POST PROCESSOR MANUAL FOR

MORI SEIKI

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

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

Table of contents:

1. Introduction

These notes are about the following post processors:

NZX1500 – 2000ST / TY2 / TY3 (formerly named NZ-T3Y3):
E12MSP_NZX-ST_2012-04-13_T1.pst: Turret 1 (upper left turret), machines on spindle 1 (main spindle) side only
E12MSP_NZX-ST_2012-04-13_T2.pst: Turret 2 (lower turret), machines on spindle 1 and spindle 2 (sub spindle) sides
E12MSP_NZX-ST_2012-04-13_T3.pst: Turret 3 (upper right turret), machines on spindle 2 side only

NZX1500 – 2000SY / Y2 (formerly named NZ-T2Y2):
E12MSP_NZX-S_2012-04-10_T1.pst: Upper turret, machines on spindle 1 and spindle 2 (sub spindle) sides
E12MSP_NZX-S_2012-04-10_T2.pst: Lower turret, machines on spindle 1 and spindle 2 (sub spindle) sides

NZX1500 – 2000SDL / Y2DL (formerly named NZ-T2Y2DL):
E12MSP_NZXDL_2012-01-11_T1.pst: Upper left turret, machines on spindle 1 side
E12MSP_NZXDL_2012-01-11_T2.pst: Upper right turret, machines on spindle 2 (sub spindle) side

Note: NZX-ST has no Y-axis; NZX-STY2 has a Y-axis on upper left and lower turrets and on NZX-STY3 all turrets have a Y-axis.
The output will match your machine configuration based on the loaded NZX Machine Setup in ESPRIT. For example, if you work on a NZX-ST, no Y-axis movements will be output in the code.
In the following manual, you can ignore any reference to the Y-axis if your machine is not equipped with this axis.

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

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

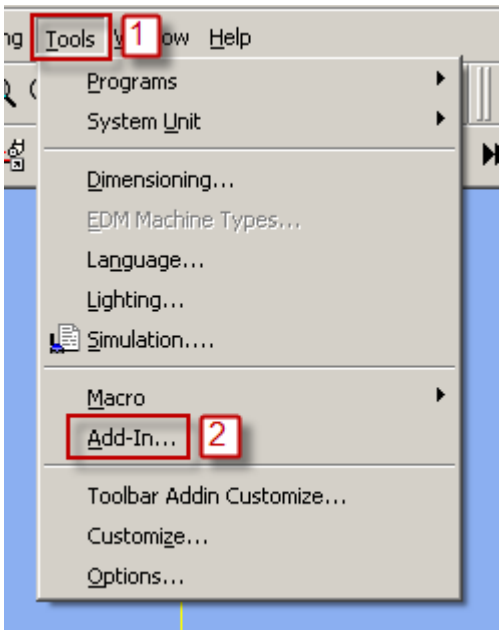
Please refer to NZX programming manual for detailed information on the related G and M-codes.

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

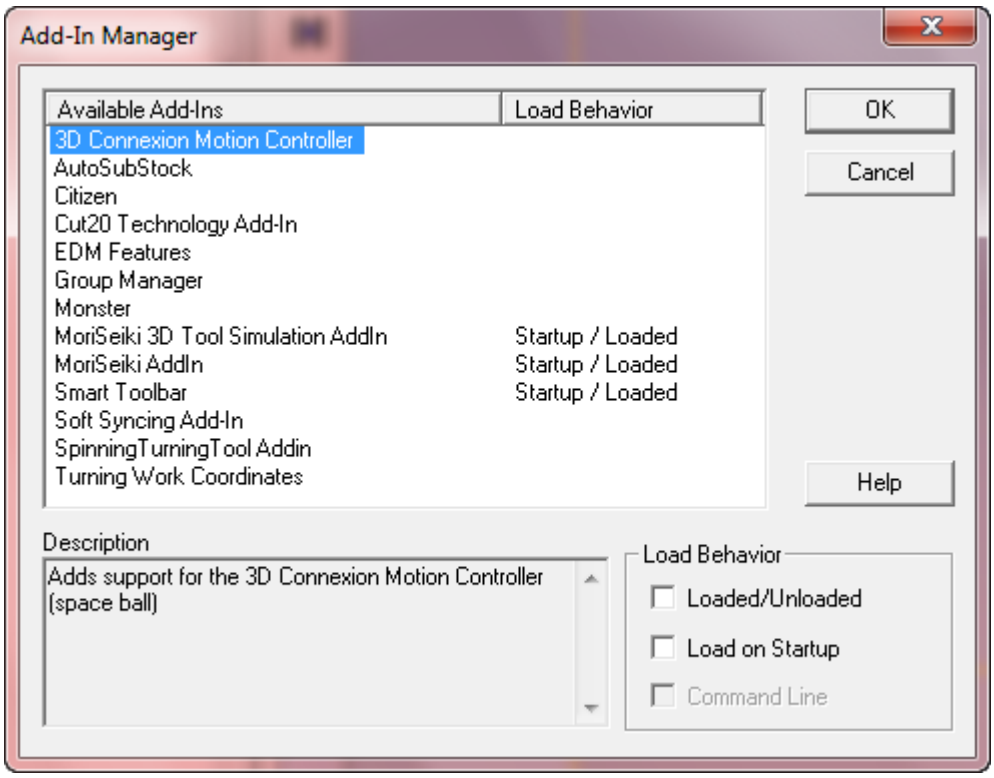
2. Required add-ins

2.1. How to load an add-in ESPRIT

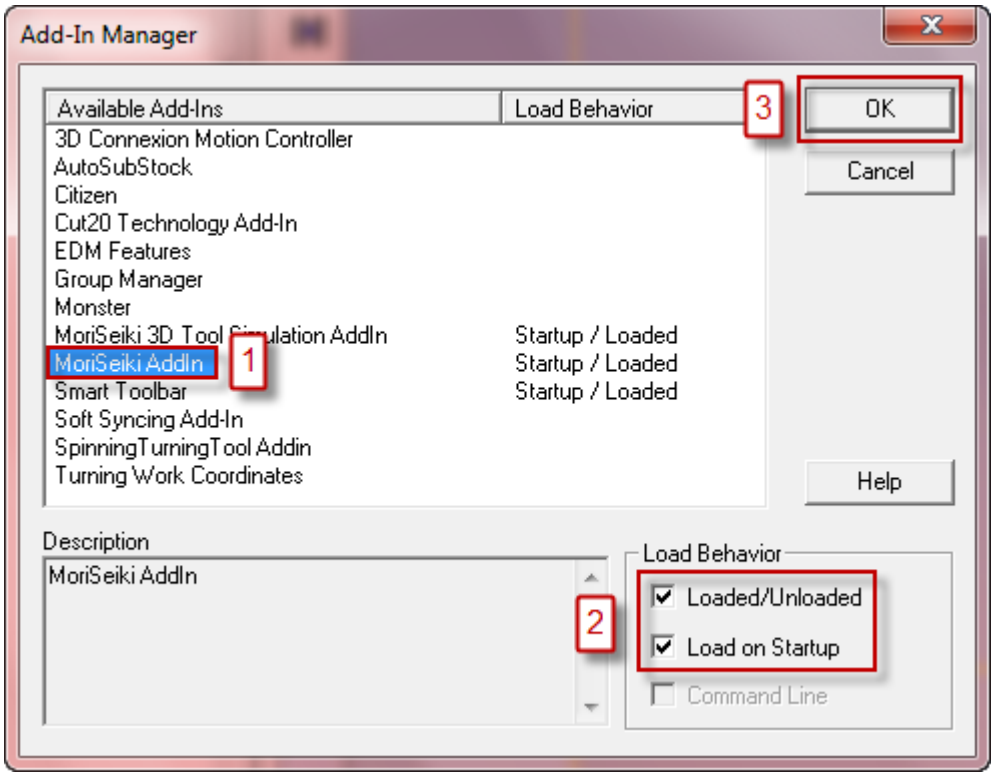
To load an add-in in ESPRIT, from the Tools menu, select Add-In...:



You will then see the add-in window:



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.



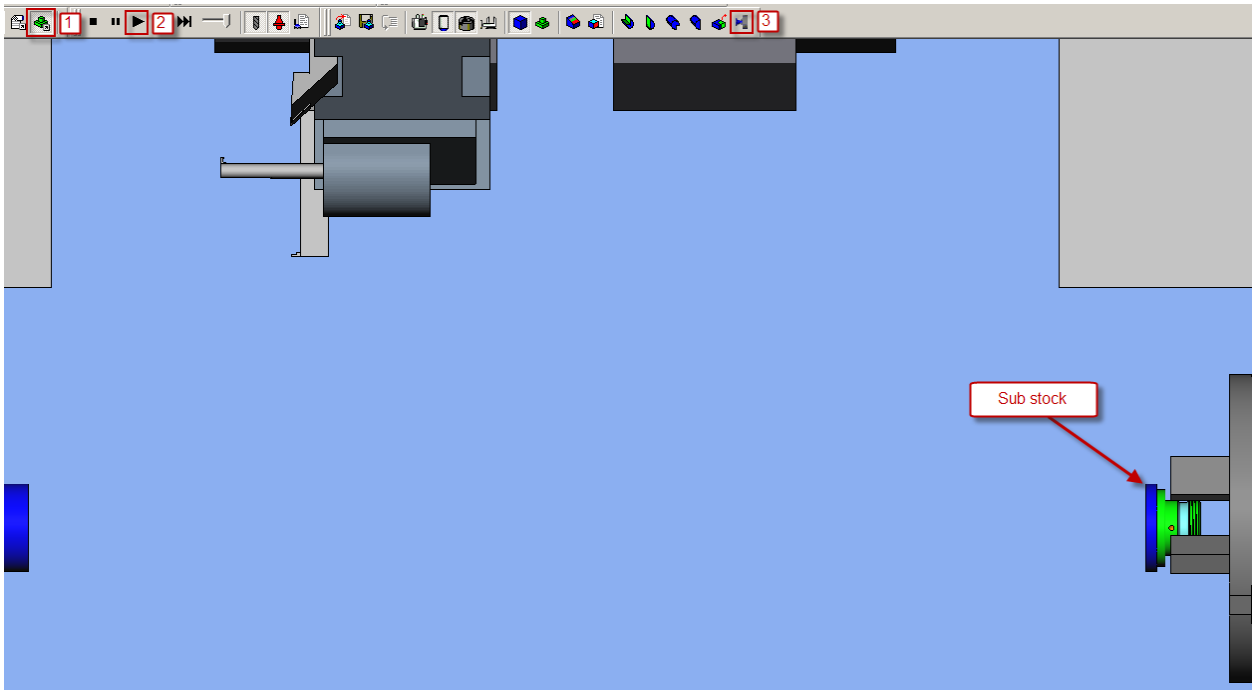
For the Mori Seiki NZX series, you will need to load the following add-ins:

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

2.2. AutoSubStock (optional)

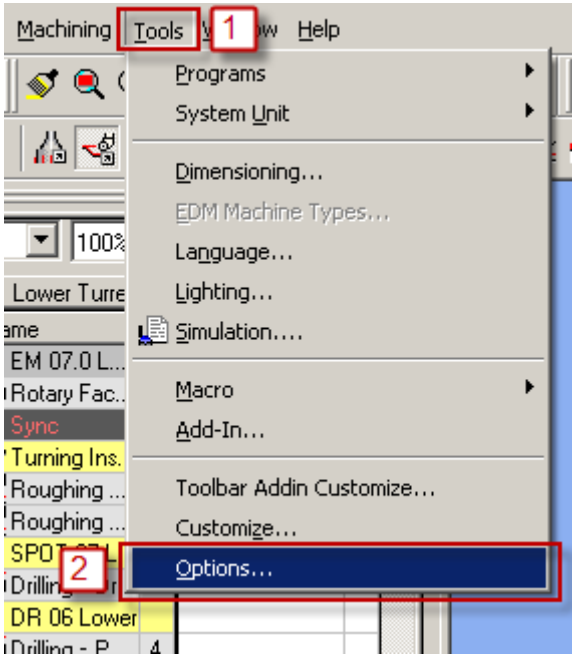
The AutoSubStock add-in needs to be turned on prior to open a file. This add-in will allow you to correctly simulate the spindle 2 machining. It will also allow the simulation of production machining, when cutting on the spindle 1 and the spindle 2 at the same time.

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

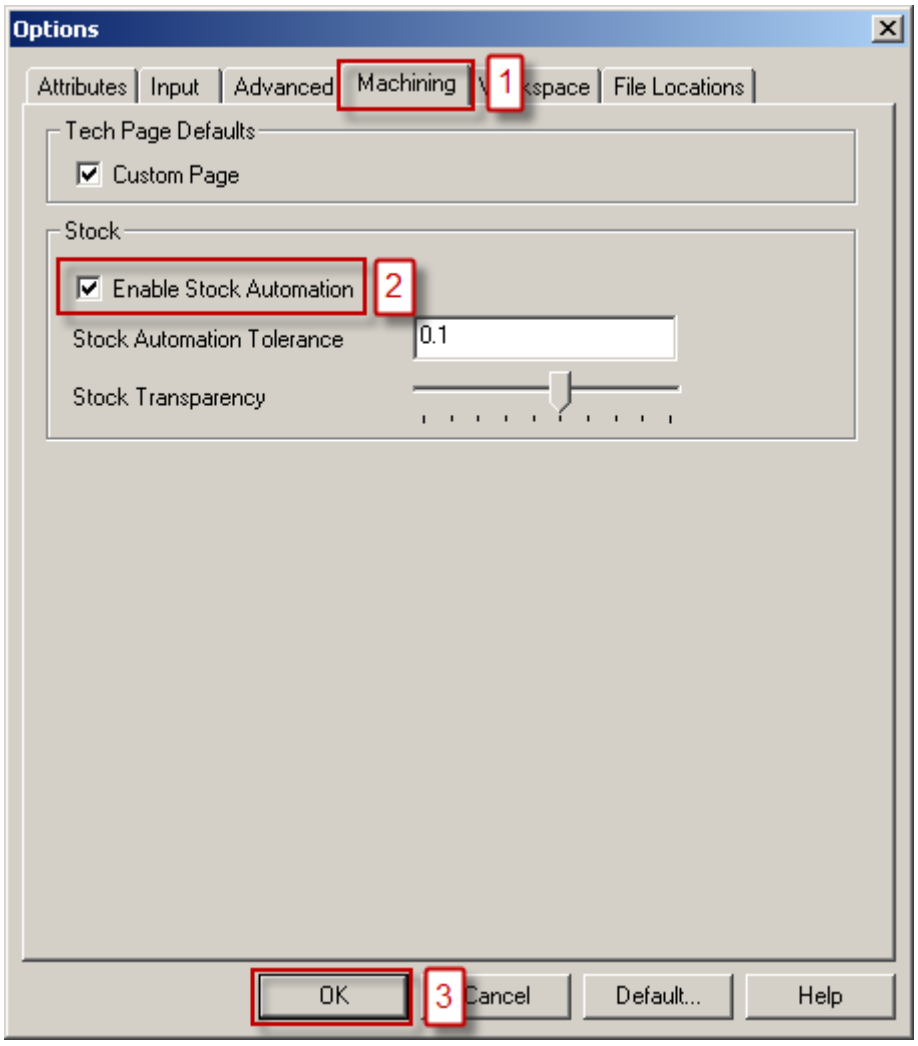


If you restart the simulation, you will be able to see the cut on the spindle 1 and spindle 2 sides all at the same time.

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



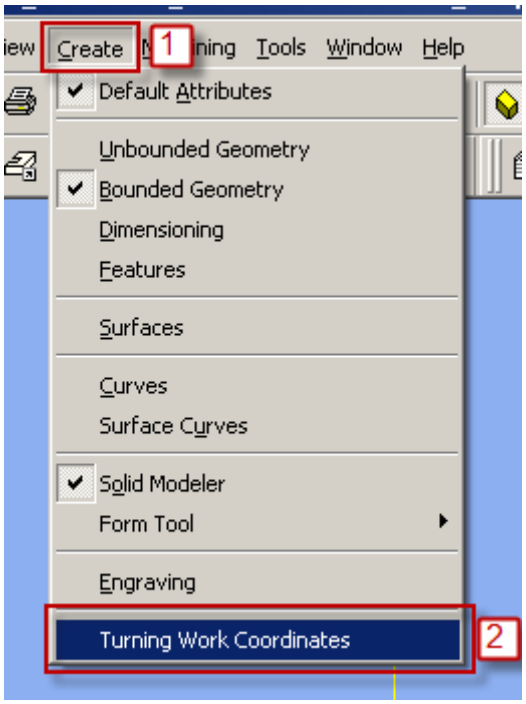
On the Machining tab, if you check Enable Stock Automation, you will not need to turn off the AutoSubStock add-in since the Stock Automation will compute the state and shape of your stock present in both the spindle 1 and spindle 2:



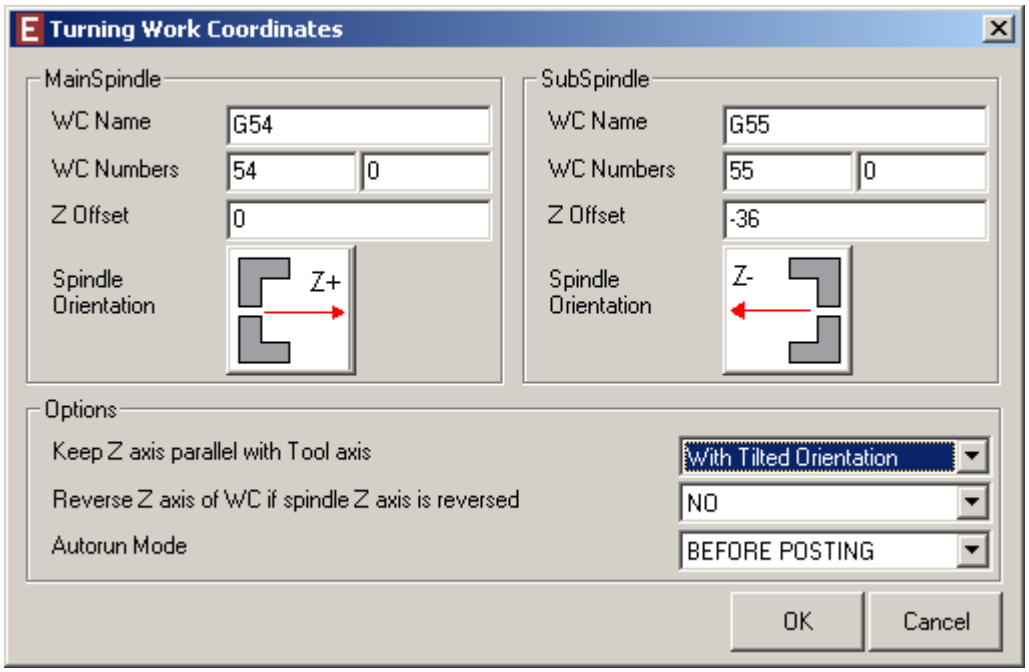
2.3. Turning Work Coordinates

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.



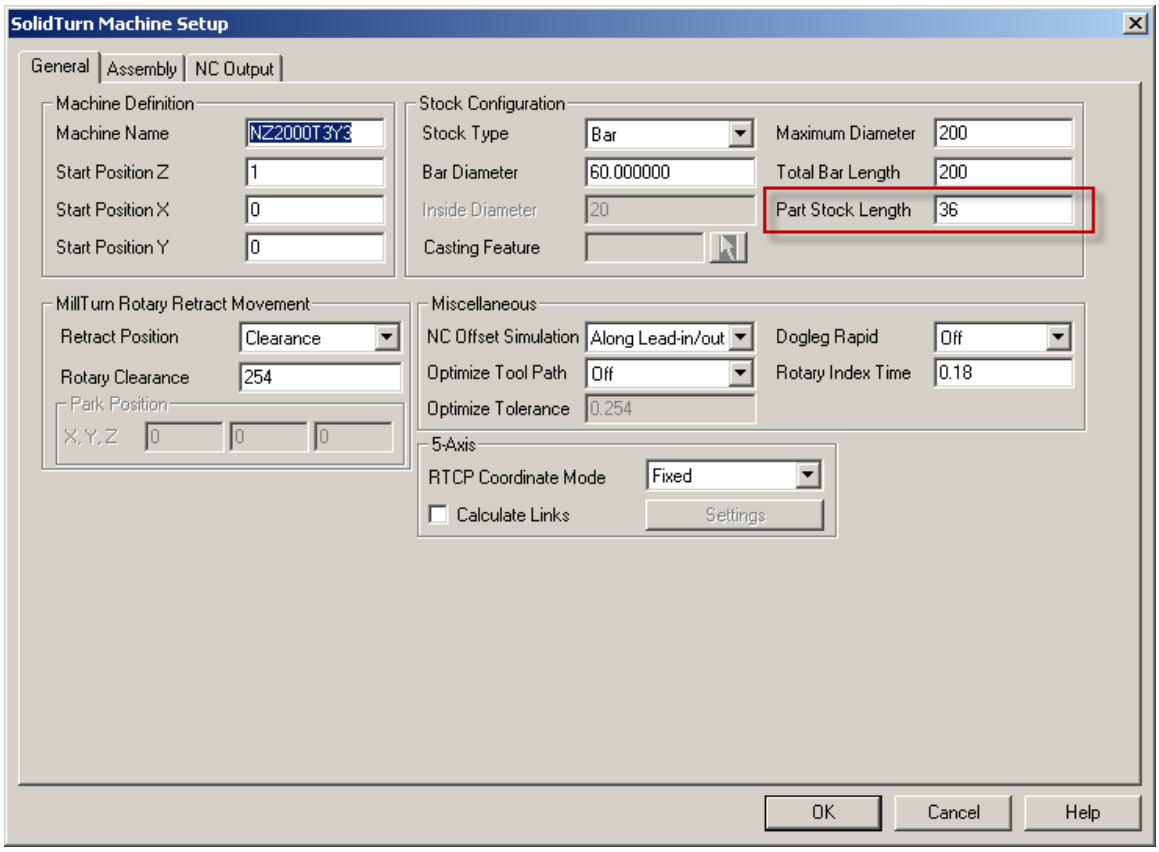
You will then see the Turning Work Coordinates dialog:



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.



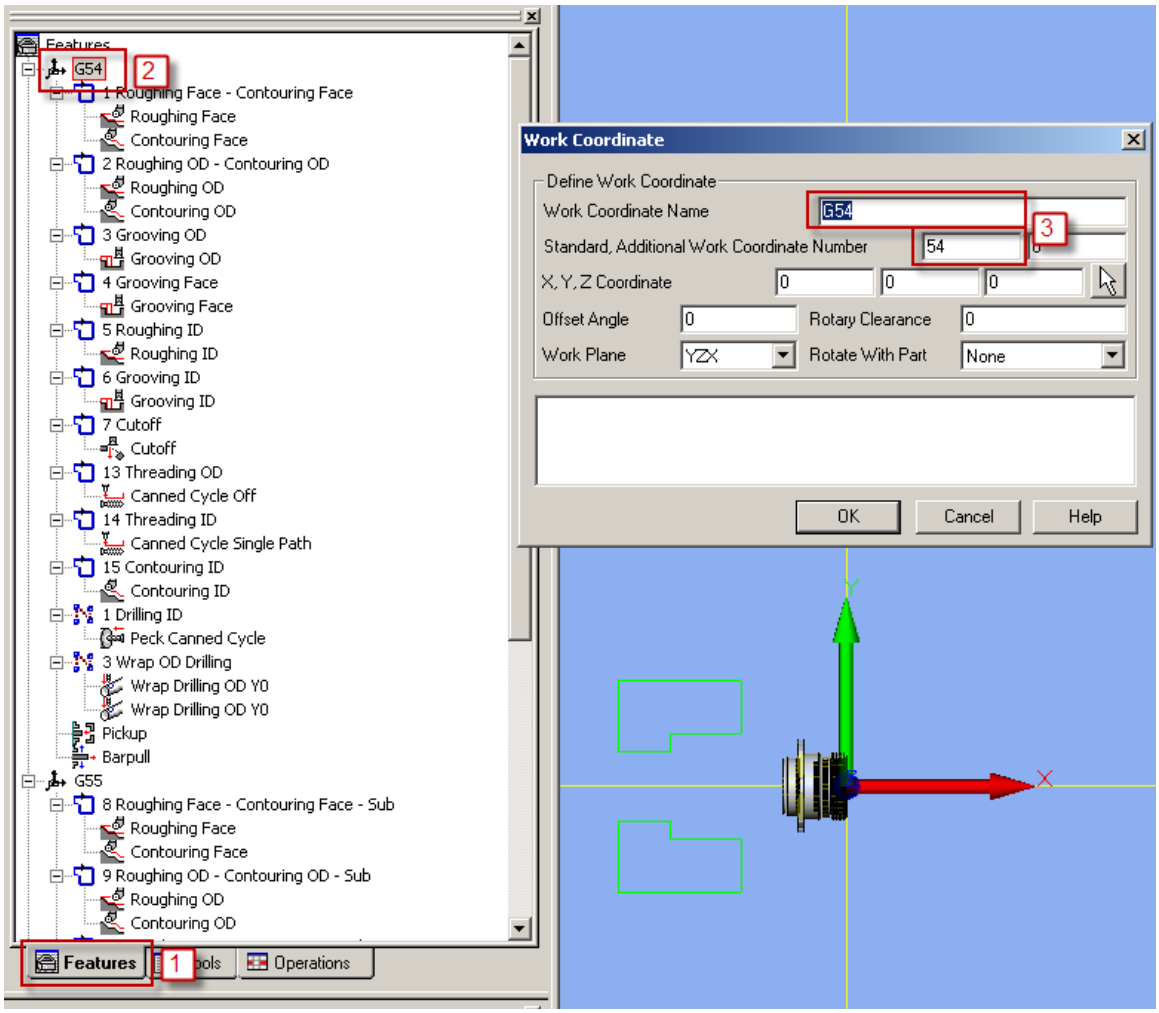
- **Spindle Orientation**: This information is coming from the Machine Setup and is just informative.

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.

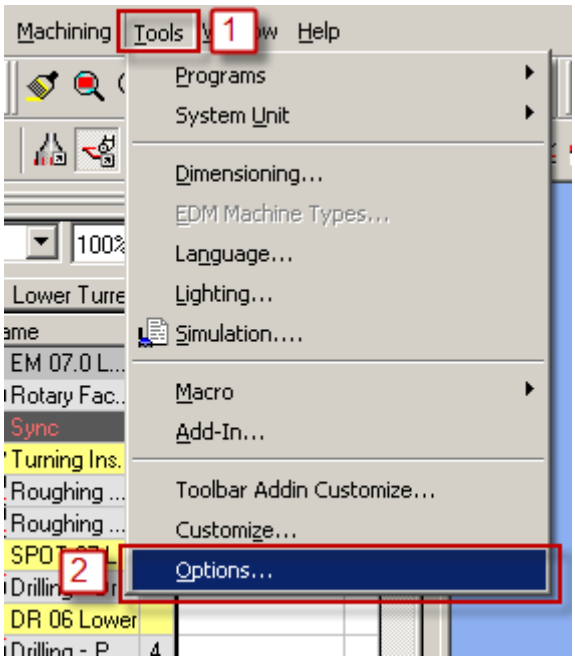
- **None**: No autorun mode selected, you have to manually run the add-in to sort operations.
- **Before posting**: Every time you will output the NC code, the add-in will be run.
- **Before simulation**: Every time you will start the simulation, the add-in will be run.

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

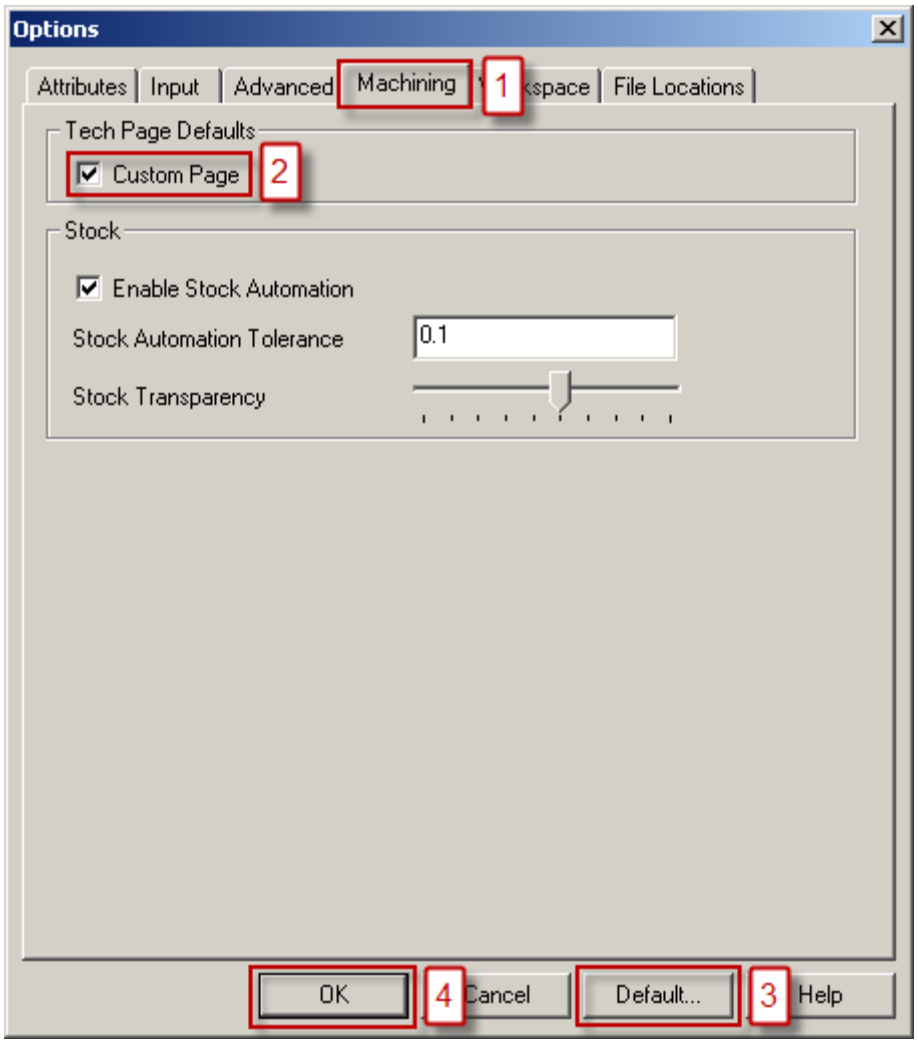
3. How to turn on Custom Pages / POST Output Configuration

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

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

4. Machine Setup

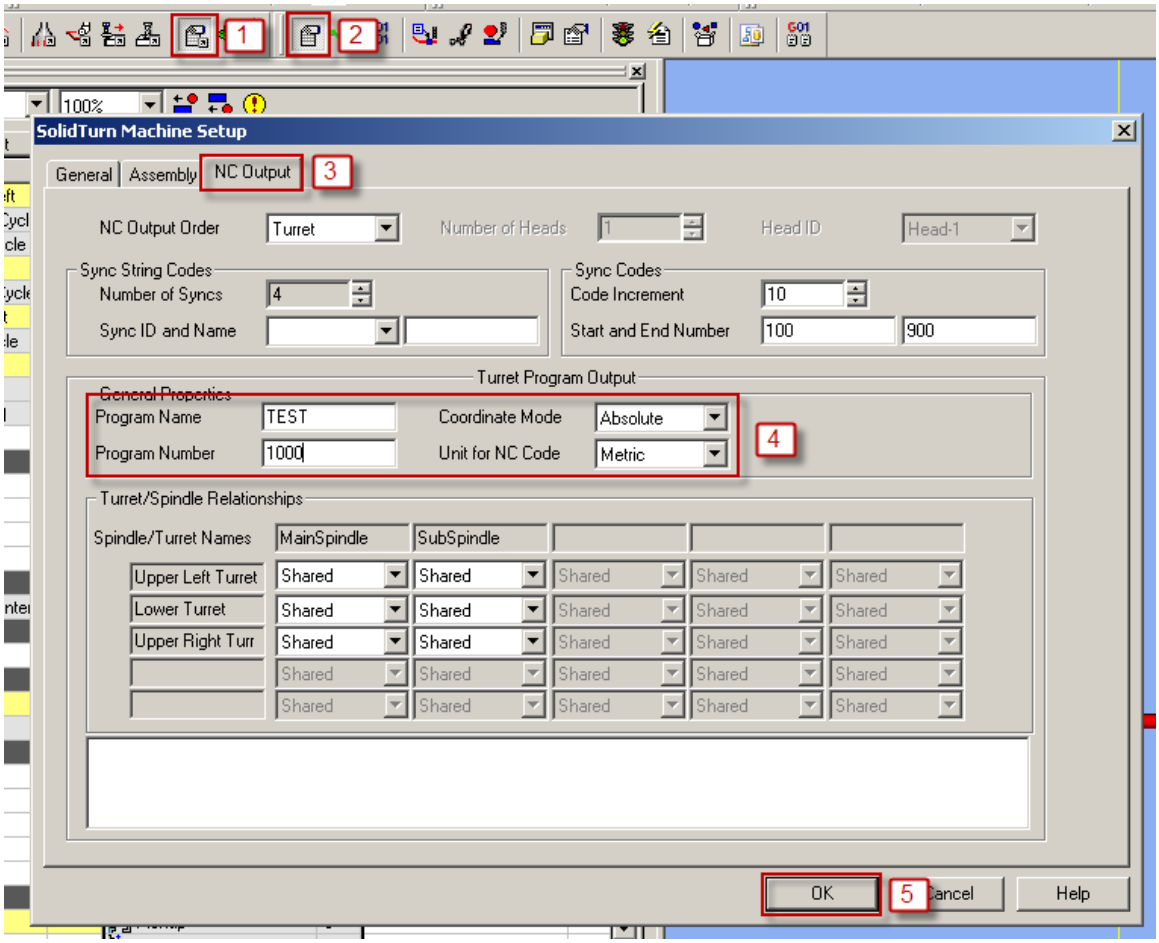
4.1. Introduction

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

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

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.

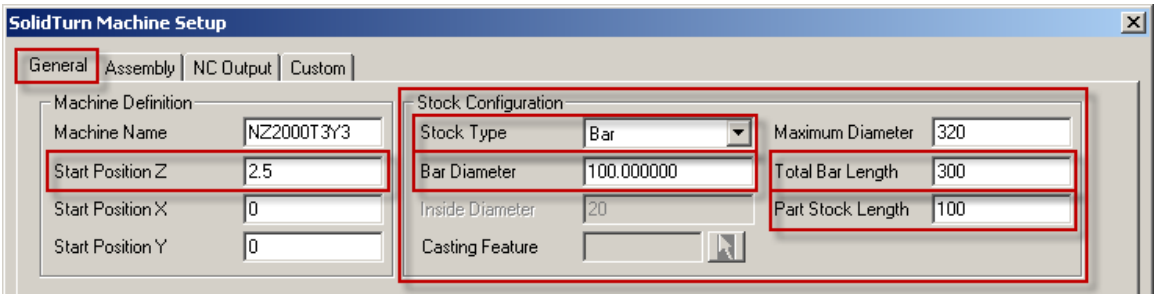


4.3. Define the turning stock

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.

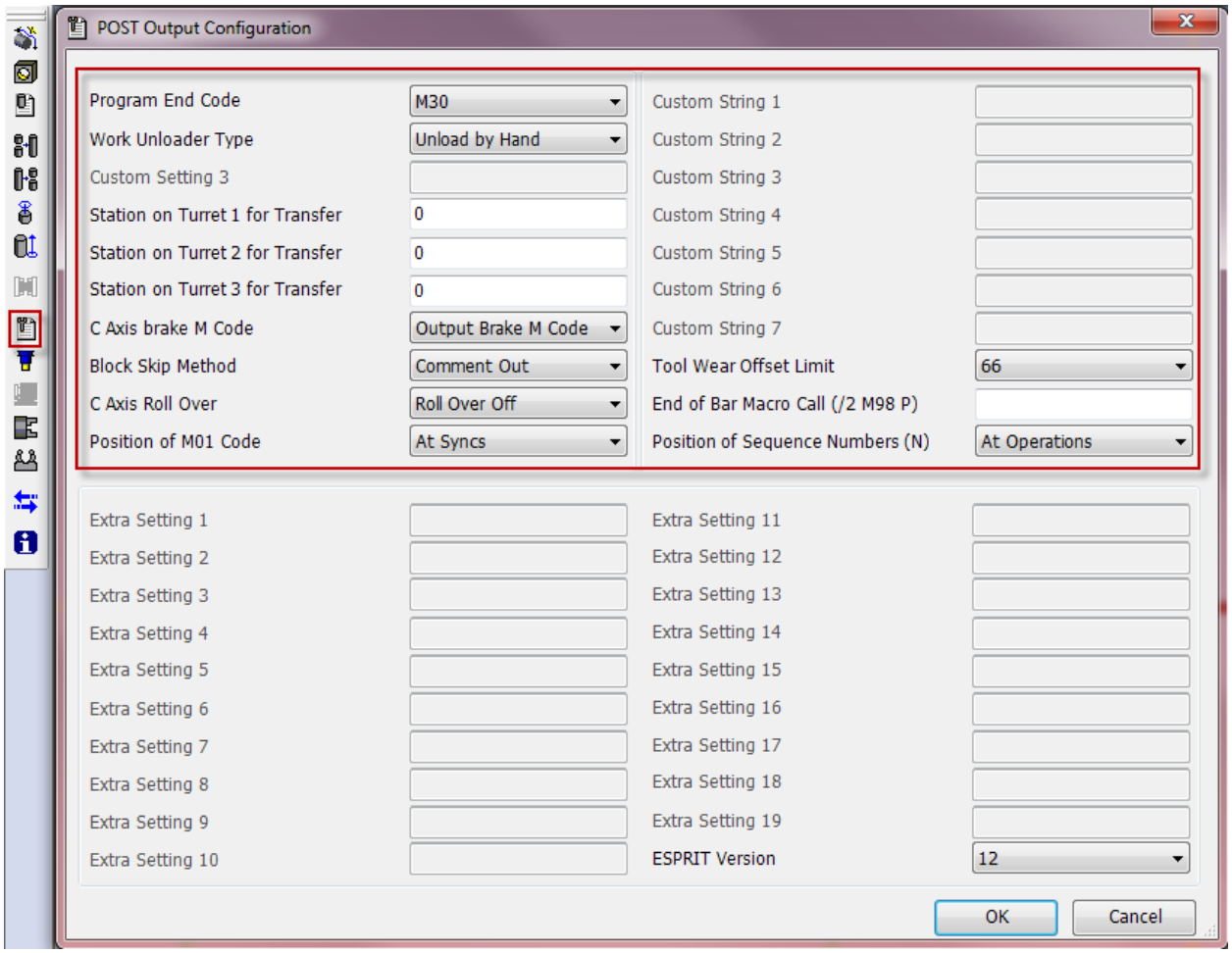


Note: The stock configuration will be detailed when needed in this manual.

4.4. Machine parameters

4.4.1. Introduction

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



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

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

Please note that M30 is the default.

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

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

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

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

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

4.4.4. Tool station for part transfer and work unloader

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

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

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

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

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

4.4.5. C-axis brake clamp/unclamp

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

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

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

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

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

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

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

4.4.7. C-axis roll over

On the POST Output Configuration, **set C-Axis Roll Over** to **Roll Over On** to turn on C-axis roll over.
Note that you will need to turn it on on the machine control as well.

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

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

In ESPRIT, C-Axis Roll Over on the POST Output Configuration is set to Roll Over Off.

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

4.4.3. Set position of sequence numbers (N)

By default, sequence numbers (N) will be output at the beginning of each operation in the NC code.
Set on the POST Output Configuration **Position of Sequence Numbers (N)** to **At Tool Changes** to output sequence numbers at tool change only.

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

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

5. Tools (T function)

5.1. T function specifications

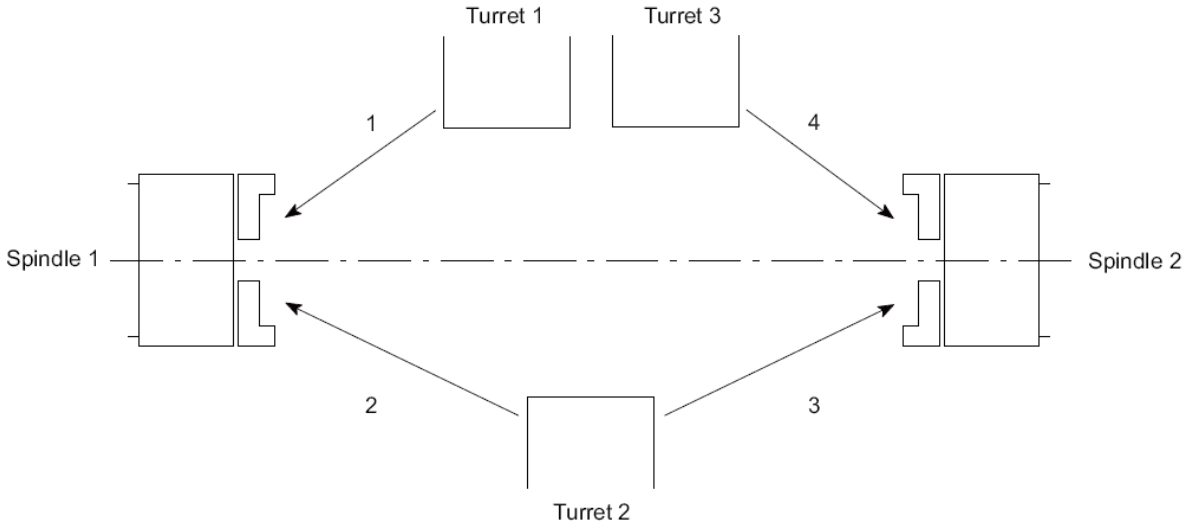
5.1.1. Introduction

T[][][][];

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

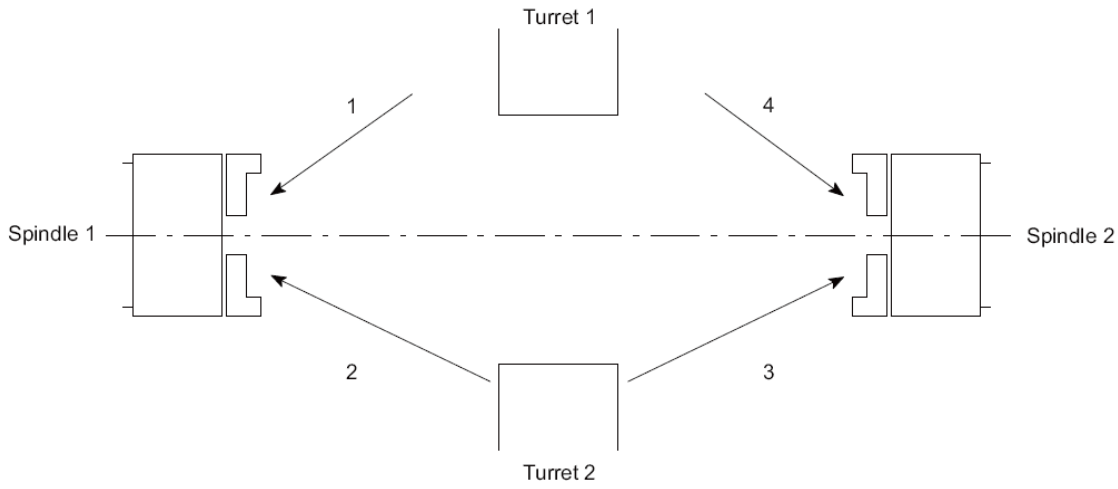
The last two digits of a T number specify the tool wear offset number.

NZX1500 – 2000ST / TY2 / TY3:



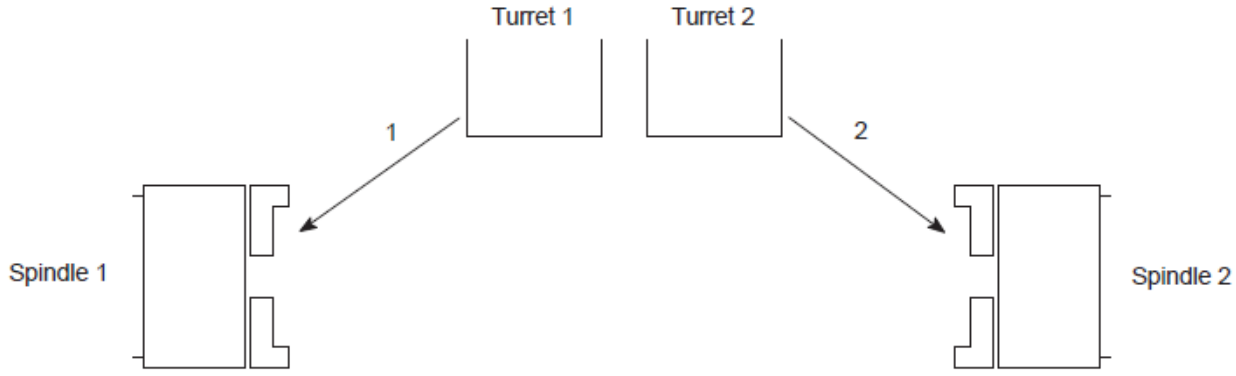
	Turret and Spindle to Use		HEAD and T Code to Specify	
1	Turret 1	Spindle 1	HEAD1	T0101 - T1616
2	Turret 2	Spindle 1	HEAD2	T0101 - T1616
3	Turret 2	Spindle 2	HEAD2	T3131 - T4646
4	Turret 3	Spindle 2	HEAD3	T3131 - T4646

NZX1500 – 2000SY / Y2:



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

NZX1500 – 2000SDL / Y2DL:



	Turret and Spindle to Use		HEAD and T Code to Specify	
1	Turret 1	Spindle 1	HEAD1	T0101 - T1616
2	Turret 2	Spindle 2	HEAD2	T0101 - T1616

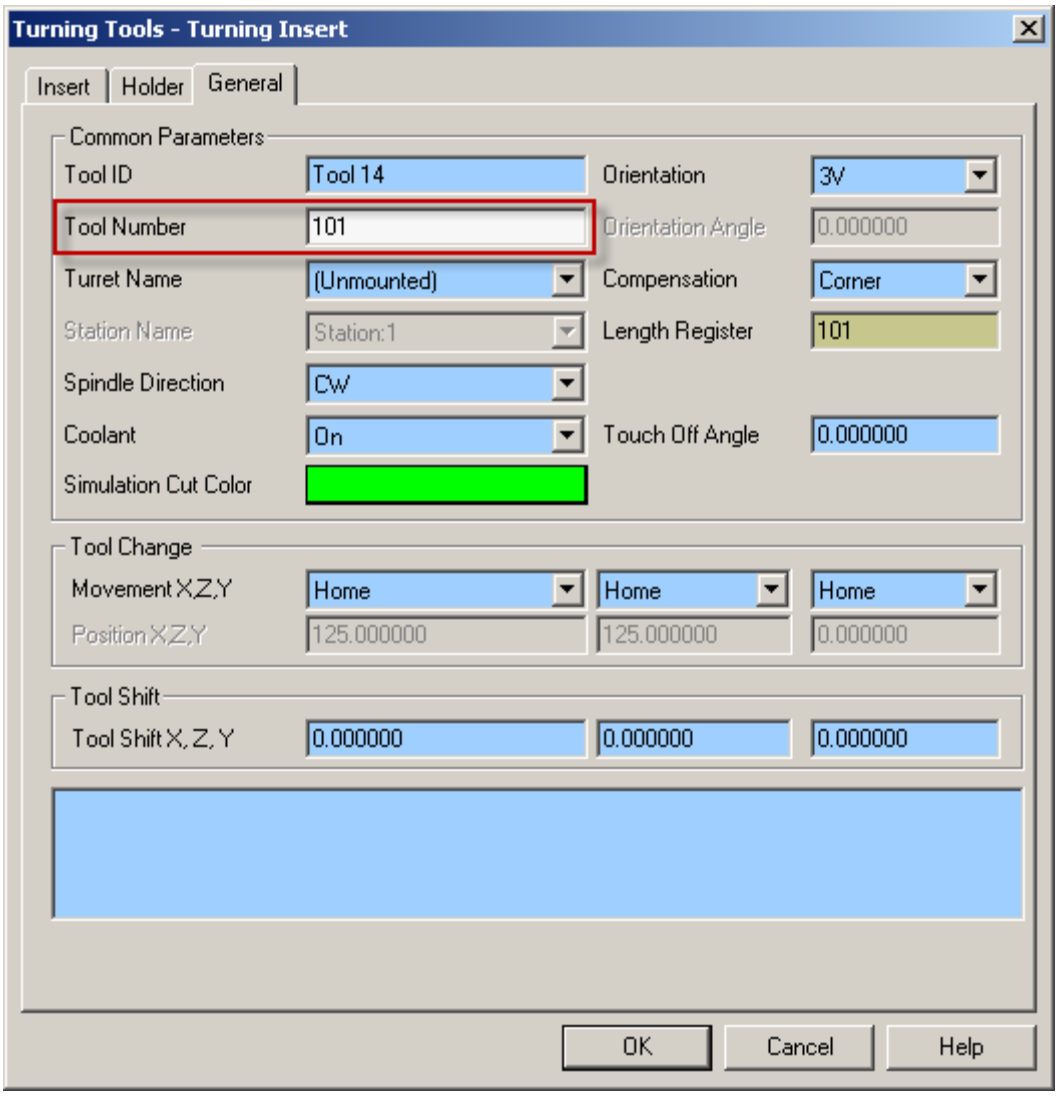
Note: It is recommended to use the same number for the tool number and tool wear offset number when specifying a T command in a program to avoid operator's errors.

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

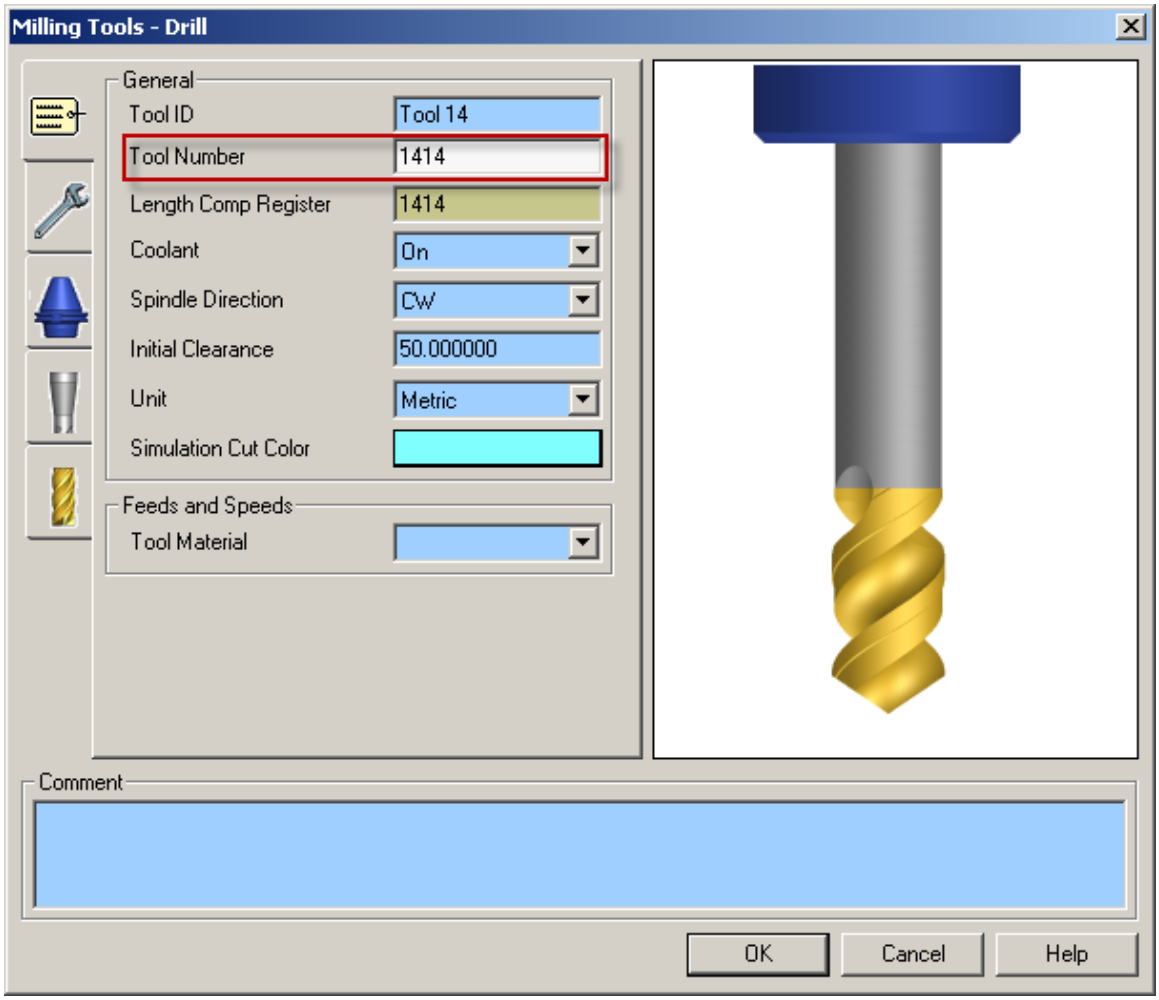
5.1.2. Specify the tool number: Method 1

A tool number in ESPRIT shall be entered as three- (101, for example) or four-digit numbers (1414, for example). It will be entered in the Tool Number dialogue box on the respective tool page. It will then be output in the NC code.
If, for example on turret 1, the Tool Number entered is 101, the tool call output will be T0101. If 1414 is entered, the tool call output will be T1414.

Turning tools:



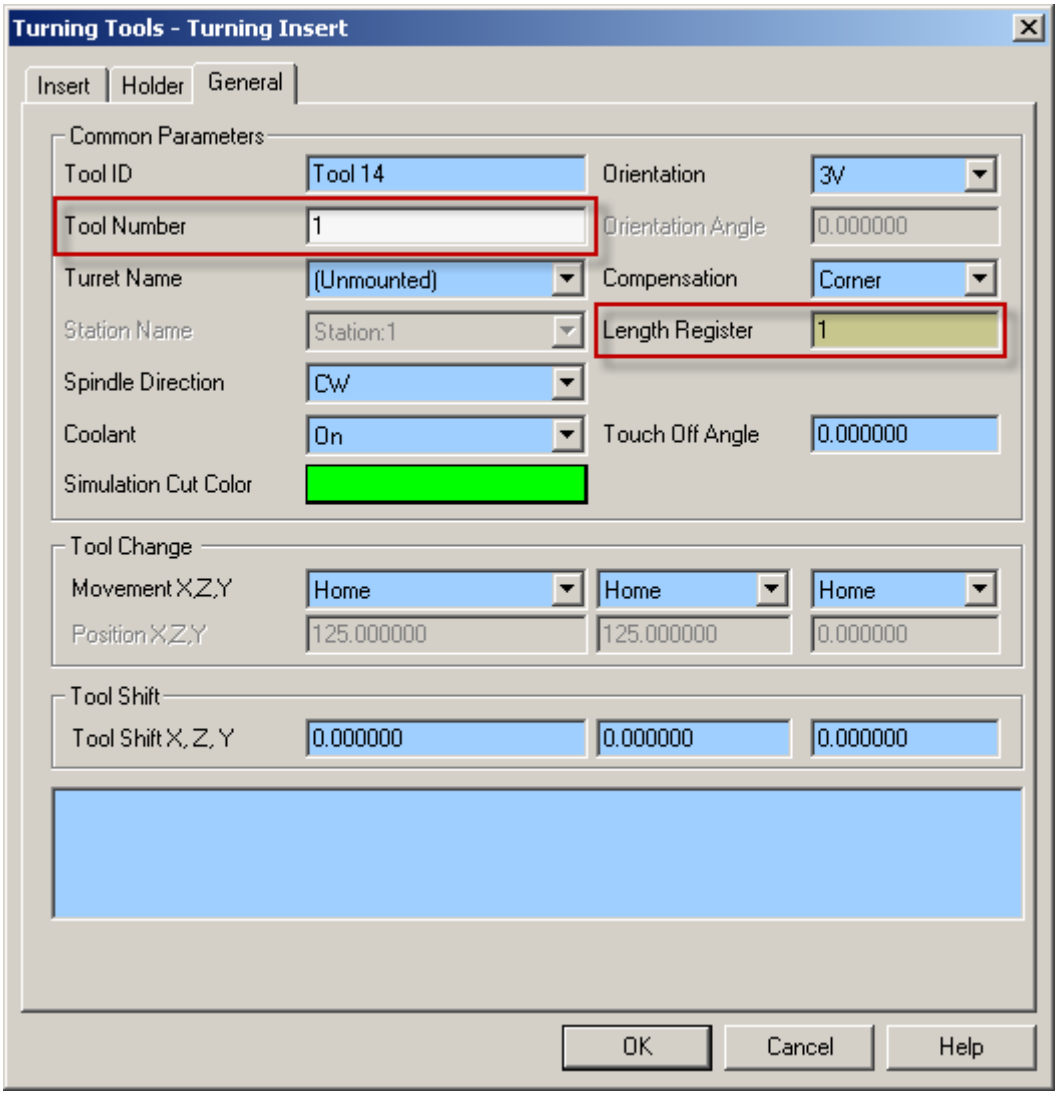
Milling tools:



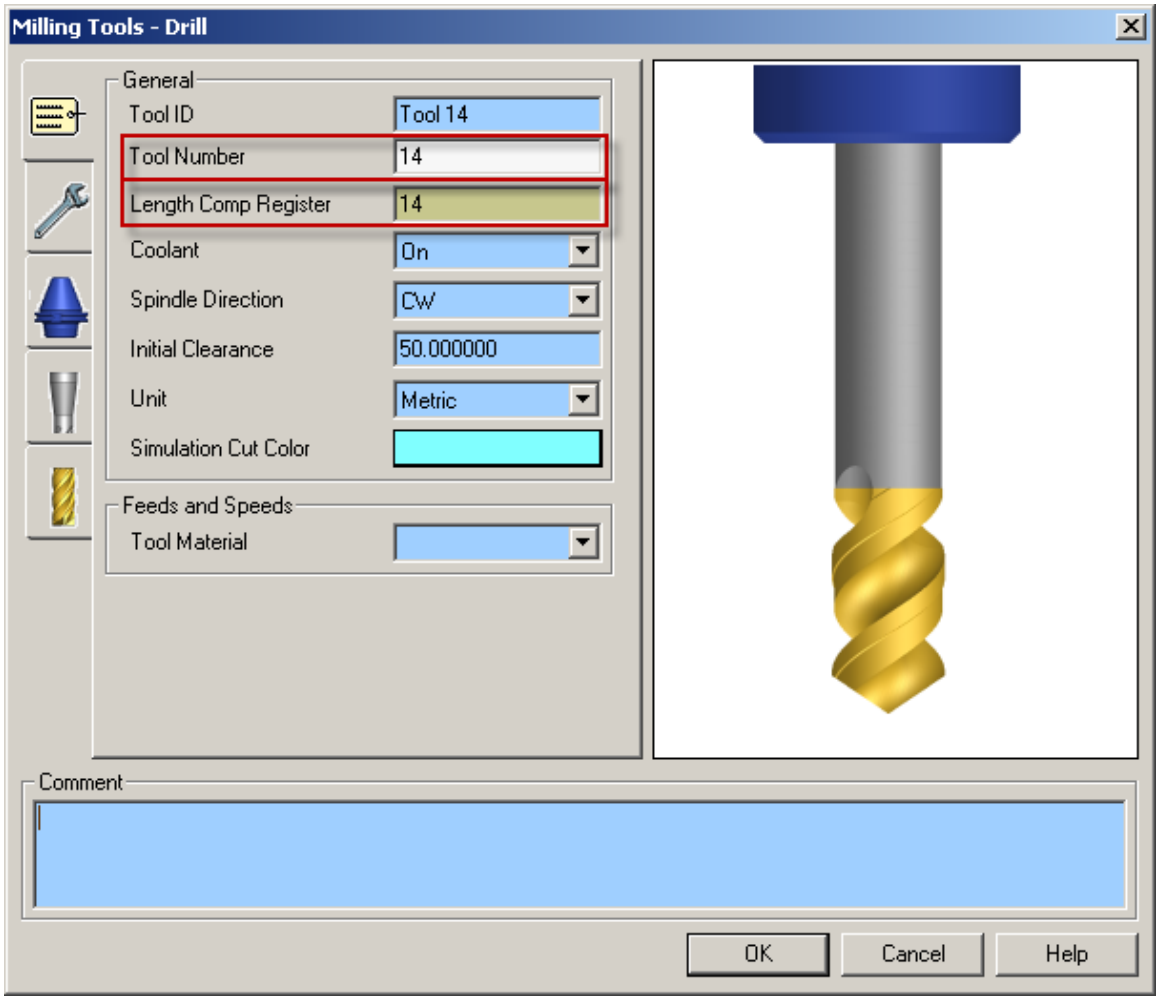
5.1.3. Specify the tool number: Method 2

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:



Milling tools:



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

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

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

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

See the examples below:

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

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

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

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

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

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

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

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

5.1.5. Restrictions

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

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

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

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

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

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

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

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

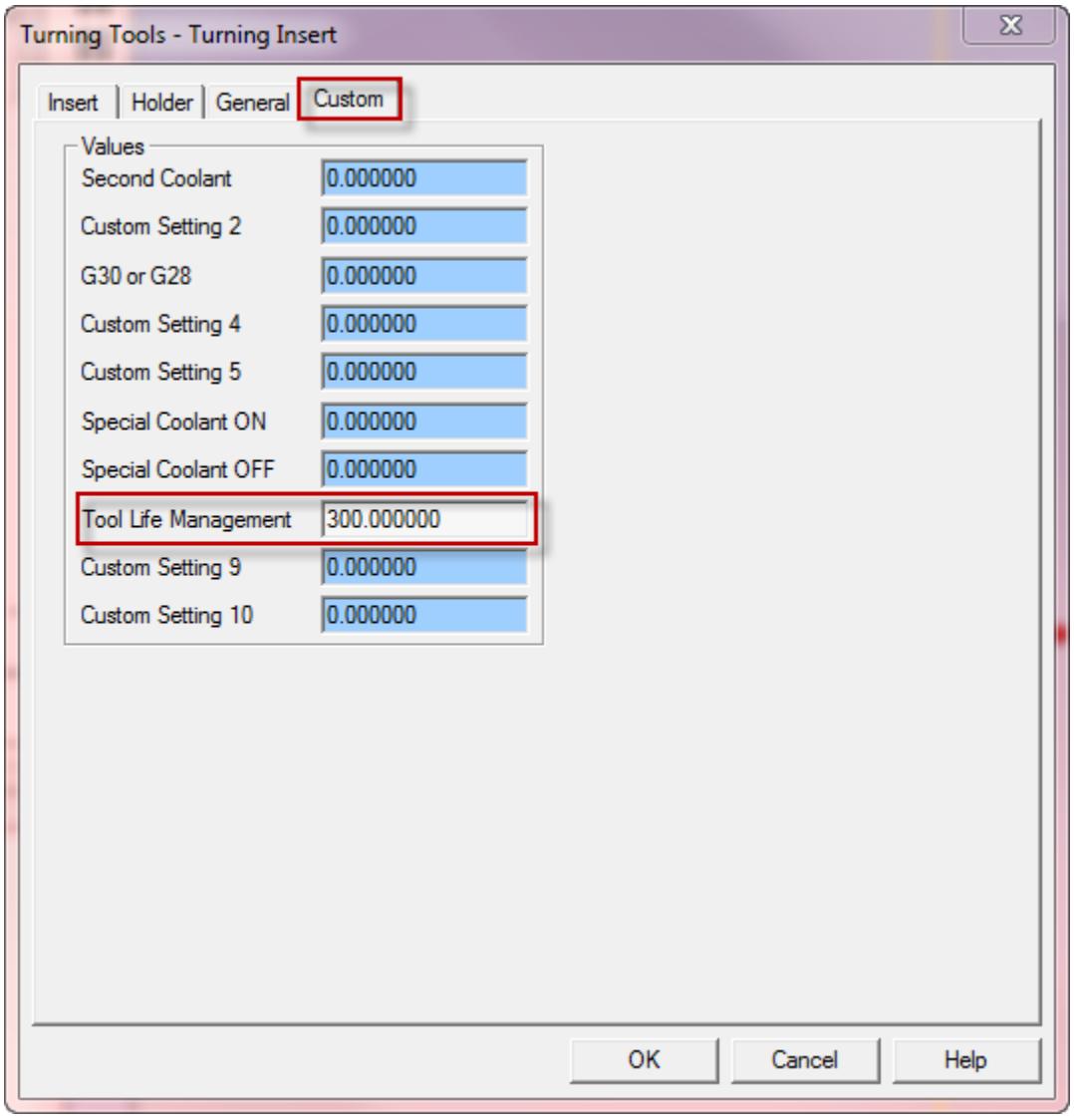
- **Head 1:** The entered Length Register number has to be contained **between 1 and 66**.
- **Head 2:** The entered Length Register number has to be contained **between 1 and 66**.

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

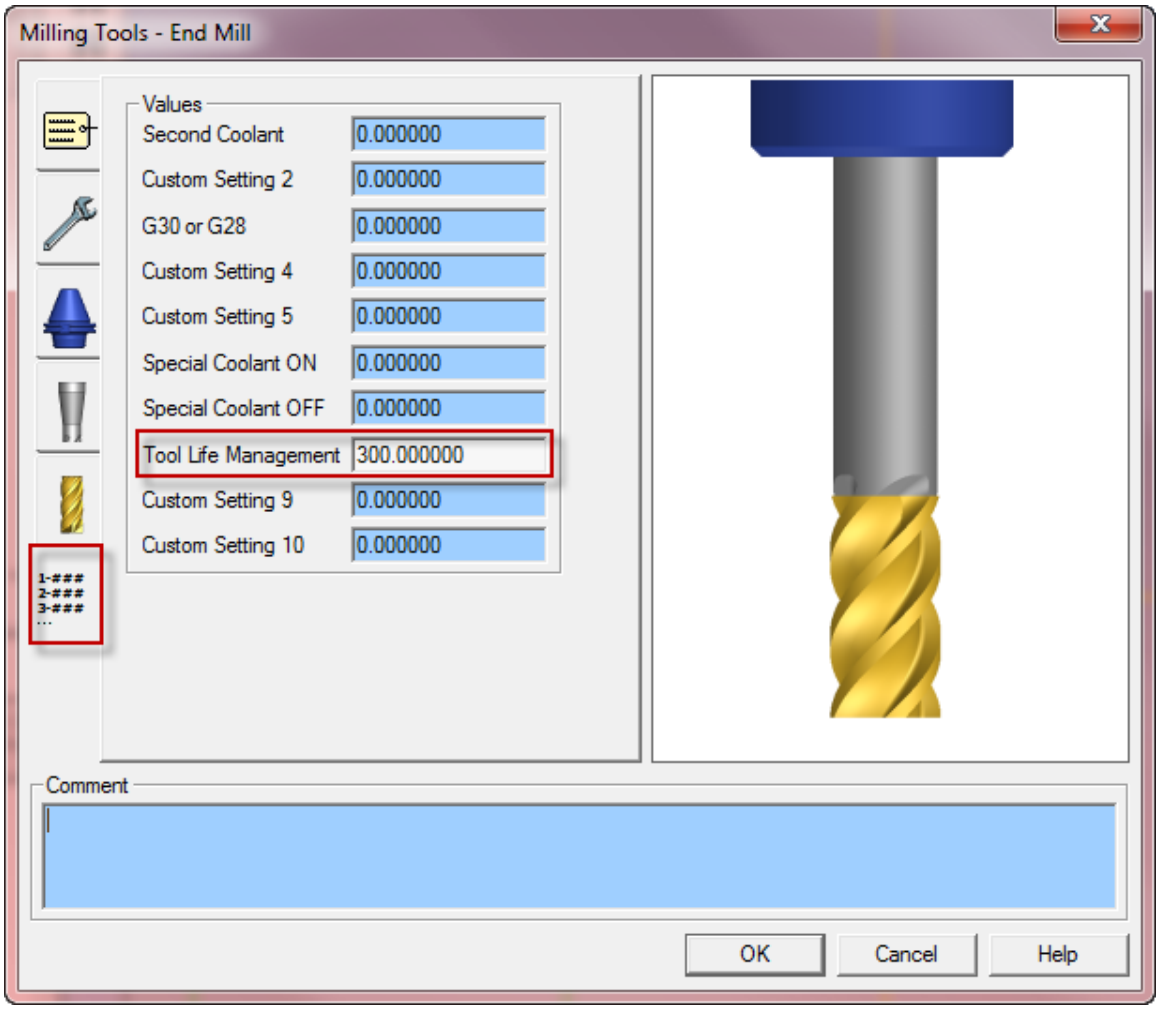
5.1.6. *Tool life management*

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

Turning tool page:



Milling tool page:



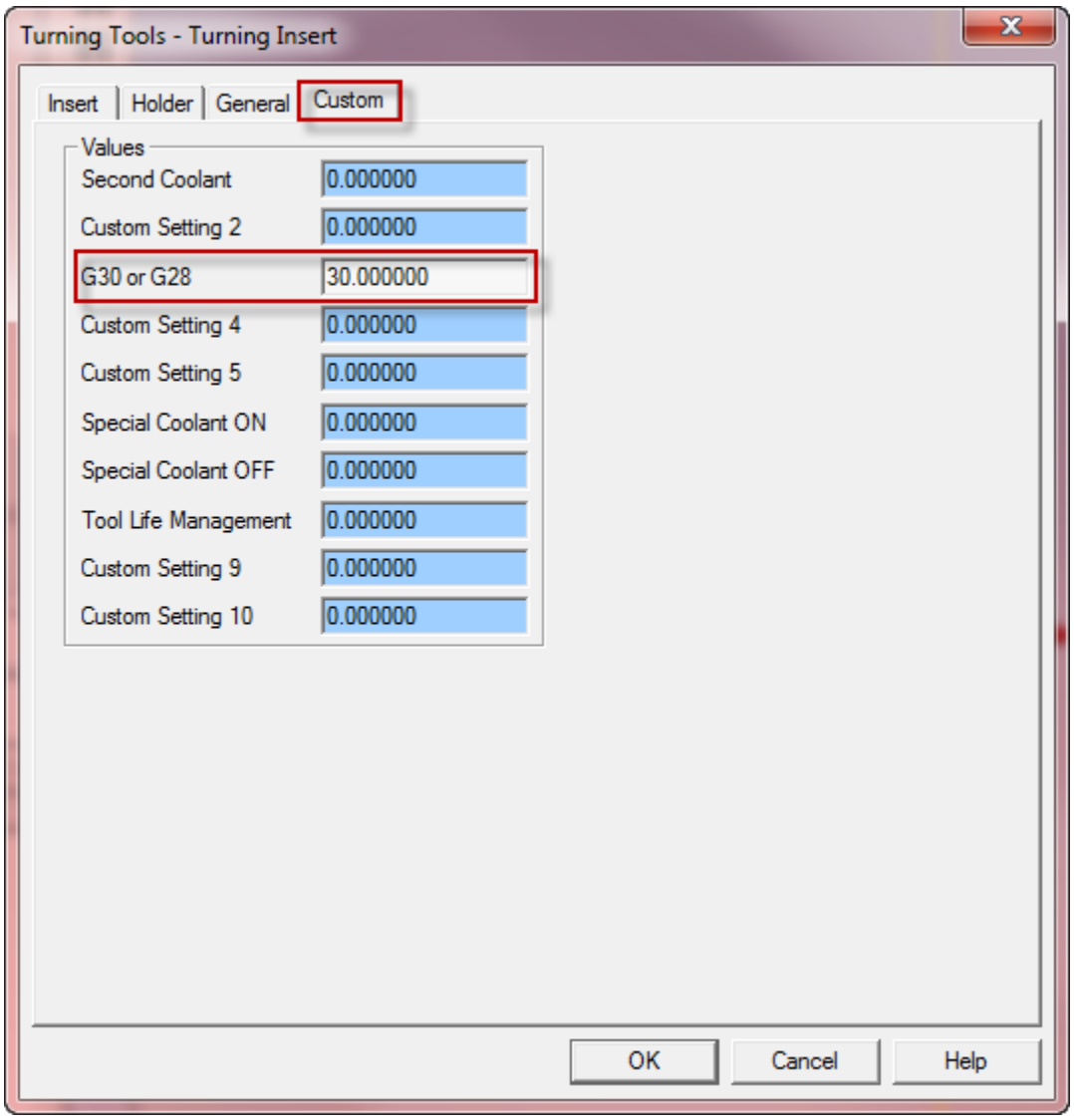
This will trigger the output of G336 (group command) and M300 (tool life count): The tool call will be made with G336 and at tool cancel M300 will be output.

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

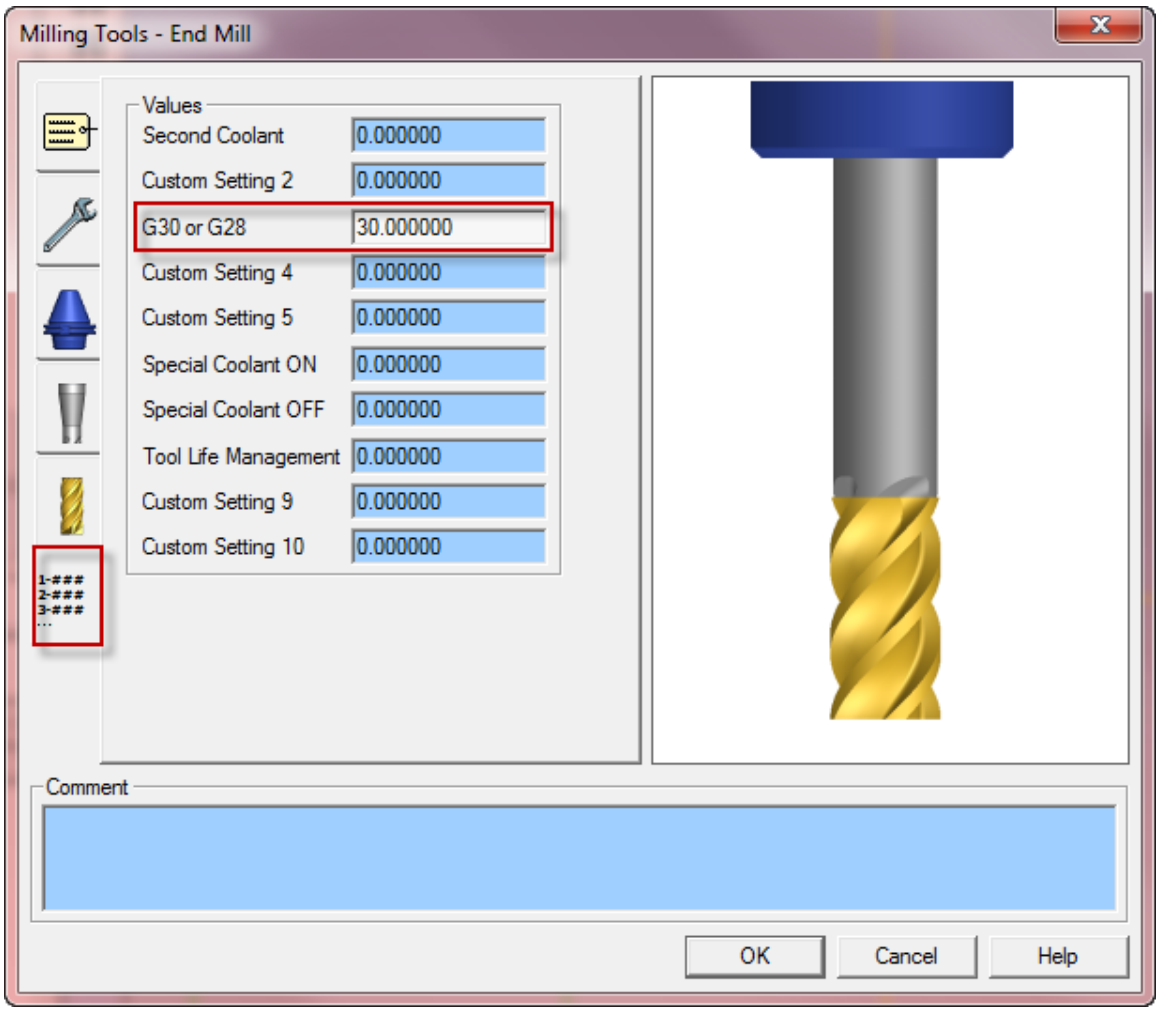
5.1.7. Second home position

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

Turning tool page:



Milling tool page:



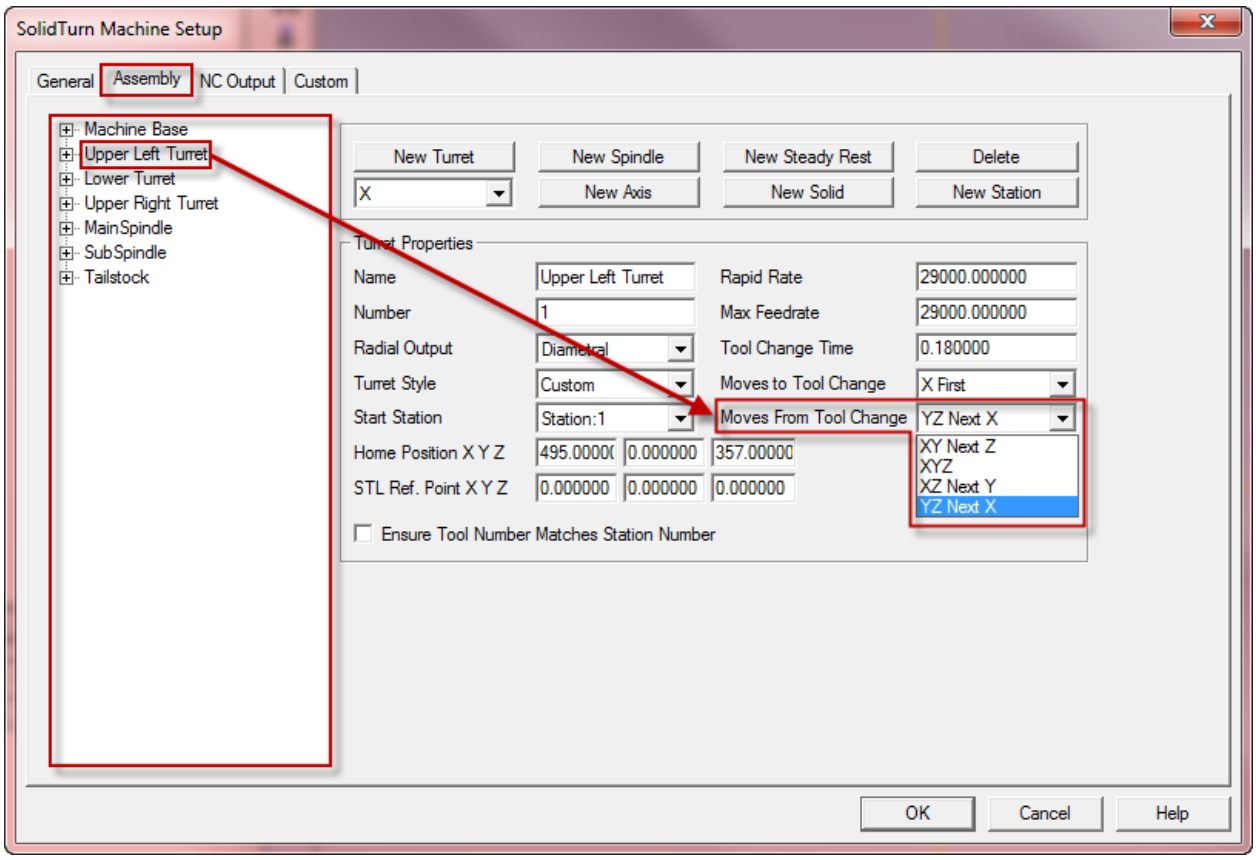
This will trigger the output of G30 (instead of G28) whenever a turret needs to be sent home.

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:

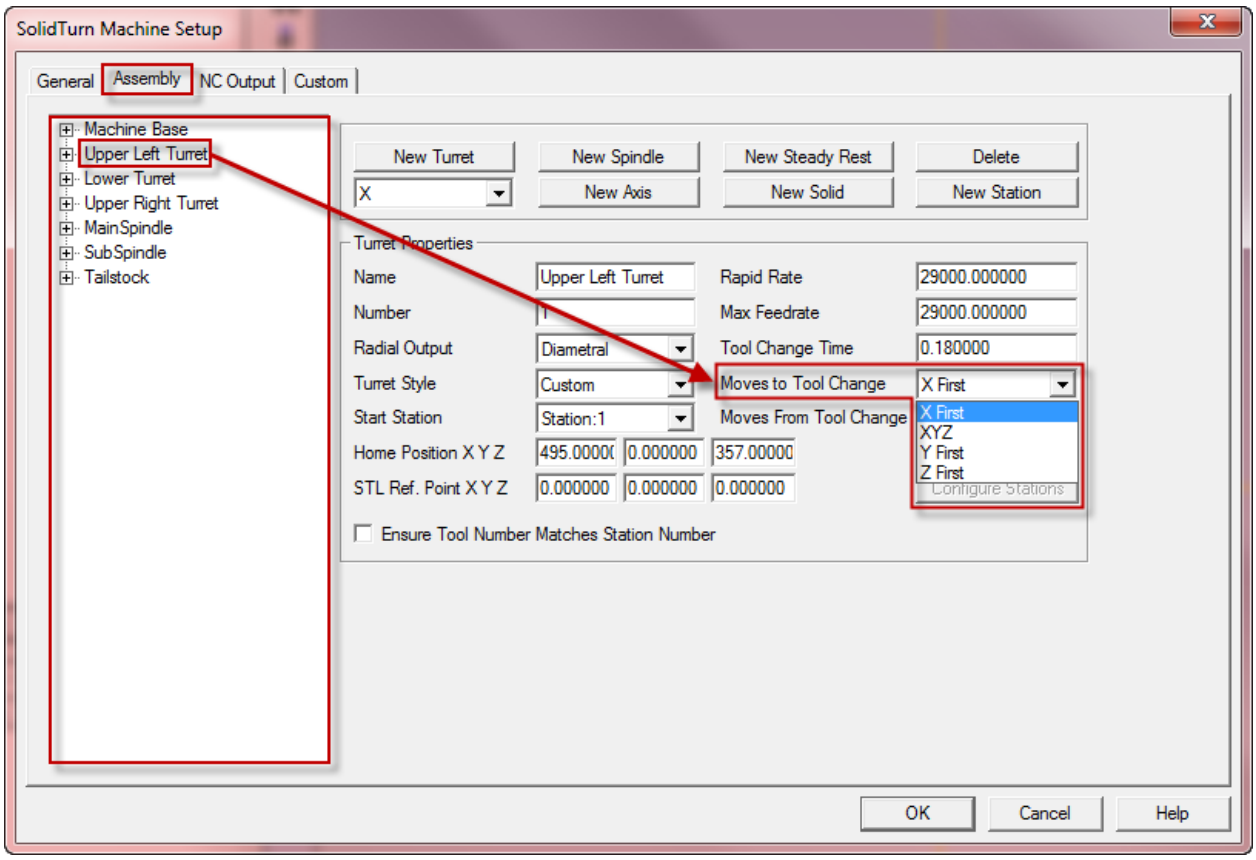
- **XY Next Z:** The tool moves along X and Y only to the entry location, then moves into position along the Z-axis.
- **XYZ:** Linear interpolation is used to move the tool from the tool change position to the start of the cut in a straight line.
- **XZ Next Y:** The tool moves along X and Z only, then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

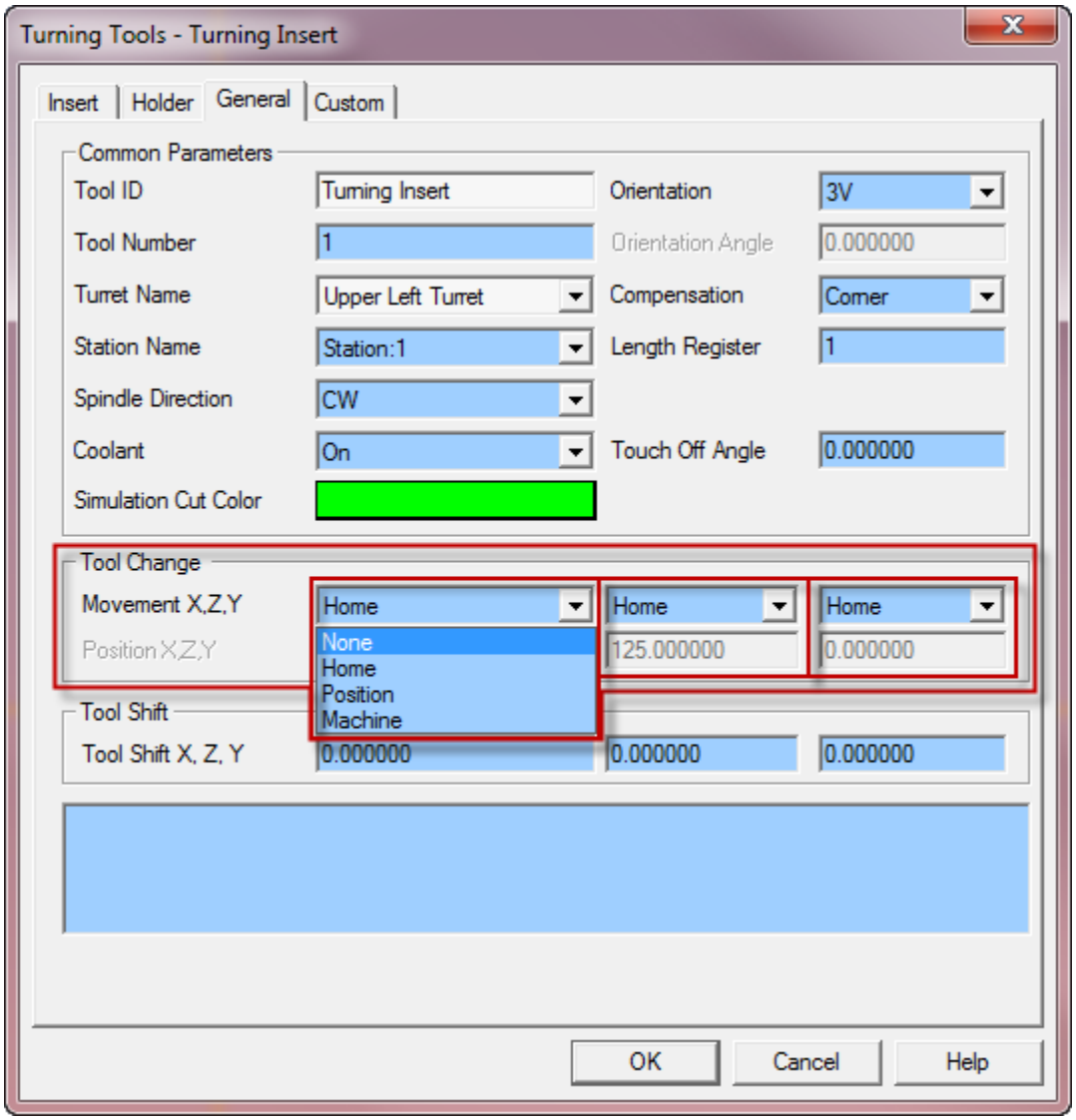
Notes:

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

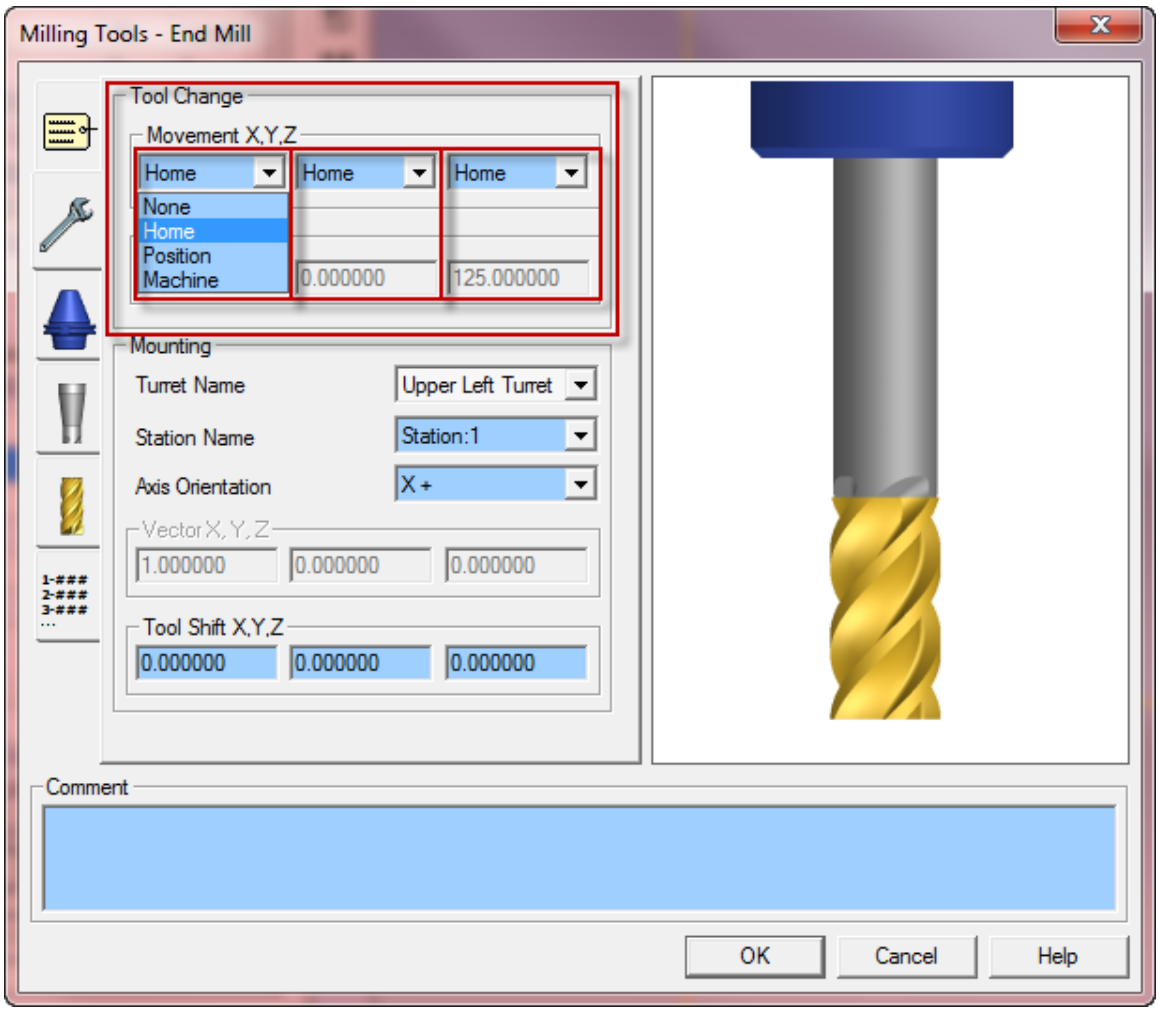
5.2.3. Moves to tool change position optimization

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:

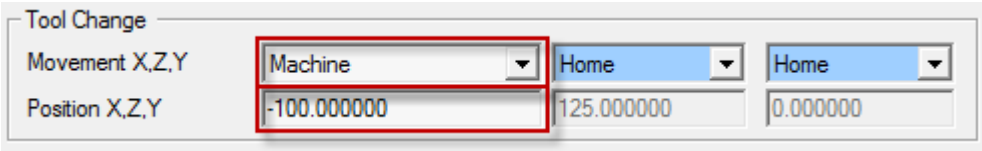


Milling tools:



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.

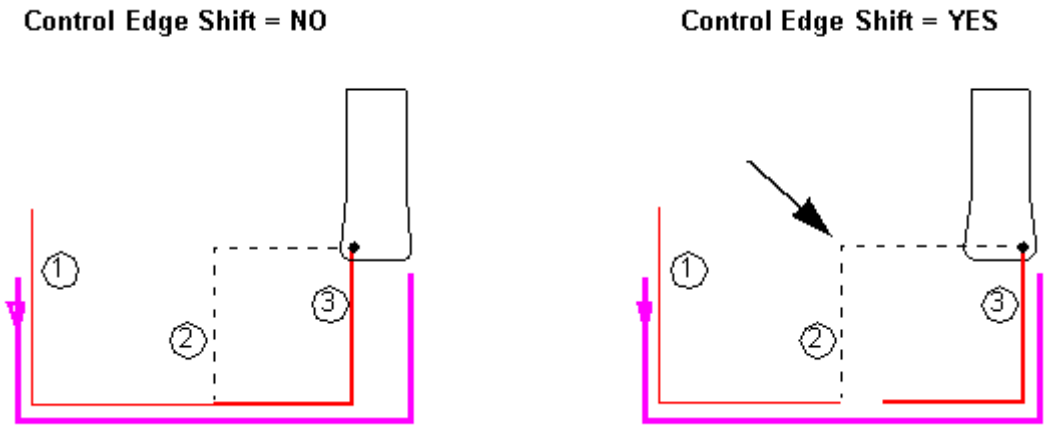
Here is an example of an impossible movement that will trigger the output of an error message: If Moves To Tool Change is set to XYZ and Movement X, Y, Z is set to Home, Machine, None, you will get this error message in the NC code: `ERROR: TOOL CHANGE MOVEMENT NOT DEFINED CORRECTLY FOR TOOL: TURNING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

Notes:

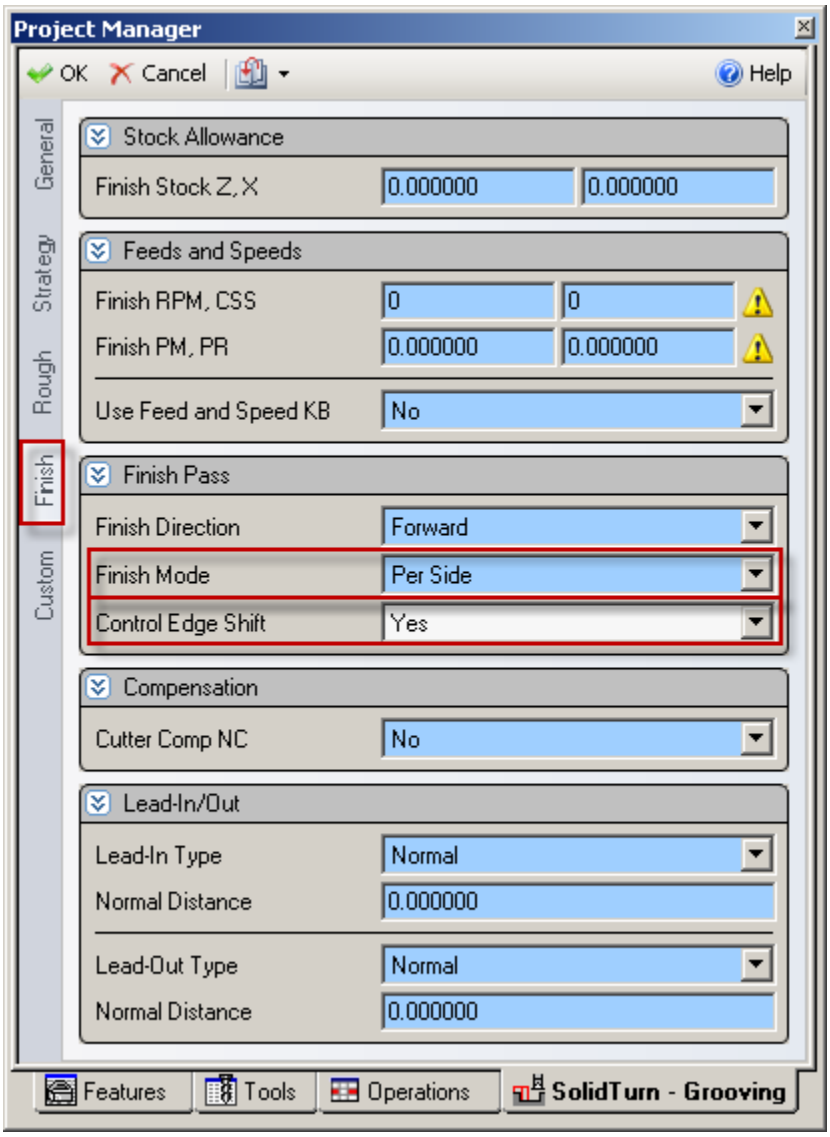
- The very first tool of a program will always be called from home (G28).
- Tool cancel position of current tool will match tool change position of the next tool.
- Movements X, Y, Z set to None, None, None can only be used for dummy tools created for park cycle.
- If your machine configuration does 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.

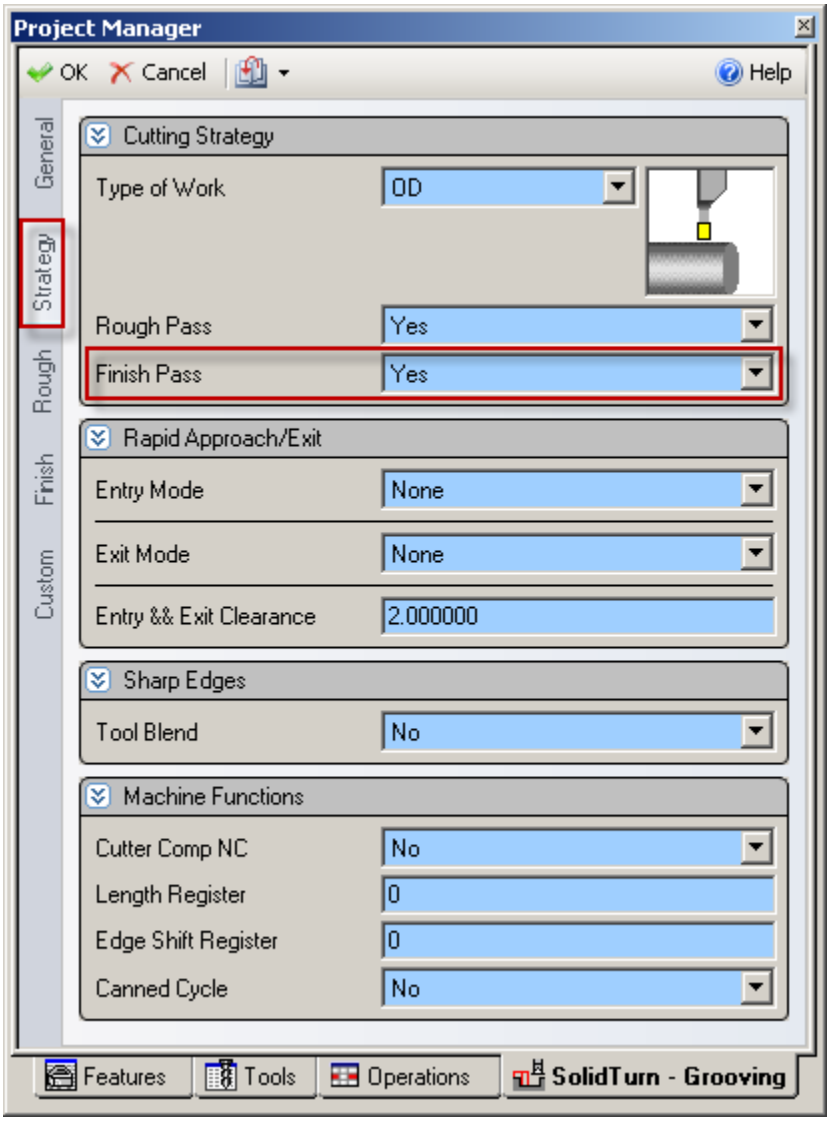
5.3. SolidTurn Grooving: Control Edge Shift

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.



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

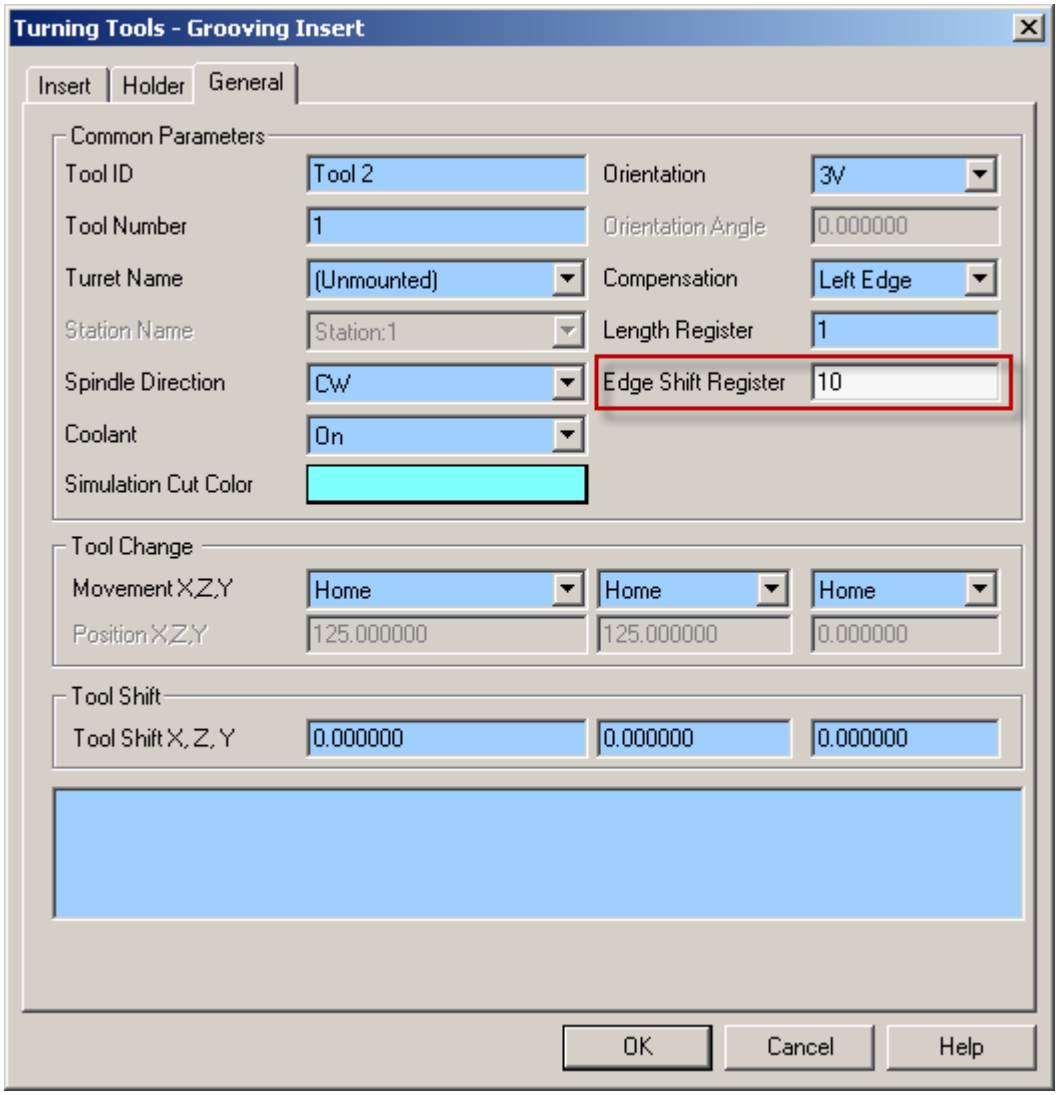

Project Manager
OK
Cancel
Help
General
Strategy
Rough
Finish
Custom
Stock Allowance
Finish Stock Z, X
0.000000
0.000000
Feeds and Speeds
Finish RPM, CSS
0
0
Finish PM, PR
0.000000
0.000000
Use Feed and Speed KB
No
Finish Pass
Finish Direction
Forward
Finish Mode
Per Side
Control Edge Shift
Yes
Compensation
Cutter Comp NC
No
Lead-In/Out
Lead-In Type
Normal
Normal Distance
0.000000
Lead-Out Type
Normal
Normal Distance
0.000000
Features
Tools
Operations
SolidTurn - Grooving



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-ST configuration:

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

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

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

Tool wear offset range of values for NZXDL configuration:

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

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

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

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

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

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

With the 133 option:

- **Head 1:** The entered Length Register number has to be contained **between 1 and 133**.
- **Head 2 machining on spindle 1 side:** The entered Length Register number has to be contained **between 1 and 67**.
- **Head 2 machining on spindle 2 side:** The entered Length Register number has to be contained **between 1 and 67**, 66 will be added to this number.
- **Head 3:** The entered Length Register number has to be contained **between 1 and 133**.

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

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

With the 133 option:

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

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

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

With the 133 option:

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

With the 200 option:

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

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

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

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

With the 99 option:

- **Head 1:** The entered Edge Shift Register number has to be contained **between 1 and 99**.
- **Head 1 with tool life management on:** The entered Edge Shift Register number has to be contained **between 1 and 99**.
- **Head 2 machining on spindle 1 side:** The entered Edge Shift Register number has to be contained **between 1 and 49**.
- **Head 2 machining on spindle 2 side:** The entered Edge Shift Register number has to be contained **between 1 and 49**, 50 will be added to this number.
- **Head 2 with tool life management on:** The entered Edge Shift Register number has to be contained **between 1 and 99**.
- **Head 3:** The entered Edge Shift Register number has to be contained **between 1 and 99**.
- **Head 3 with tool life management on:** The entered Edge Shift Register number has to be contained **between 1 and 99**.

With the 133 option:

- **Head 1:** The entered Edge Shift Register number has to be contained **between 1 and 133**.
- **Head 1 with tool life management on:** The entered Edge Shift Register number has to be contained **between 1 and 133**.
- **Head 2 machining on spindle 1 side:** The entered Edge Shift Register number has to be contained **between 1 and 67**.
- **Head 2 machining on spindle 2 side:** The entered Edge Shift Register number has to be contained **between 1 and 67**, 66 will be added to this number.
- **Head 2 with tool life management on:** The entered Edge Shift Register number has to be contained **between 1 and 133**.

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

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

With the 99 option:

- **Head 1 machining on spindle 1 side:** The entered Edge Shift Register number has to be contained **between 1 and 49**.
- **Head 1 machining on spindle 2 side:** The entered Edge Shift Register number has to be contained **between 1 and 49**, 50 will be added to this number.
- **Head 1 with tool life management on:** The entered Edge Shift Register number has to be contained **between 1 and 99**.
- **Head 2 machining on spindle 1 side:** The entered Edge Shift Register number has to be contained **between 1 and 49**.
- **Head 2 machining on spindle 2 side:** The entered Edge Shift Register number has to be contained **between 1 and 49**, 50 will be added to this number.
- **Head 2 with tool life management on:** The entered Edge Shift Register number has to be contained **between 1 and 99**.

With the 133 option:

- **Head 1 machining on spindle 1 side:** The entered Edge Shift Register number has to be contained **between 1 and 67**.
- **Head 1 machining on spindle 2 side:** The entered Edge Shift Register number has to be contained **between 1 and 67**, 66 will be added to this number.
- **Head 1 with tool life management on:** The entered Edge Shift Register number has to be contained **between 1 and 133**.
- **Head 2 machining on spindle 1 side:** The entered Edge Shift Register number has to be contained **between 1 and 67**.
- **Head 2 machining on spindle 2 side:** The entered Edge Shift Register number has to be contained **between 1 and 67**, 66 will be added to this number.
- **Head 2 with tool life management on:** The entered Edge Shift Register number has to be contained **between 1 and 133**.

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

With the 99 option:

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

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

With the 133 option:

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

With the 200 option:

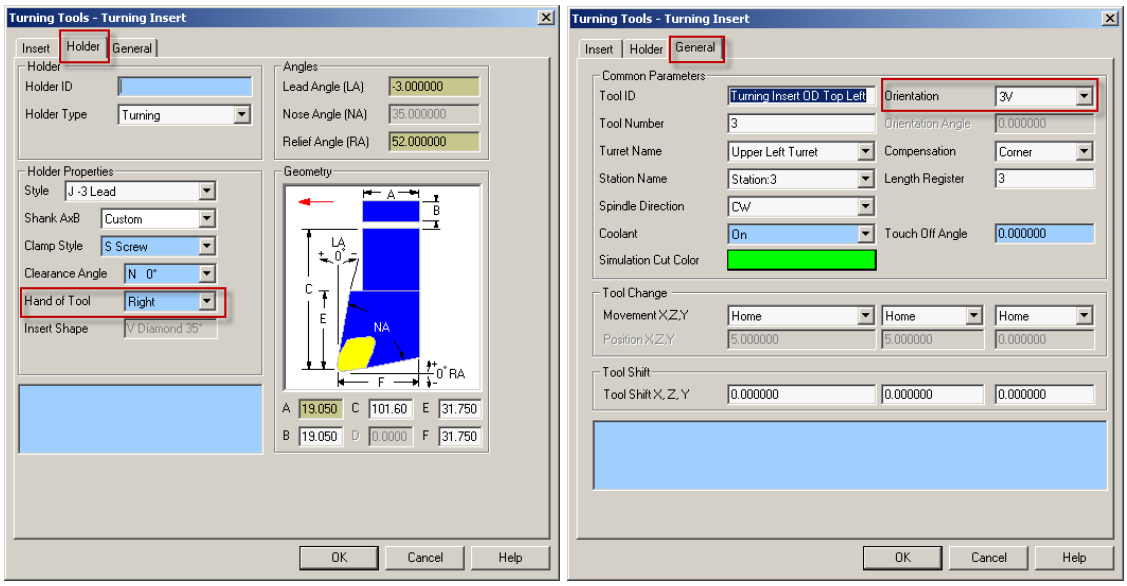
- **Head 1:** The entered Edge Shift Register number has to be contained **between 1 and 200**.
- **Head 1 with tool life management on:** The entered Edge Shift Register number has to be contained **between 1 and 200**.
- **Head 2:** The entered Edge Shift Register number has to be contained **between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

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

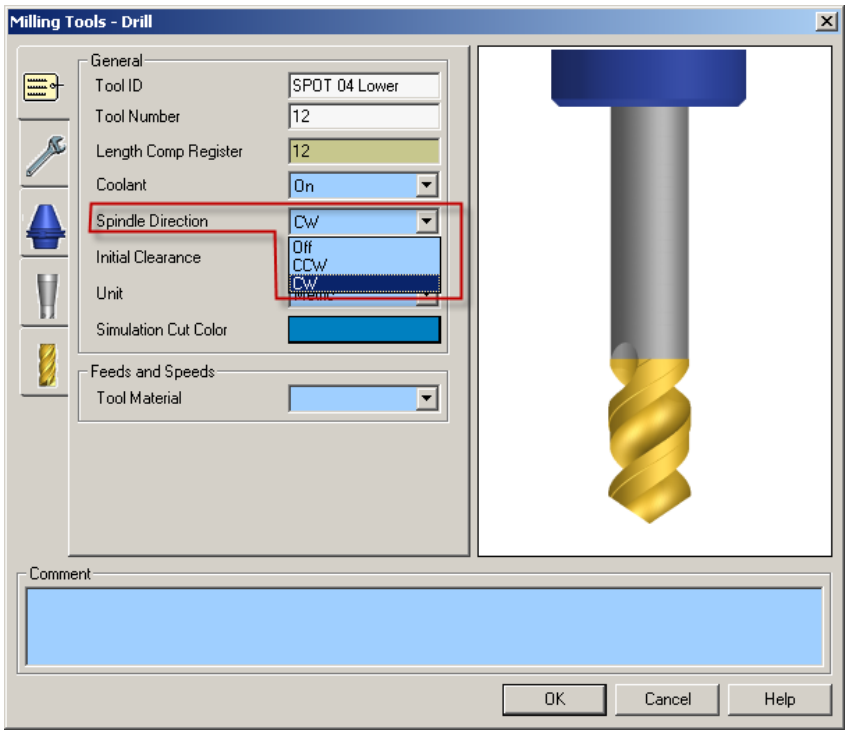
6. Spindle direction, speed (S function) and feedrate (F function)

6.1. Turning spindle direction and milling tool spindle direction

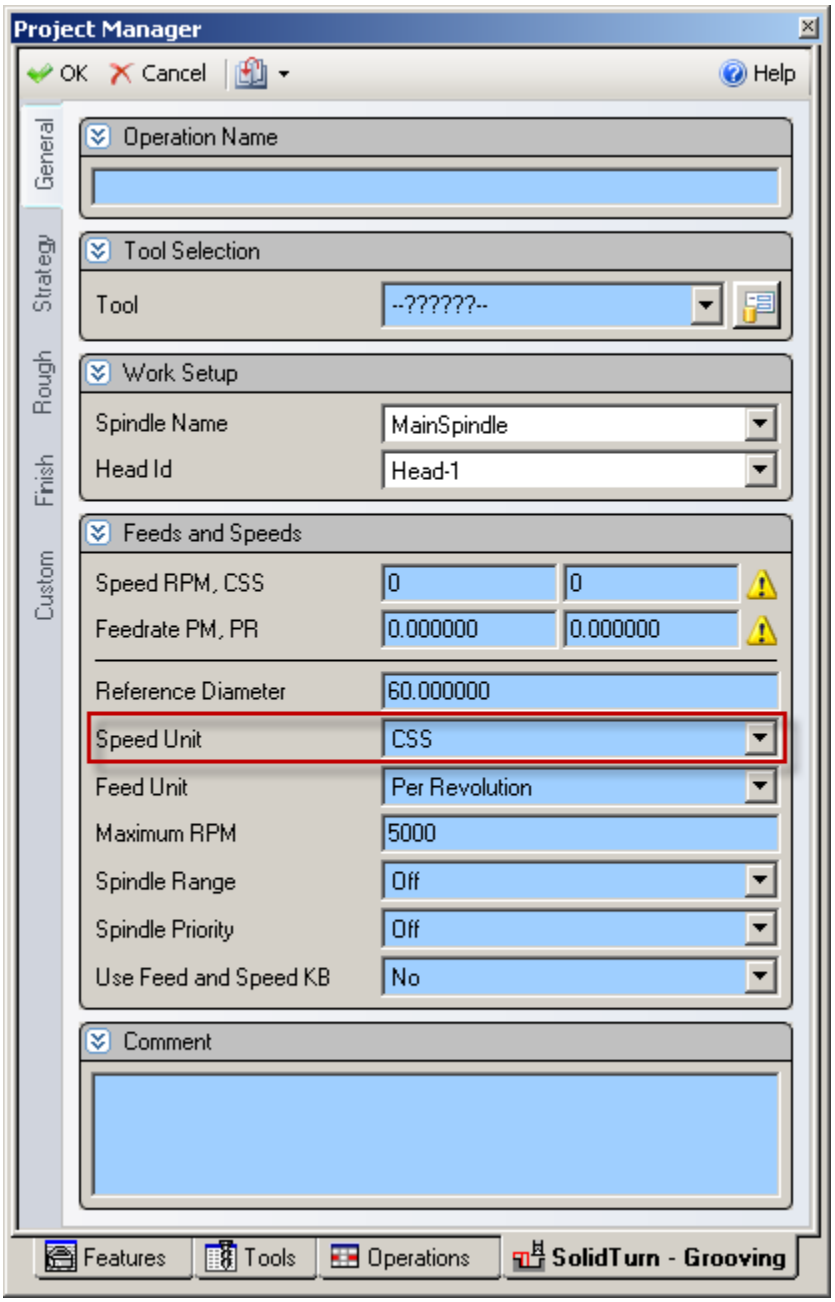
For **turning operations** using a cutting insert (**Roughing, Balanced Roughing, Contouring, Grooving and Threading**), the spindle direction will automatically be computed by the post processors and will output the correct M-code (M03, M04, M203 or M204) based on the hand of the tool on the holder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



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. Spindle speed output for CSS unit



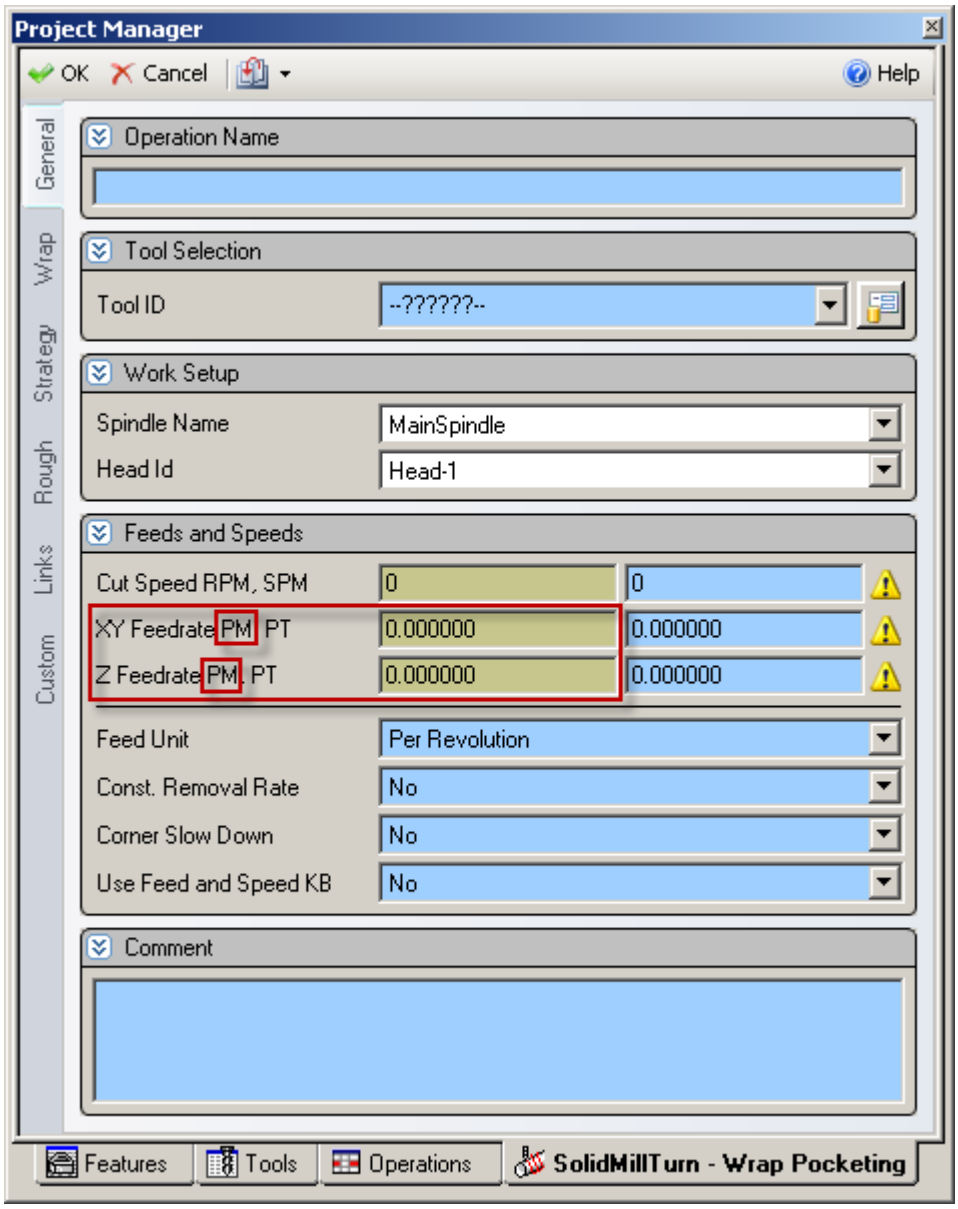
On a turning operation page, if you choose to output the turning spindle speed in the unit CSS (Constant Surface Speed), a warm up speed will first be output with the G97 code before the first positioning move. The speed value is computed at the first diameter from where the CSS command will be turned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

6.3. Feedrate for 4-axis milling operations

For Wrap Pocketing and Wrap Contouring operations with Cylindrical Interpolation set to No, Rotary Face Pocketing and Rotary Face Contouring operations with Polar Interpolation set to No or 5-axis operations with 5th axis locked with Z-axis (5-axis operation becomes a 4-axis operation), a rotary feedrate will be computed by the post processor, based on the linear and rotary (C-axis) moves of the cut.

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

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

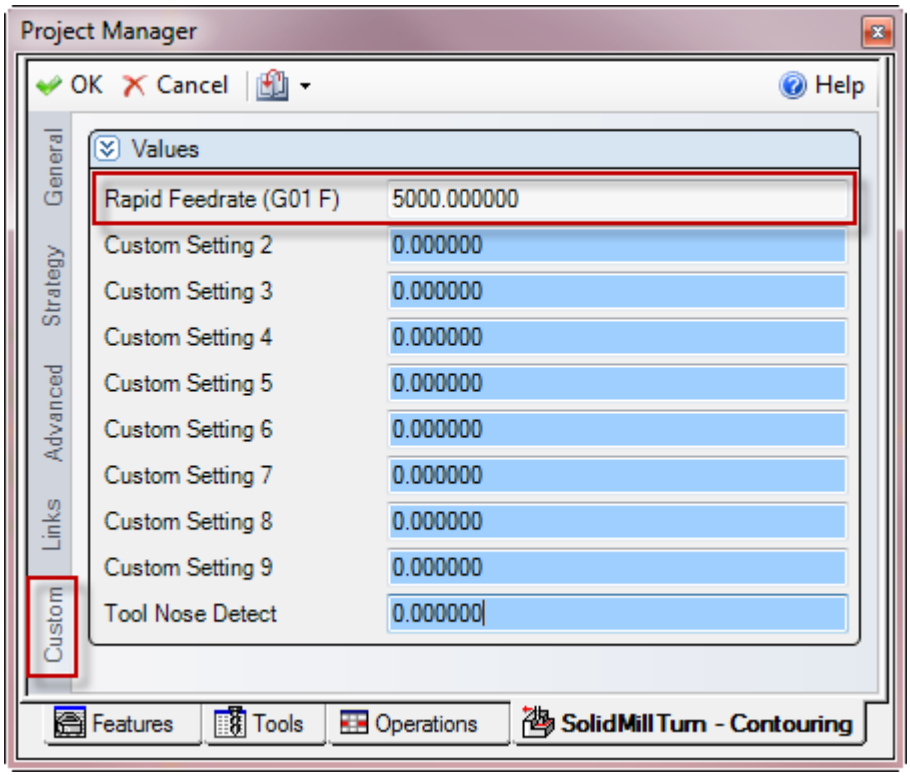


6.4. Output of rapid positioning moves (G00) with linear interpolation moves (G01 F)

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

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

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



7. How to output coolant codes

7.1. Introduction

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

For NZX-ST & NZX-S configurations:

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

For NZXDL configuration:

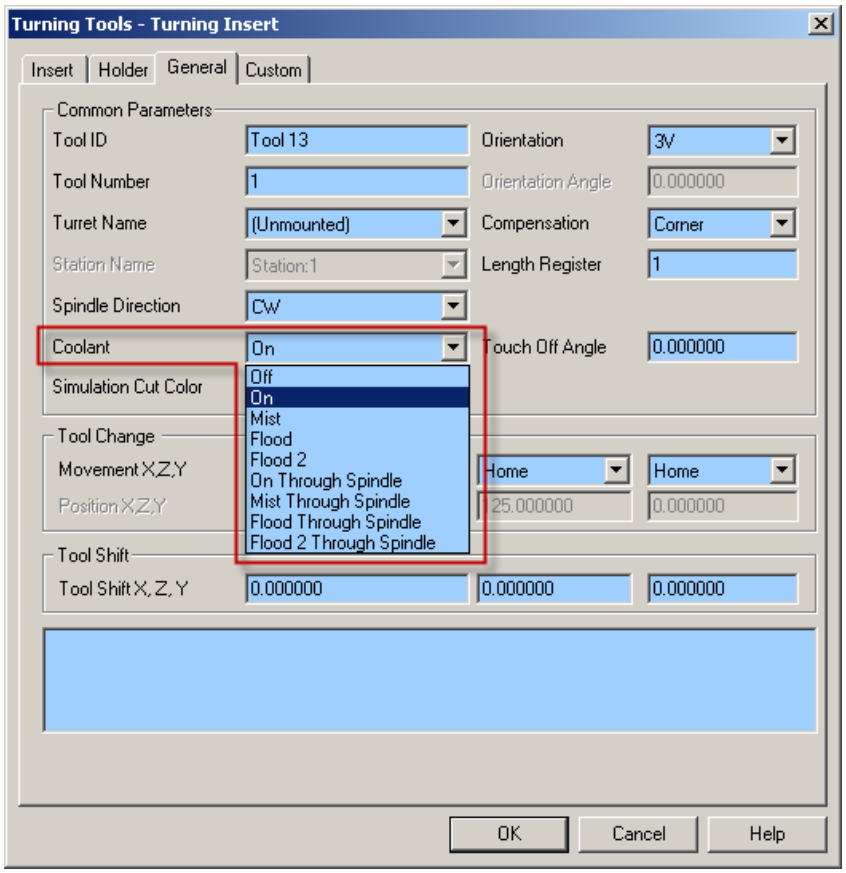
Code	Function
M08	Coolant ON
M09	Coolant OFF
M382	Shower coolant ON
M383	Shower coolant OFF
M478	Through-spindle coolant ON
M479	Through-spindle coolant OFF
M621	Super high pressure coolant ON
M622	Super high pressure coolant OFF
M651	Chuck top coolant ON
M652	Chuck top coolant OFF

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

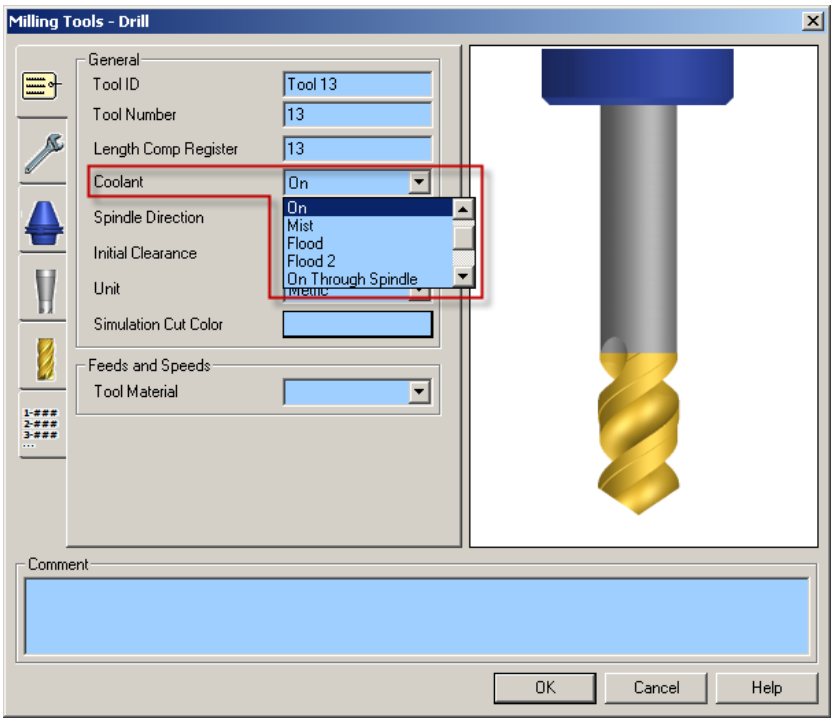
7.2. Set the first coolant code

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

Turning tool page:



Milling tool page:



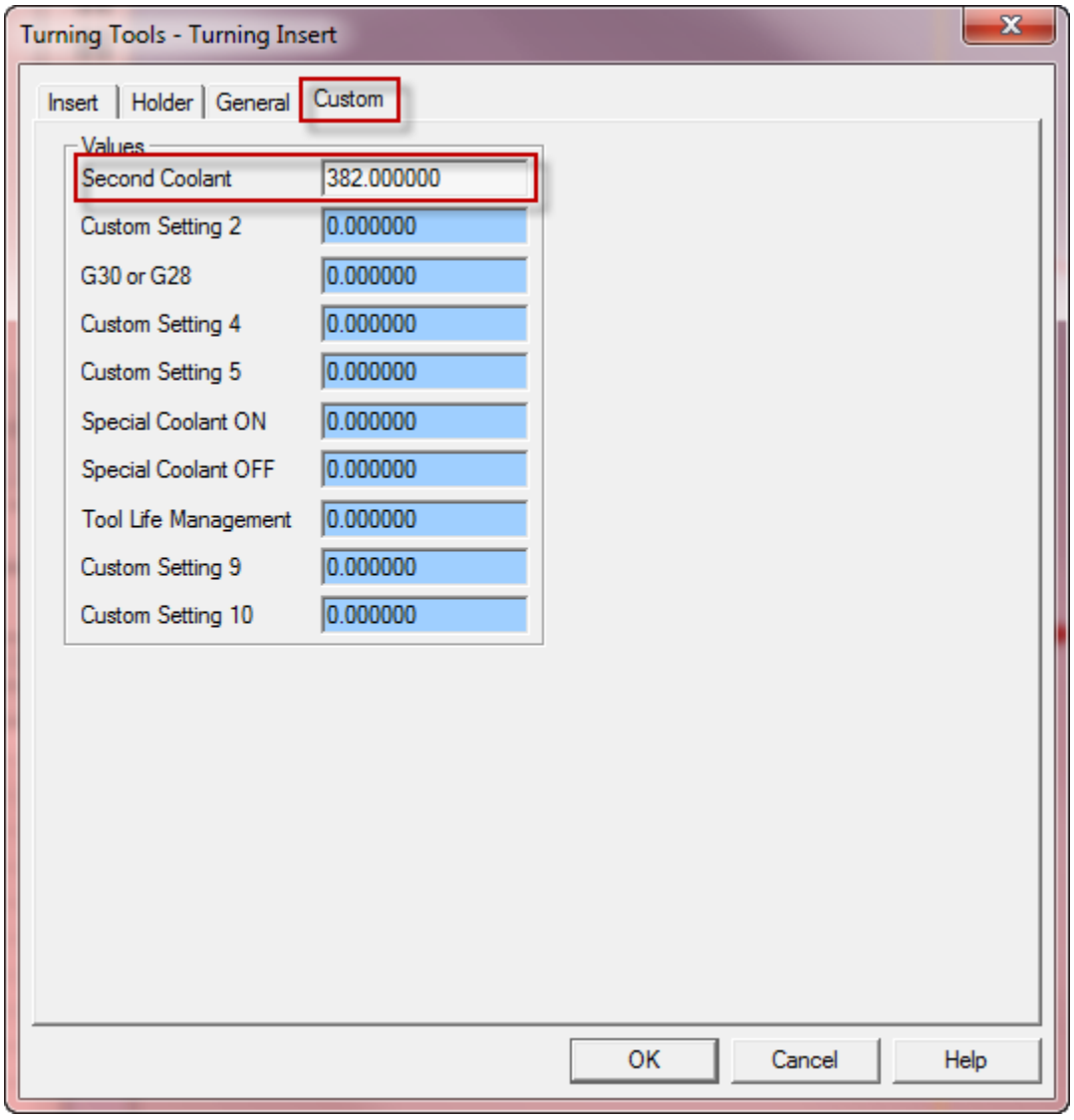
Set **Coolant** to:

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

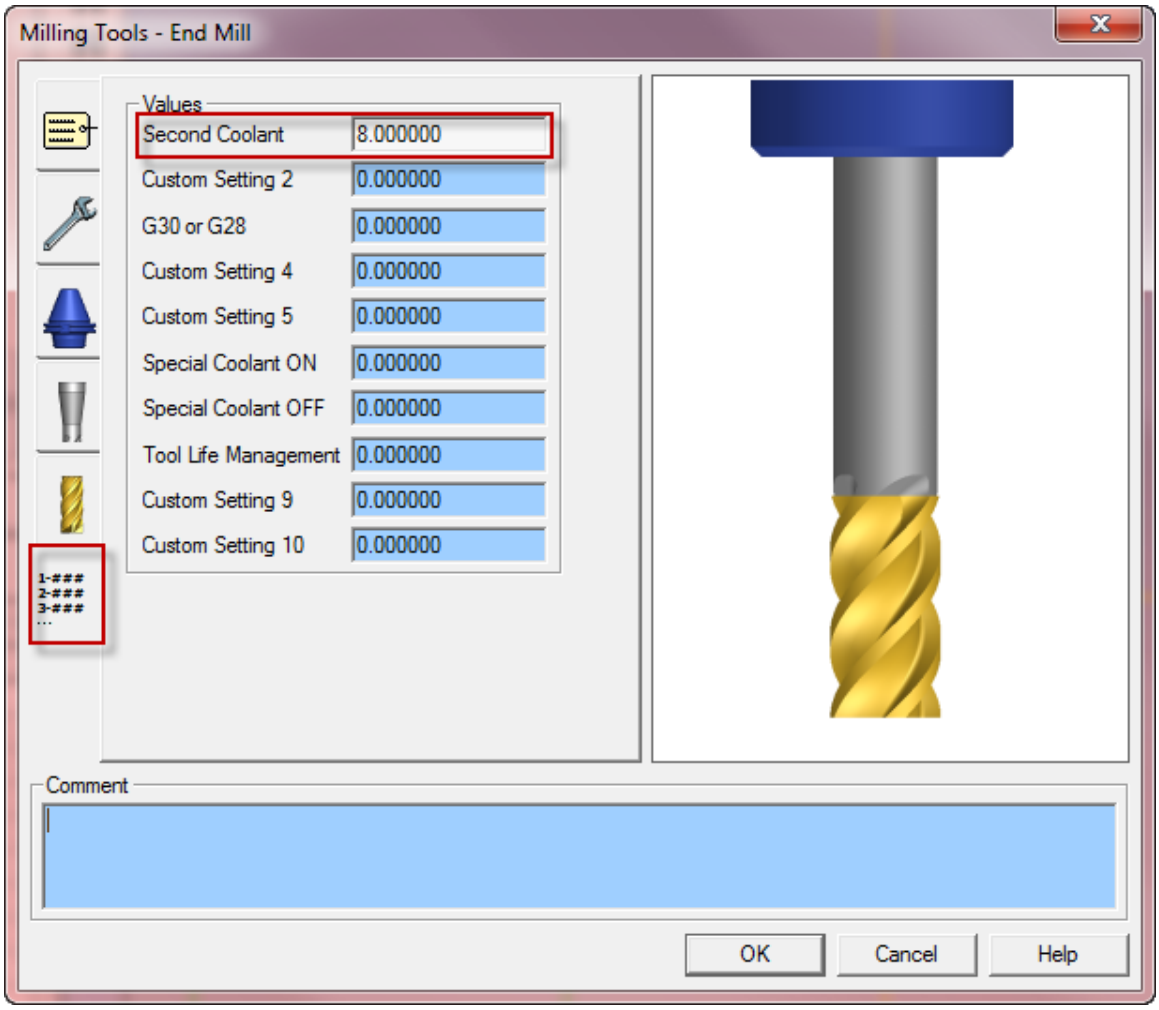
7.3. Set the second coolant code (optional)

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

Turning tool page:



Milling tool page:



Set **Second Coolant** to:

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

7.4. Set a special coolant code (optional)

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

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

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

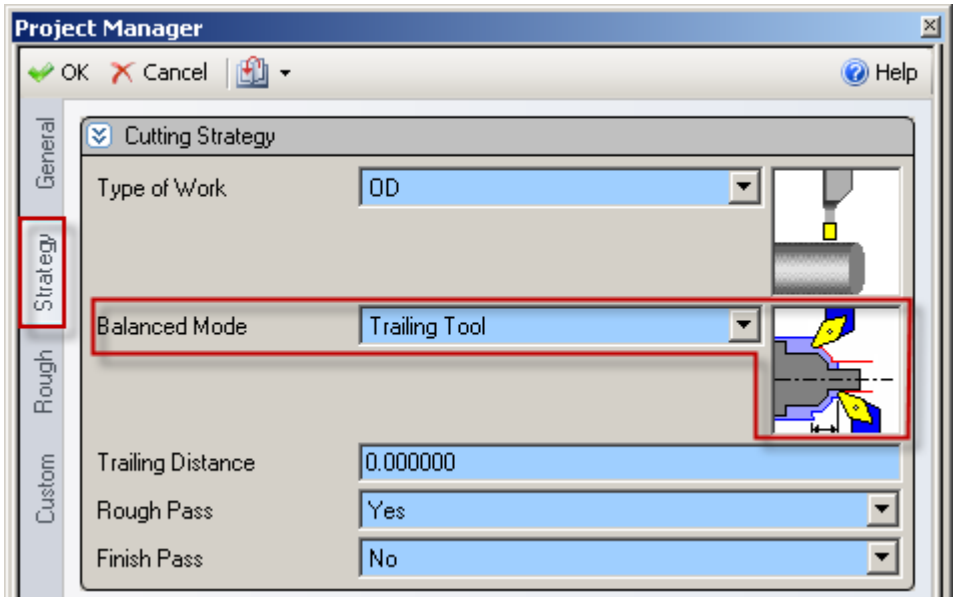
8. Simultaneous machining (for NZX-ST & NZX-S configurations only)

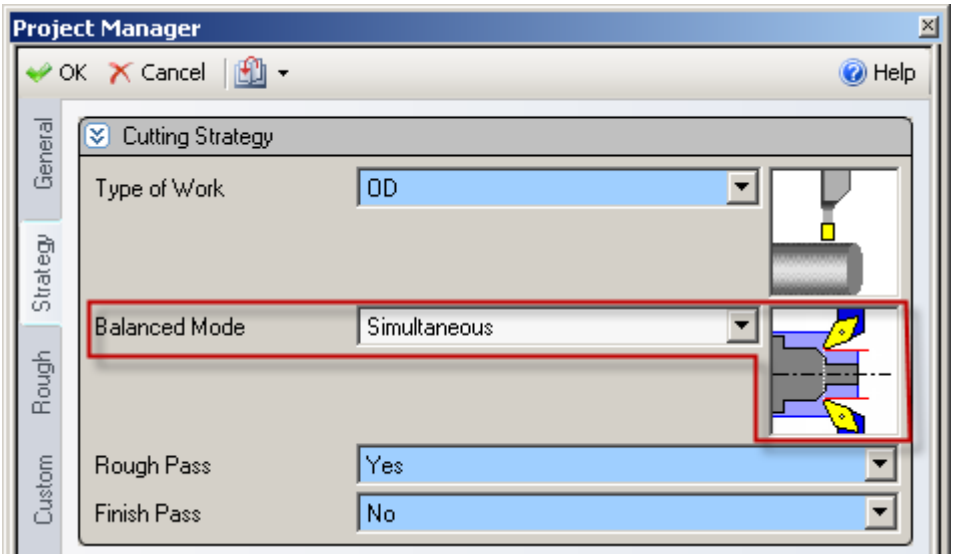
8.1. Simultaneous turning operations: Turning on the same spindle with 2 different turrets

8.1.1. Balanced roughing cycle

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

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





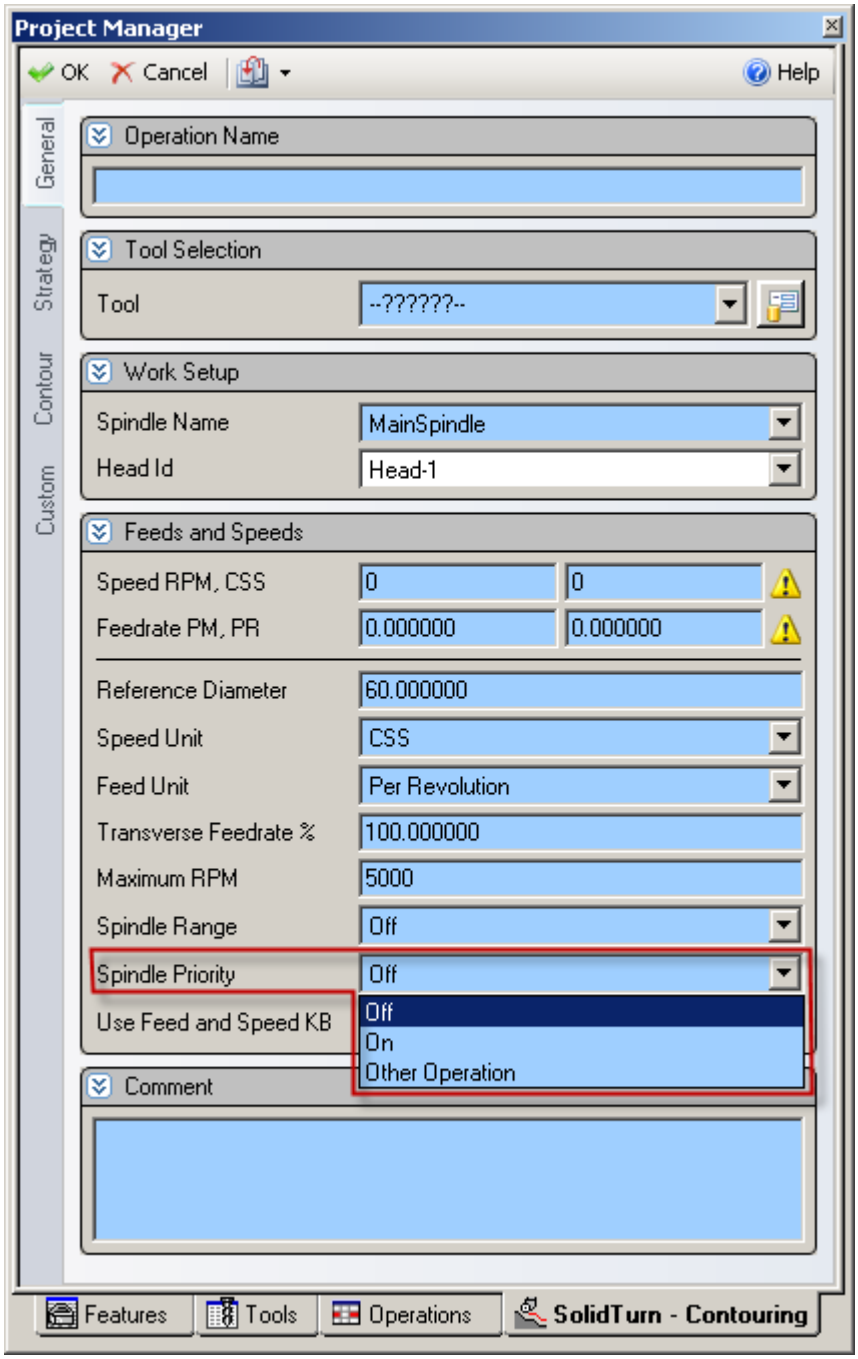
For the Trailing Tool balanced mode, a wait code (soft sync) will be output in between each pass. For the Simultaneous balanced mode, the Balanced Cut Mode (G68) will be turned on on both involved heads and will be canceled at the end of the cut by the Balanced Cut Mode Cancel code (G69).
The spindle speed will only be output in channel 1 for head 1 / head 2 balanced turning operation (or in channel 3 for head 2 / head 3 balanced turning operation for NZX-ST configuration), not in channel 2.

8.1.2. Other turning cycles

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

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

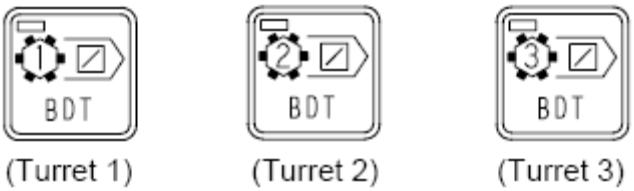
The head controlling the spindle and the head "slave" are set on the operation page of turning operations, using the Spindle Priority setting:



Set Spindle Priority to:

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

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



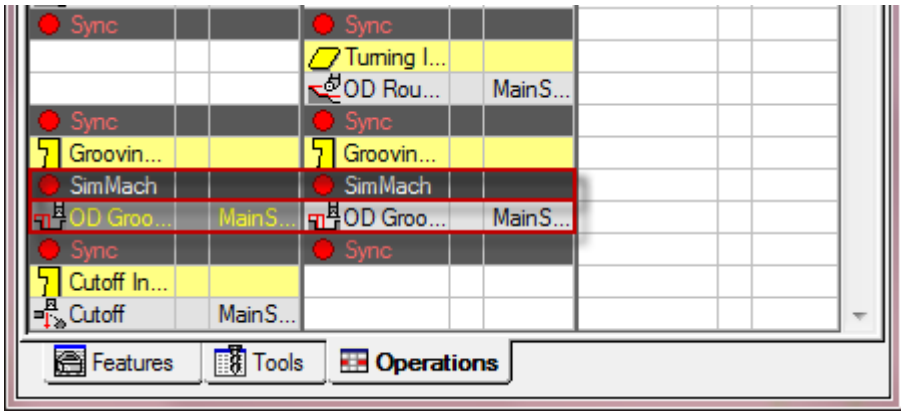
(Turret 1) (Turret 2) (Turret 3)

8.1.3. Other turning cycles in Balanced Cut Mode

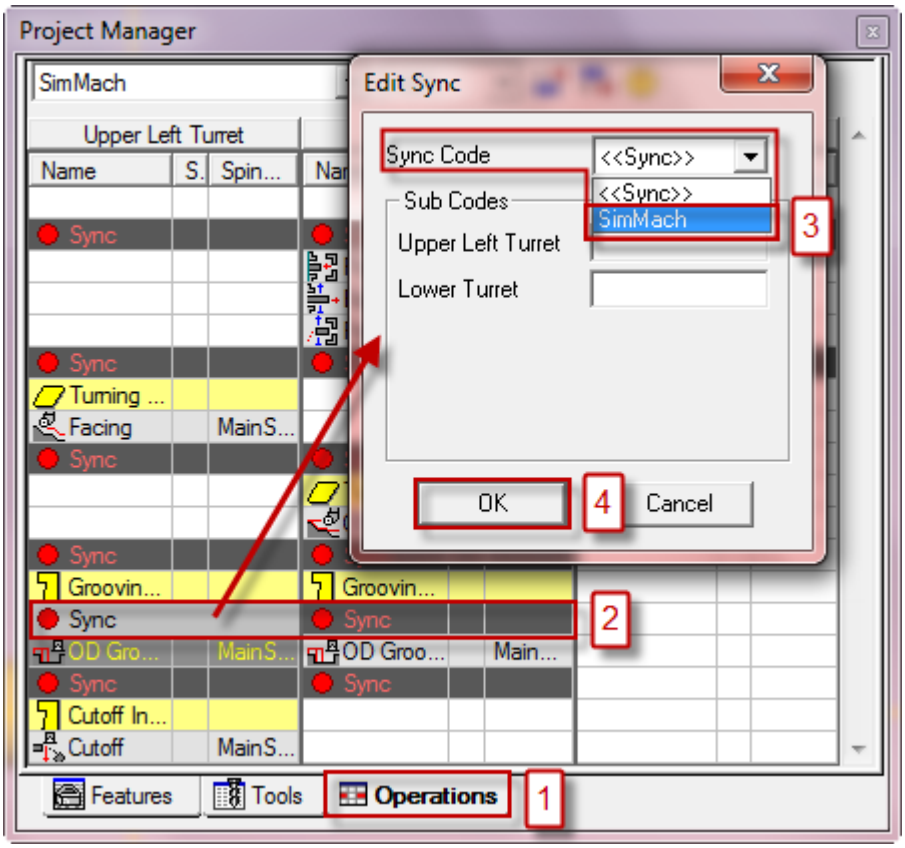
CAUTION !!! Do not use with **SolidTurn Drilling**.

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

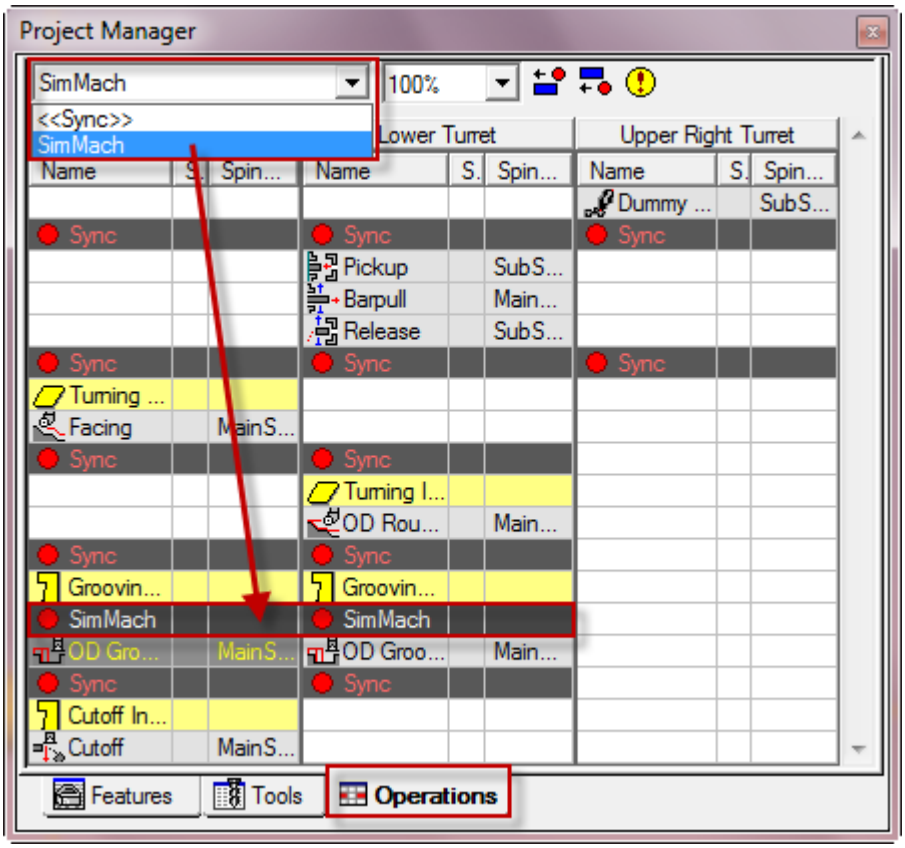
This functionality is typically used to perform pinch grooving operations.



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

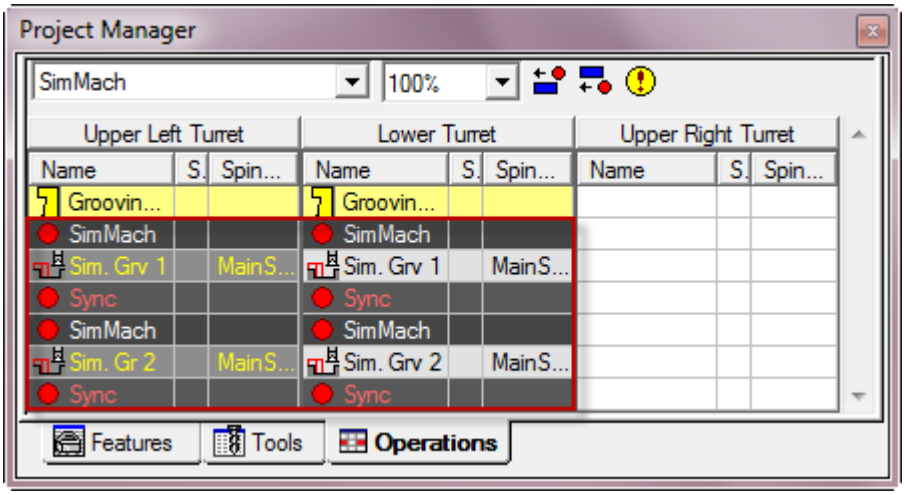


Or, you can also directly select SimMach from the pull-down on top of the Operations tab of the Project Manager. Once selected, simply program your sync: it will be a SimMach Sync Code.



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 **8.1.2. Other turning cycles**.

Notes:

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

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

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

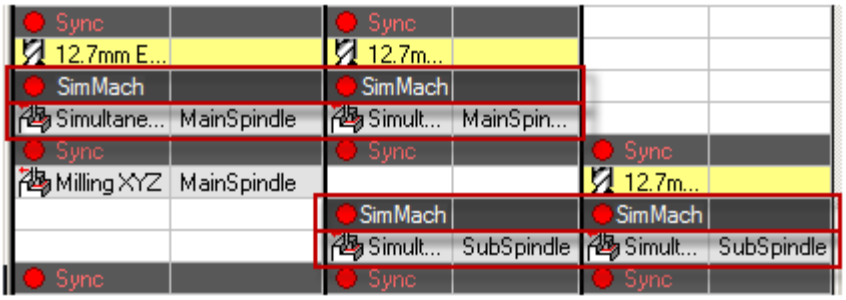
Simultaneous milling works for C-axis indexing milling operations and not 4-axis wrap or rotary milling operations.

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

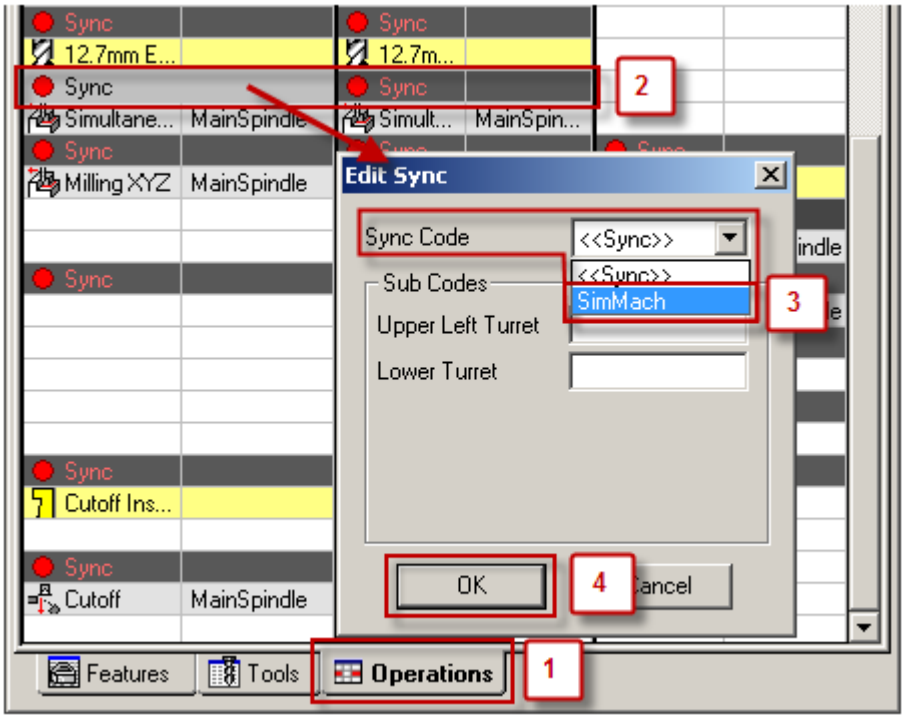
Turret 2 and turret 3 can simultaneously mill on spindle 2 side. In this case, the C-axis is controlled by head 3.

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

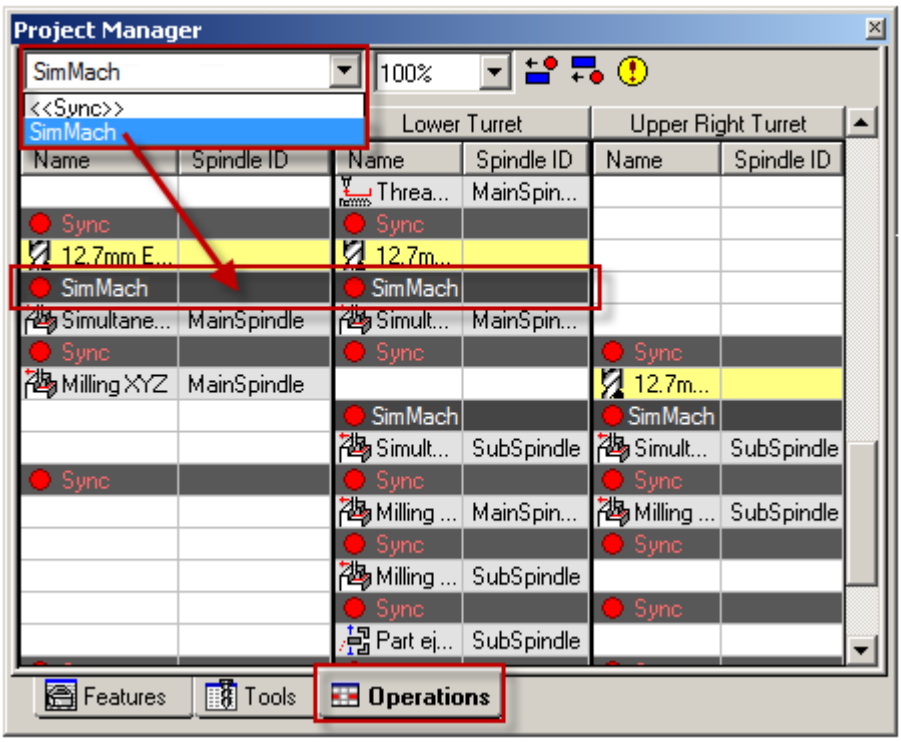
To program simultaneous milling in ESPRIT, you will need to synchronize head 1 & head 2 (or head 2 & head 3 for NZX-ST configuration) prior to the 3-axis milling operations with the **SimMach** Sync Code. It will trigger the simultaneous milling mode.



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

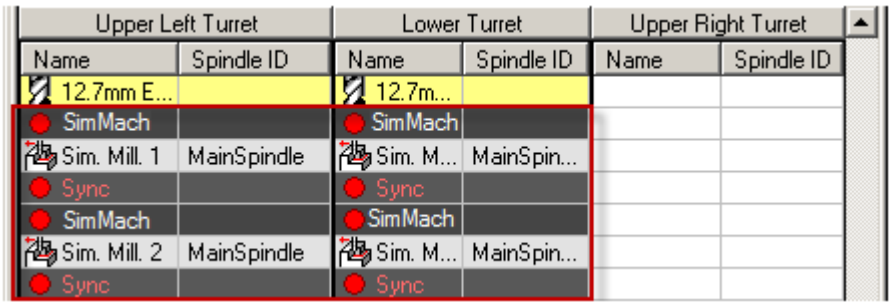


Or, you can also directly select SimMach from the pull-down on top of the Operations tab of the Project Manager. Once selected, simply program your sync: it will be a SimMach Sync Code.



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

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



Notes:

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

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

9. Operation synchronization

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

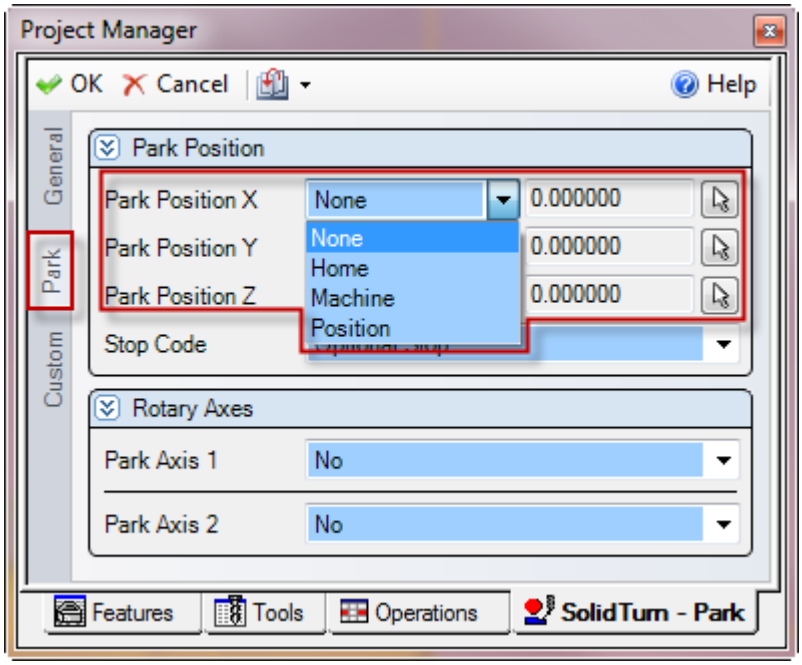
10. Park cycle

10.1. Introduction

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

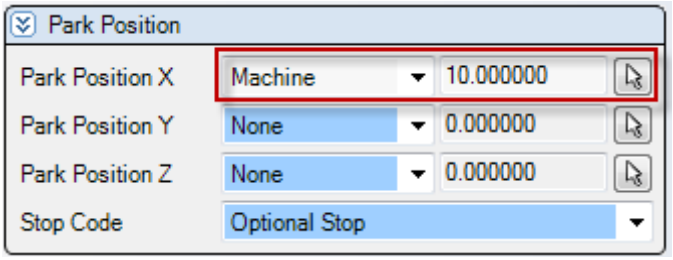
10.2. How to park a tool

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

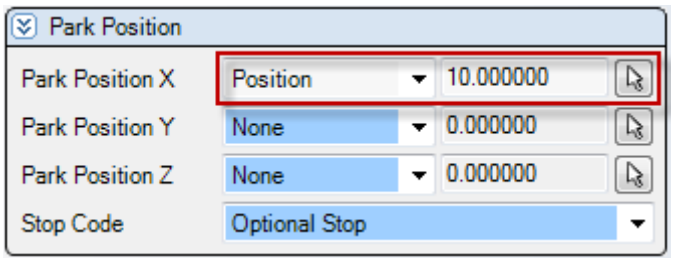


Set Park Position X, Y or Z to:

- **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.

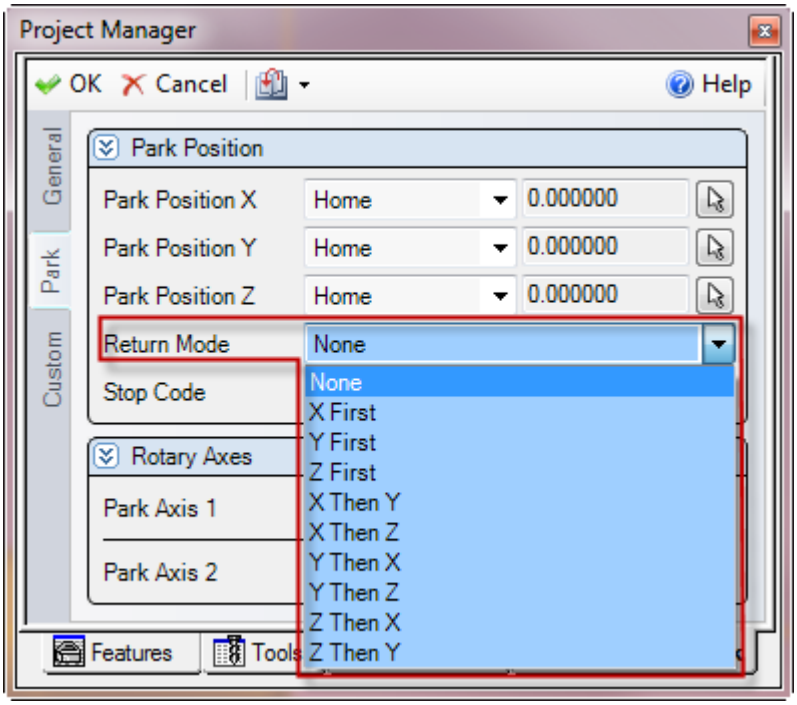


- **Position:** The tool will be parked at the specified position in Position X, Y, Z fields at rapid traverse rate (G00).



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:*

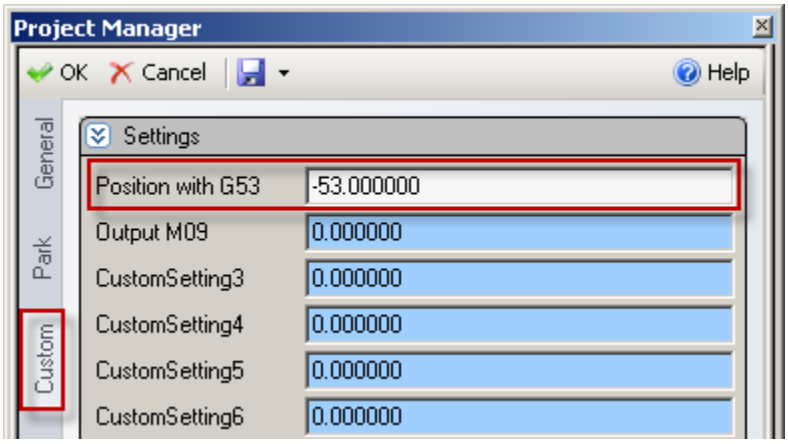
  If Park Position X, Y & Z are all set to Home: all Return Modes become available.

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

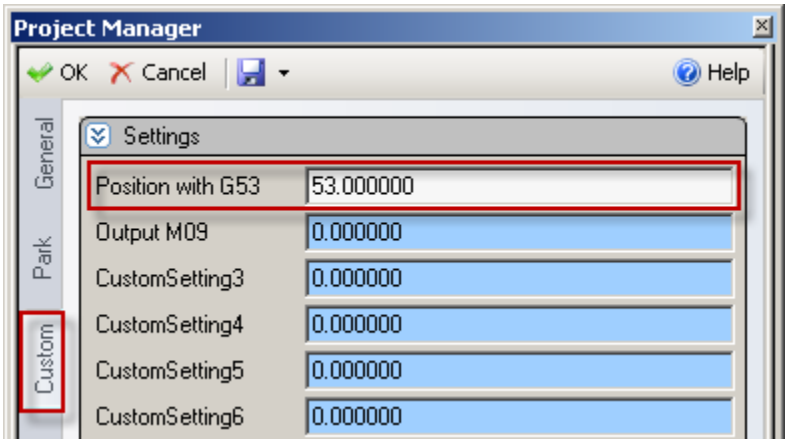
10.3. Park a turret to its maximal travel position (G53)

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-ST configuration:

Upper left turret T1:

Left position: -295 mm
Right position: 95 mm

Lower turret T2:

Left position: -400 mm
Right position: 400 mm

Upper right turret T3:

Left position: -95 mm
Right position: 295 mm

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

Upper turret T1:

Left position: -400 mm
Right position: 400 mm

Lower turret T2:

Left position: -400 mm
Right position: 400 mm

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

Upper left turret T1:

Left position: -280 mm
Right position: 0 mm

Upper right turret T2:

Left position: 0 mm
Right position: 280 mm

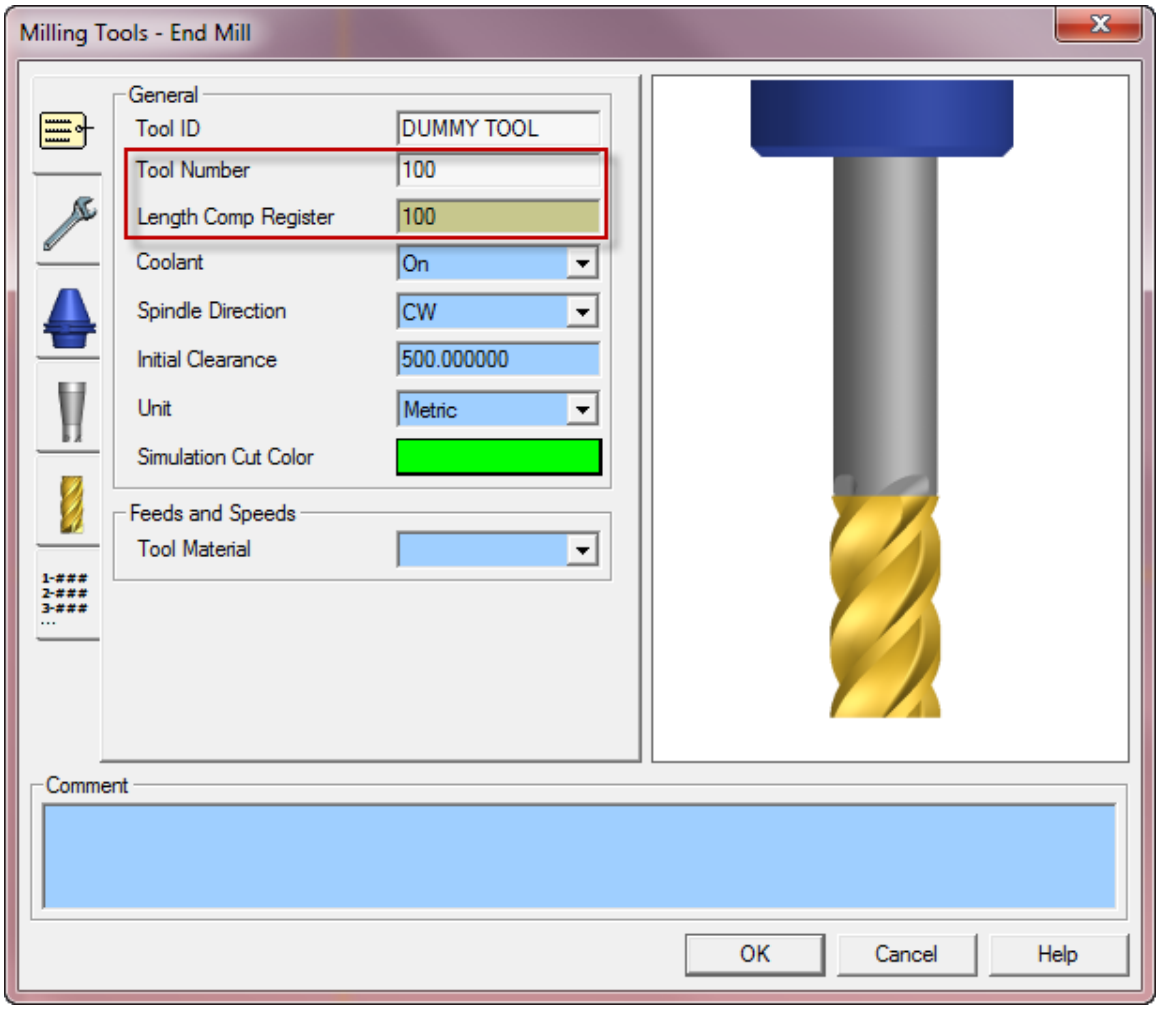
Note that if the tool you are parking is the same as the previous tool in use, the turret will be sent home in Y (Y-axis specifications only) then X before the G53 output as a safety.

10.4. Park a turret prior to a transfer

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

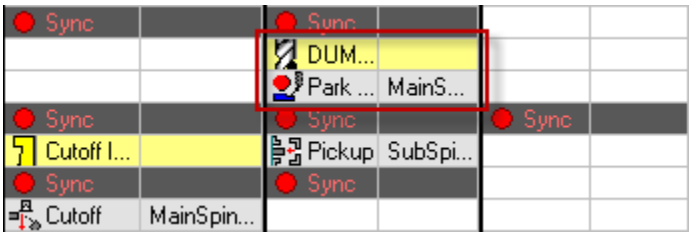
Once correctly setup, you can park turrets using the Park cycle (with G53) prior to a transfer.

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



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

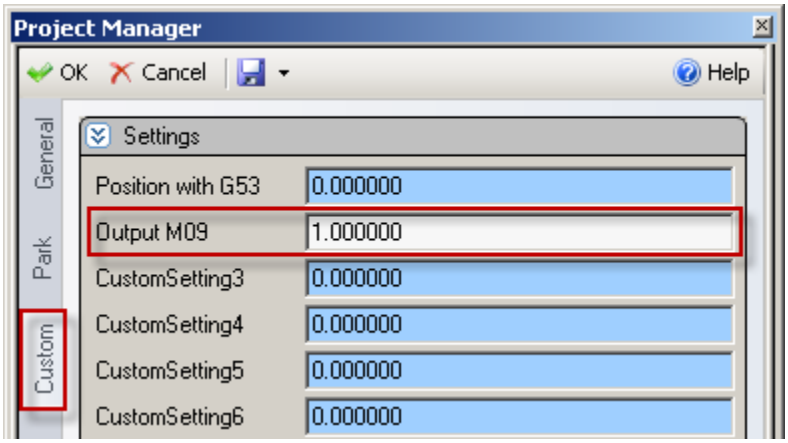
number, you will need to set Position with G53 to 53 (or -53, see part **10.3. Park a turret to its maximal travel position (G53)**) on the Park operation Custom tab. The turret will be parked using G53.



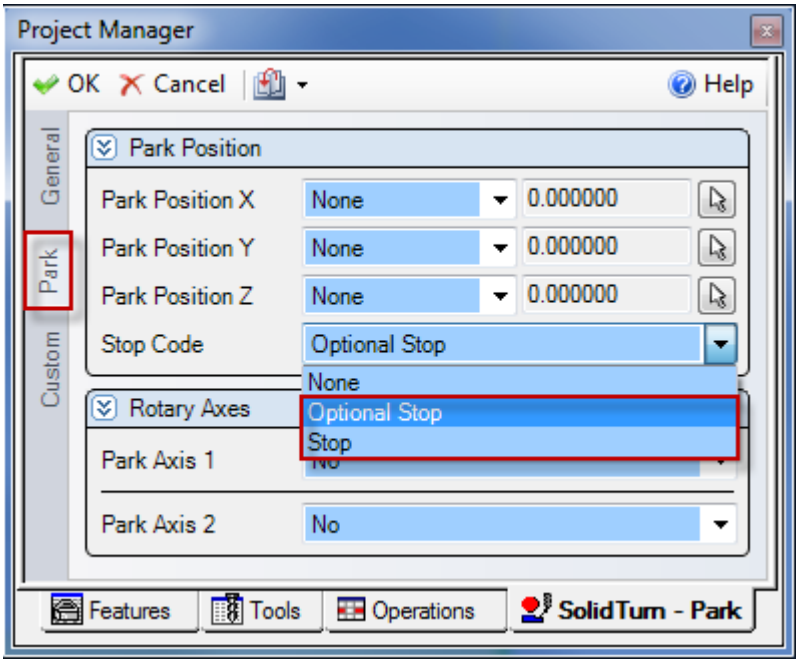
Note: If you omit to enter 53 in the Position with G53 field of the Park operation, you will get the following error message in your NC code: `ERROR: NO TOOL WEAR OFFSET: PARK WITH POSITION (USING G00) CANNOT BE USED`.

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

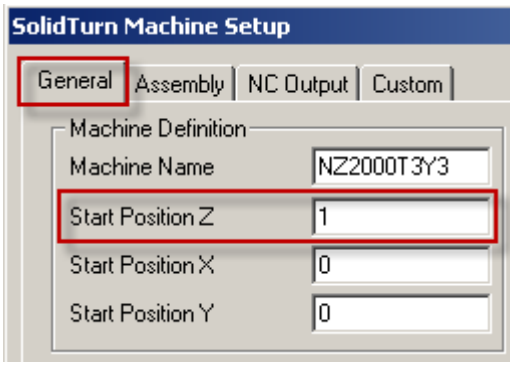
In this following part, you will be given detailed instructions on how to manually program various catching of finished part and various part transfers in ESPRIT. **A correct program in ESPRIT is necessary to have a correct NC code.**
Please note that the following part catching and part transfer scenarios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

11.2. Finished part catching

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

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

Note: On the General tab of the Machine Setup, set the Start Position Z:
Start Position Z = Length of stock for facing the front side of the part



The regular steps to follow are:
1. Machining the part on the spindle 1 side
2. Perform a cut-off operation

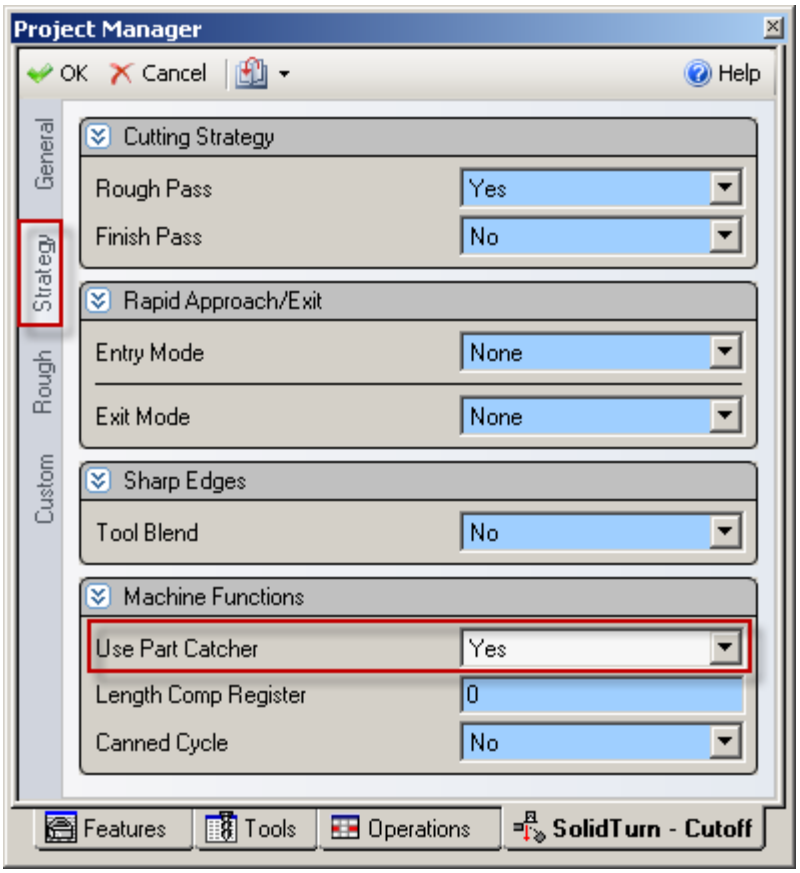
Steps to program this part ejection type in ESPRIT:

1. Machining the part on the spindle 1 side.

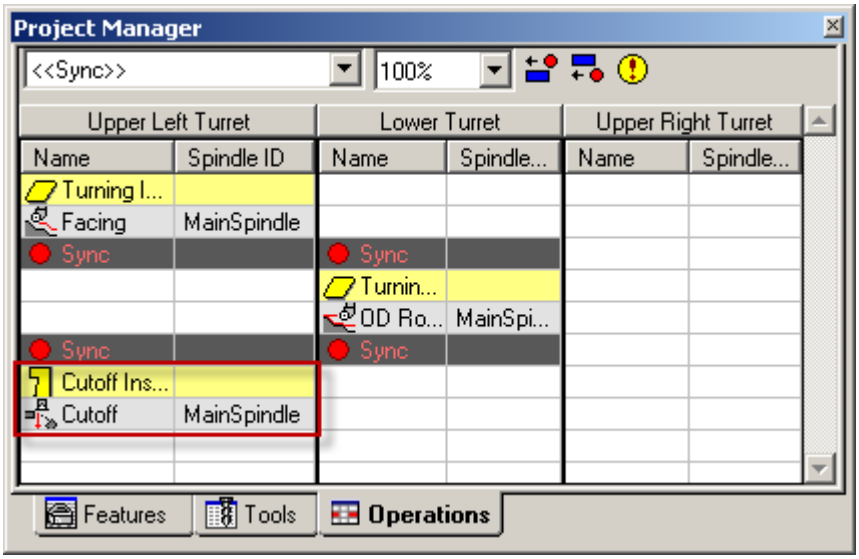
2. Cut-off:
   Use operation **Cutoff** using **Upper** (**Left** for NZX-ST & NZXDL configurations) **Turret** tool on **MainSpindle** (**Spindle Name**).

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

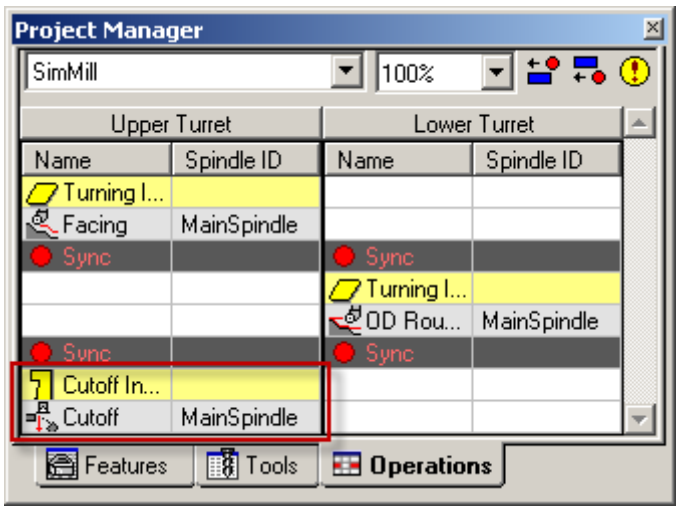
   To trigger the part catching output, set Use Part Catcher to Yes on the Strategy tab.



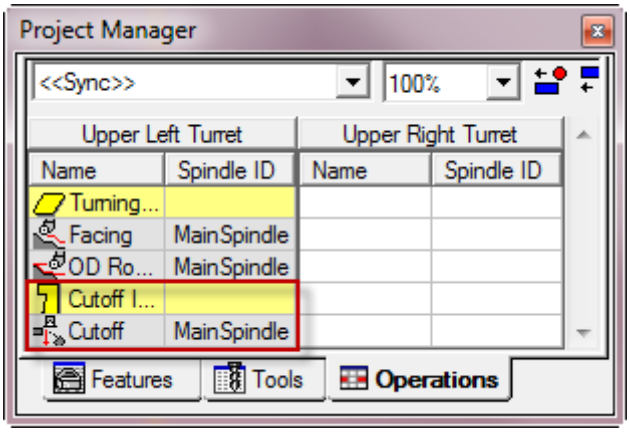
Sample operation list for NZX-ST configuration:



Sample operation list for NZX-S configuration:



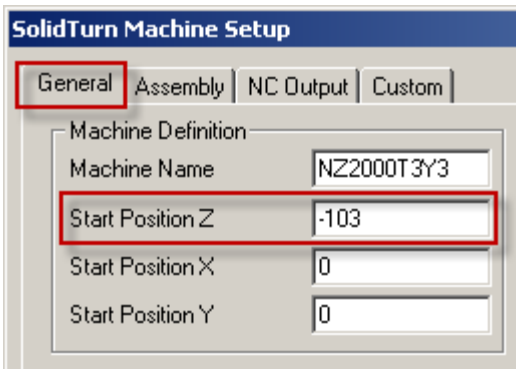
Sample operation list for NZXDL configuration:



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

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

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



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

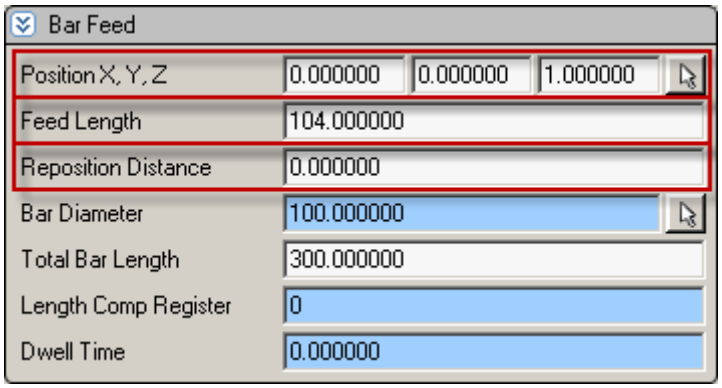
1. Perform a barfeed operation
2. Machining the part on the spindle 1 side
3. Perform a cut-off operation
4. Looping for next parts

Steps to program this part ejection type in ESPRIT:

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

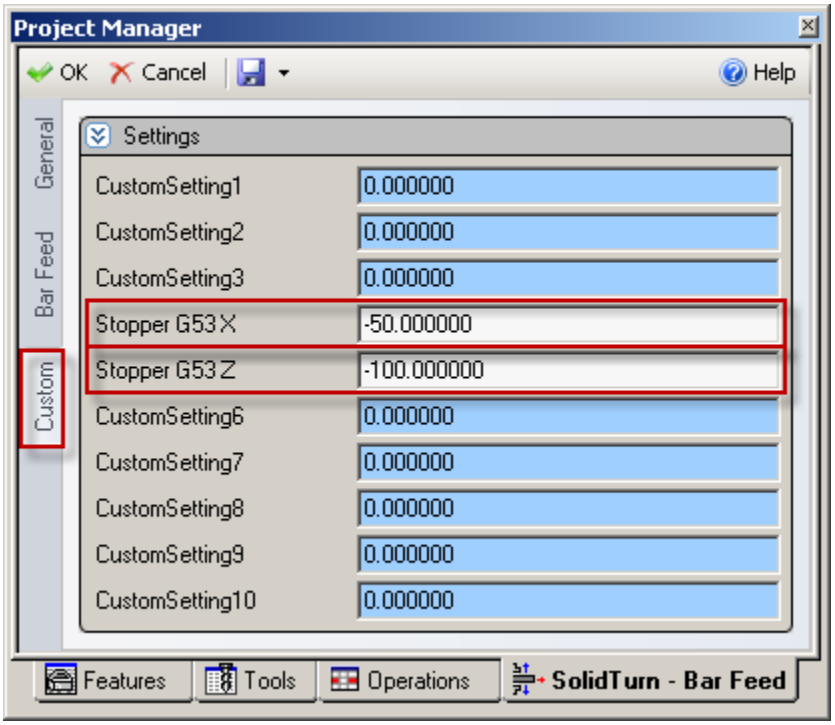
   On Bar Feed tab, set the Feed Length and Reposition Distance:
   Feed Length (=Barfeed distance) = Length of the finished part (Part Stock Length) + Cut-off tool width + Length of stock for facing the front side and, if needed, the back side of the part
   Reposition Distance = 0

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



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

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



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

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

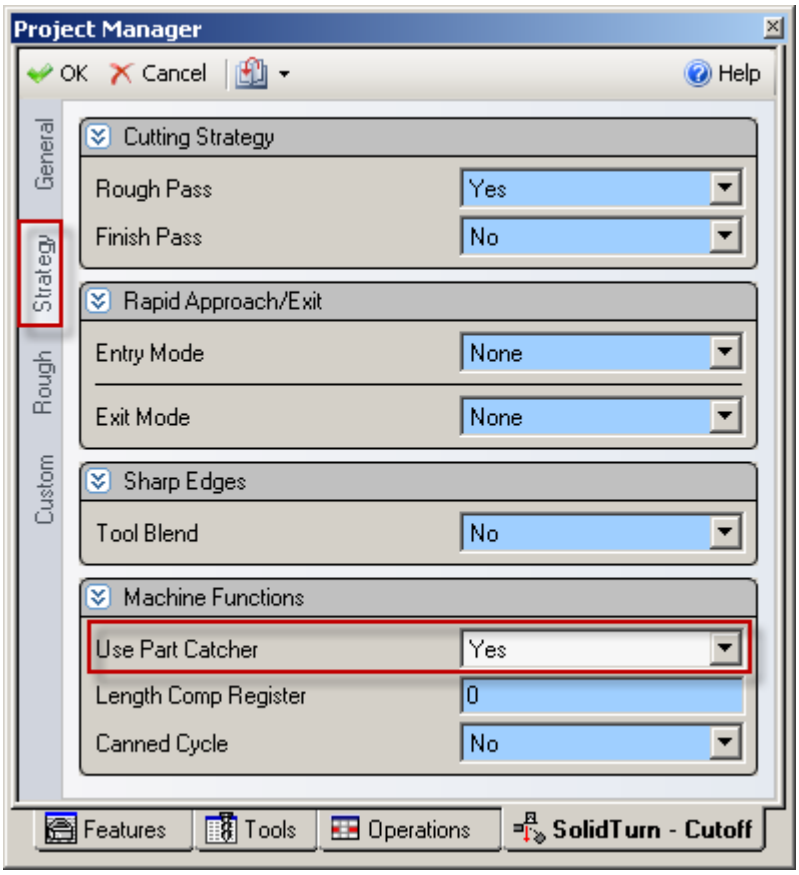
2. Machining the part on the spindle 1 side.

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

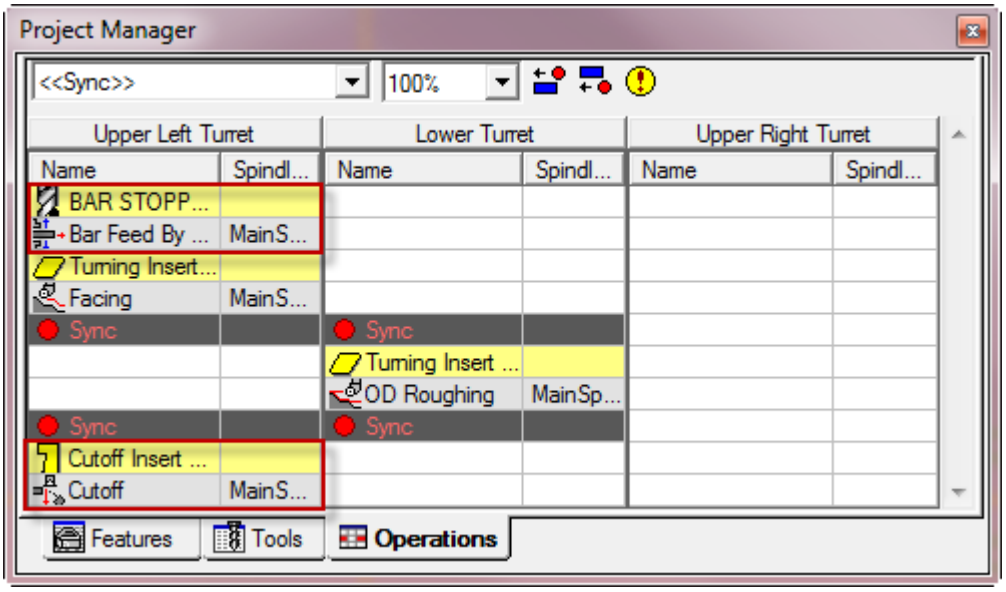
   Cut-off tool is always loaded in the upper left turret for Mori Seiki NZX-ST & NZXDL configurations. The post processors handle cut-off operation

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

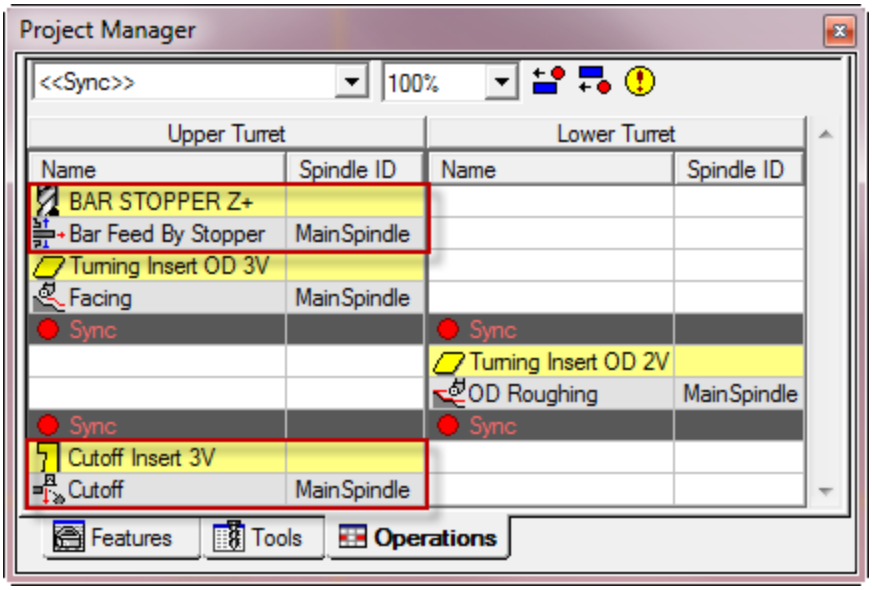
To trigger the part catching output, set Use Part Catcher to Yes on the Strategy tab.



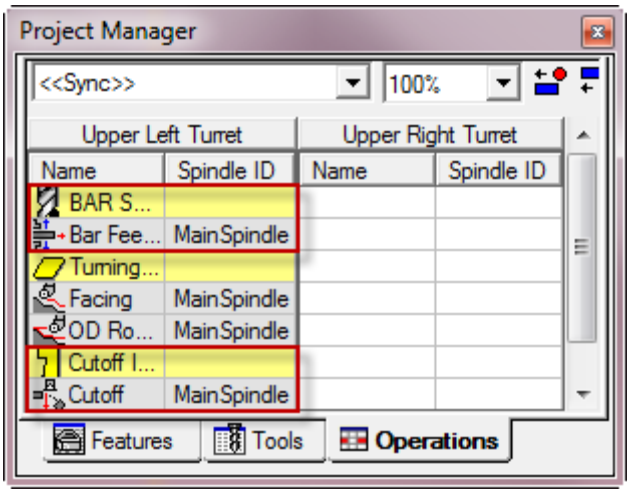
Sample operation list for NZX-ST configuration:



Sample operation list for NZX-S configuration:



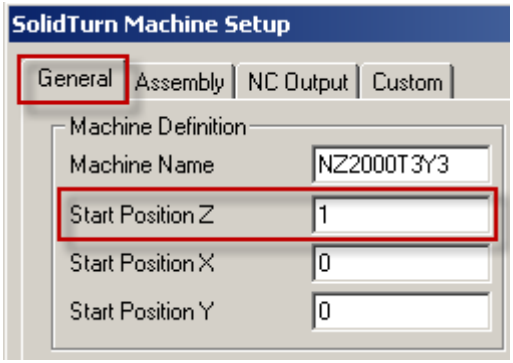
Sample operation list for NZXDL configuration:



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

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

Note: On the General tab of the Machine Setup, set the Start Position Z:
Start Position Z = Length of stock for facing the front side of the part



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

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

Steps to program this part ejection type in ESPRIT:

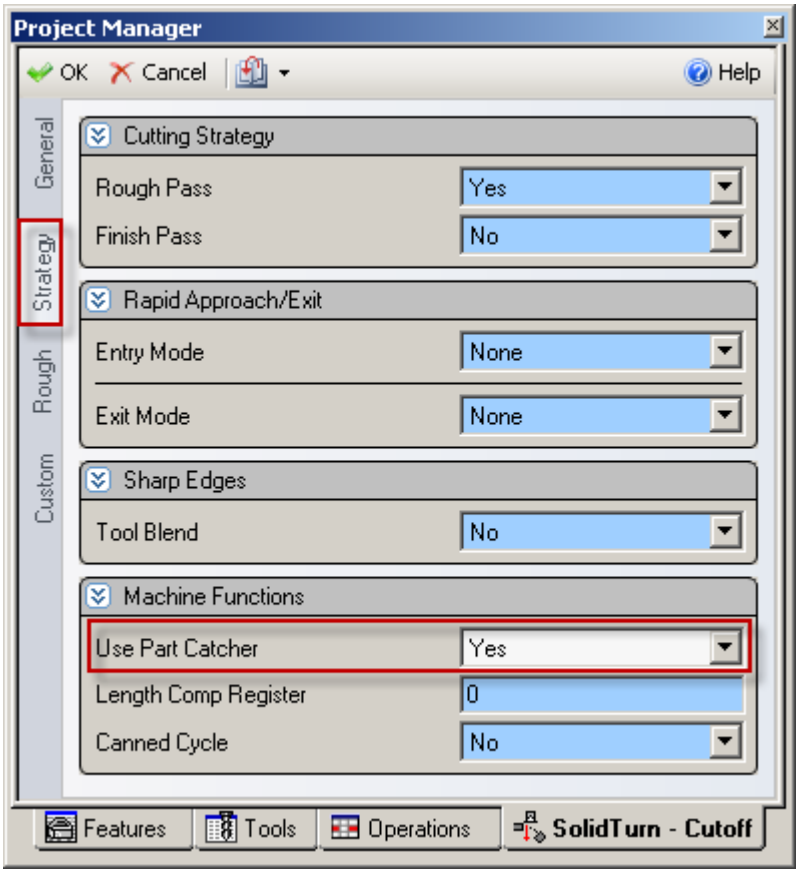
1. Machining the part on the spindle 1 side.

2. Cut-off:
   Use operation **Cutoff** using **Upper** (**Left** for NZX-ST & NZXDL configurations)

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

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

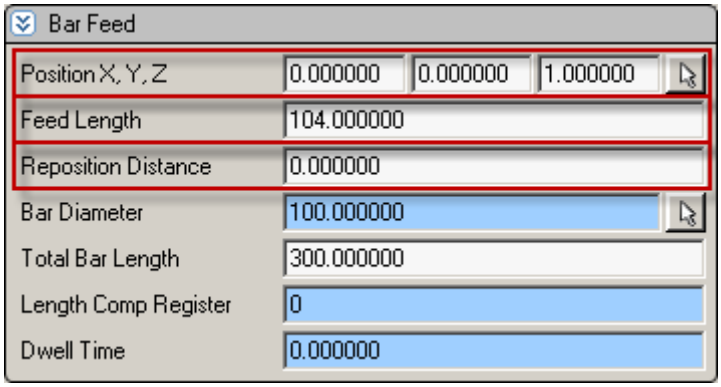
To trigger the part catching output, set Use Part Catcher to Yes on the Strategy tab.



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

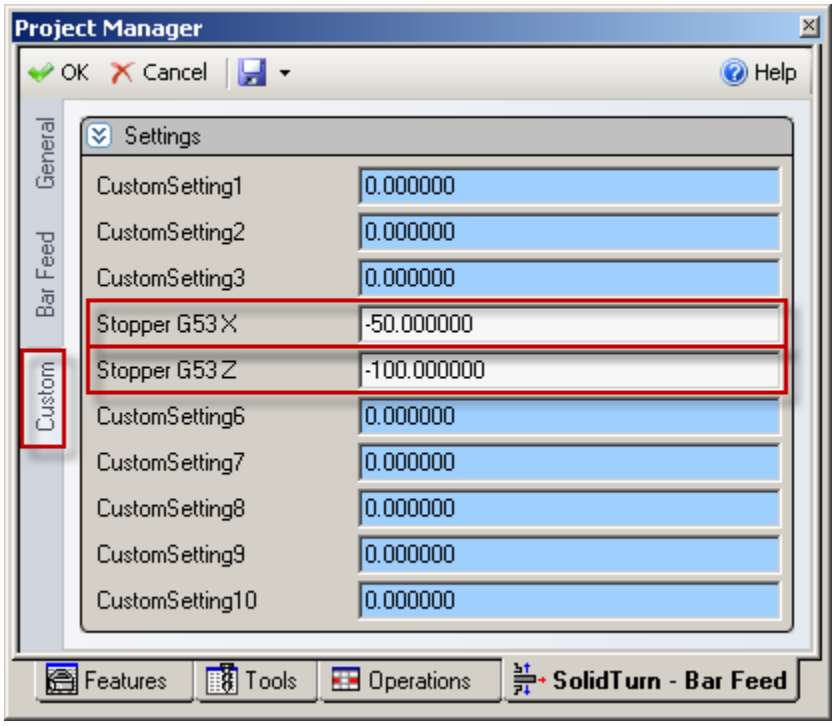
   On Bar Feed tab, set the Feed Length and Reposition Distance:
   Feed Length (=Barfeed distance) = Length of the finished part (Part Stock Length) + Cut-off tool width + Length of stock for facing the front side and, if needed, the back side of the part
   Reposition Distance = 0

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



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

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

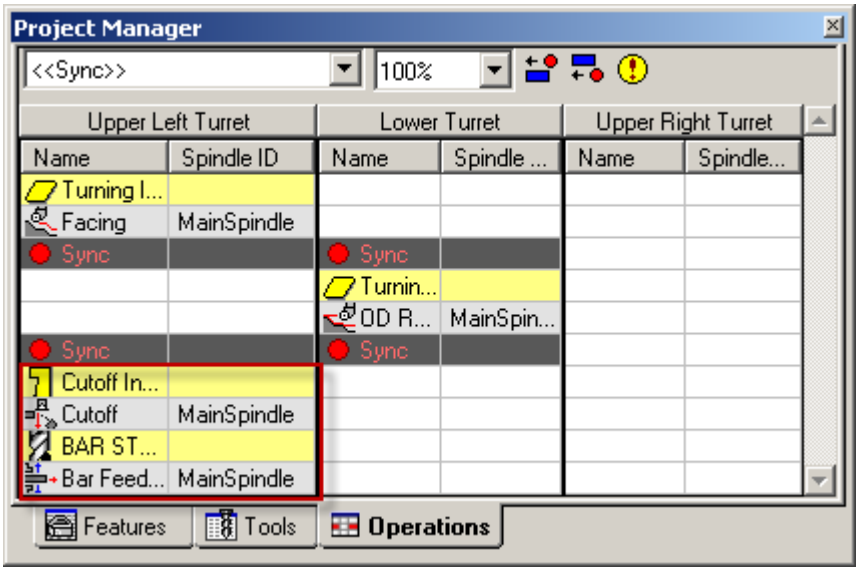


*To use current machine work coordinate defined in ESPRIT (see part **2.3. Turning Work Coordinates**):* If no value is specified on the Custom tab in the

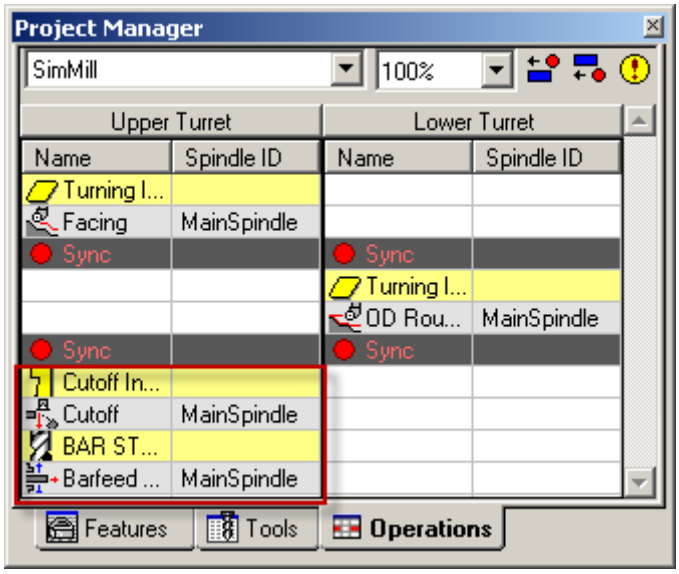
Stopper G53 X and Stopper G53 Z fields (fields left equal to 0), the stopper tool will be positioned at the programmed position (Position X, Y, Z specified on Bar Feed tab) in the current active machine work coordinate.

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

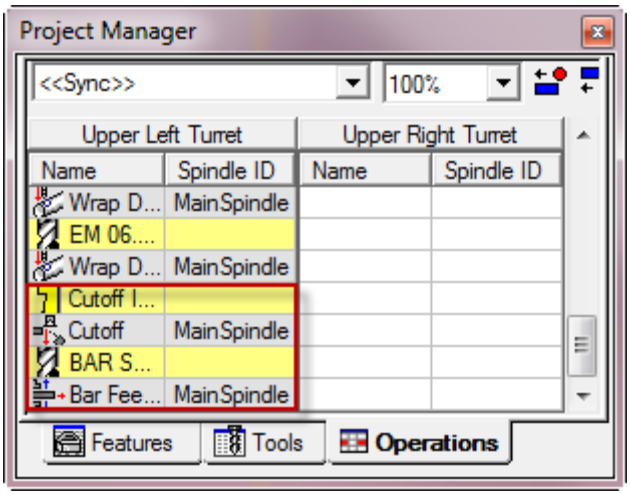
Sample operation list for NZX-ST configuration:



Sample operation list for NZX-S configuration:



Sample operation list for NZXDL configuration:



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

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

The regular steps to follow are:

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

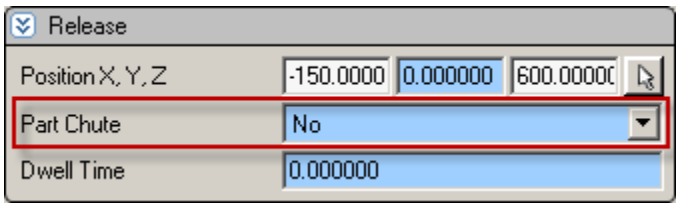
Steps to program this part ejection type in ESPRIT:

1. Machining the part on the spindle 1 side.

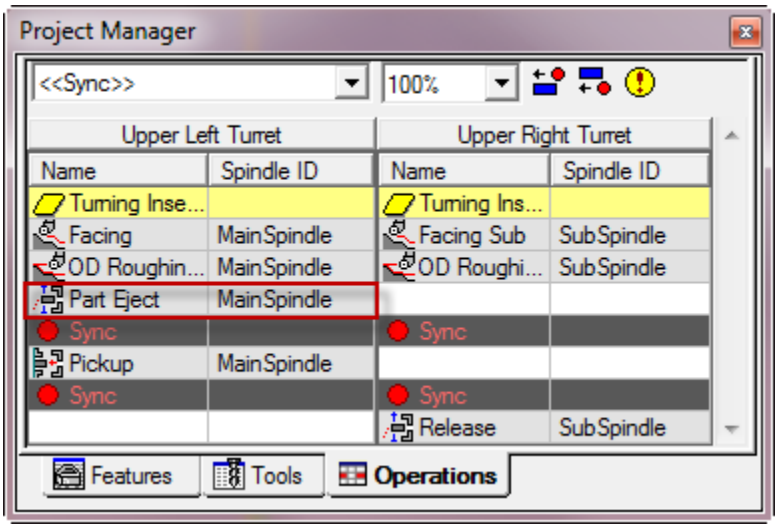
2. Manually catching the finished part in the spindle 1:
   To be able to manually catch the finished part in the spindle 1, head 1 needs to be stopped. To do so, the M-code M00 (program stop) needs to be output in head 1.

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

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



<u>Sample operation list of manual finished part catching:</u>



11.2.5. Catching finished part on spindle 2 side

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

The regular steps to follow are:

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

Steps to program this part ejection type in ESPRIT:

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

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

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

2. <u>For NZX-ST configuration:</u>
   Program a wait code between head 2 and head 3 (P23 sync code) or across the three heads (P123 sync code):
   A sync across the three heads is necessary if you want to manually catch the finished part in the spindle 2 (machine is stopped with M00).

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

OUTPUT M00.

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

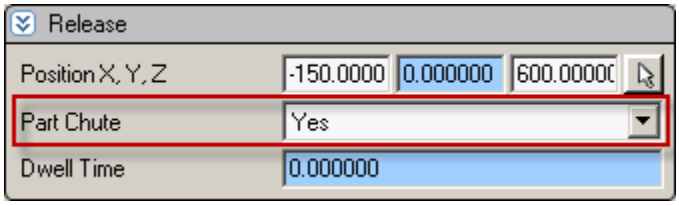
Note: **For manual part catching**, if you do not program a sync above your part release, you will get the following error message in your NC code (in head 2): ERROR: PROGRAM A SYNC ABOVE PART EJECT OPERATION TO OUTPUT M00.

3. Catching the finished part in the spindle 2:
   Use operation **Release** with **SubSpindle** (**Spindle Name**) on **Lower** (**Upper Right** for NZXDL configuration) **Turret** (**Turret Name**).

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

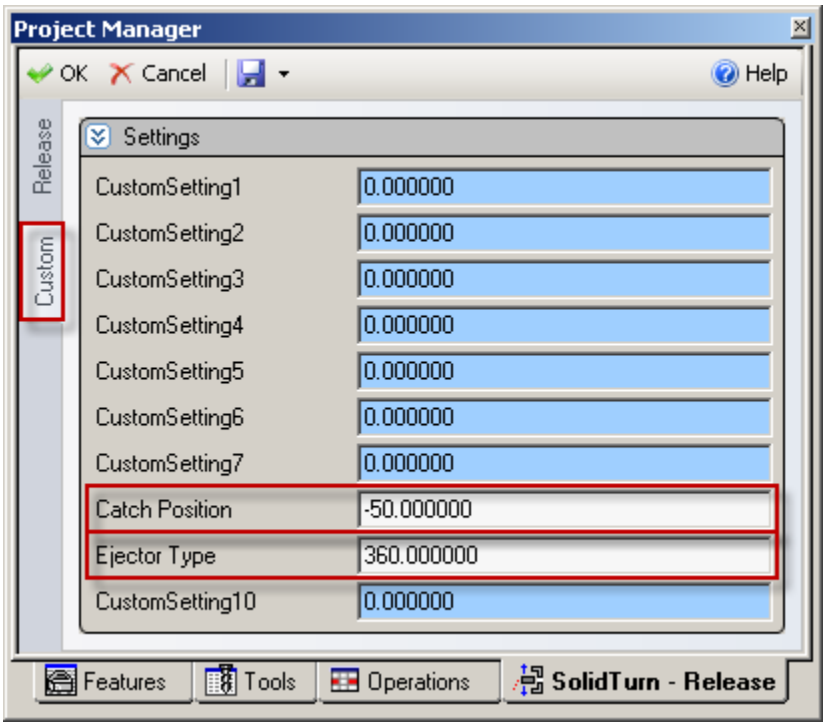
   Based on the setting set for Work Unloader Type on the POST Output Configuration (see part **4.4.3. Work unloader on spindle 2 side: hand or receiver**), the NC output will differ to match your machine specifications: hand or receiver equipped.

   Catching the finished part in the spindle 2 on a machine equipped with a hand or a receiver:
   To trigger the part catching output, set Part Chute to Yes on the Release tab.



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

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

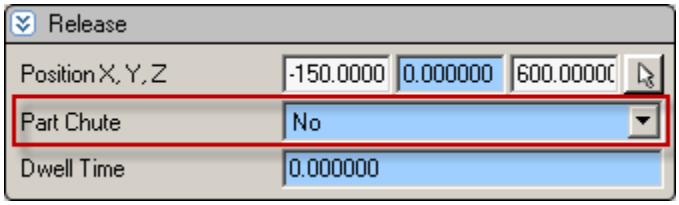


Manually catching the finished part in the spindle 2 (machine is stopped with M00 for NZX-ST & NZX-S configurations and head 2 is stopped with M00 for NZXDL configuration):

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

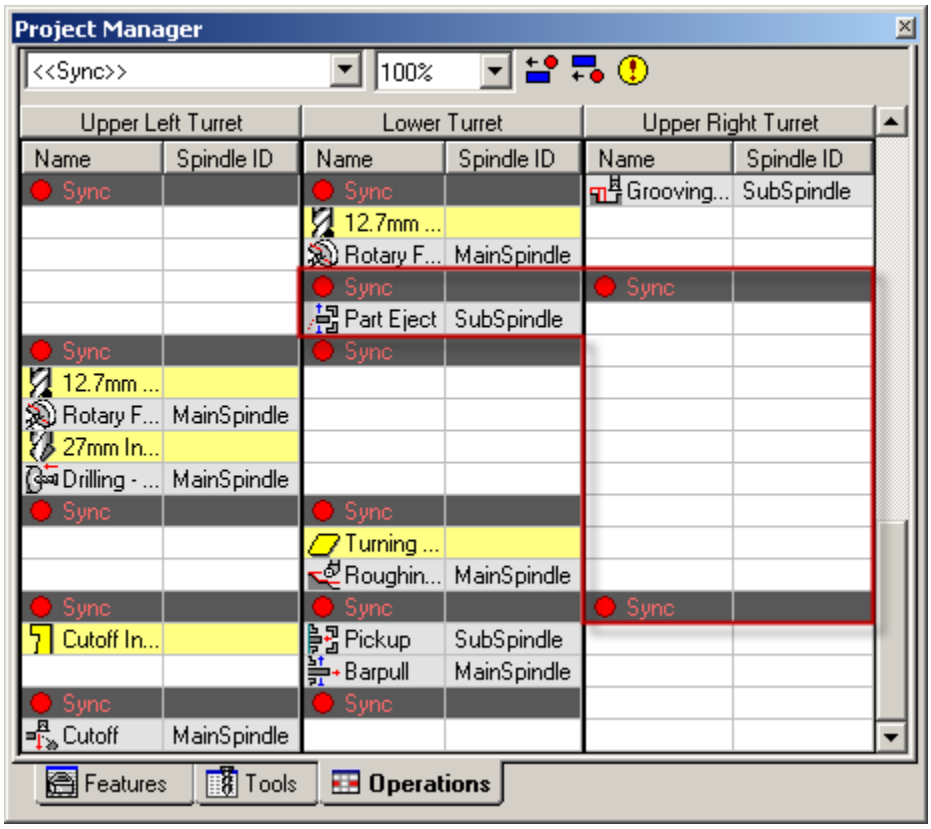
To be able to manually catch the finished part in the spindle 2 for NZXDL configuration, head 2 needs to be stopped. To do so, the M-code M00 (program stop) needs to be output in head 2.

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

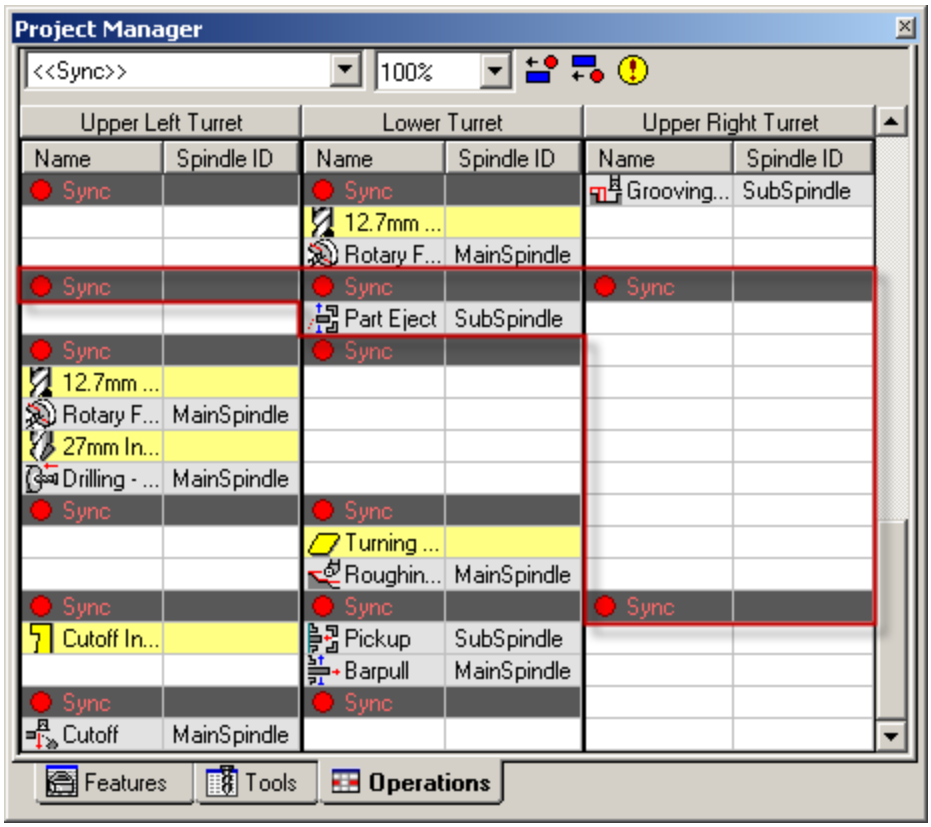


4. In head 3 for NZX-ST configuration, to avoid possible collisions, no operation should be programmed after the P23 (or P123) wait code until the next programmed wait code.

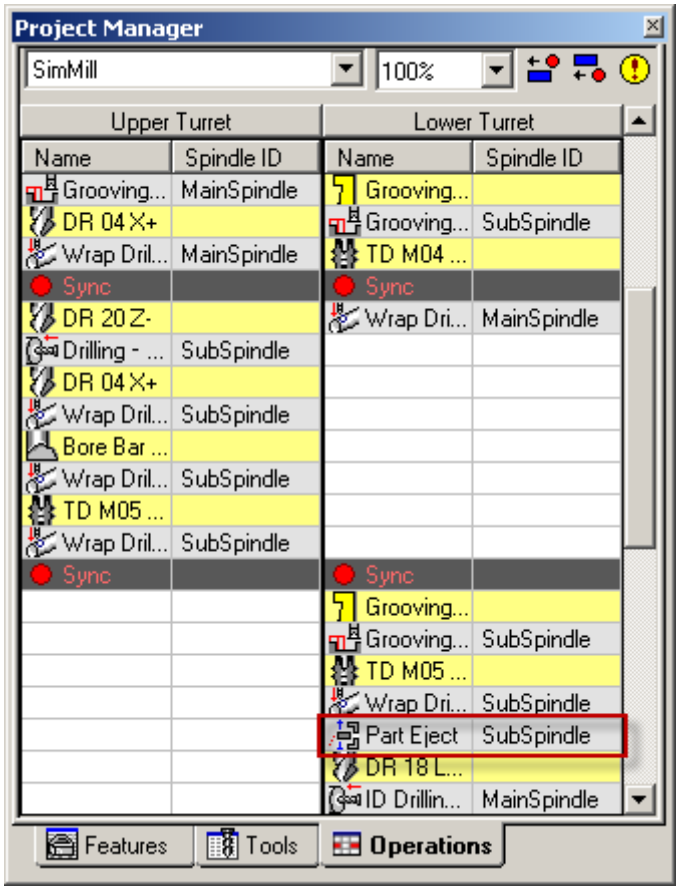
Sample operation list of finished part catching with arm or receiver for NZX-ST configuration:



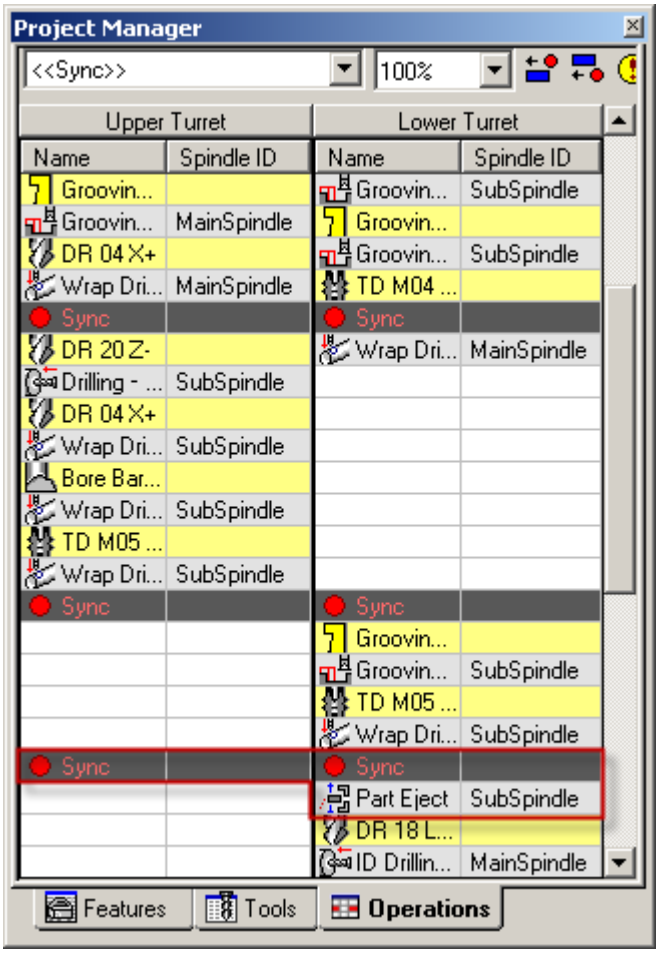
Sample operation list of manual part catching for NZX-ST configuration:



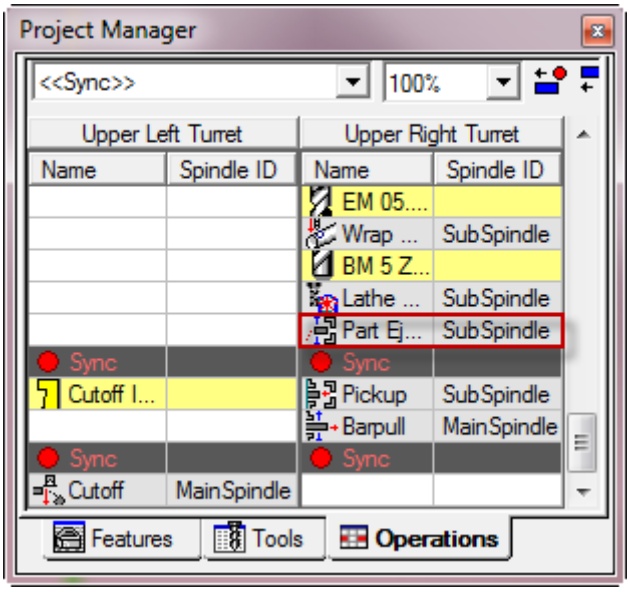
Sample operation list of finished part catching with arm or receiver for NZX-S configuration:



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



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

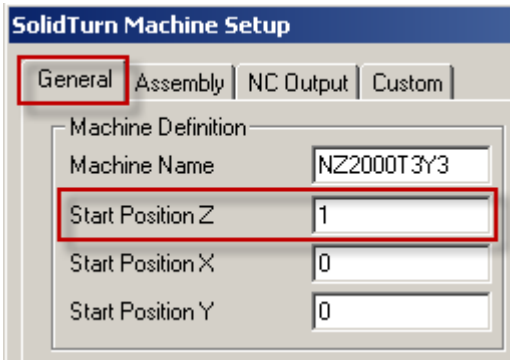


11.3. Part transfer

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

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

Note: On the General tab of the Machine Setup, set the Start Position Z:
Start Position Z = Length of stock for facing the front side of the part



The regular steps to follow are:

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

Steps to program this transfer type in ESPRIT:

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

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

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

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).

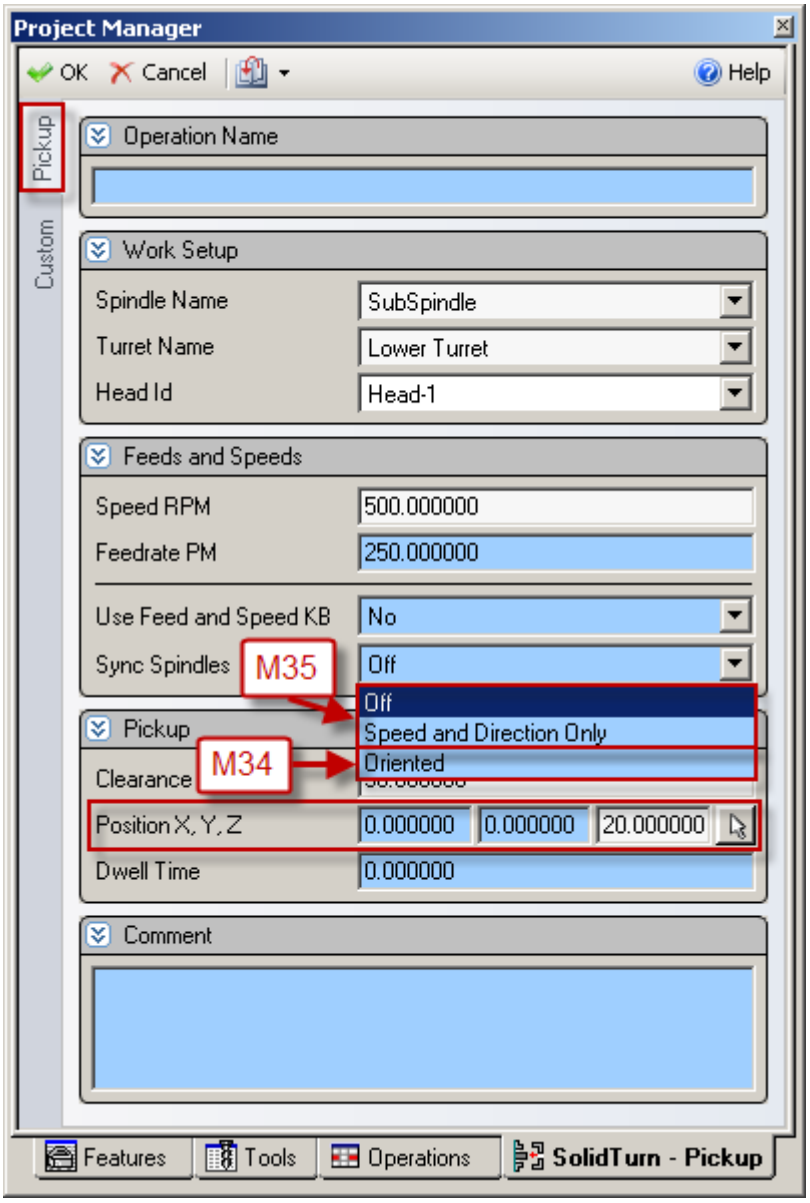
3. Program a wait code across all heads.

4. Picking the part in the spindle 1 with the spindle 2:
   Use operation **Pickup** with **SubSpindle** (**Spindle Name**) on **Lower** (**Upper Right** for NZXDL configuration) **Turret** (**Turret Name**). With Position X, Y, Z, set the pickup point.

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

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

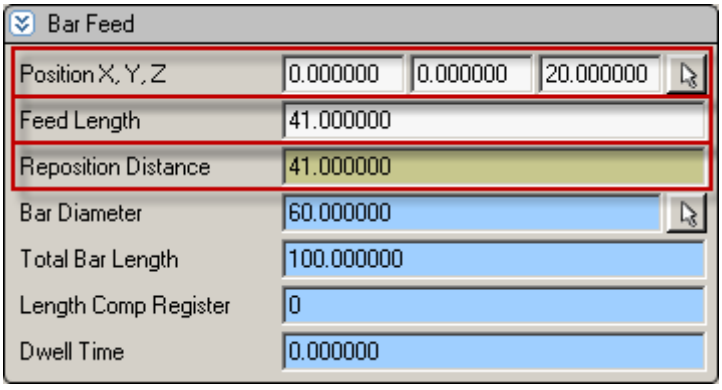
   Enter the Feedrate PM of the spindle 2 B-axis and its clearance (from the pickup point); it will be output in head 2 program (since head 2 controls the B-axis).



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

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

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



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

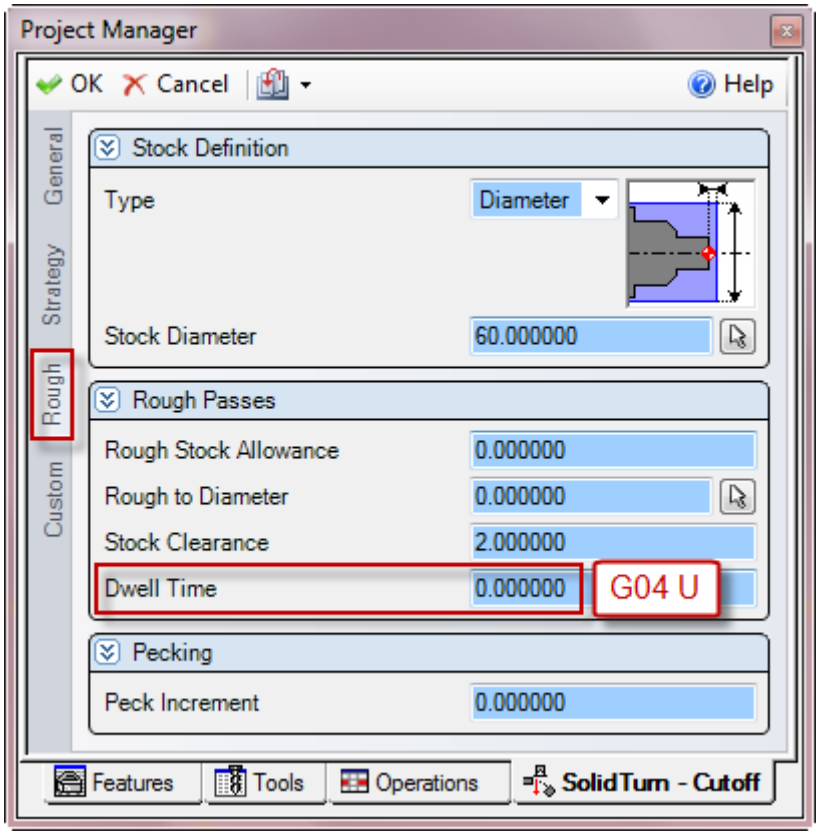
On the General tab, enter the Feedrate PM of the spindle 2 B-axis, it will be output in head 2 program (since head 2 controls the B-axis).

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

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

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

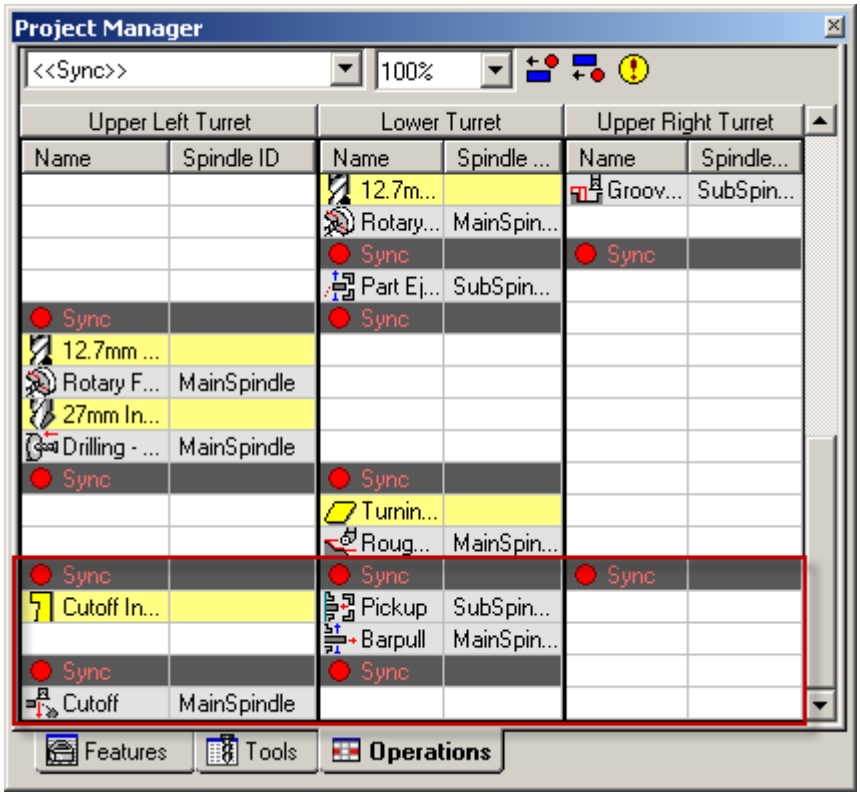
   To output a dwell (G04 U) above workpiece cut-off detection (M80), enter in the Dwell Time field on the Rough tab of the Cutoff operation the dwell time in s.



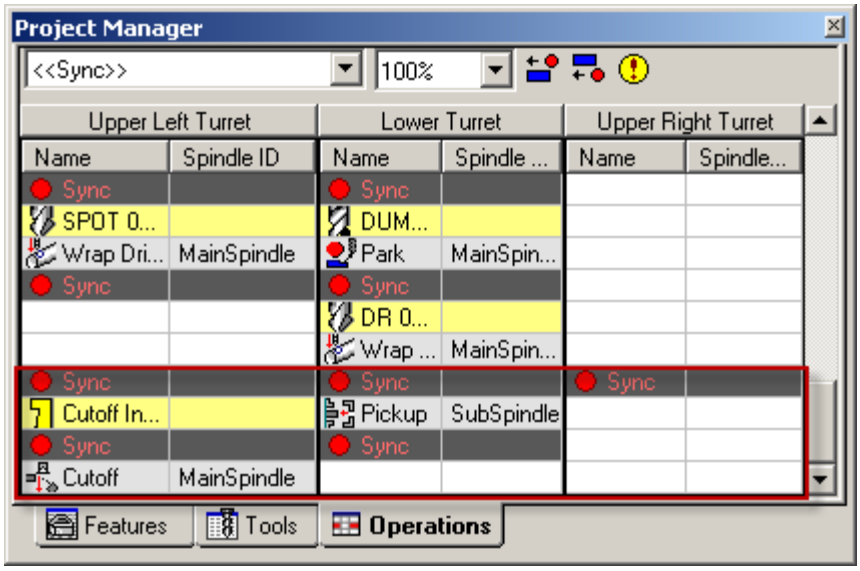
The spindle 2 automatically goes home after cut-off completion.

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

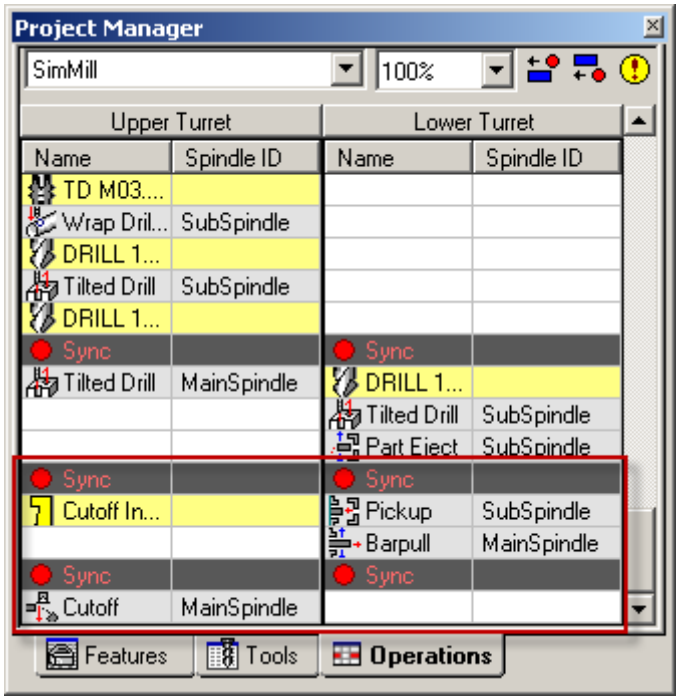
Sample operation list with transfer (with barpull) and cut-off for NZX-ST configuration:



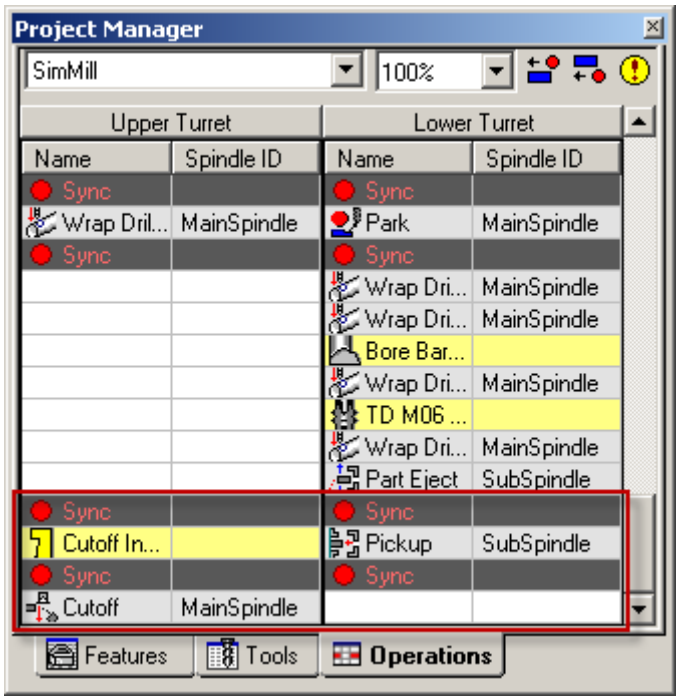
Sample operation list with transfer (without barpull) and cut-off for NZX-ST configuration:



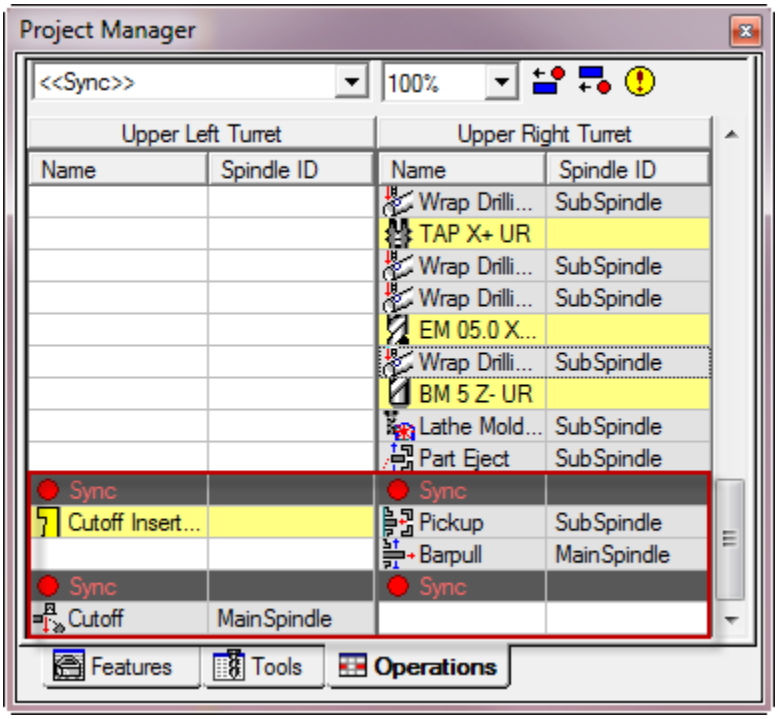
Sample operation list with transfer (with barpull) and cut-off for NZX-S configuration:



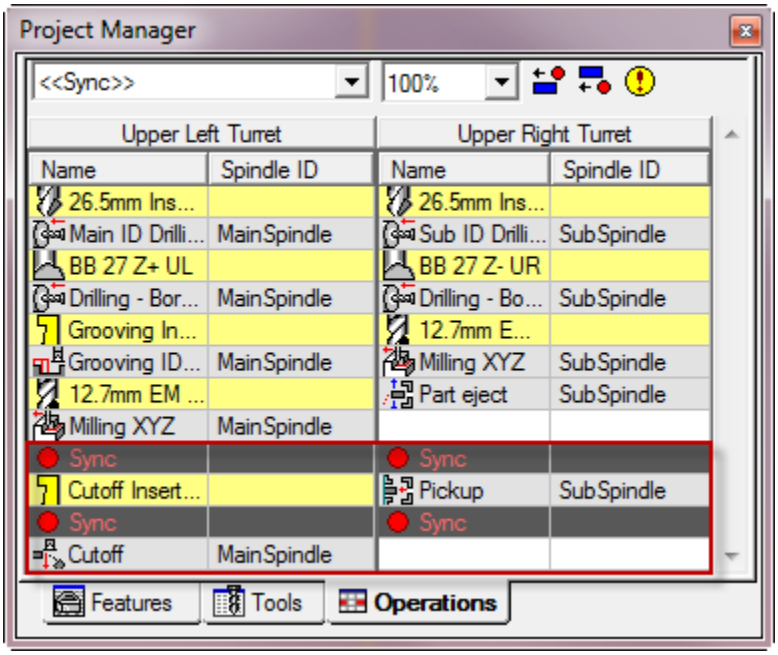
Sample operation list with transfer (without barpull) and cut-off for NZX-S configuration:



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



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

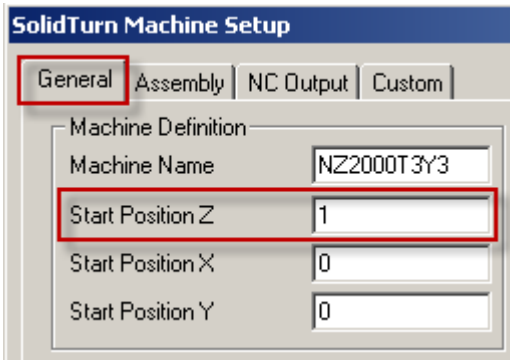


11.3.2. Part transfer from spindle 1 to spindle 2 with cut-off (when machine equipped with a barfeeder)

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

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

Note: On the General tab of the Machine Setup, set the Start Position Z:
Start Position Z = Length of stock for facing the front side of the part



The regular steps to follow are:

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

Steps to program this transfer type in ESPRIT:

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

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

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

2. Catching the finished part in the spindle 2:
   Spindle 2 needs to be empty before transferring the workpiece from spindle 1 to

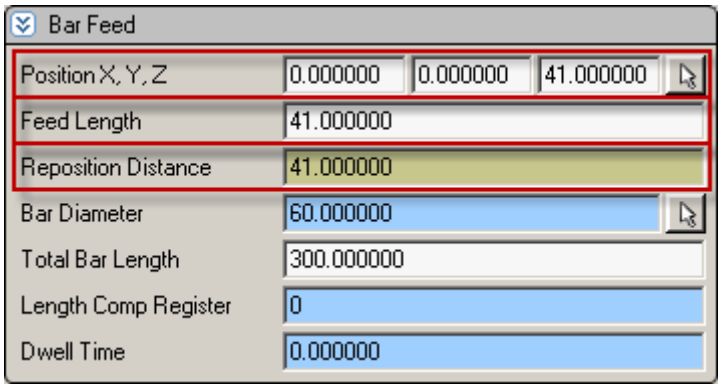
spindle 2 (see part **11.2.5. Catching finished part on spindle 2 side** for programming details).

3. Program a wait code across all heads.

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

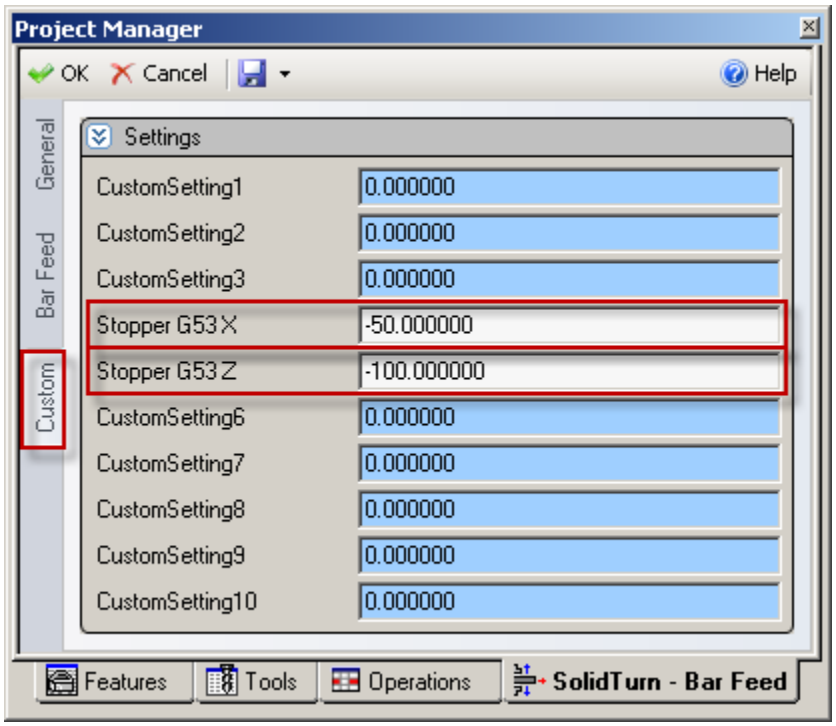
   On Bar Feed tab, set the Feed Length and Reposition Distance:
   Feed Length (=Barfeed distance = Reposition Distance) = Length of the finished part (Part Stock Length) + Cut-off tool width + Length of stock for facing the front side and the back side of the part

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



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

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



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

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

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

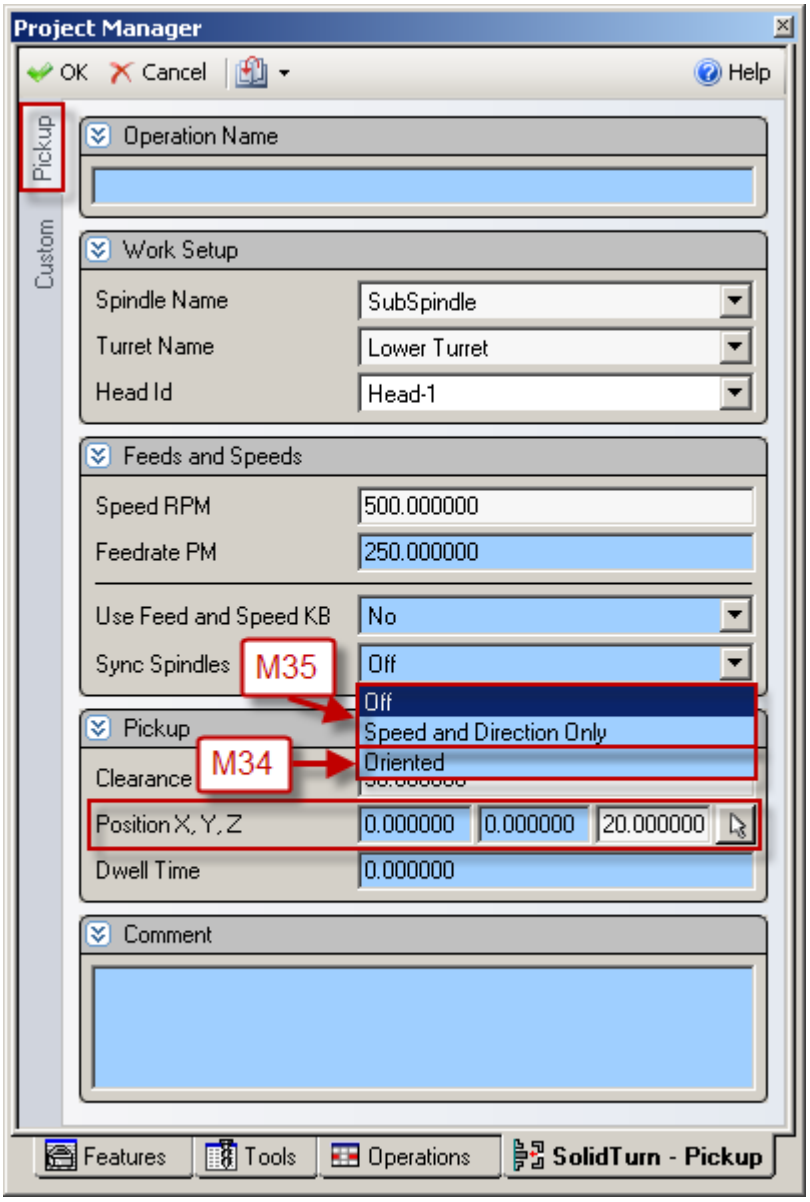
6. Picking the part in the spindle 1 with the spindle 2:
   Use operation **Pickup** with **SubSpindle** (**Spindle Name**) on **Lower** (**Upper Right** for NZXDL configuration) **Turret** (**Turret Name**). With Position X, Y, Z, set the pickup point. Note that the pickup point will be shifted by the reposition distance amount entered on Bar Feed tab of Bar Feed operation.

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

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

output on head 1 after synchronization of the spindles.

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

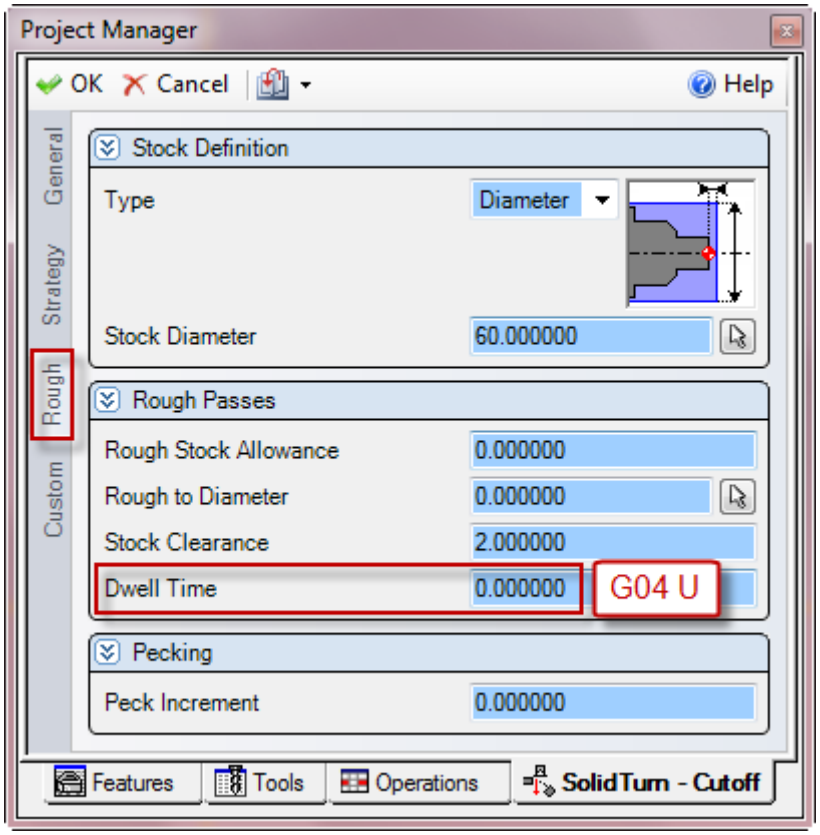


7. Program a wait code between upper left turret and lower turret (P12 sync code) for NZX-ST configuration.
   Program a wait code across the two heads for NZX-S & NZXDL configurations.

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

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

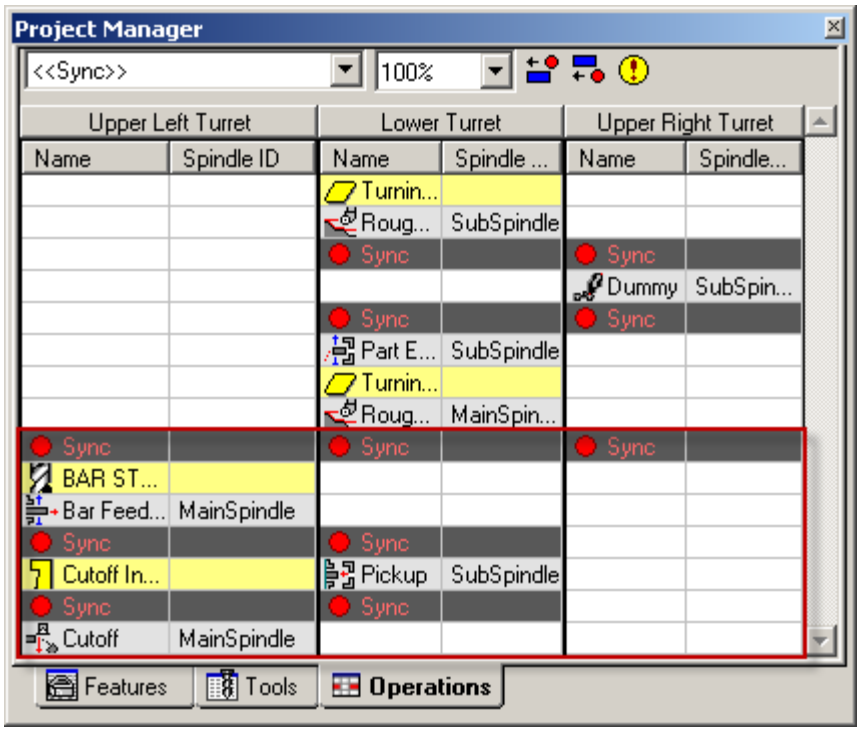
To output a dwell (G04 U) above workpiece cut-off detection (M80), enter in the Dwell Time field on the Rough tab of the Cutoff operation the dwell time in s.



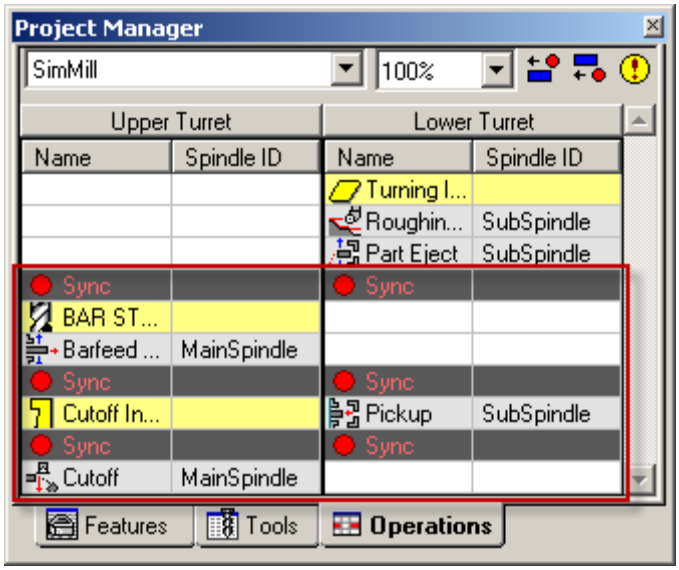
The spindle 2 automatically goes home after cut-off completion.

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

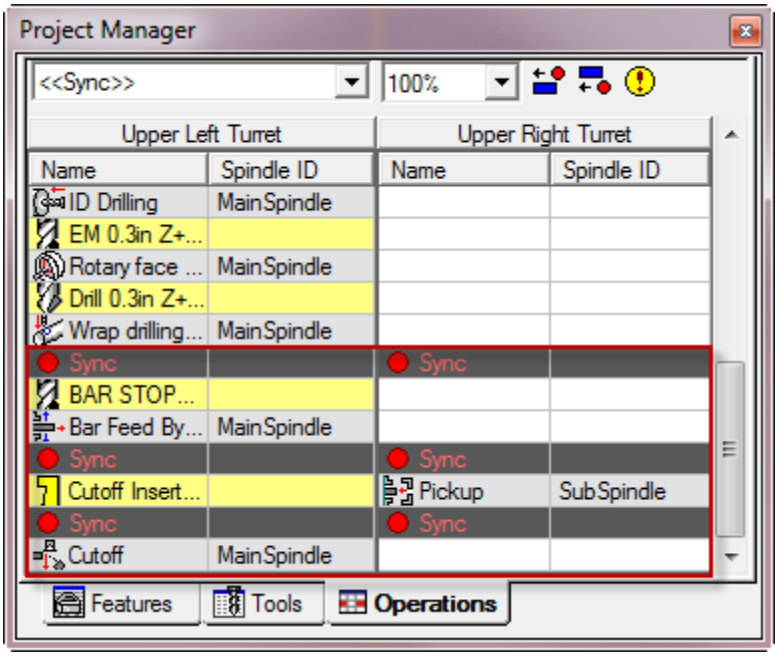
Sample operation list for NZX-ST configuration:



Sample operation list for NZX-S configuration:



Sample operation list for NZXDL configuration:

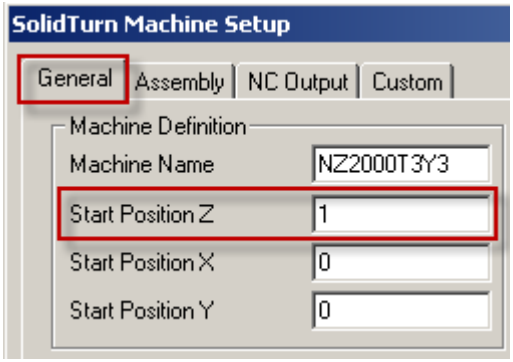


11.3.3. Part transfer from spindle 1 to spindle 2 with cut-off and stock repositioning (programmed last)

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

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

Note: On the General tab of the Machine Setup, set the Start Position Z:
Start Position Z = Length of stock for facing the front side of the part



The regular steps to follow are:

1. Machining the parts on the spindle 1 and spindle 2 sides
2. Catching the finished part in the spindle 2
3. Picking the part in the spindle 1 with the spindle 2

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

Steps to program this transfer type in ESPRIT:

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

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

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

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).

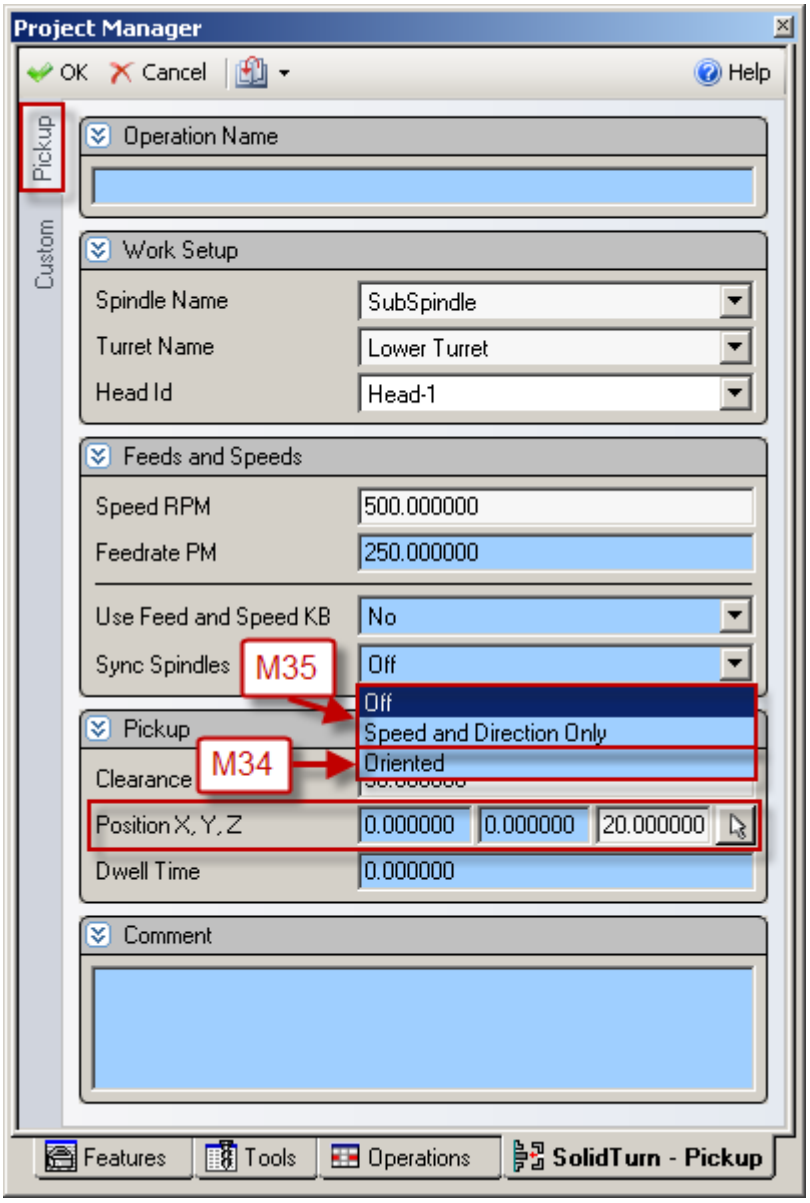
3. Program a wait code across all heads.

4. Picking the part in the spindle 1 with the spindle 2:
   Use operation **Pickup** with **SubSpindle** (**Spindle Name**) on **Lower** (**Upper Right** for NZXDL configuration) **Turret** (**Turret Name**). With Position X, Y, Z, set the pickup point.

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

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

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



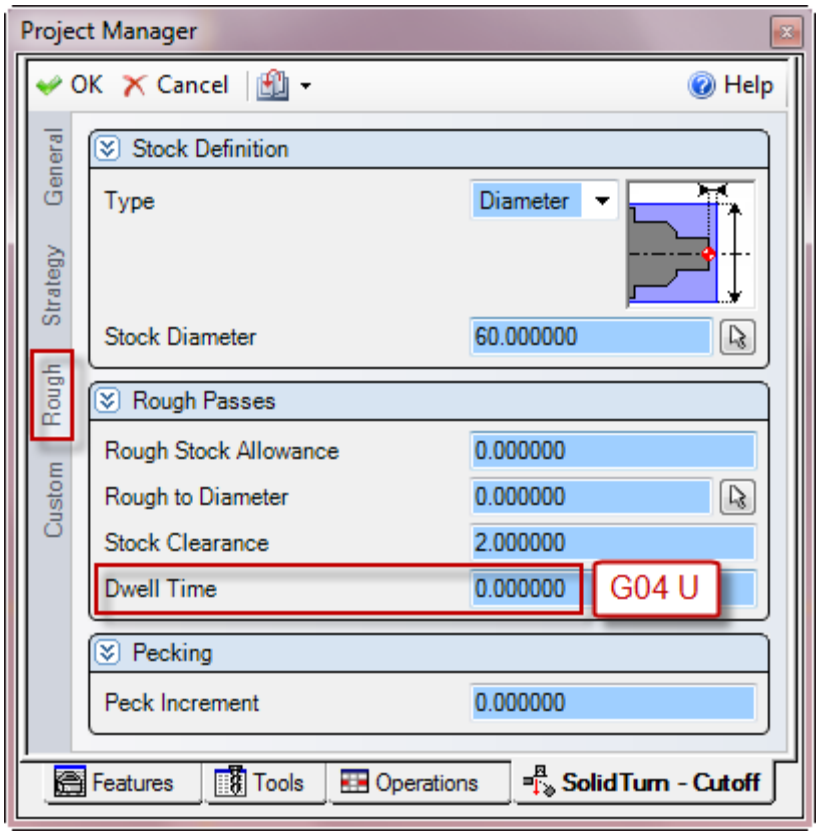
5. Program a wait code between upper left turret and lower turret (P12 sync code) for NZX-ST configuration.
   Program a wait code across the two heads for NZX-S & NZXDL configurations.

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

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

configuration.

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



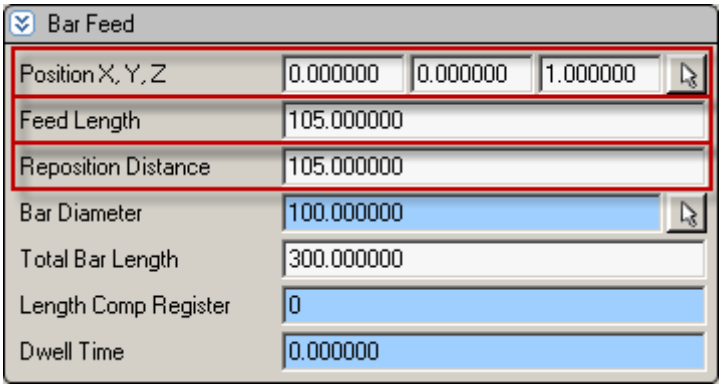
The spindle 2 automatically goes home after cut-off completion.

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

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

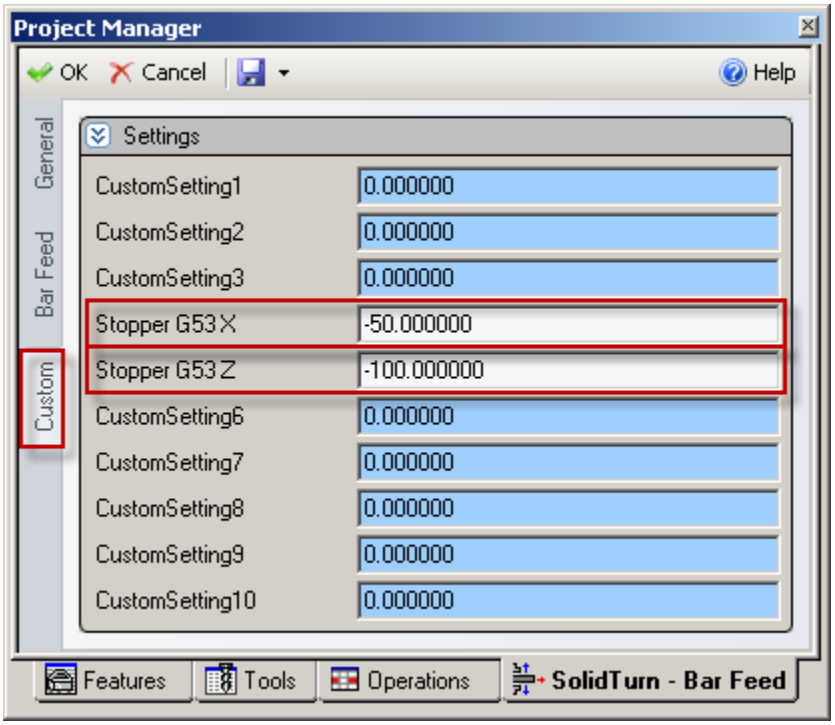
   On Bar Feed tab, set the Feed Length and Reposition Distance:
   Feed Length (=Barfeed distance = Reposition Distance) = Length of the finished part (Part Stock Length) + Cut-off tool width + Length of stock for facing the front side and the back side of the part

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



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

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

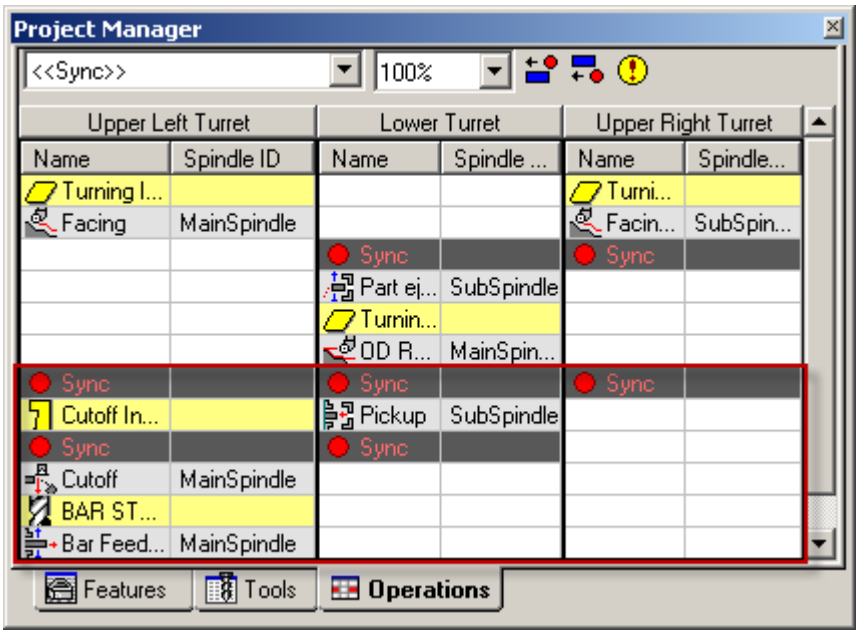


*To use current machine work coordinate defined in ESPRIT (see part **2.3. Turning Work Coordinates**):* If no value is specified on the Custom tab in the Stopper G53 X and Stopper G53 Z fields (fields left equal to 0), the stopper tool will be positioned at the programmed position (Position X, Y, Z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

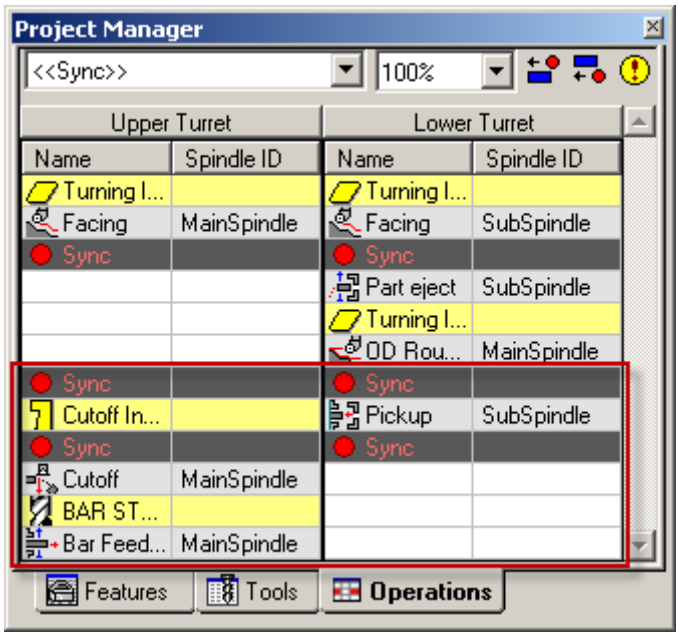
Note that using End of Bar Macro Call (/2 M98 P) on the POST Output

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

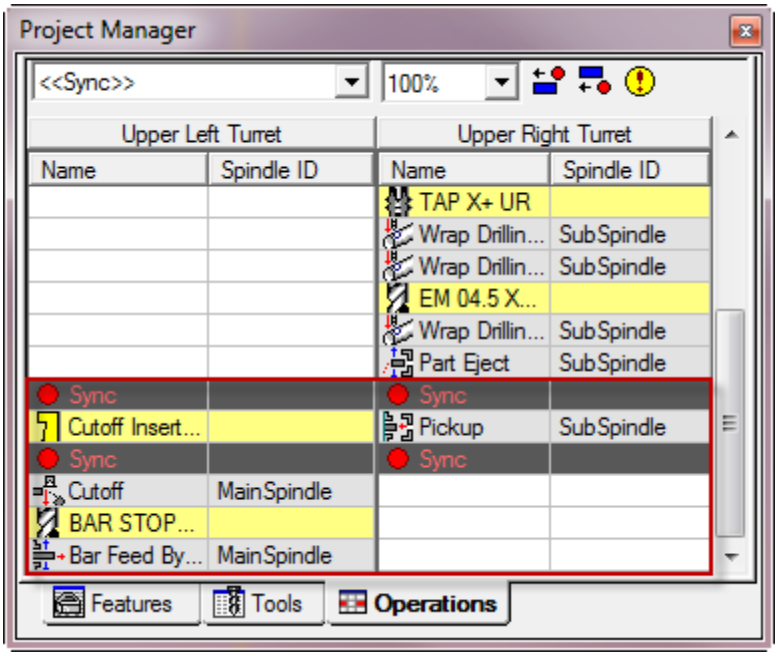
Sample operation list with transfer, cut-off and barfeed (last) for NZX-ST configuration:



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



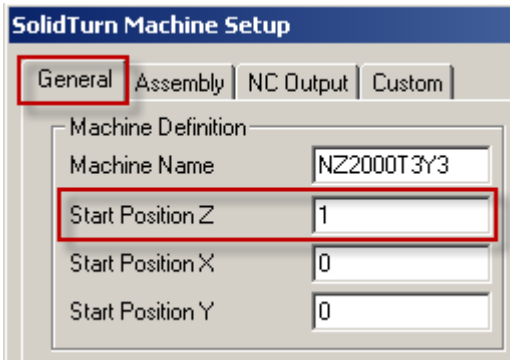
Sample operation list with transfer, cut-off and barfeed (last) for NZXDL configuration:



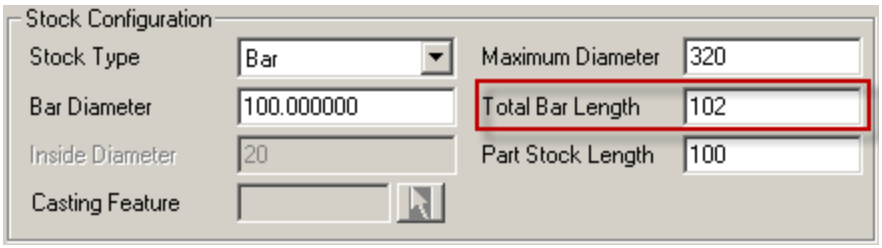
11.3.4. Part transfer from spindle 1 to spindle 2 without cut-off

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

Note: On the General tab of the Machine Setup, set the Start Position Z and Total Bar Length:
Start Position Z = Length of stock for facing the front side of the part



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

Steps to program this transfer type in ESPRIT:

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

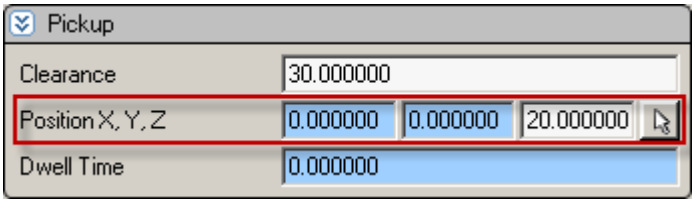
   The setting "Part Stock Length" from the Machine Setup is used as default value for Turning Work Coordinates Sub Spindle Z Offset.

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

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. Program a wait code across all heads.

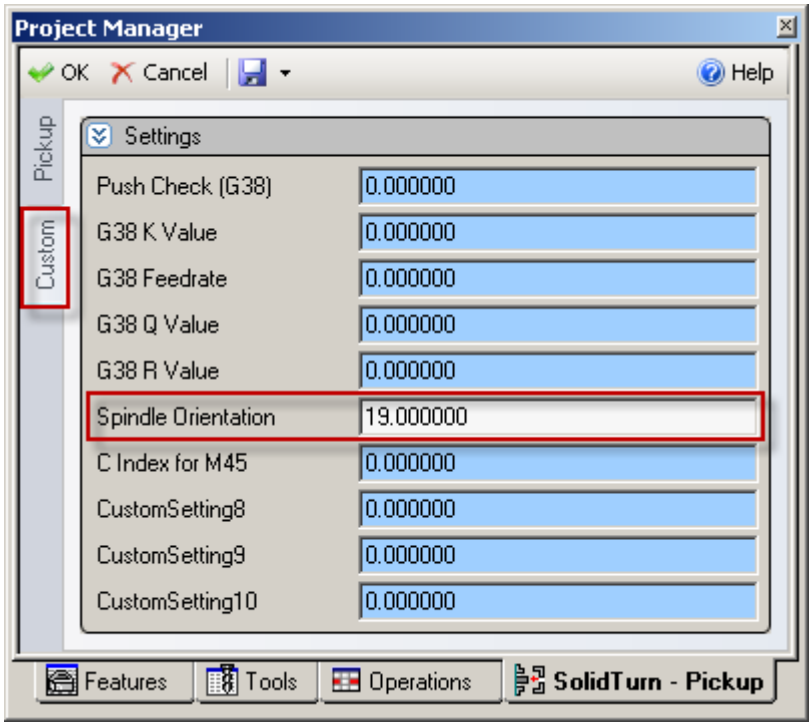
4. Picking the part in the spindle 1 with the spindle 2:
   Use operation **Pickup** with **SubSpindle** (**Spindle Name**) on **Lower** (**Upper Right** for NZXDL configuration) **Turret** (**Turret Name**). With Position X, Y, Z, set the pickup point.

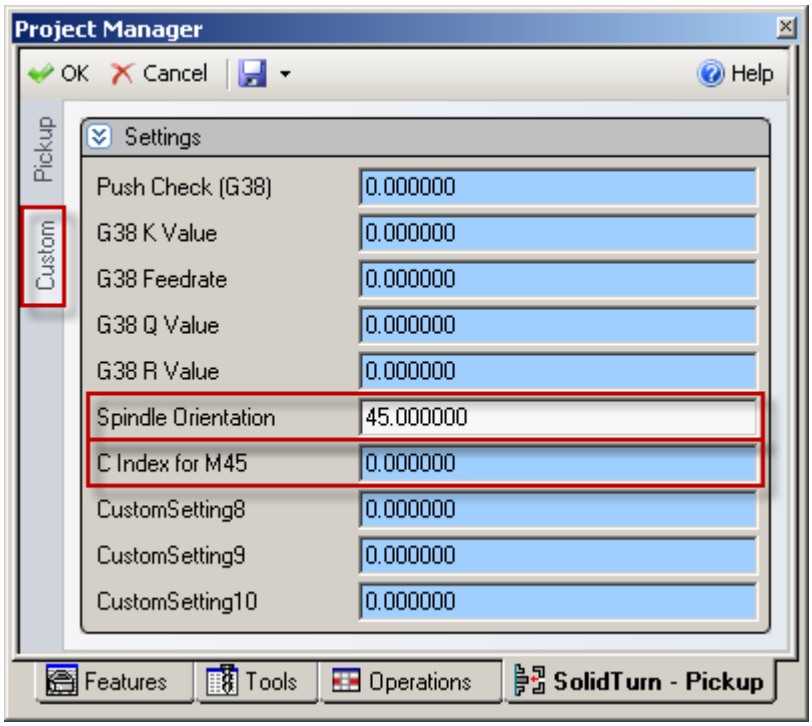


The pickup operation has to be programmed with lower (upper right for NZXDL configuration) turret because the lower turret controls the B-axis (spindle 2).

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

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

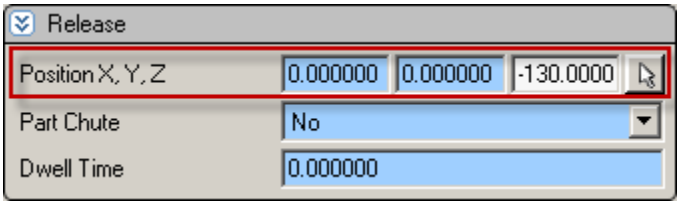




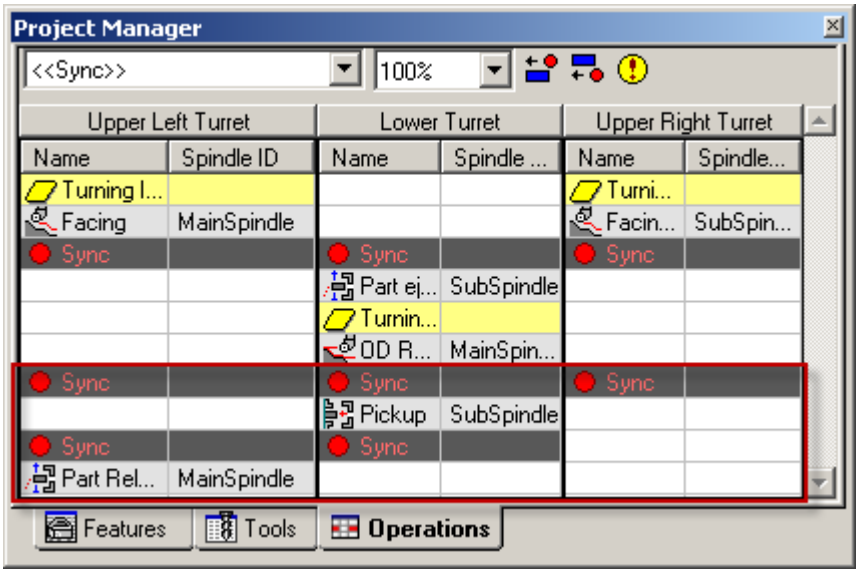
Note that the milling C-axis mode is the default. So if you omit to enter a value in the Spindle Orientation field or enter a wrong value, the spindles will be indexed using this mode.

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

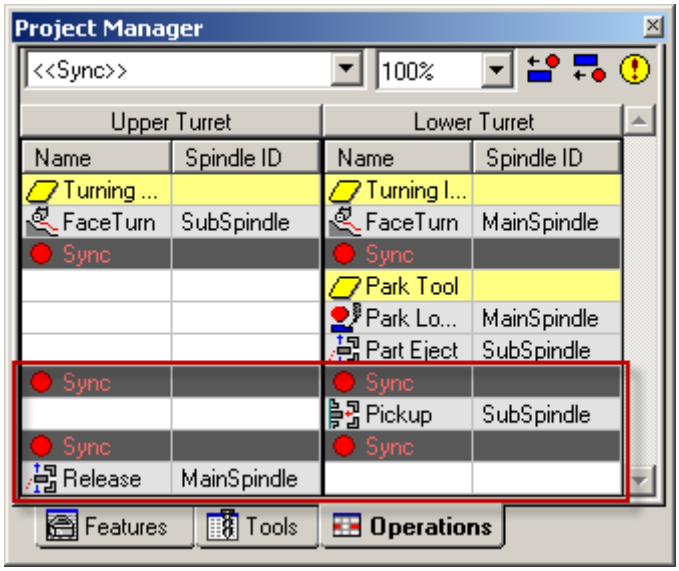
6. Releasing the part from the spindle 1:
   Use operation **Release** with **MainSpindle** (**Spindle Name**) on **Upper** (**Left** for NZX-ST & NZXDL configurations) **Turret** (**Turret Name**). With Position X, Y, Z, set the release point.



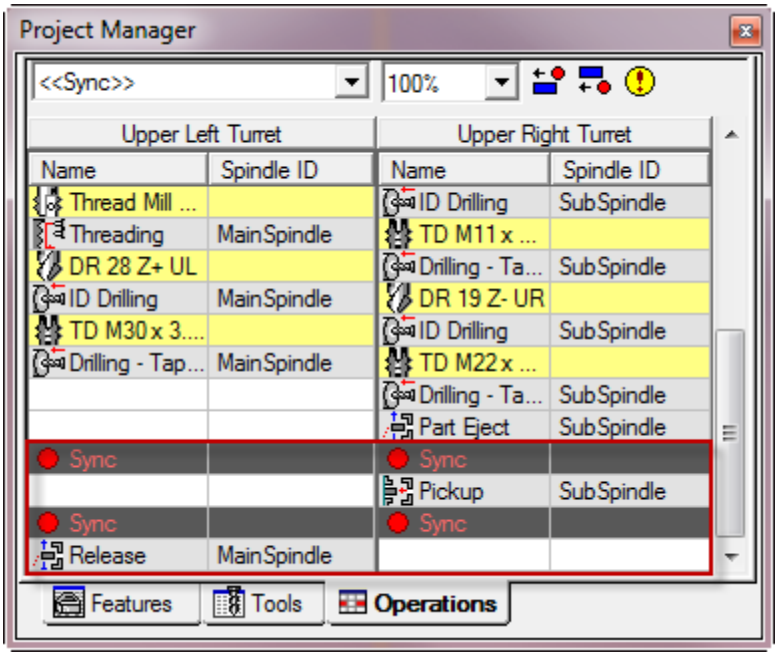
Sample operation list for NZX-ST configuration:



Sample operation list for NZX-S configuration:



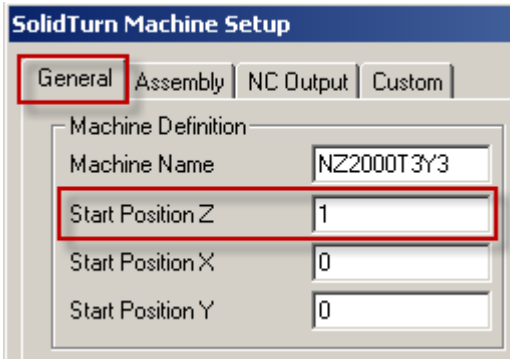
Sample operation list for NZXDL configuration:



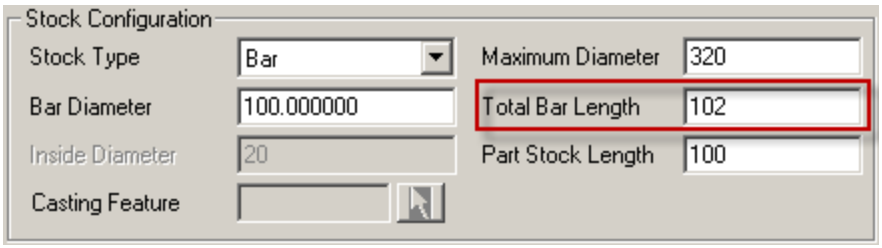
11.3.5. Part transfer from spindle 2 to spindle 1 without cut-off

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

Note: On the General tab of the Machine Setup, set the Start Position Z:
Start Position Z = Length of stock for facing the front side of the part



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

Steps to program this transfer type in ESPRIT:

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

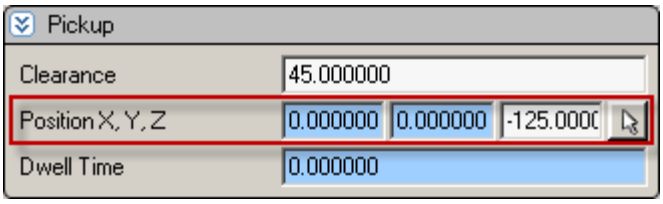
   The setting "Part Stock Length" from the Machine Setup is used as default value for Turning Work Coordinates Sub Spindle Z Offset.

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

2. Manually unloading the finished part in the spindle 1:
   Spindle 1 needs to be empty before transferring the workpiece from spindle 2 to spindle 1 (see part **11.2.4. Manually catching finished part on spindle 1 side** for programming details for NZXDL configuration).
   Note for NZX-ST & NZX-S configurations: A **Release** operation with **MainSpindle** (**Spindle Name**) on **Upper** (**Left** for NZX-ST configuration) **Turret** (**Turret Name**) can be programmed for simulation purpose (part eject). It will not affect the NC code.

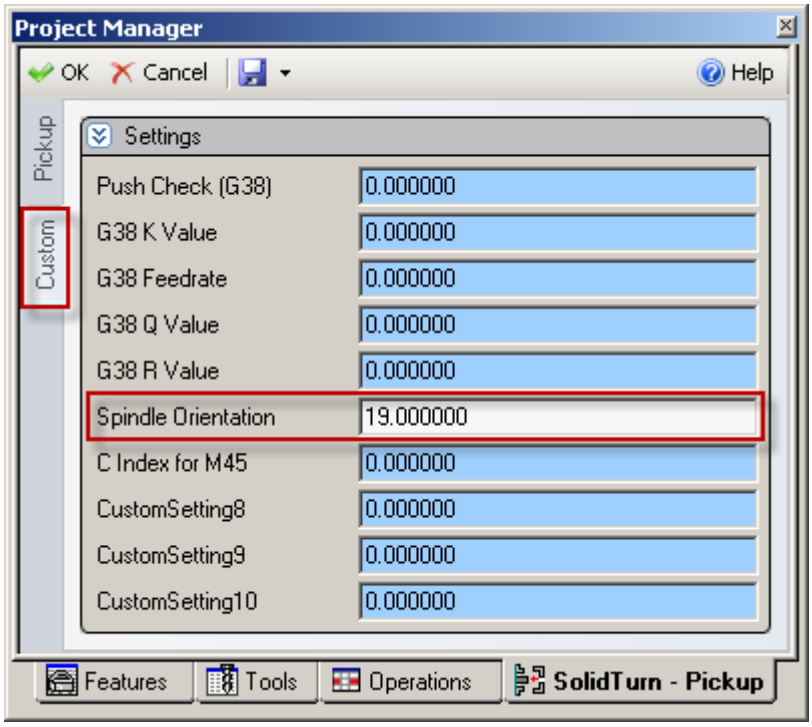
3. Program a wait code across all heads.

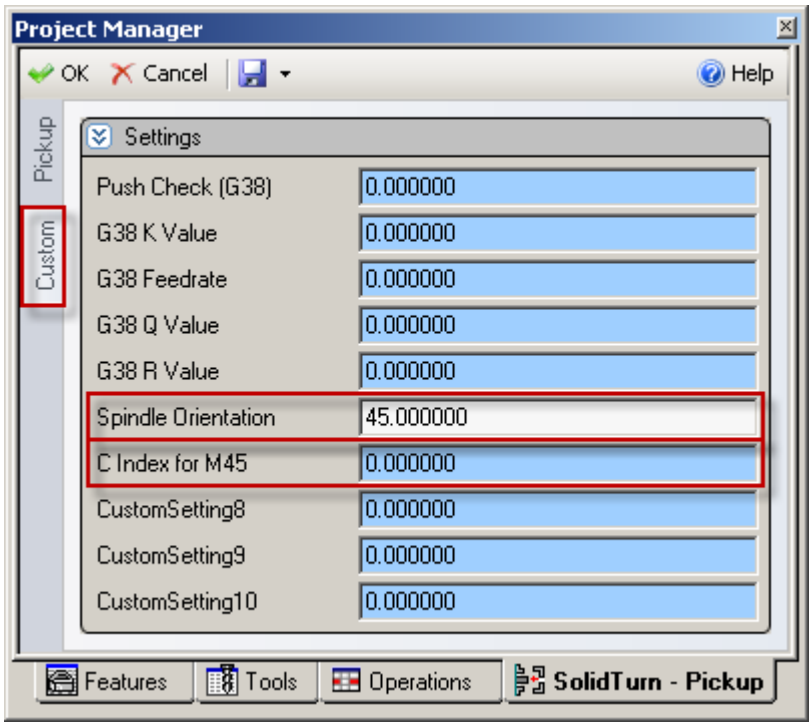
4. Picking the part in the spindle 2 with the spindle 1:
   Use operation **Pickup** with **MainSpindle** (**Spindle Name**) on **Upper** (**Left** for NZX-ST & NZXDL configurations) **Turret** (**Turret Name**). With Position X, Y, Z, set the pickup point: point that you would like the back of spindle 1 to reach while picking up the part (usually the zero point of the part in the spindle 2).



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

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

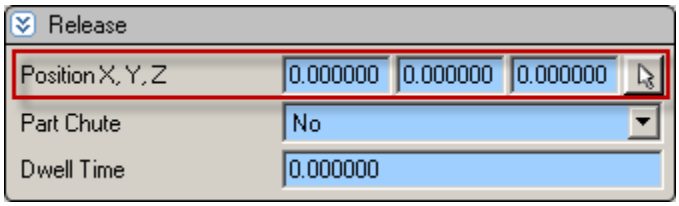




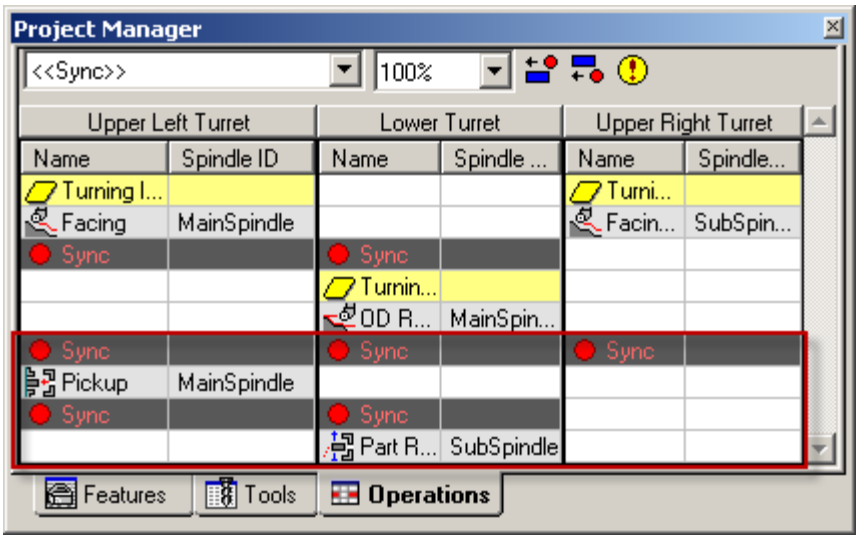
Note that the milling C-axis mode is the default. So if you omit to enter a value in the Spindle Orientation field or enter a wrong value, the spindles will be indexed using this mode.

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

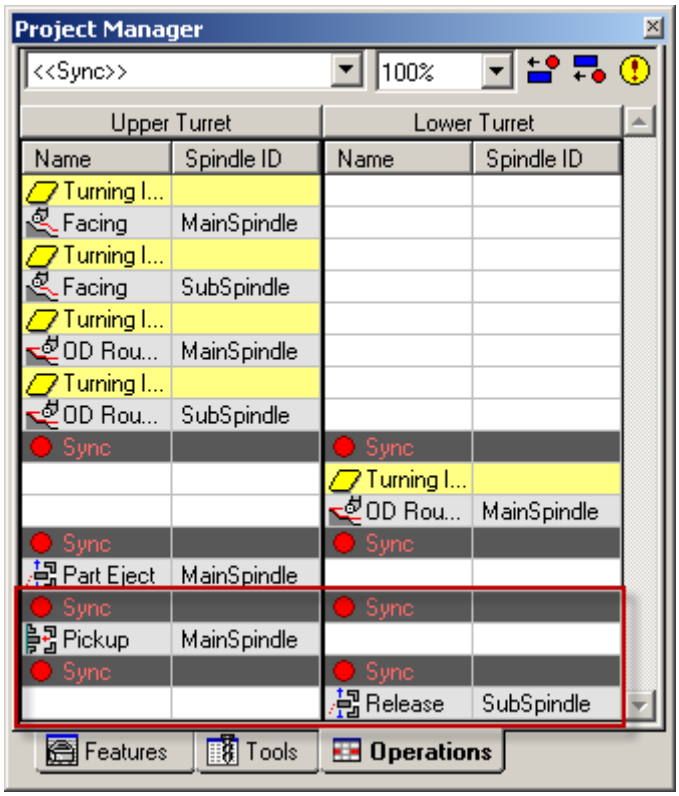
6. Releasing the part from the spindle 2:
   Use operation **Release** with **SubSpindle** (**Spindle Name**) on **Lower** (**Upper Right** for NZXDL configuration) **Turret** (**Turret Name**). With Position X, Y, Z, set the release point.



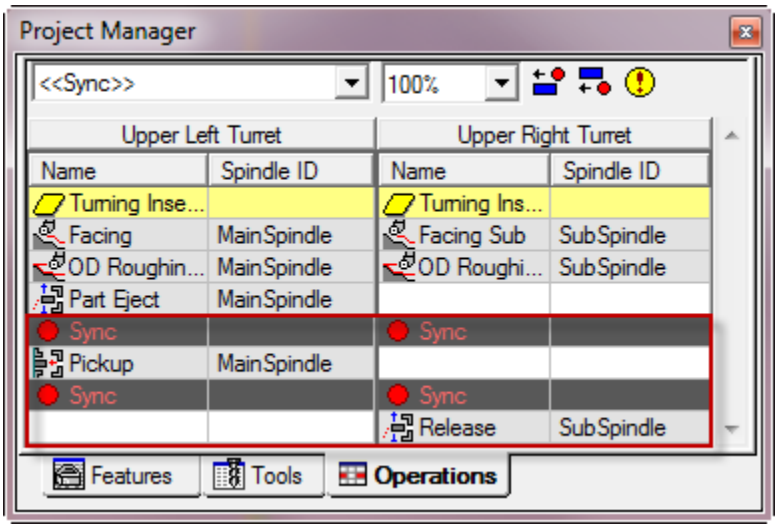
Sample operation list for NZX-ST configuration:



Sample operation list for NZX-S configuration:



Sample operation list for NZXDL configuration:



11.3.6. Stock repositioning

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

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

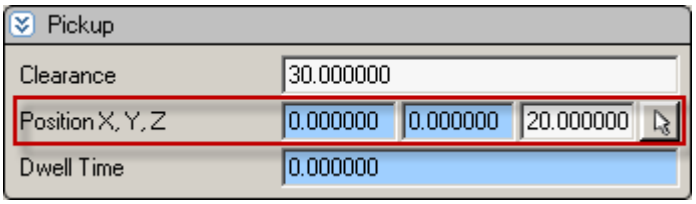
The regular steps to follow are:

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

Steps to program this transfer type in ESPRIT:

1. Program a wait code across all heads.

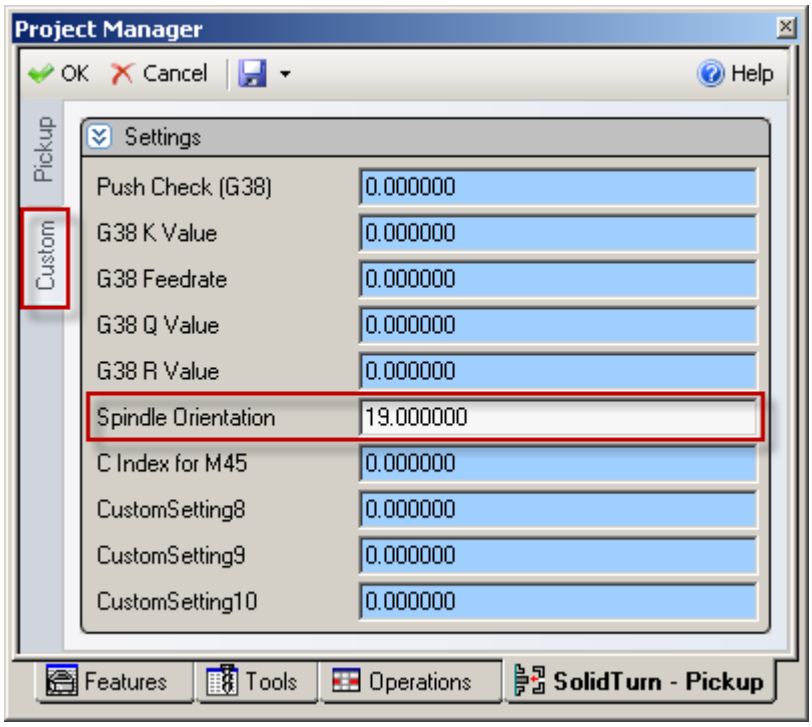
2. Picking the part in the spindle 1 with the (empty) spindle 2:
   Use operation **Pickup** with **SubSpindle** (**Spindle Name**) on **Lower** (**Upper Right** for NZXDL configuration) **Turret** (**Turret Name**). With Position X, Y, Z, set the pickup point.

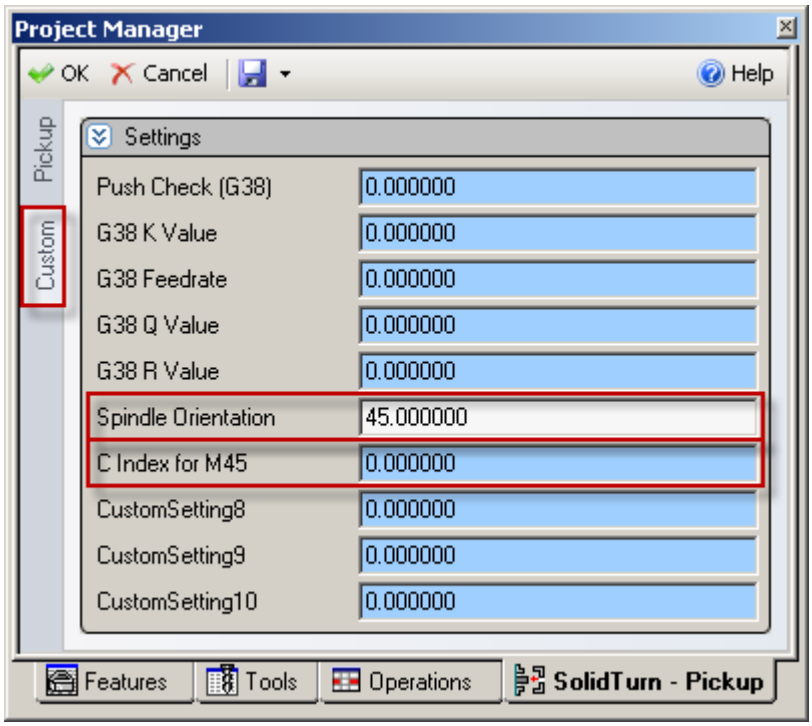


   The pickup operation has to be programmed with lower (upper right for NZXDL configuration) turret because it controls the B-axis (spindle 2).

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

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



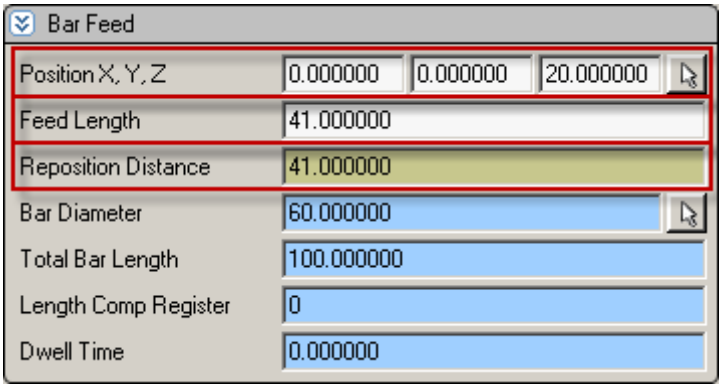


Note that the milling C-axis mode is the default. So if you omit to enter a value in the Spindle Orientation field or enter a wrong value, the spindles will be indexed using this mode.

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

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

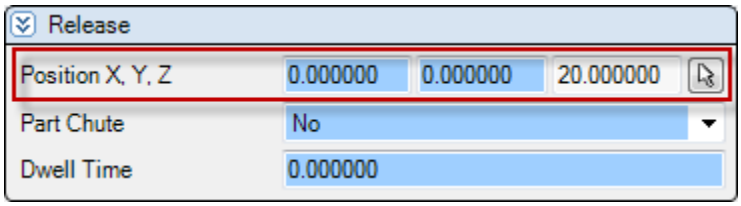
   With Position X, Y, Z, **set the same point as for the pickup operation** (If the point is different, chucks will unclamp and clamp before pulling the bar).



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

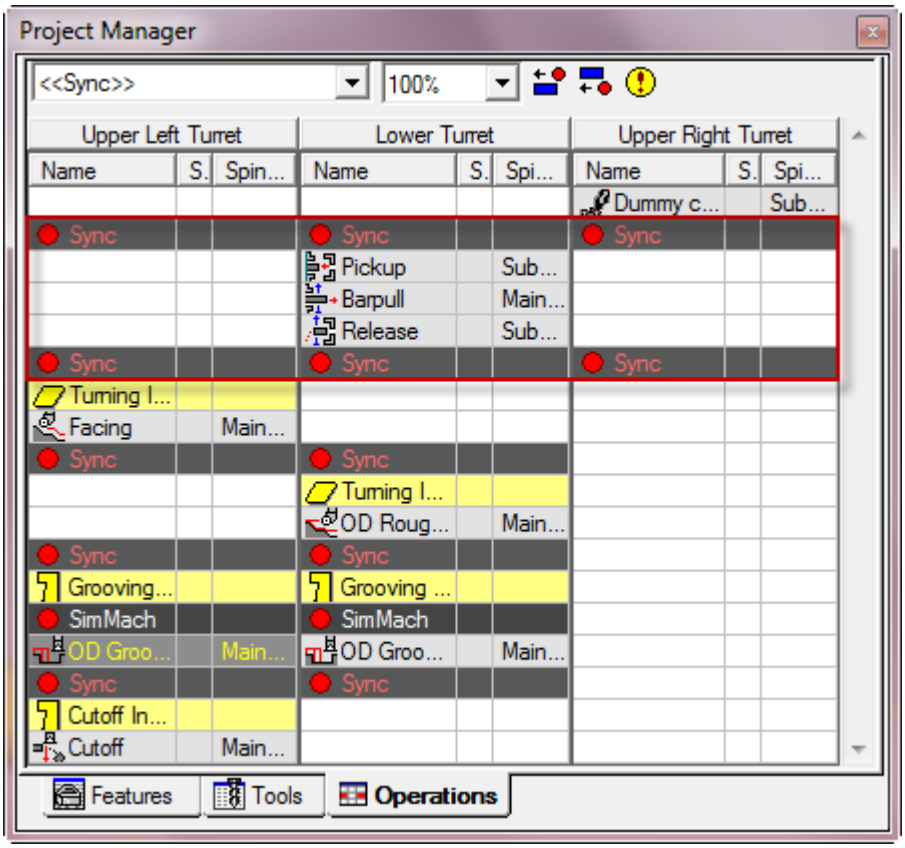
On the General tab, enter the Feedrate PM of the spindle 2 B-axis, it will be output in head 2 program (since head 2 controls the B-axis).

4. Releasing the part from the spindle 2:
   Use operation **Release** with **SubSpindle** (**Spindle Name**) on **Lower** (**Upper Right** for NZXDL configuration) **Turret** (**Turret Name**). With Position X, Y, Z, set the release point.

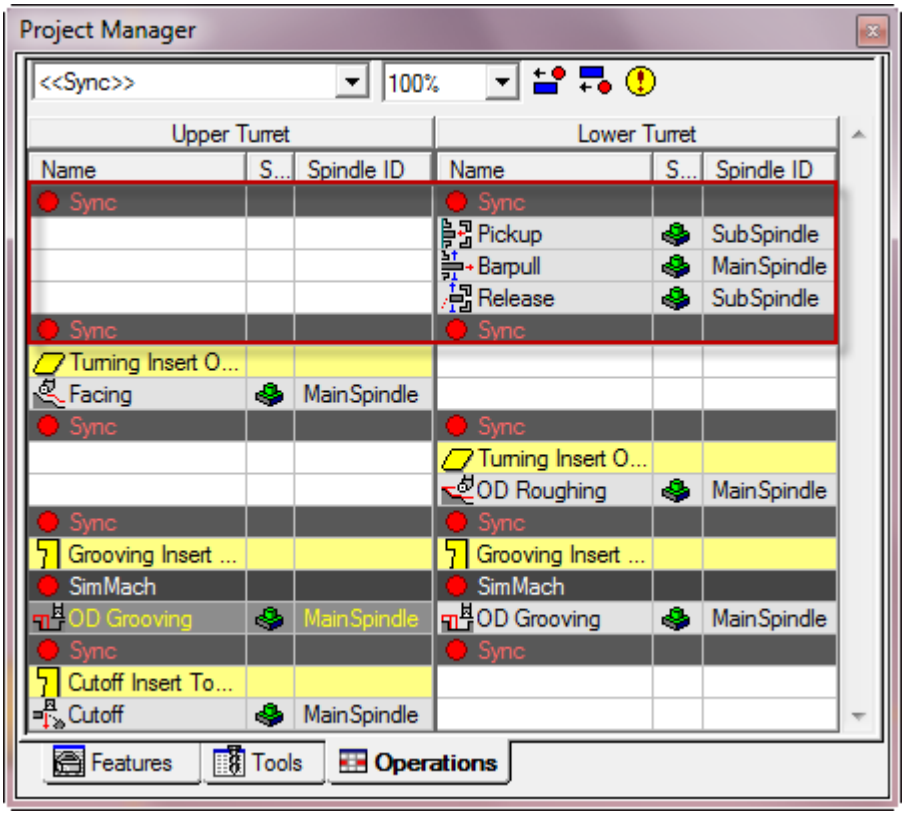


5. Program a wait code across all heads.

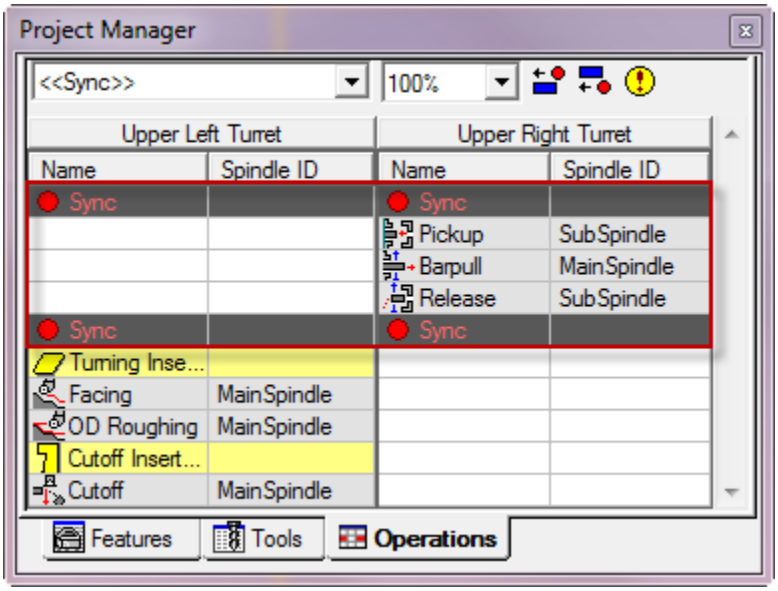
<u>Sample operation list for NZX-ST configuration:</u>



Sample operation list for NZX-S configuration:



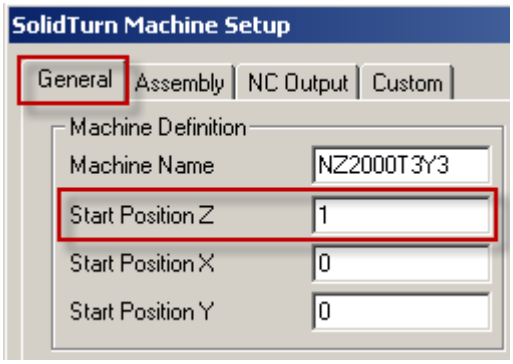
Sample operation list for NZXDL configuration:



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

To support the stock with both spindles, you first need to pick it up (and, if needed, pull it) with the spindle 2. You can after program your operations (only in head 1 for NZXDL configuration) and in the code will be output the necessary spindles synchronization and unsynchronization codes. Finally, once machining is completed, you can program one of the transfer scenarios described in the previous parts at the end of the operation list.

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



The regular steps to follow are:

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

Steps to program this transfer type in ESPRIT:

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

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

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

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

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).

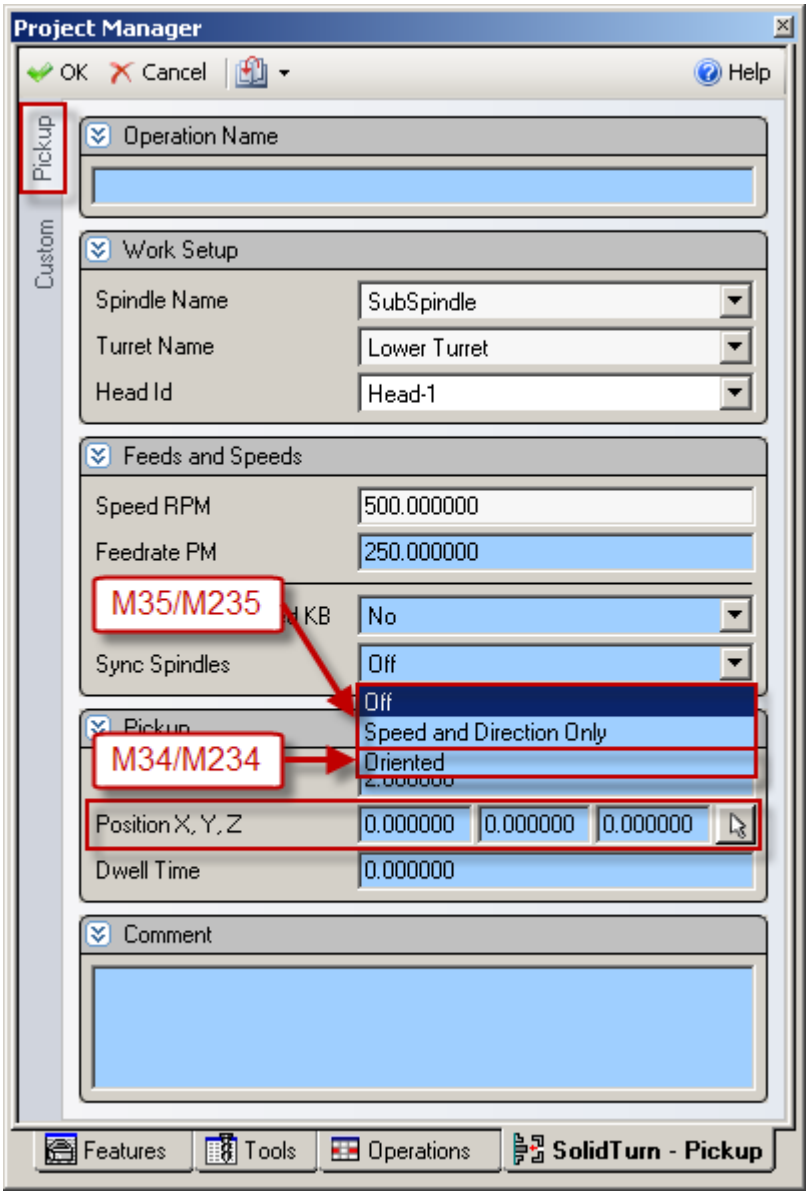
3. Program a wait code across all heads.

4. Picking the part in the spindle 1 with the spindle 2:
   Use operation **Pickup** with **SubSpindle** (**Spindle Name**) on **Lower** (**Upper Right** for NZXDL configuration) **Turret** (**Turret Name**). With Position X, Y, Z, set the pickup point.

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

   Based on the following operation in head 1 (or 2 for NZX-ST & NZX-S configurations), the correct synchronization code will be output in the code:
   - **M34** (or **M35**): Turning operation is next on head 1; the spindle speed entered on the SolidTurn Pickup page will be output on head 1 after synchronization of the spindles; the spindle direction code will be output on head 1 based on the direction of the following turning operation.
   - **M234** (or **M235**) for NZX-ST & NZX-S configurations only: Turning operation is next on head 2; the spindle speed entered on the SolidTurn Pickup page will be output on head 2 after synchronization of the spindles; the spindle direction code will be output on head 2 based on the direction of the following turning operation.
   - **M480**: Milling operation is next on head 1 (or 2 for NZX-ST & NZX-S configurations); the spindles are stopped.

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

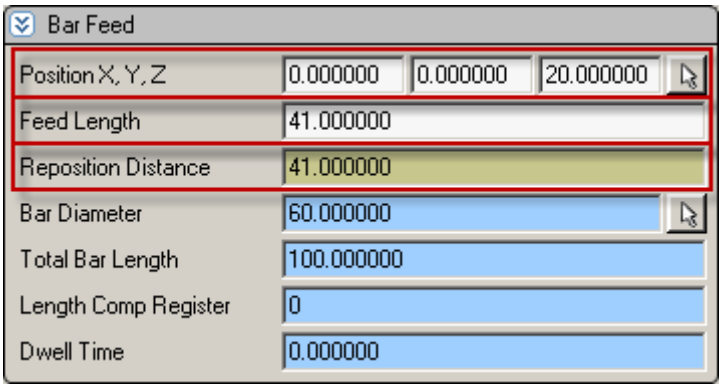


Note: The synchronization mode selected (as shown on picture above) on the pickup operation page will be used for all the following turning operations while the spindles are both holding the part.

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

   On Bar Feed tab, set the Feed Length and Reposition Distance:
   Feed Length (= Barpull distance = Reposition Distance)

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



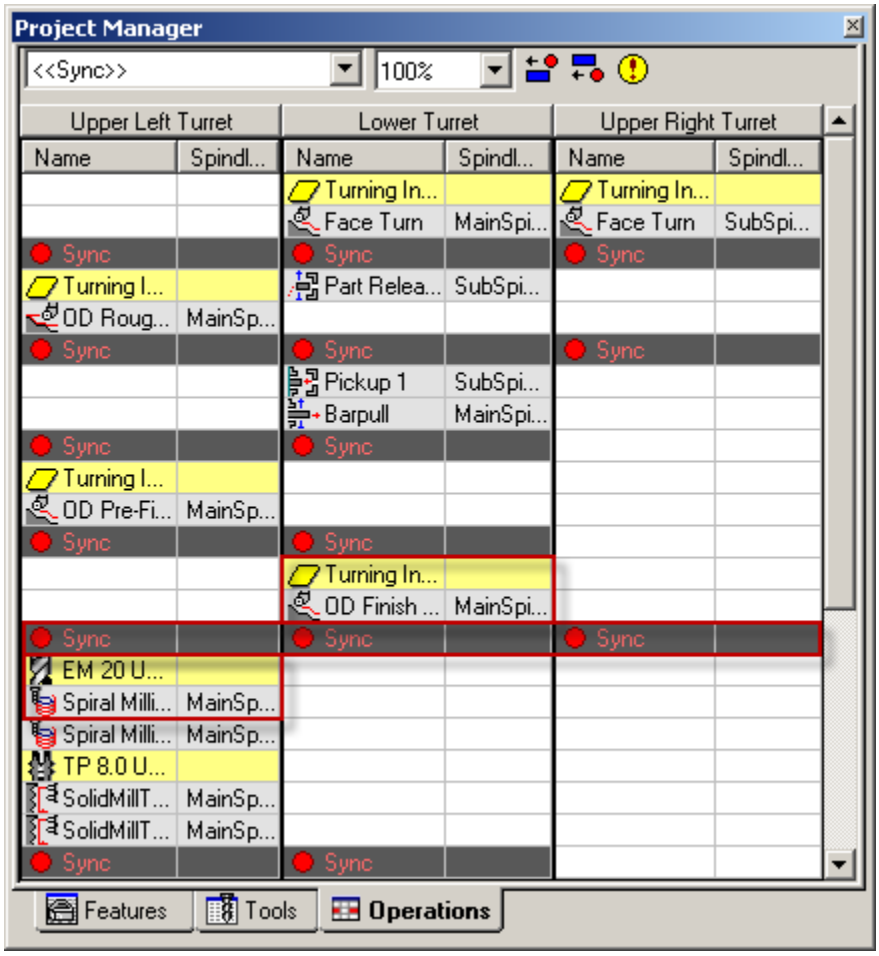
Programming a pickup before the barpull makes the spindle 2 stay at the same location after the barpull. Without pickup operation, spindle 2 goes home after the barpull.

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

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

7. Machining the part on the spindle 1 side (only with head 1 for NZXDL configuration) while the spindles are synchronized and are both supporting the part:
   The correct synchronization (and unsynchronization) codes will be output in the NC code based on the programmed operations.

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

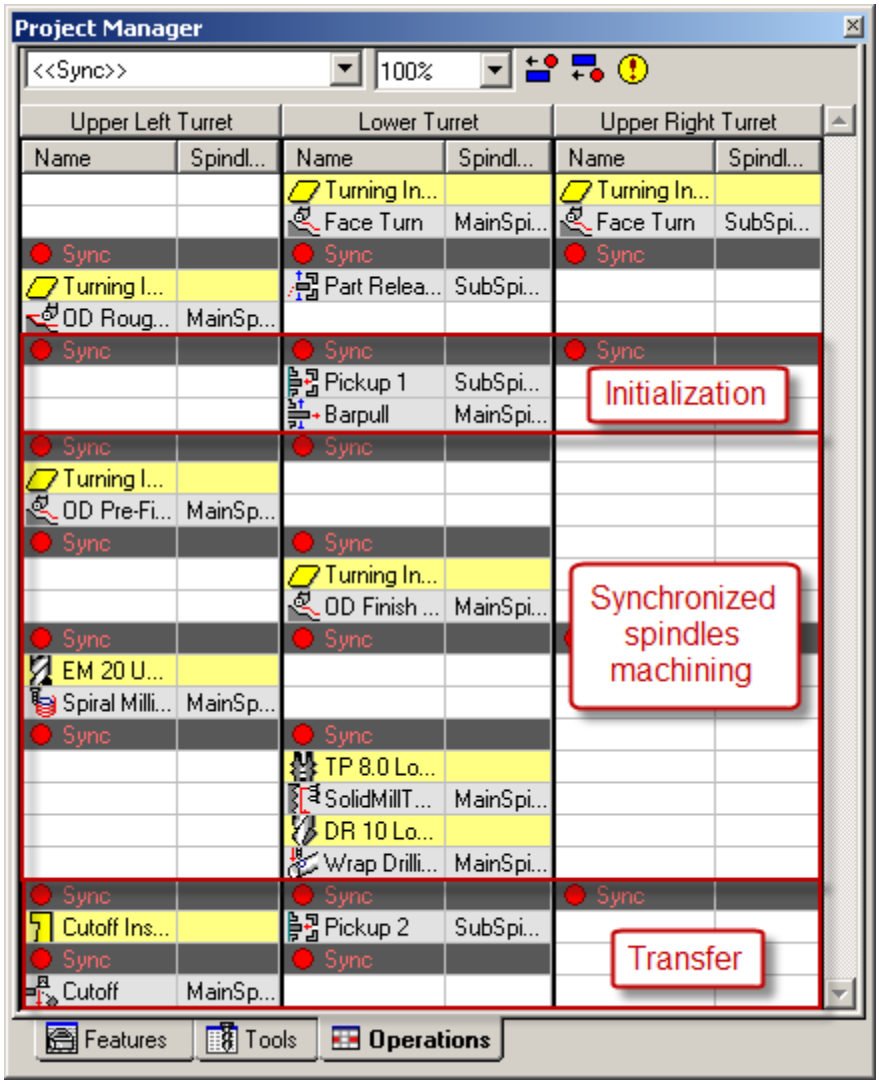


Also note that, for NZX-ST & NZX-S configurations, in M480 mode, C-axis values of milling operations programmed on head 2 cannot be output in head 2 and so will be output in head 1.

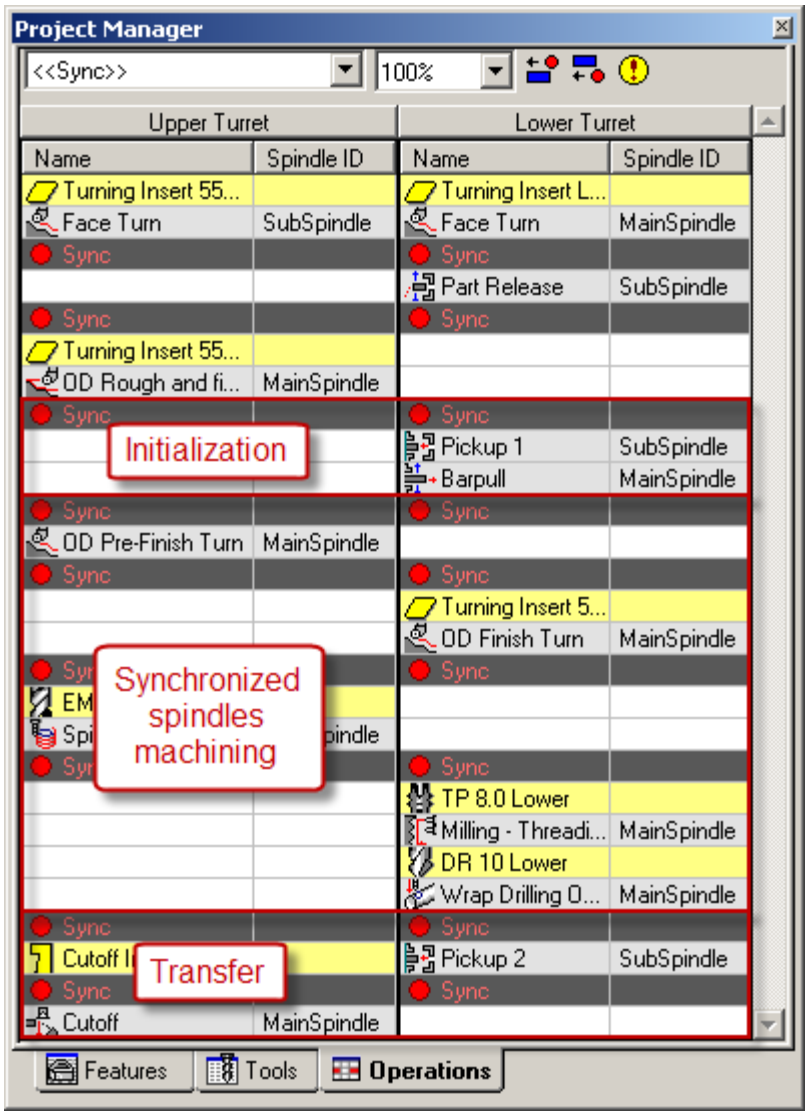
8. Program a wait code across all heads.

9. Perform one of the previously described transfer scenarios to transfer the part to spindle 2 side.

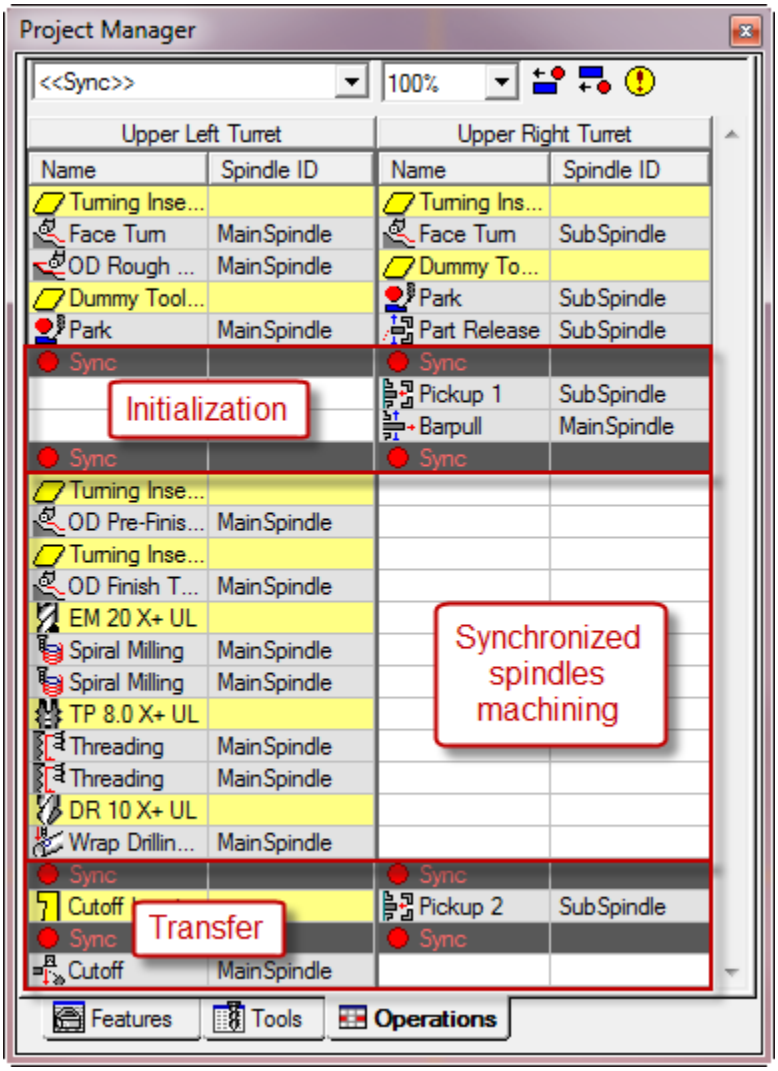
Sample operation list for NZX-ST configuration:



Sample operation list for NZX-S configuration:



Sample operation list for NZXDL configuration:



11.3.8. Workpiece Pushing Check (G38)

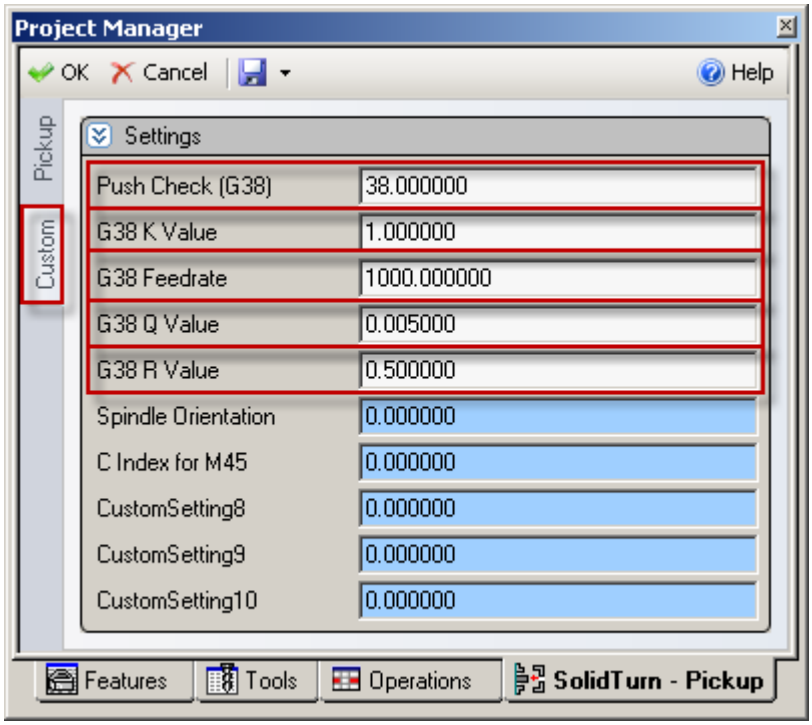
G38 B_ K_ F_ Q_ R_ ;

The G38 command can be specified when transferring a workpiece from chuck 1 to chuck 2 (or from chuck 2 to chuck 1).
In the workpiece transfer operation, when the position error of the B-axis servomotor of headstock 2 reaches the parameter set value, the workpiece is judged to have been brought into contact with the reference face of chuck 2 or chuck 1. Then, chuck 2 or chuck 1 clamps the workpiece and the program advances to the next block.

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

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

You can specify in the G38 K Value (Custom Setting 2) field the value of K (B-axis retraction stroke, unsigned value). If no value is entered, the B-axis is automatically retracted by the servo error amount in the uni-direction positioning mode.
You can specify in the G38 Feedrate (Custom Setting 3) field the value of F (B-axis feedrate (mm/min) from R). If no value is entered, 30 mm/min will be used by the machine.
You can specify in the G38 Q Value (Custom Setting 4) field the value of Q (Tolerance for workpiece transfer position). If no value is entered, 1 mm will be used by the machine.
You can specify in the G38 R Value (Custom Setting 5) field the value of R (B-axis feedrate changing position). If no value is entered, 1 mm will be used by the machine.



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

11.3.9. Park turret for transfer

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

12. Multiple repetitive cycles

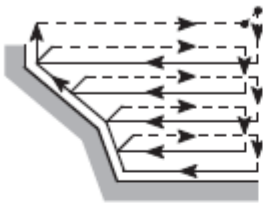
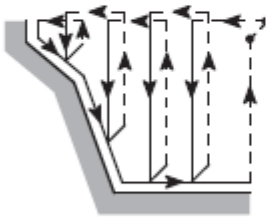
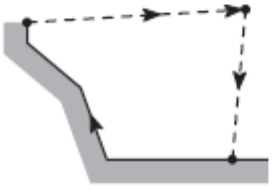
12.1. Introduction

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

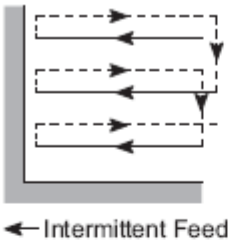
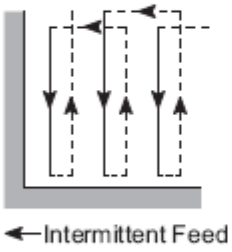
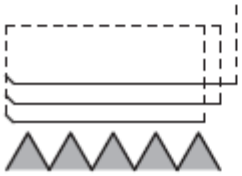
Roughing processes that require several blocks of commands can be specified by a single block of commands preceded by a G code calling a multiple repetitive cycle, and blocks that define the finished shape. The tool paths for rough cutting cycles are automatically determined.

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

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

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

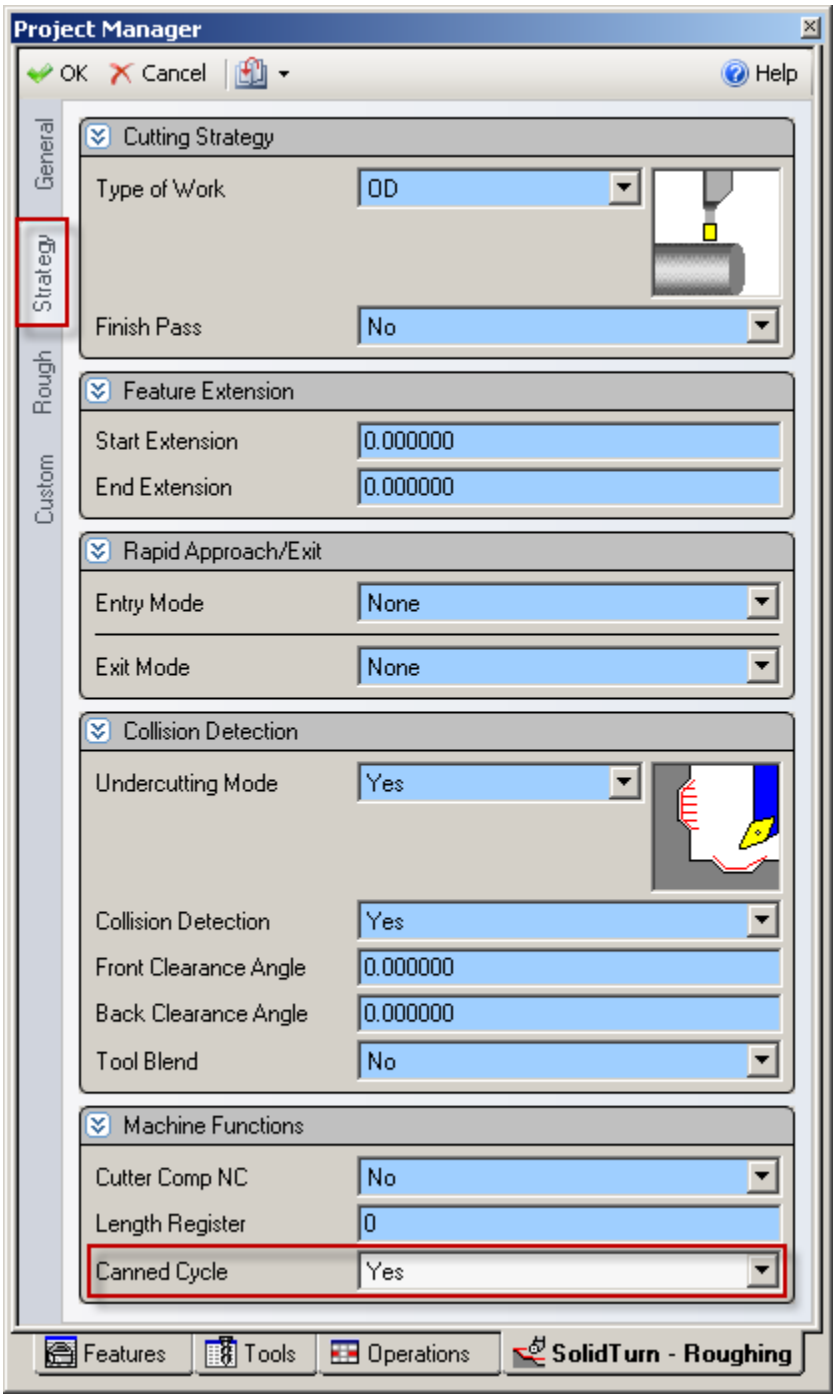
O.D./I.D. groove, face groove and threading cycles:

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

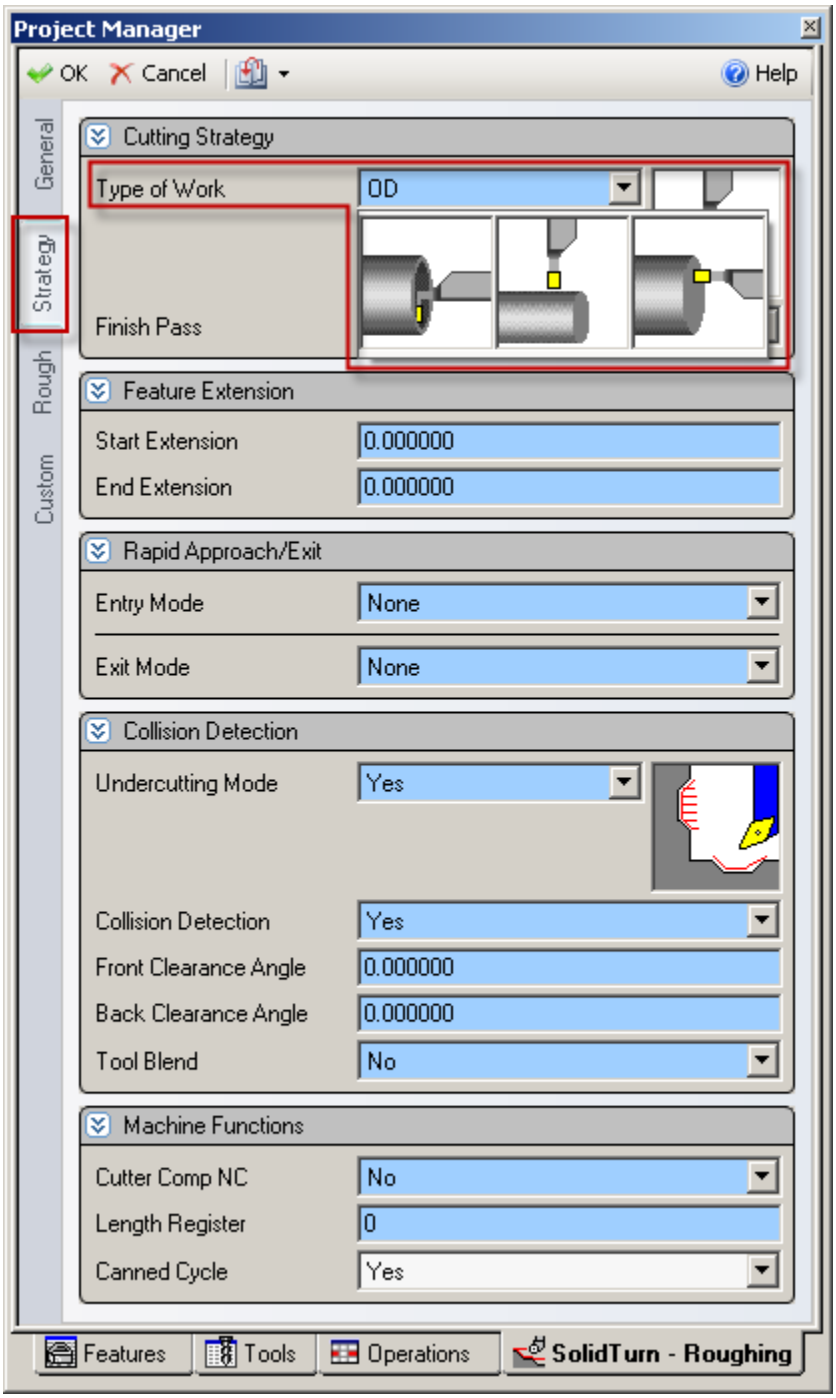
12.2. Roughing cycle

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.

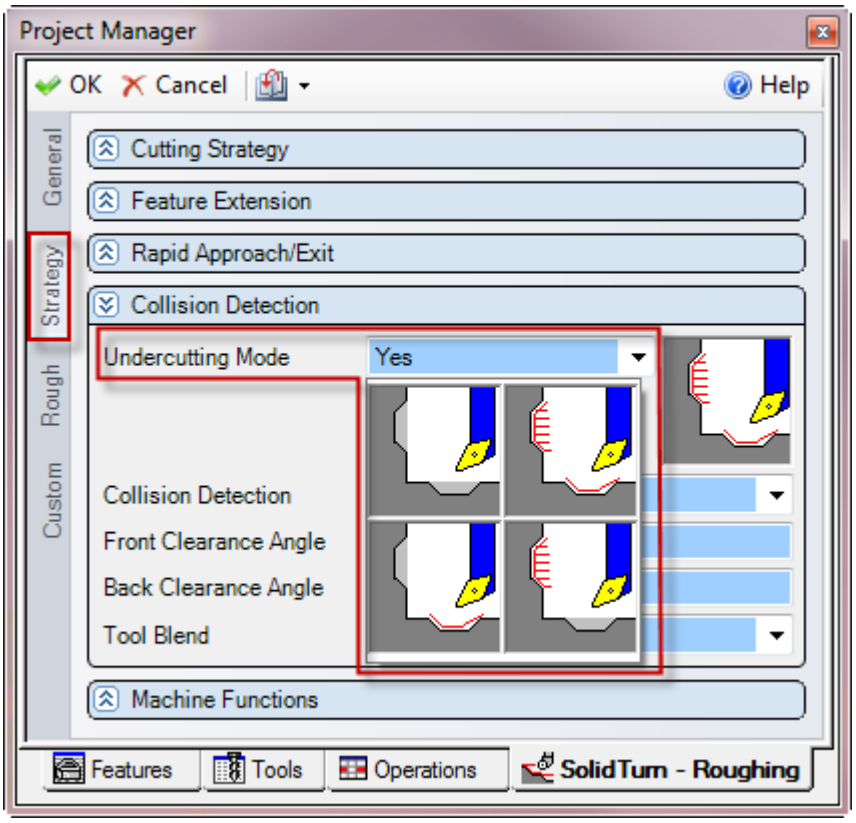


Based on the selected work strategy (Type of Work), the output G-code will be different. G71 will be output if your profile is on the O.D. or the I.D. of the part. G72 will be output if you cut on the face of the part.



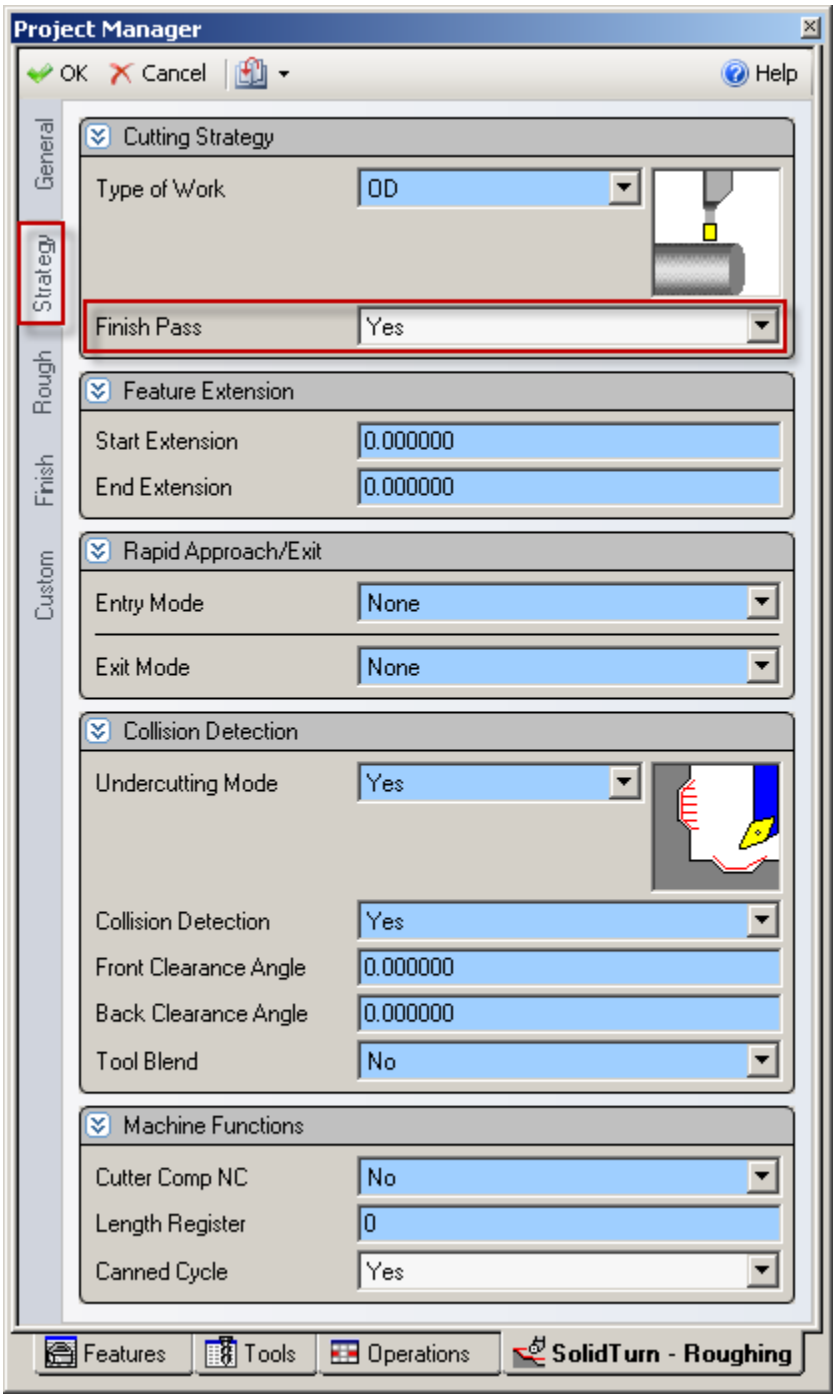
Note: Pockets on a turning profile will be cut by the roughing canned cycle, if Undercutting Mode on the Strategy tab is set to:

- **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.

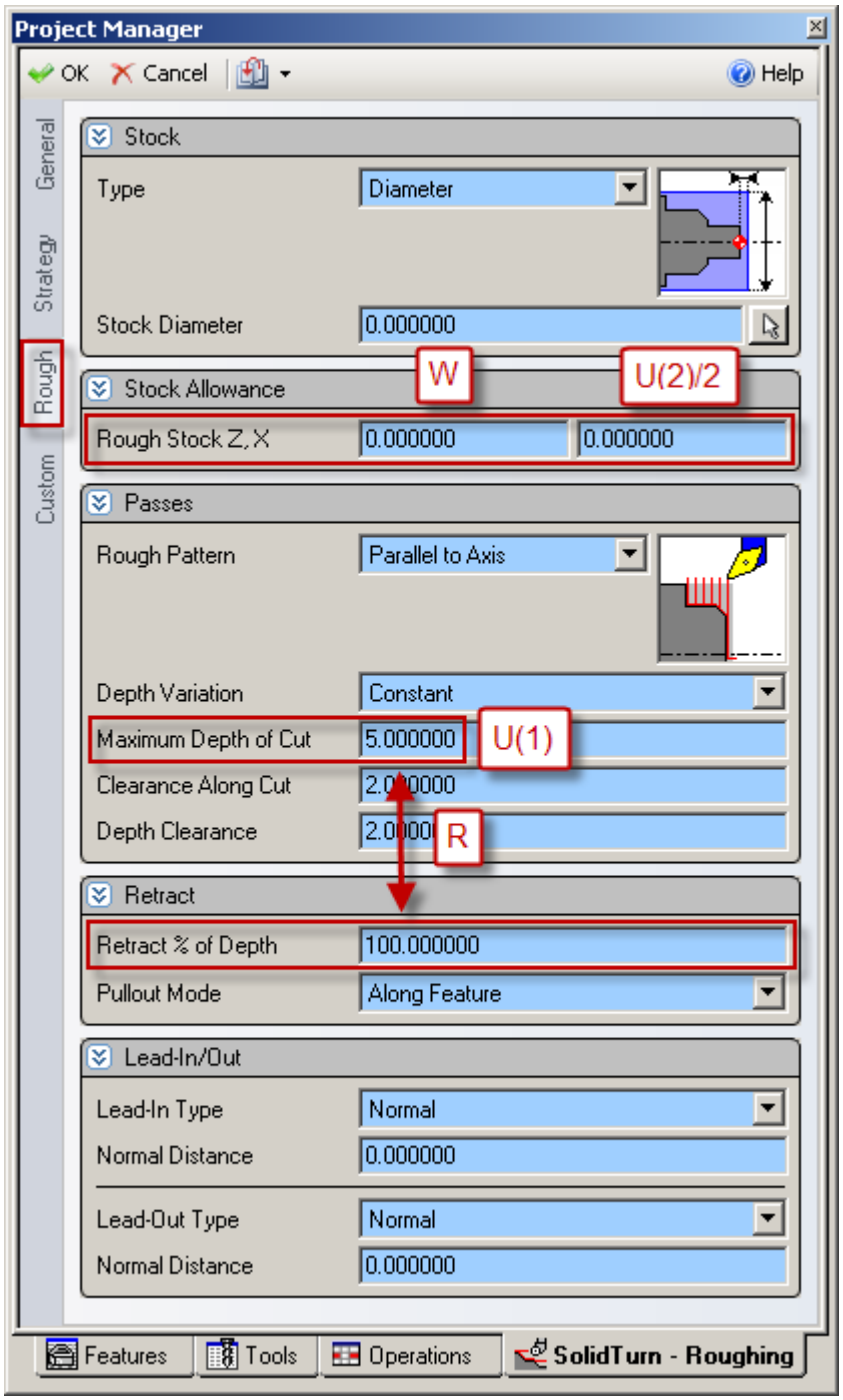


Information about some of the parameters to enter for the following cycles:

- 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_ ;

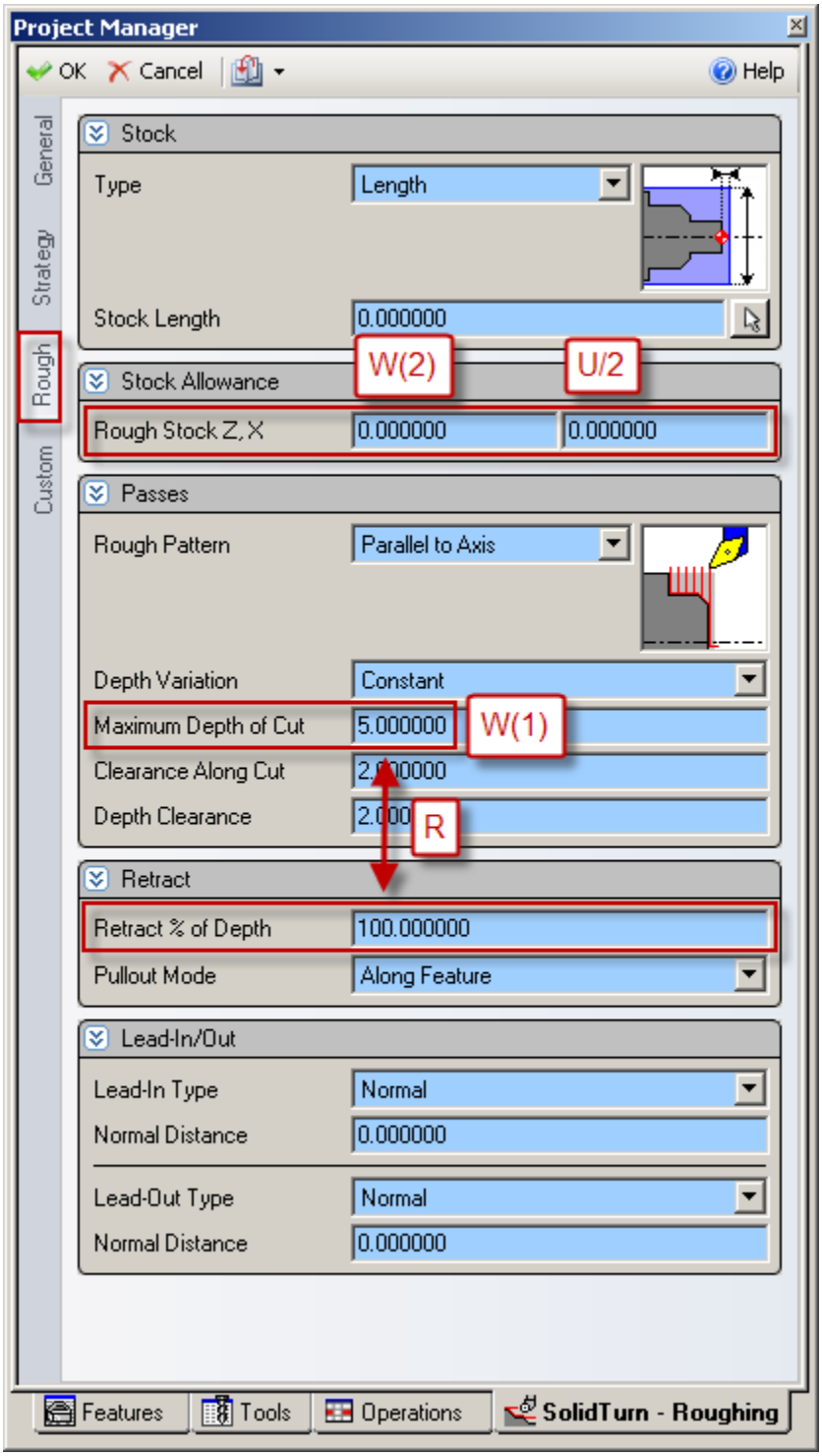


U(1) and **W** will be output as entered in ESPRIT. **U(2)/2** will be multiplied by 2 to have the correct value in the NC code **U(2)**. **R** will be computed based on the entered

Maximum Depth of Cut and Retract % of Depth value: **R** = Maximum Depth of Cut * Retract % of Depth / 100.
For example, if you set Maximum Depth of Cut to 5 and Retract % of Depth to 75, R output in the NC code will be R3.75 (5*75/100).

12.2.3. Face roughing

G72 W(1) R_ ;
G72 P_ Q_ U_ W(2) F_ ;

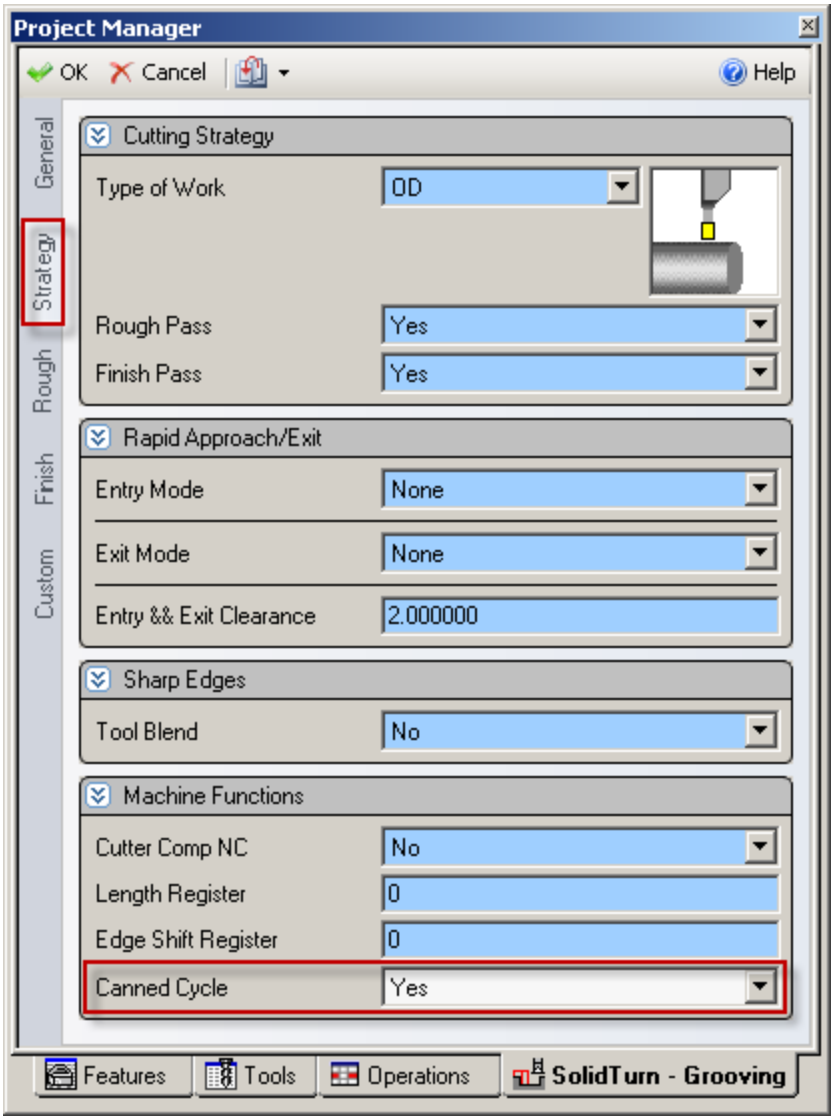


W(1) and **W(2)** will be output as entered in ESPRIT. **U/2** will be multiplied by 2 to have the correct value in the NC code **U**. **R** will be computed based on the entered Maximum Depth of Cut and Retract % of Depth value: **R** = Maximum Depth of Cut * Retract % of Depth / 100.

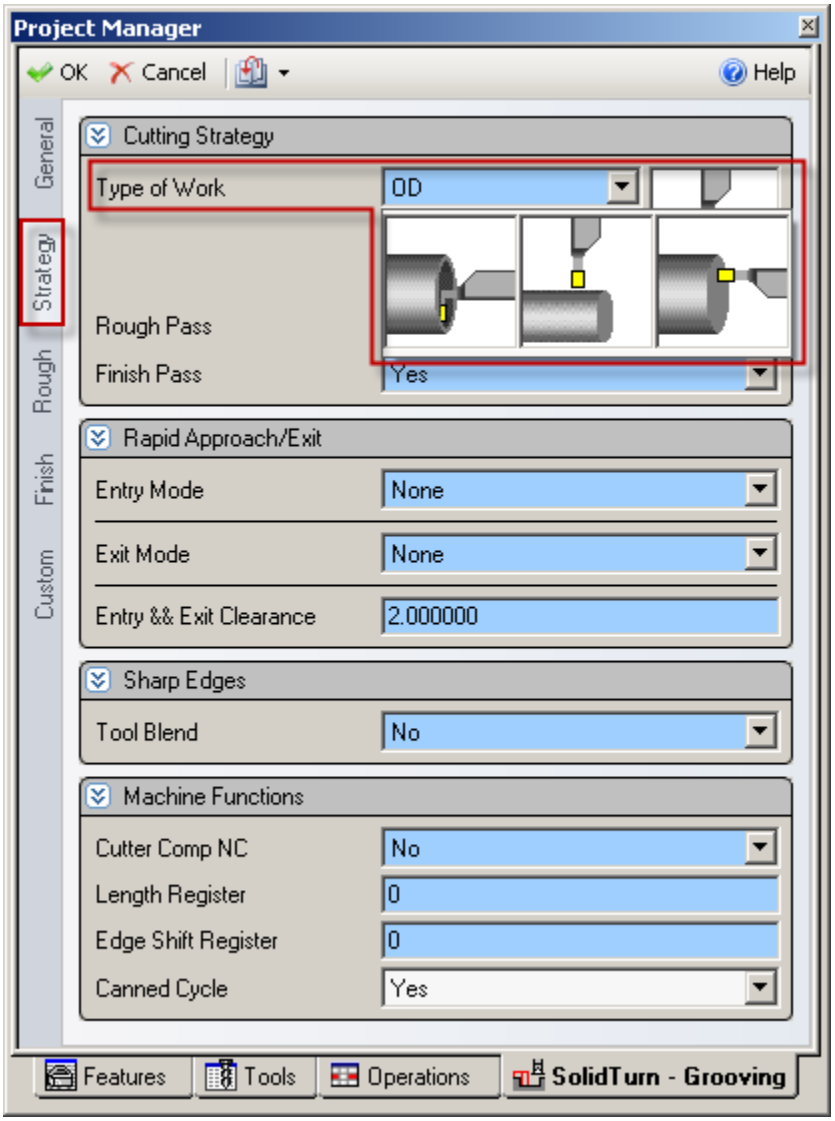
12.3. Grooving cycle

12.3.1. Introduction

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



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.

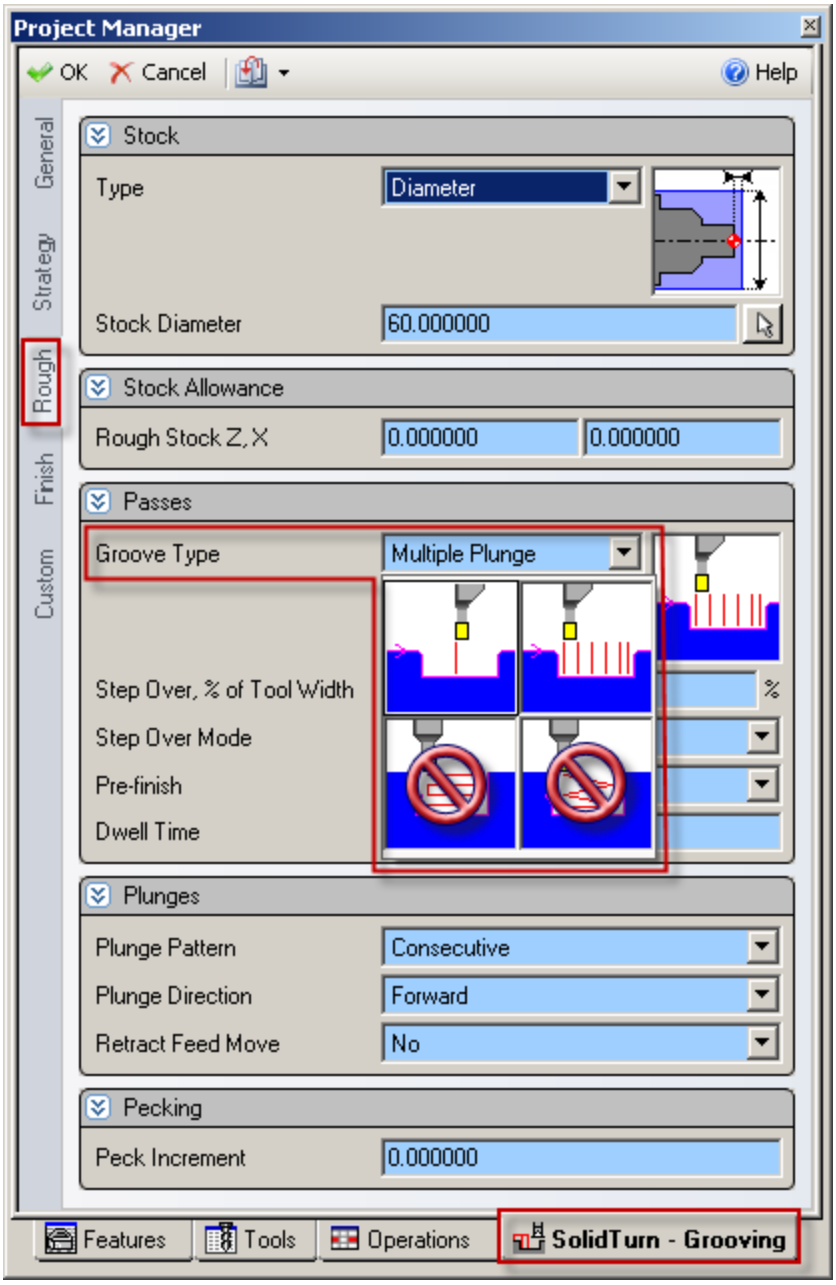


Information about some of the parameters to enter for the following cycles:

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

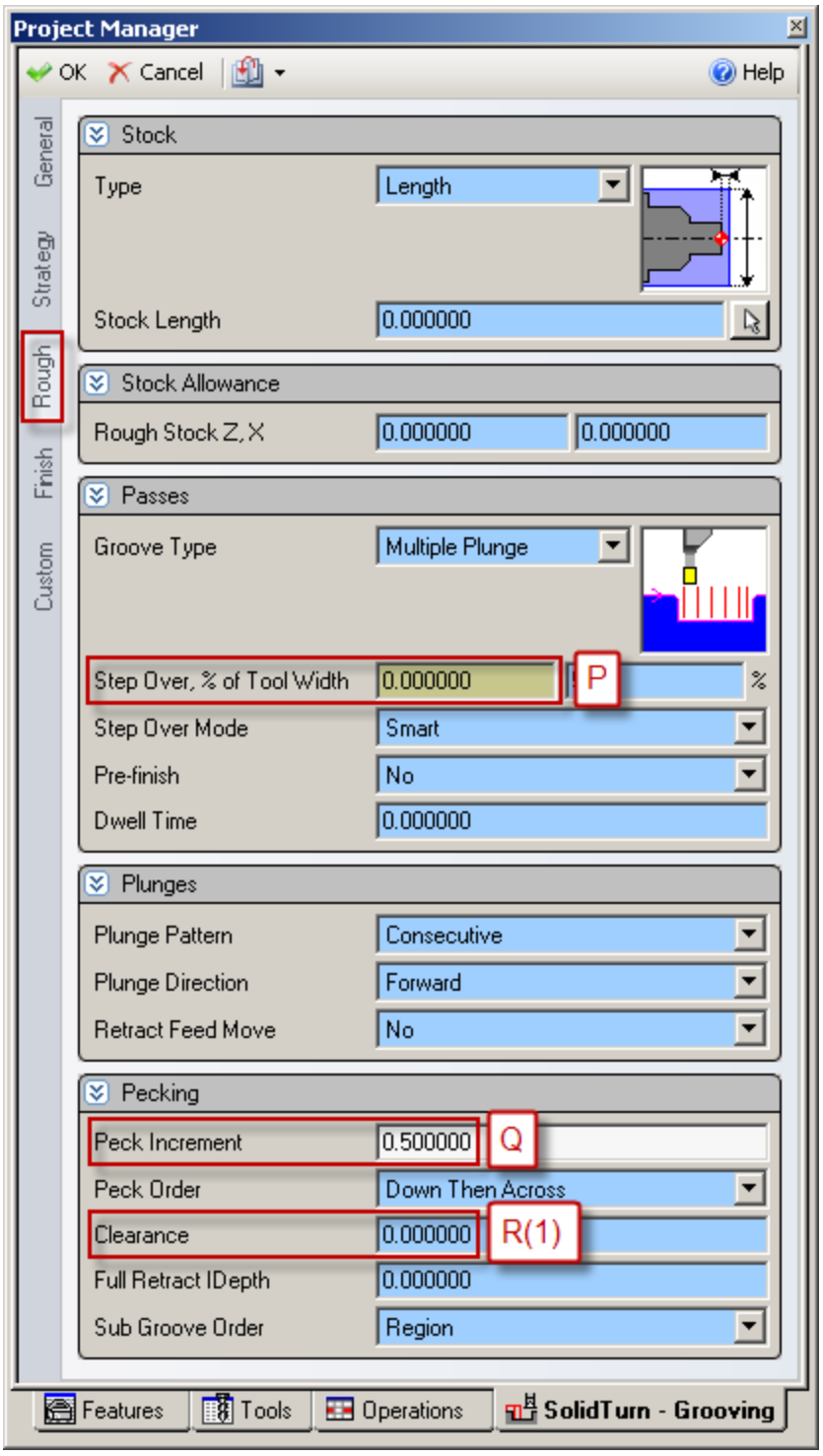
Notes: Grooving canned cycles can only be used on grooves with straight wall geometry. Grooves with a side taper angle must be programmed with a regular groove operation.

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`.



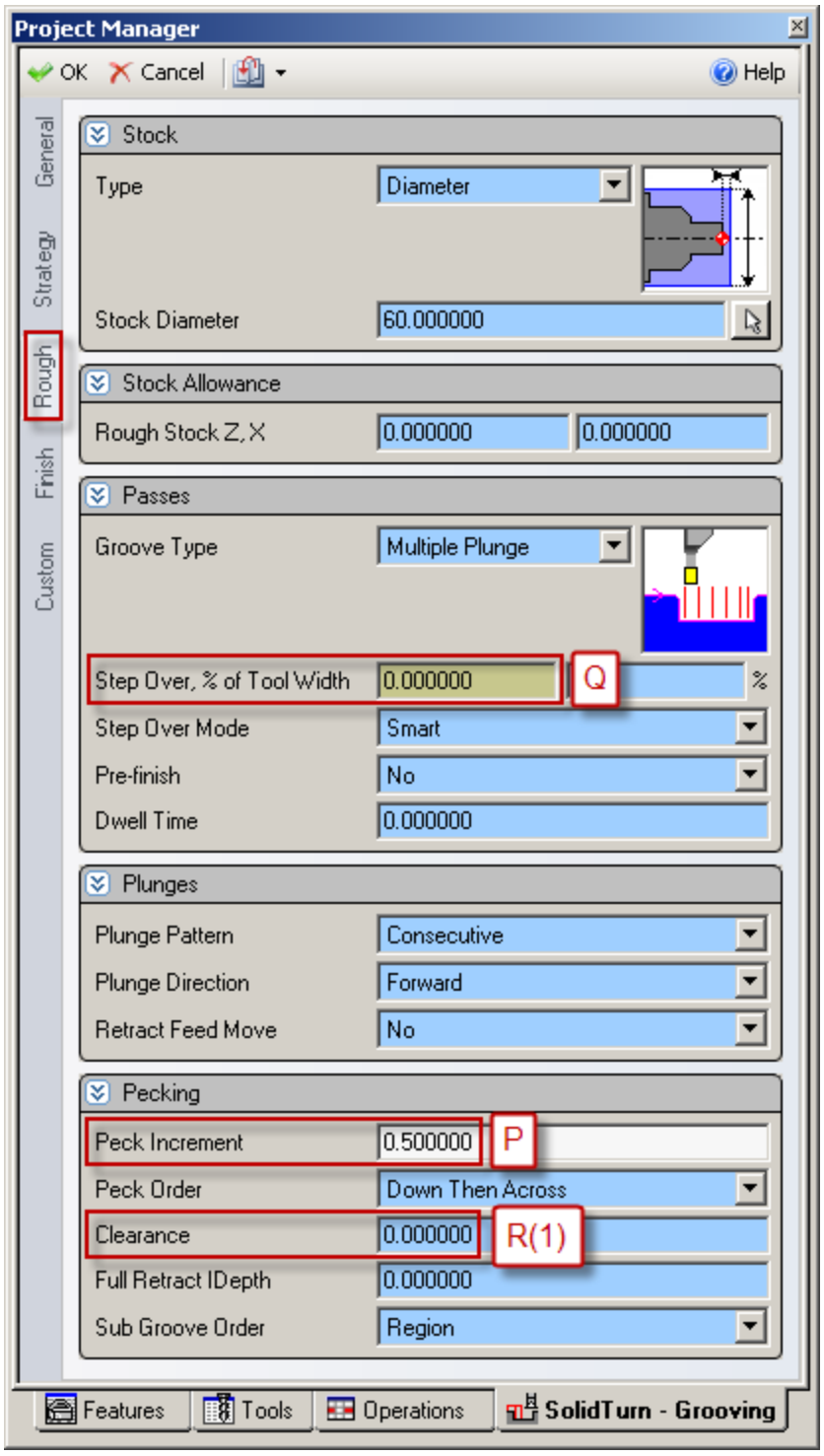
12.3.2. Face grooving

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



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

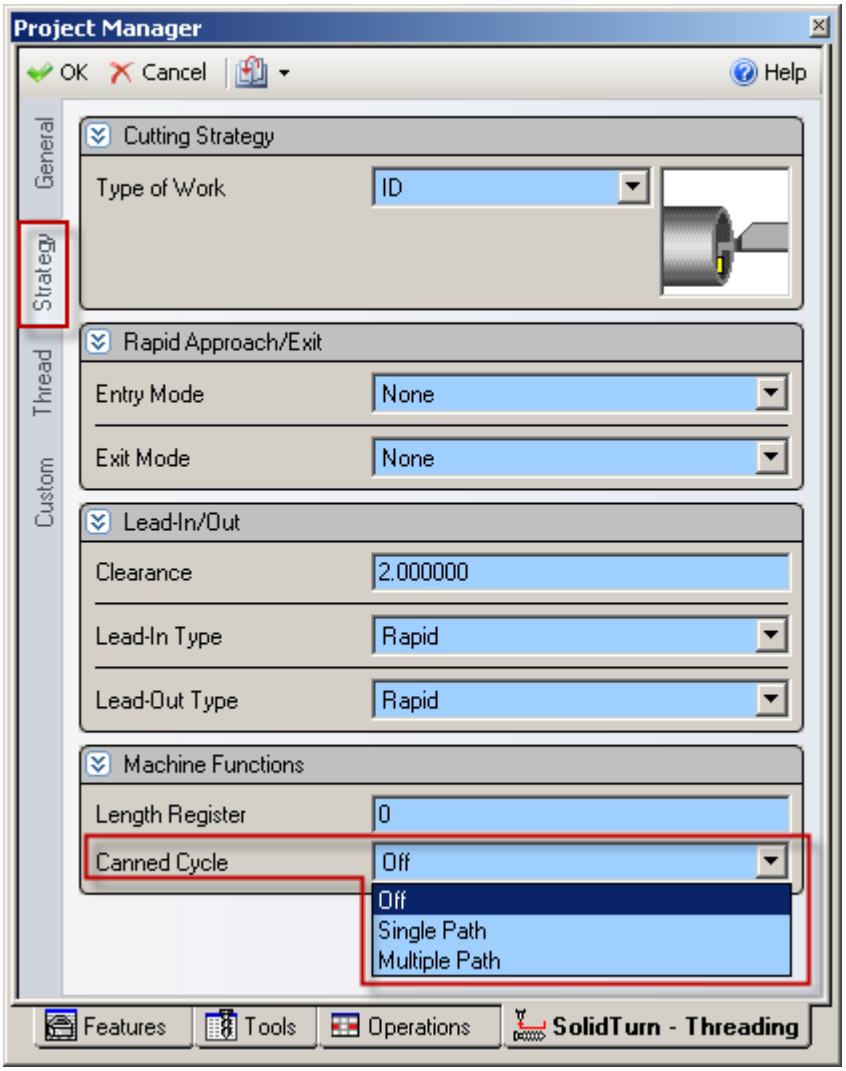
G75 R(1) ;
G75 X_ Z_ P_ Q_ F_ ;


Project Manager
OK
Cancel
Help
General
Strategy
Rough
Finish
Custom
Stock
Type
Diameter
Stock Diameter
60.000000
Stock Allowance
Rough Stock Z, X
0.000000
0.000000
Passes
Groove Type
Multiple Plunge
Step Over, % of Tool Width
0.000000
Q
%
Step Over Mode
Smart
Pre-finish
No
Dwell Time
0.000000
Plunges
Plunge Pattern
Consecutive
Plunge Direction
Forward
Retract Feed Move
No
Pecking
Peck Increment
0.500000
P
Peck Order
Down Then Across
Clearance
0.000000
R(1)
Full Retract IDepth
0.000000
Sub Groove Order
Region
Features
Tools
Operations
SolidTurn - Grooving

12.4. Threading cycle

12.4.1. Introduction

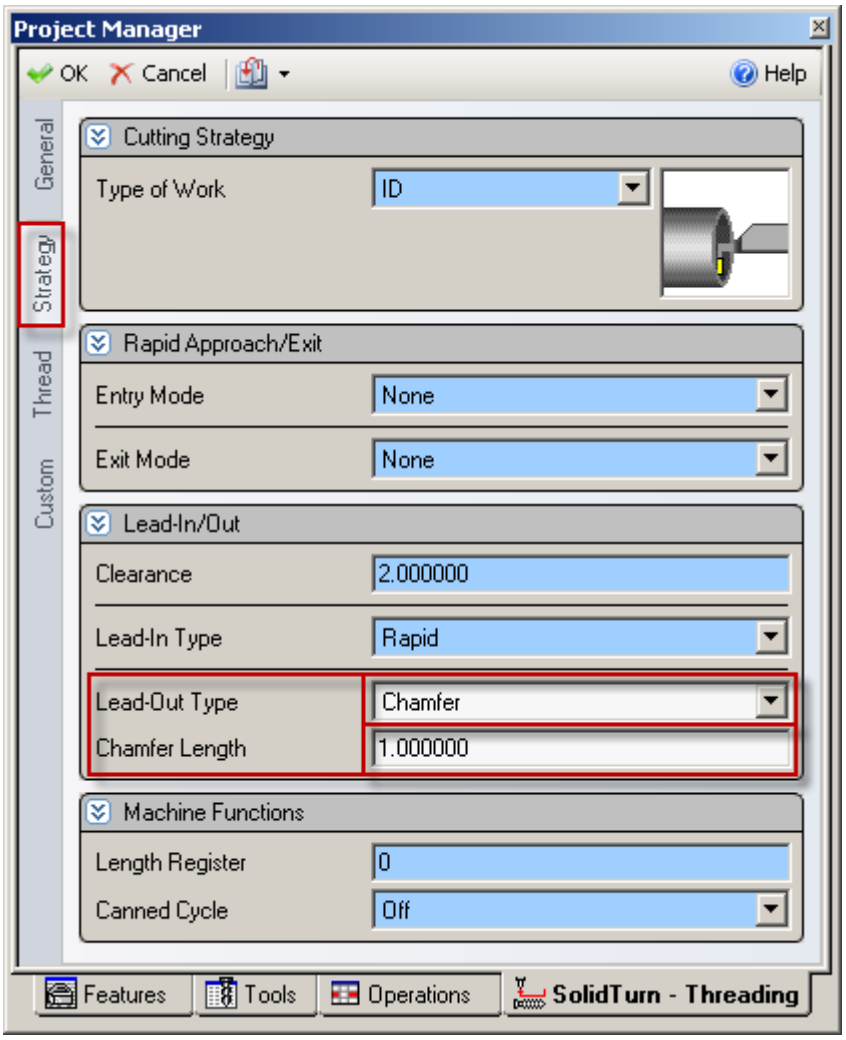
In ESPRIT, on the Strategy tab of the SolidTurn Threading operation page, set **Canned Cycle** to **Single Path** to trigger the G76 multiple repetitive cycle output. If you set this parameter to **Off**, the threading cycle will be output with G32 or G34 if Lead Variation is different from 0. When Canned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



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

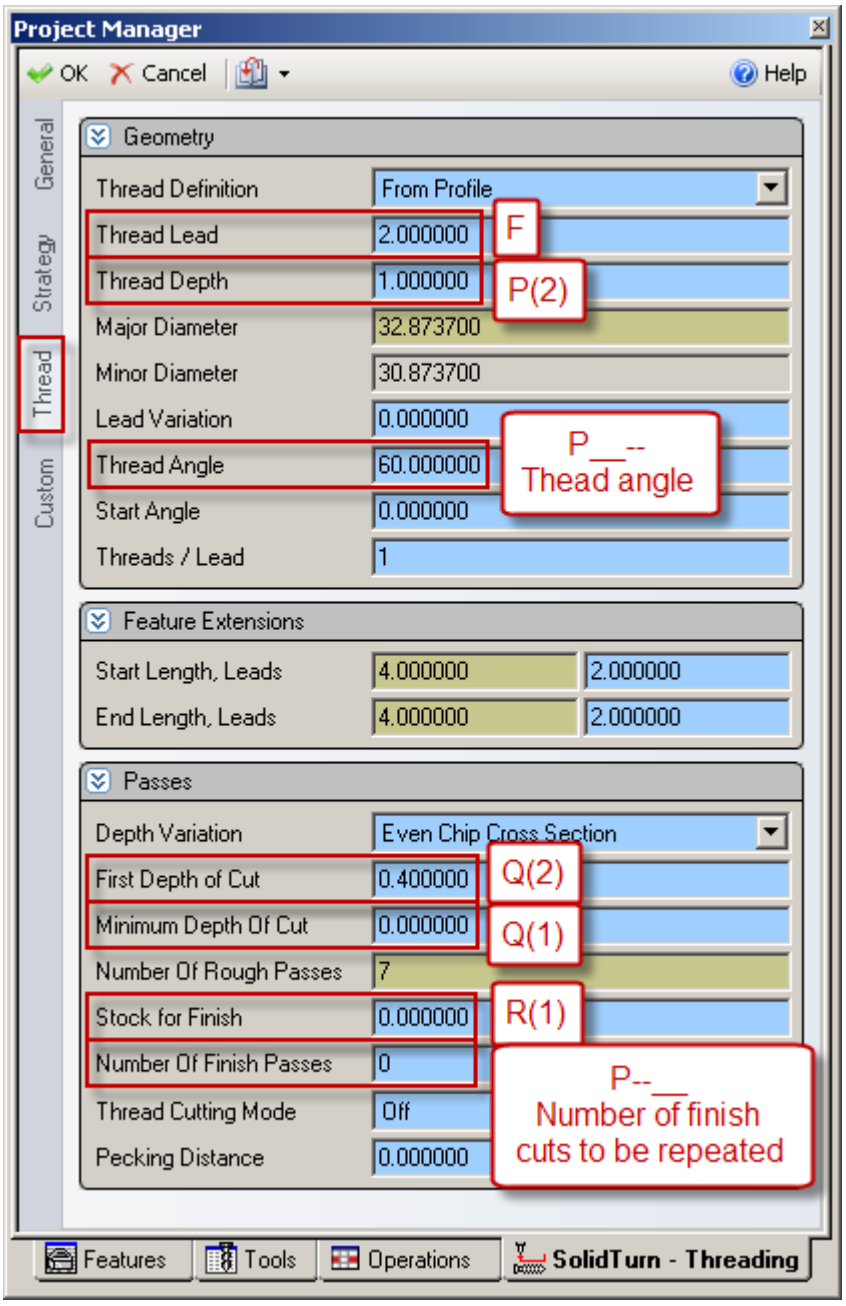
- The lead of the thread **F** will be output as entered in ESPRIT.
- Specify the angle of shift of the thread cutting start angle **Q** in °, it will then be output in the units of 0.001° without a decimal point. For example, if you enter 2 for Q in ESPRIT, the output will be in the NC code Q2000.
- The cut speed **S** can be specified on the General tab of the operation.

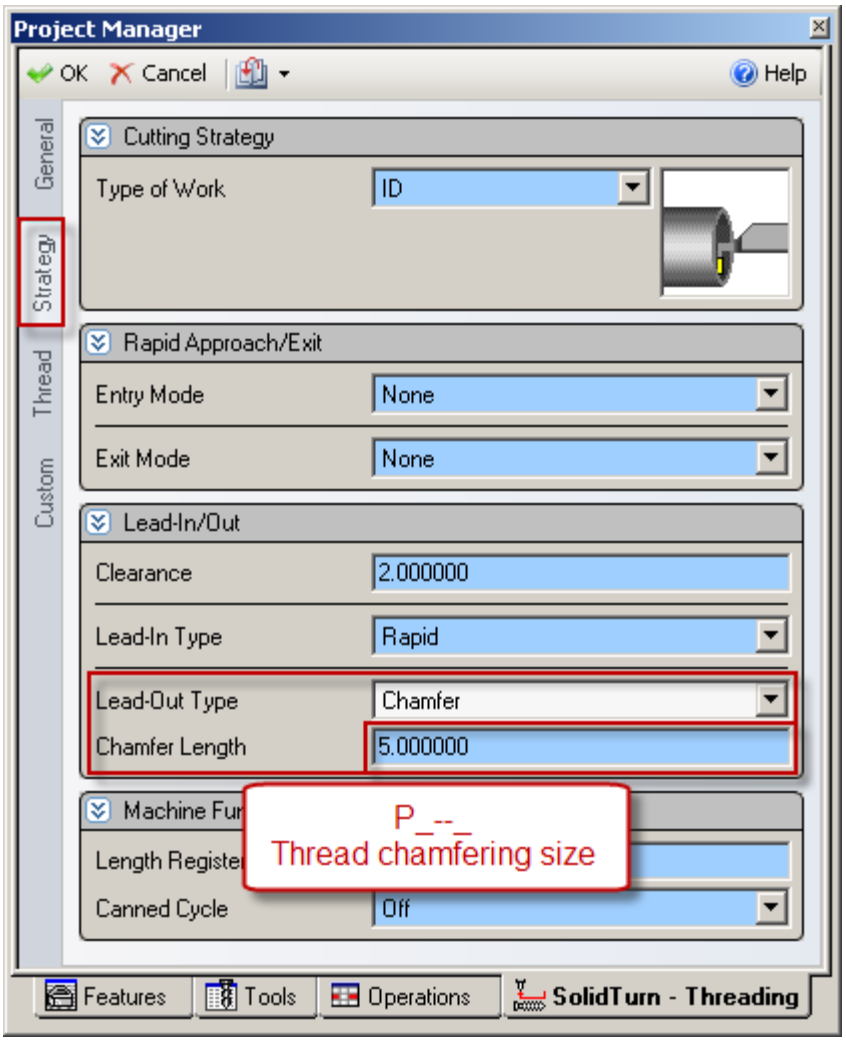
On the Strategy tab, if you set the Lead-Out Type to Chamfer and the Chamfer Length greater than 0, you will have **M23** (chamfer ON) output prior to the **G76** or **G92** commands. With a different lead-out set, the M-code output will be **M24** (chamfer OFF).



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_ ;

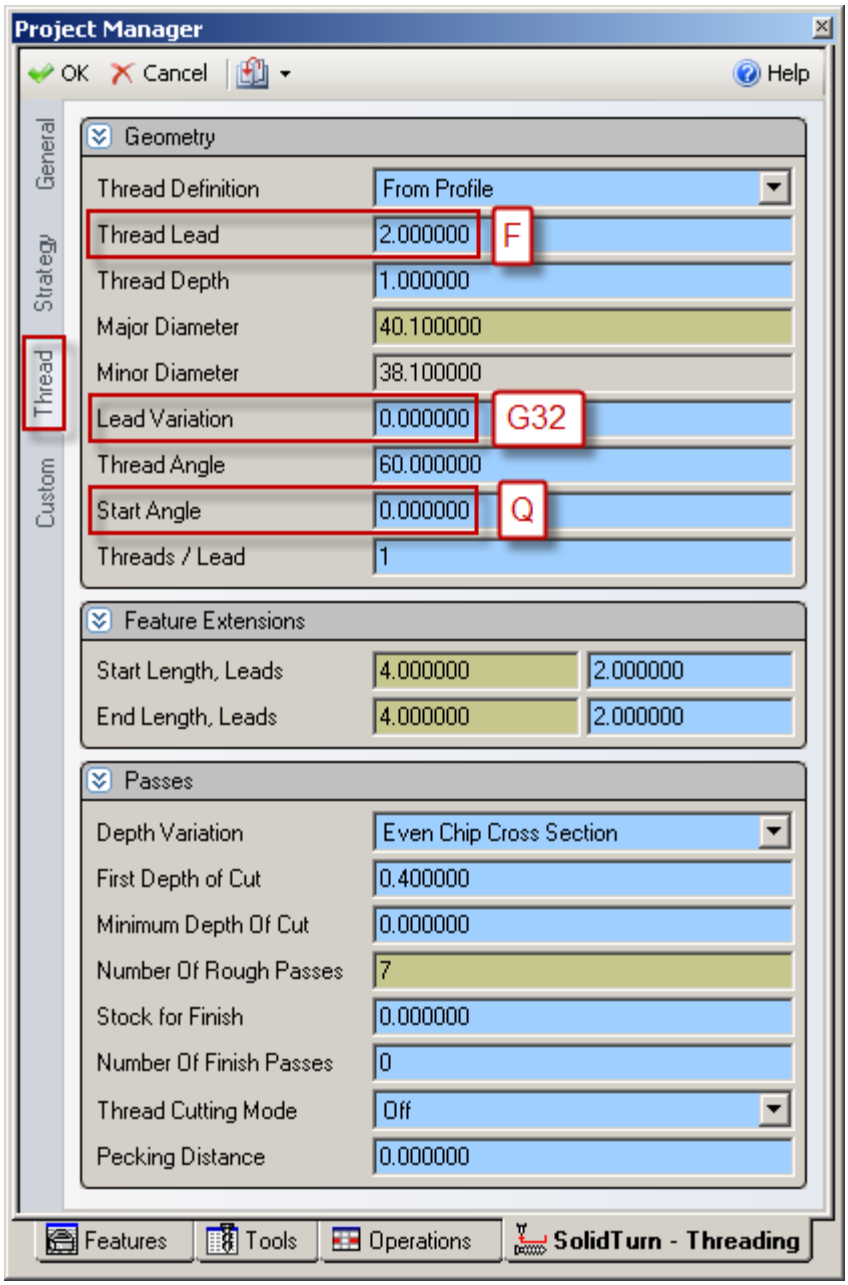

Project Manager
OK
Cancel
Help
General
Strategy
Thread
Custom
Geometry
Thread Definition
From Profile
Thread Lead
2.000000
F
Thread Depth
1.000000
P(2)
Major Diameter
32.873700
Minor Diameter
30.873700
Lead Variation
0.000000
Thread Angle
60.000000
P__--
Thead angle
Start Angle
0.000000
Threads / Lead
1
Feature Extensions
Start Length, Leads
4.000000
2.000000
End Length, Leads
4.000000
2.000000
Passes
Depth Variation
Even Chip Cross Section
First Depth of Cut
0.400000
Q(2)
Minimum Depth Of Cut
0.000000
Q(1)
Number Of Rough Passes
7
Stock for Finish
0.000000
R(1)
Number Of Finish Passes
0
P--__
Number of finish cuts to be repeated
Thread Cutting Mode
Off
Pecking Distance
0.000000
Features
Tools
Operations
SolidTurn - Threading



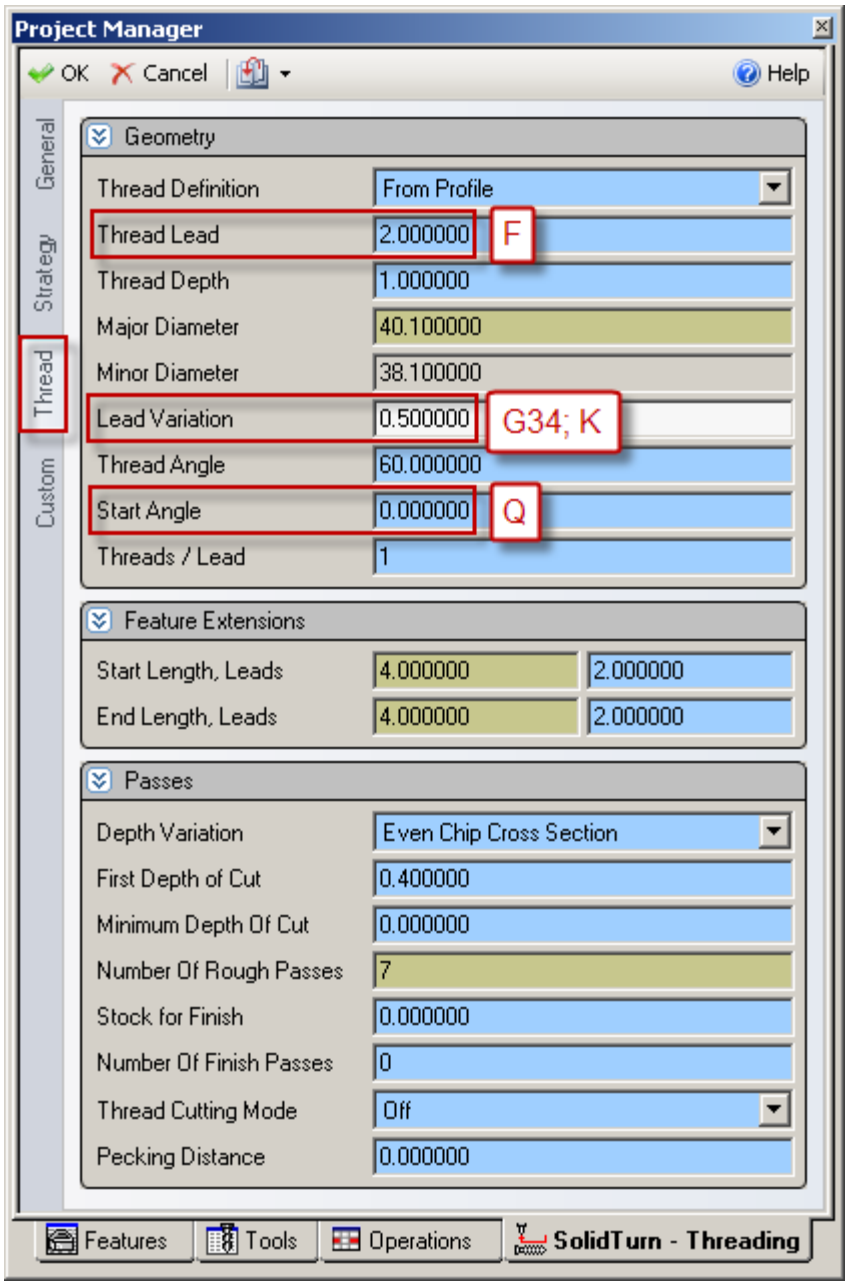
R(1) will be output as entered in ESPRIT. **Q(1)**, **P(2)** and **Q(2)** are unsigned values, in radius. Specify them in mm, they will then be output in units of 0.001 mm without decimal point. For example, if you enter 2 for Q(1) in ESPRIT, the output will be in the NC code Q2000.

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 (Option on machine):
G34 X_ Z_ F_ K_ Q_ ;

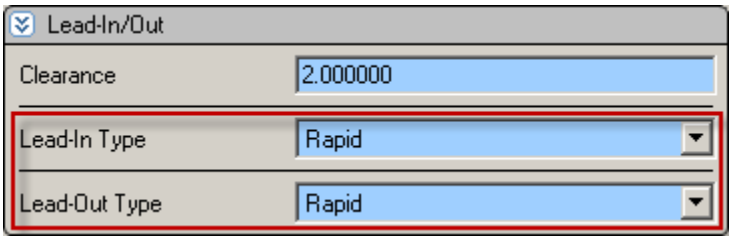


K will be output as entered in ESPRIT.

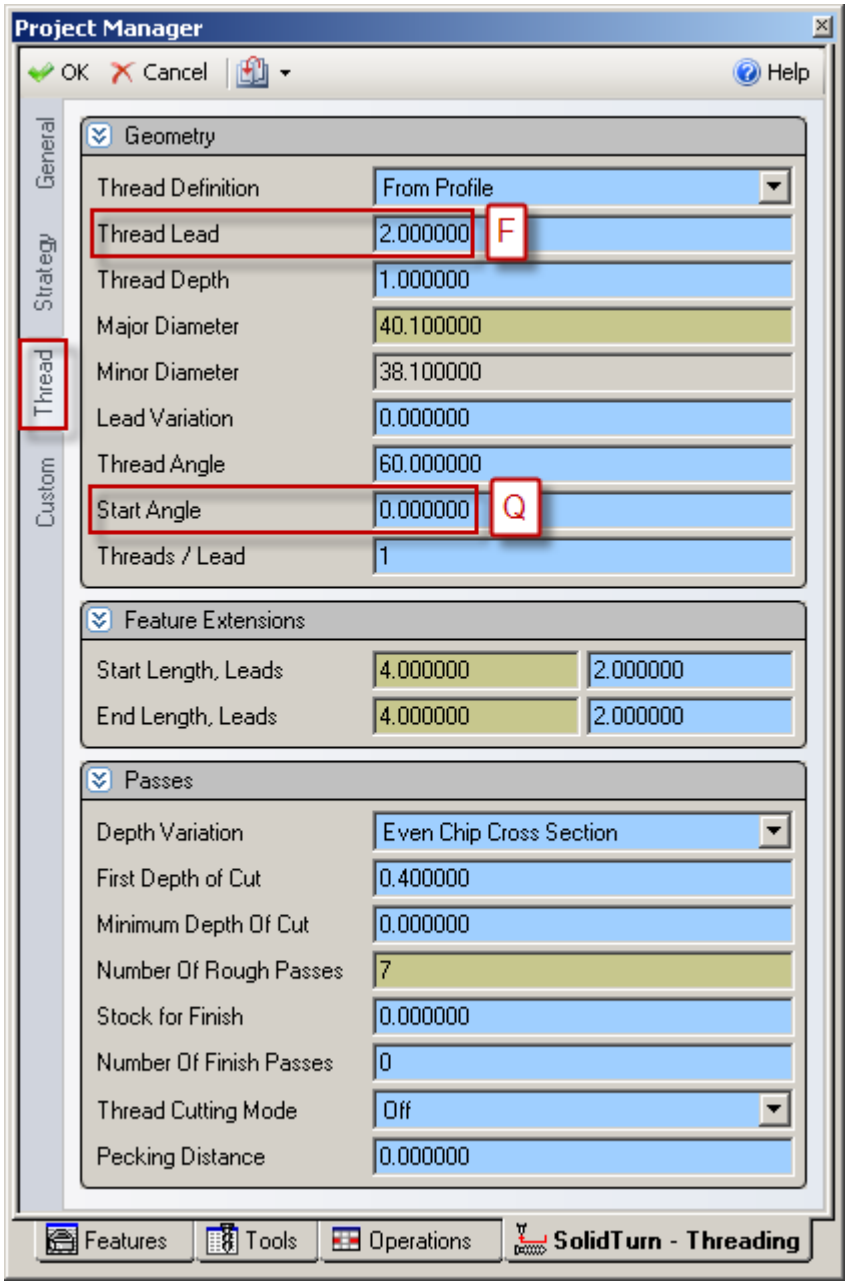
12.4.4. Threading with canned cycle set to multiple path

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

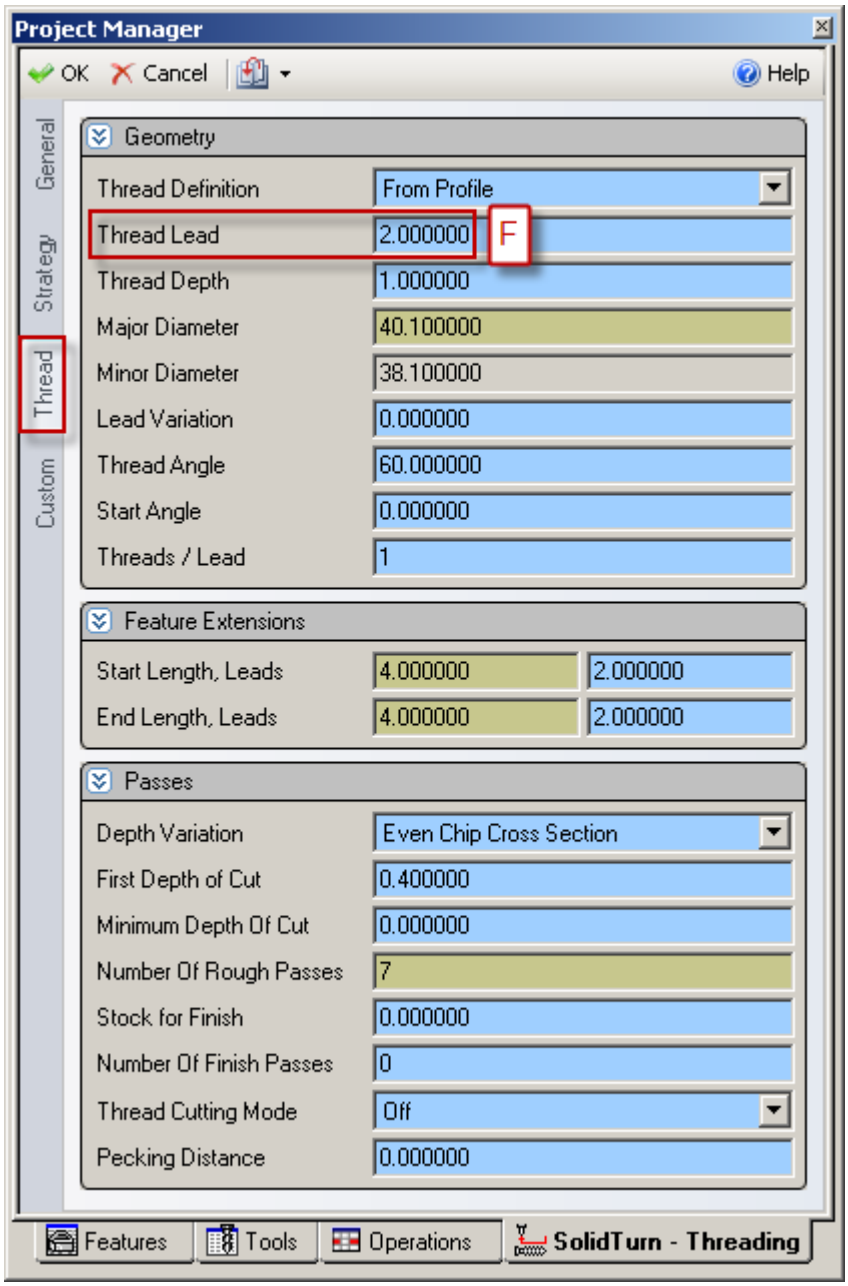
Note: If you set the Lead-In Type to Feed or Chamfer or the Lead-Out Type to Feed, you will get the following error message in your NC code: `ERROR: WRONG LEAD-IN OR LEAD-OUT TYPE SPECIFIED`.



Straight thread cutting:
G92 X_ Z_ F_ Q_ ;



Tapered thread cutting:
G92 X_ Z_ R_ F_ ;



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

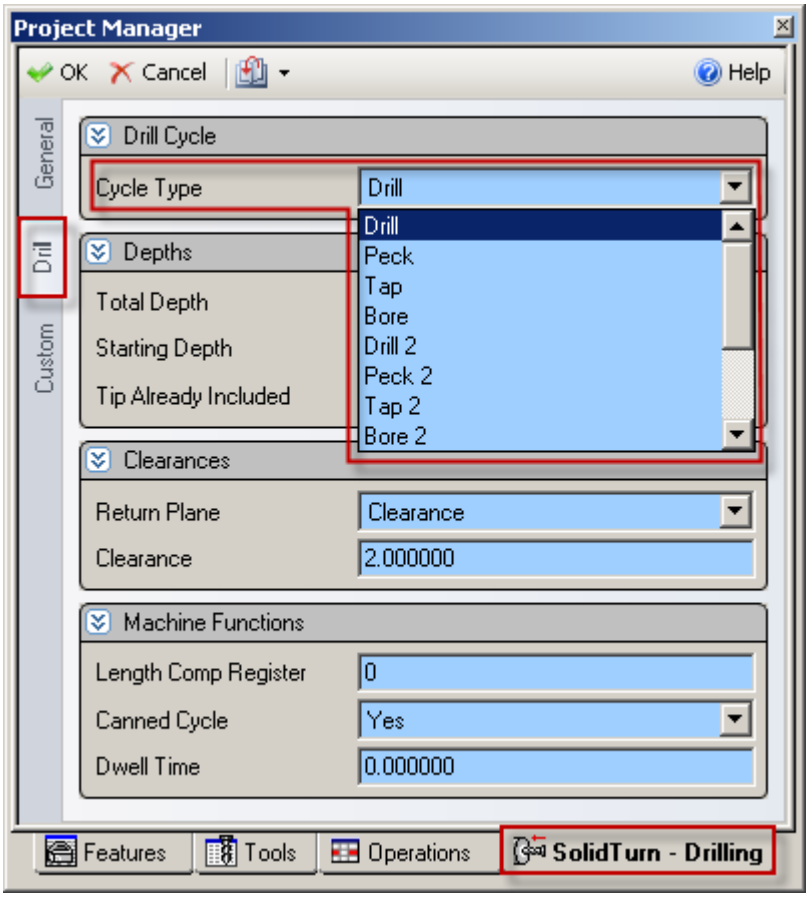
13.1. Introduction

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:

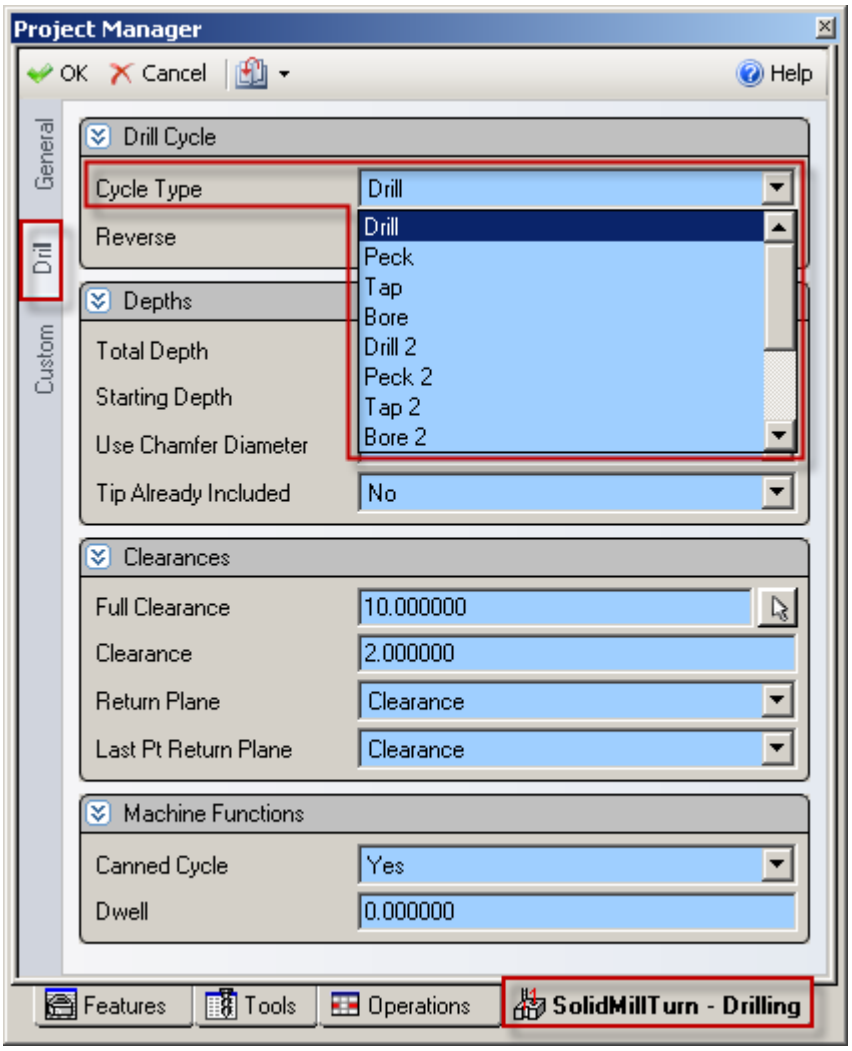
The Face Hole Machining Canned Cycle:
Hole machining is executed by the Z-axis with positioning executed by the combination of the X-, C- and Y-axes (Y-axis specifications only).

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

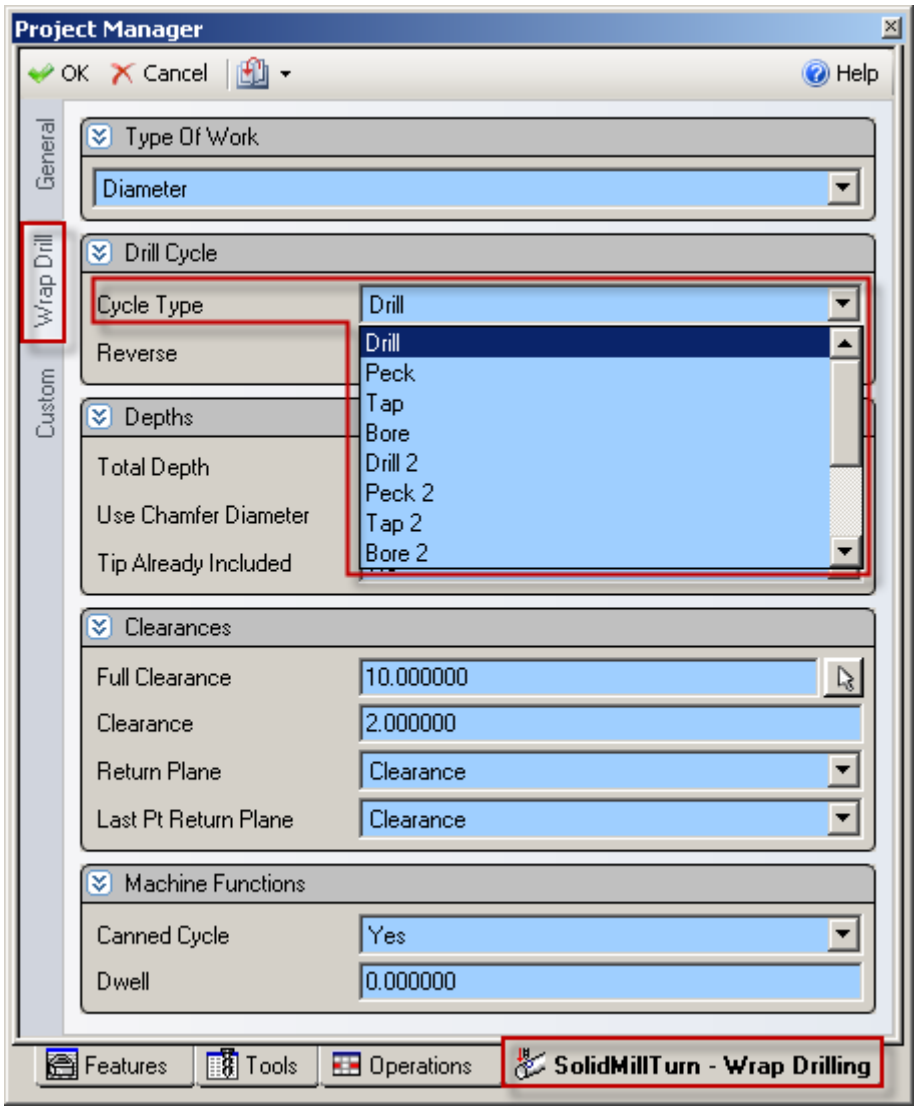
In ESPRIT, you can choose between the different drilling canned cycles using the Cycle Type pull-down on the Drill tab for the SolidTurn Drilling cycle:



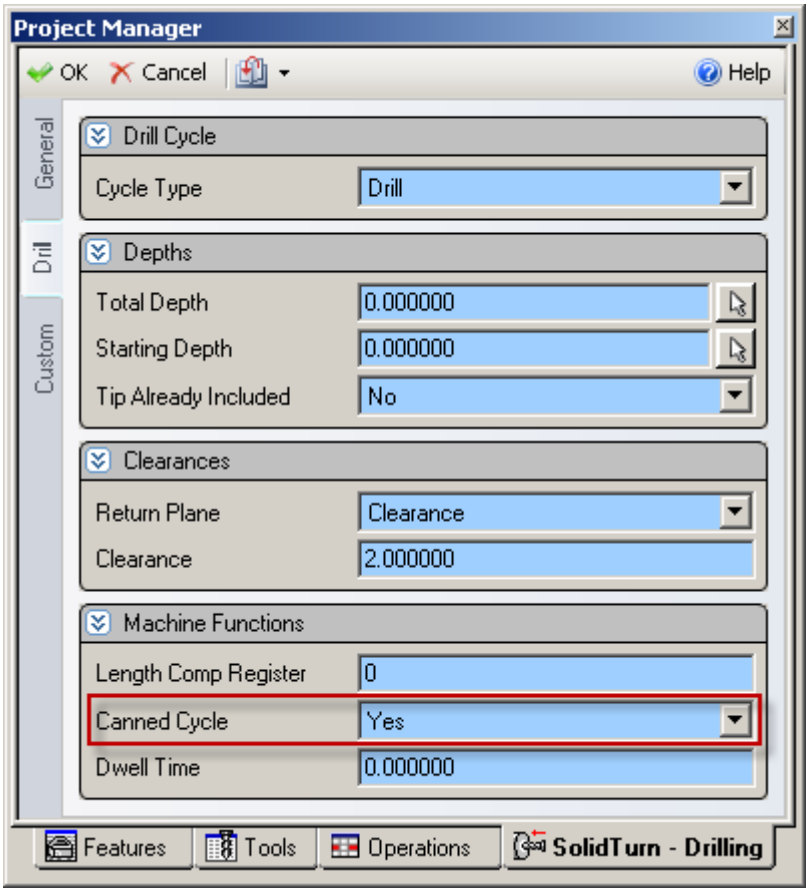
On the Drill tab for the SolidMillTurn Drilling cycle:



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:

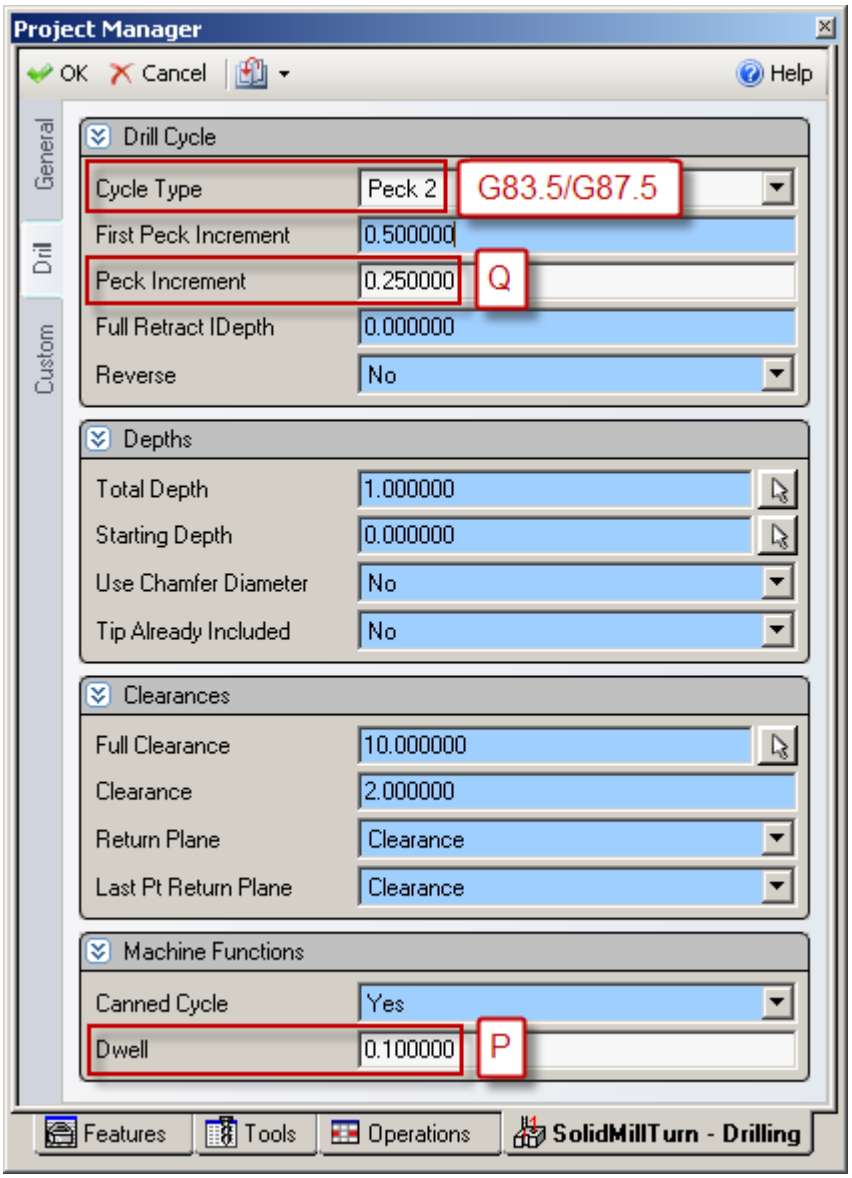
- The depth of cut per one infeed motion **Q** is an unsigned value, in radius for side drilling. Specify Q in mm, it will then be output in units of 0.001 mm without decimal point. For example, if you enter 2 for Q in ESPRIT, the output will be in the NC code Q2000.
- Enter the dwell **P** in s, it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

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

13.2.1. Face and side high-speed deep hole drilling (G83.5/G87.5)

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

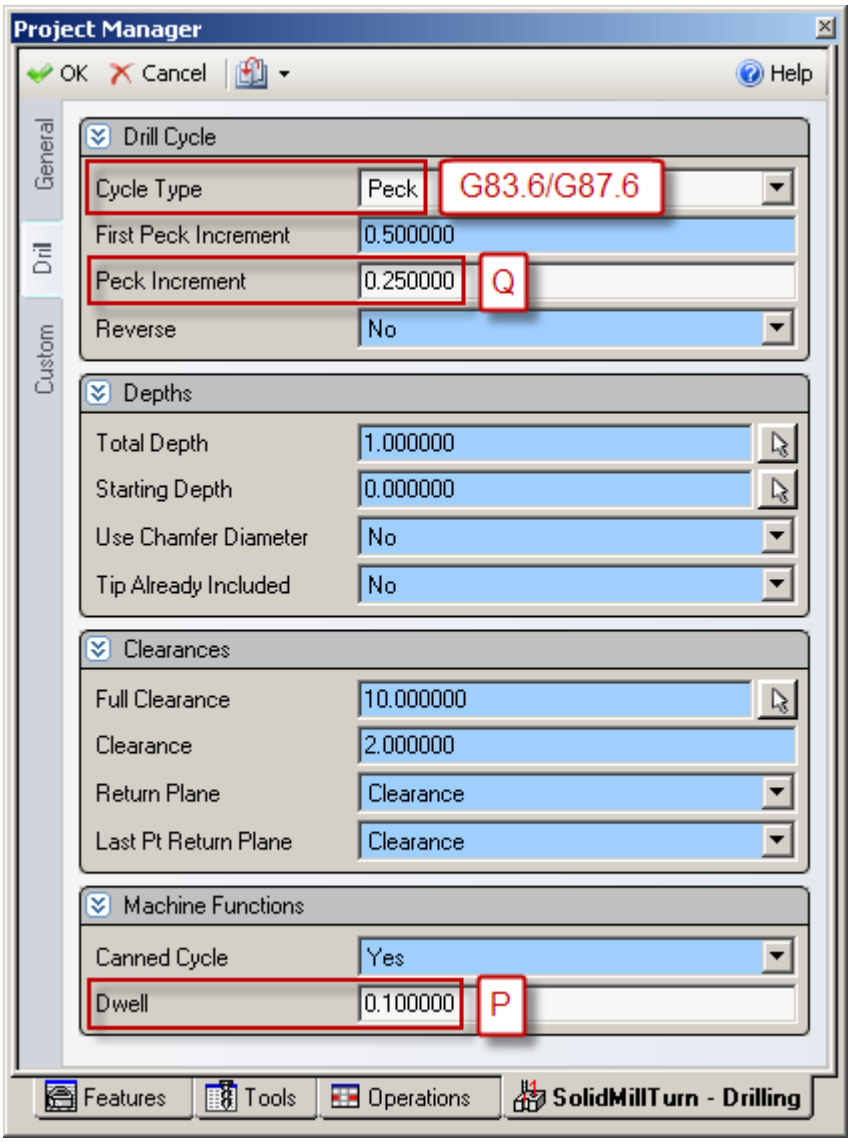
G83.5 X_ (Y_) C_ Z_ R_ Q_ P_ F_ ;
G87.5 Z_ (Y_) C_ X_ R_ Q_ P_ F_ ;



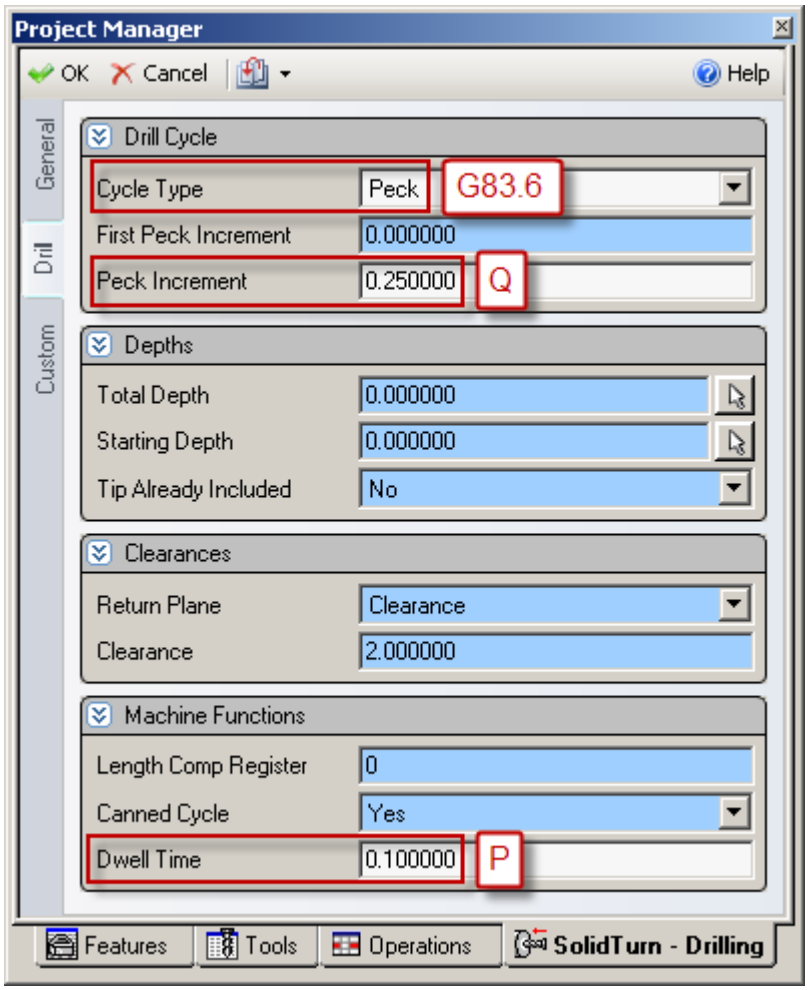
13.2.2. Face and side deep hole drilling (G83.6/G87.6)

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

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



Settings for SolidTurn Drilling:

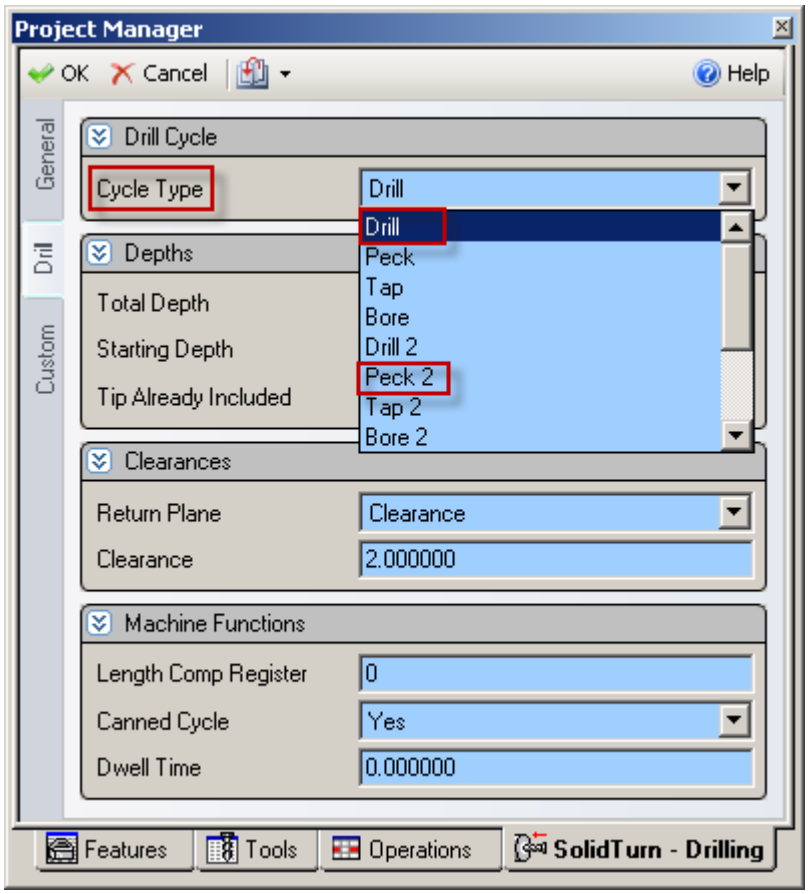


13.2.3. Deep hole drilling with G74

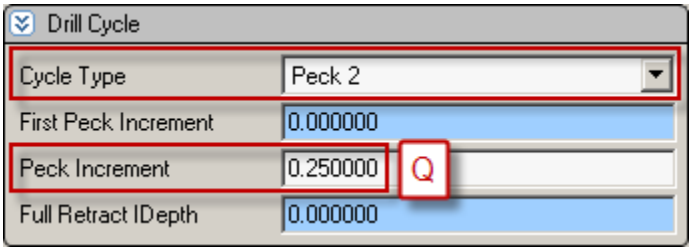
CAUTION !!! Use **SolidTurn Drilling**

G74 R(1)
G74 Z_ Q_ F_ ;

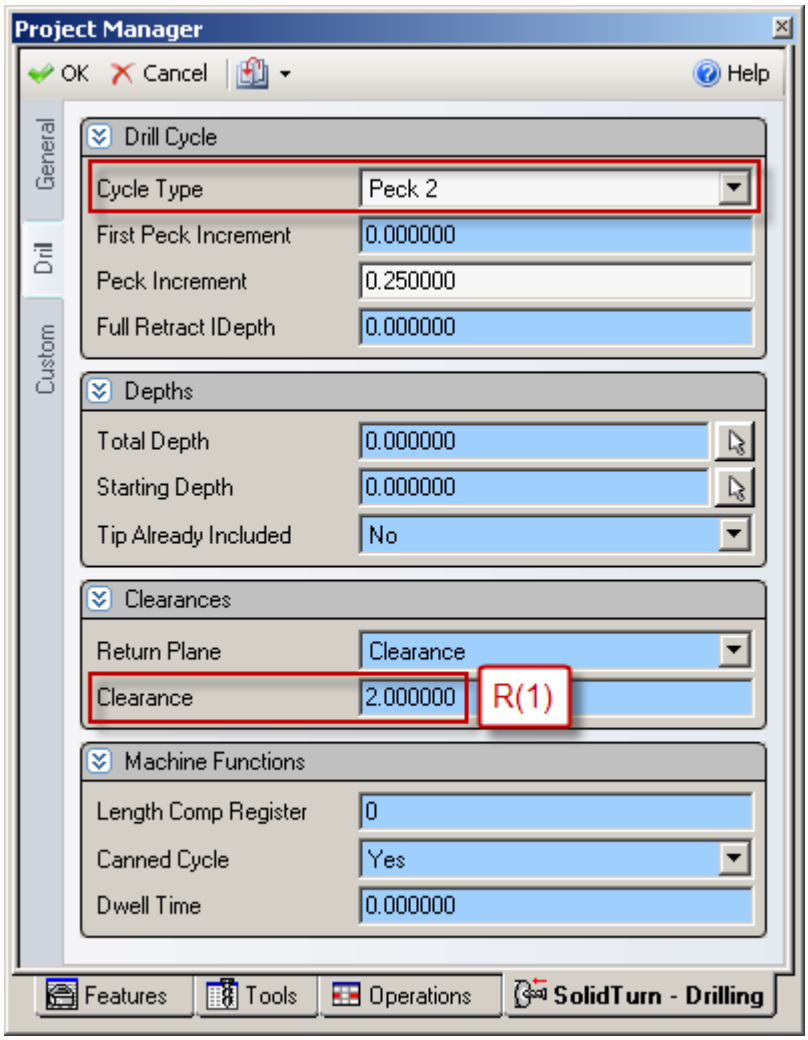
On the SolidTurn Drilling cycle, set the Cycle Type to Drill or Peck 2 to output your drilling cycle with G74.



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:



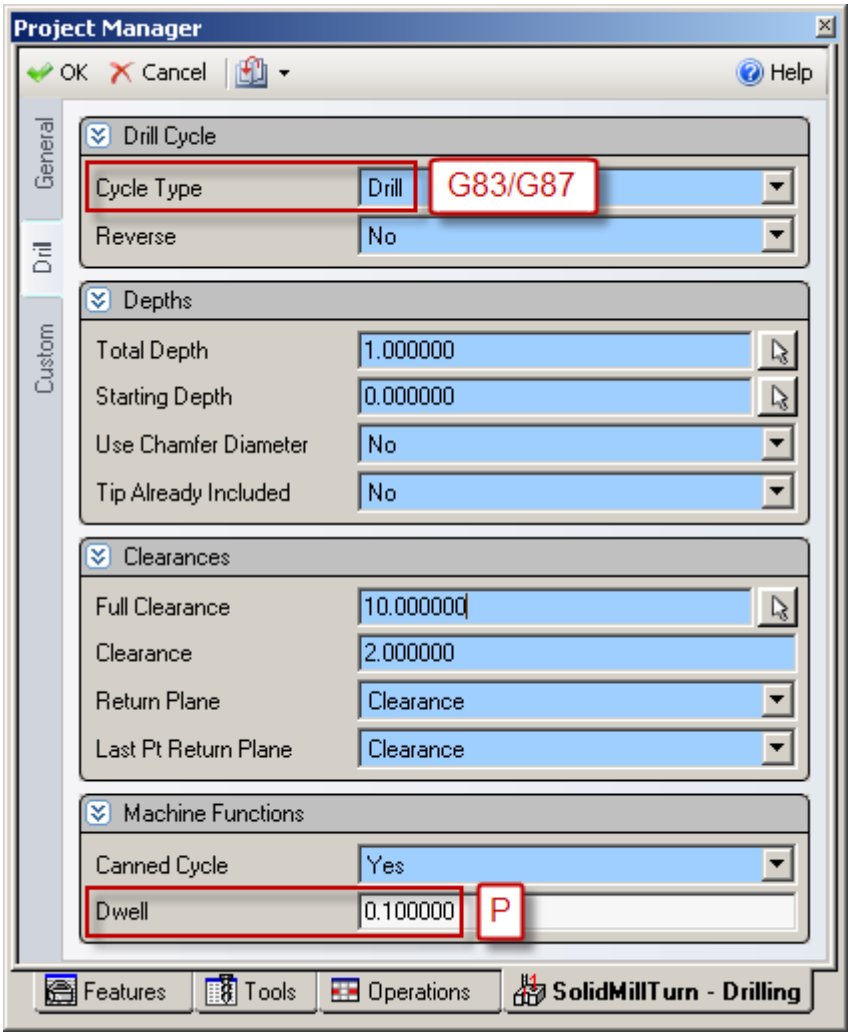
R(1) corresponds to return amount. It will be output as entered in ESPRIT when Peck 2 is selected. R0.0 will be output when Drill is selected.



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_ ;

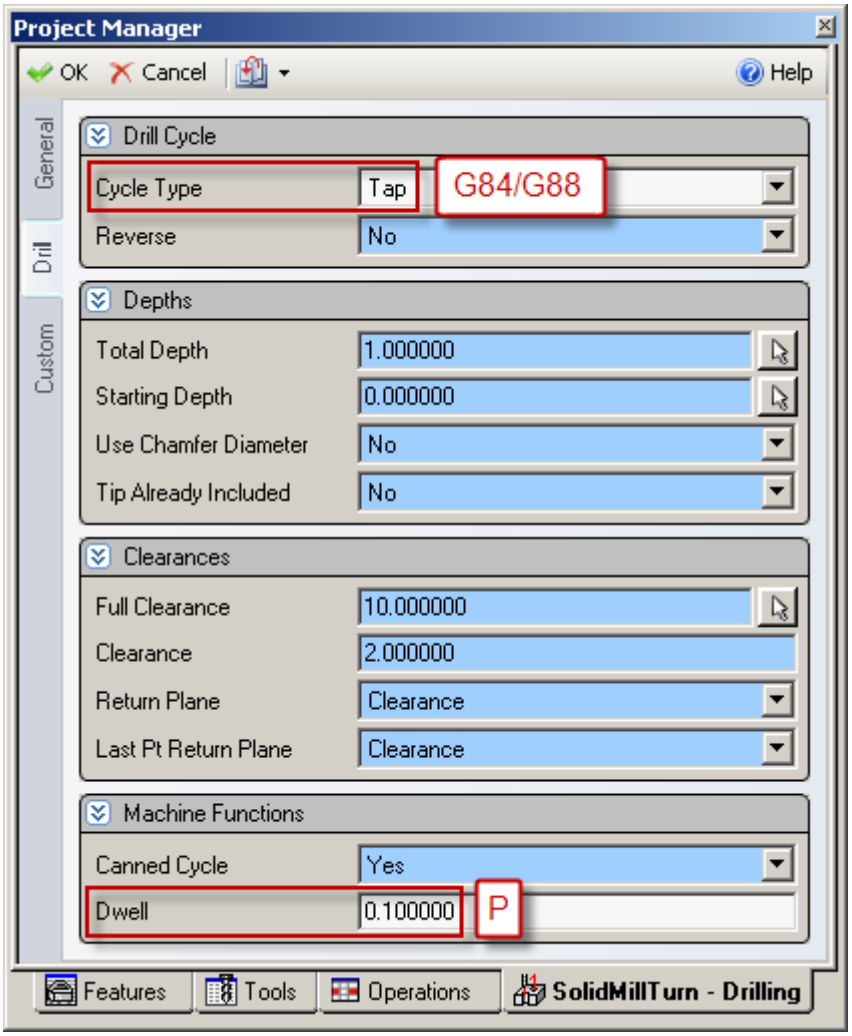


13.4. Face and side tapping / Tapping at center of spindle

13.4.1. Face and side tapping (G84/G88)

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

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

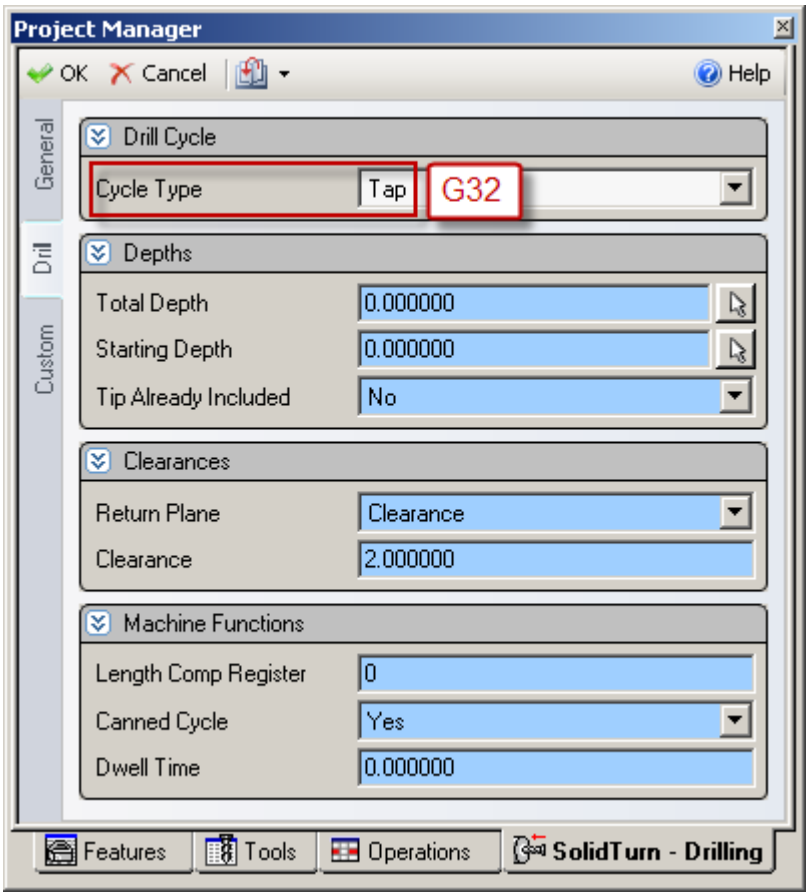


13.4.2. Tapping at center of spindle (G32)

CAUTION !!! Use **SolidTurn Drilling**

G32 Z_ F_ ;

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



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

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

13.5.1. Restriction of synchronized tapping

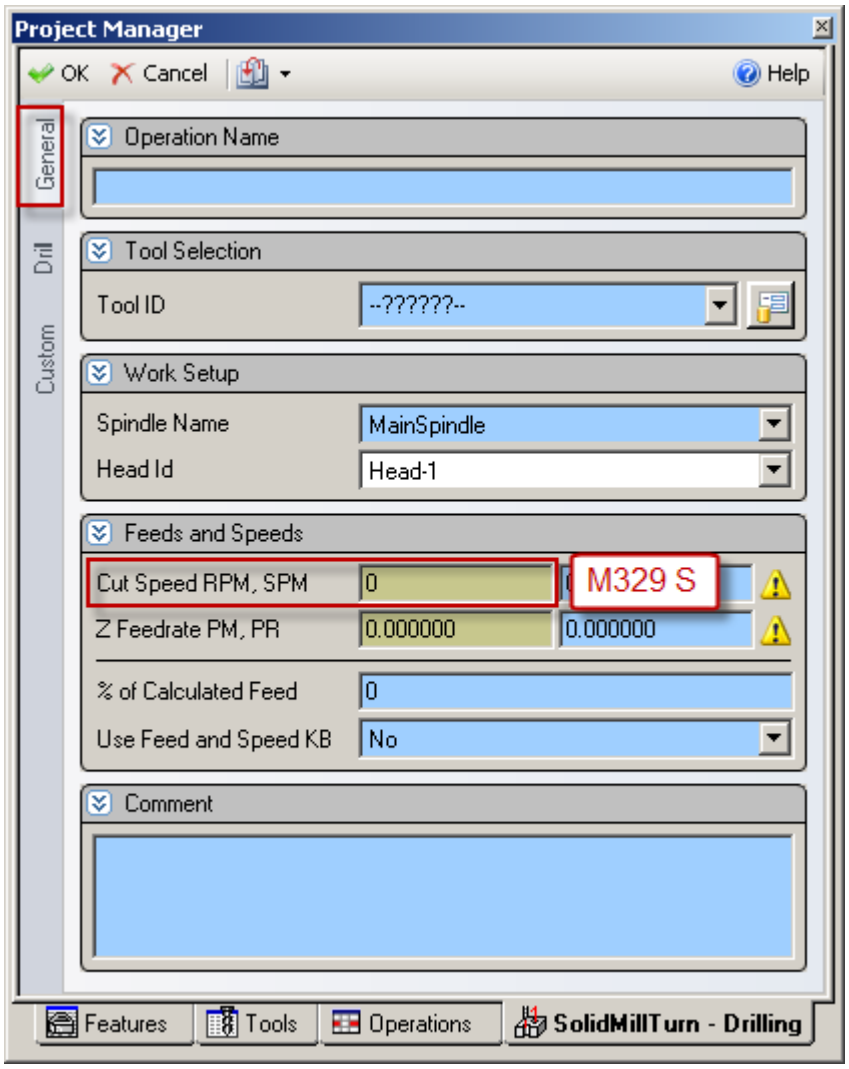
CAUTION !!! During the synchronized tapping cycle, the maximum speeds for the rotary tool is 3000 min^{-1}.
If the maximum speed for the rotary tool holder is less than the above, specify a speed that is less than the maximum speed for the holder.

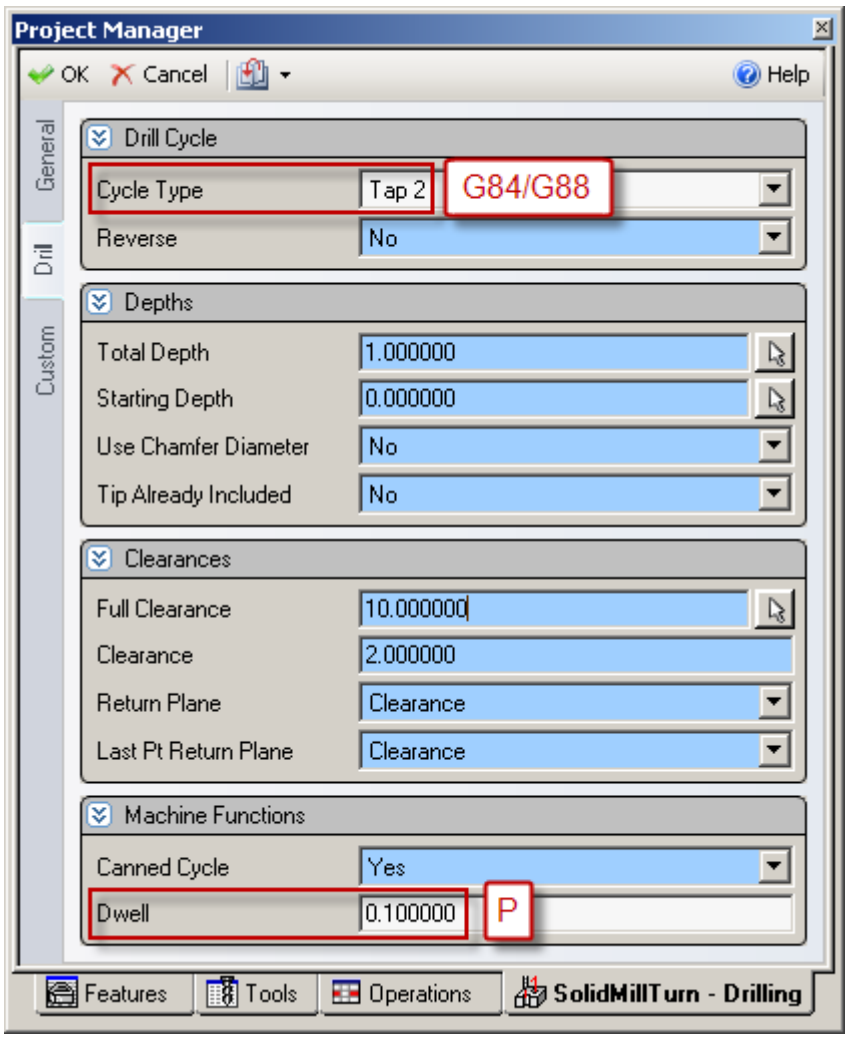
13.5.2. Face and side synchronized tapping (M329 G84/M329 G88)

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

M329 S_ ;

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

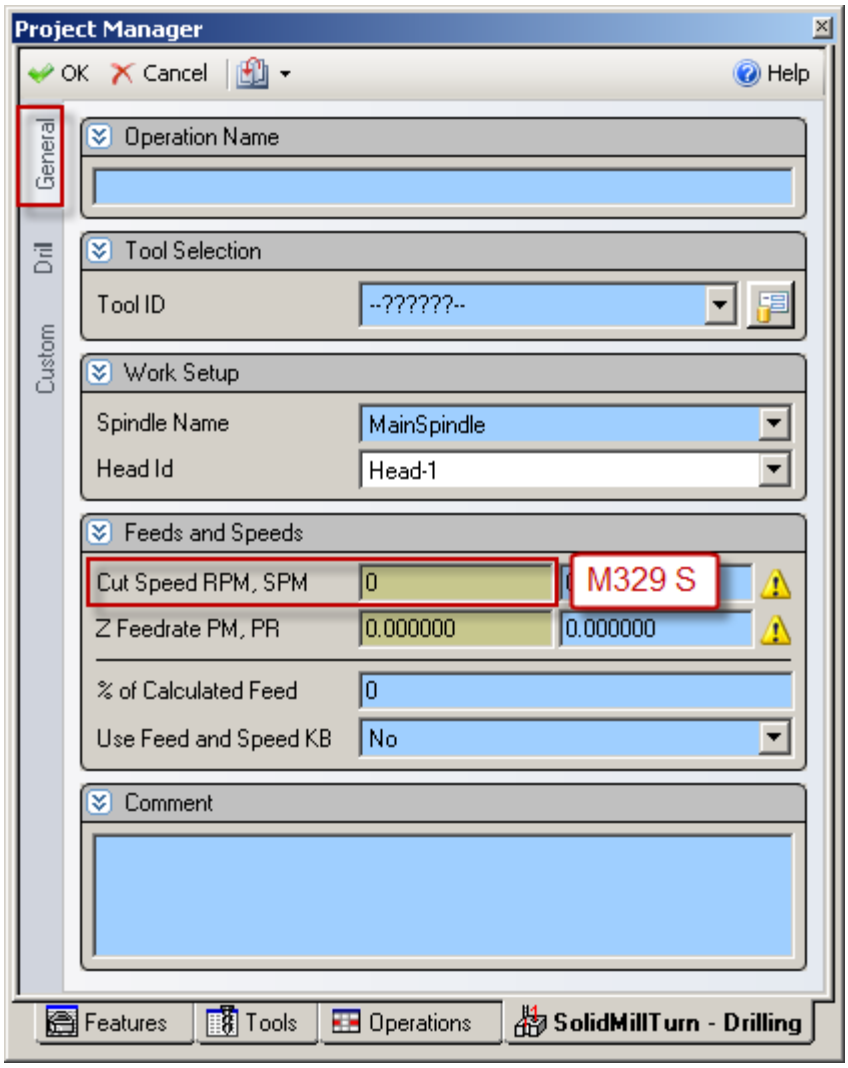


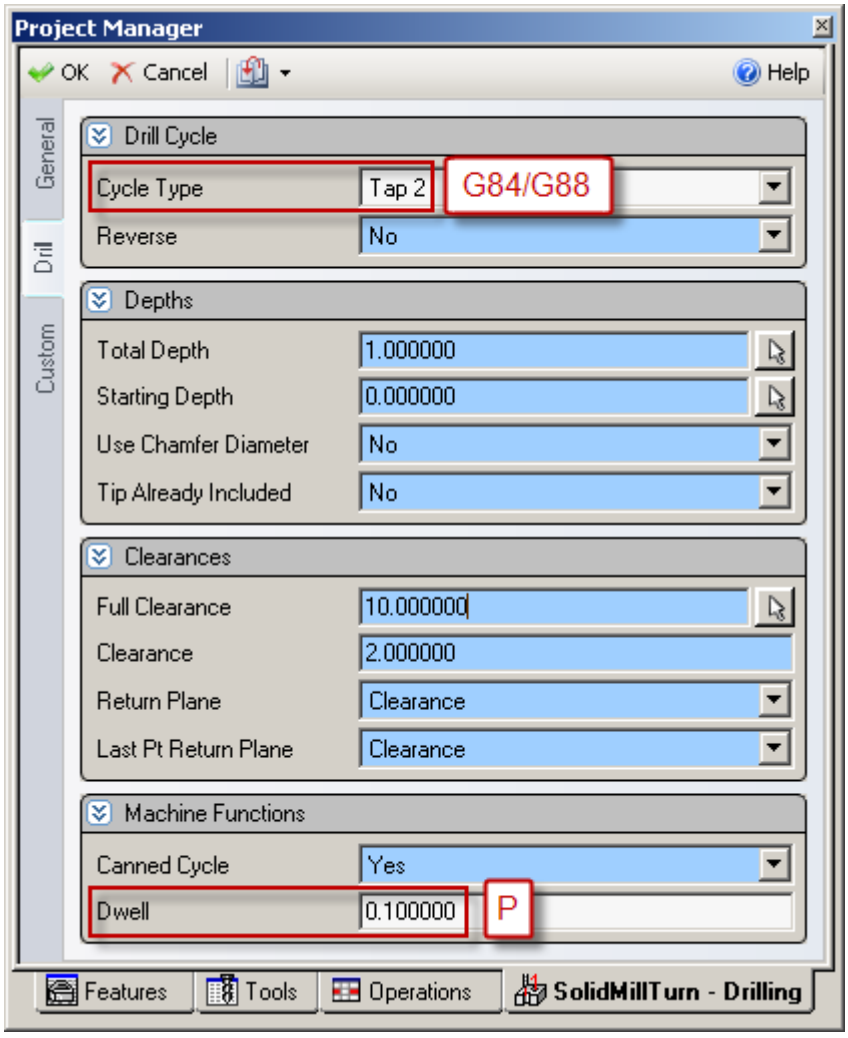


13.5.3. Face and side (high-speed) deep hole synchronized tapping (M329 G84/M329 G88)

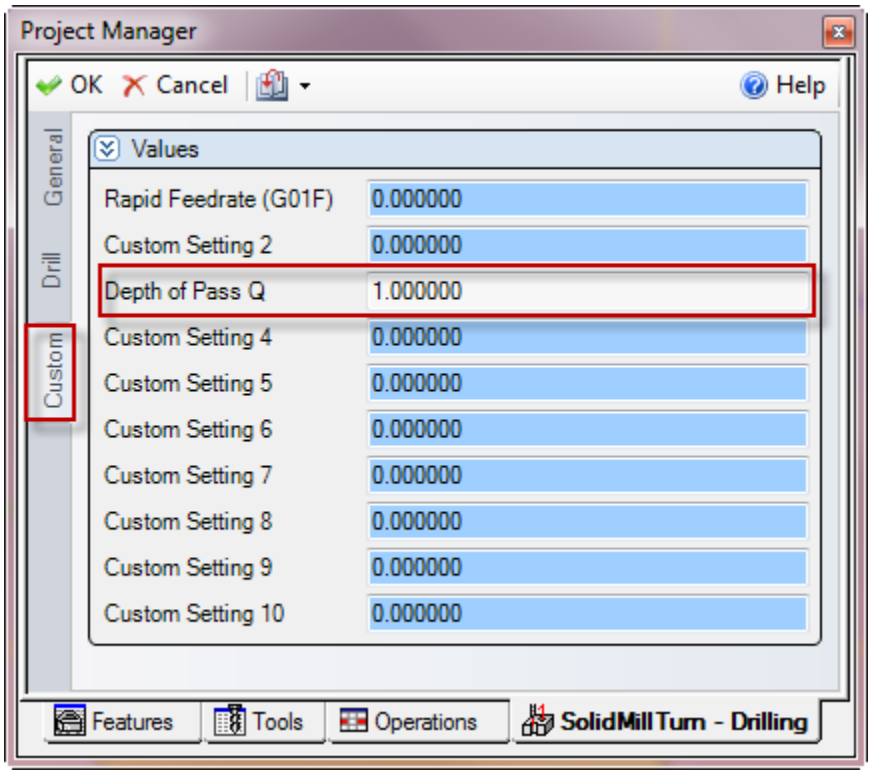
CAUTION !!! Use **SolidMillTurn Drilling** or **SolidMillTurn Wrap Drilling**

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


Project Manager
OK
Cancel
Help
General
Drill
Custom
Operation Name
Tool Selection
Tool ID
--??????--
Work Setup
Spindle Name
MainSpindle
Head Id
Head-1
Feeds and Speeds
Cut Speed RPM, SPM
0
M329 S
Z Feedrate PM, PR
0.000000
0.000000
% of Calculated Feed
0
Use Feed and Speed KB
No
Comment
Features
Tools
Operations
SolidMillTurn - Drilling



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



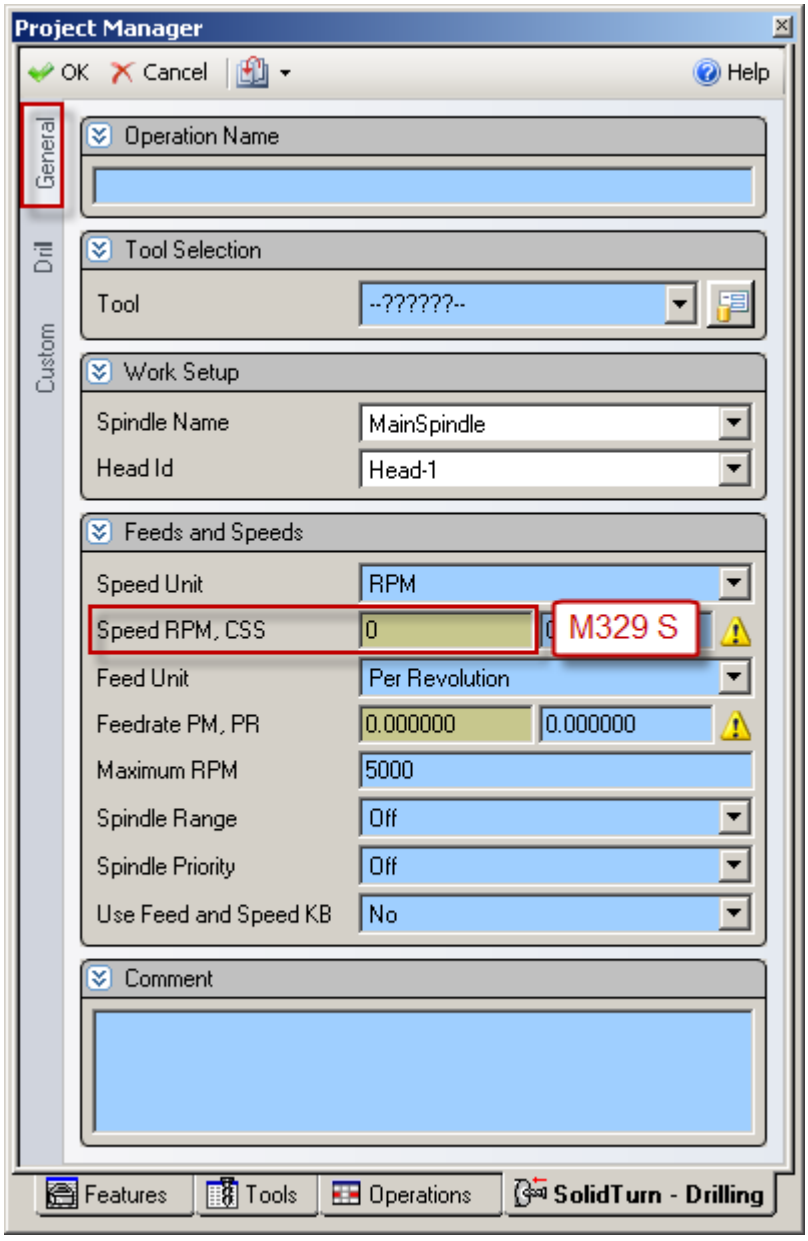
13.5.4. Spindle synchronized tapping (M329 G84) (Option on machine)

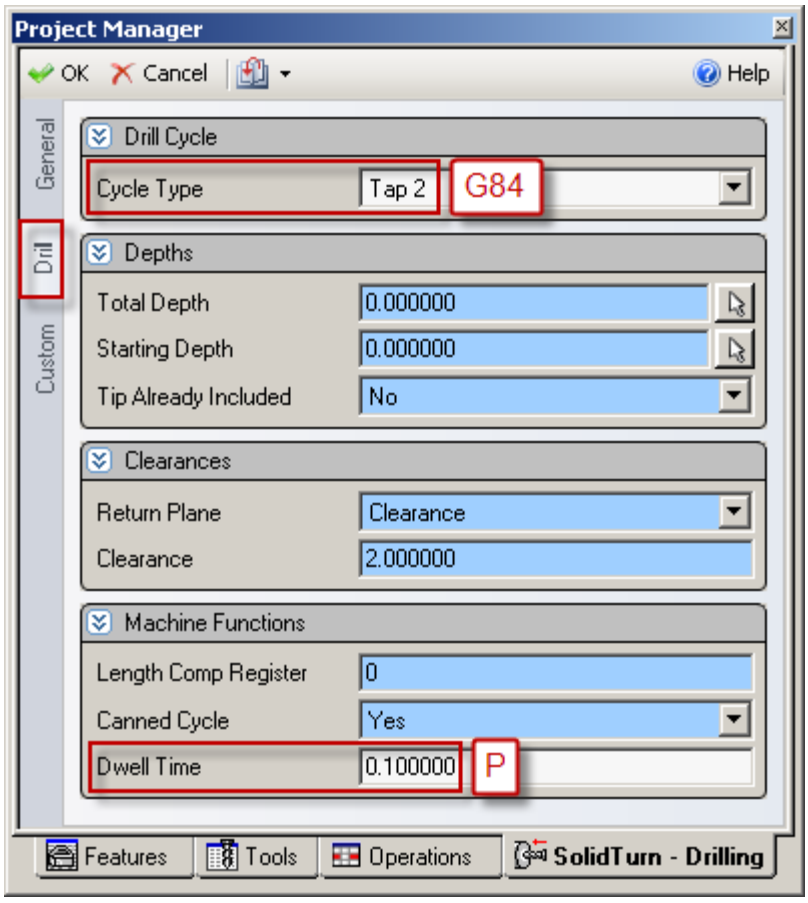
CAUTION !!! Use **SolidTurn Drilling**

M329 S_ ;
G84 X_ Z_ R_ P_ F_ ;

The M329 G84 command is used to execute a tapping operation at the center of the spindle (rotational center of workpiece).
This tapping cycle allows the Z-axis feed to be always synchronized with spindle revolution.

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


Project Manager
OK
Cancel
Help
General
Drill
Custom
Operation Name
Tool Selection
Tool
--??????--
Work Setup
Spindle Name
MainSpindle
Head Id
Head-1
Feeds and Speeds
Speed Unit
RPM
Speed RPM, CSS
0
M329 S
Feed Unit
Per Revolution
Feedrate PM, PR
0.000000
0.000000
Maximum RPM
5000
Spindle Range
Off
Spindle Priority
Off
Use Feed and Speed KB
No
Comment
Features
Tools
Operations
SolidTurn - Drilling

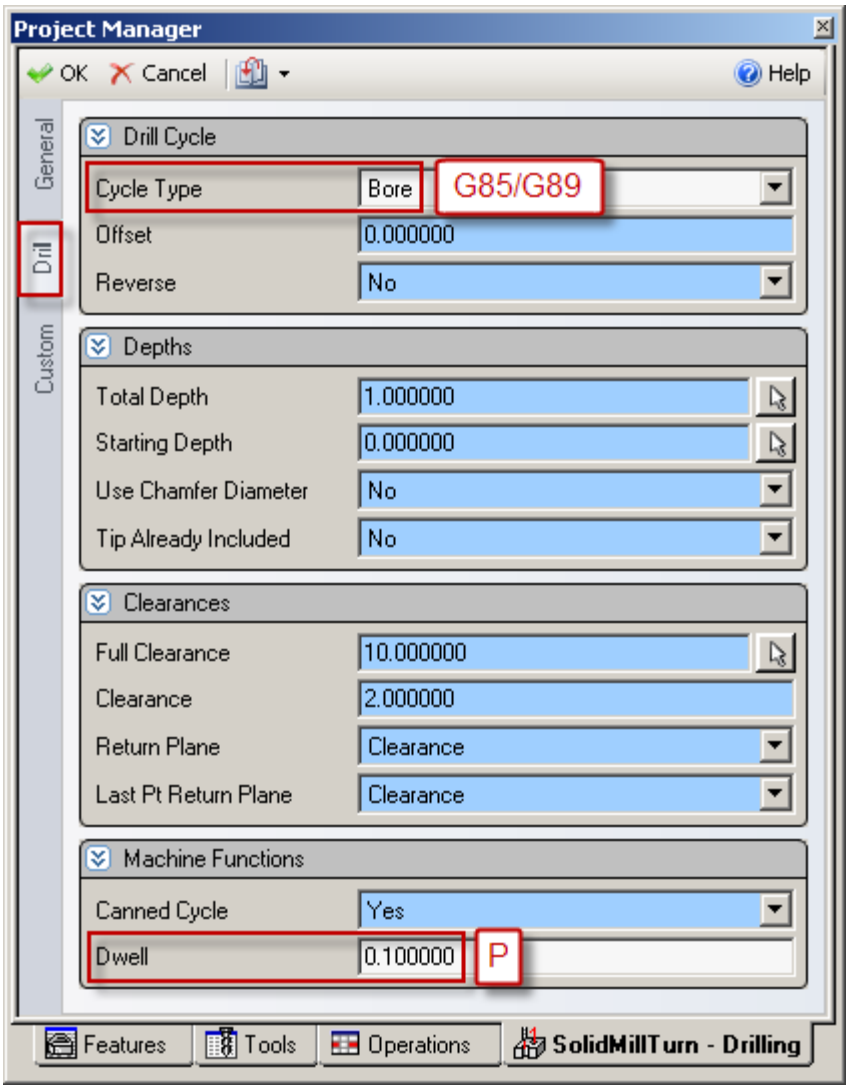


13.6. Face and side boring / Boring in turning mode

13.6.1. Face and side boring (G85/G89)

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

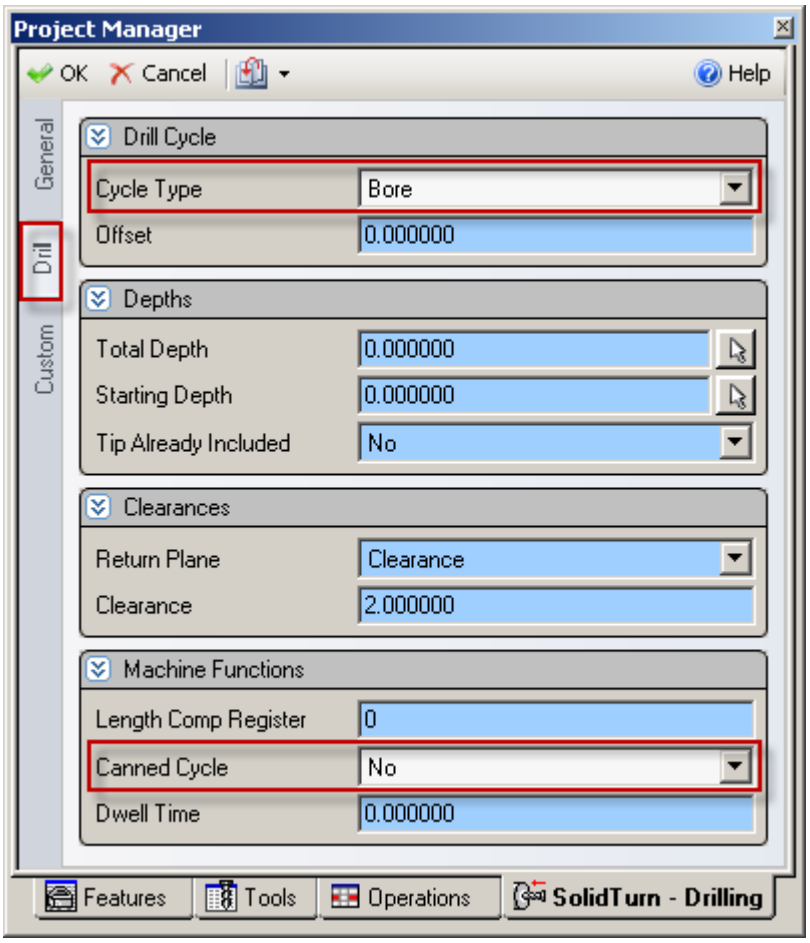
G85 X_ (Y_) C_ Z_ R_ P_ F_ ;
G89 Z_ (Y_) C_ X_ R_ P_ F_ ;



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 BORING CYCLE`.

13.7. Spindle / Rotary Tool Spindle Simultaneous Operation Mode

CAUTION !!! Use **SolidTurn Drilling**

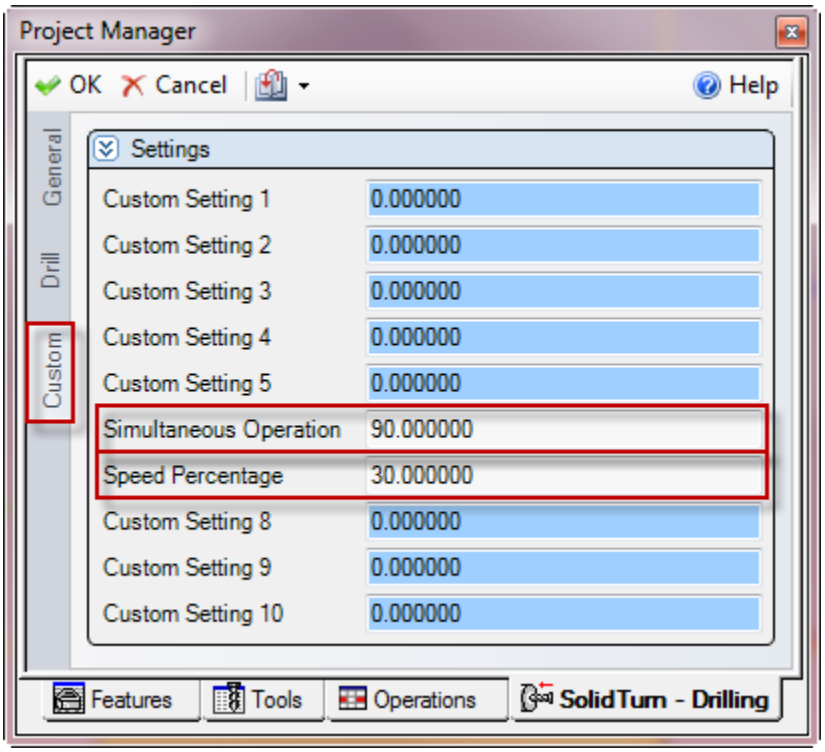
M90 S_ ;
M91;
M290 S_ ;
M291;

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

On the Custom tab of the SolidTurn Drilling operation, set Simultaneous Operation (Custom Setting 6) to 90 to trigger the output of the M90 (or M290 if machining on the spindle 2 side for NZX-ST & NZX-S configurations) command.

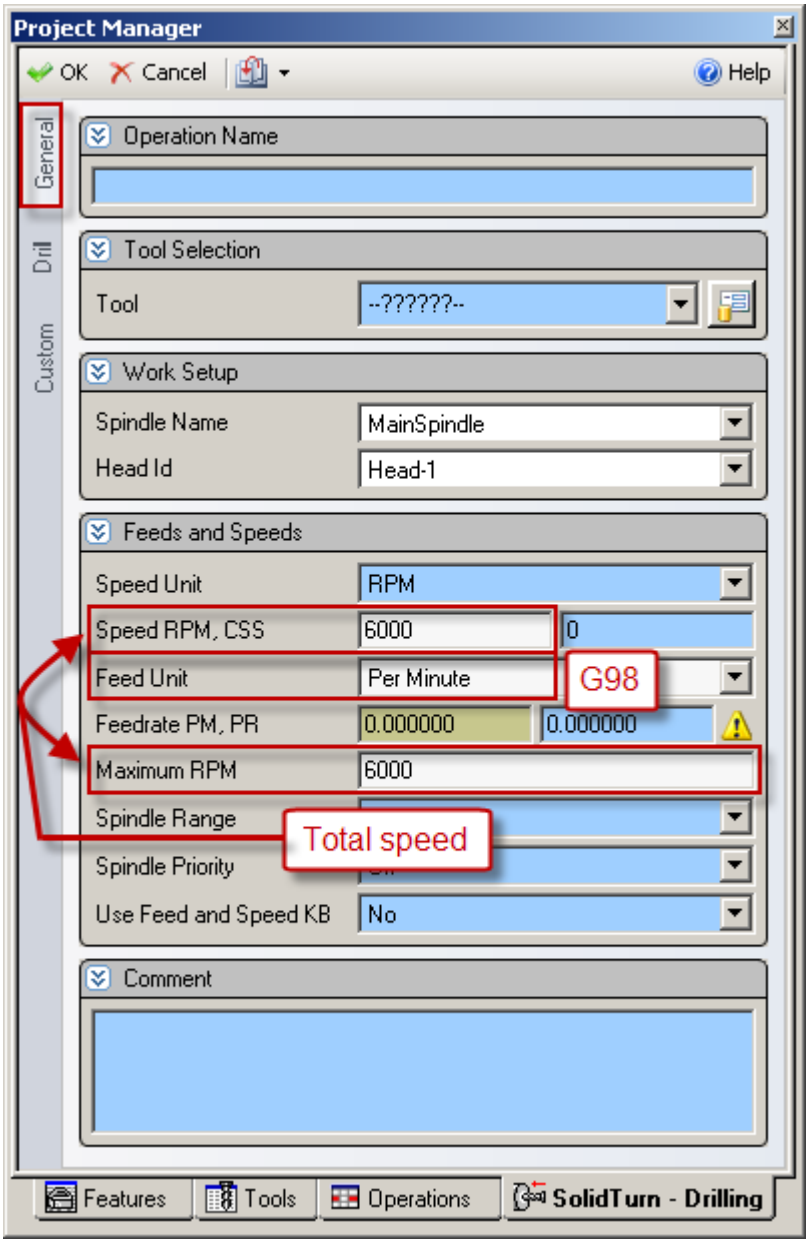
On the General tab, set in Speed RPM the desired total speed, which mean speed of the turning spindle plus speed of the rotary tool.
You can then specify with Speed Percentage (Custom Setting 7) the percentage of the total speed you want to apply to the rotary tool.

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



Notes:

- Set on the General tab, the **Feed Unit** to **Per Minute**: You can determine the feedrate of the tool assuming the spindle speed of 6000 min^{-1}, which is the sum of the turning spindle speed (4200 min^{-1}) and the rotary tool speed (1800 min^{-1}).
- On the General tab, make sure to **change** the **Maximum RPM** value according to the specified Speed RPM value. If Speed RPM is set to 6000 rpm but Maximum RPM to 5000 rpm, the output will be computed using 5000 rpm, as a safety.
- The direction of rotation has to be specified on the Tool page by the pull-down Spindle Direction.



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

14.1. Introduction

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

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

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

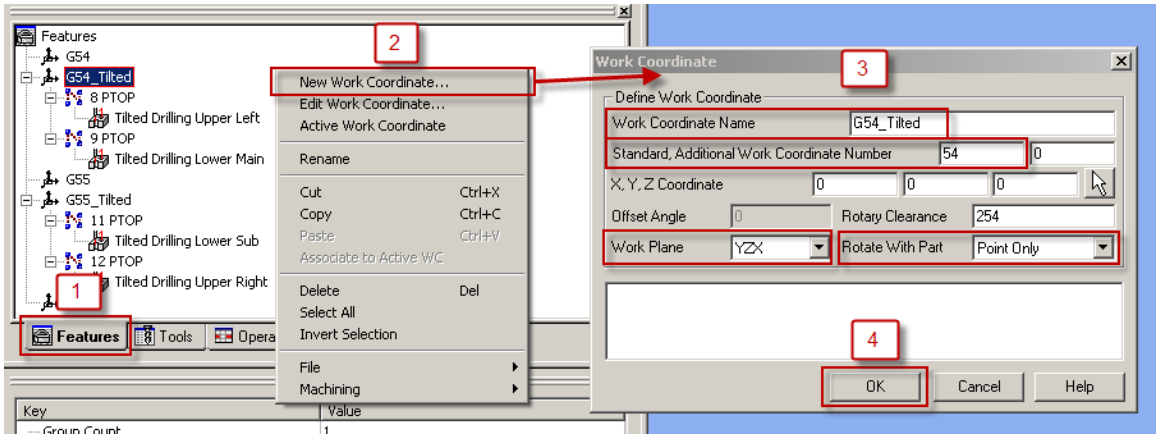
By using the program created assuming that machining is executed on the ZX plane, the programmed machining can be executed on the plane after rotation by converting the coordinate system by specifying the center of rotation, the direction of the center axis of rotation, and the angle of rotation.

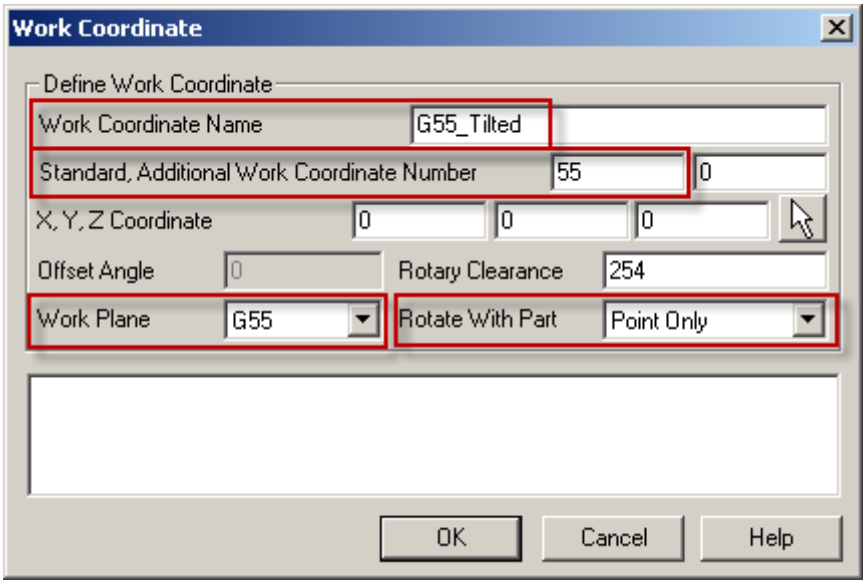
Note that the center of rotation will always be (X0; Y0; Z0) and the direction of the center axis of rotation (I0; J1; K0): the coordinate system will be tilted around the Y-axis. The angle of rotation R_ will automatically be computed for you based on the angle of the tilted holder.

14.2. How to output 3D coordinate conversion

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

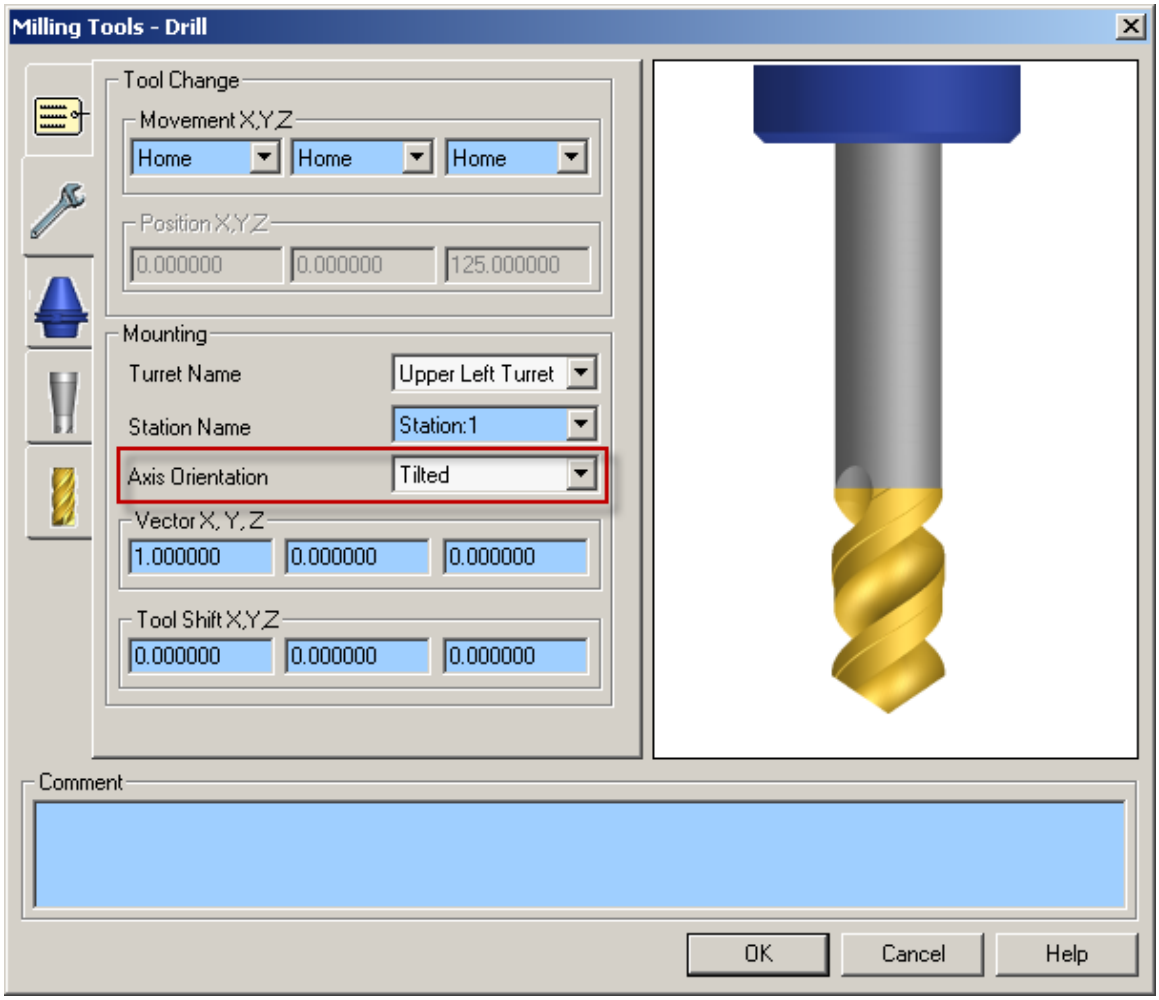
- Enter the Work Coordinate Name (for spindle 1 side: G54_Tilted and for spindle 2 side G55_Tilted, for example).
- Enter the Standard Work Coordinate Number: Usually 54 for spindle 1 side and 55 for spindle 2 side.
- Set the Work Plane to YZX if you machine on the spindle 1 side and to G55 if you machine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





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

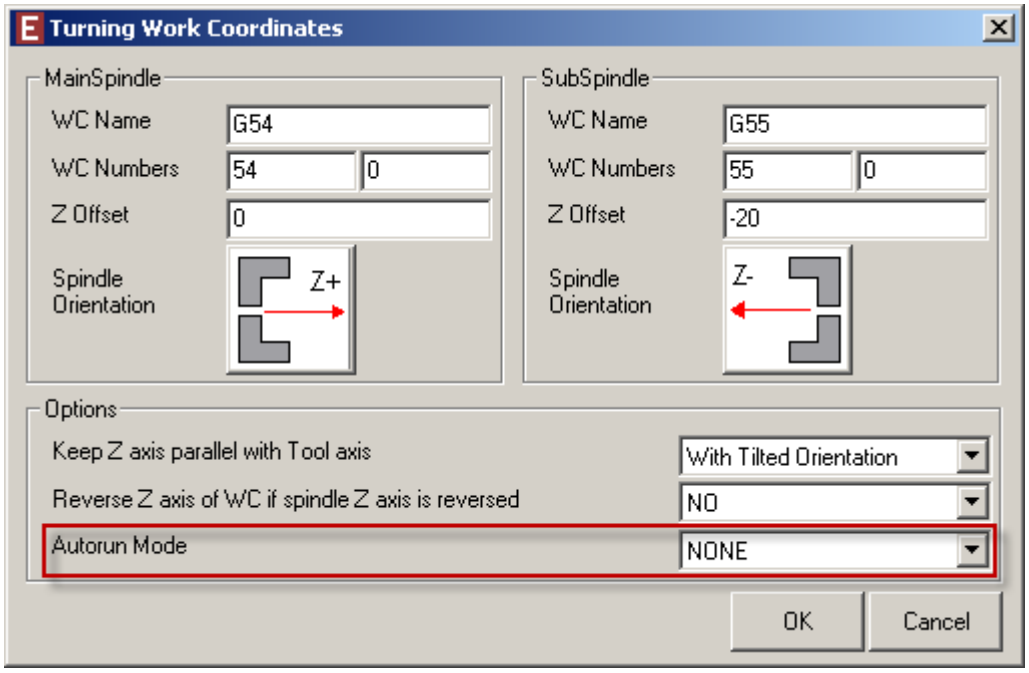
Your work coordinate is now correctly set for 3D coordinate conversion. If you **create a Tilted** tool in ESPRIT, the NC code output when using this tool will be with 3D coordinate conversion.



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

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

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



15. 4-axis wrap milling cycles with interpolation

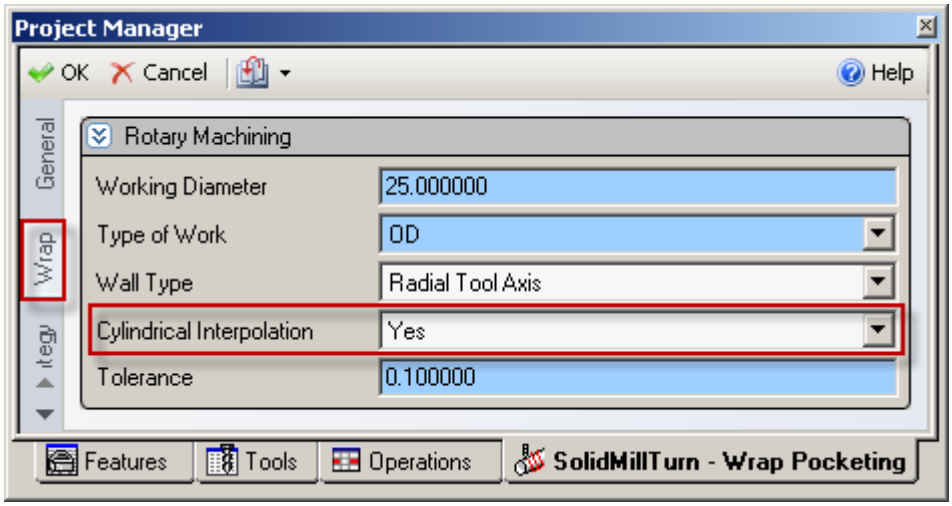
15.1. Introduction

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

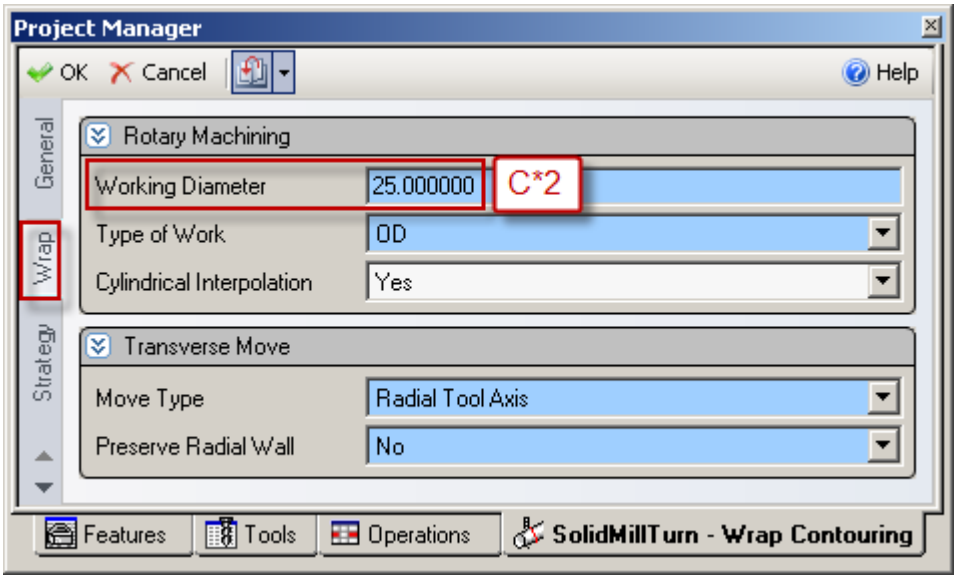
15.2. Cylindrical interpolation

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

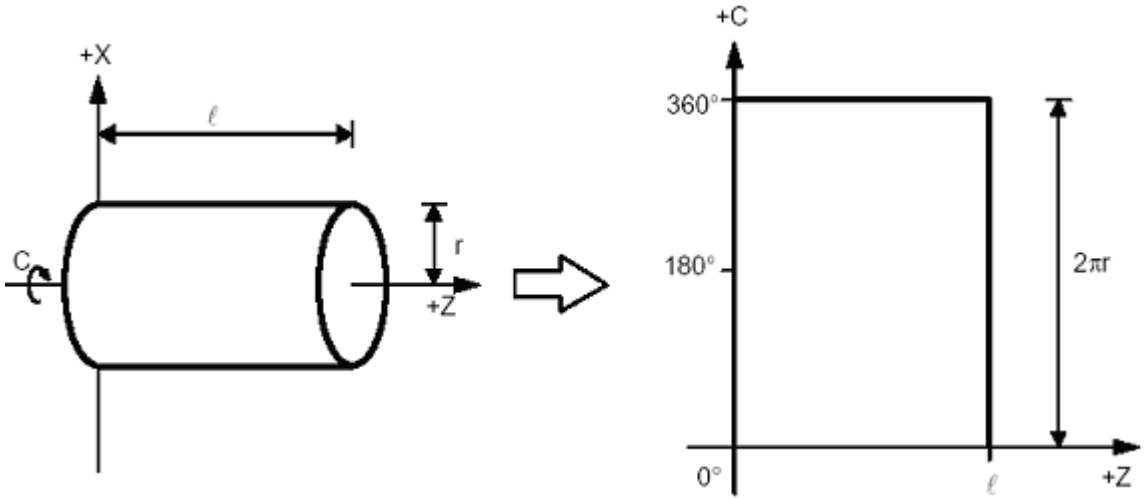
On the Wrap tab of the operation pages, set Cylindrical Interpolation to Yes to output 4-axis wrap milling operations with cylindrical interpolation.



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.

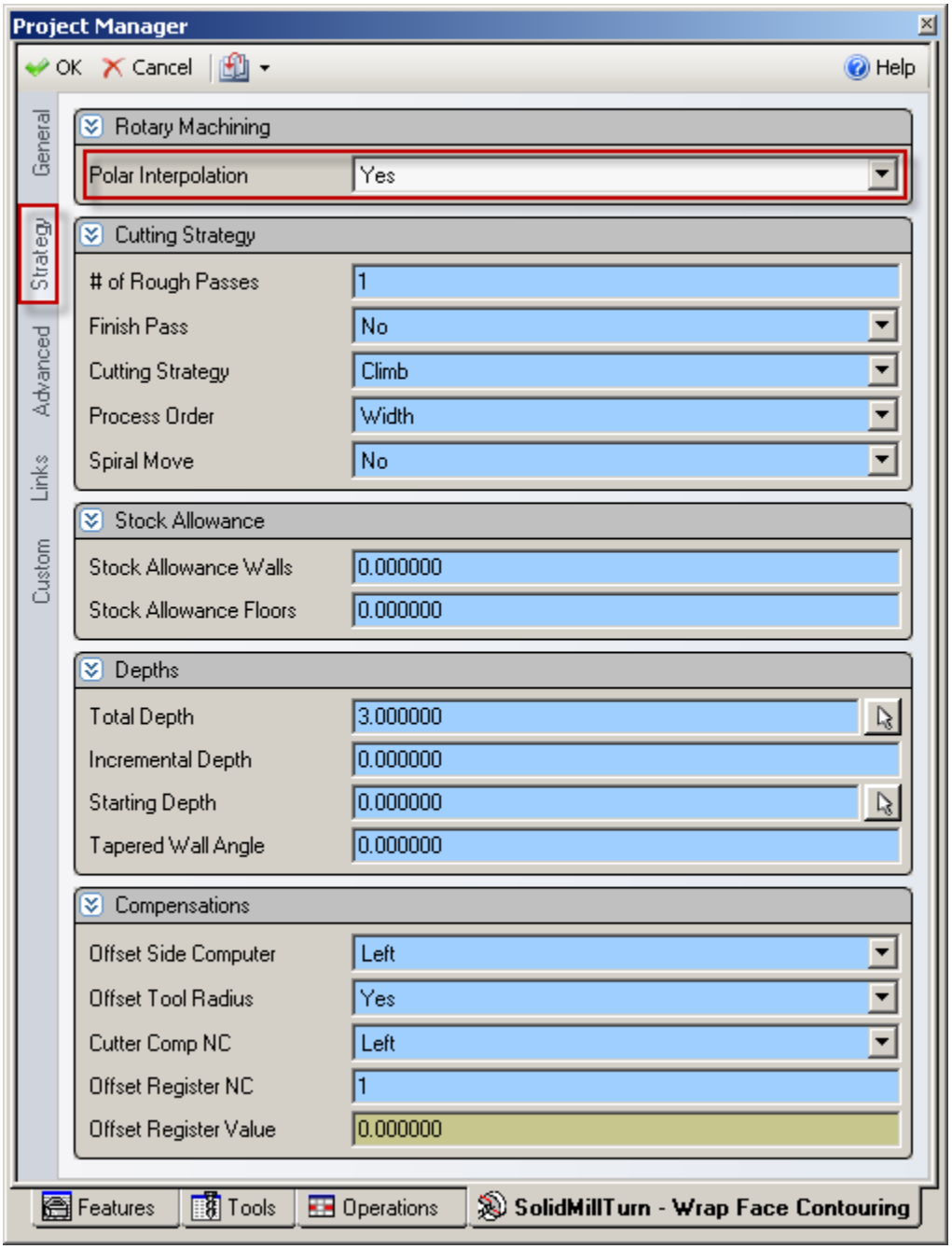


Using cylindrical interpolation can shorten the program length because ESPRIT outputs the circular motions in the cylindrical plane.
Without cylindrical interpolation, circular motions in the cylindrical plane are approximated by small linear moves (quality varies with the tolerance of the operation page).

15.3. Polar coordinate interpolation (Notching)

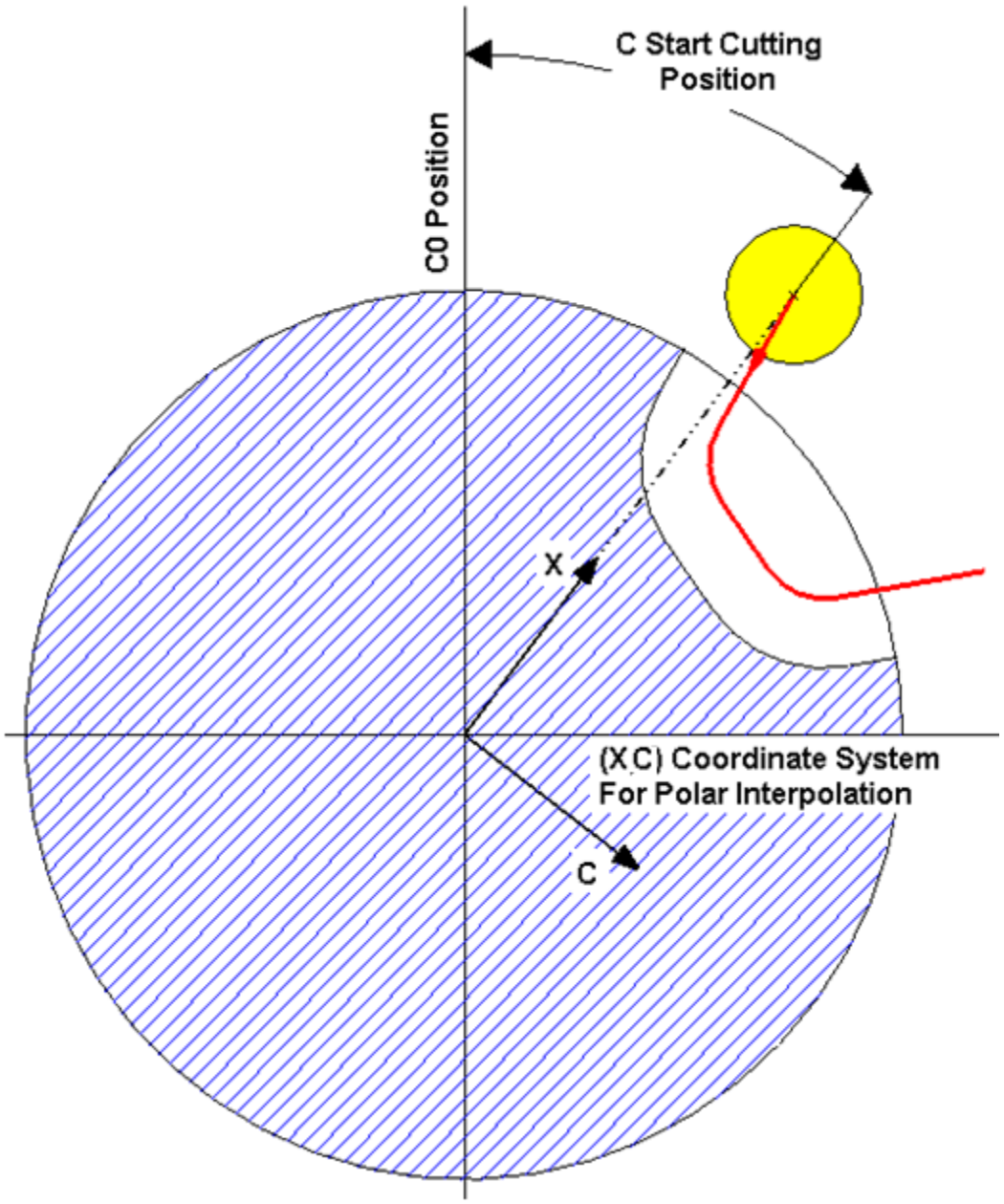
The polar coordinate interpolation can be turned on the SolidMillTurn Wrap Face Pocketing and Wrap Face Contouring pages.

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



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

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



16. Custom Settings index

16.1. On operation pages

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

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

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

16.2. On tool pages

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

16.3. In Machine Setup

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

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