POST PROCESSOR MANUAL FOR

MORI SEIKI

NT(X) Series

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

Table of Contents

1. Introduction

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

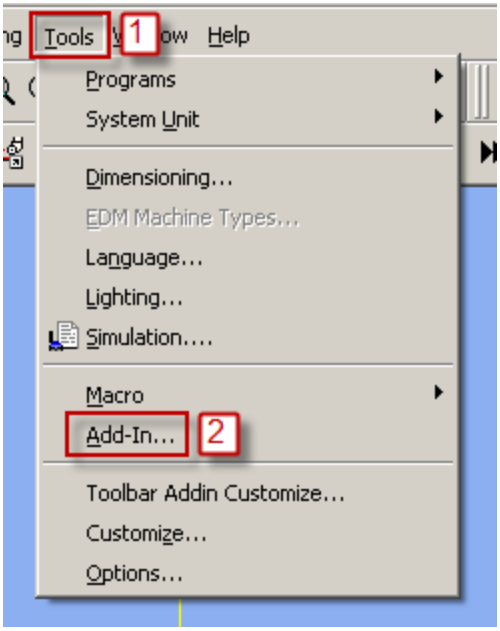
Please refer to NT programming manual for detailed information on the related G and M-codes.

2. Setting up the Esprit Environment

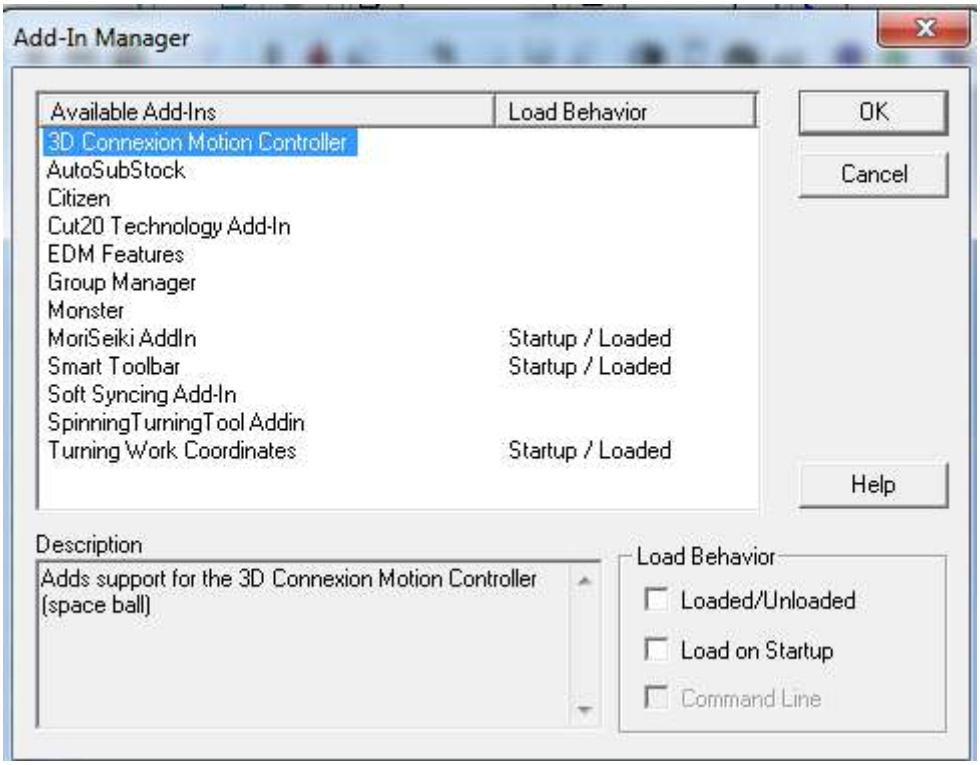
2.1. Required Add-ins

2.1.1. How to Load an Add-in in ESPRIT

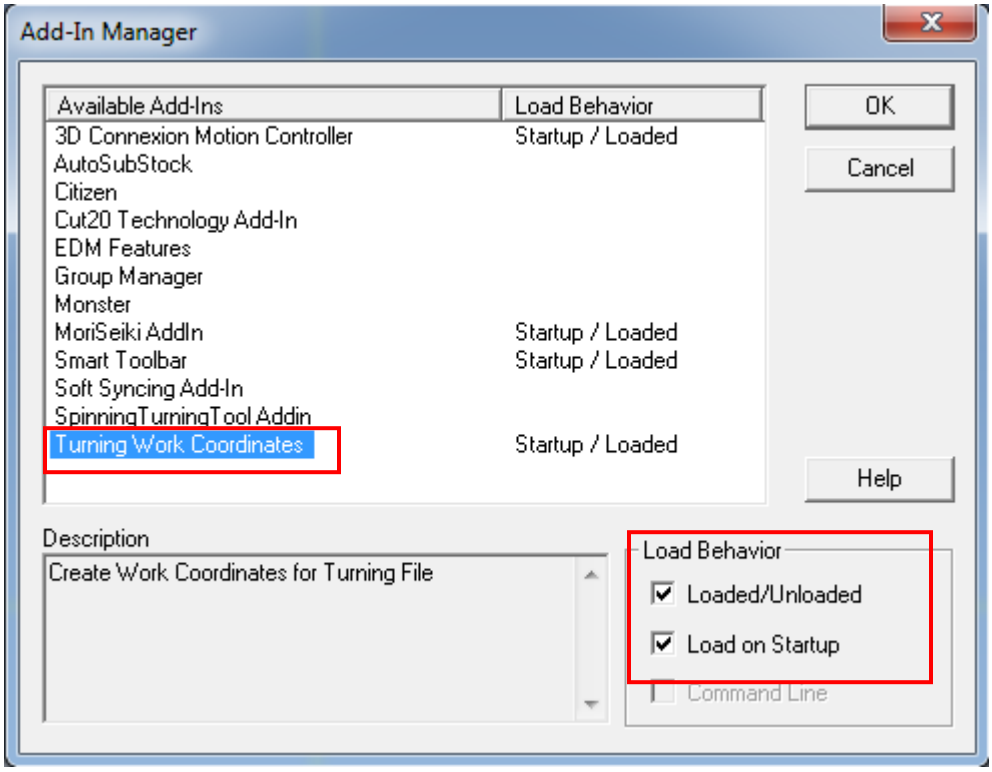
To load an add-in in ESPRIT, from the Tools menu, select Add-In...:



You will then see the add-in window:



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



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

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

- Turning Work Coordinates

Optional Addins that can also be loaded if needed:

- Spinning Turning Tool Addin
- AutoSubStock

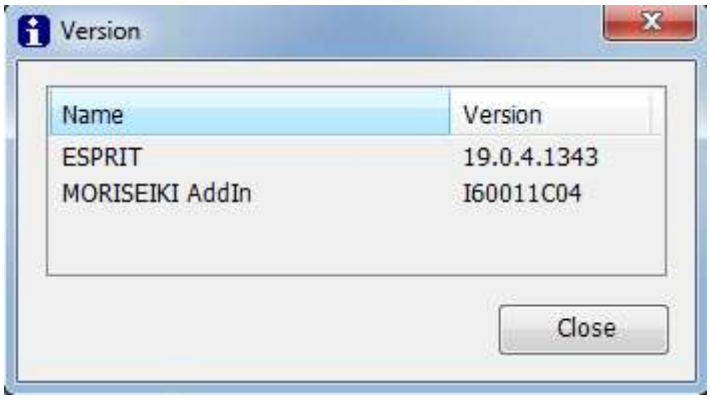
2.2. Mori Seiki ESPRIT for Mapps Addin

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

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

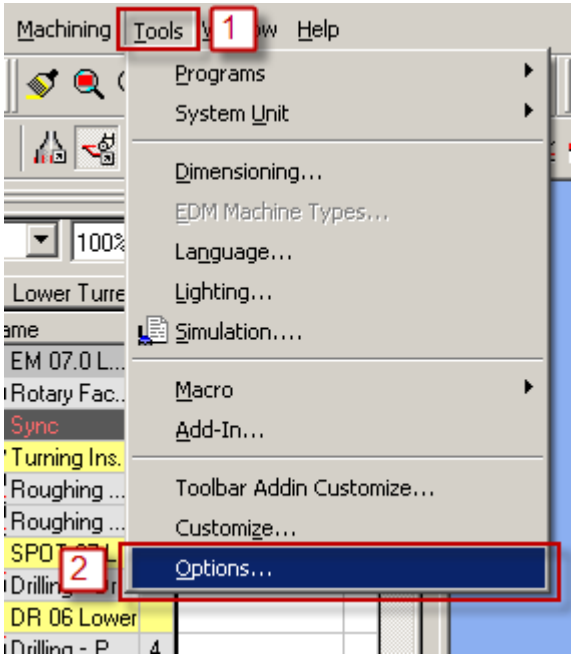
The Mori Seki AutoPark addin is integrated with the ESPRIT Utilities.

ESPRIT for MAPPS addin version I60011C04 or newer is required



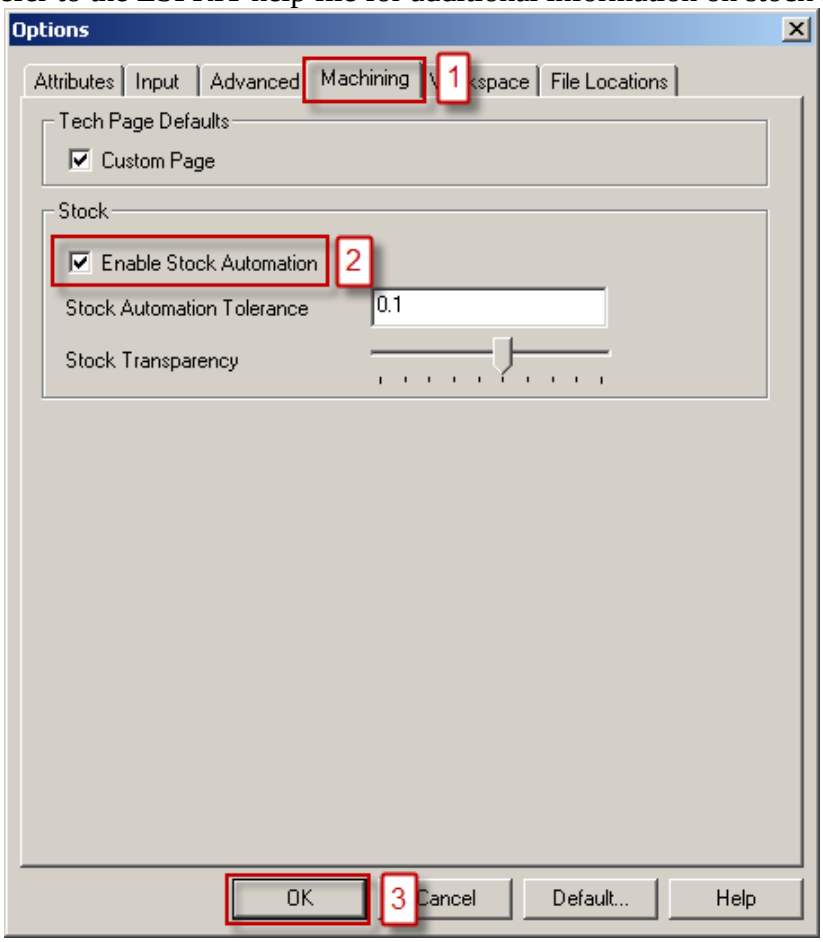
2.3. Stock Automation

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



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

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

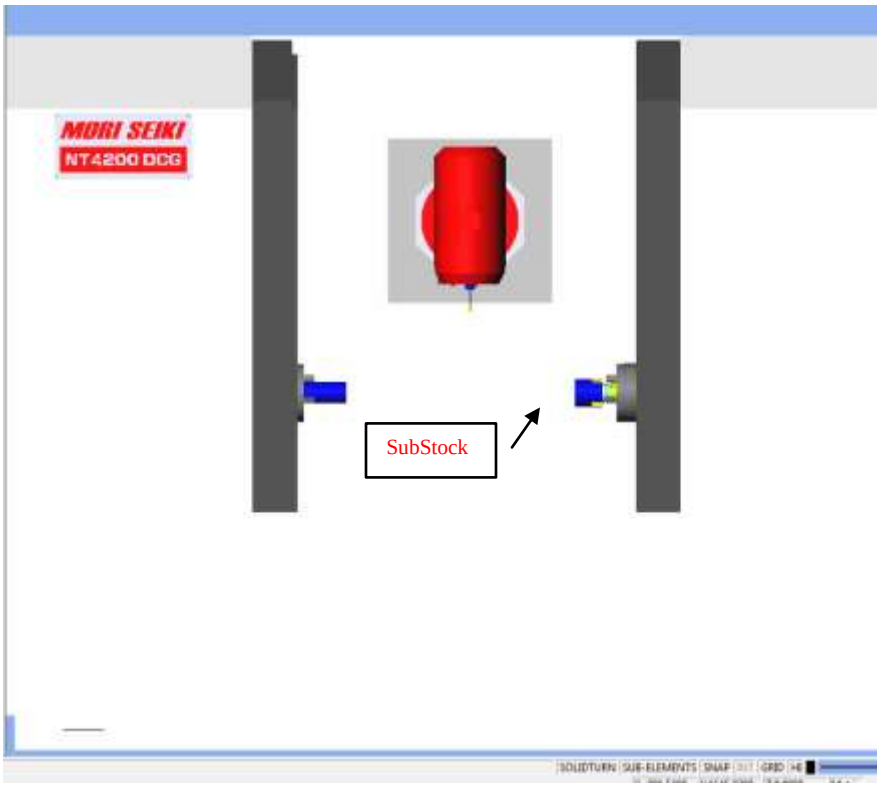


2.4. AutoSubStock (optional)

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

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

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



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

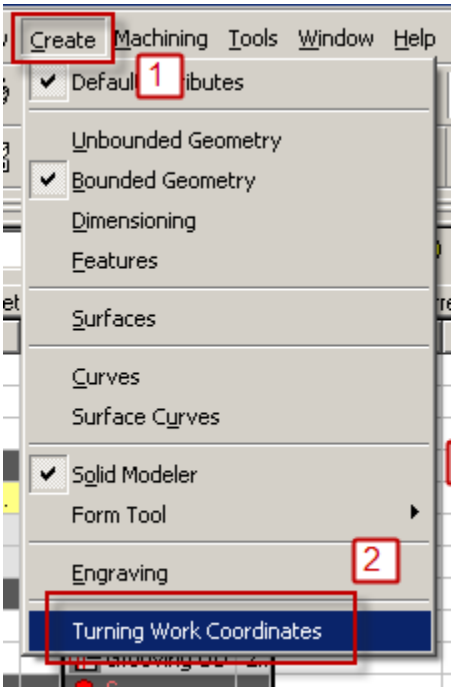
2.5. Turning Work Coordinates Addin

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

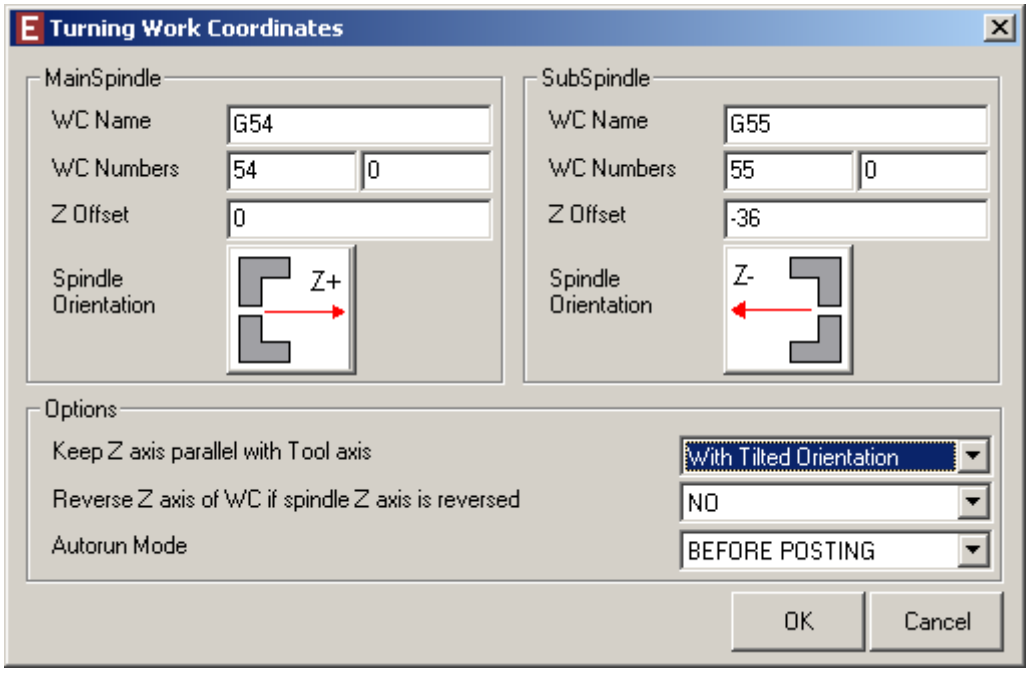
When the Addin is run the operations that have the Main Spindle selected will go under the Work Coordinate (WC) created for the Main Spindle and the operations that have the Sub Spindle. If an operation(s) are machined with the B axis at an angle other than -90, 0, +90 will be placed in a “Baxis” version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

2.5.1. Setting Up the Turning WC addin

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



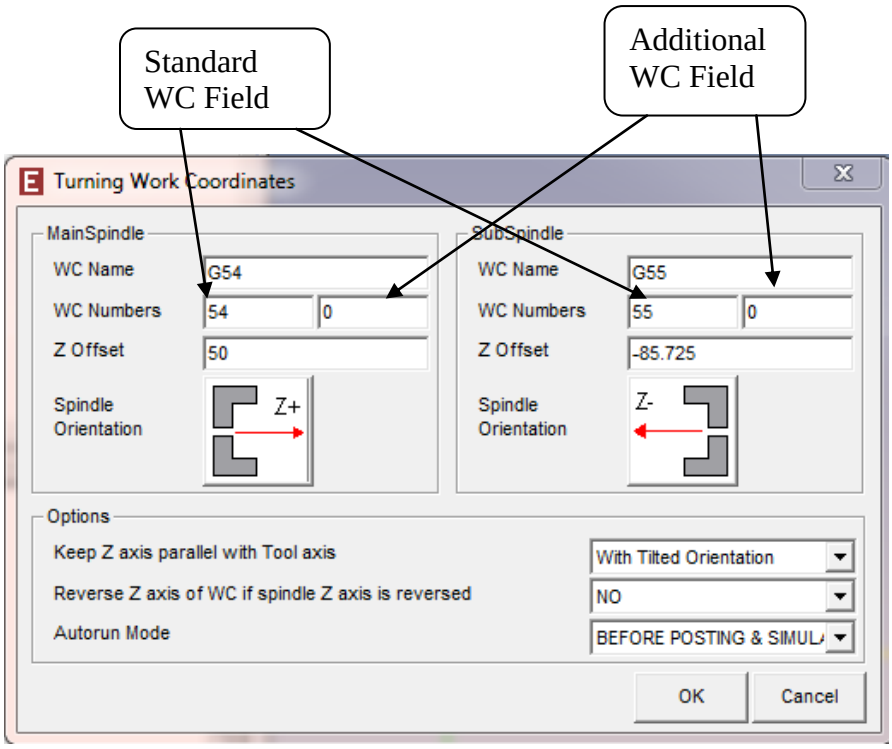
You will then see the Turning Work Coordinates dialog:



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

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

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



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

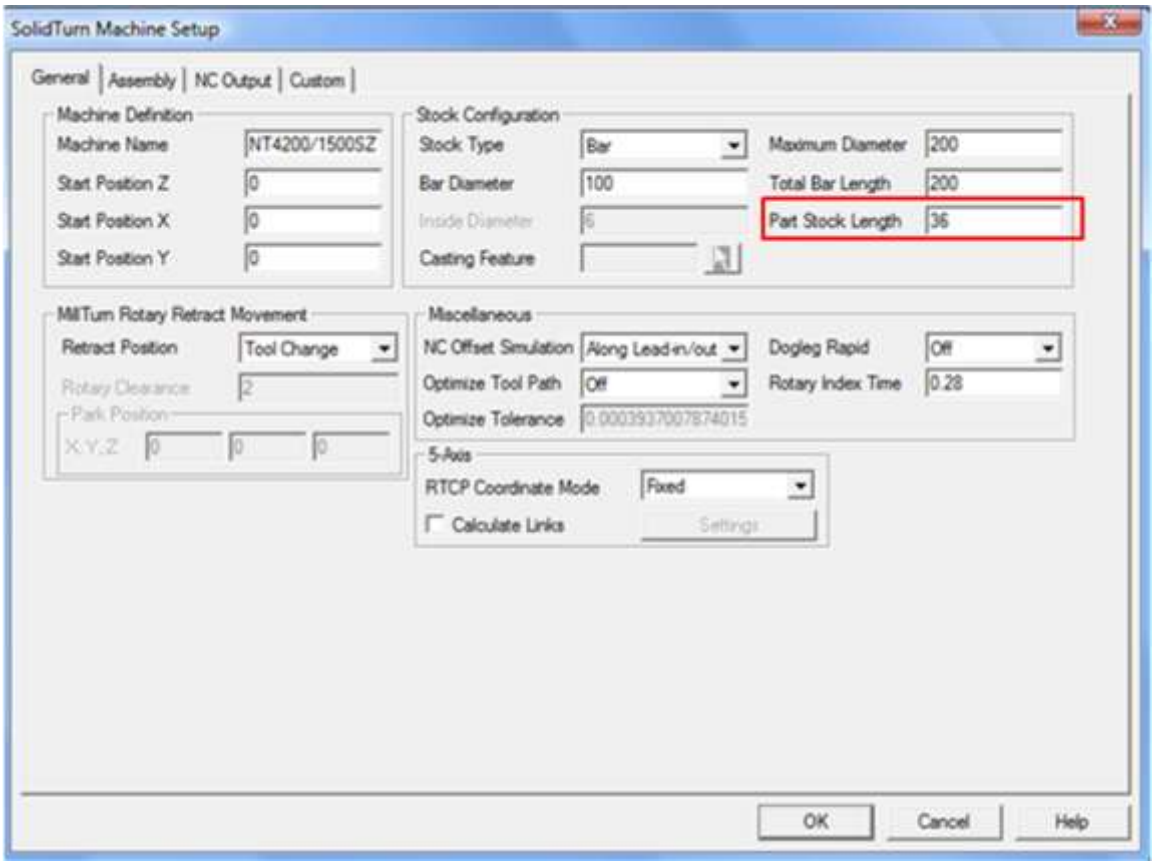
  Note: The NT1000 post has G59 is reserved for use on the Sub spindle when it is tilted. If programming an NT1000 do not enter 59 into a WC in Esprit) .

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

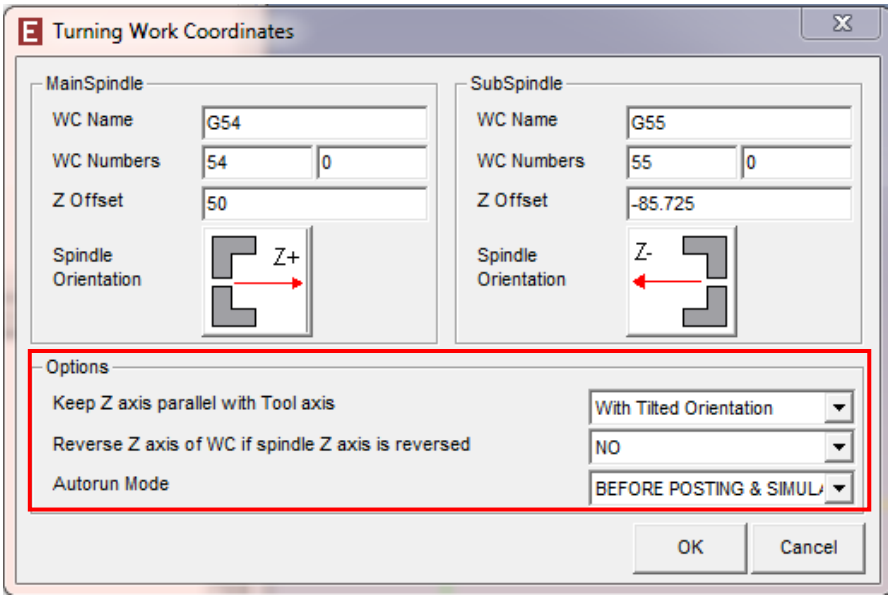
- Z Offset: Used to setup the location of the work coordinates in Esprit. This is most commonly only used for the sub spindle.
  - Main Spindle: Typically set to zero. This means the program zero for main spindle is P0 (World Zero) in Esprit.

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

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



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



Options Section:

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

2.6. AutoPark Addin (required on NT1000 machines)

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

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

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

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

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

3. Machine Setup

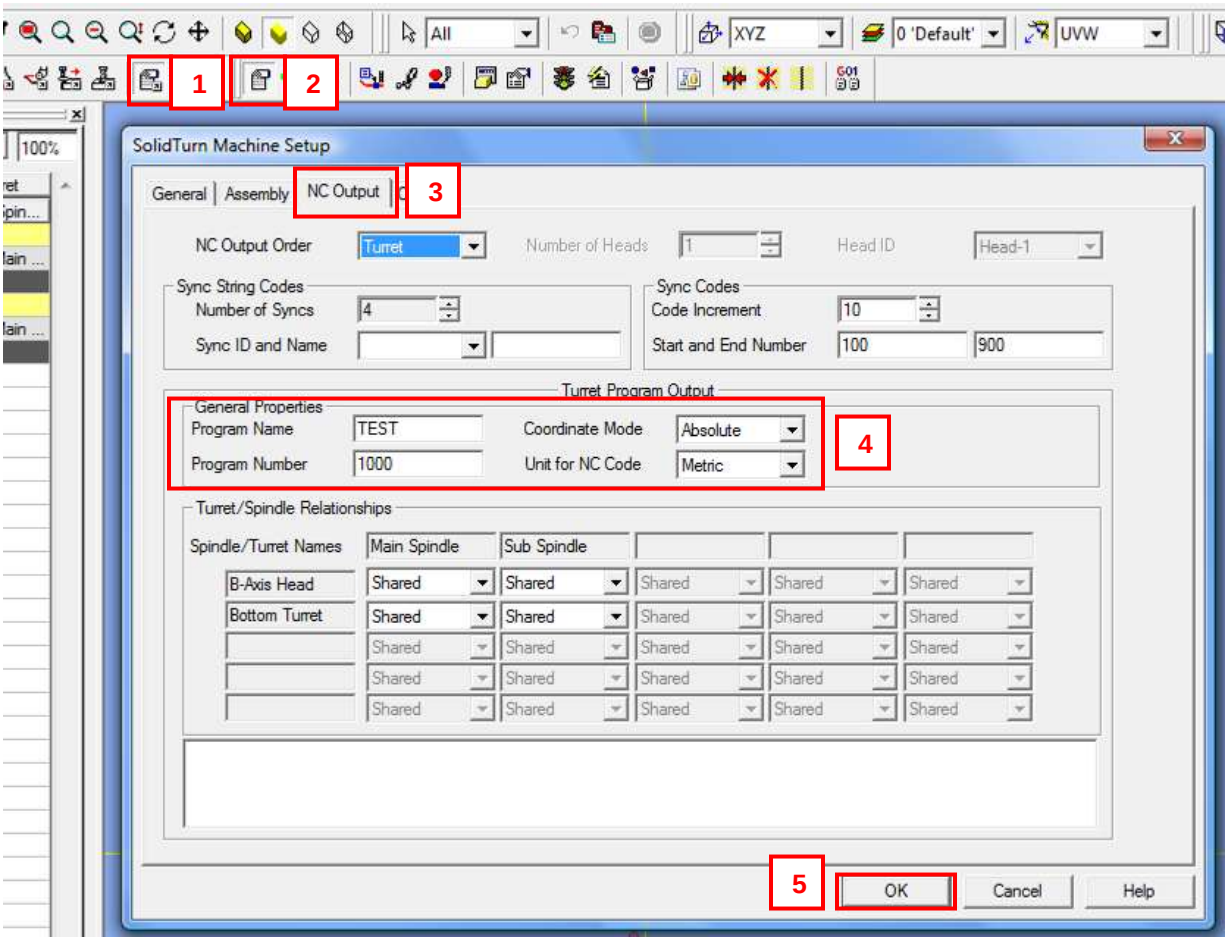
3.1. Introduction

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

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

3.2. Set the Program Name, Program Number

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



3.3. Define the Turning Stock

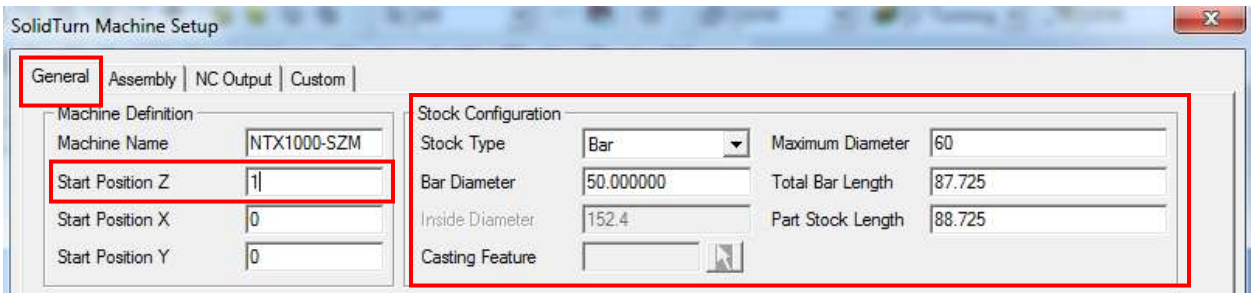
On the General tab of the Machine Setup, you can define your turning stock. For turning operations, if you are using the Stock Type Automation, your NC output will be linked to the defined turning stock.

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

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

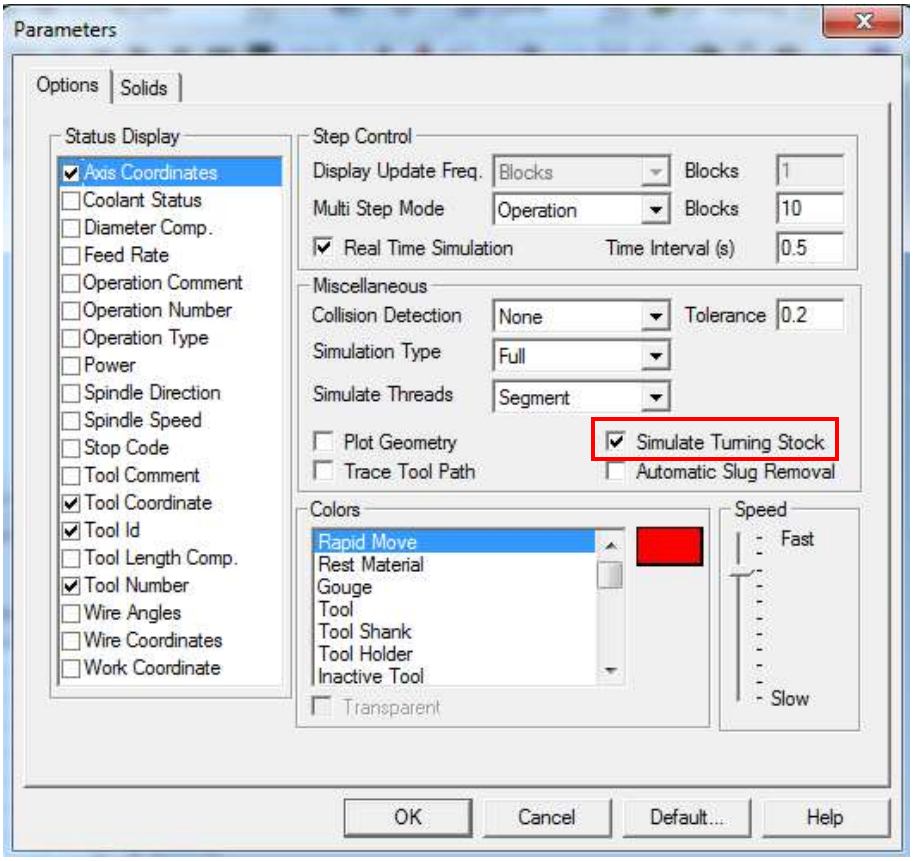
Use Bar Diameter to specify the diameter of your stock.
Use Total Bar Length to define the total length of your stock.

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



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

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

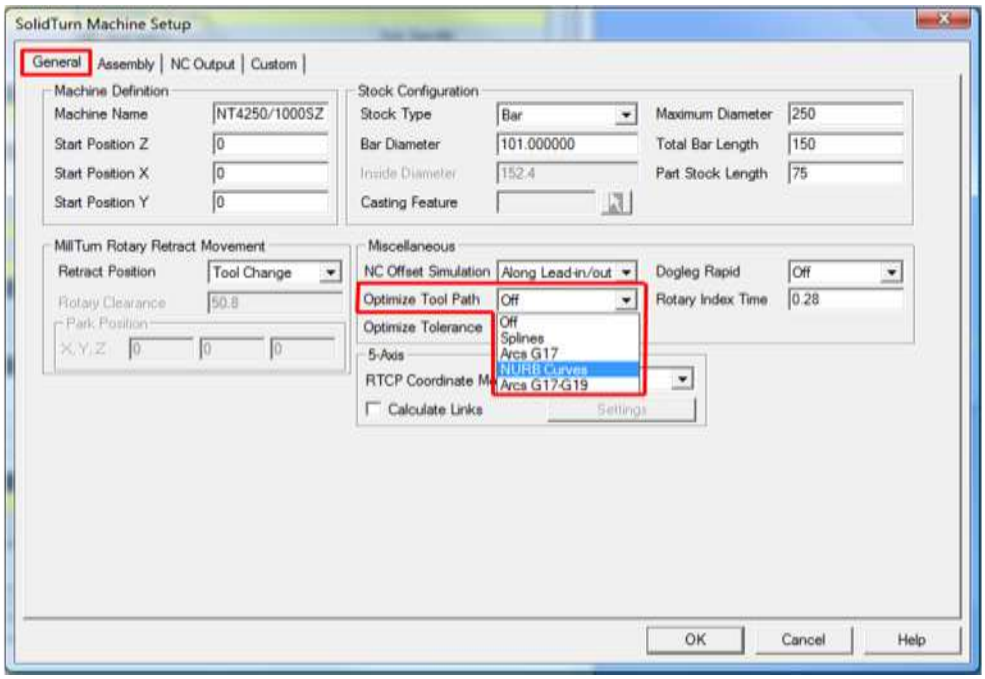


3.4. Tool Path Optimization with NURBS

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

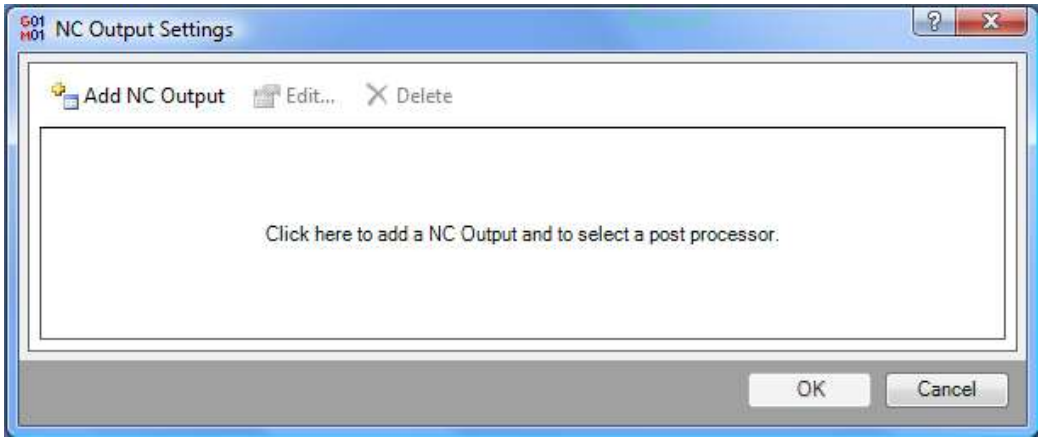
To turn on NURBS Optimization in ESPRIT, navigate to the Machine Setup page and the General tab. Under "Miscellaneous" there is a pull down labeled "Optimize Tool Path" open this and scroll down to "NURBS Curves" to select NURBS optimization. Click OK to complete this function.

Machine Setup General Tab

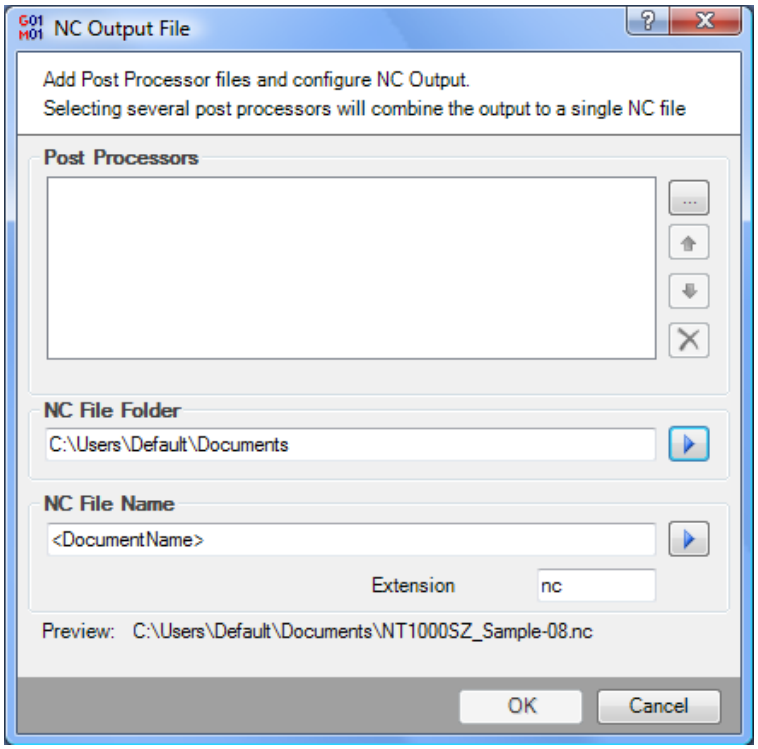


3.5. Selecting Post Processors and Outputting NC Code

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



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

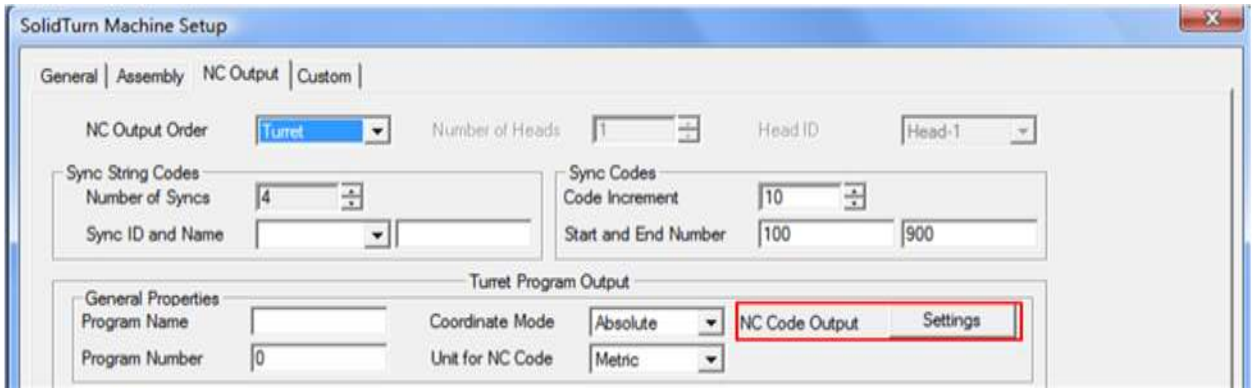


For multi-turret machines such as the NT-SZ, NT1000WZ and NT1000SZ, the user should select each turrets post processor separately and give each file a unique NC File Name. Once the user has selected one post processor, they are able to select Add NC Output again and create a new NC Output file for the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

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

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

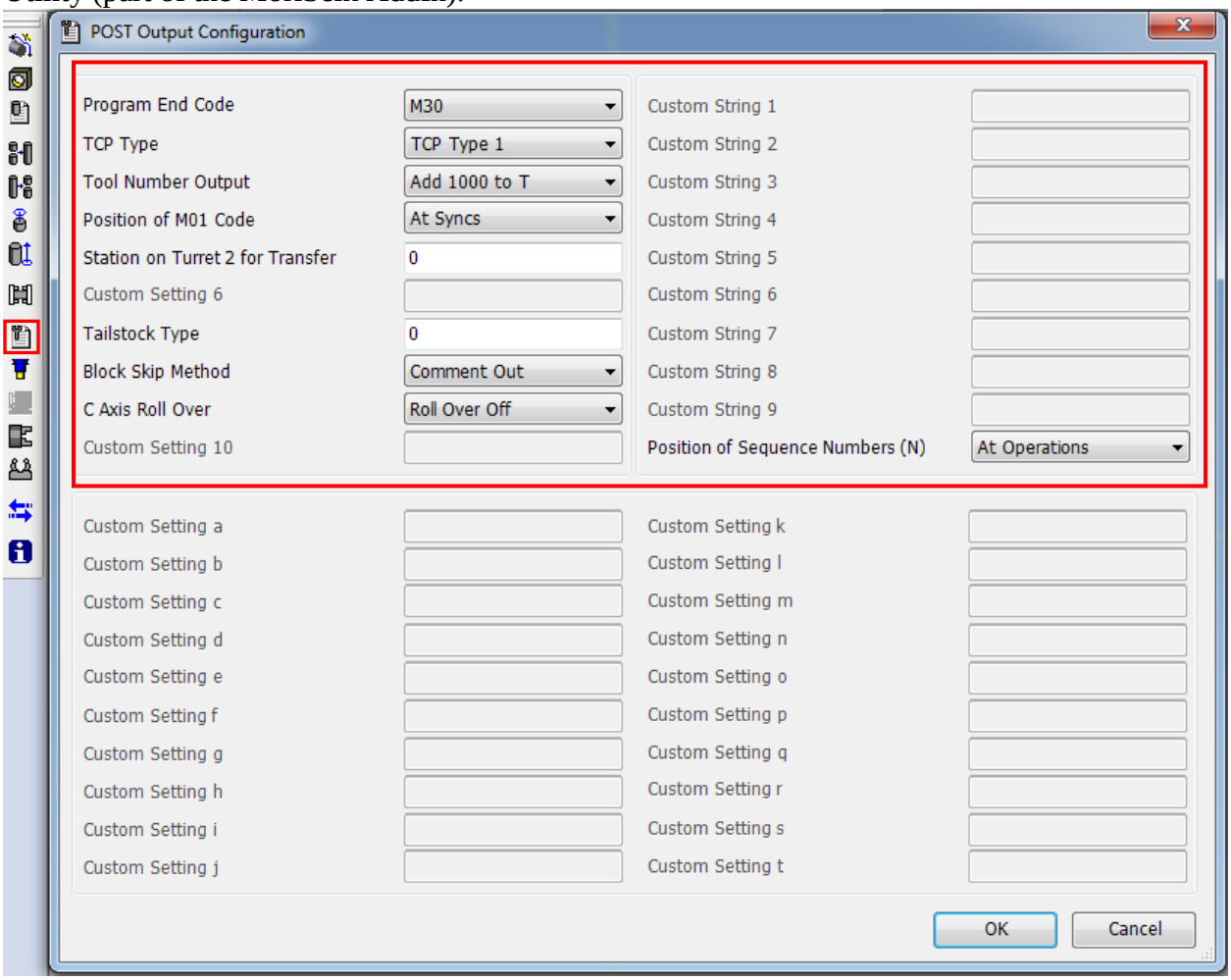
Note: If the user receives new post processors or would like to make any changes to the file location or file name, they are able to do this by navigating to the Machine Setup in the Common Machining toolbar. On the "NC Output" tab, there is a section on the right hand side called NC Output Settings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



3.6. Machine Parameters

3.6.1. Introduction

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



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

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

Please note that M30 is the default.

3.6.3. TCP Type

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

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

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

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

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

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

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

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

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

3.6.5. Station on Turret 2 for Transfer

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

By default this value is 0

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

For additional information see the Park Cycles section of this manual.

3.6.6. Tail Stock Type

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

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

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

By default this setting is 0

For additional information see the Tailstock section of this manual.

3.6.7. Block Skip Method

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

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

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

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

By default Comment out is used.

3.6.8. C-Axis Rollover

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

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

By default Roll Over Off is used.

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

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

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

In ESPRIT use **Roll Over Off**

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

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

In ESPRIT use **Roll Over On**

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

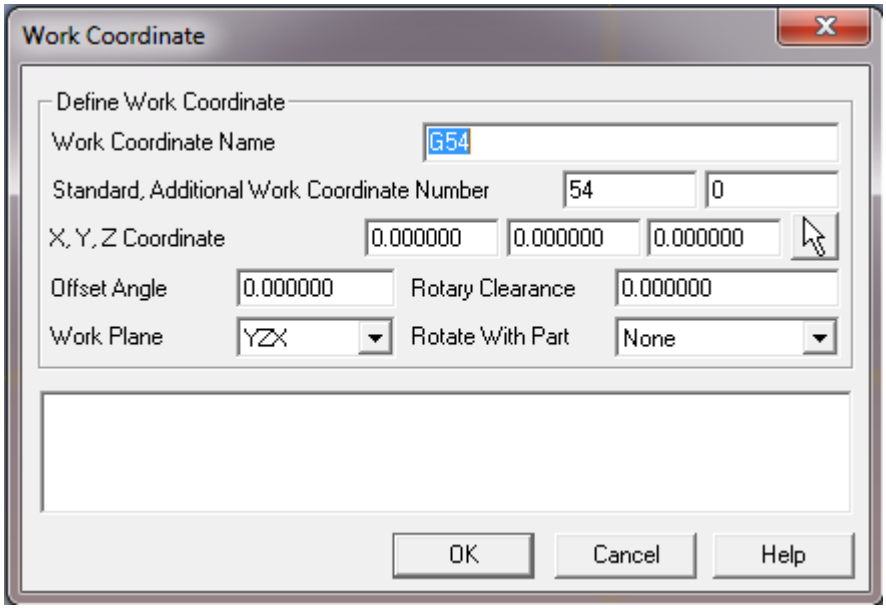
- C-axis will be indexed with C values.
- SolidMillTurn 4-axis wrap and rotary operations with interpolation off cannot be programmed.

4. Work Coordinates

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

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

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



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

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

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

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

- Offset Angle: Must be set to 0

- Rotary Clearance: is not currently used by the NT posts

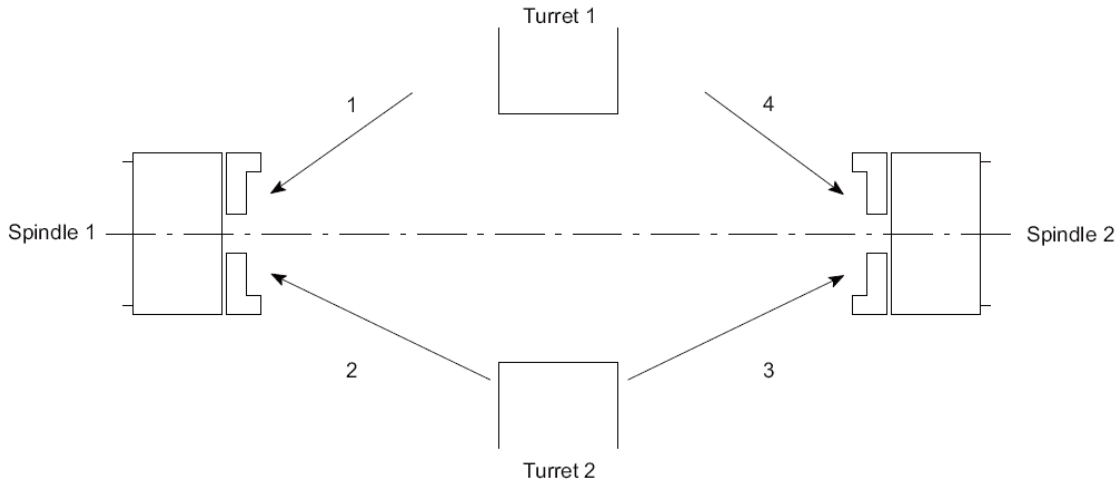
- Work Plane: it is Critical that the Work plane selected is either YZX or a plane that is parallel to the YZX plane.

- Rotate with Part MUST be set as follows:
    - For turning operations any MillTurn operations at B-90, B0, and B90 set this to "None"
    - For any MillTurn operation that is NOT at B-90, B0, and B90, set this to "Point only"

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

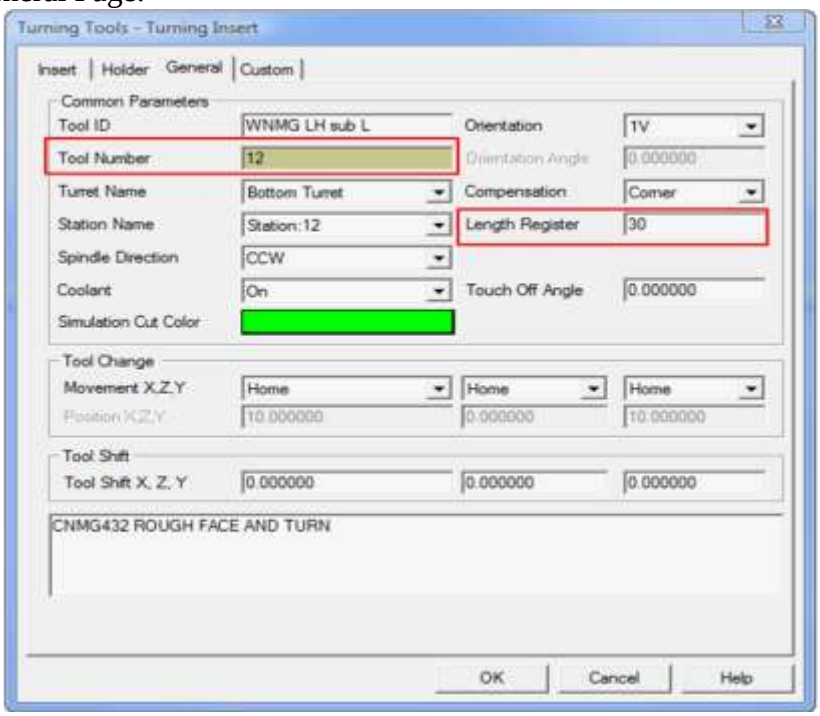
5. Tools (T-function)

5.1. T-function Specifications

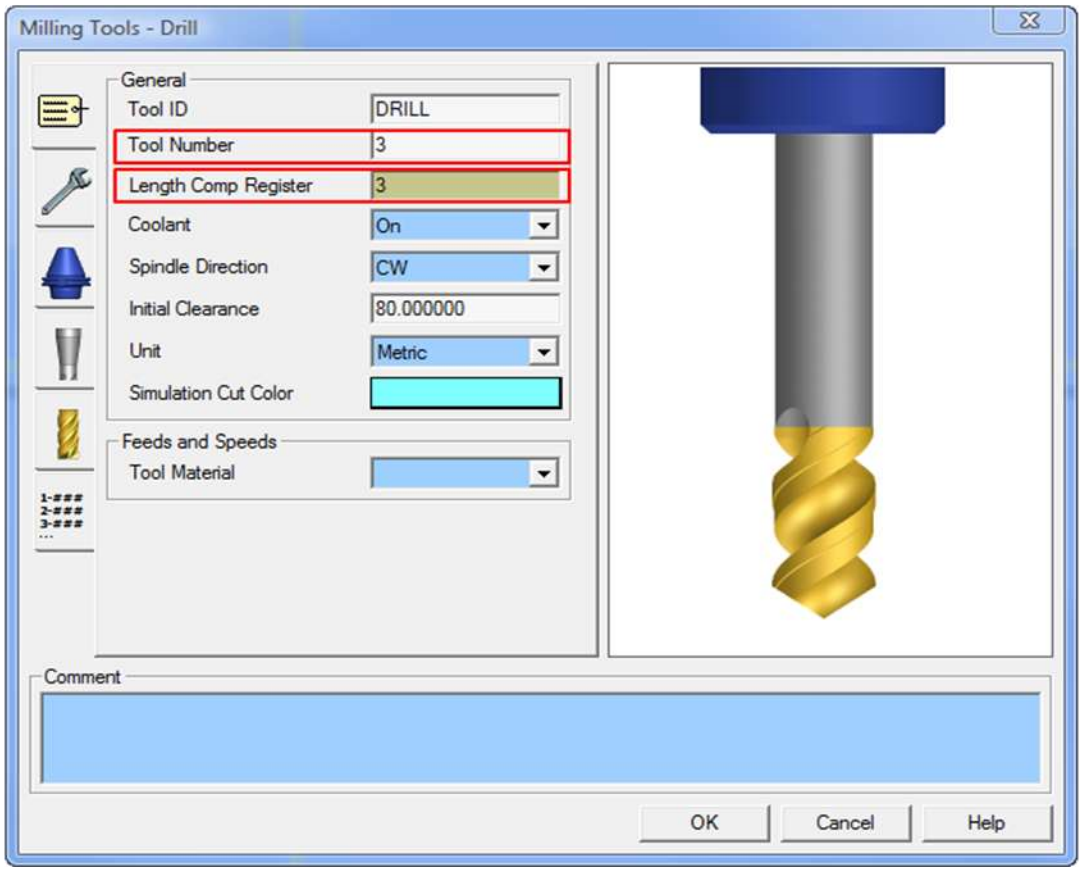


	Turret and Spindle to Use		T Code to Specify
1	Turret 1	Spindle 1	T1001 – T1099
2	Turret 2	Spindle 1	T0101-T1212
3	Turret 2	Spindle 2	T0131 – T1242
4	Turret 1	Spindle 2	T1001 – T1099

Turning Tool General Page:



Milling Tool General Page:

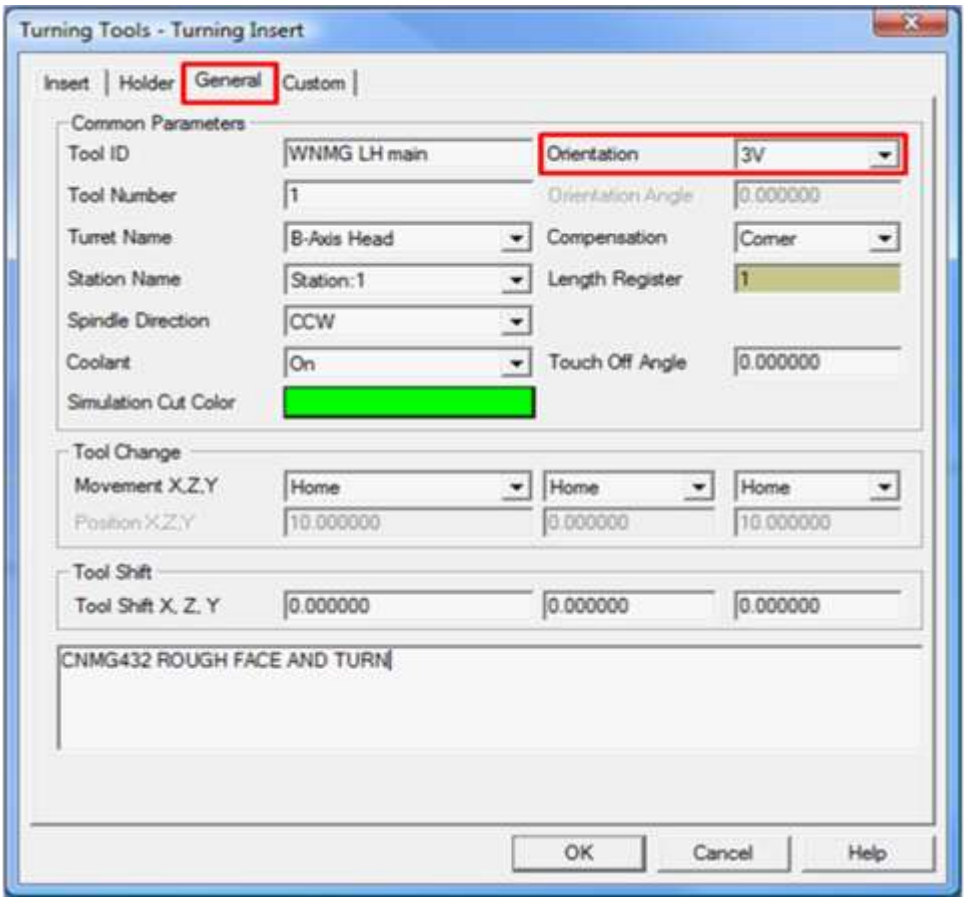


5.1.1. Upper Head (B-Axis Head):

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

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

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



5.1.2. Lower Turret:

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

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

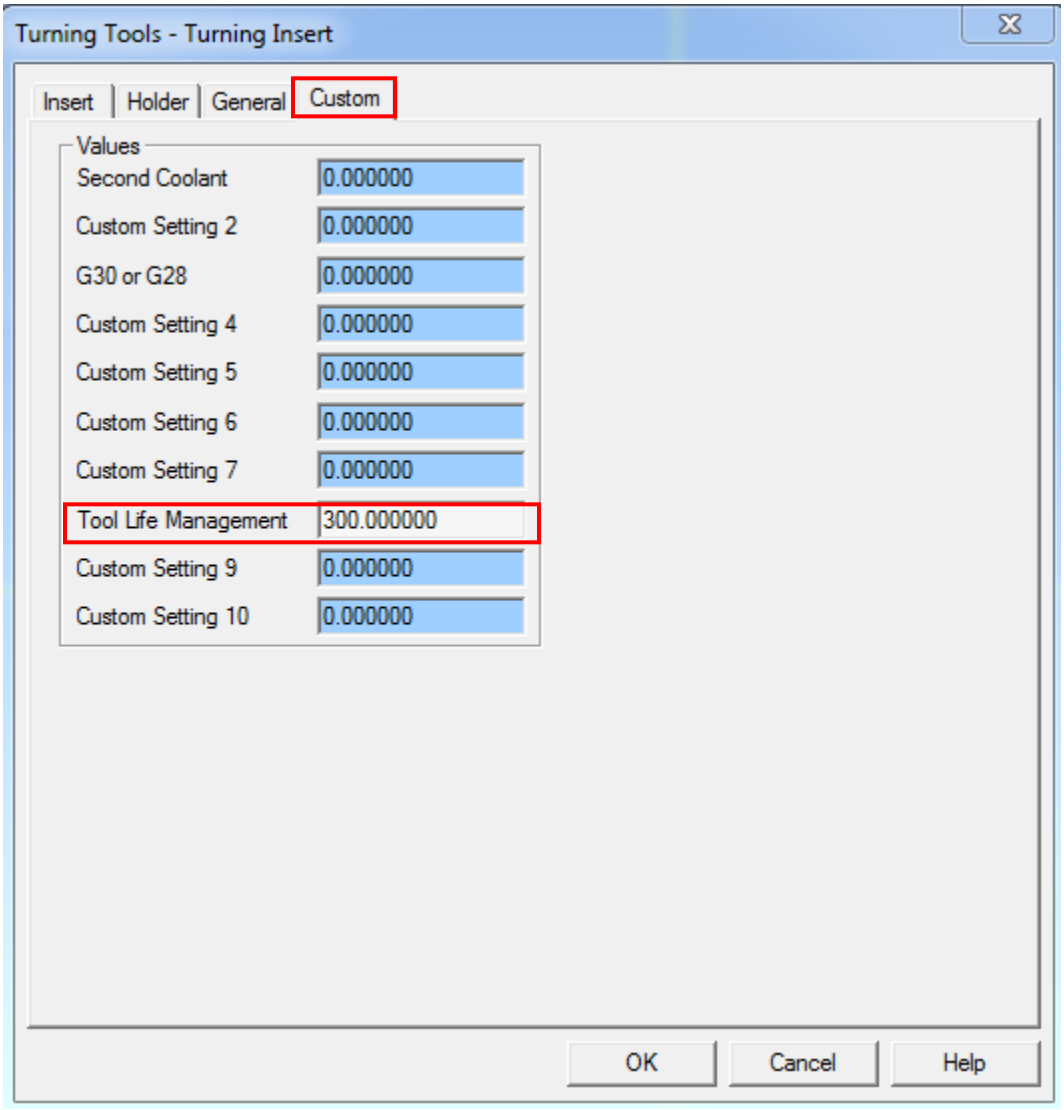
For the NT machine series when machining on spindle 2 (Sub Spindle), the post processor will add 30 to the tool number entered in the Tool Number Dialogue Box if that tool is mounted on the Lower Turret (Head 2/Turret 2). If 1010 is entered in the Tool Number Dialogue Box, T1040 will be output in the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

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

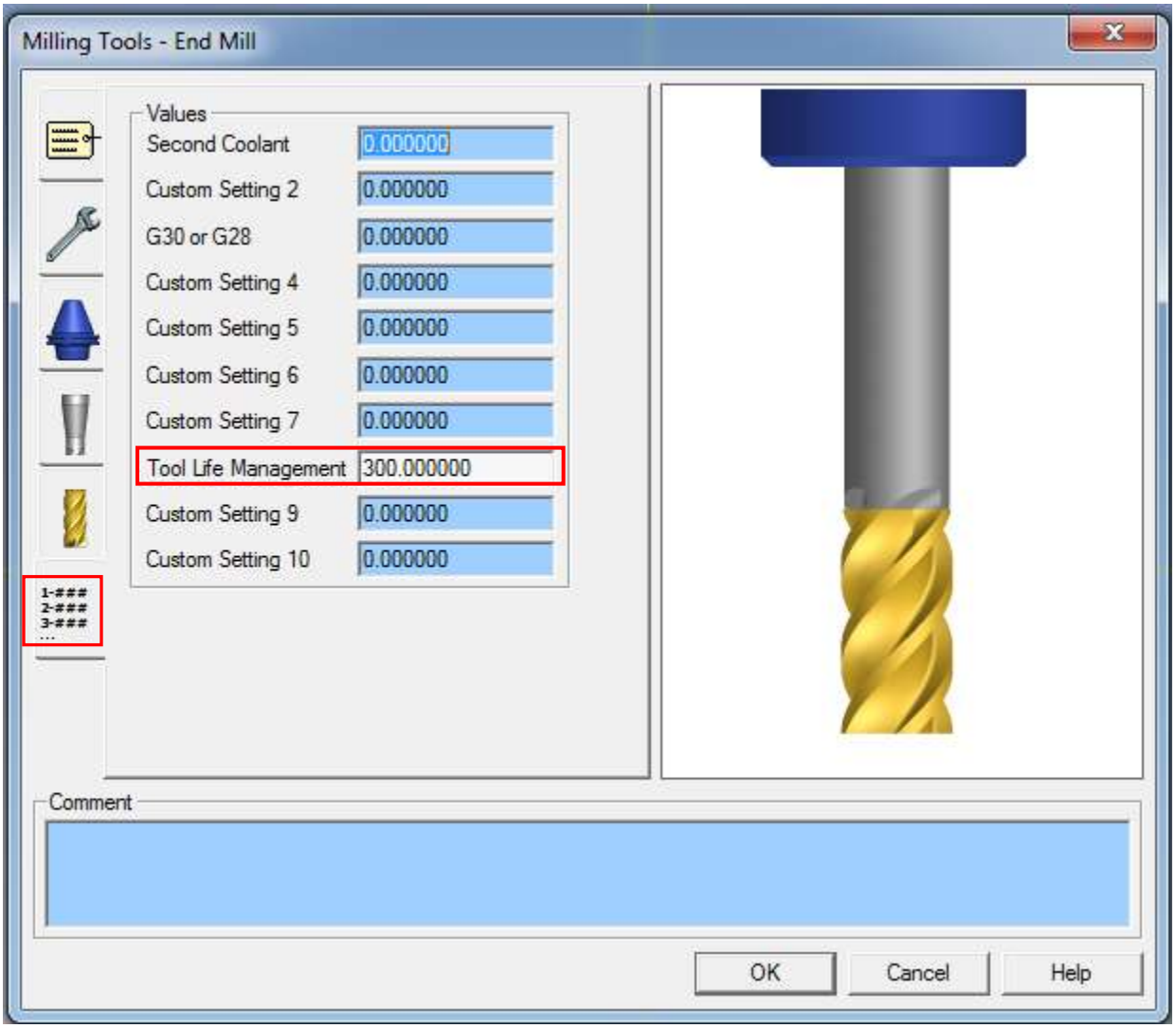
5.2. Tool Life Management

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

Turning tool page:

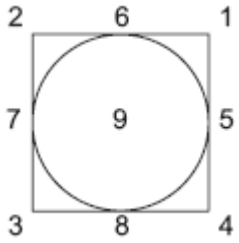


Milling tool page:



5.3. Tool Nose Direction (G43 H_ T_)

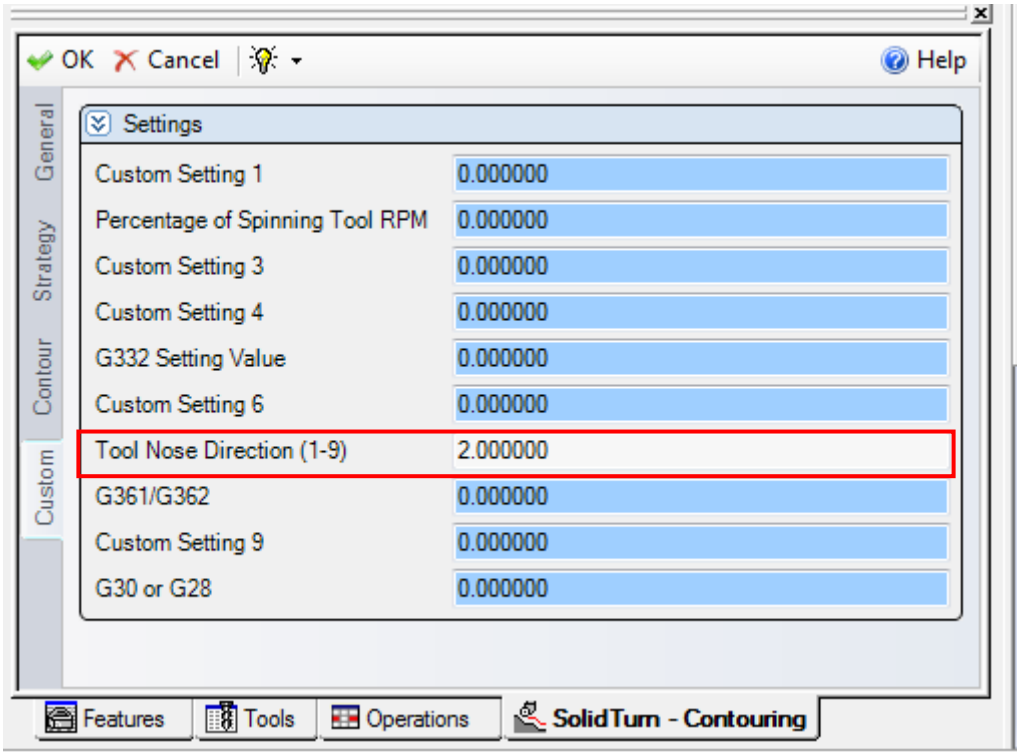
When using a turning tool in the B-axis head of the Mori Seiki NT, it is possible to specify the direction of the tool nose to be offset according to the following diagram:



The corresponding G code is:

G43 H_. T_.

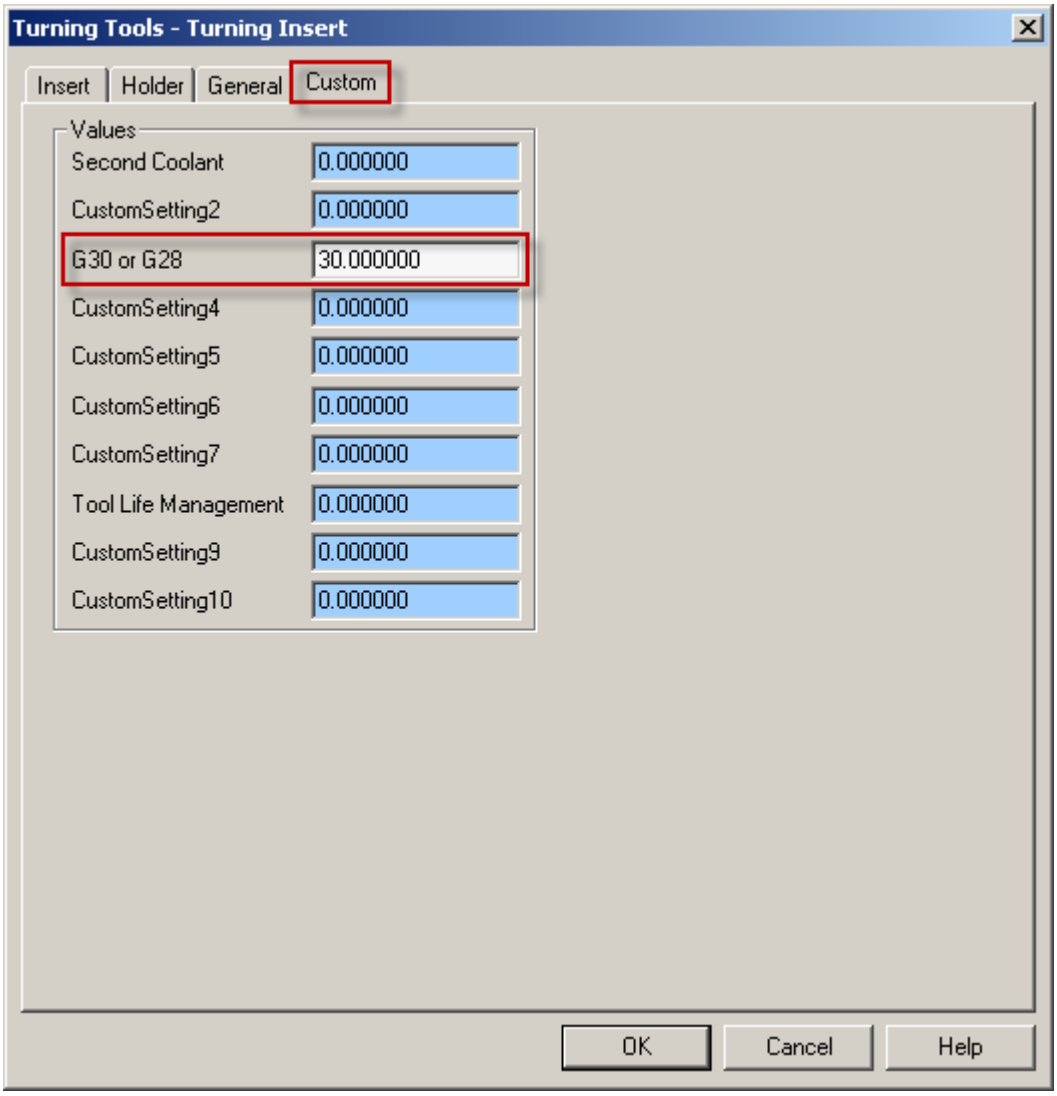
To program this in ESPRIT, go to the Custom tab of any operation using a turning tool and set the Tool Nose Direction (1-0) (Custom Setting 7) field to any value between 1 and 9. Values >9 or <1 will omit the T_ output.



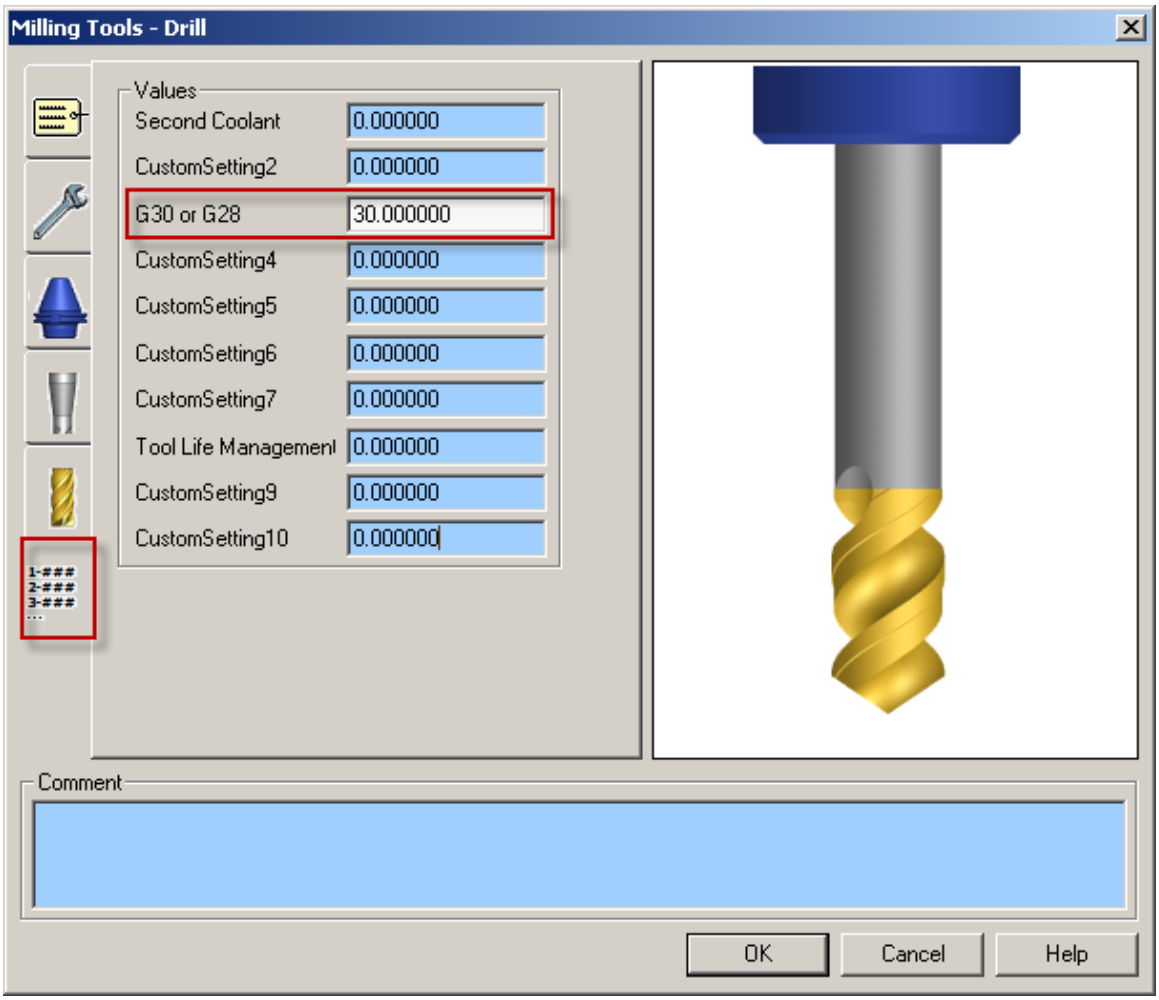
5.4. Second home position

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

Turning tool page:



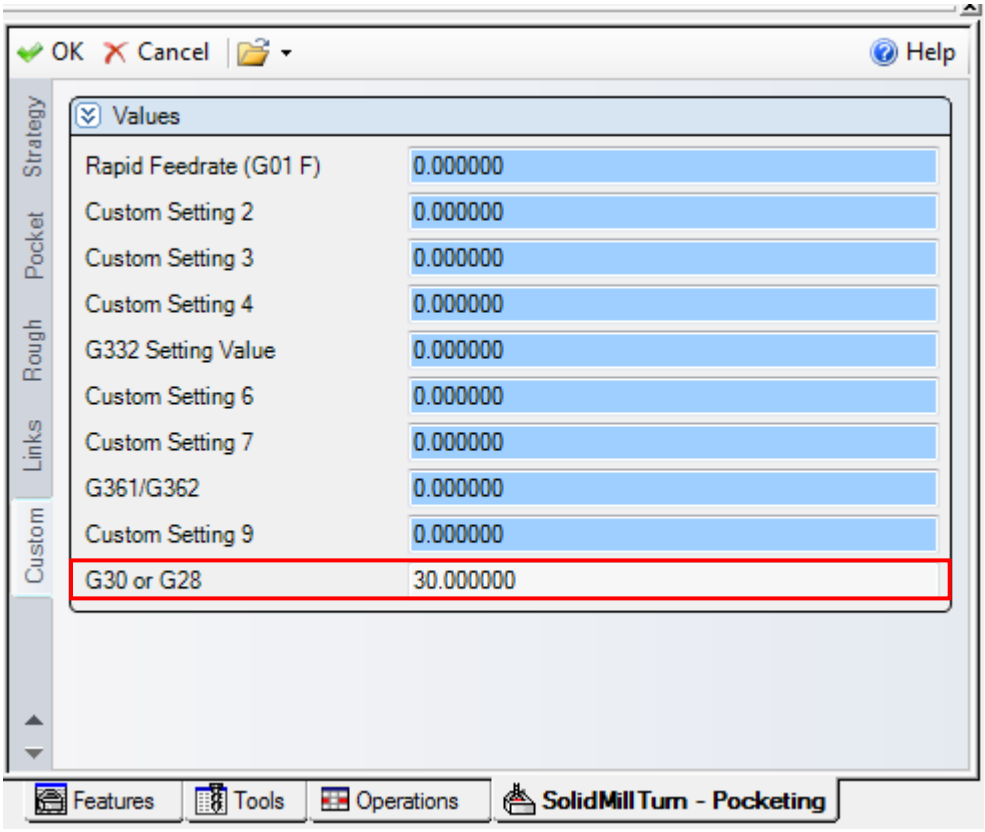
Milling tool page:
:



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

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

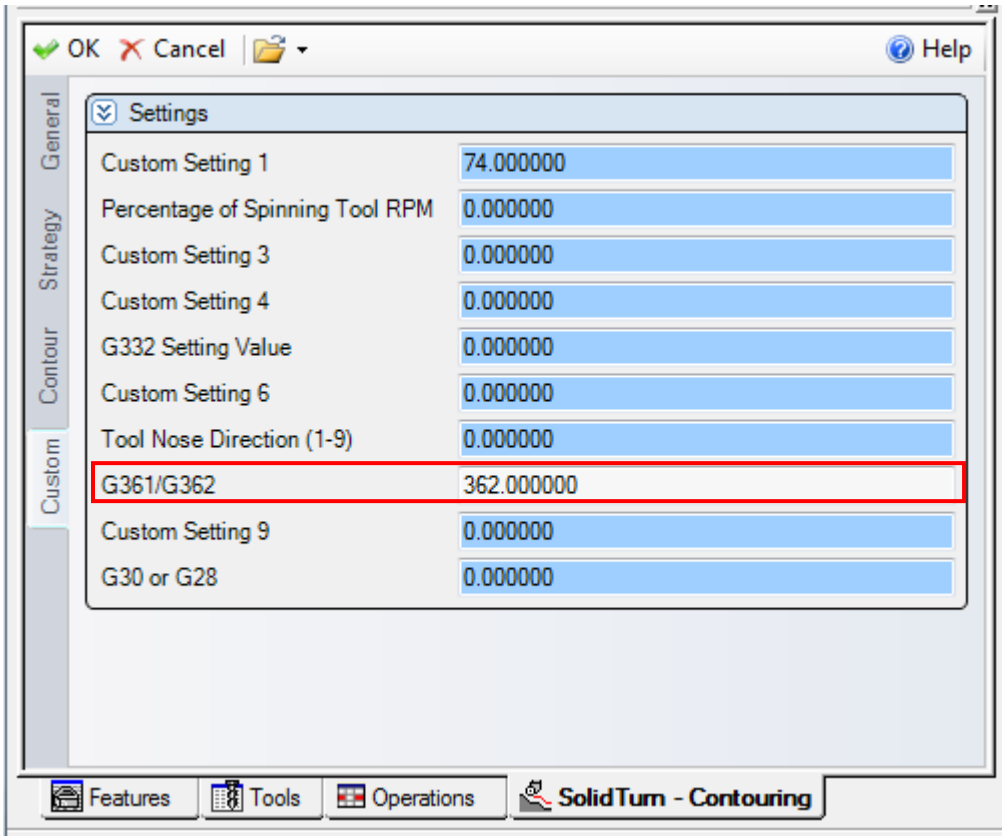
This can also be set within a specific operation page as well as shown below with the “G30 or G28” (Operation Custom setting 10)



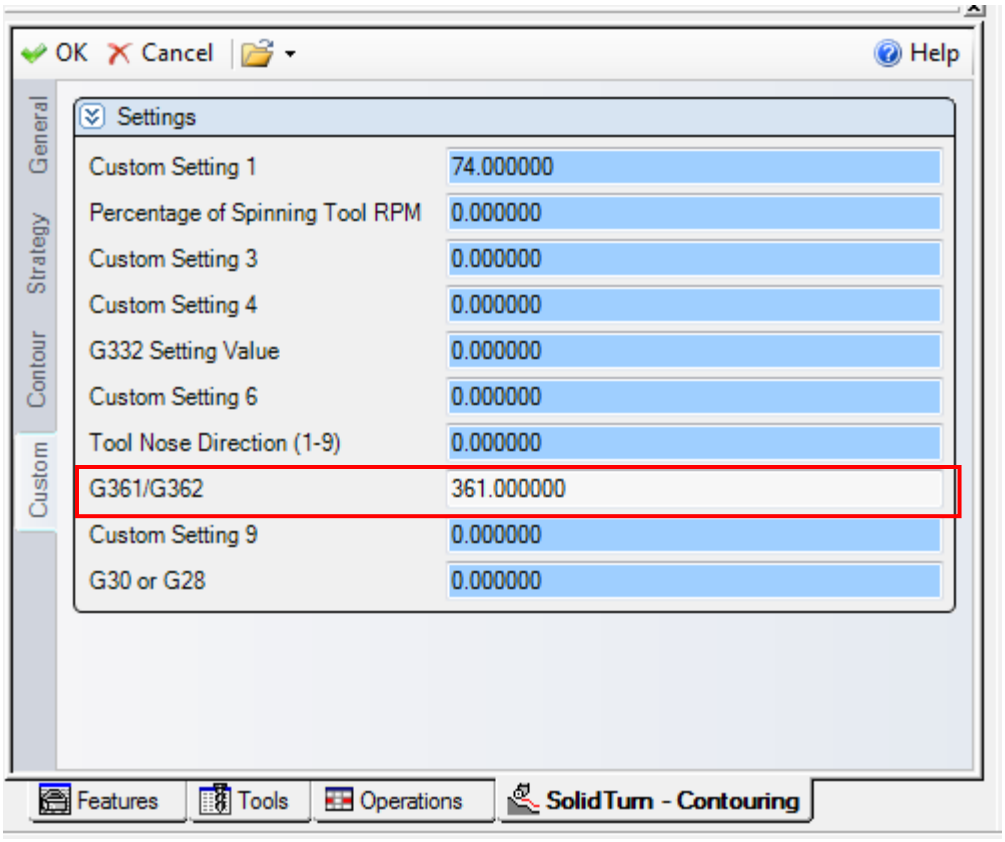
5.5. G361/362 Tool Change Command Setting

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

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

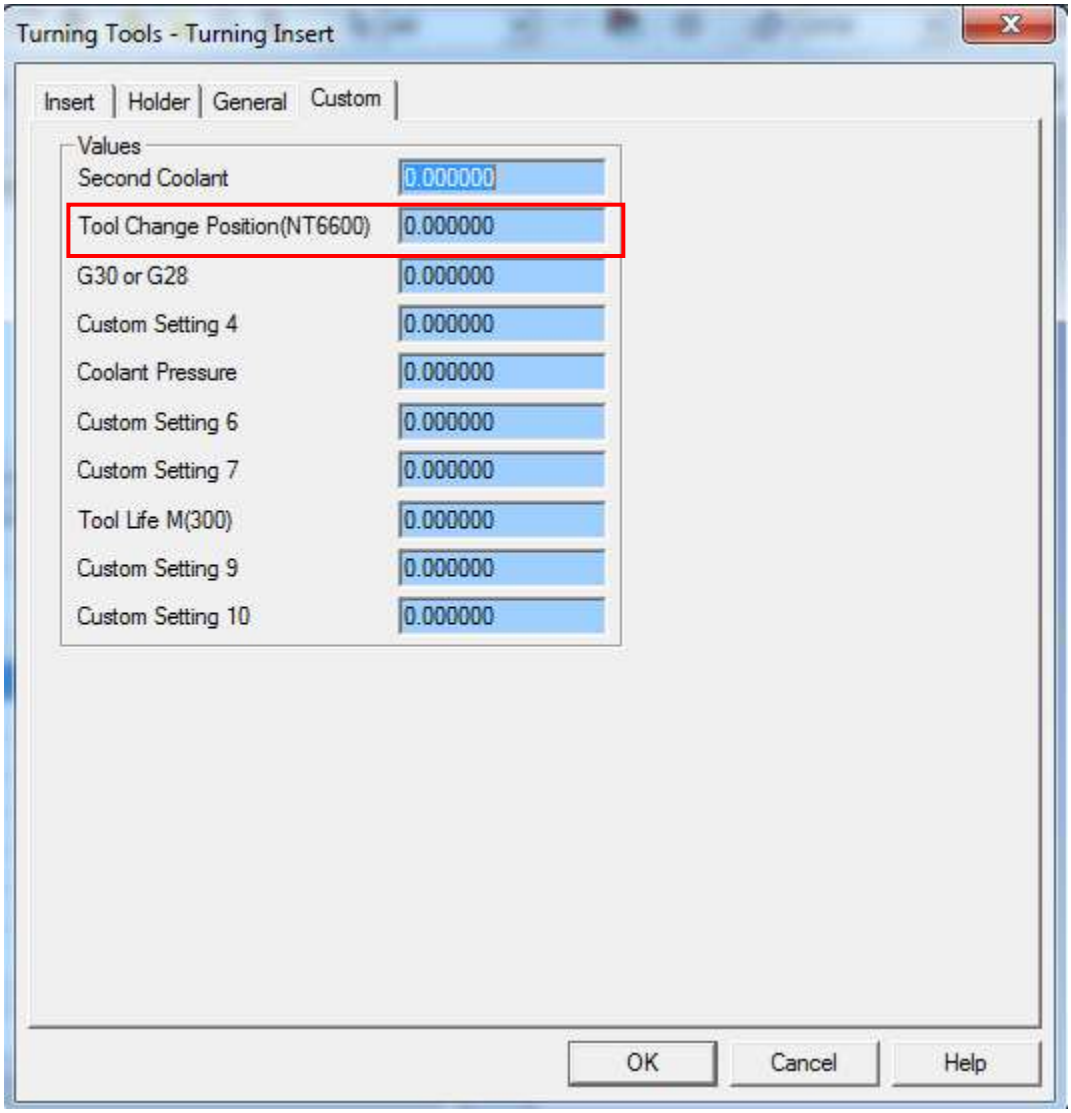


For operations where both spindles are holding the part (simultaneous machining), the G362 tool change call is standard. If the user would like to instead output G361, set the "G361/G32" to 361. If any value other than 361 is input, G362 will be output in the NC code.


OK
Cancel
Help
General
Strategy
Contour
Custom
Settings
Custom Setting 1
74.000000
Percentage of Spinning Tool RPM
0.000000
Custom Setting 3
0.000000
Custom Setting 4
0.000000
G332 Setting Value
0.000000
Custom Setting 6
0.000000
Tool Nose Direction (1-9)
0.000000
G361/G362
361.000000
Custom Setting 9
0.000000
G30 or G28
0.000000
Features
Tools
Operations
SolidTurn - Contouring

5.6. Tool Change position

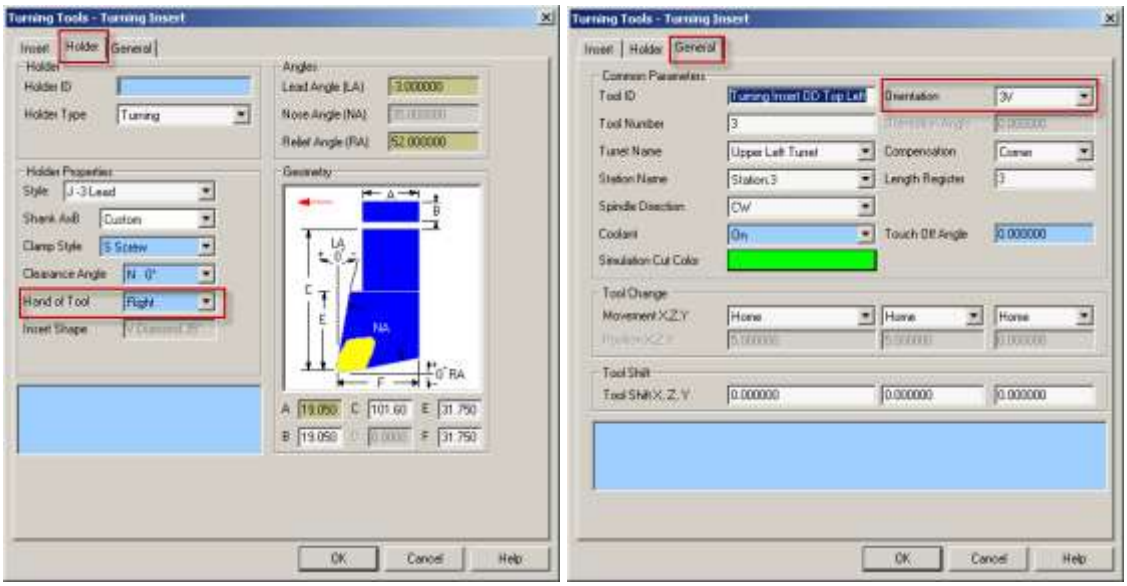
NT-S 6600 machines have the ability to change the tool change position. This is done with an R value on G361/G362 line. This can be programed by setting the desired position number into the Tool Change Position (Custom Setting 2) field.



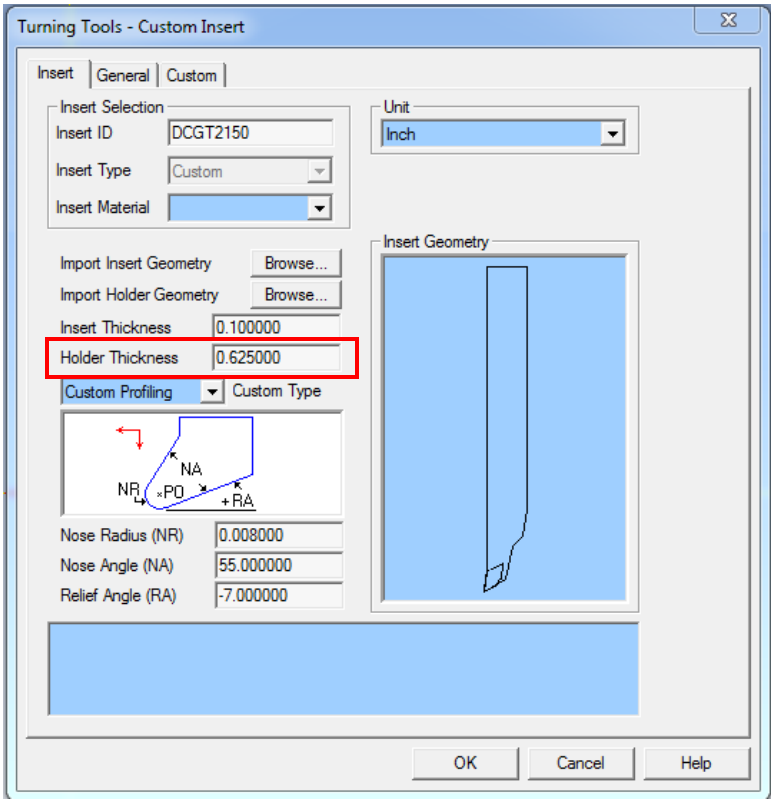
6. Spindle Direction, Speed, and Feedrate

6.1. Turning Spindle Direction and Milling Tool Spindle Direction

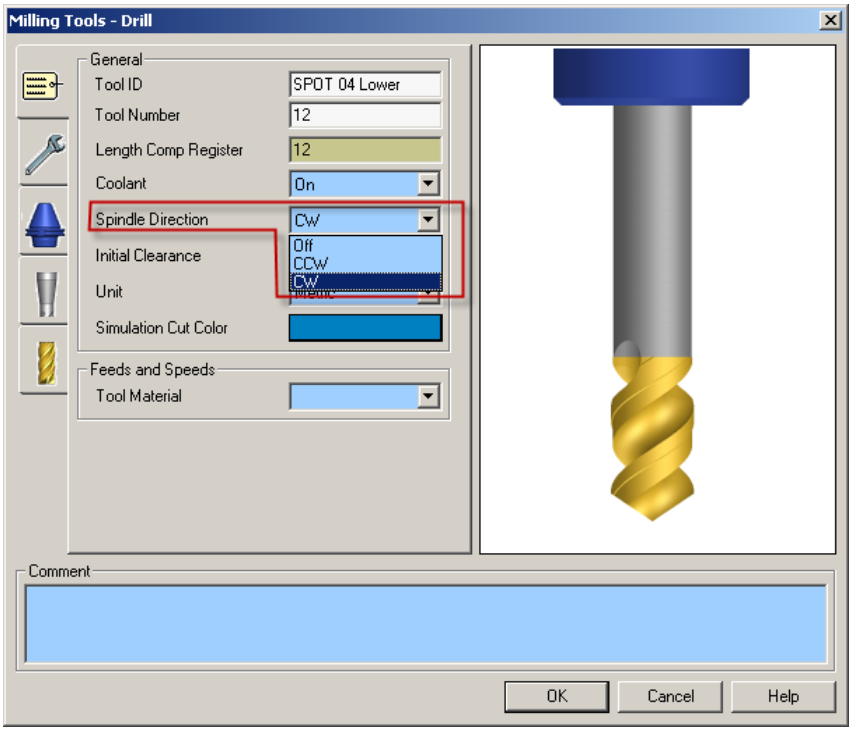
For turning operations using a cutting insert (Roughing, Balanced Roughing, Contouring, Grooving and Threading), the spindle direction will automatically be computed by the post processors and will output the correct M-code (M03, M04, M203 or M204) based on the hand of the tool on the holder and the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



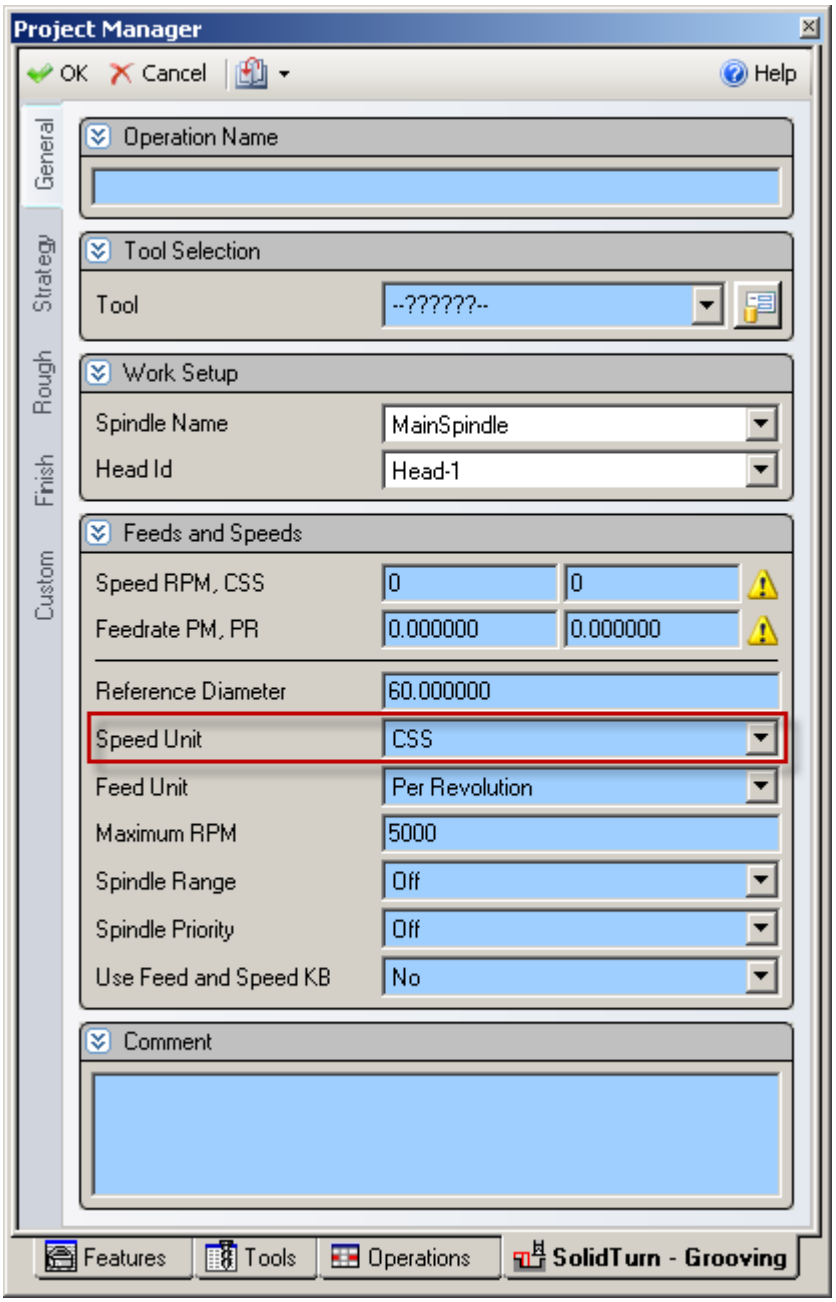
Custom Turning tools do not have the option to select if the tool is a right or left hand tool. In this case we use the Holder Thickness field. If this value is a positive value the post will know this is a left hand tool, if it is a negative value it is a right hand tool.



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



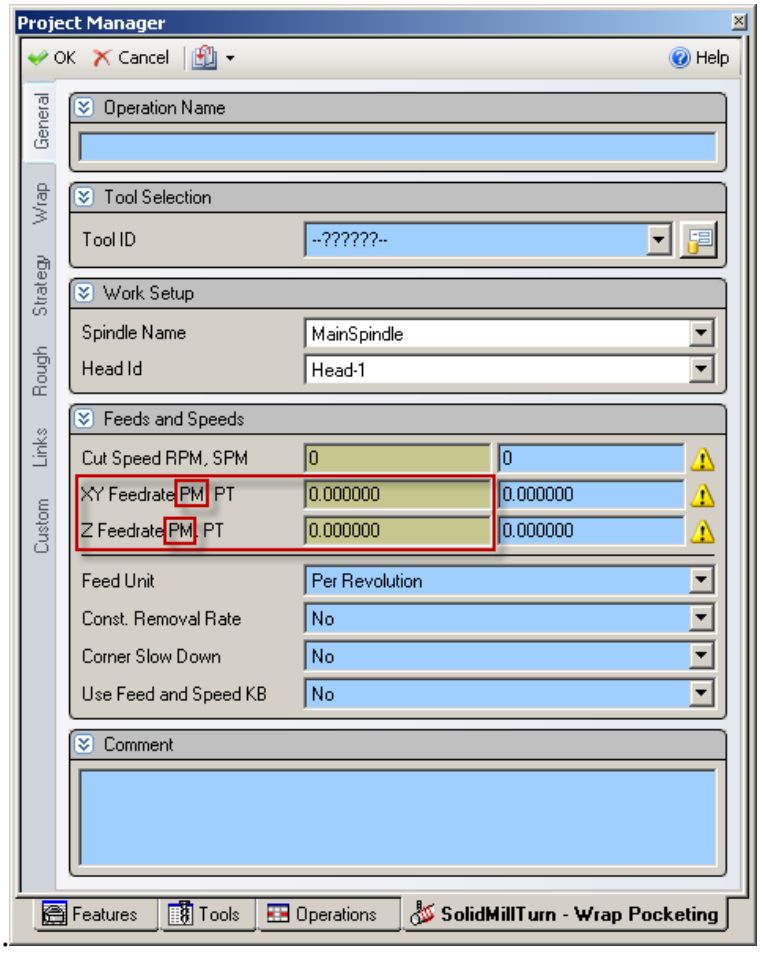
On a turning operation page, if you choose to output the turning spindle speed in the unit CSS (Constant Surface Speed), a warm up speed will first be output with the G97 code before the first positioning move. The speed value is computed at the first diameter from where the CSS command will be turned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

6.3. Feedrate for 4-axis Milling Operations

For Wrap Pocketing and Wrap Contouring operations with Cylindrical Interpolation set to No or Rotary Face Pocketing and Rotary Face Contouring operations with Polar Interpolation set to No, a rotary feedrate will be computed by the post processor, based on the linear and rotary (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 inch/min (or mm/min) and rotary feedrates (angular moves around the C-axis) in deg/min.

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

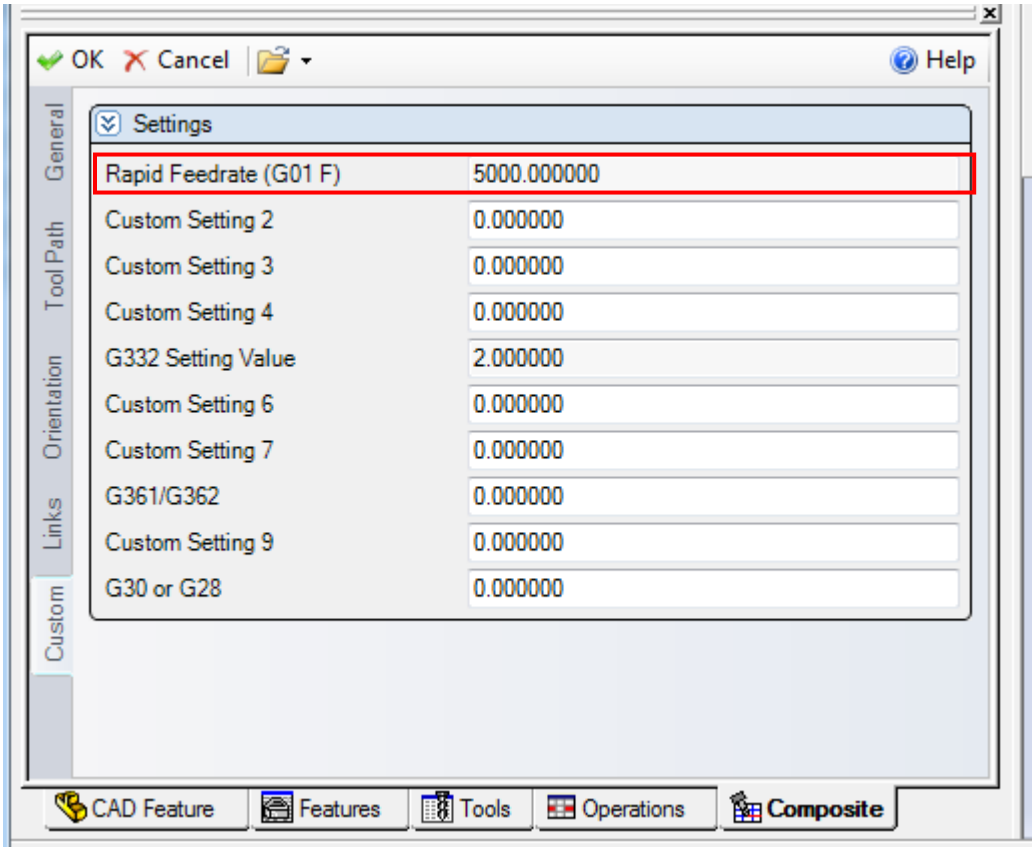


6.4. Feedrate option for rapid moves in 5 axis operations

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

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

If this value is left 0 G00 will be output for the rapid moves.
This setting is only active in the operation programed. If you want to use the G1 F option for multiple operations you will need to set this in each operation.



6.5. Turning on AI Contour Control

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

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

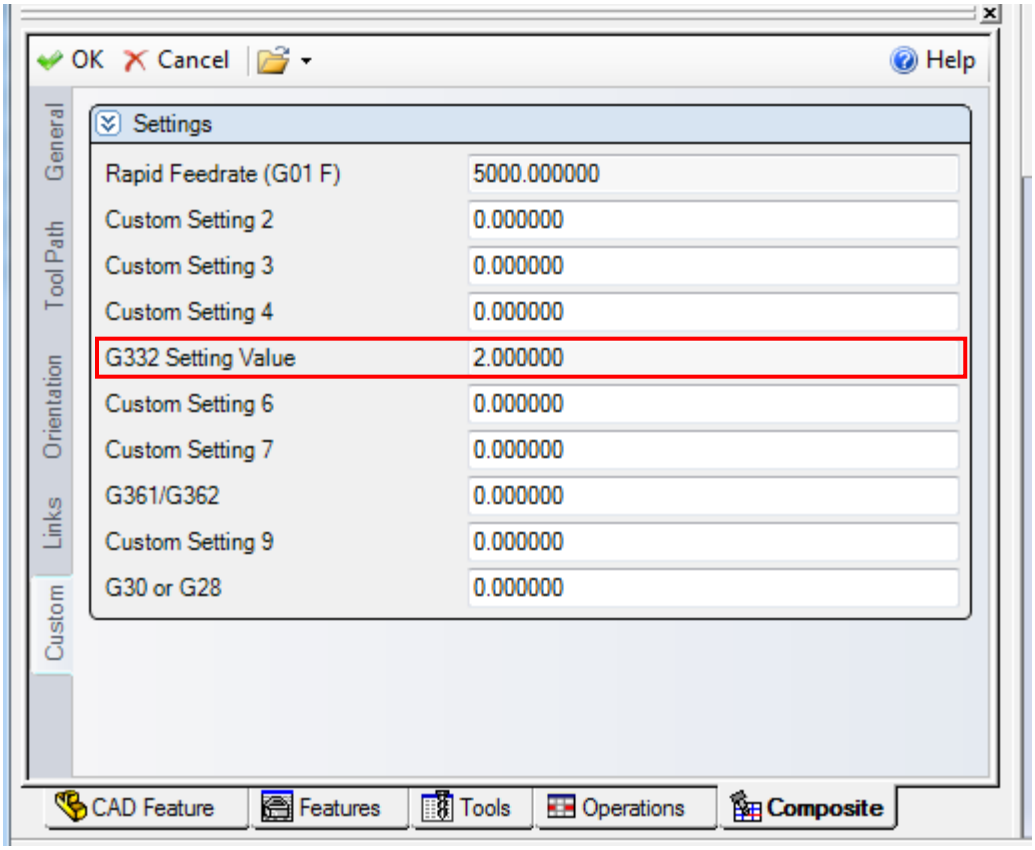
- R1: Time Priority Mode
- R2: Middle Mode
- R3: Accuracy Priority Mode
- R4: Custom Mode

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

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

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

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



7. 5-axis Simultaneous Milling Options

7.1. Introduction

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

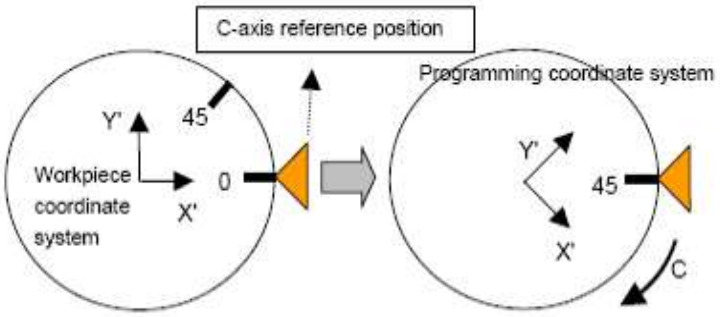
- Fixed Type1 Mode (G43.4, XYZ BC WKP=1)
- Rotate with Part Type1 Mode (G43.4, XYZ BC WKP=0)
- Rotate with Part Type2 Mode (G43.5, XYZ IJK WKP=0)

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

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

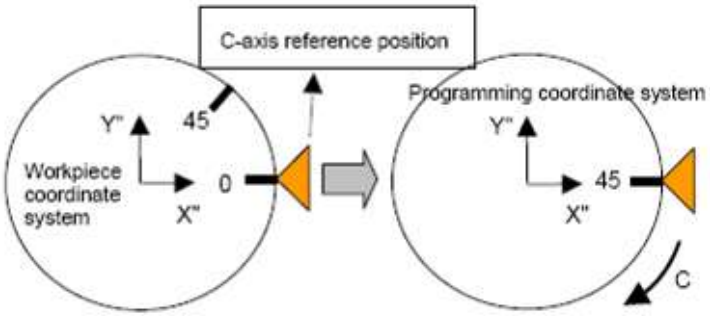
7.2. Definitions

7.2.1. Rotate With Part



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

7.2.2. Fixed

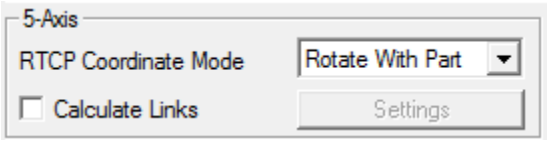


- Coordinates are output relative to the current work piece origin (local work coordinate in ESPRIT). In this case, the programming coordinate system does NOT rotate as the table turns around but remains fixed on the workpiece coordinate system.
- This mode changes the coordinates output by the postprocessor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

7.3. Operation Types

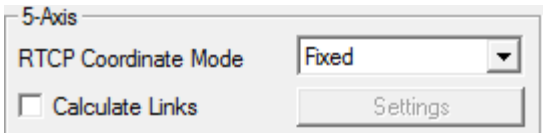
7.3.1. RTCP Type 1 WKP=0 (G43.4)

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



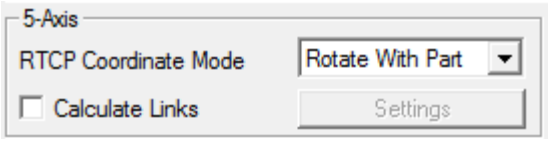
7.3.2. RTCP Type 1 WKP=1 (G43.4)

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



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

TCP Type must be set to “TCP Type 2” in the Post Output Configuration Utility as well as ‘RTCP Coordinate Mode’ must be set to Rotate With Part on the Machine Setup\General Tab
.



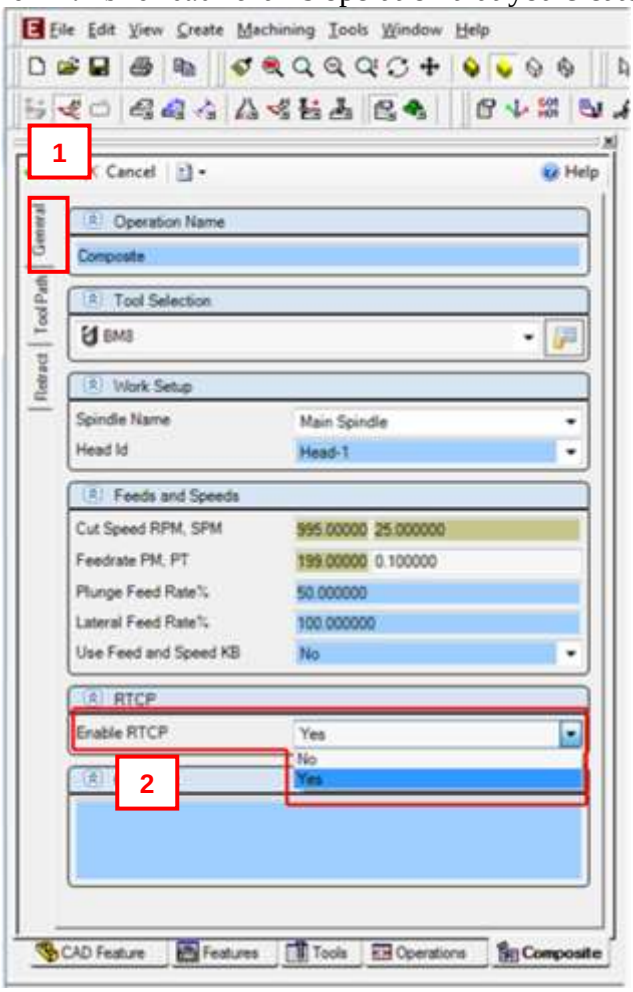
7.4. Additional Information

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

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

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

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

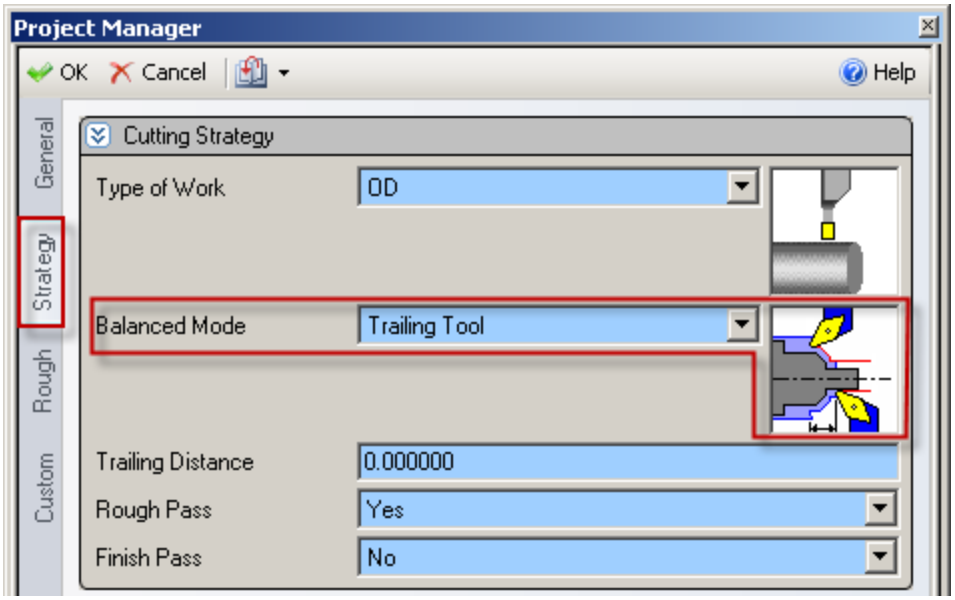


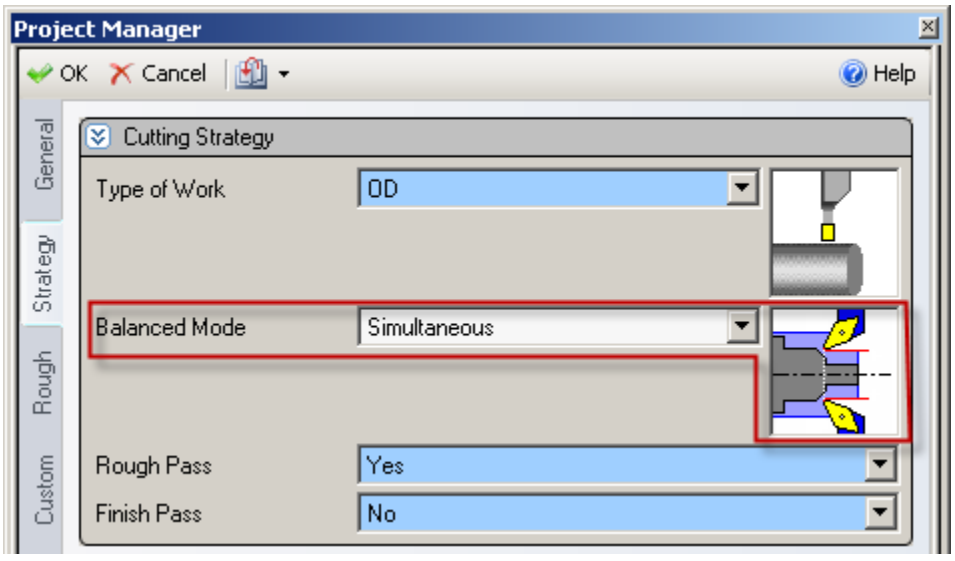
8. Simultaneous Turning Operations: Turning on the same spindle with two turrets

8.1. Balance Roughing Cycle

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

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





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

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

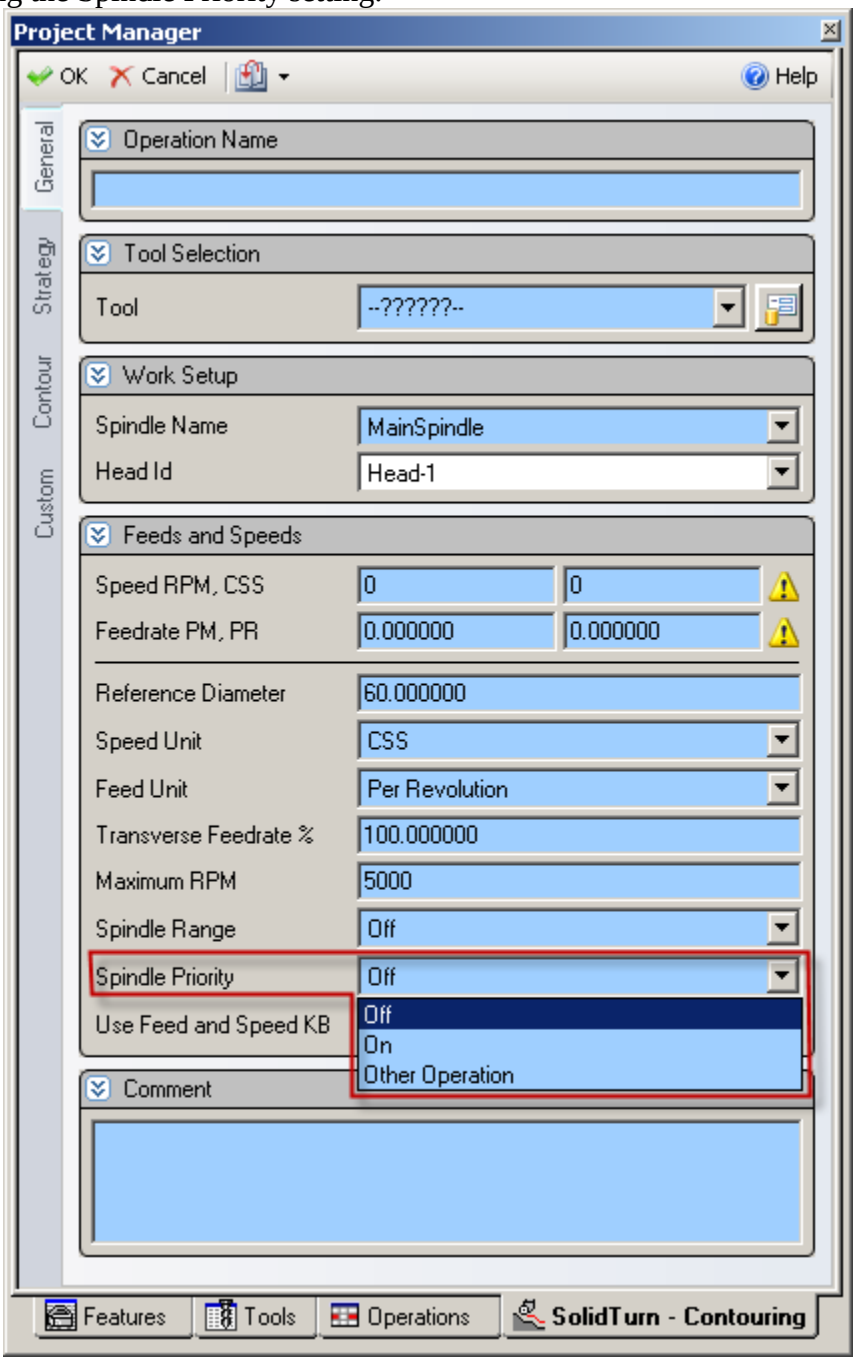
The spindle speed will only be output in Head 1 (B-axis), not in Head 2.

8.2. Other Turning Cycles

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

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

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



Set Spindle Priority to:

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

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

9. Park Operation

9.1. Introduction

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

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

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

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

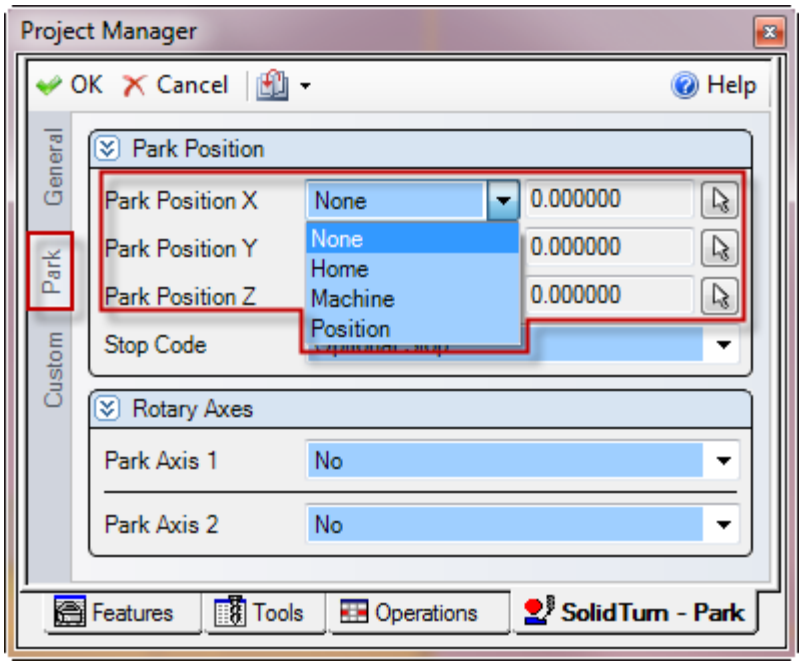
When using the Park operation with the upper head, the Position type park modes are not supported as the simulation will not match the machine movements. The Home and Machine modes are fully supported.

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

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

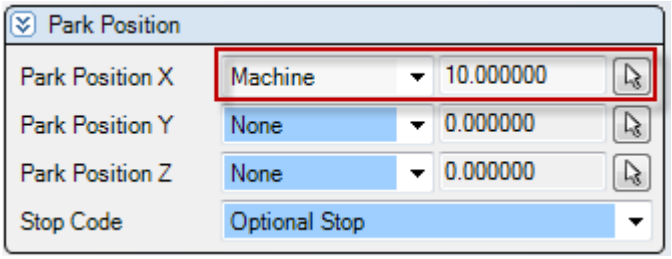
9.2. How to park a tool

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

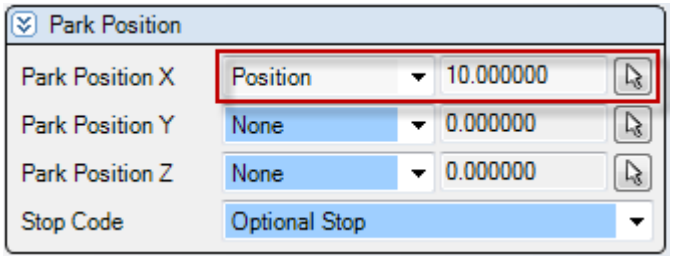


Set Park Position X, Y or Z to:

- **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 Second home position section for additional information.
- **Machine:** The tool will move along the selected axis in the machine work coordinate (G53) to the specified position in Position X, Y, Z fields.

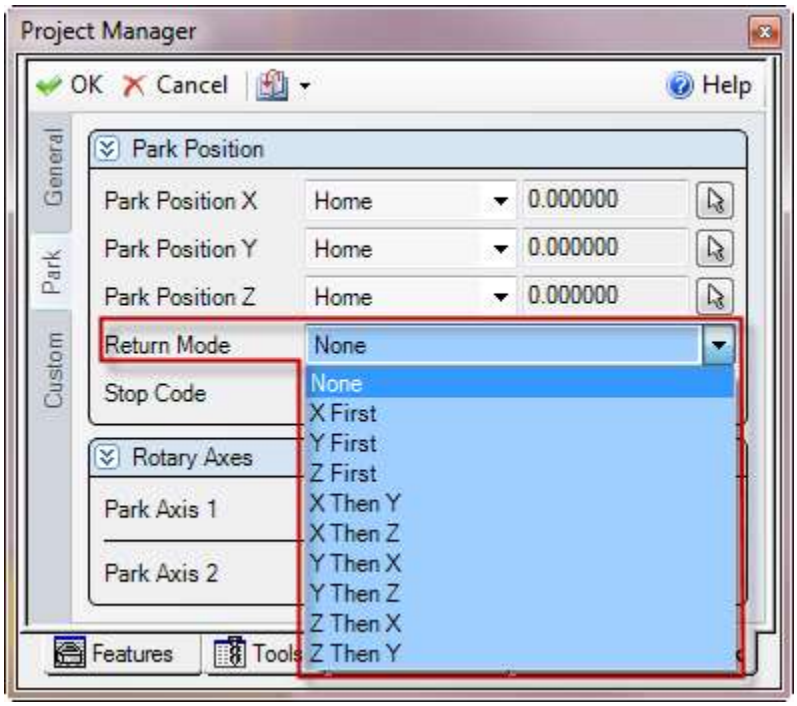


- **Position:** The tool will be parked at the specified position in Position X, Y, Z fields at rapid traverse rate (G00). Position mode is only supported by the lower turret.



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

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



The different existing Return Modes are:

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

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

- *Example 1:*

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

- *Example 2:*

If Park Position X is set to Home, Park Position Y set to Home and Park Position Z set to Machine: the choices for Return Mode become Z First, X Then Y, X Then Z, Y Then X, Y Then Z, Z Then X or Z Then Y.
Since movements in machine work coordinate system (G53) cannot be output in the NC code on the same line as movements home (G28), ESPRIT is filtering the forbidden combinations. These combinations will always have Z movement output on its own line in the NC code.

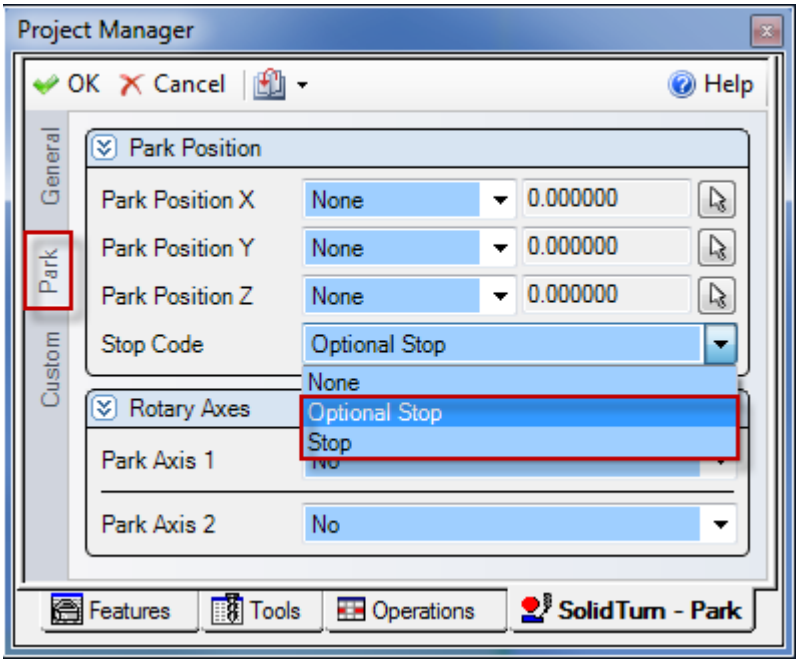
- *Example 3:*

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

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

9.3. Stop Code Options

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



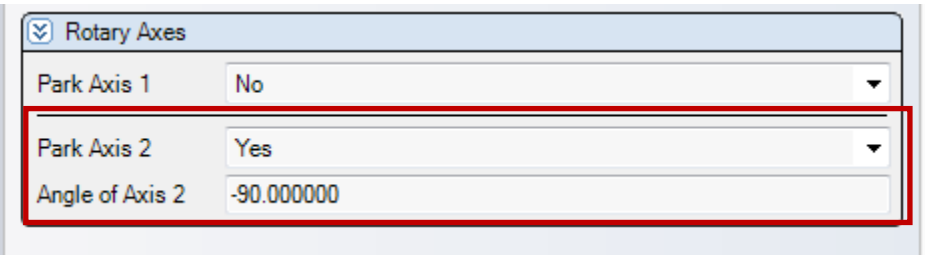
9.4. Setting the B axis angle

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

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

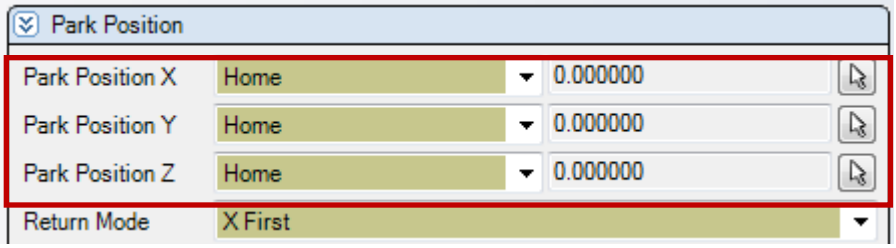
If the park operation does not make a tool change the B axis will only be moved Park Axis 2 field is set to Yes, otherwise the B axis will be left at its current angle.

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



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

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



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

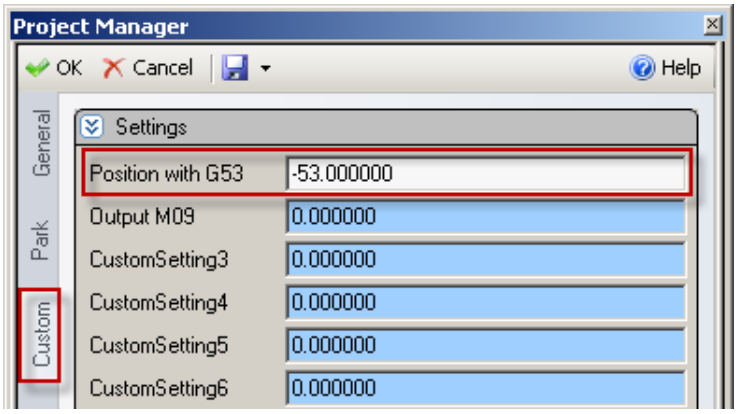
9.5. Park head/ turret to its minimum travel position (G53)

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

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

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

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



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

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

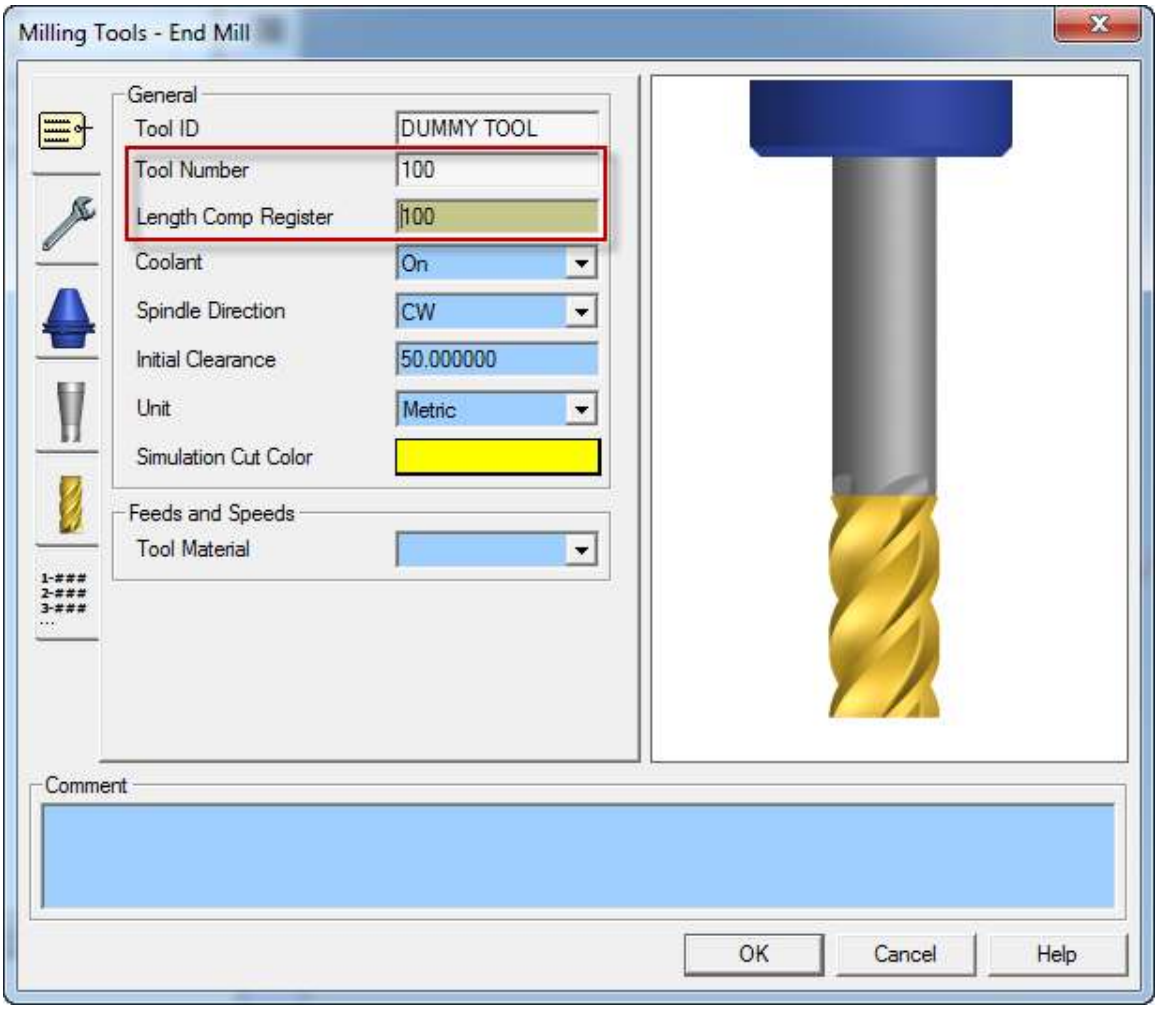
Notes
That if the tool selected in the park operation is not the same as the previous tool the turret will be sent to the tool change location, change the tool, then move to the park location.

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

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

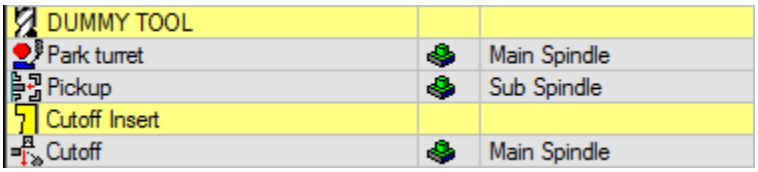
Once correctly setup, you can park the turret using the Park cycle (with G53) prior to a transfer.

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



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

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



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

9.7. Setting up the park operation with the AutoPark addin

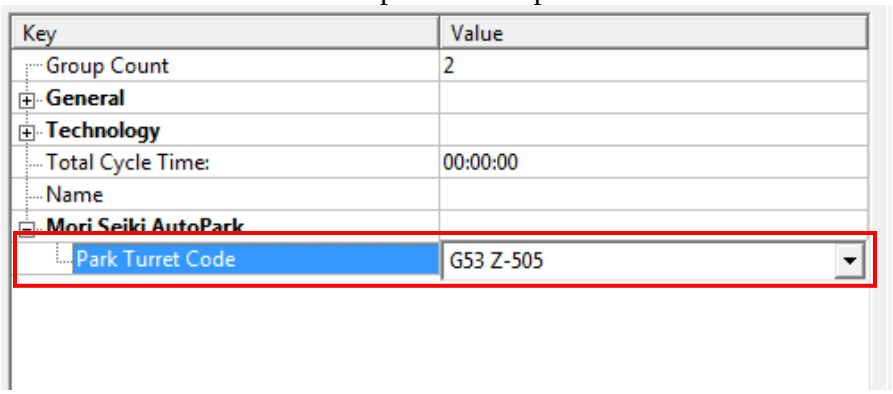
The AutoPark addin is an alternative method of setting up the park operation to move the Head\Turret to the minimum travel limits. The advantages of the addin is that it automatically setups up your park locations of the head/turret for both correct NC code and simulation.

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

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

To use the AutoPark addin perform the following steps:

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



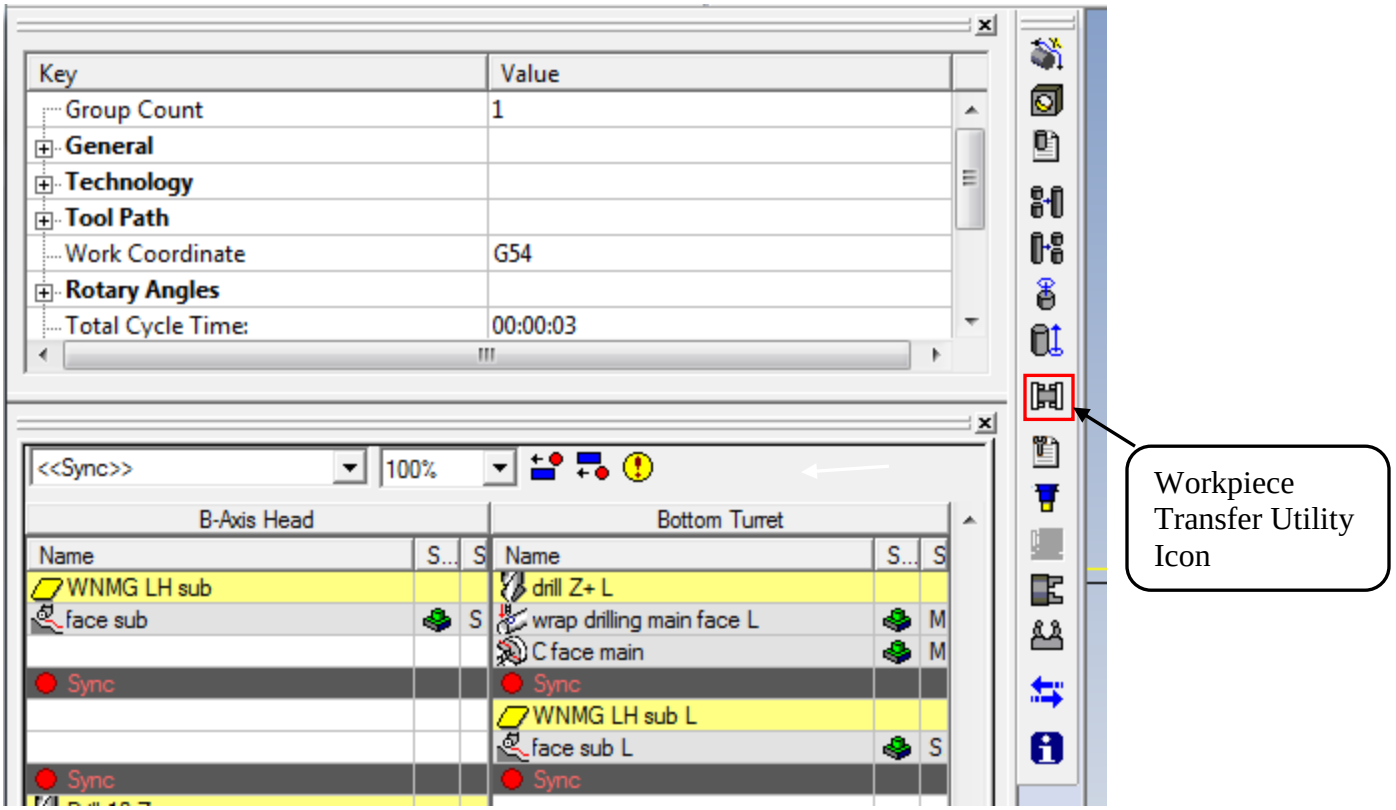
10. How to Program a Part Transfer

10.1. Introduction

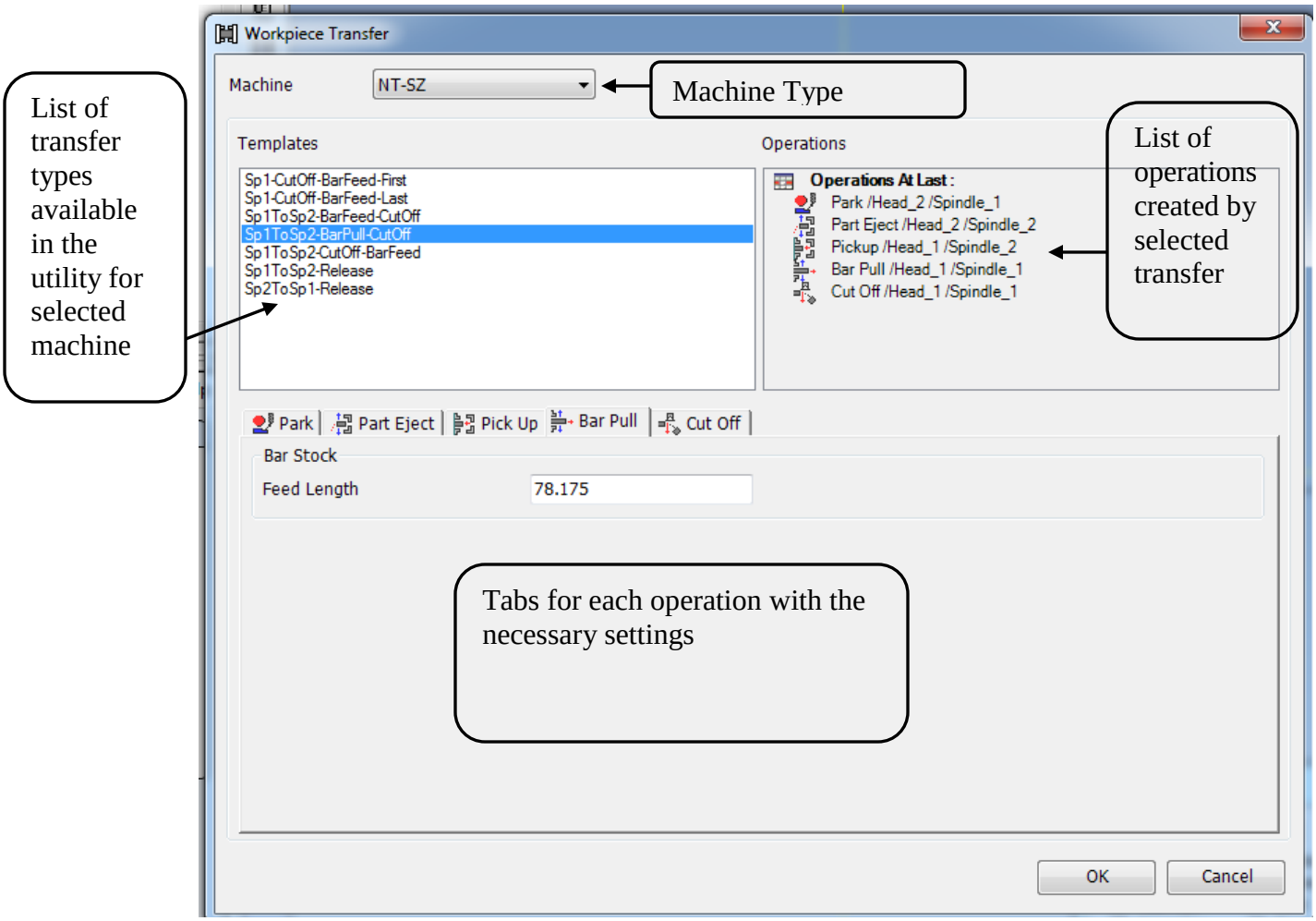
The part transfers can be programed using the Mori Seiki Workpiece Transfer utility. This utility streamlines the creation of the multiple operations used a typical transfers on the NT machines.

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

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



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



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

Notes:

- Below are a summary of each of the transfer types supported by the utility.
- In the Operation Tabs in the Workpiece Transfer Utility section is a detailed expiation of the settings in operation tabs.
- The Workpiece Transfer Utility does not cover every situation yet. You can always modify what has been created to suite your specific needs. If you find errors or missing transfers please contact support and let us know so we can update and improve the utility.

- If your machine type is not selected you are not using the correct Machine Setup file. You will need to down load the latest machine setup file and update your ESPRIT file.

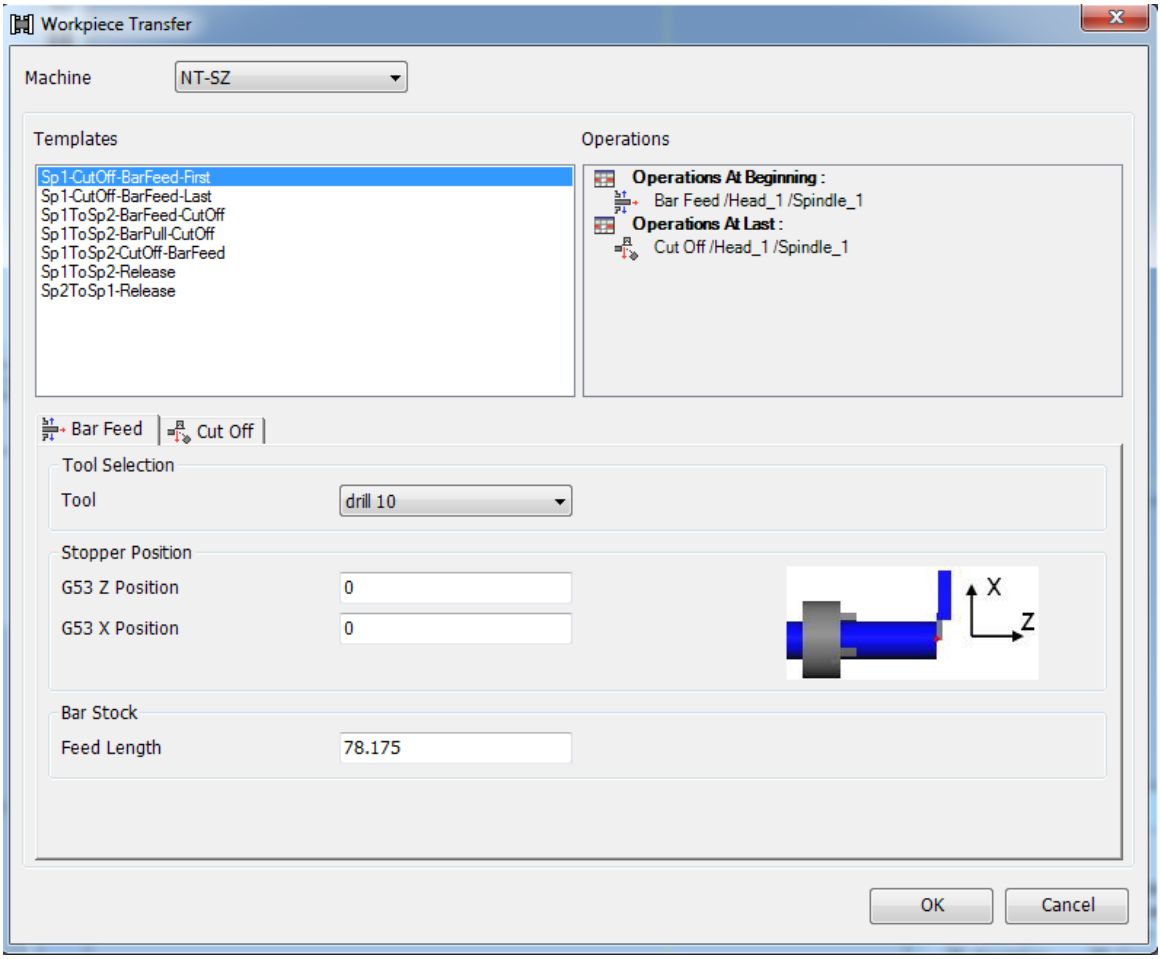
- In addition if you do not see the list of transfers you have an installation problem and you should contact support for help in resolving this matter.

10.2. SP1-CutOff-BarFeed-First

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

The following operations will be created:

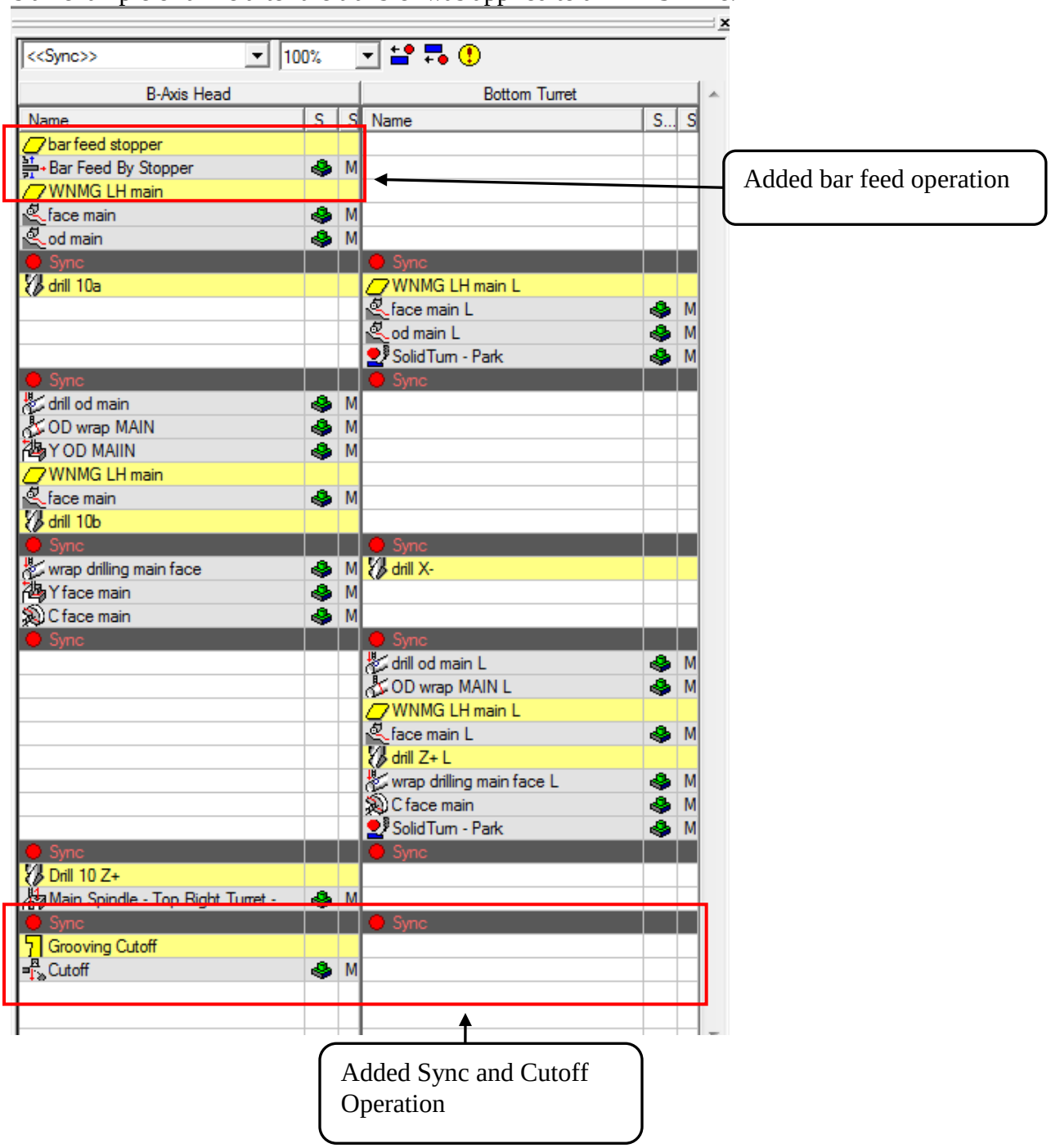
- Bar feed by stopper on upper head at the top of the operation list
- Cutoff Operation. On the upper head.



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

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

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



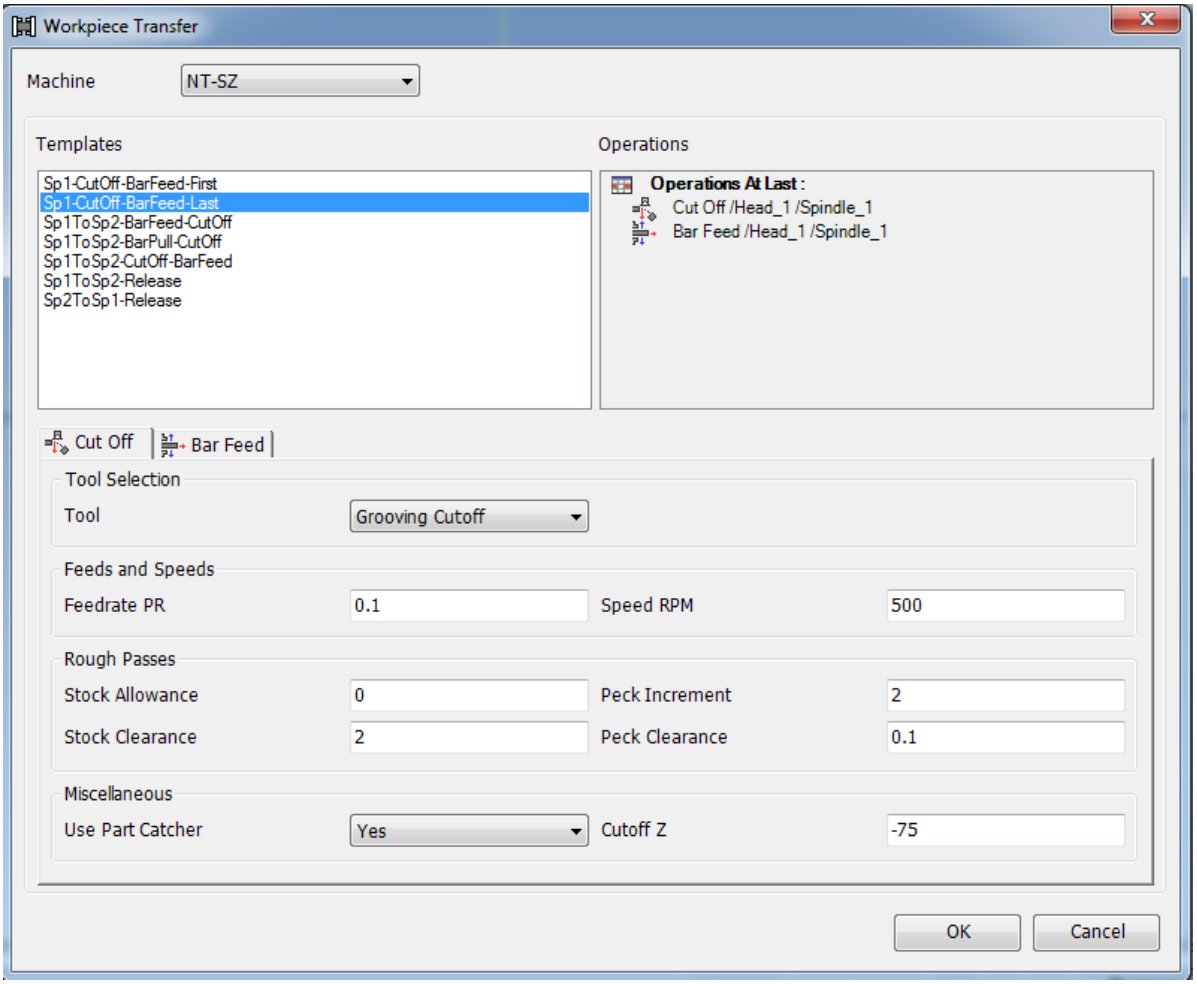
Note: This same option exists on all of the NT machines, non-lower turret machines will not have the sync added, otherwise they are the same.

10.3. SP1-CutOff-BarFeed-Last

SP1-CutOff-BarFeed-last is very similar to the SP1-CutOff-BarFeed-First shown above.

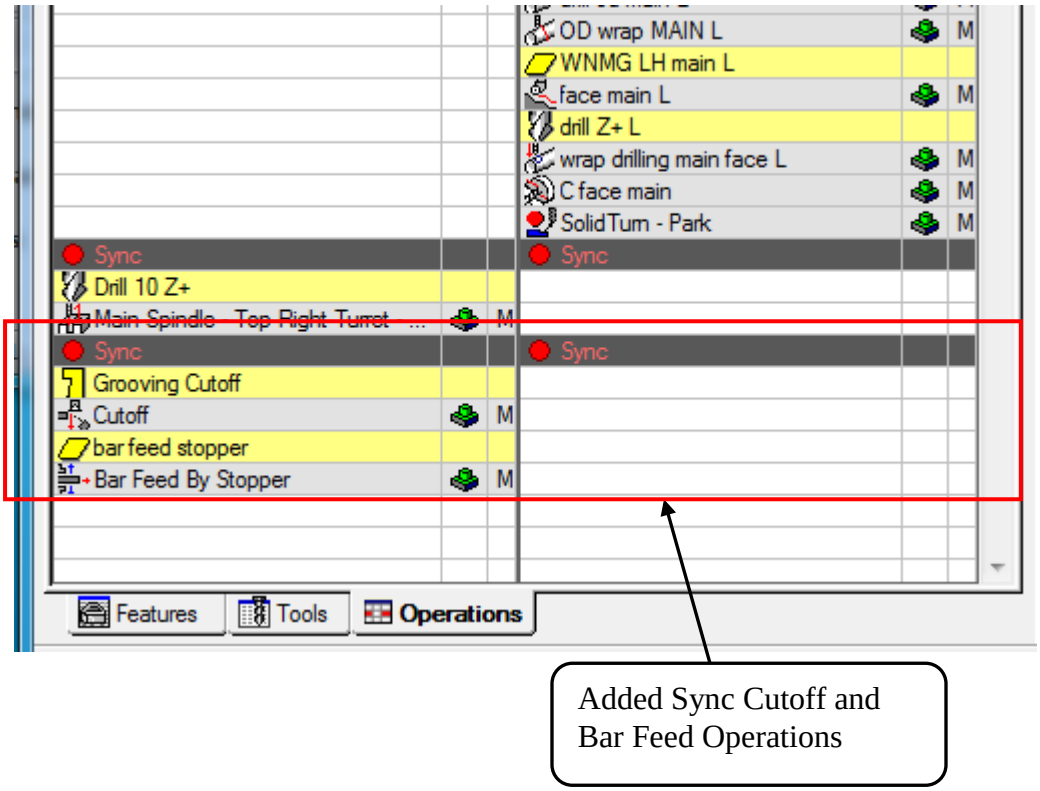
The following operations will be created:

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



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

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



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

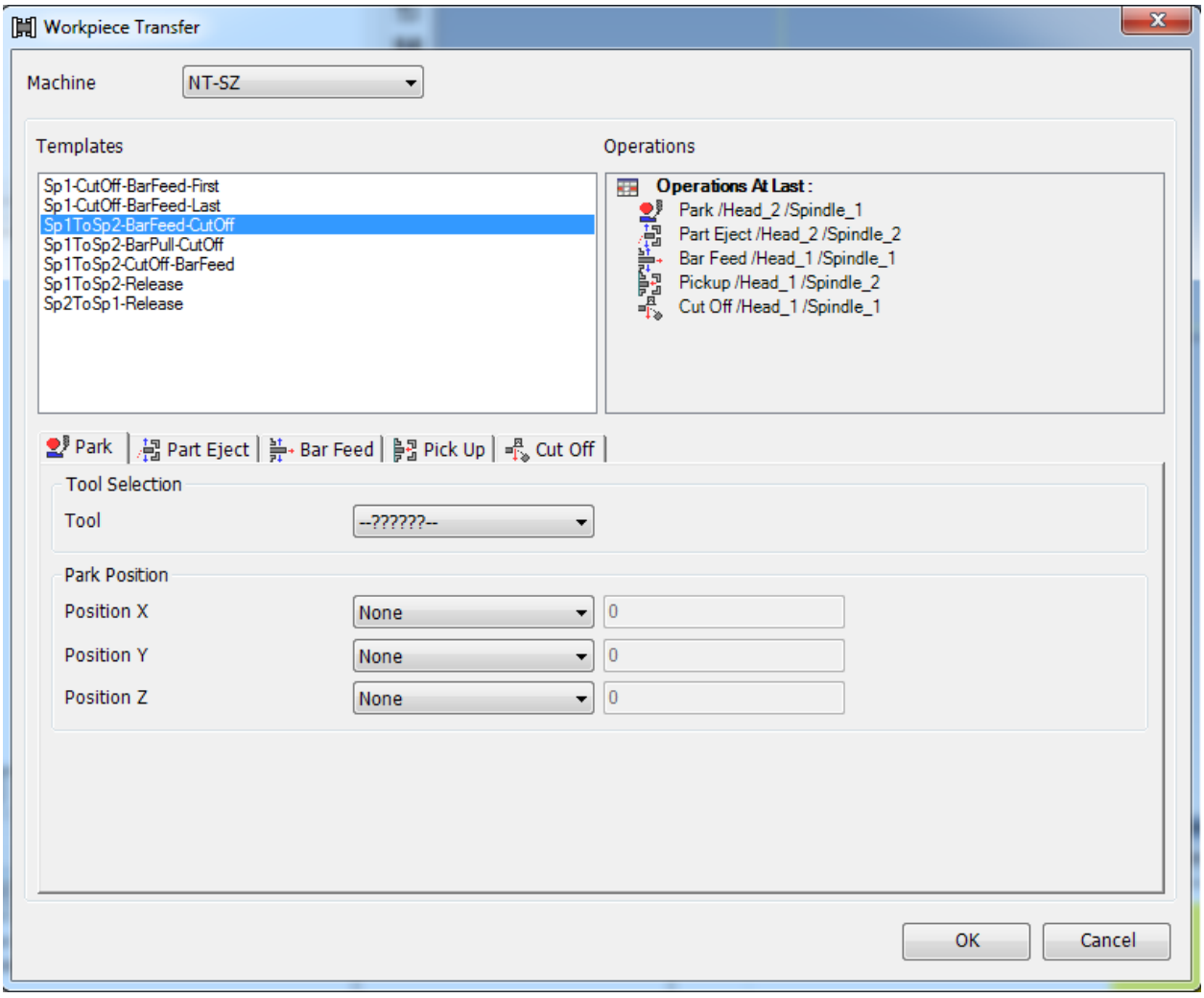
Refer to the Operation Tabs in the Workpiece Transfer Utility section for a detailed expiation of the settings in operation tabs.

10.4. SP1 To SP2- BarFeed-Cutoff

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

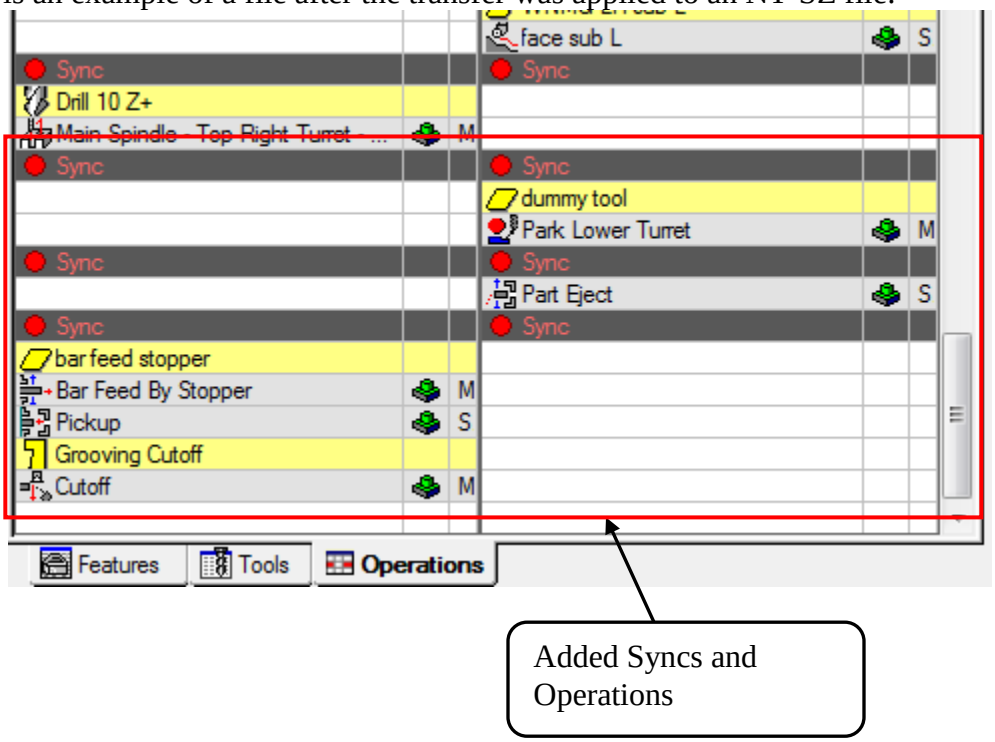
The following operations will be created:

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



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

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



Notes:

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

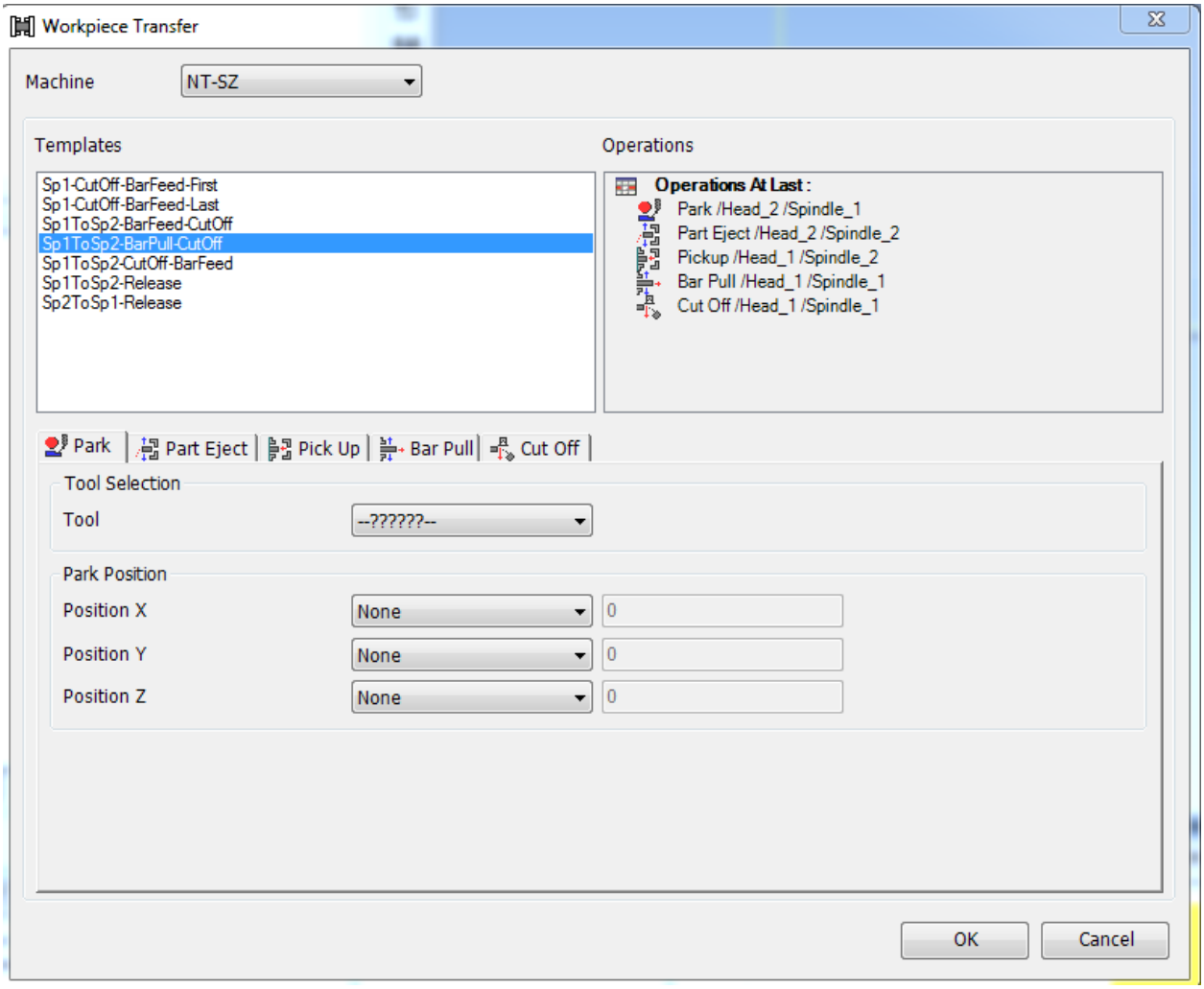
NT000-SZ and NT000-WZ machines will also have the part eject cycle on the upper head

10.5. SP1 To SP2- BarPull-Cutoff

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

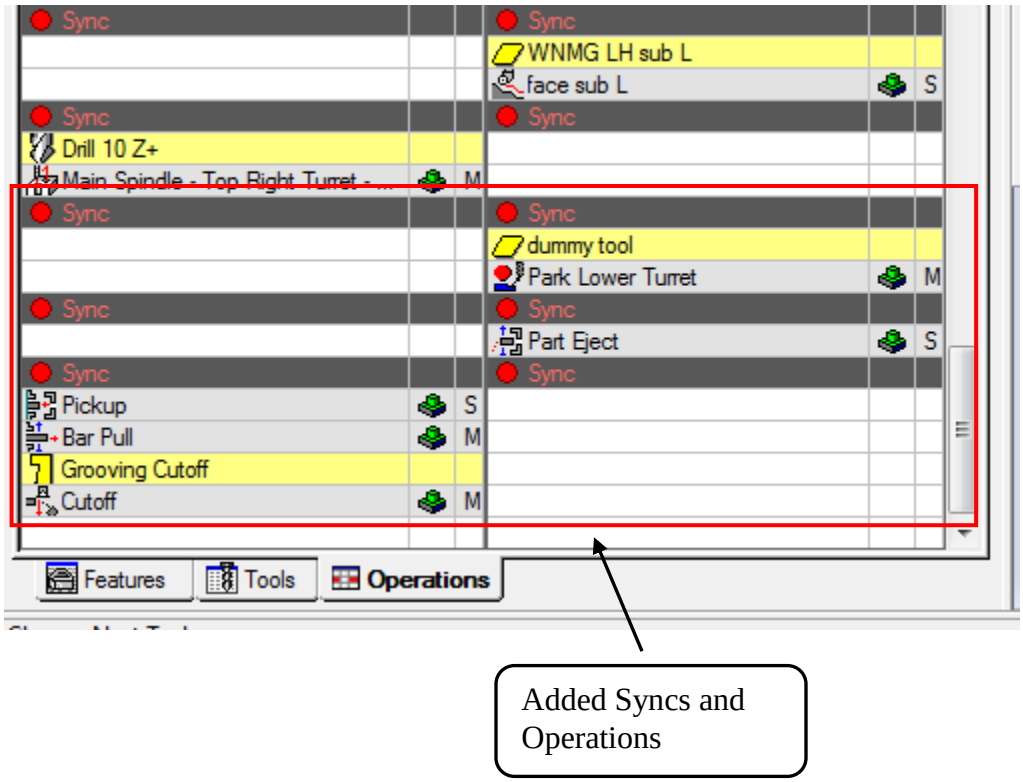
The following operations will be created:

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



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

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



Notes:

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

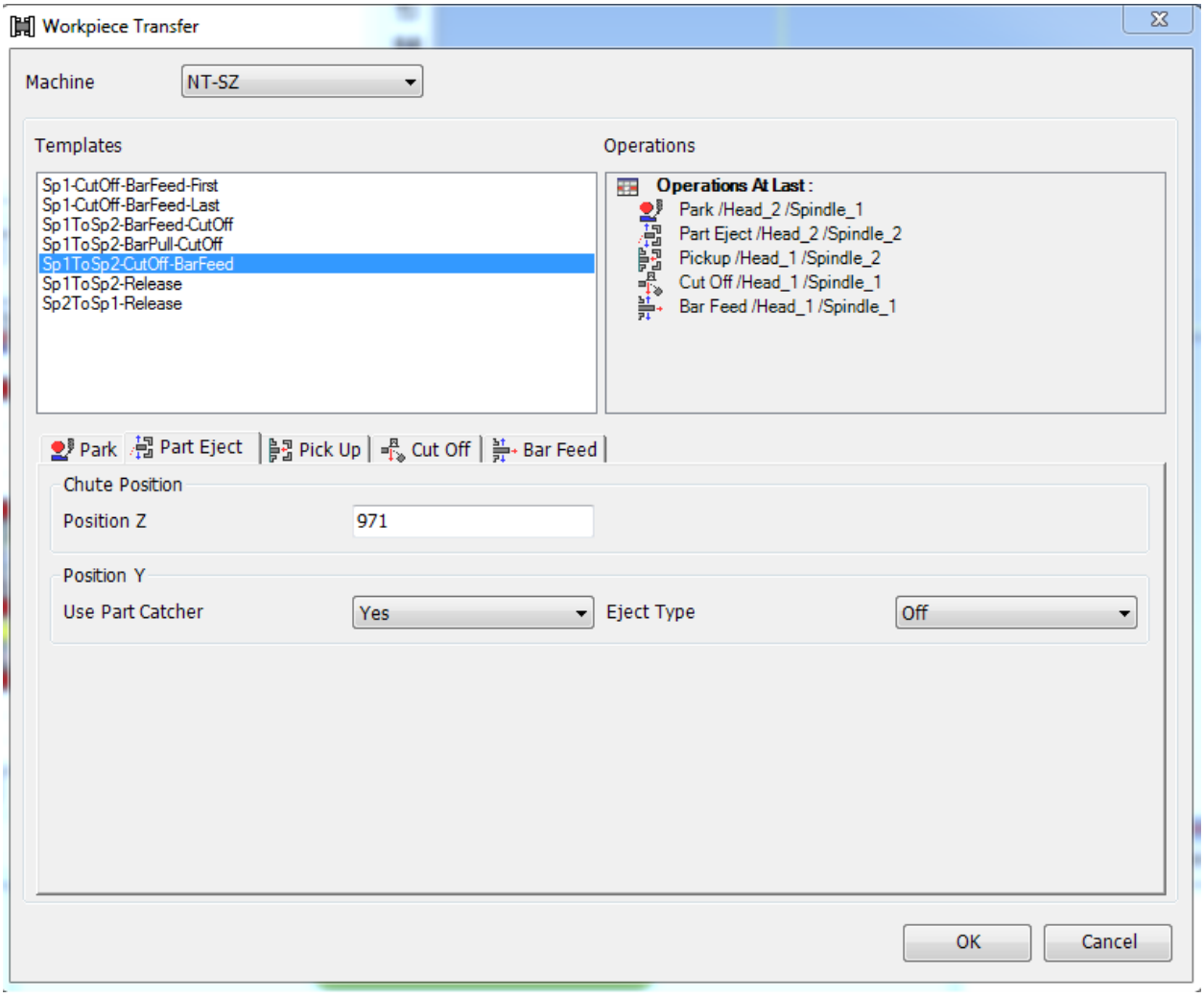
NT000-SZ and NT000-WZ machines will also have the part eject cycle on the upper head

10.6. SP1 To SP2-Cutoff-BarFeed

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

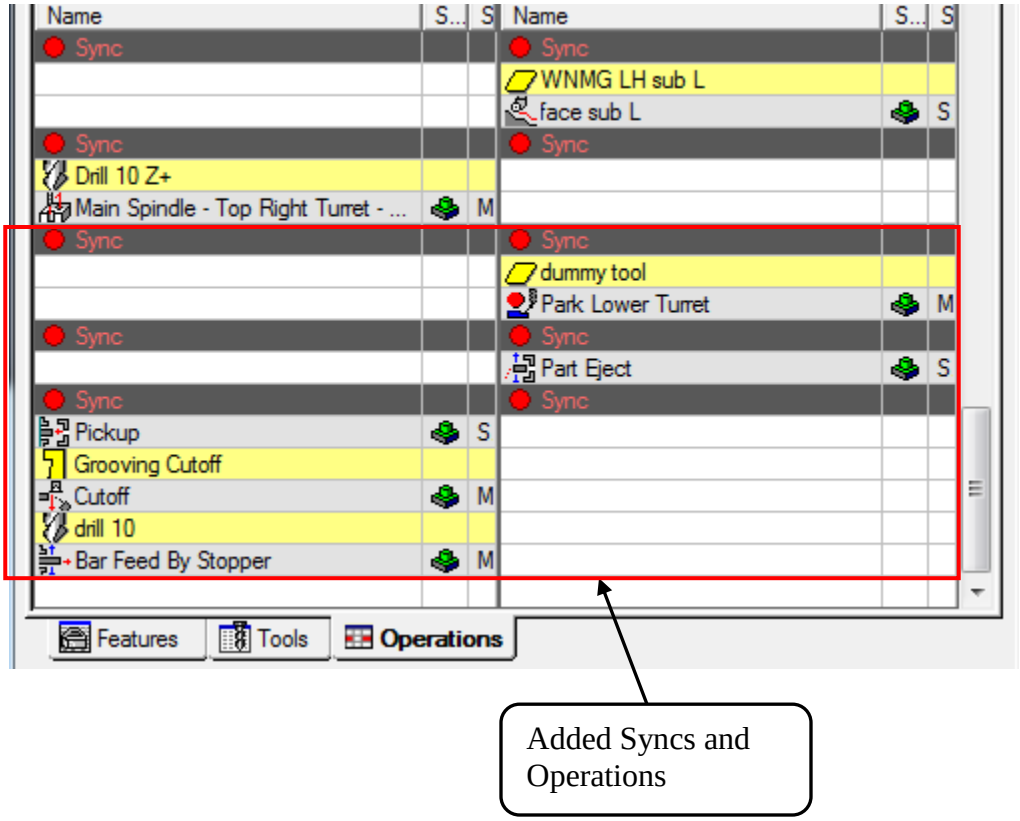
The following operations will be created:

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



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

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



Notes:

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

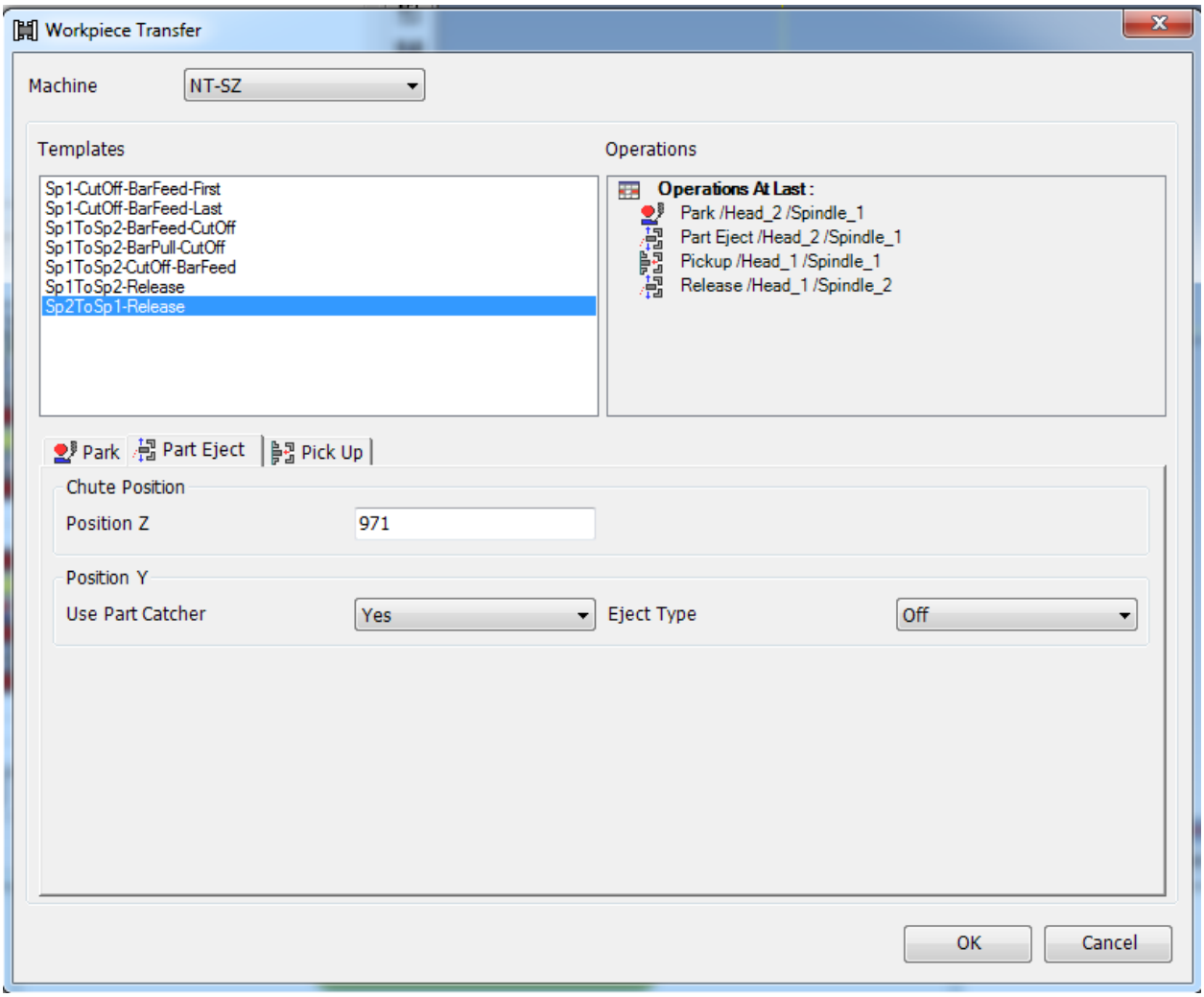
NT000-SZ and NT000-WZ machines will also have the part eject cycle on the upper head

10.7. SP1 To SP2-Release

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

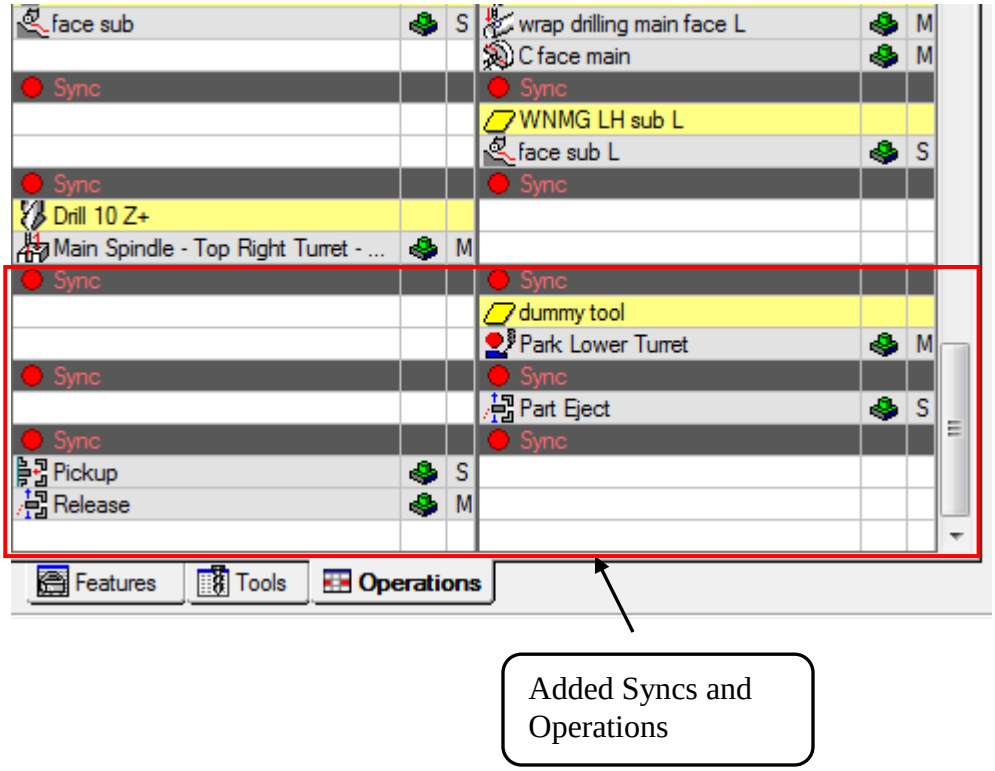
The following operations will be created:

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



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

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



Notes:

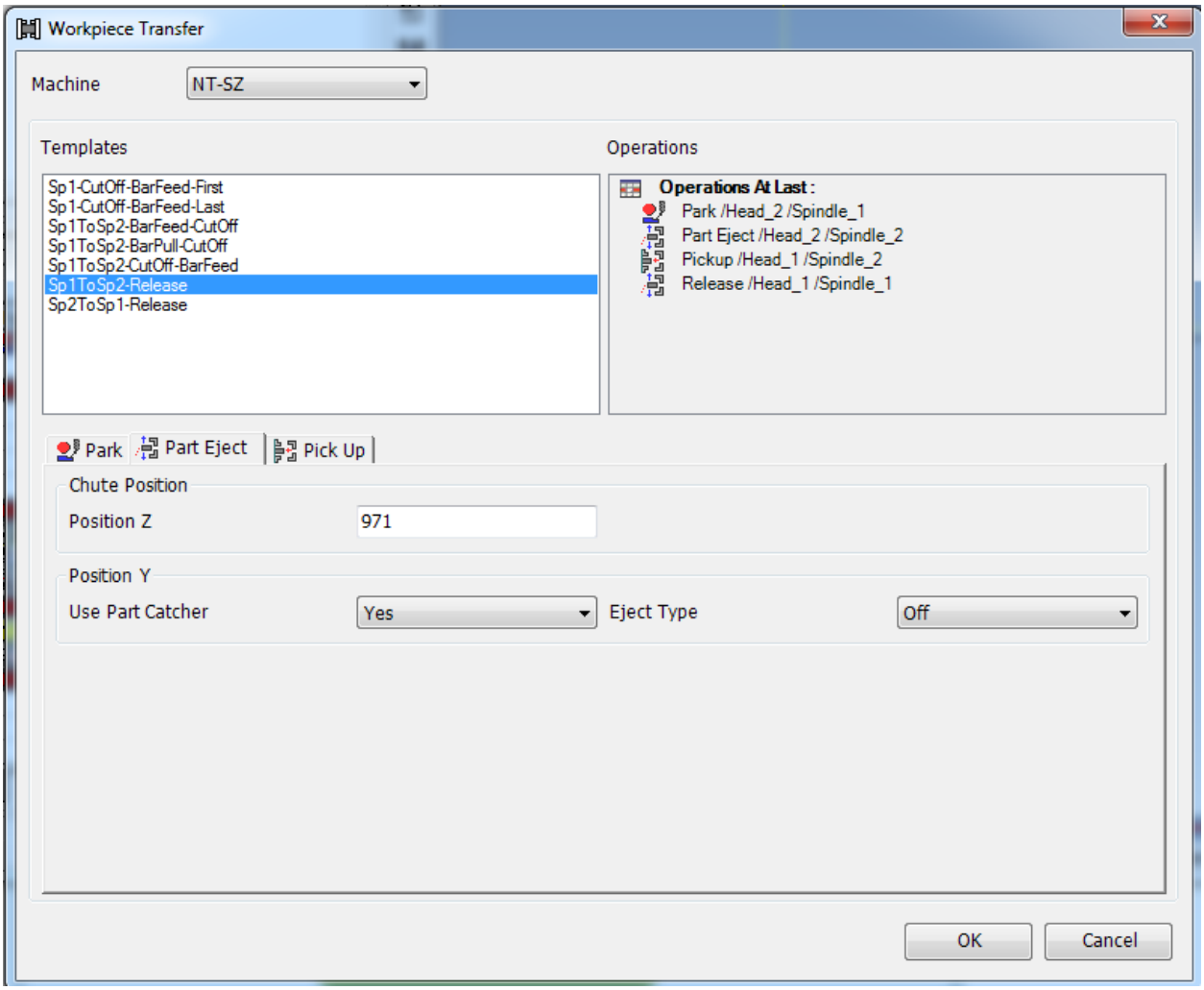
This same option exists on all of the NT machines, non-lower turret machines will not have the syncs or park operation added and the part eject cycle will be on the upper turret, otherwise they are the same.

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

10.8. SP2 To SP1-Release

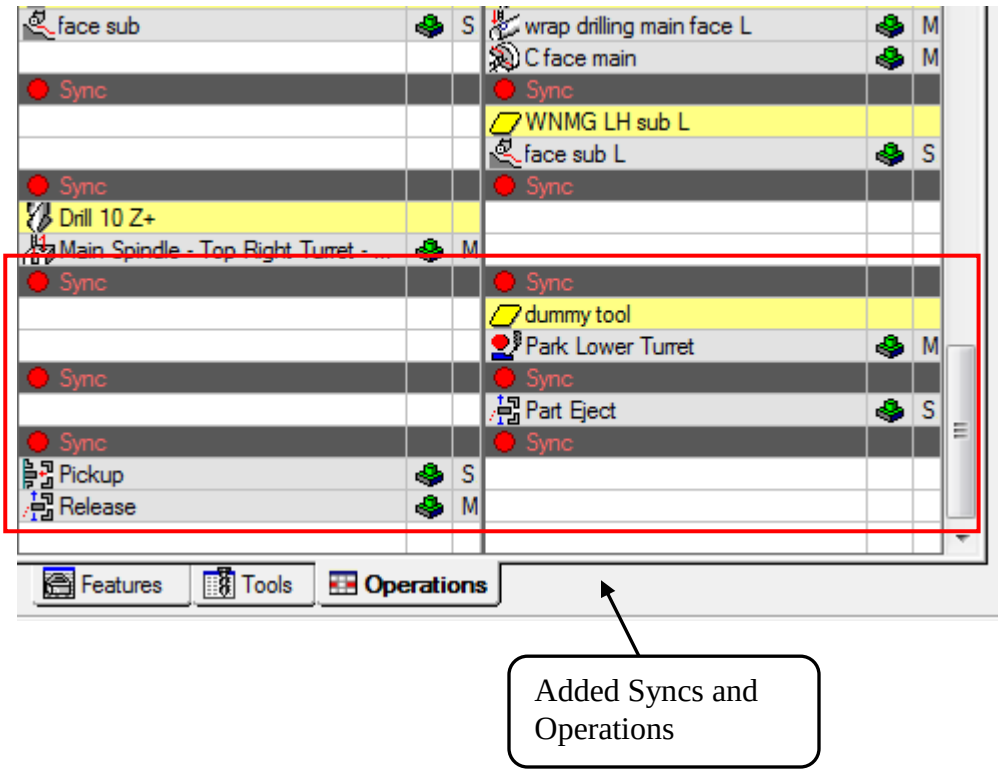
This is a transfer from the main spindle to the sub spindle. The following operations will be created:

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



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

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



Notes:

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

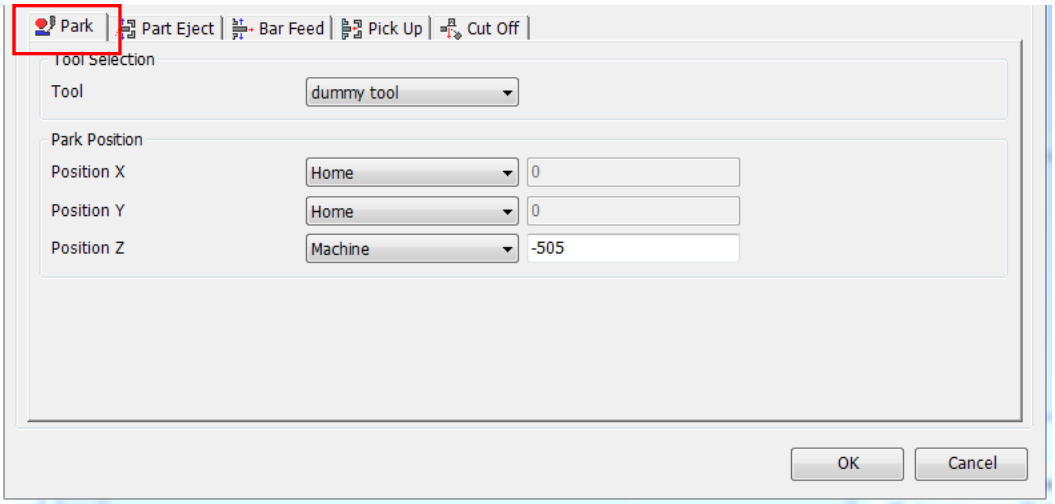
NT000-SZ and NT000-WZ machines will also have the part eject cycle on the upper head

10.9. Operation Tabs in the Workpiece Transfer Utility

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

10.9.1. Park

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

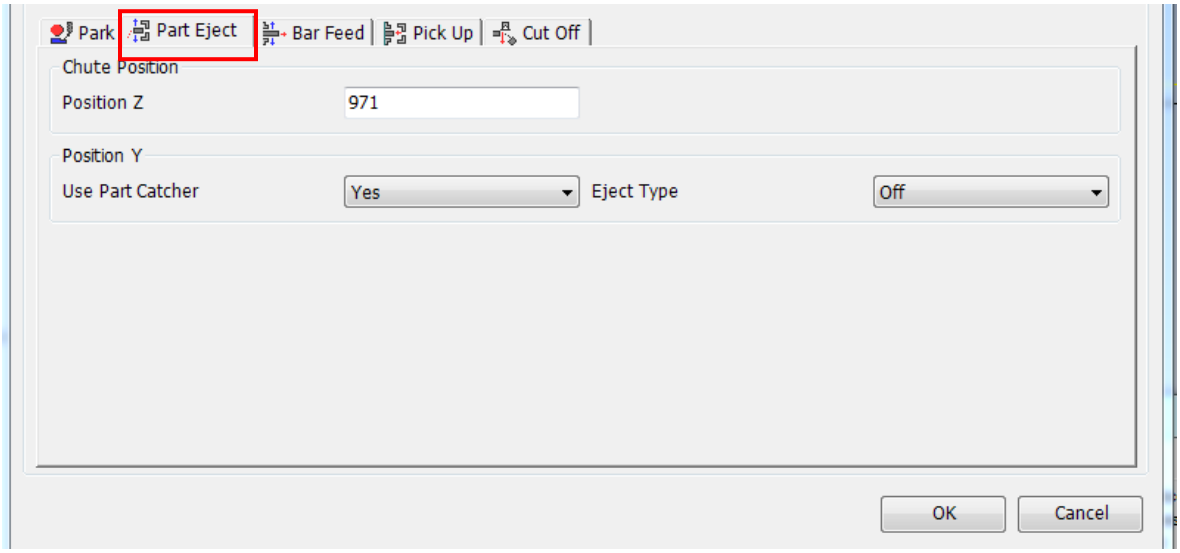


The Park Tab has the following settings:

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

10.9.2. Part Eject

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



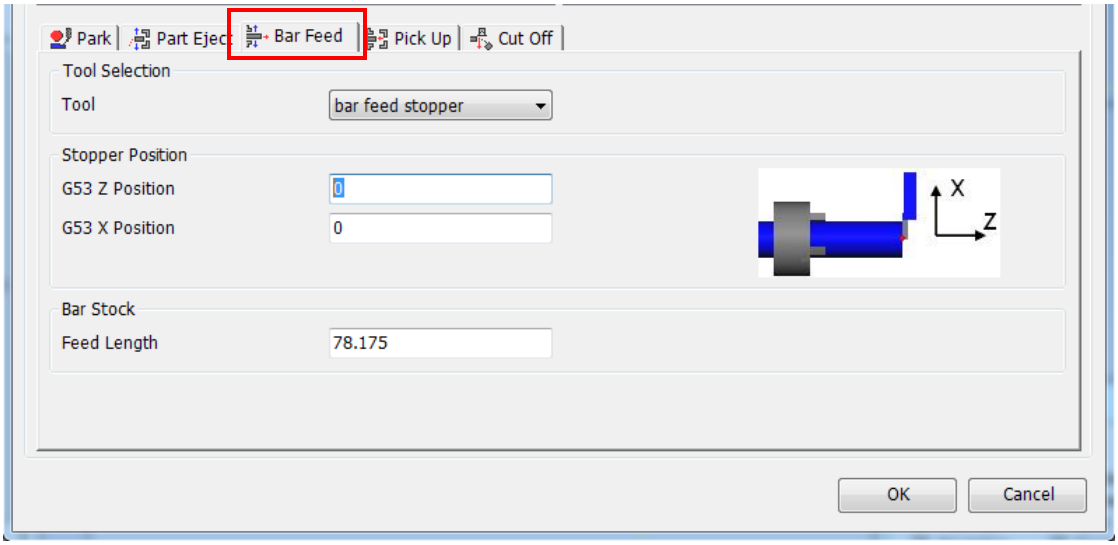
The Eject Tab has the following settings:

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

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

10.9.3. Bar Feed

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



The Bar Feed Tab has the following settings:

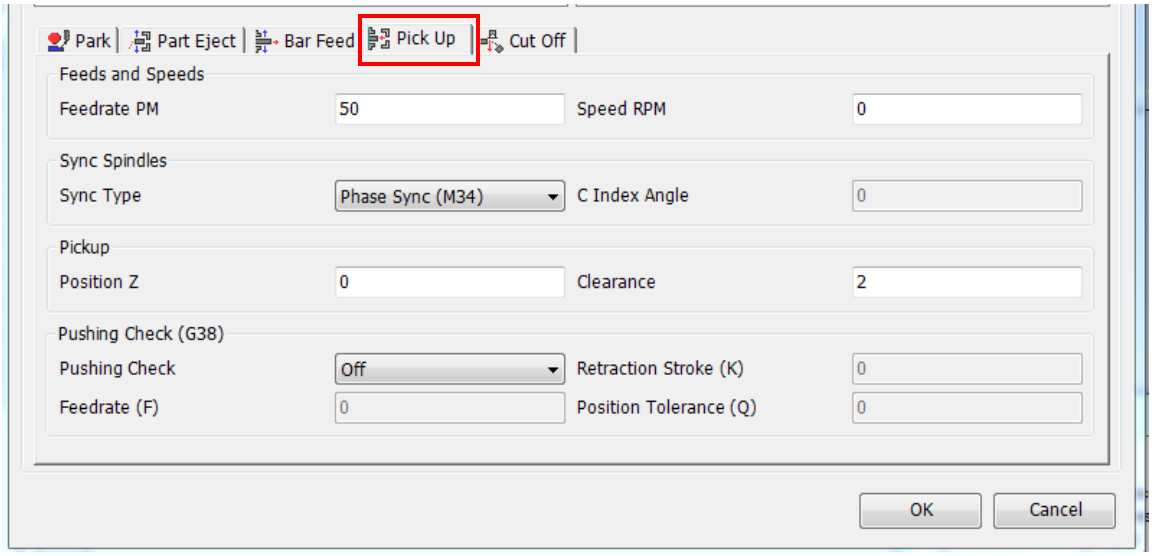
- Tool: This is the tool used for the bar stopper
- Stopper Position G53 Z position: Position in the NC code for the bar stopper location
- Stopper Position G53 X position: Position in the NC code for the bar stopper location
    - Note if the Stopper position is left zero the NC code will move the bar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

Notes:

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

10.9.4. Pick Up

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



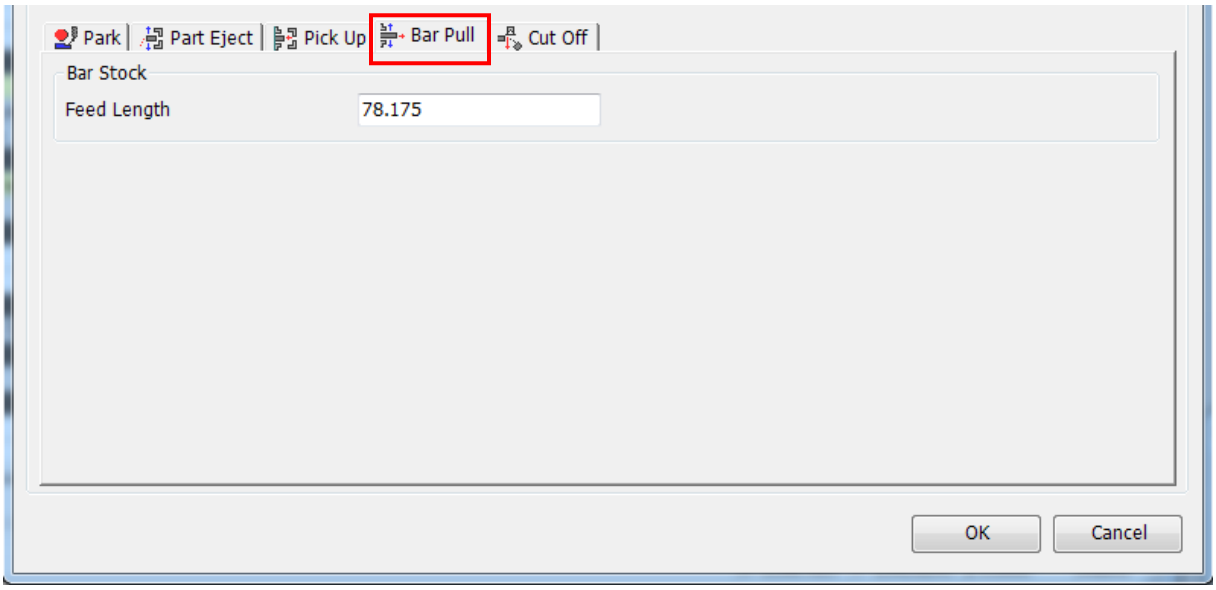
The Pick Up Tab has the following settings:

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

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

10.9.5. Bar Pull

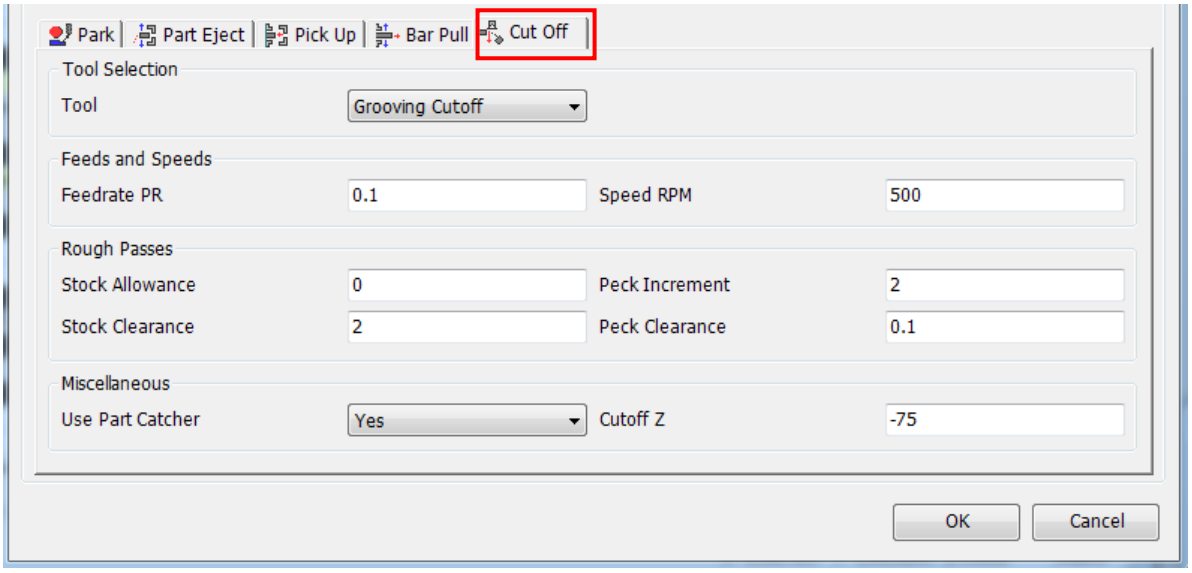
When programming a transfer type with a barpull with the sub spindle, the barpull tab only has one setting. The settings on this page will be used in the creation of a pickup operation.



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

10.9.6. Cutoff

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



The Cutoff Tab has the following settings:

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

10.10. Custom Settings for Part Transfer Cycles

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

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

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

10.10.1. Workpiece Pushing Check (G38)

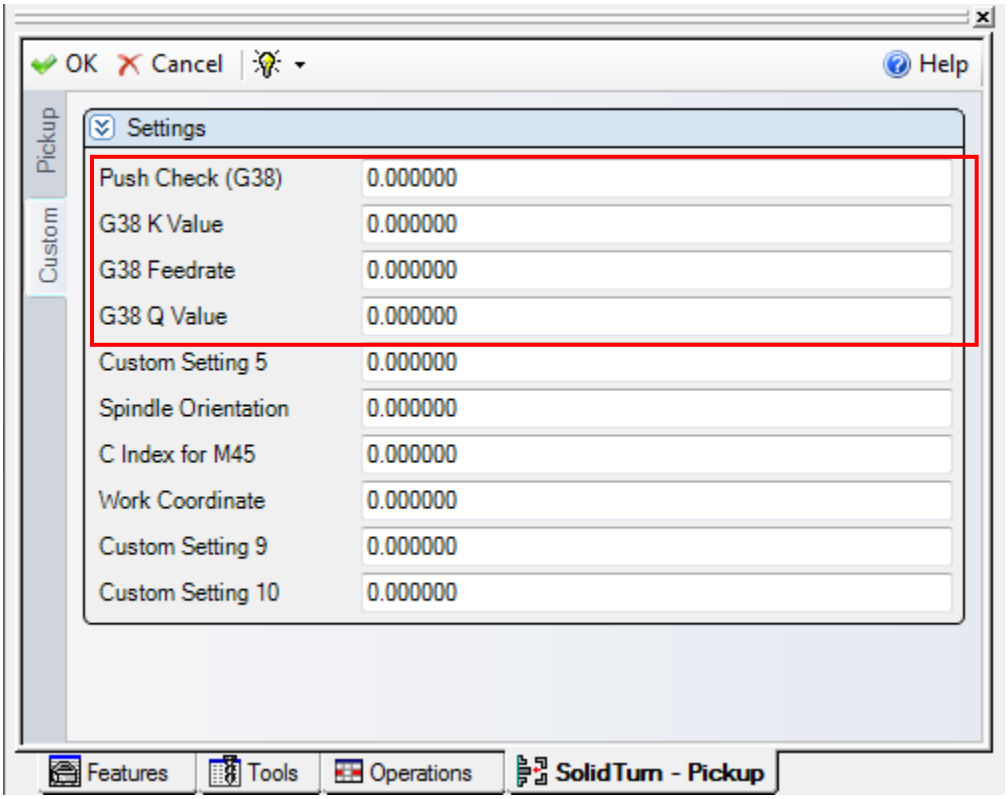
In the workpiece transfer operation, when the position error of the A-axis servomotor of headstock 2 reaches the parameter set value, the workpiece is judged to have been brought into contact with the reference face of chuck 2 or chuck 1. Then, chuck 2 or chuck 1 clamps. In ESPRIT, it is possible to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

G38 A_ K_ F_ Q_

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

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

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

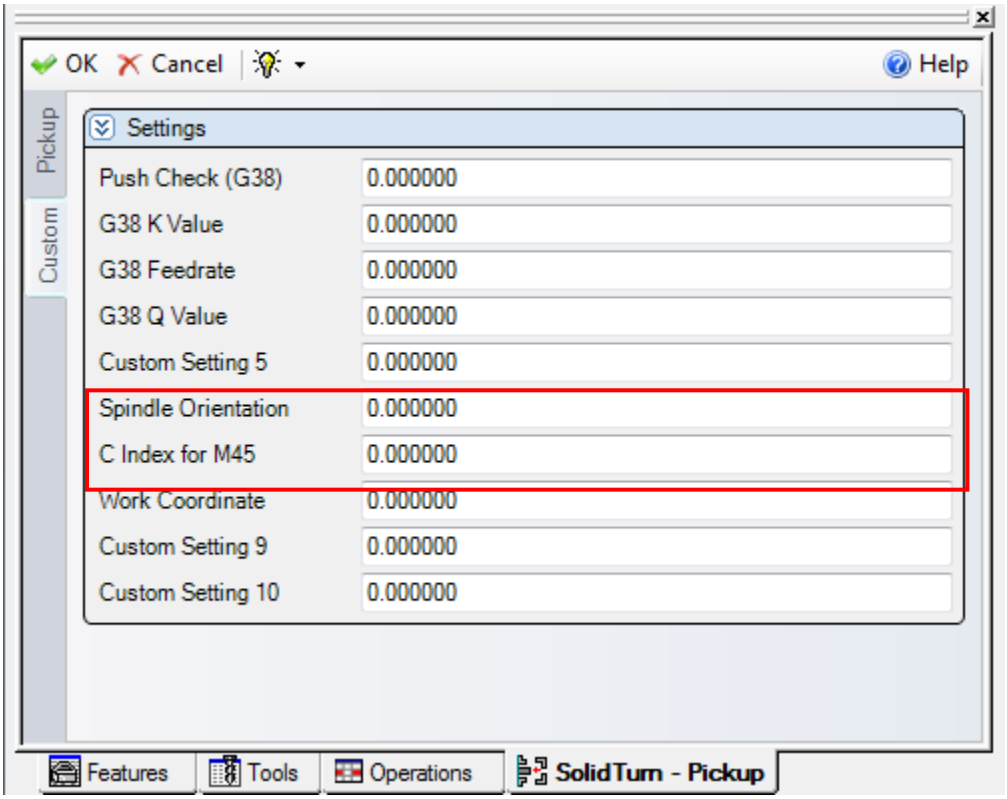


10.10.2. C axis angle for pick operation

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

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

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



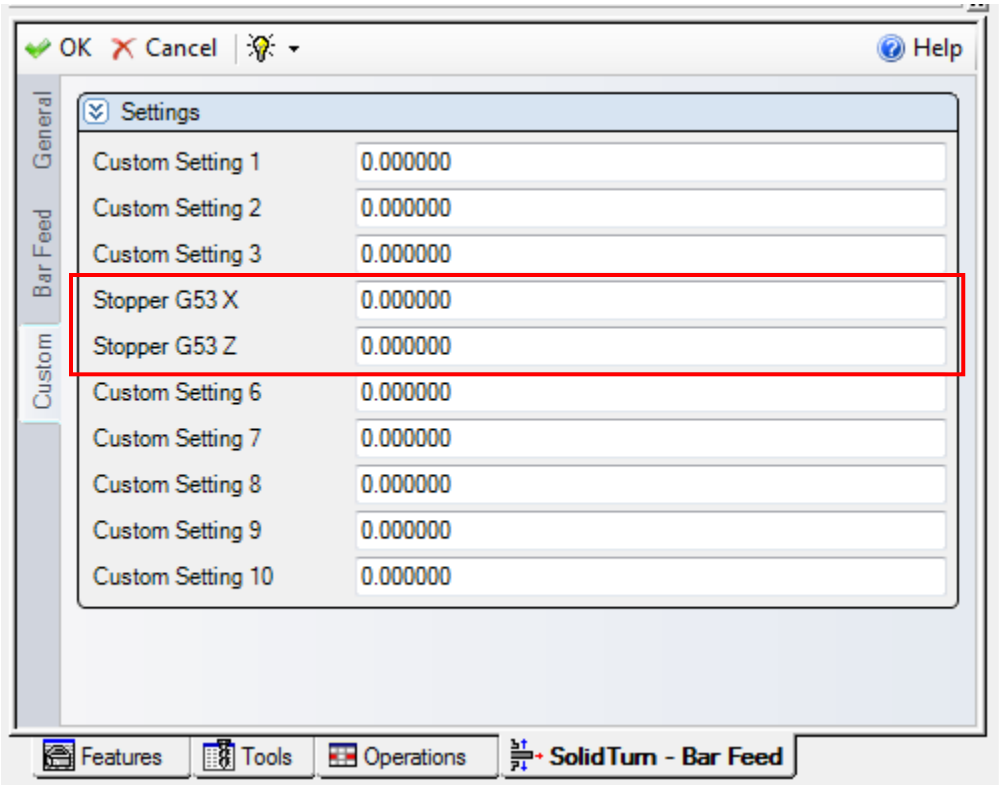
NOTE: The milling C-axis mode is the default. So if the user omits a value in the Spindle Orientation field (leaves it set to zero) or enters a wrong value, the spindles will be indexed using this mode.

10.10.3. Bar Feed positions relative to G53

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

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

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



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

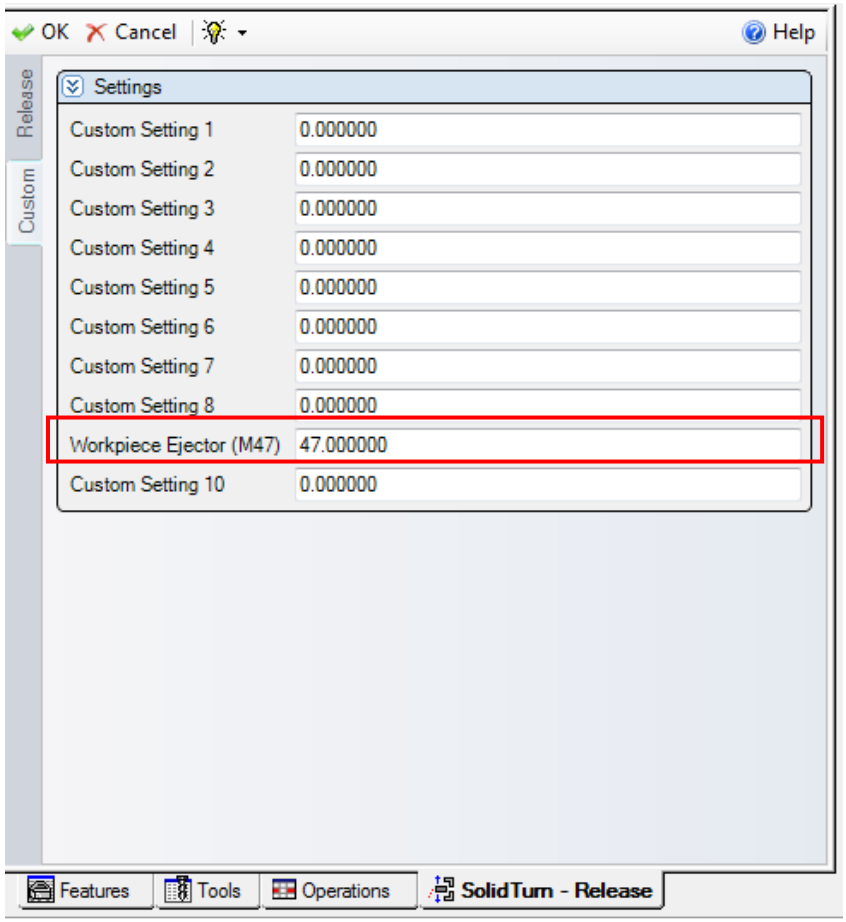
In the case that the workpiece is small or delicate, it is possible to use the Workpiece Ejector to ensure that the workpiece has been removed from the chuck. To initiate this function, navigate to the Custom tab of the Release operation used to eject the part from the sub spindle.

10.10.4.1. NT-S

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

10.10.4.2. NT-SZ

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

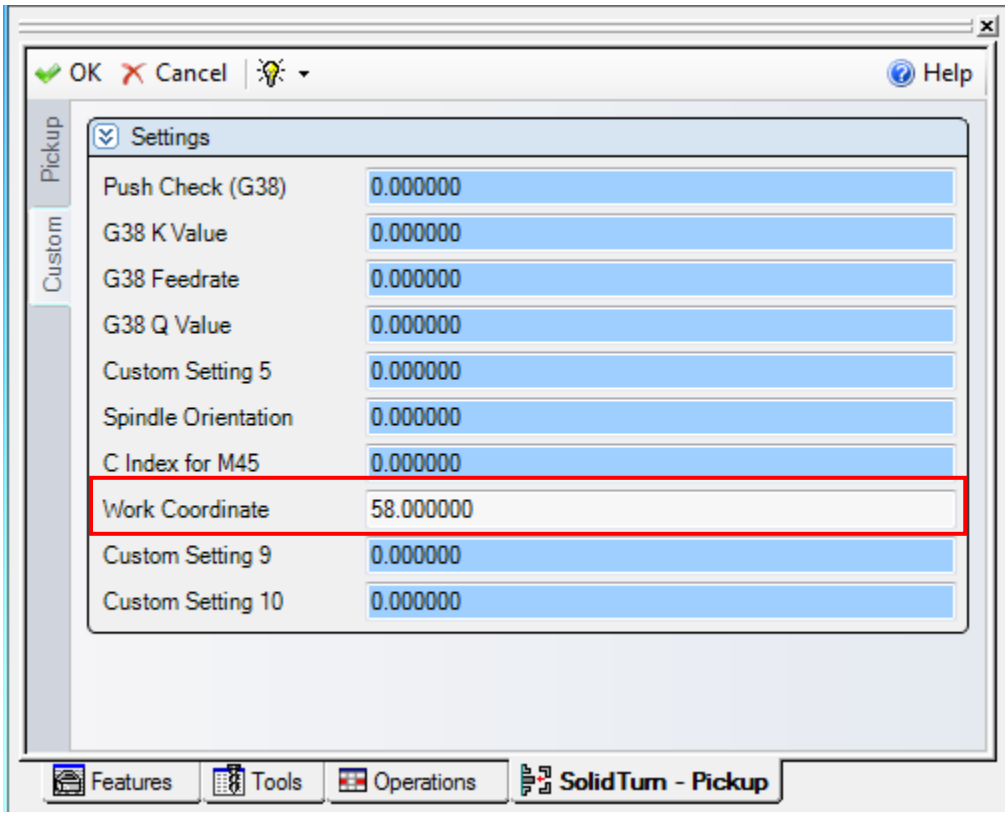


10.10.5. Work Coordinate for Pickup Operation

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

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

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



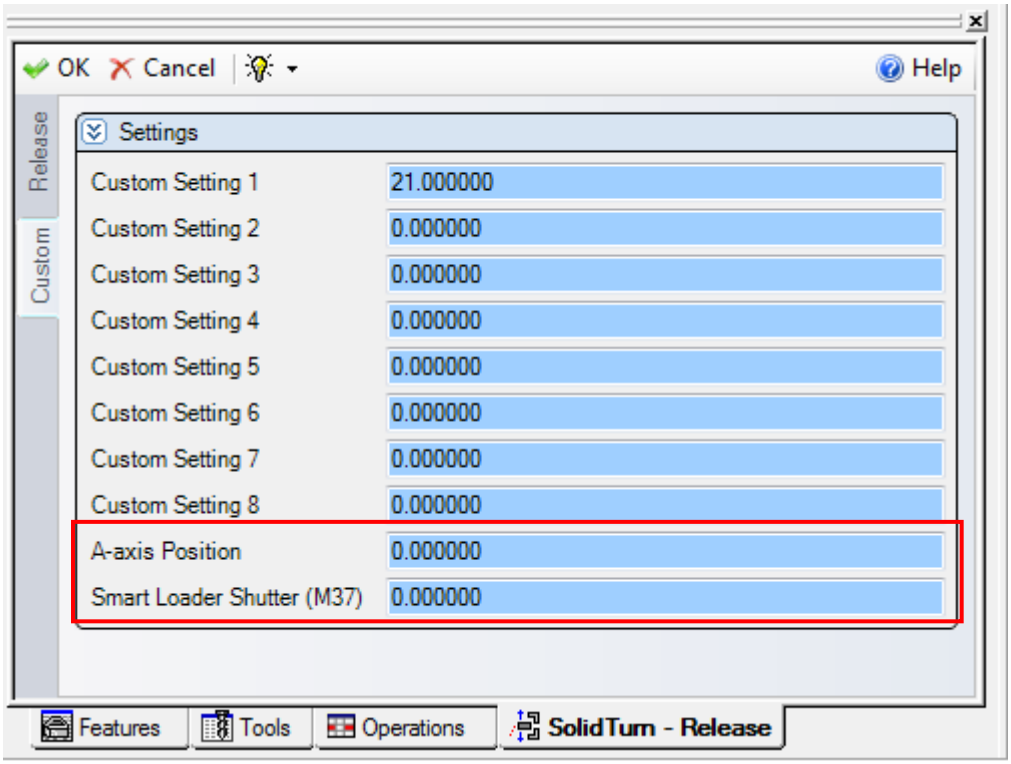
A axis position for part release (NT1000 only)

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

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

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

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

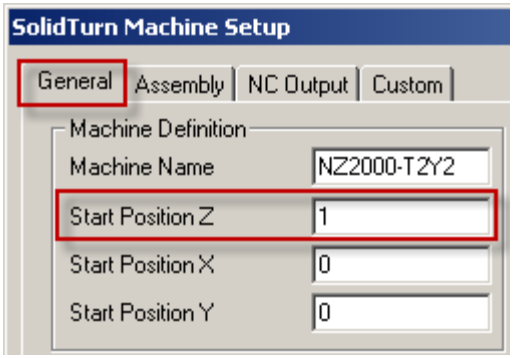


11. Working with Long Parts: Pickup\Barpull then Simultaneous Machining

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

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

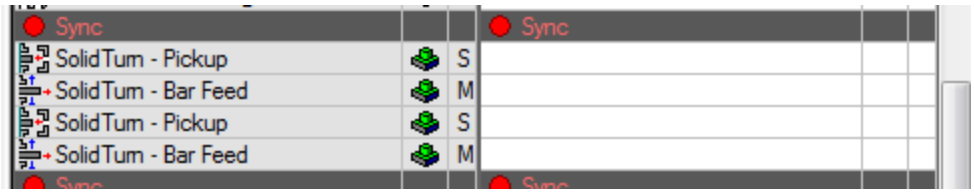
Note: On the General tab of the Machine Setup, set the Start Position Z:
Start Position Z = Length of stock for facing the front side of the part or position of the stock in spindle 1 before pick up for synchronized machining



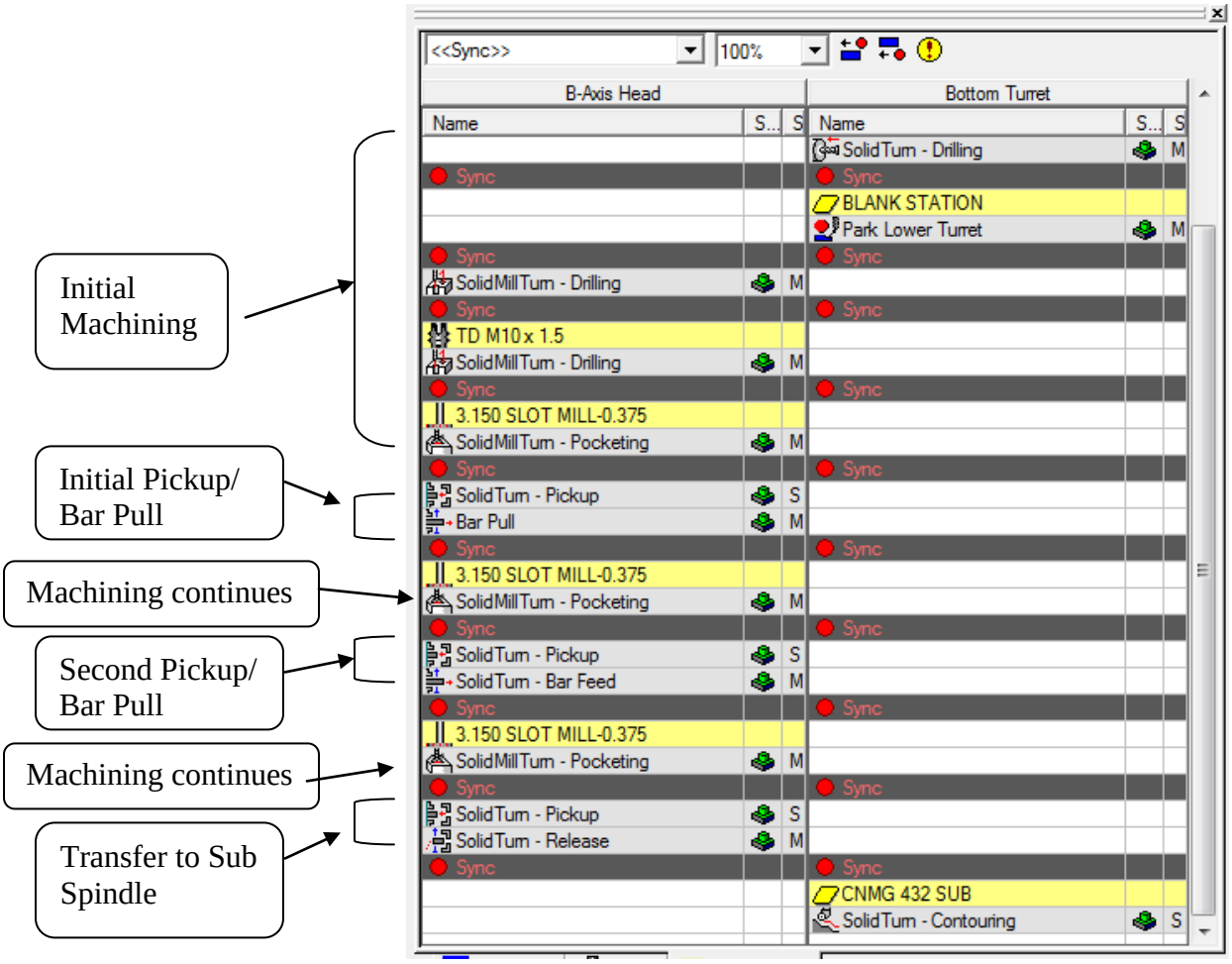
The typical steps to follow are:

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

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



It is possible to program several Pickup\Bar Pulls with machining between them to machine the part at different locations as you pull the part through the machines
If multiple pickup\bar pulls are programed the C axis orientation options will only be read from the first pickup operation.



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

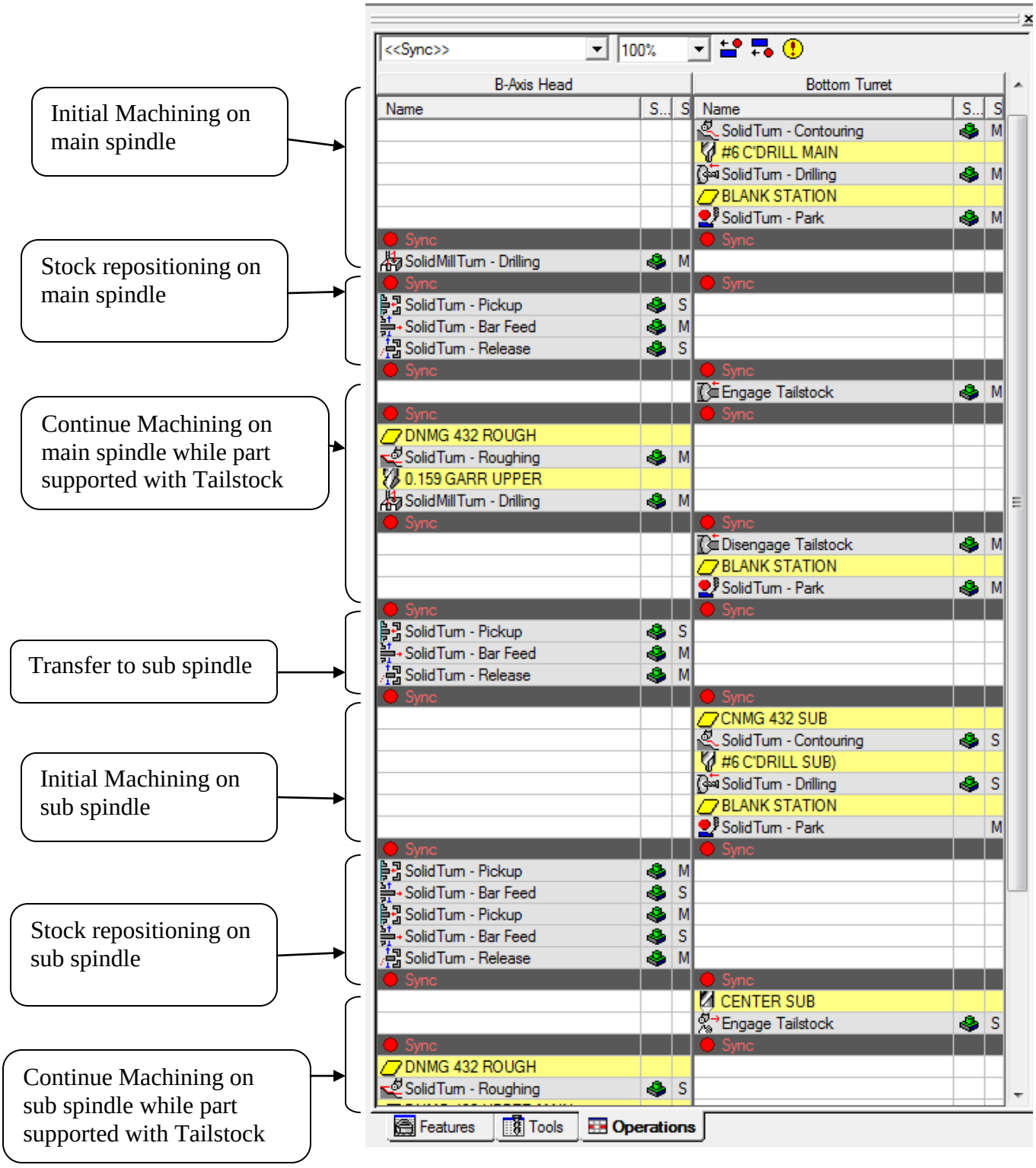
12. Stock Repositioning

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

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

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

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



13. How to Output Coolant Codes

13.1. Introduction

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

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

.

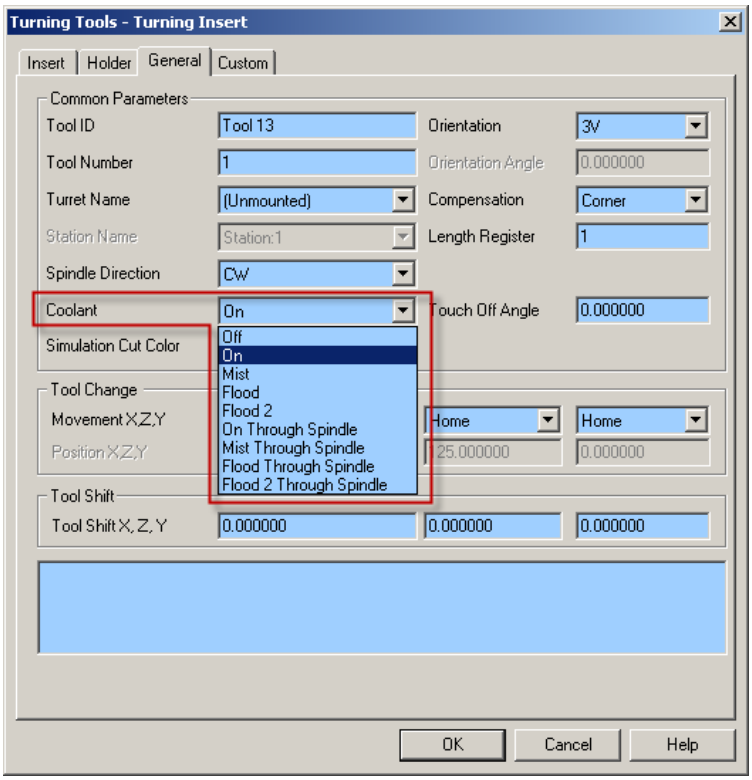
13.2. Setting First Coolant Code

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

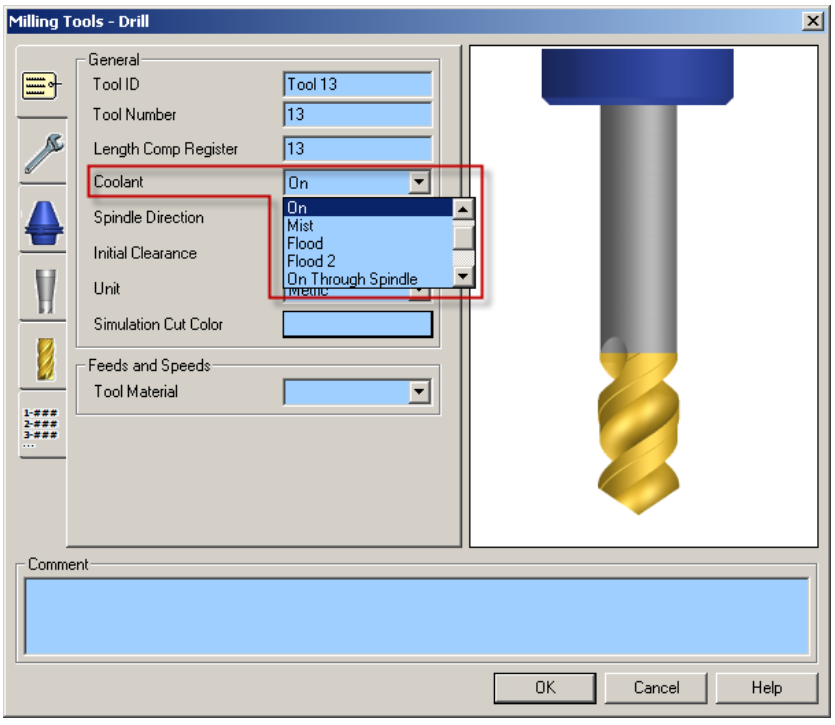
Coolant options supported by the NT Post:

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

Turning tool page:



Milling tool page:



13.3. Setting the second coolant code (optional)

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

13.4. Setting the High Pressure Coolant Pressure options

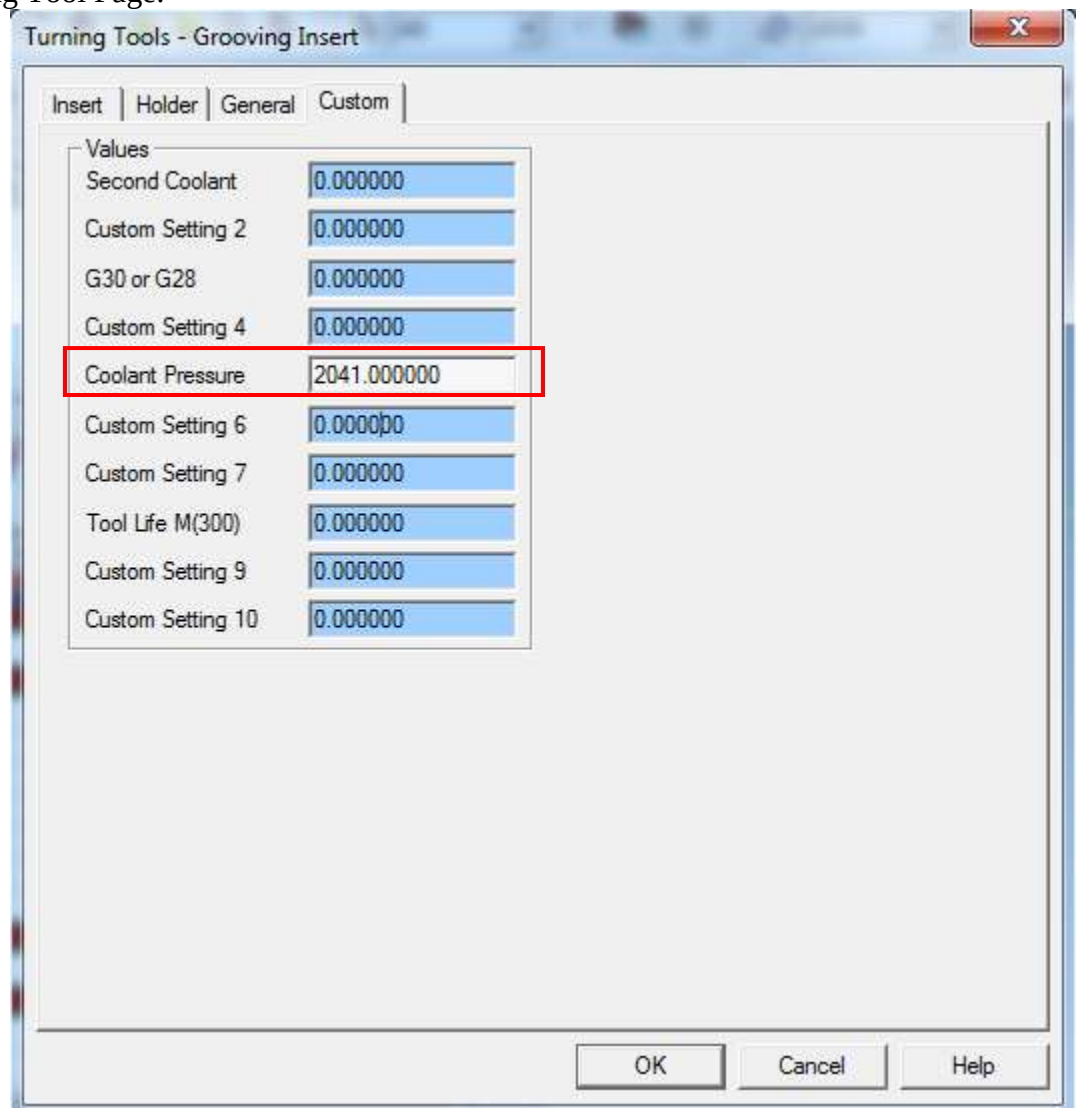
13.4.1. Upper Head Option

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

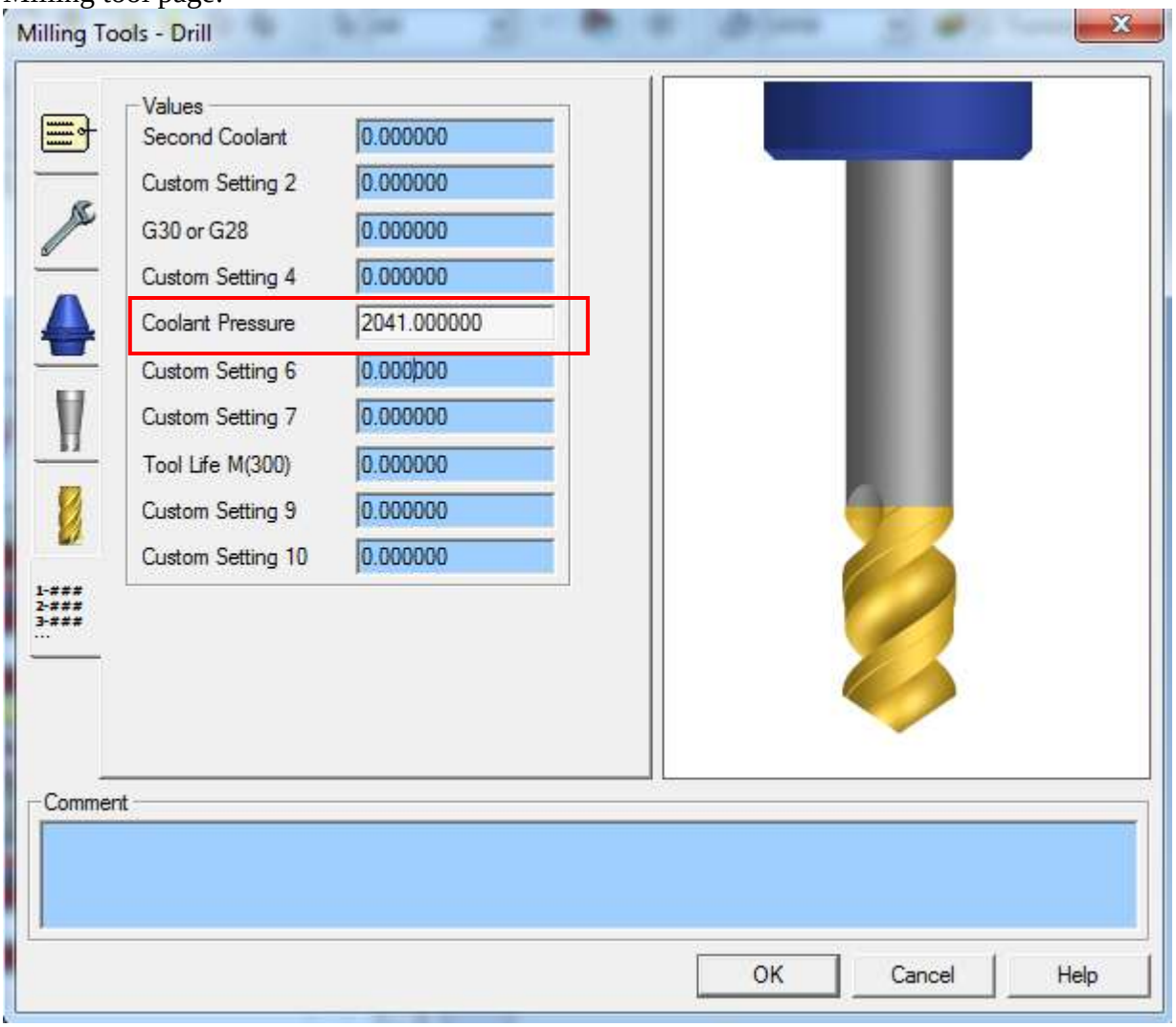
13.4.2. Lower Turret Options

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

Turning Tool Page:



Milling tool page:



14. Multiple Repetitive (Canned) Cycles

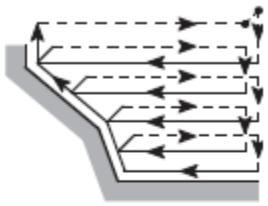
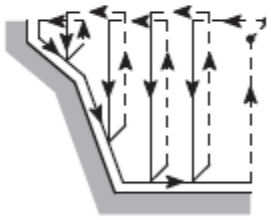
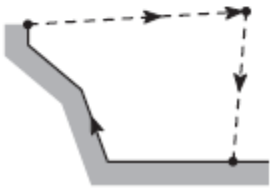
14.1. Introduction

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

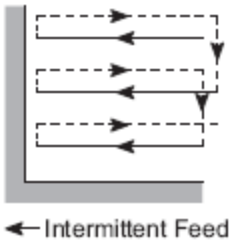
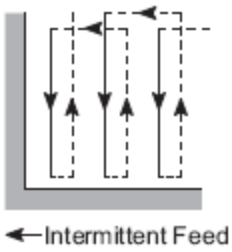
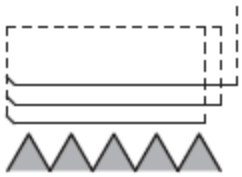
Roughing processes that require several blocks of commands can be specified by a single block of commands preceded by a G code calling a multiple repetitive cycle, and blocks that define the finished shape. The tool paths for rough cutting cycles are automatically determined.

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

O.D./I.D. and 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.

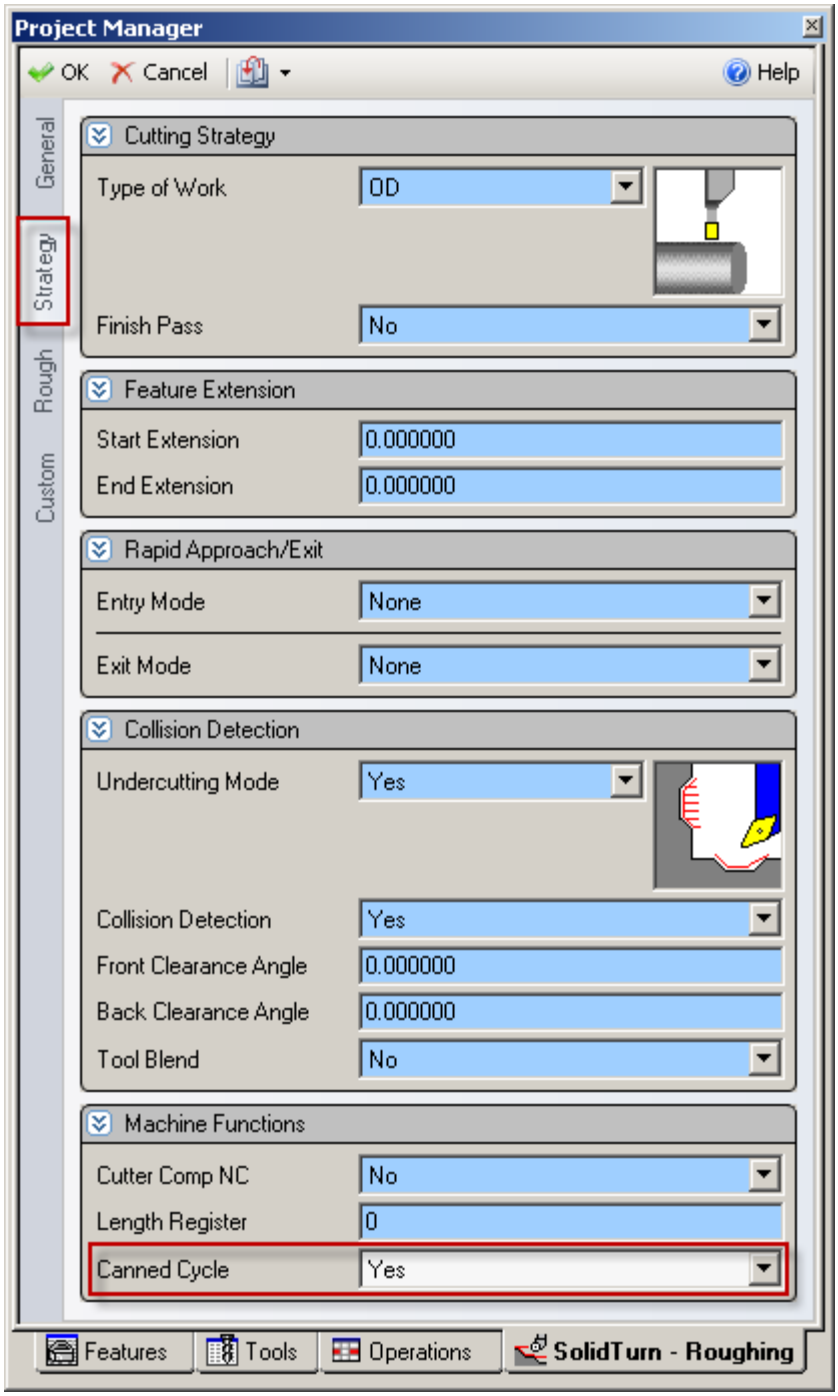
O.D./I.D. and face groove, cutoff and threading cycles:

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.

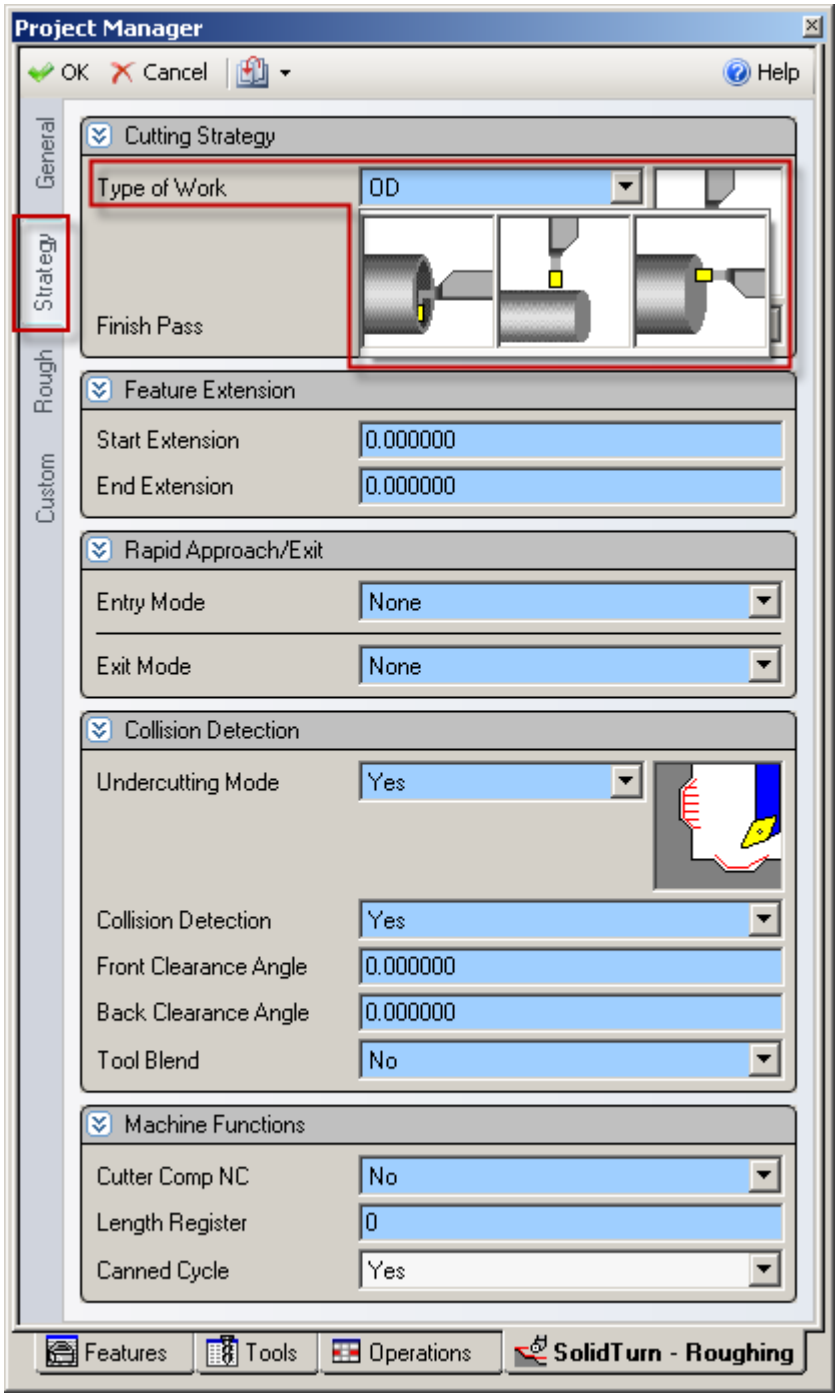
14.2. Roughing Cycle

14.2.1. Introduction

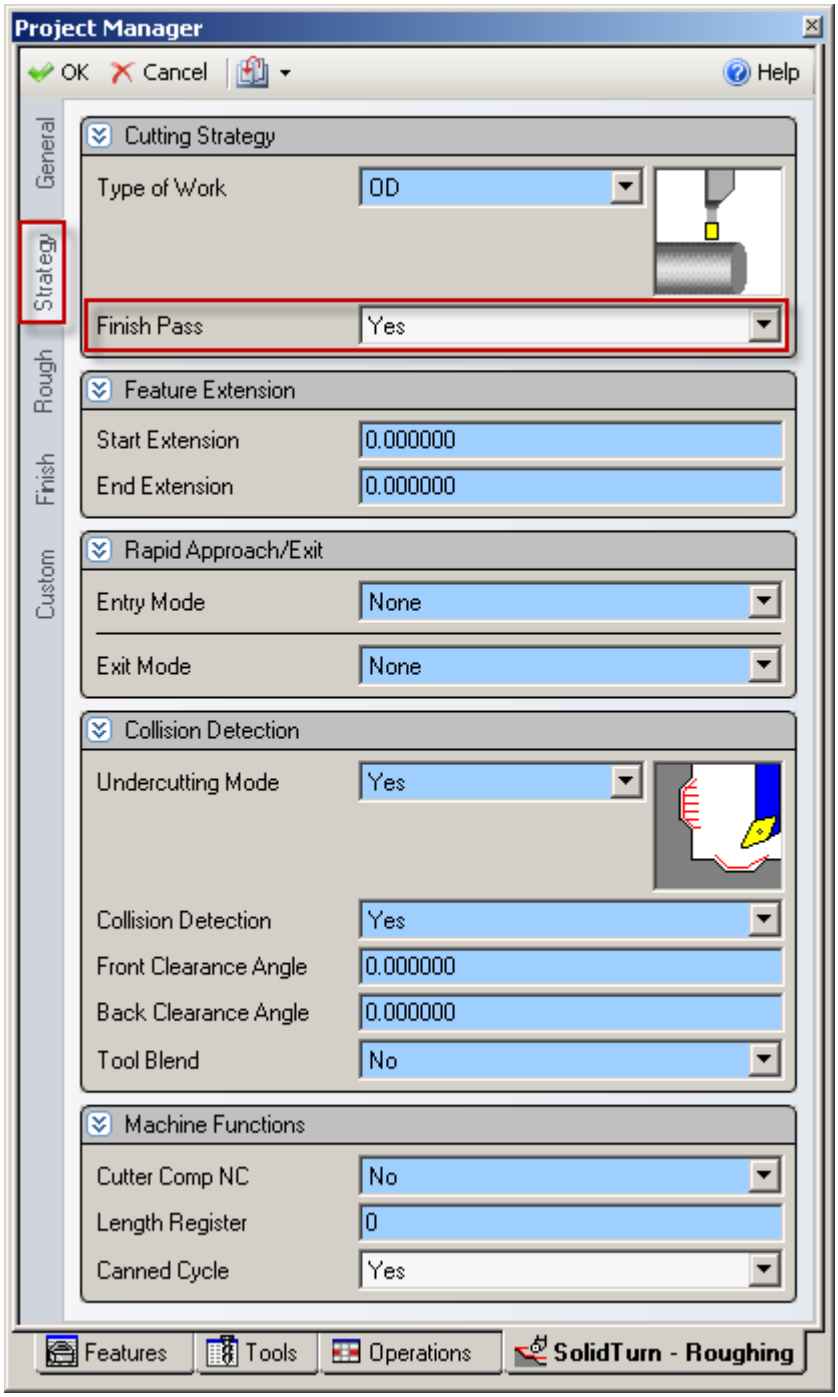
In ESPRIT, on the Strategy tab of the SolidTurn Roughing operation page, set Canned Cycle to Yes to trigger the multiple repetitive cycle output.



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



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

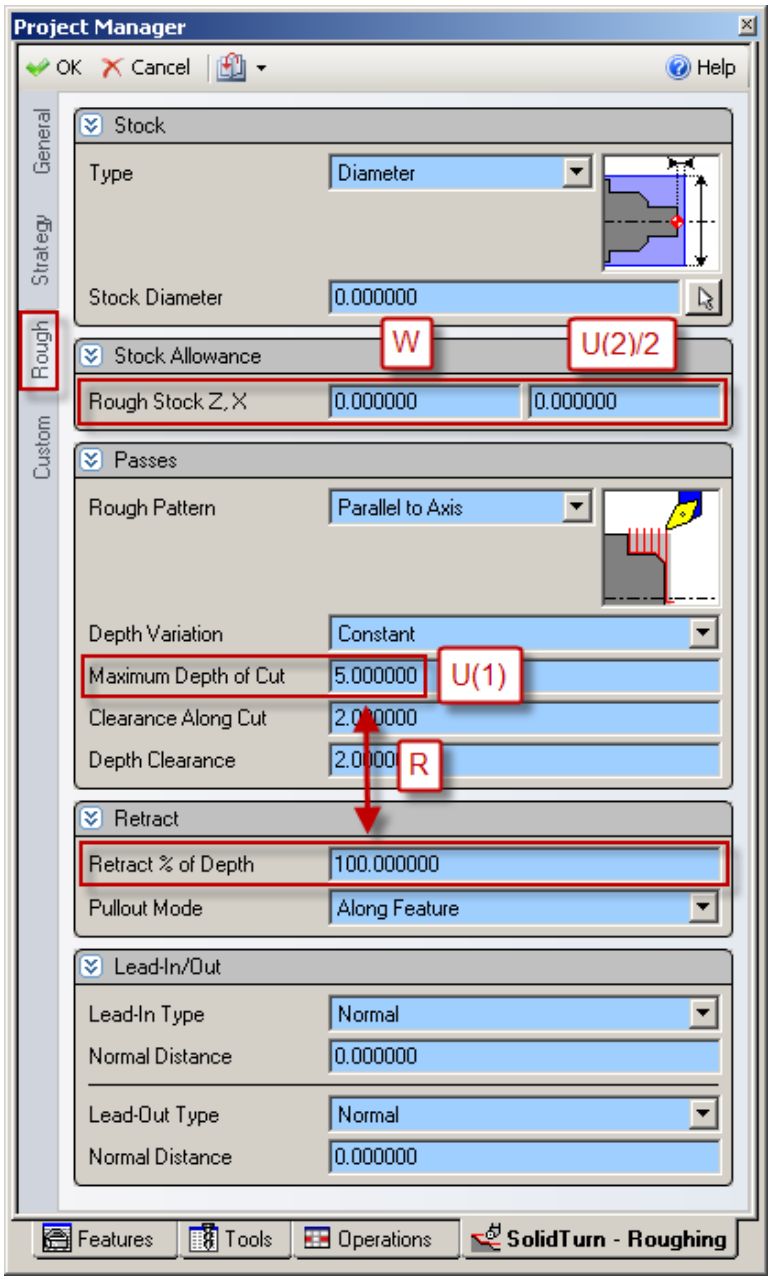


Information about some of the parameters to enter for the following cycles:

- 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.

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

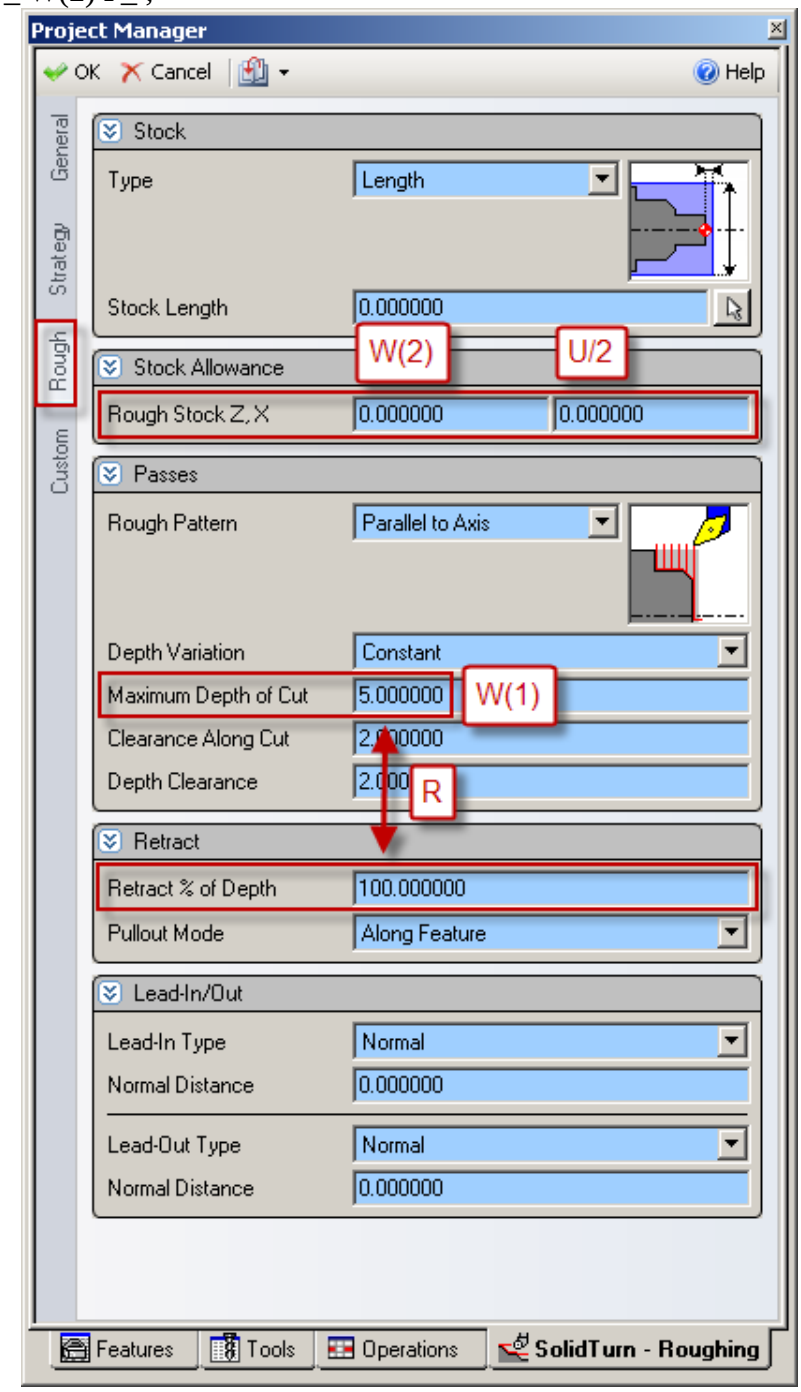
G71 U(1) R_ ;
G71 P_ Q_ U(2) W_ F_ ;



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

14.2.3. Face Roughing

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

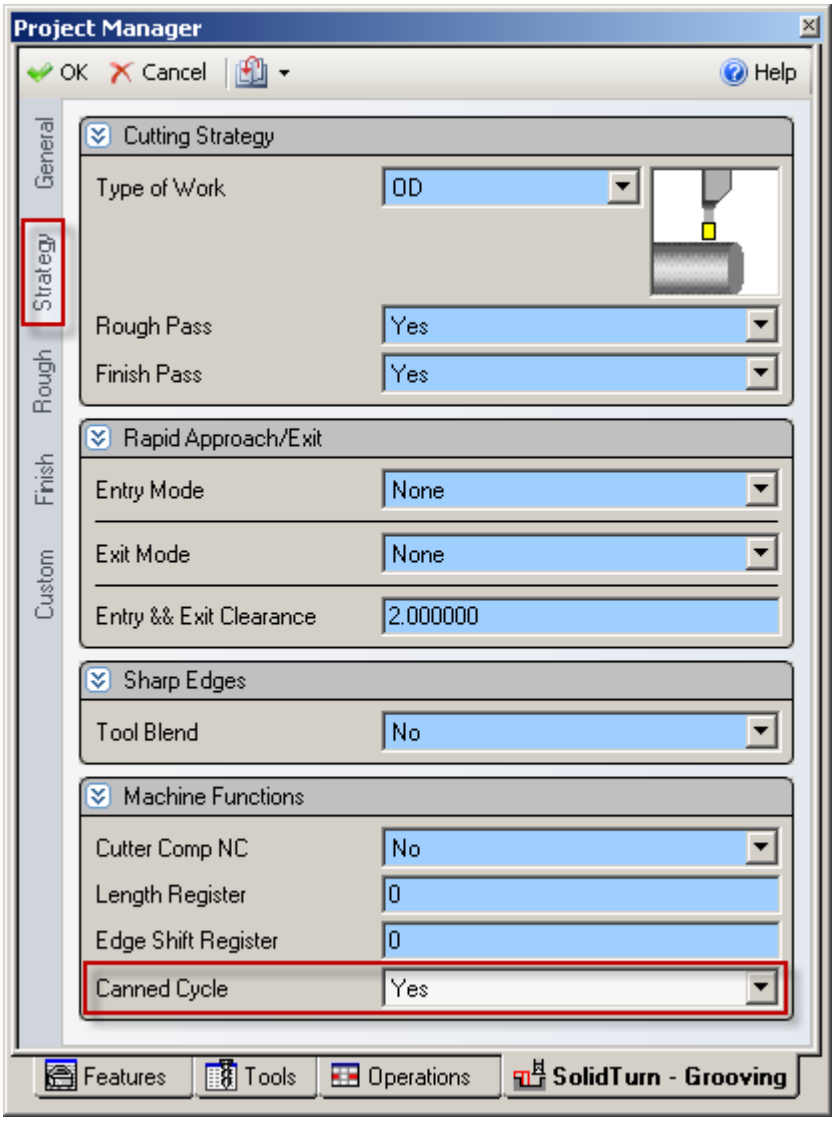


W(1) and W(2) will be output as entered in ESPRIT. U/2 will be multiplied by 2 to have the correct value in the NC code U. R will be computed based on the entered Maximum Depth of Cut and Retract % of Depth value: R = Maximum Depth of Cut * Retract % of Depth / 100.

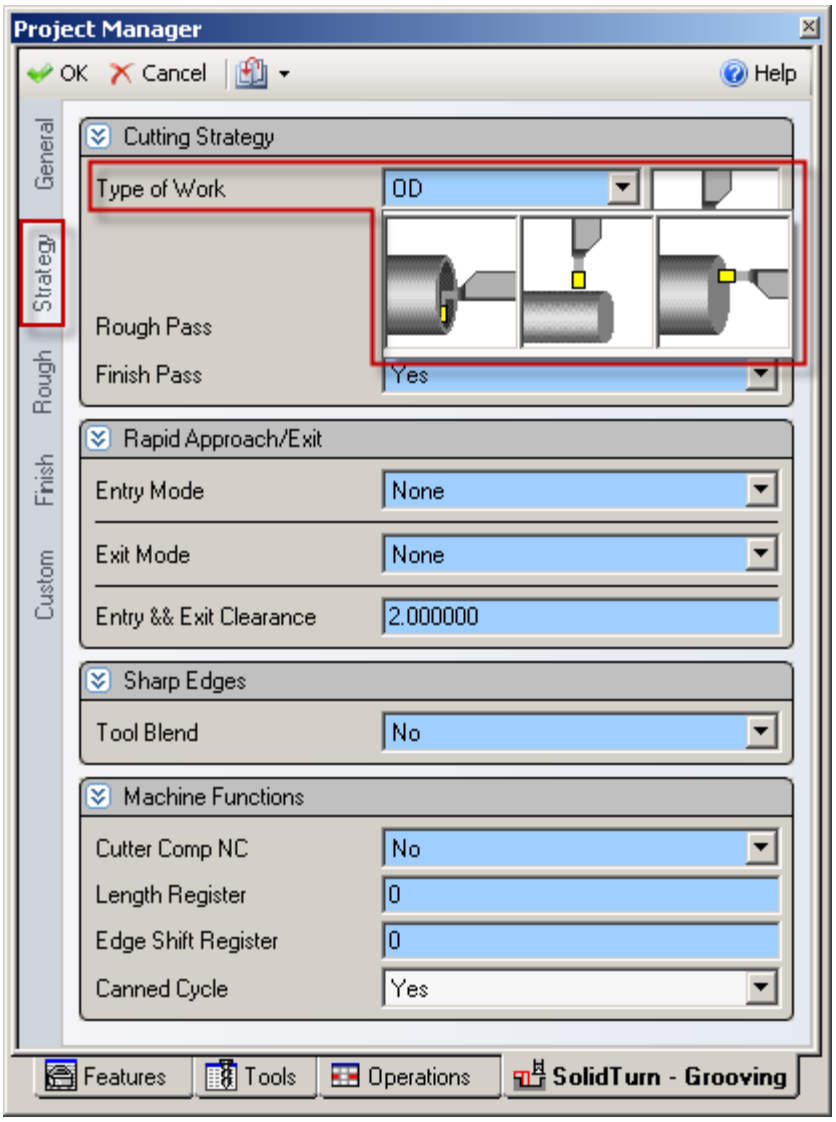
14.3. Grooving Cycle

14.3.1. Introduction

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



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.



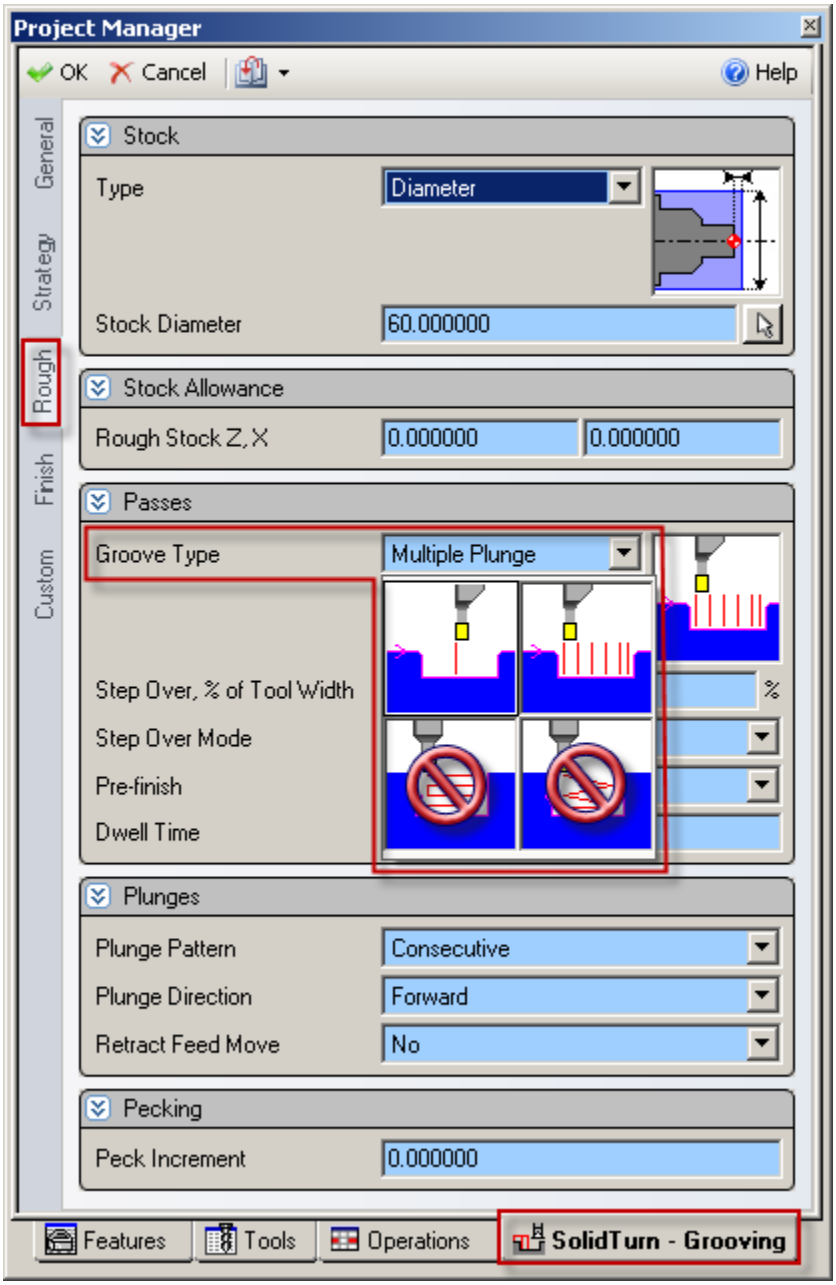
Information about some of the parameters to enter for the following cycles:

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

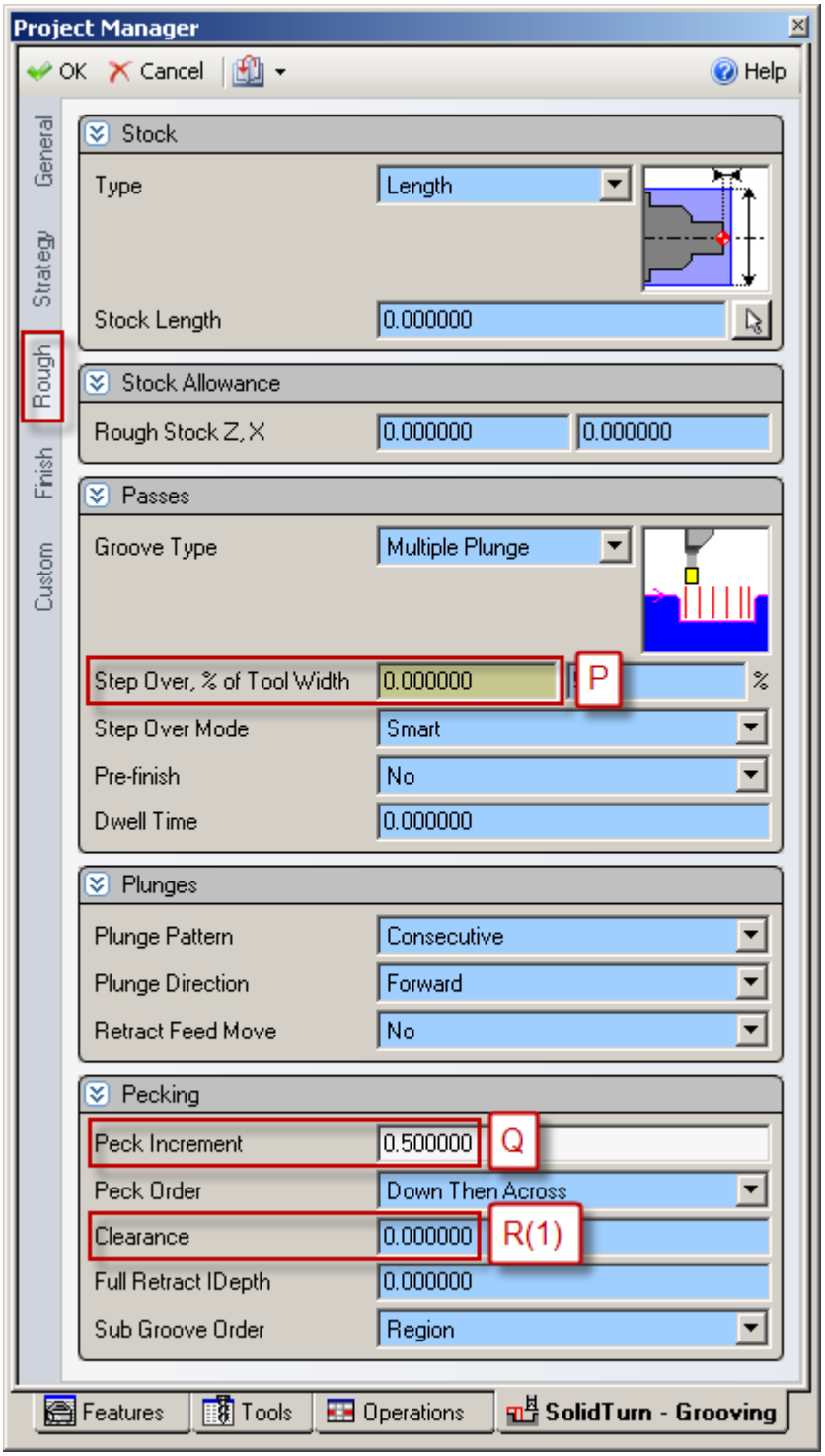
NOTE: Grooving canned cycles can only be used on grooves with straight wall geometry. Grooves with a side taper angle must be programmed with a regular groove operation.

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



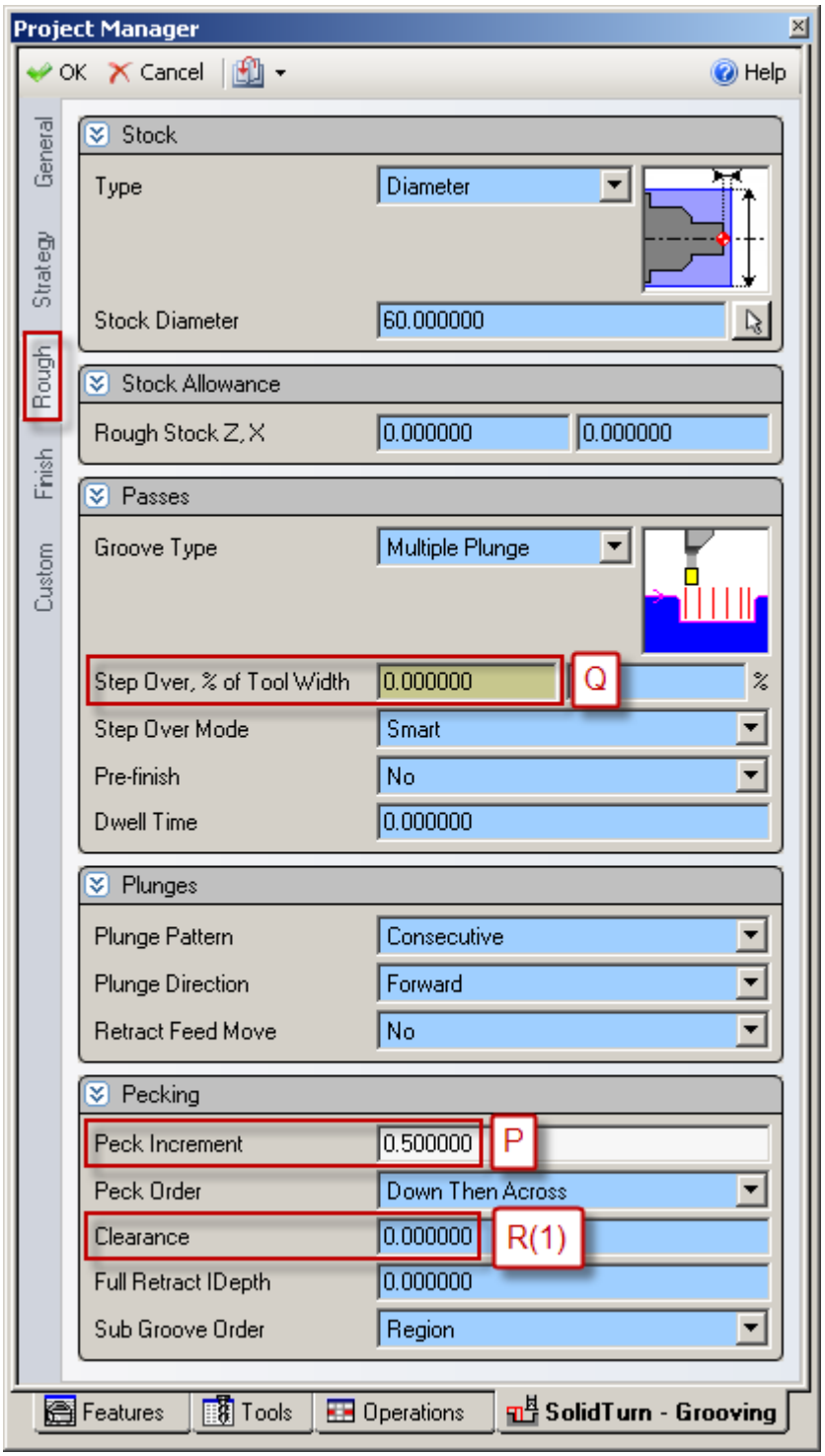
14.3.2. Face Grooving

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



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

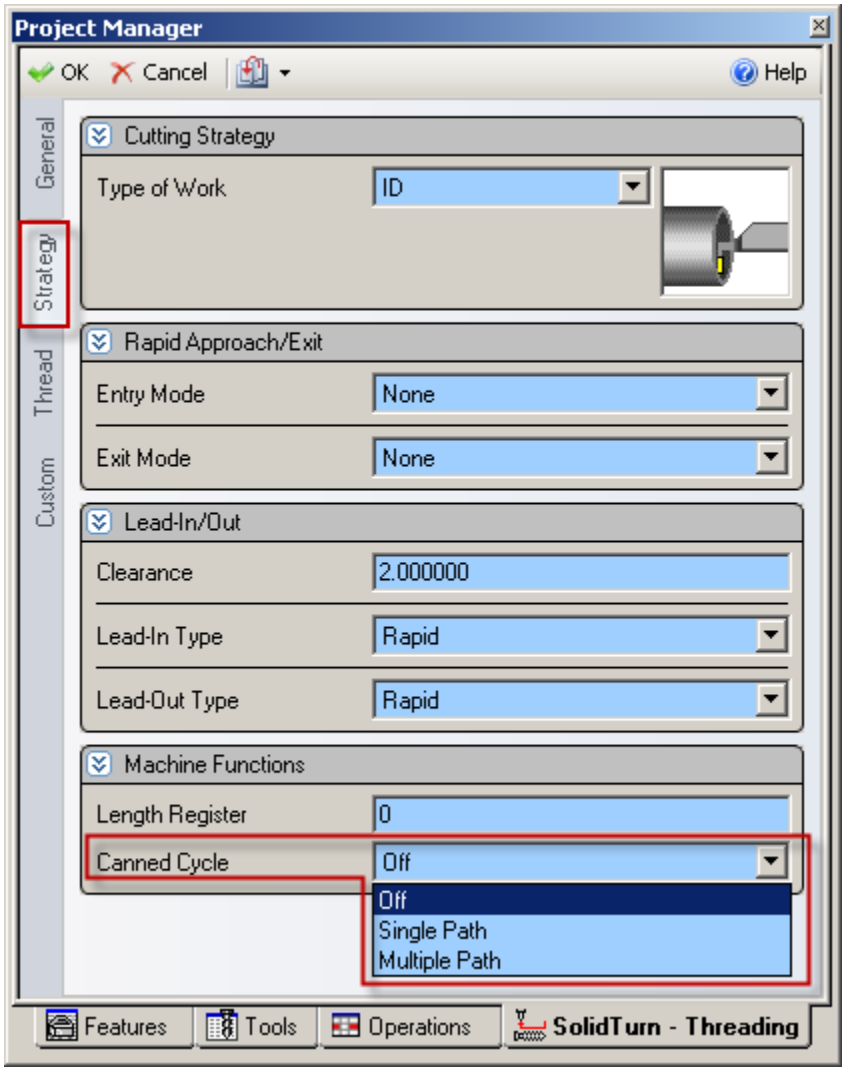
G75 R(1) ;
G75 X_ Z_ P_ Q_ F_ ;



14.4. Threading Cycle

14.4.1. Introduction

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

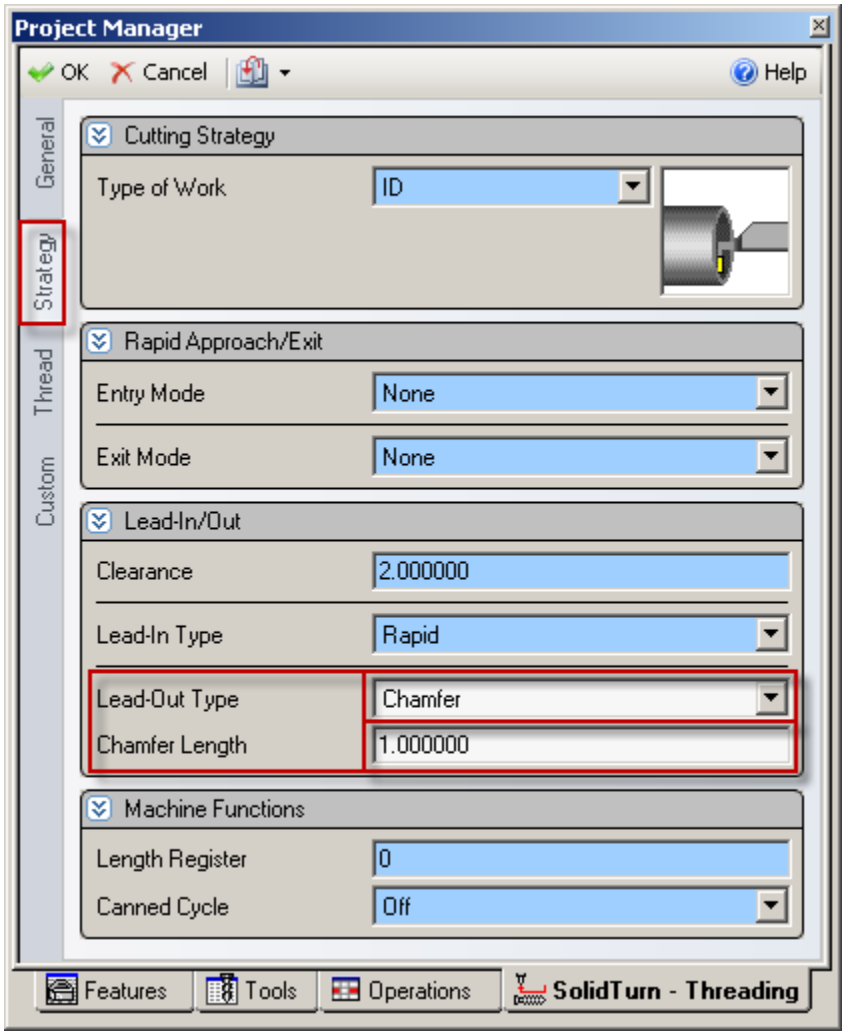


Information about some of the parameters to enter for the following cycles:

- 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.

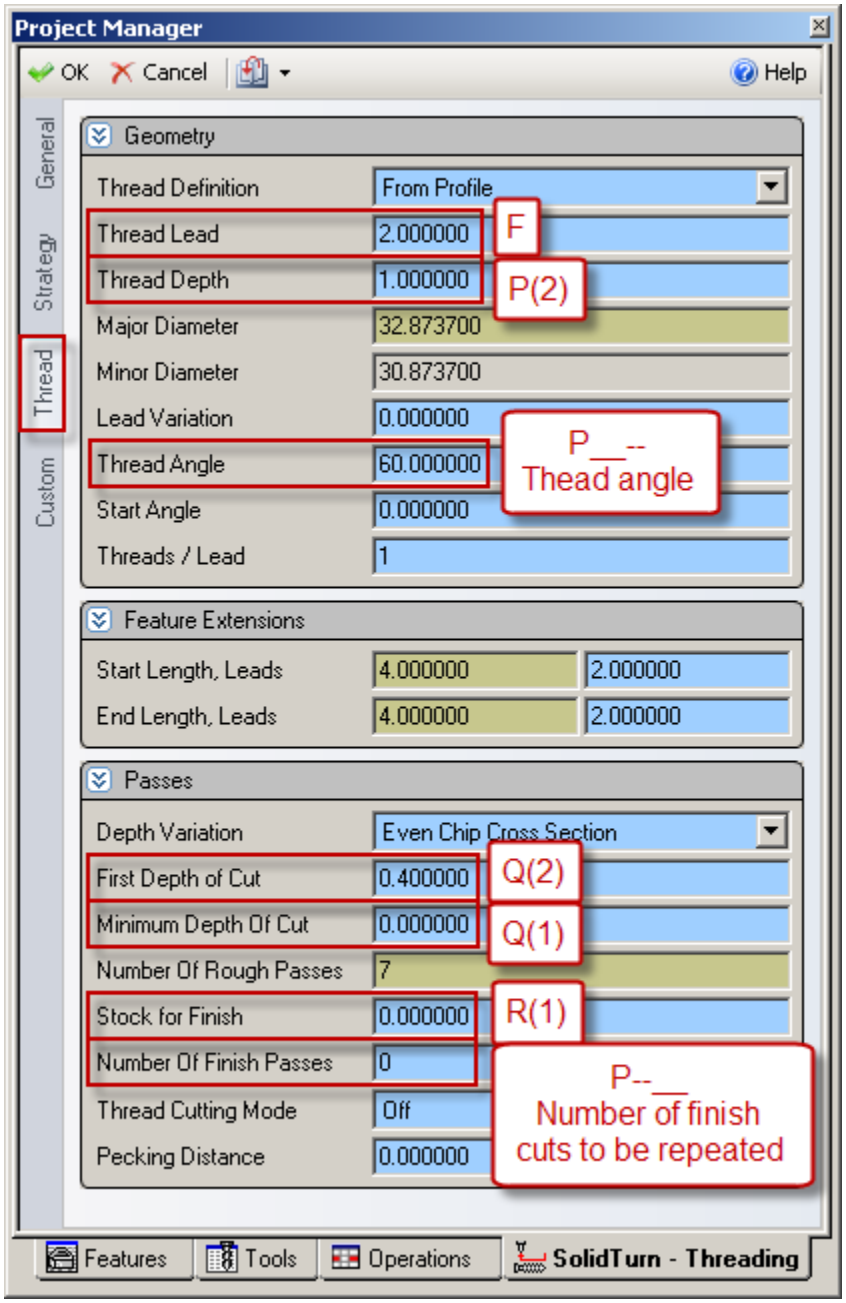
- The cut speed S can be specified on the General tab of the operation.

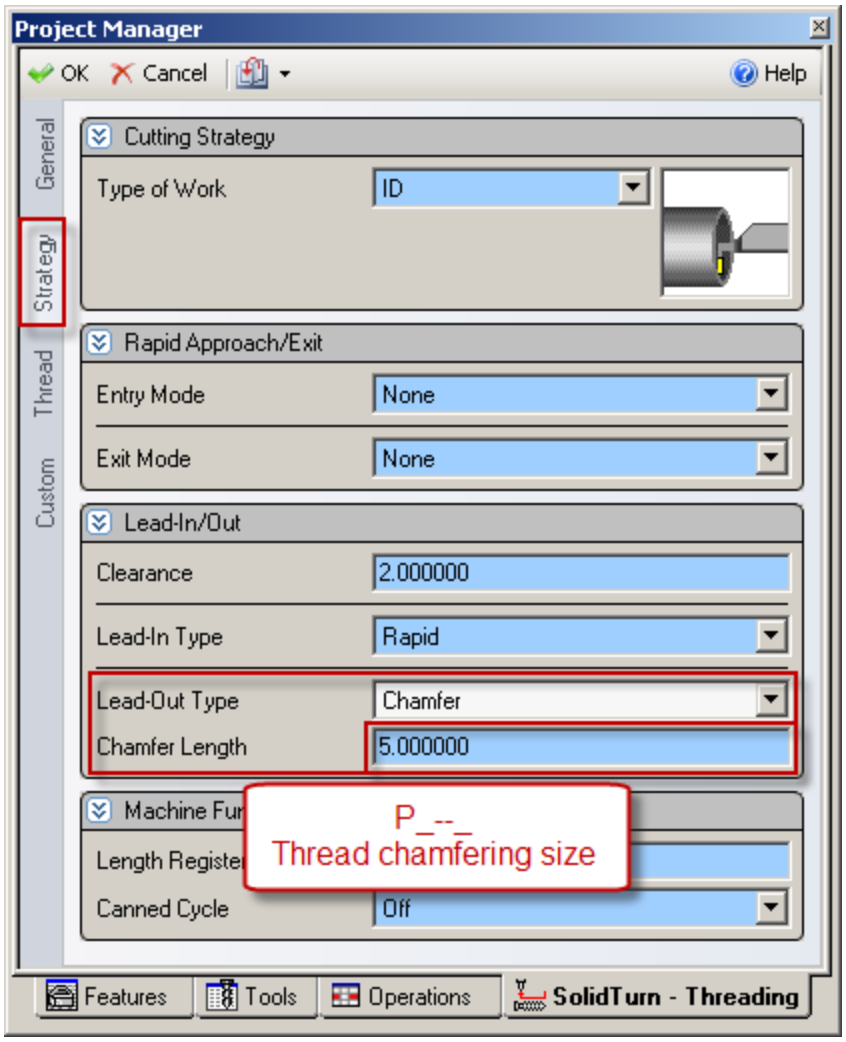
On the Strategy tab, if you set the Lead-Out Type to Chamfer and the Chamfer Length greater than 0, you will have M23 (chamfer ON) output prior to the G76 or G92 commands. With a different lead-out set, the M-code output will be M24 (chamfer OFF).



14.4.2. Threading with Canned Cycle Set to Single Path

G76 P(1) Q(1) R(1) ;
G76 X_ Z_ R(2) P(2) Q(2) F_ ;



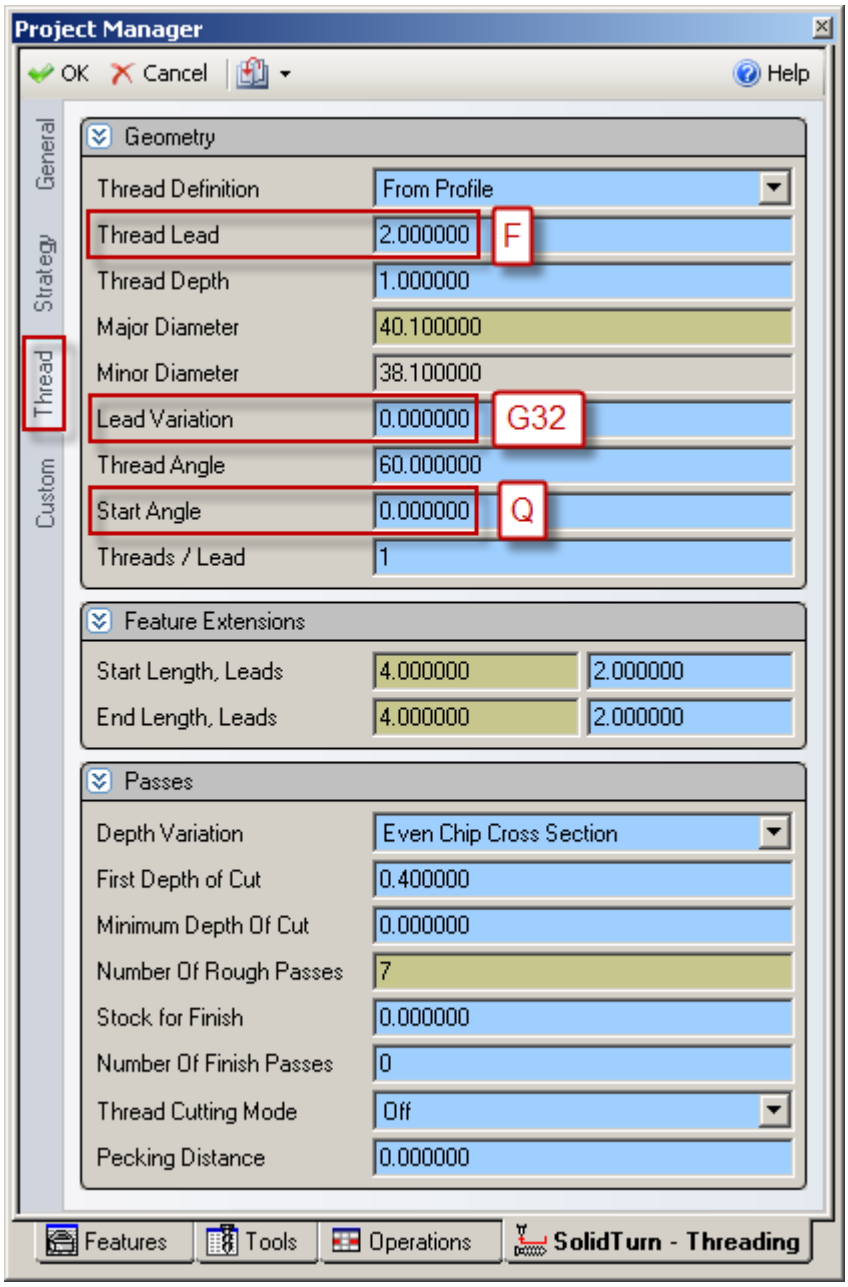


R(1) will be output as entered in ESPRIT. Q(1), P(2) and Q(2) are unsigned values, in radius. Specify them in mm, they will then be output in units of 0.001 mm without decimal point. For example, if you enter 2 for Q(1) in ESPRIT, the output will be in the NC code Q2000.

14.4.3. Threading with Canned Cycle Set to OFF

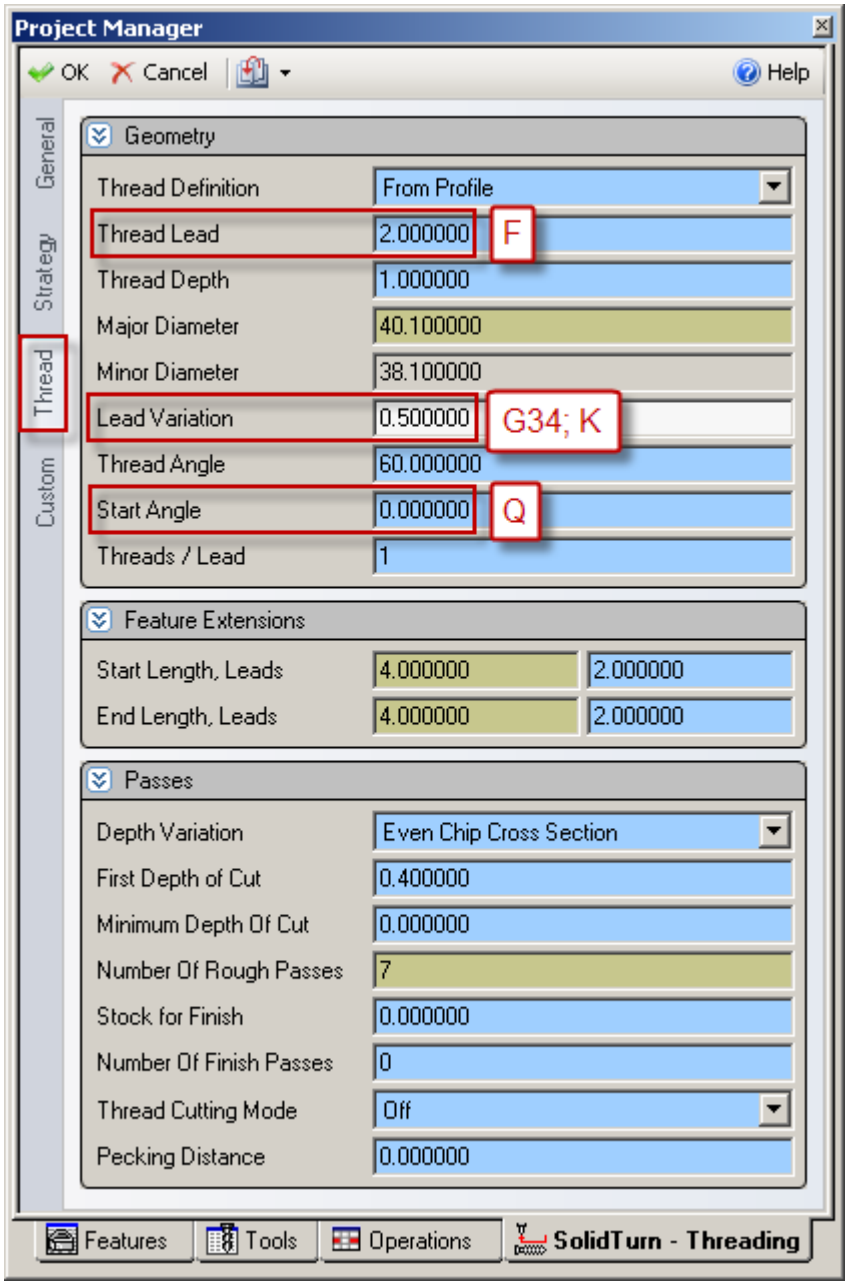
14.4.3.1. Lead Variation Set to Zero (0)

G32 X_ Z_ F_ Q_ ;



14.4.3.2. Lead Variation Not Set to Zero (<>0)

G34 X_ Z_ F_ K_ Q_ ;



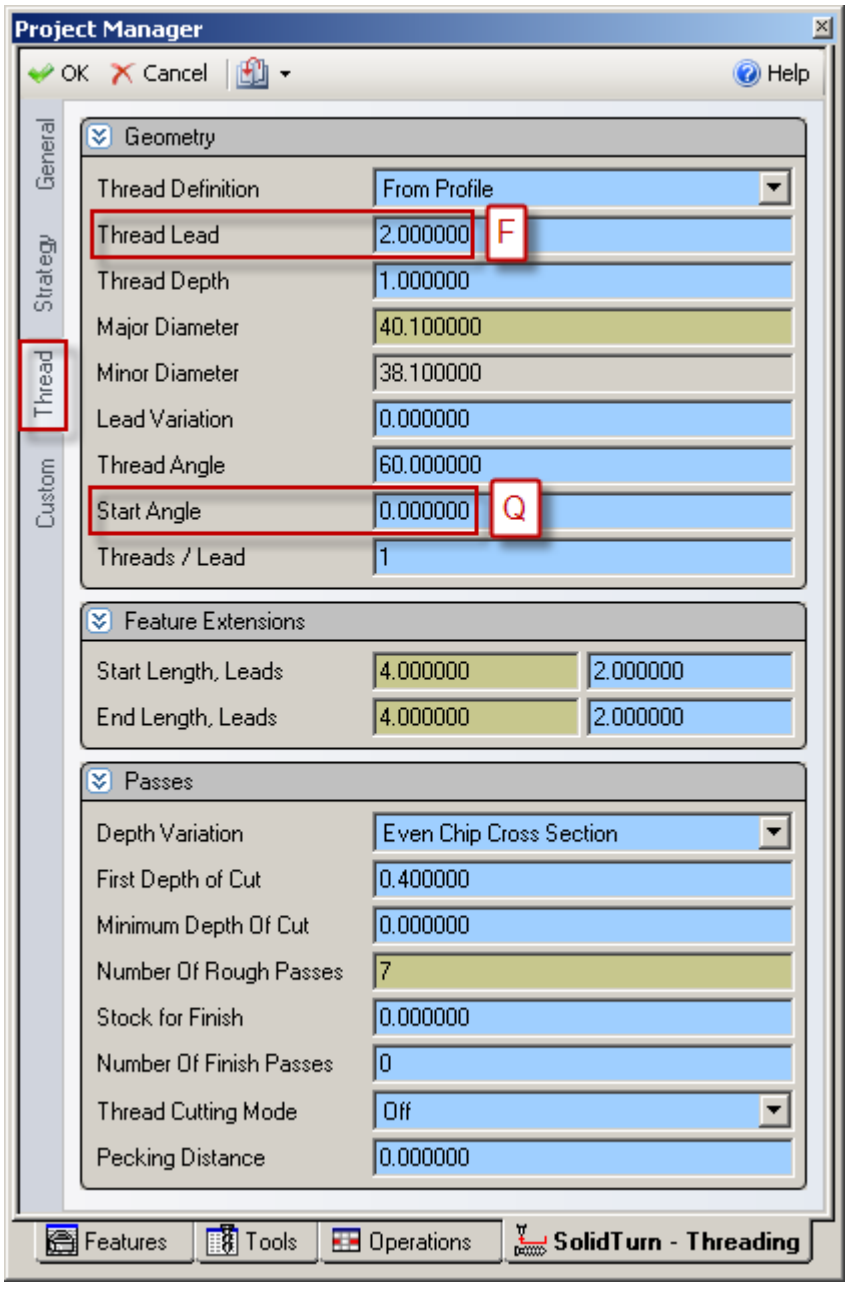
K will be output as entered in ESPRIT.

14.4.4. Threading with Canned Cycle Set to Multiple Path

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

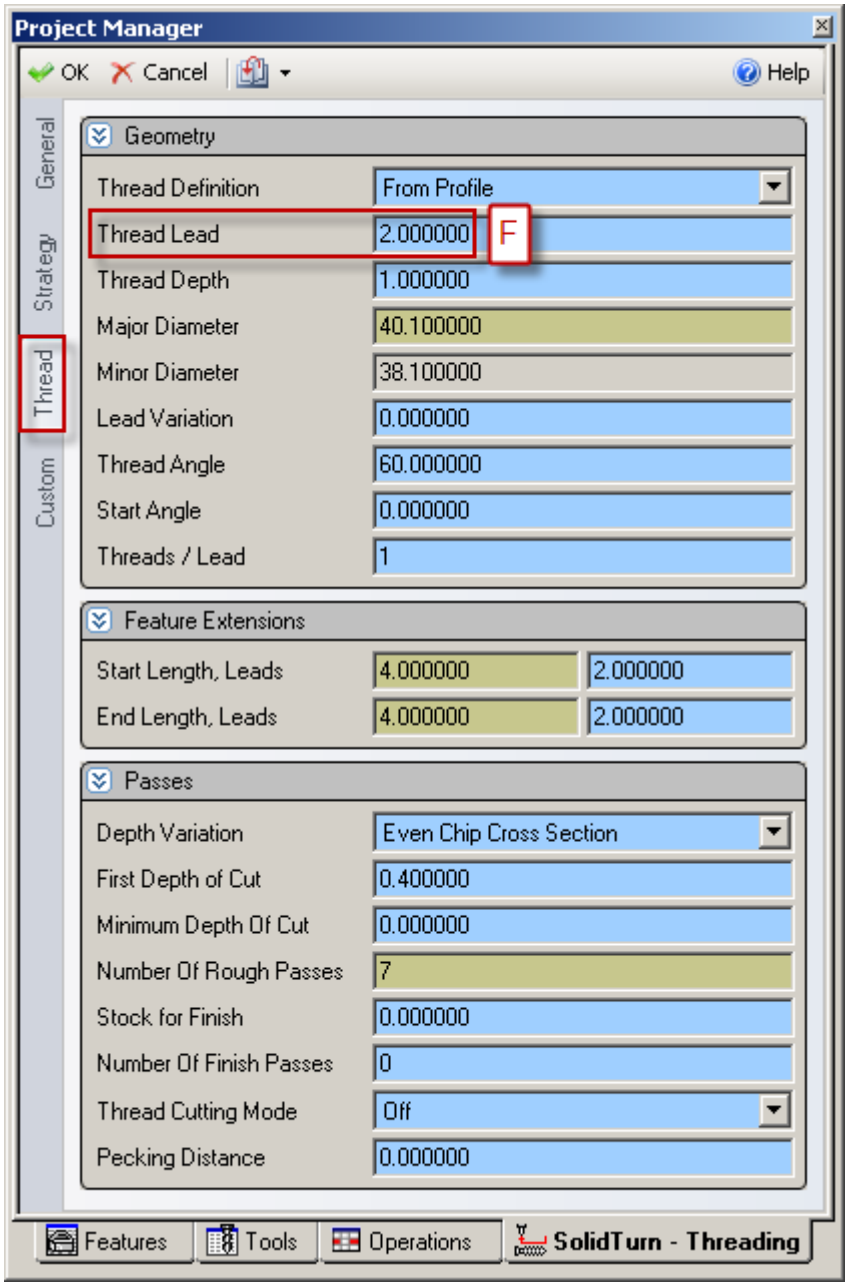
14.4.4.1. Straight Thread Cutting

G92 X_ Z_ F_ Q_ ;



14.4.4.2. Tapered Thread Cutting

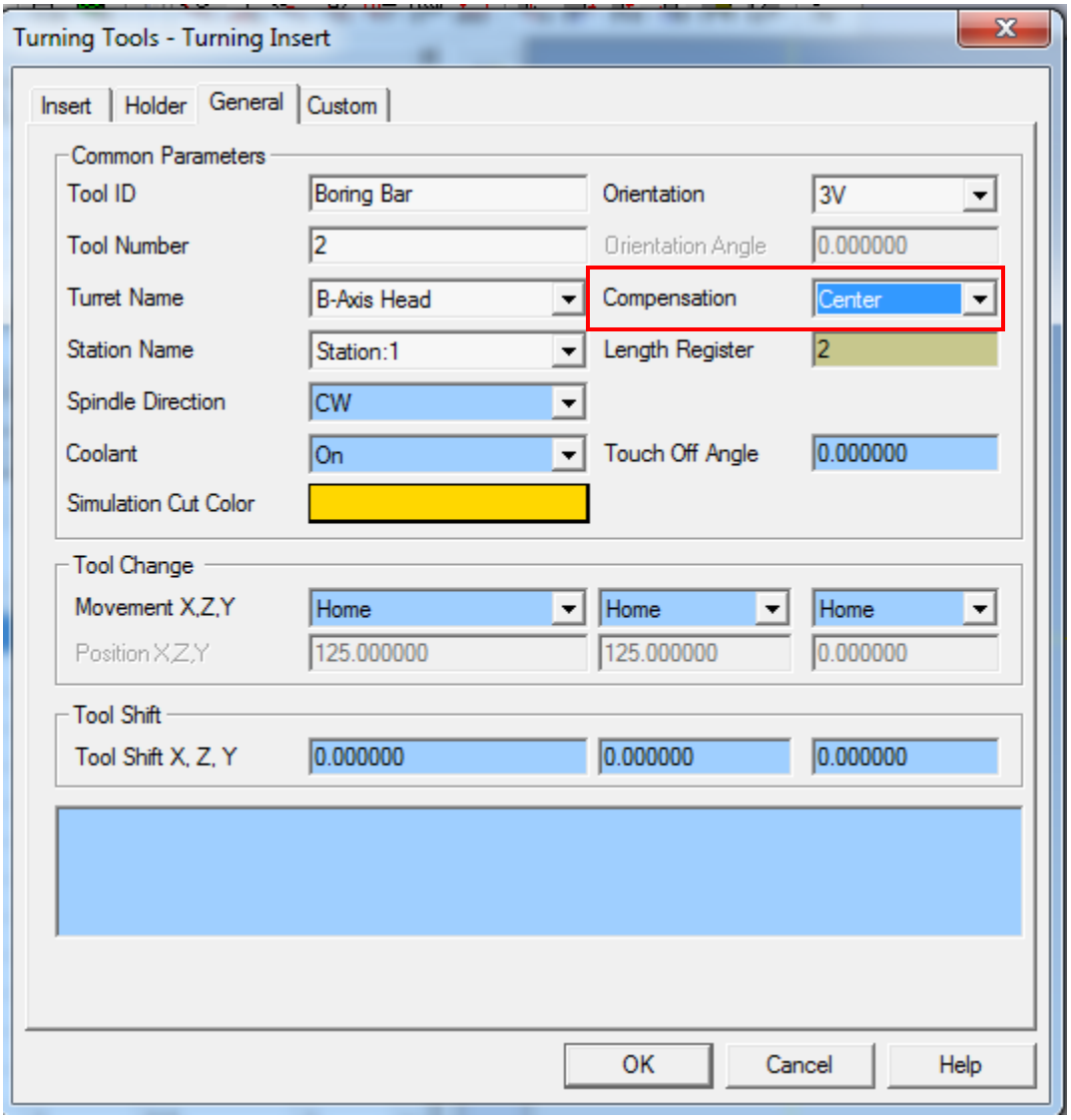
G92 X_ Z_ R_ F_ ;



15. B axis Turing Cycle

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

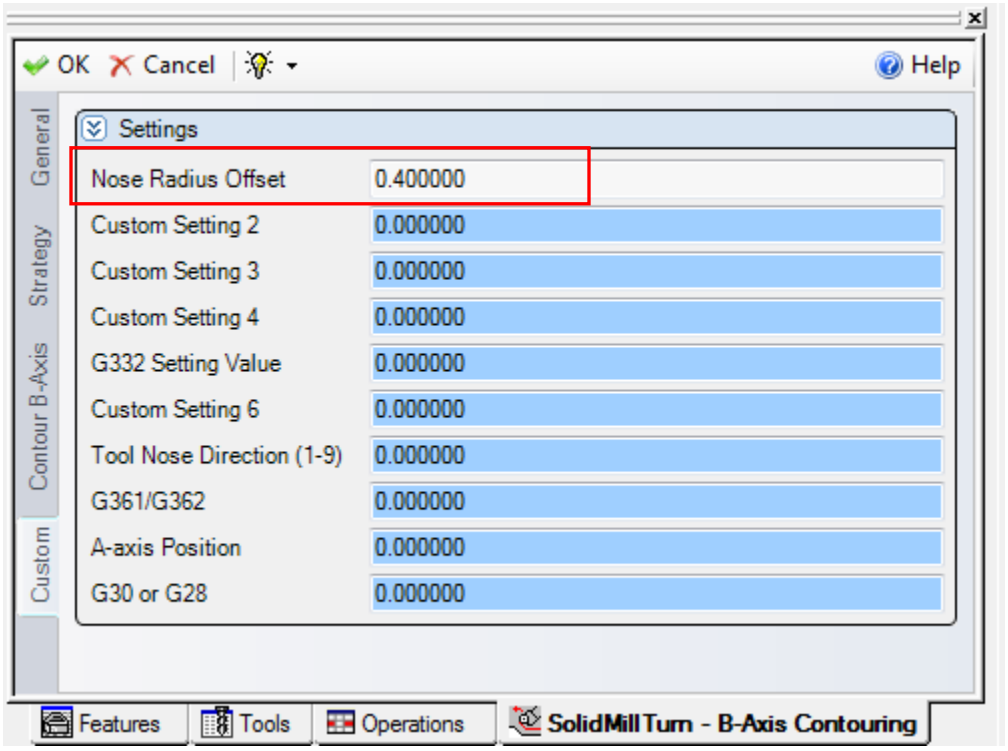
The tool Compensation filed MUST be set to “Center” for ESPRIT to correctly calculate the tool path for a B axis turning cycle.



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

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

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



16. Hole Machining Canned Cycles

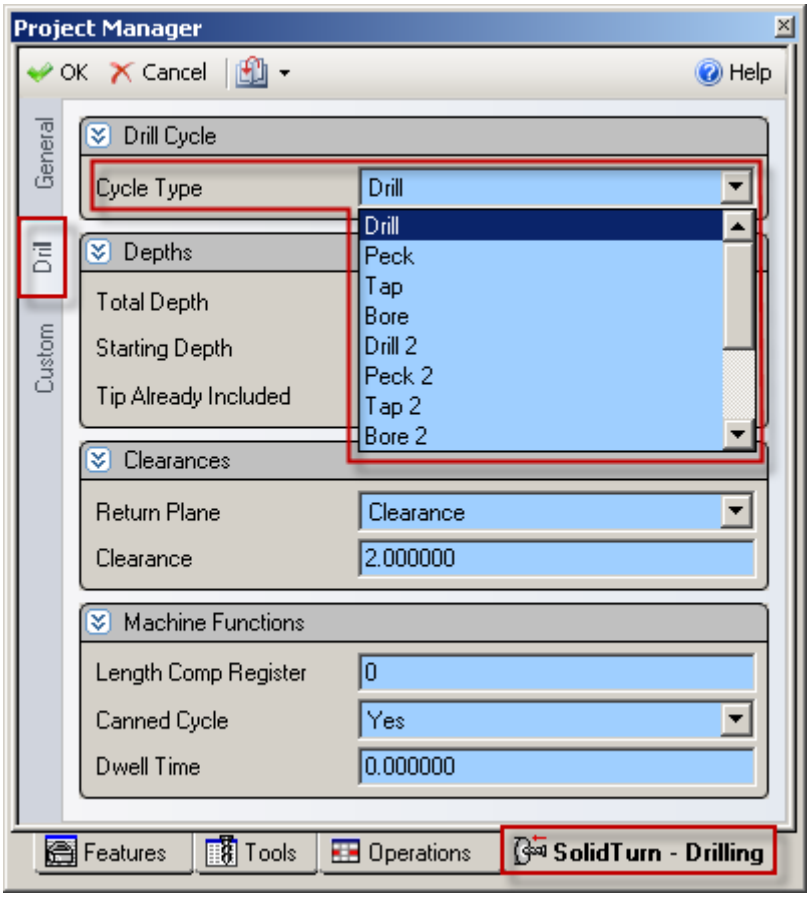
16.1. Introduction

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

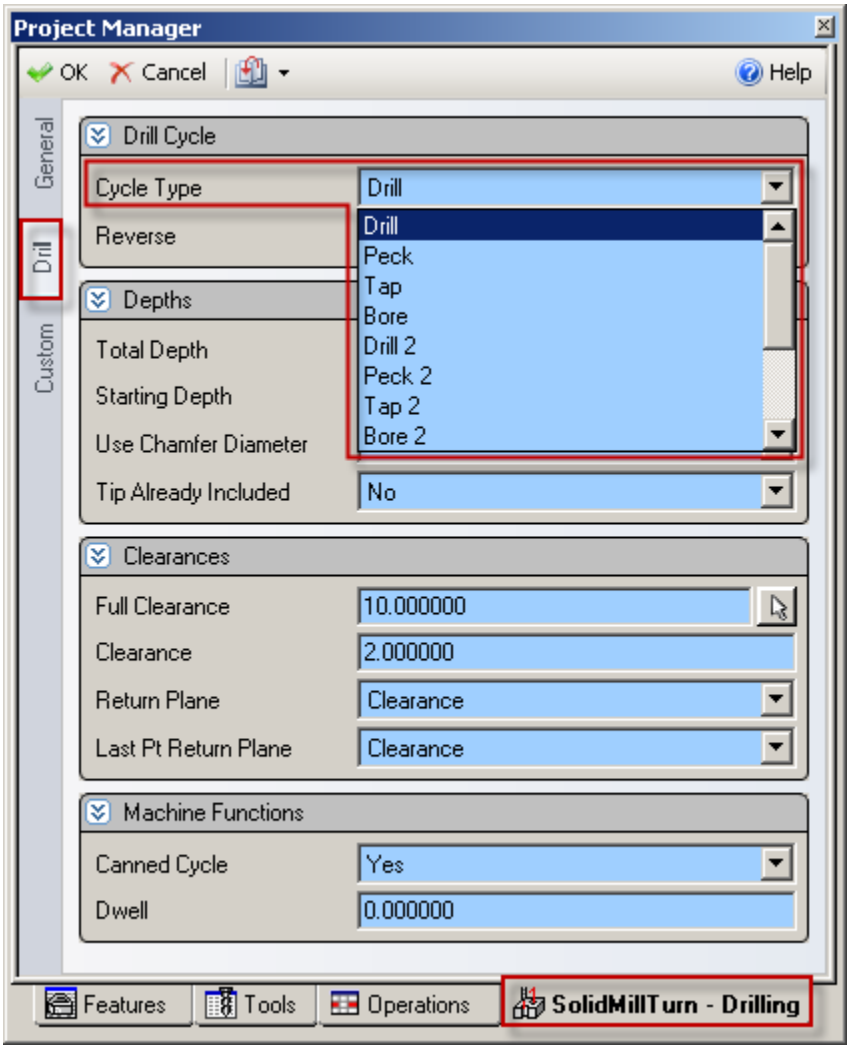
The Face Hole Machining Canned Cycle:
Hole machining is executed by the Z-axis with positioning executed by the combination of the X-, C- and Y-axes.

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

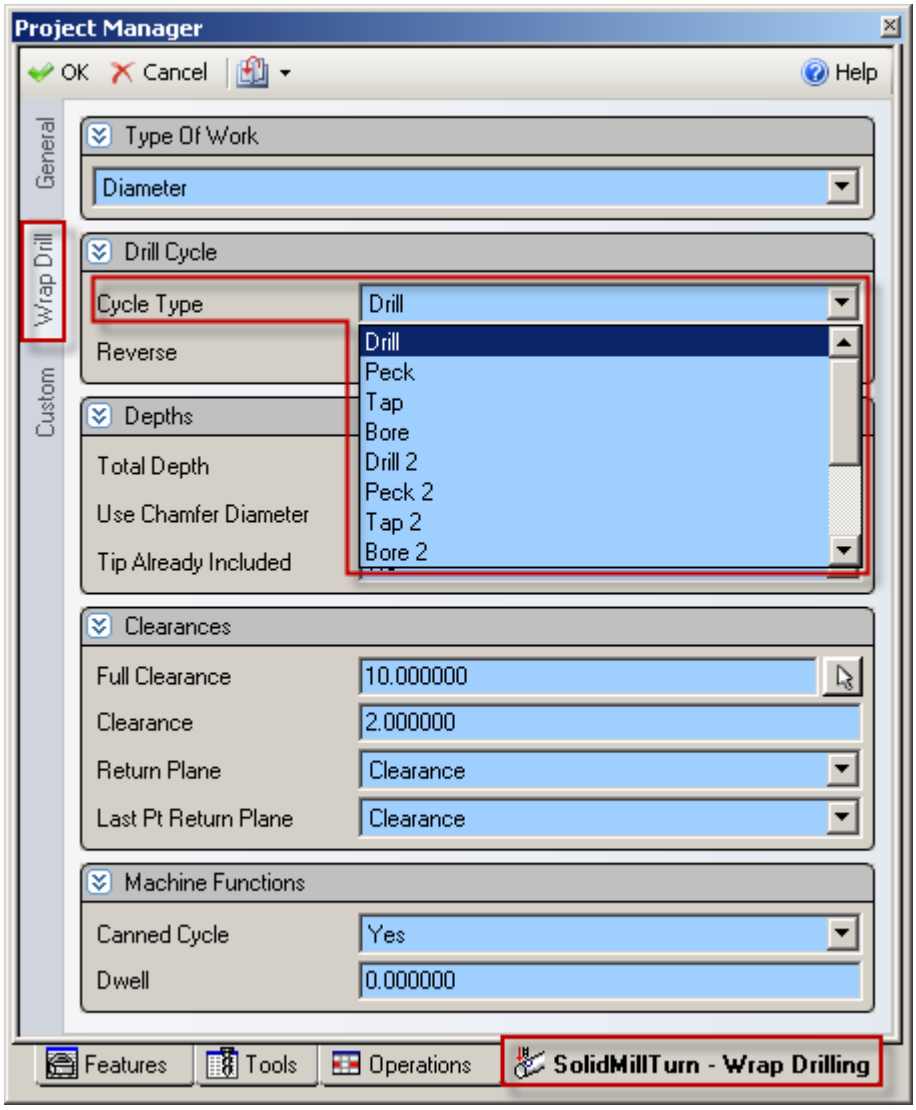
In ESPRIT, you can choose between the different drilling canned cycles using the Cycle Type pull down on the Drill tab for the SolidTurn Drilling cycle:



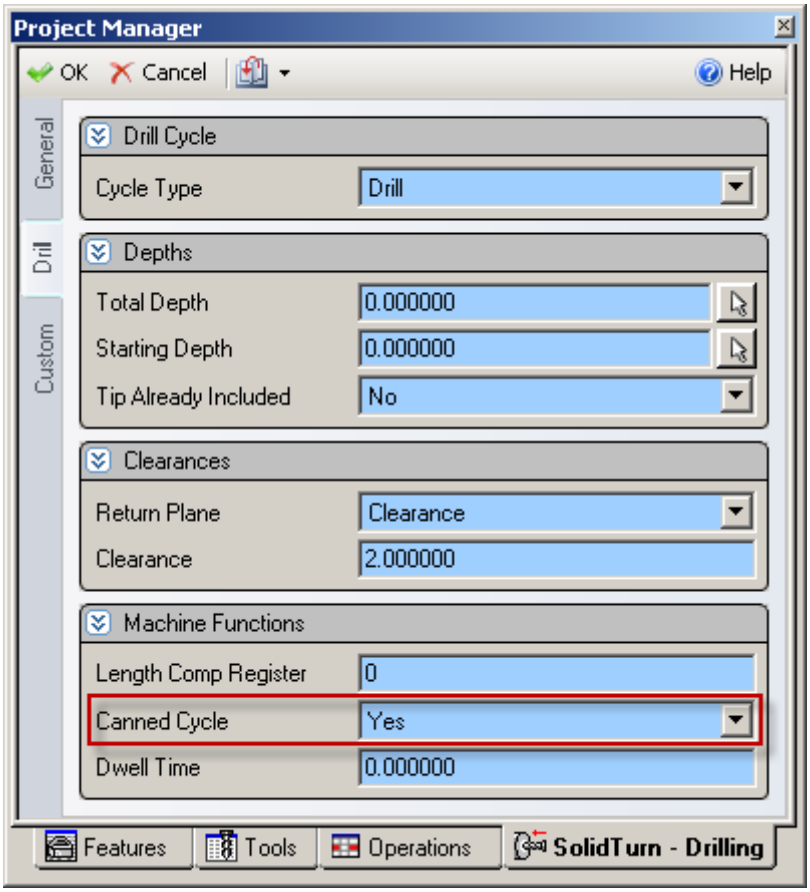
On the Drill tab for the SolidMillTurn Drilling cycle:



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:

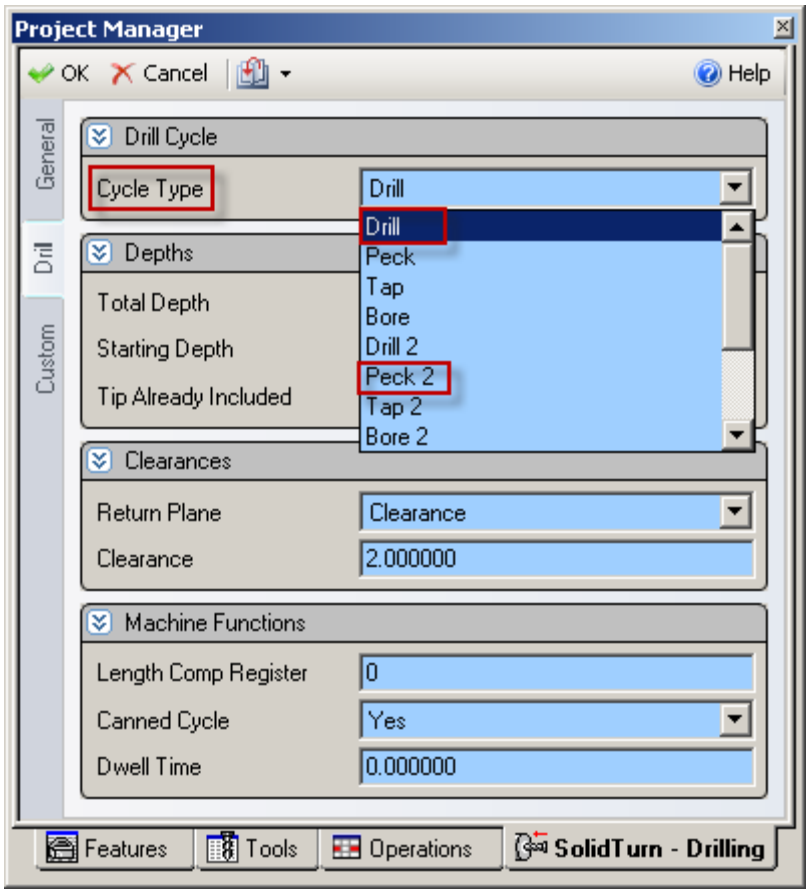
- The depth of cut per one infeed motion Q is an unsigned value, in radius for side drilling. Specify Q in mm, it will then be output in units of 0.001 mm without decimal point. For example, if you enter 2 for Q in ESPRIT, the output will be in the NC code Q2000.
- Enter the dwell P in s, it will then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

16.2. Deep Hole Drilling Cycle (G74)

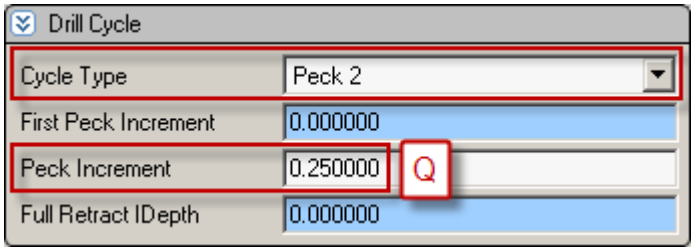
CAUTION !!! Use SolidTurn Drilling

G74 R(1)
G74 Z_ Q_ F_ ;

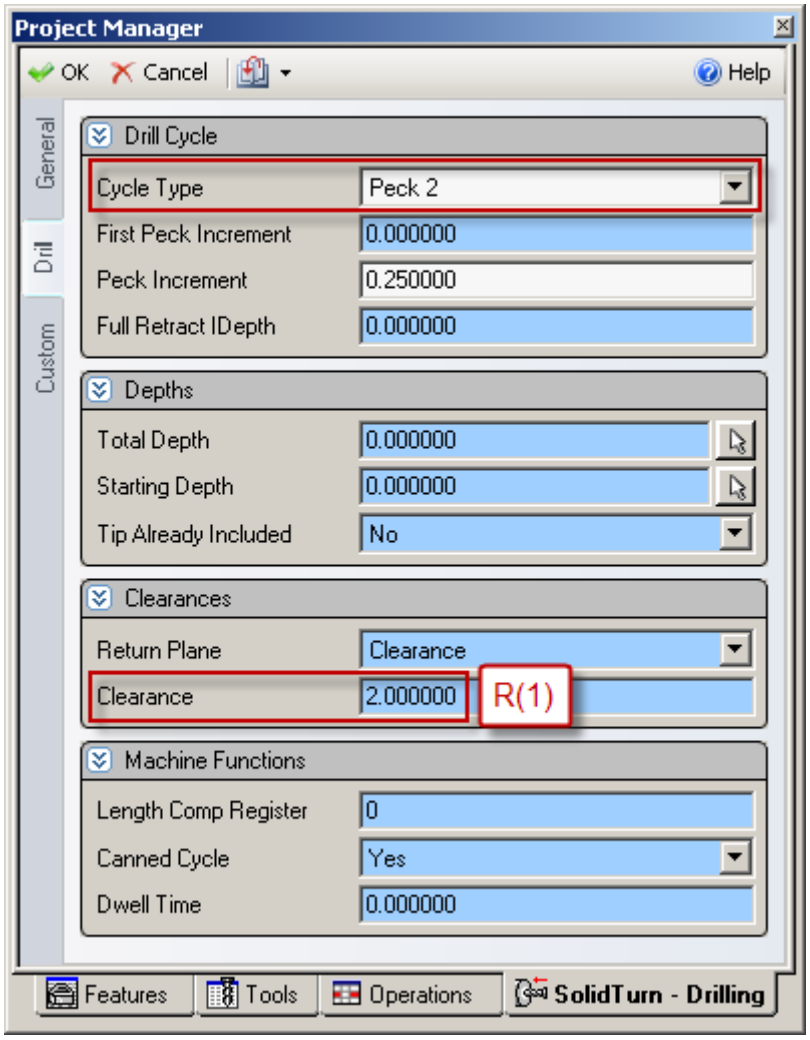
On the SolidTurn Drilling cycle, set the Cycle Type to Drill or Peck 2 to output your drilling cycle with G74.



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:



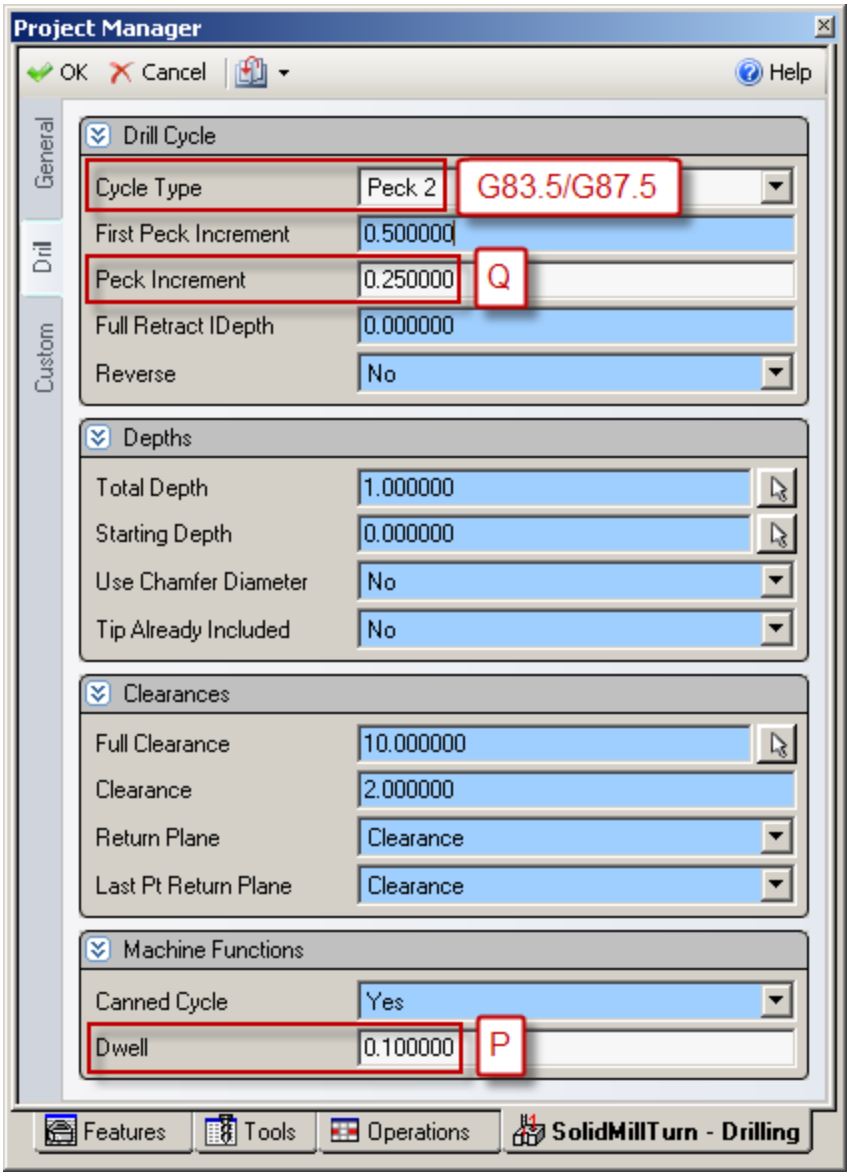
R(1) corresponds to return amount. It will be output as entered in ESPRIT when Peck 2 is selected. R0.0 will be output when Drill is selected.



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

CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

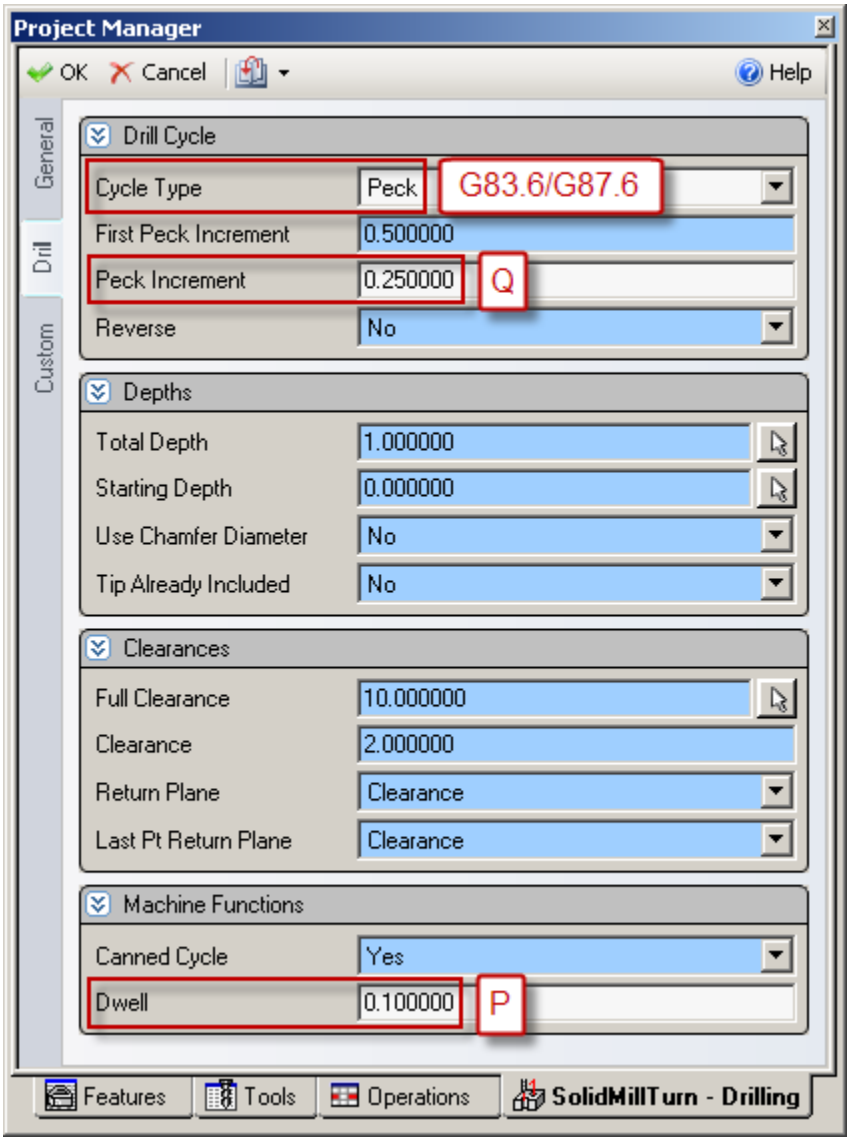
G83.5 X_ (Y_) C_ Z_ R_ Q_ P_ F_ ;
G87.5 Z_ (Y_) C_ X_ R_ Q_ P_ F_ ;



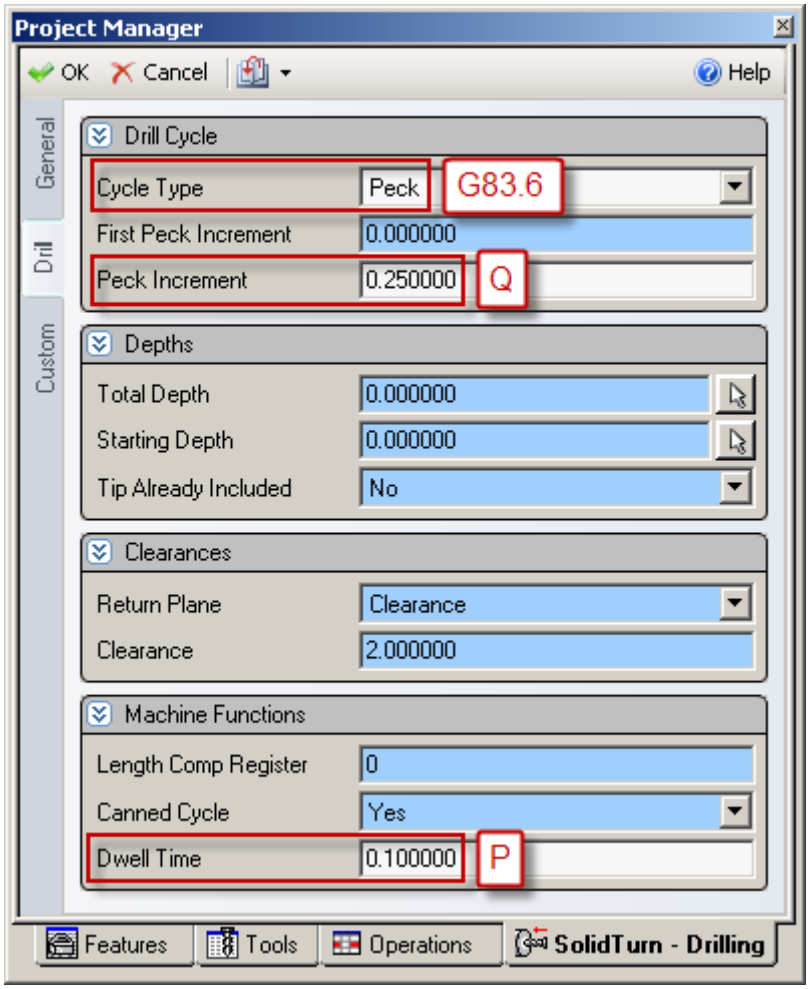
16.4. Face and Side Deep Hole Drilling Cycle (G83.6/G87.6)

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_ ;



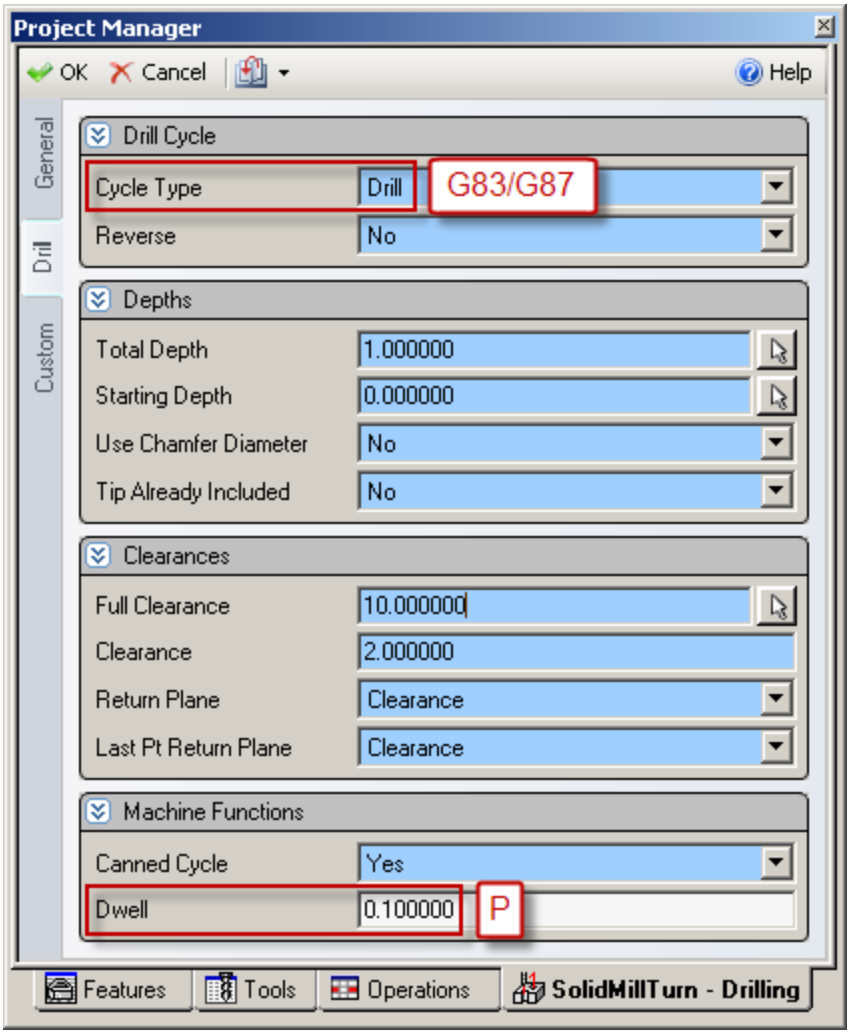
Settings for SolidTurn Drilling:



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

CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

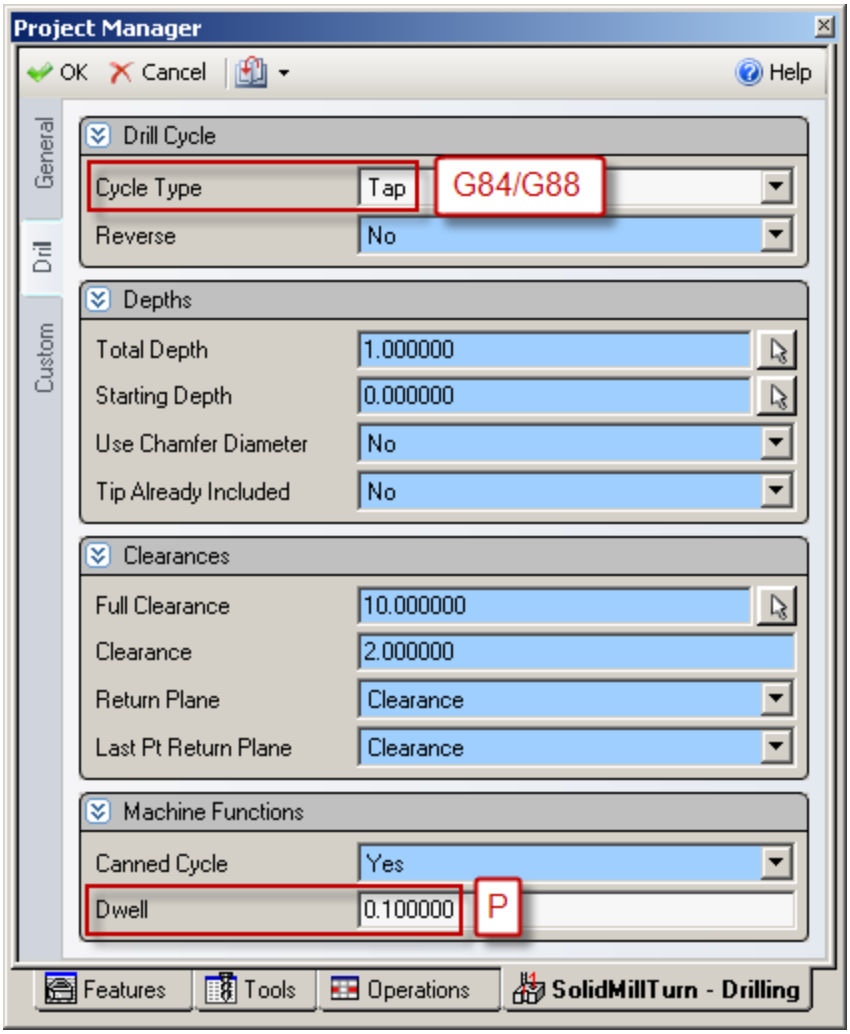
G83 X_ (Y_) C_ Z_ R_ P_ F_ ;
G87 Z_ (Y_) C_ X_ R_ P_ F_ ;



16.6. Face and Side Tapping Cycle (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_ ;

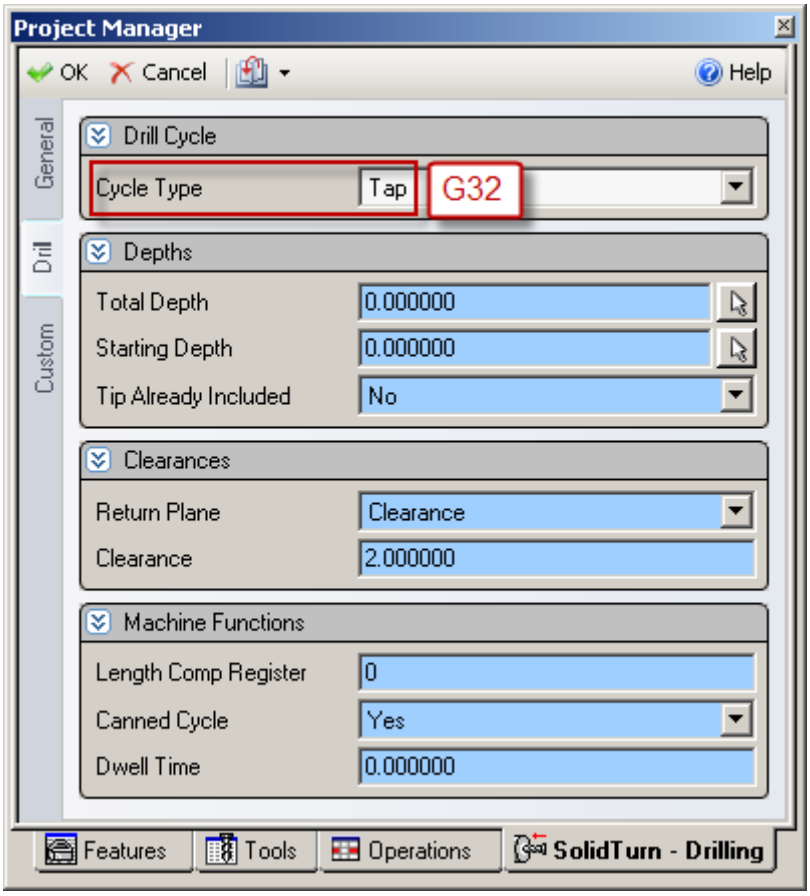


16.7. Tapping at Center of Spindle (G32)

CAUTION !!! Use SolidTurn Drilling

G32 Z_ F_ ;

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



16.8. Synchronized Tapping (M329)

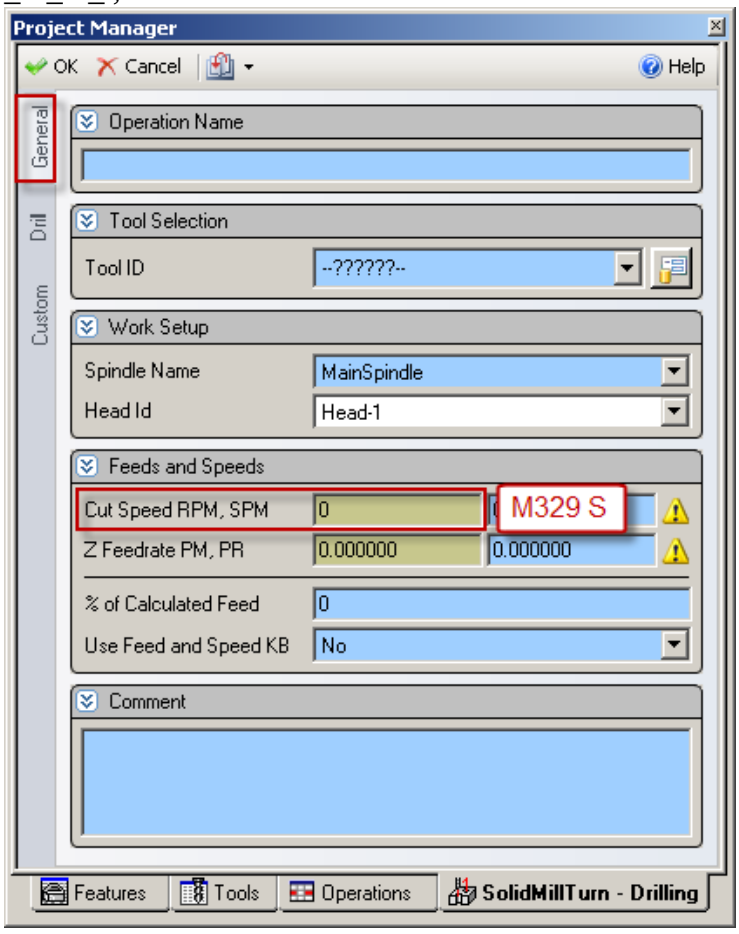
16.8.1. Limitations of Synchronized Tapping

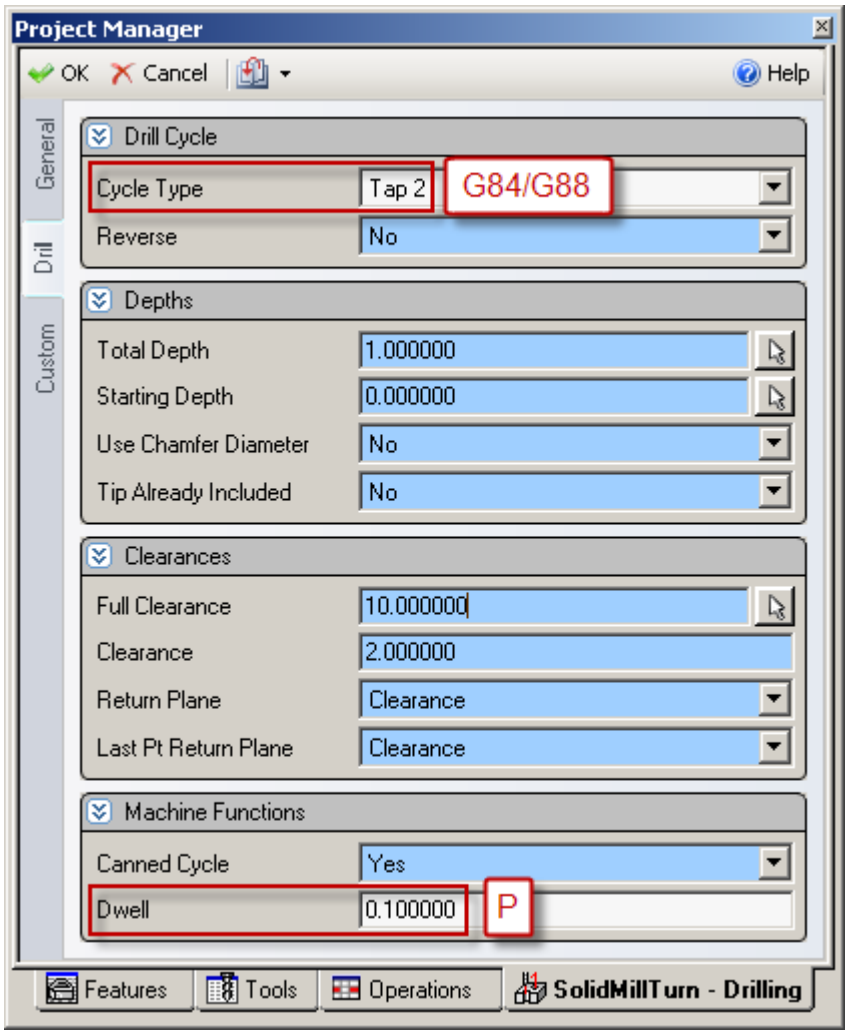
CAUTION !!! During the synchronized tapping cycle, the maximum speeds for the rotary tool is 3000 min^{-1}.
If the maximum speed for the rotary tool holder is less than the above, specify a speed that is less than the maximum speed for the holder.

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

CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

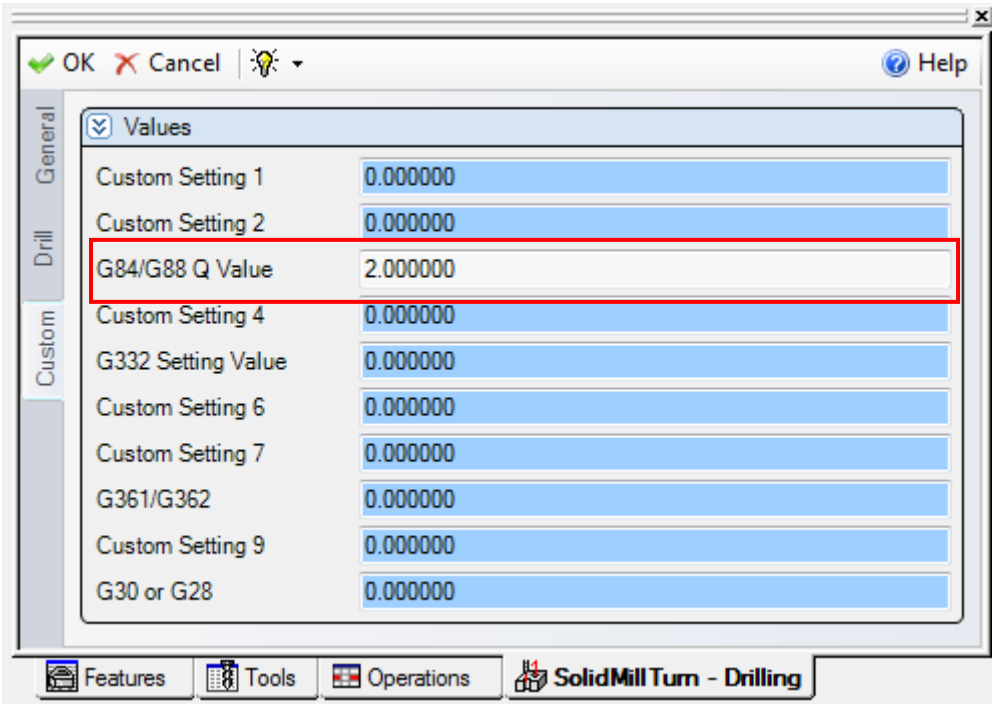
M329 S_ ;
G84 X_ (Y_) C_ Z_ R_ P_ F_ ;
M329 S_ ;
G88 Z_ (Y_) C_ X_ R_ P_ F_ ;




Project Manager
OK
Cancel
Help
General
Drill
Custom
Drill Cycle
Cycle Type
Tap 2
G84/G88
Reverse
No
Depths
Total Depth
1.000000
Starting Depth
0.000000
Use Chamfer Diameter
No
Tip Already Included
No
Clearances
Full Clearance
10.000000
Clearance
2.000000
Return Plane
Clearance
Last Pt Return Plane
Clearance
Machine Functions
Canned Cycle
Yes
Dwell
0.100000
P
Features
Tools
Operations
SolidMillTurn - Drilling

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

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



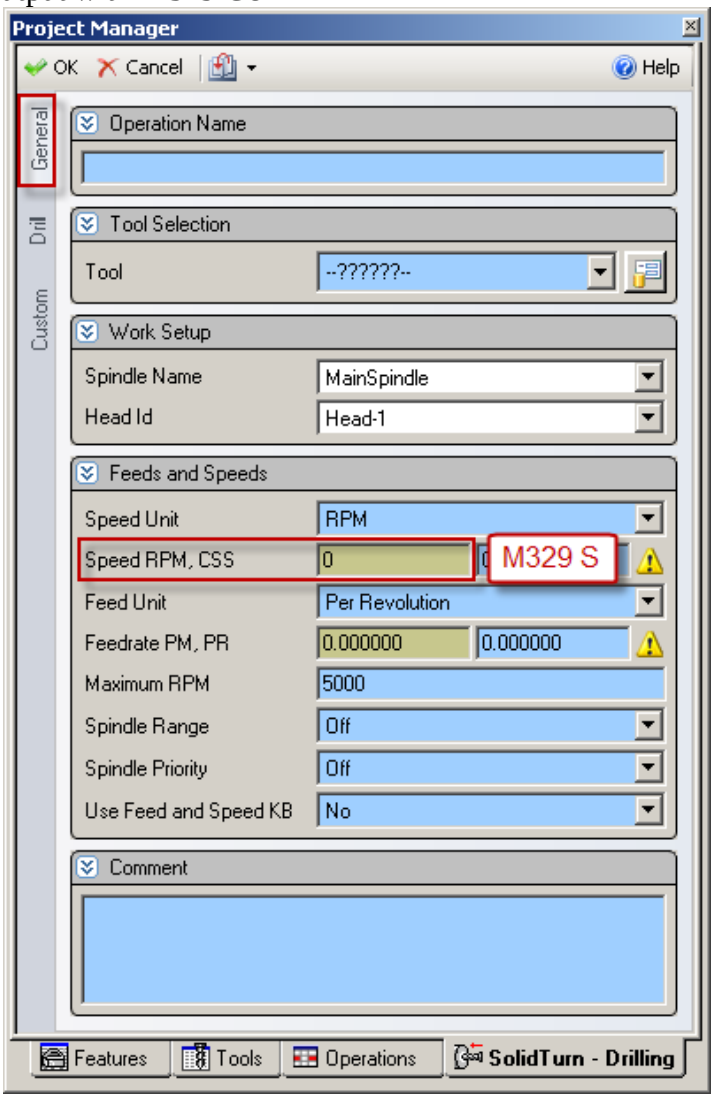
16.8.4. Spindle Synchronized Tapping (M329 G84) (Option)

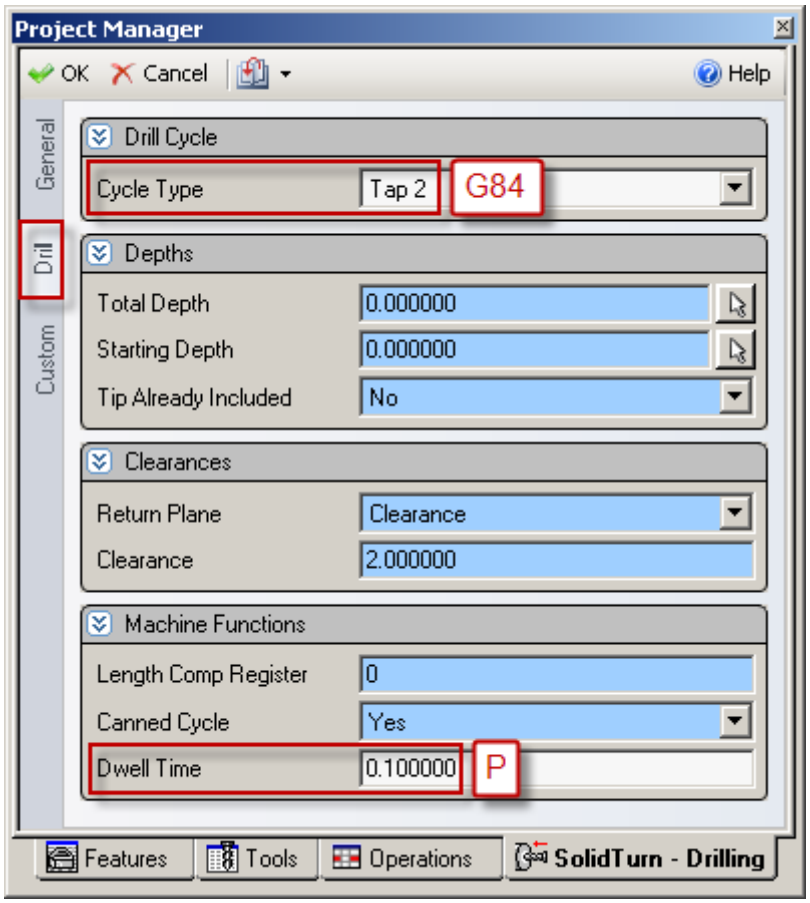
CAUTION !!! Use SolidTurn Drilling

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.

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

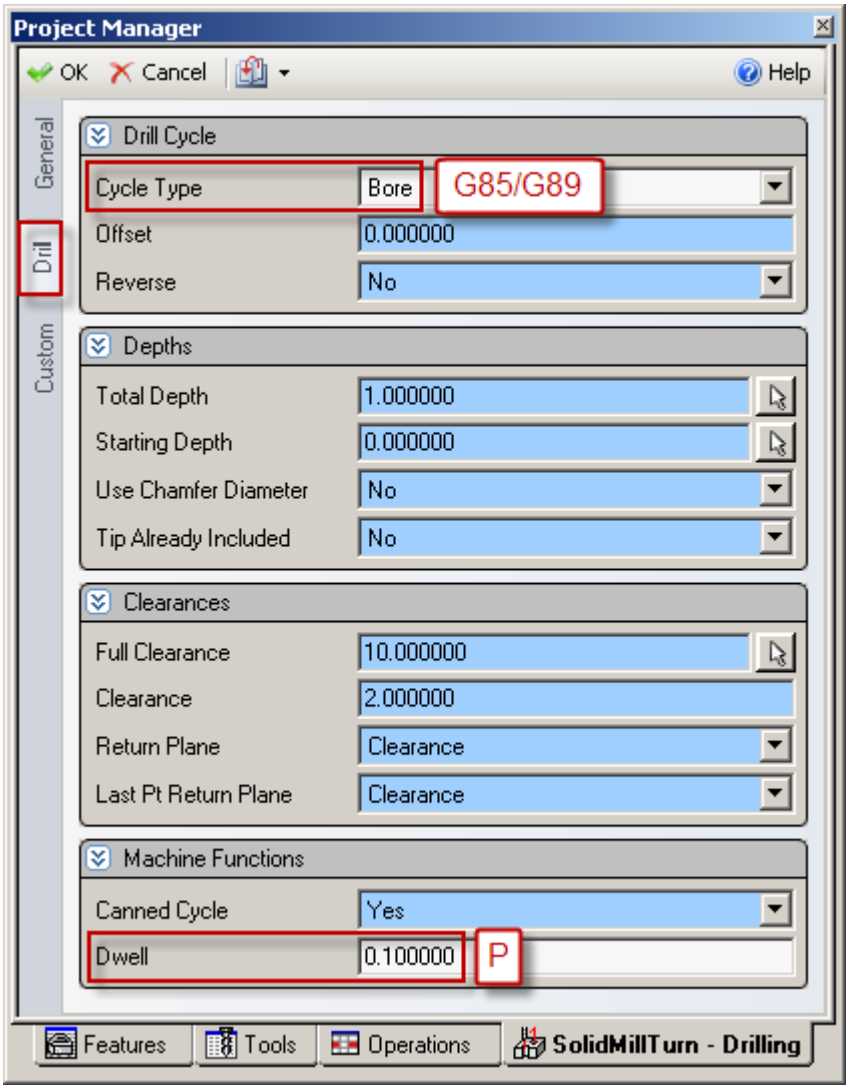



Project Manager
OK
Cancel
Help
General
Drill
Custom
Drill Cycle
Cycle Type
Tap 2
G84
Depths
Total Depth
0.000000
Starting Depth
0.000000
Tip Already Included
No
Clearances
Return Plane
Clearance
Clearance
2.000000
Machine Functions
Length Comp Register
0
Canned Cycle
Yes
Dwell Time
0.100000
P
Features
Tools
Operations
SolidTurn - Drilling

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

CAUTION !!! Use SolidMillTurn Drilling or SolidMillTurn Wrap Drilling

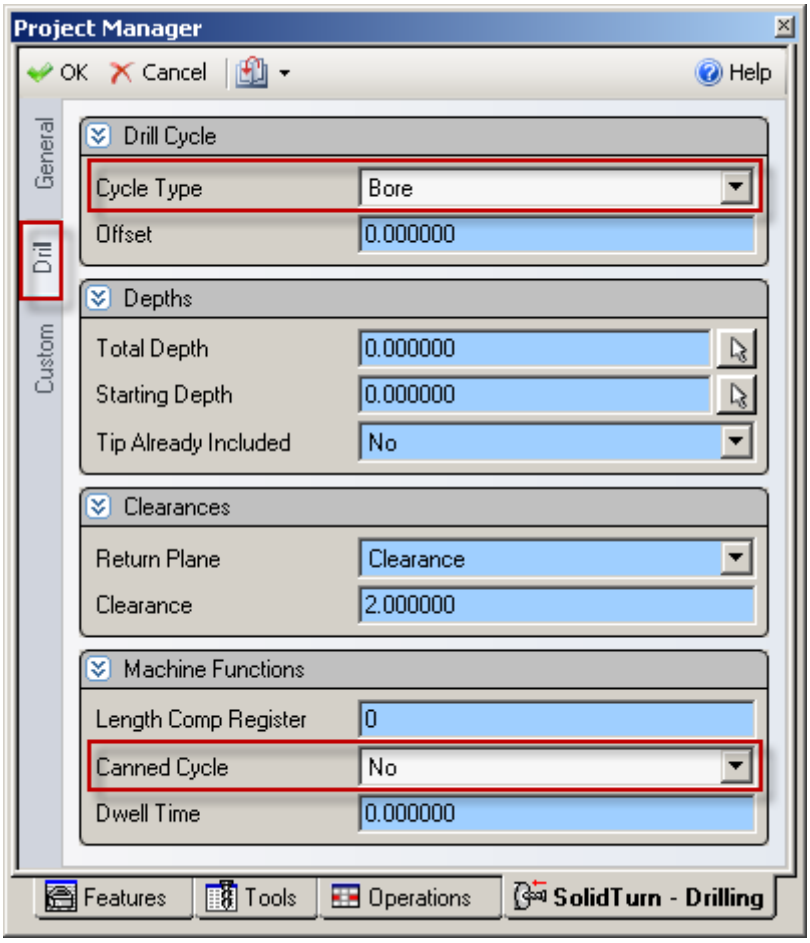
G85 X_ (Y_) C_ Z_ R_ P_ F_ ;
G89 Z_ (Y_) C_ X_ R_ P_ F_ ;



16.8.6. Boring in Turning Mode

CAUTION !!! Use SolidTurn Drilling

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



16.9. Spindle/Rotary Tool Spindle Simultaneous Operation Mode

CAUTION !!! Use SolidTurn Drilling

M90 S_ ;
M91;
M290 S_ ;
M291;

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

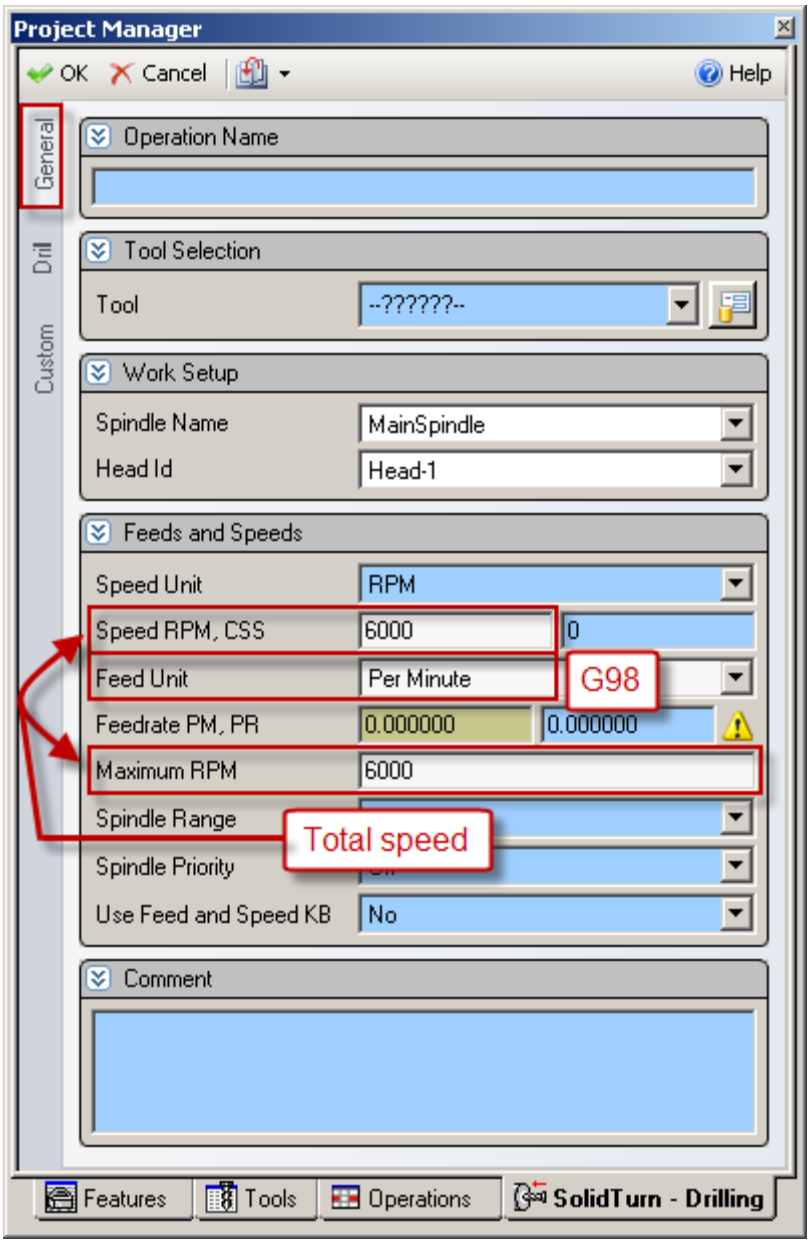
On the Custom tab of the SolidTurn Drilling operation, set the Simultaneous Operation (Custom Setting 6) field to 90 to trigger the output of the M90 (or M290 if machining on the spindle 2 side) command.

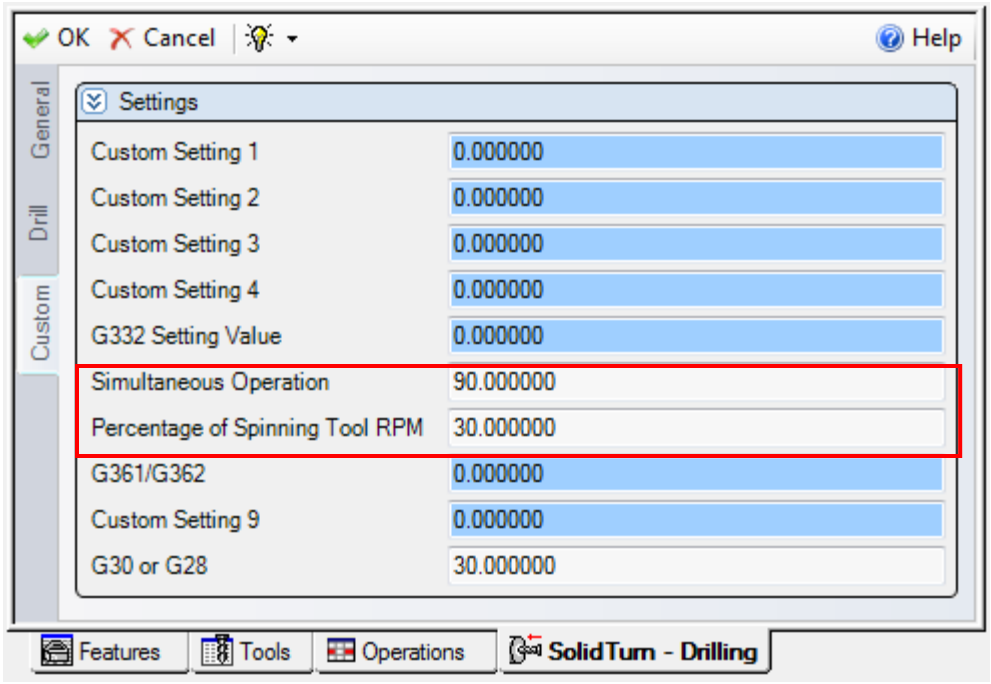
On the General tab, set in Speed RPM to the desired total speed, which mean speed of the turning spindle plus speed of the rotary tool.
You can then specify with the Percentage of Spinning Tool RPM (Custom Setting 7) field the percentage of the total speed you want to apply to the rotary tool.

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

Notes:

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


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
6000
0
Feed Unit
Per Minute
G98
Feedrate PM, PR
0.000000
0.000000
Maximum RPM
6000
Spindle Range
Total speed
Spindle Priority
Use Feed and Speed KB
No
Comment
Features
Tools
Operations
SolidTurn - Drilling


OK
Cancel
Help
General
Drill
Custom
Settings
Custom Setting 1
0.000000
Custom Setting 2
0.000000
Custom Setting 3
0.000000
Custom Setting 4
0.000000
G332 Setting Value
0.000000
Simultaneous Operation
90.000000
Percentage of Spinning Tool RPM
30.000000
G361/G362
0.000000
Custom Setting 9
0.000000
G30 or G28
30.000000
Features
Tools
Operations
SolidTurn - Drilling

17. 3D Coordinate Conversion (G68.1)

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

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

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

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

18. 4-Axis Wrap Milling Cycles with Interpolation

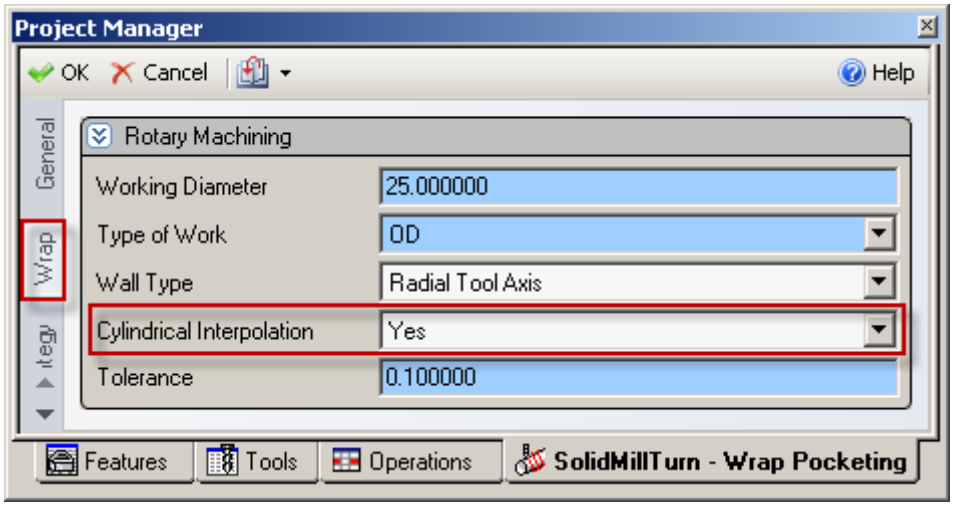
18.1. Introduction

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

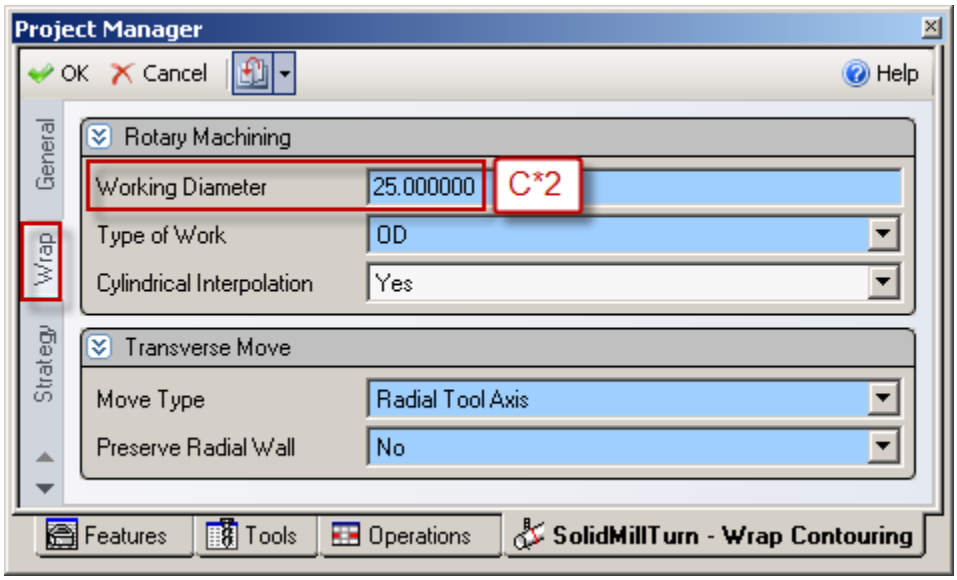
18.2. Cylindrical Interpolation

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

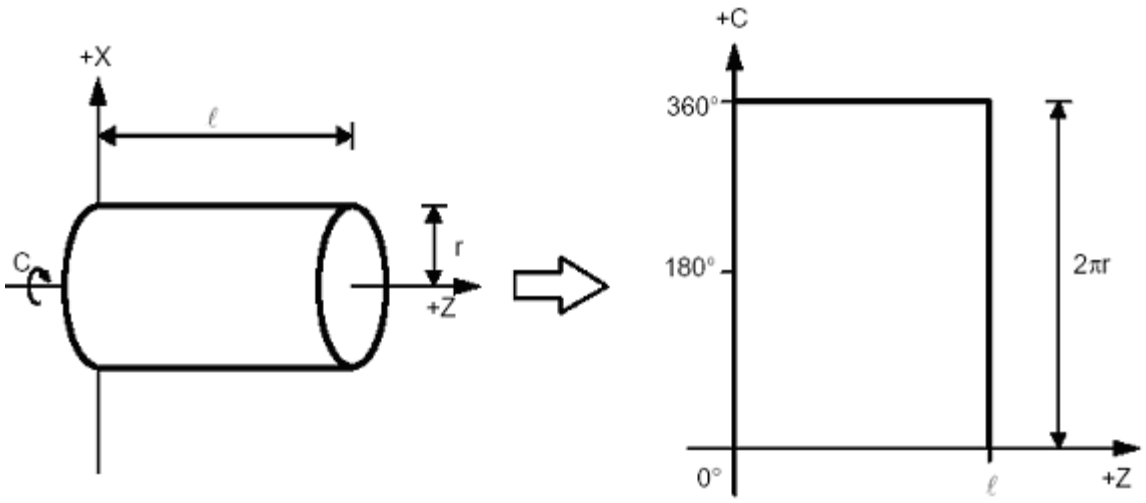
On the Wrap tab of the operation pages, set Cylindrical Interpolation to Yes to output 4-axis wrap milling operations with cylindrical interpolation.



It will be turned on in the NC code by G107 C_ and turned off by G107 C0.
C corresponds to the radius of the workpiece (groove bottom). It can be specified (as a diameter value) on the Wrap tab in the Working Diameter field of the operation pages:



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

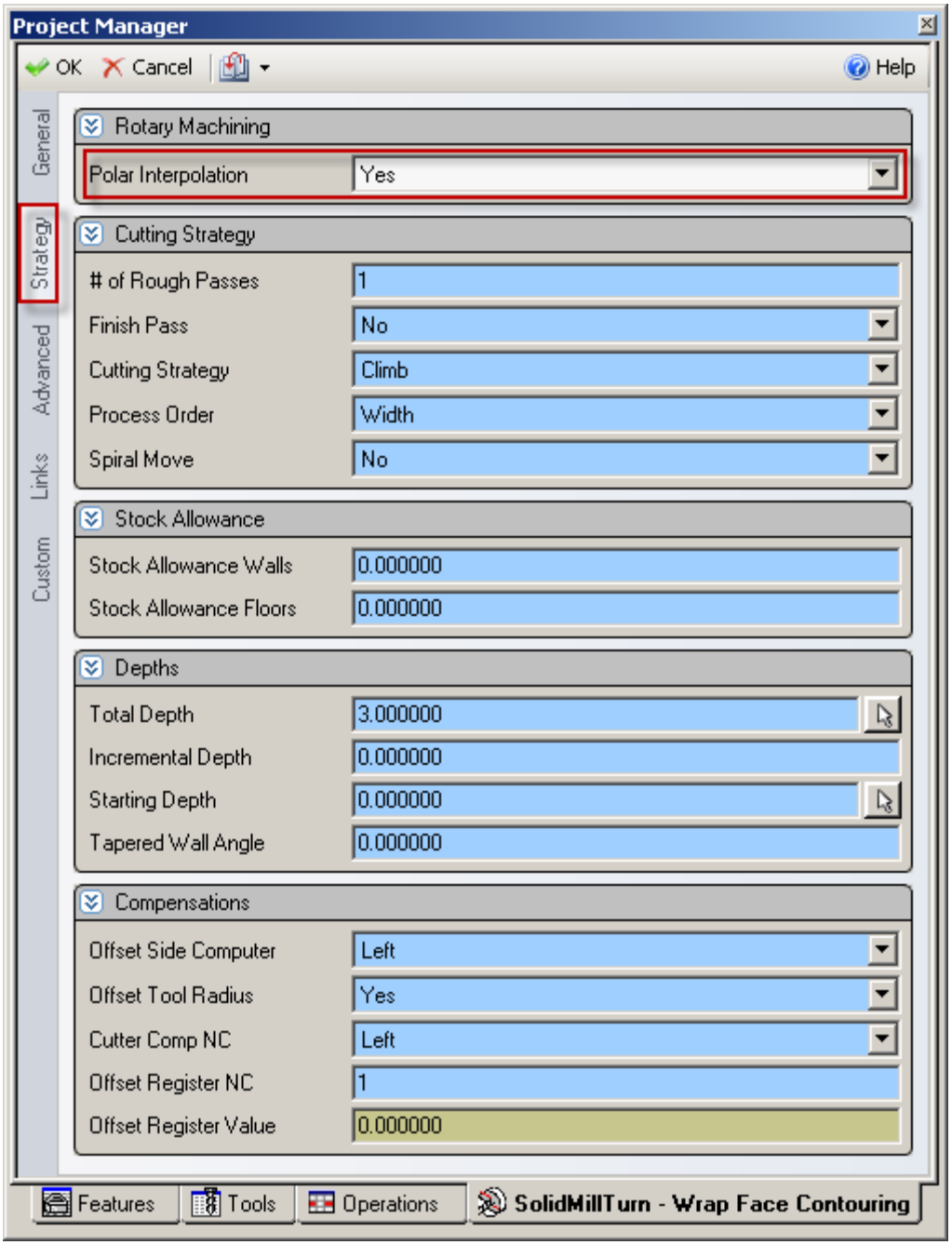


Using cylindrical interpolation can shorten the program length because ESPRIT outputs the circular motions in the cylindrical plane.
Without cylindrical interpolation, circular motions in the cylindrical plane are approximated by small linear moves (quality varies with the tolerance of the operation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

18.3. Polar Interpolation

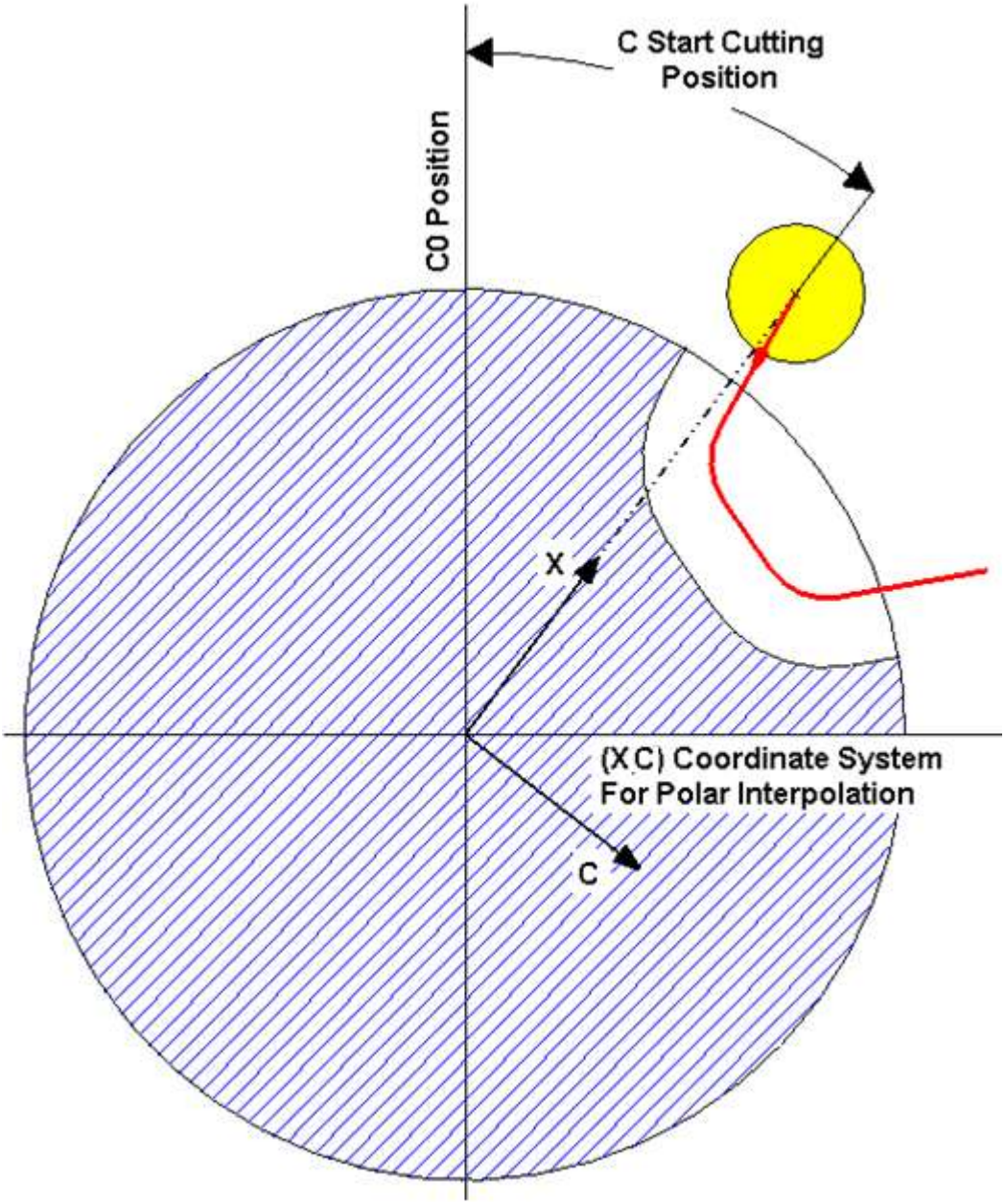
The polar coordinate interpolation can be turned on the SolidMillTurn Wrap Face Pocketing and Wrap Face Contouring pages.

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



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

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



19. Tailstock

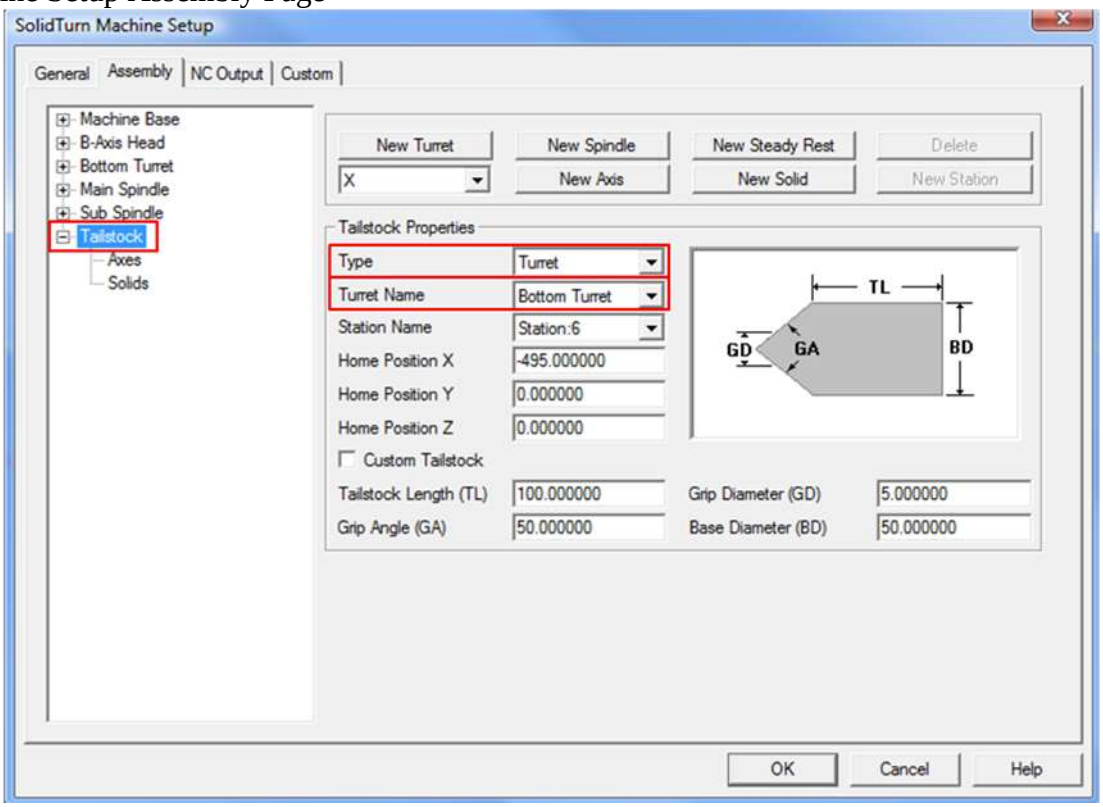
19.1. Live Center on Lower Turret

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

19.1.1. Defining Tailstock in Machine Setup

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

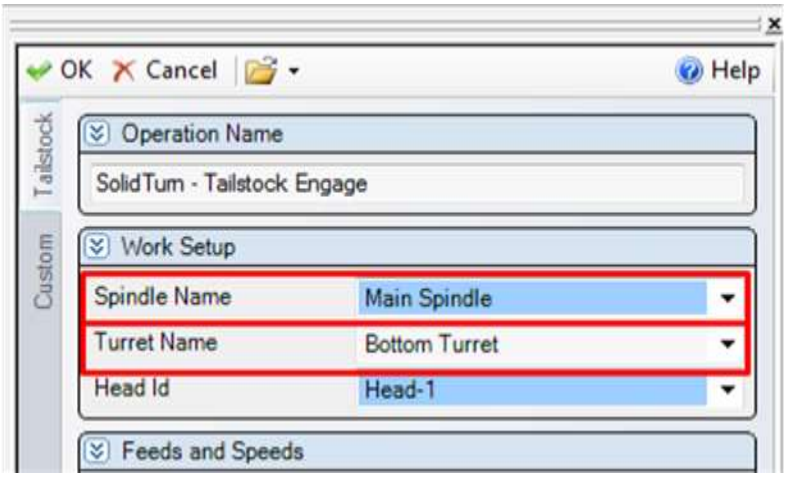
Machine Setup Assembly Page



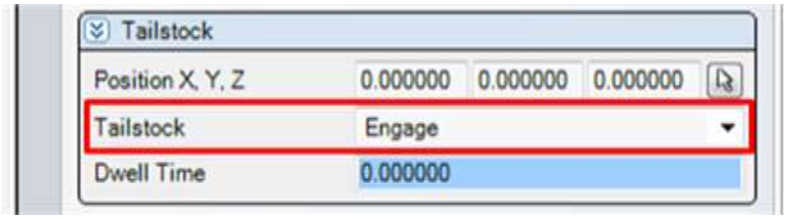
19.1.2. Creating Tailstock Operation

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

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



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



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

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

19.2. Second Live Center on Lower Turret

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

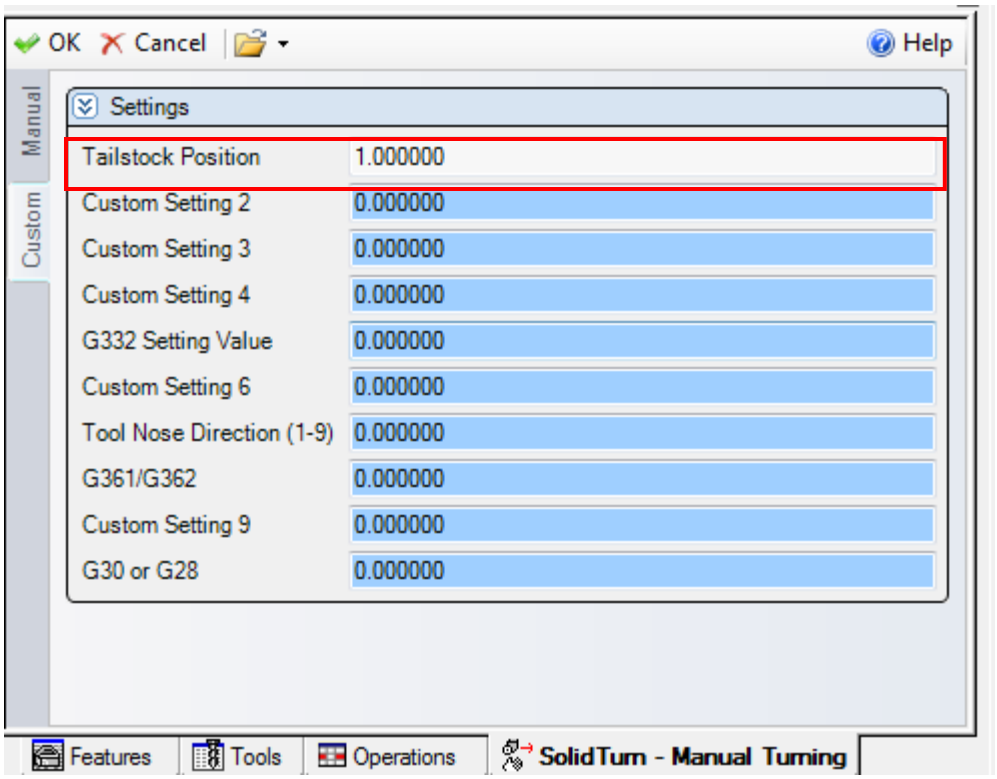
We have created a method by which you can use the manual turning operation to program a second center on the lower turret by setting the Tail Stock Position field (Custom Setting 1) to either a 1 (engage) or -1 (disengage).

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

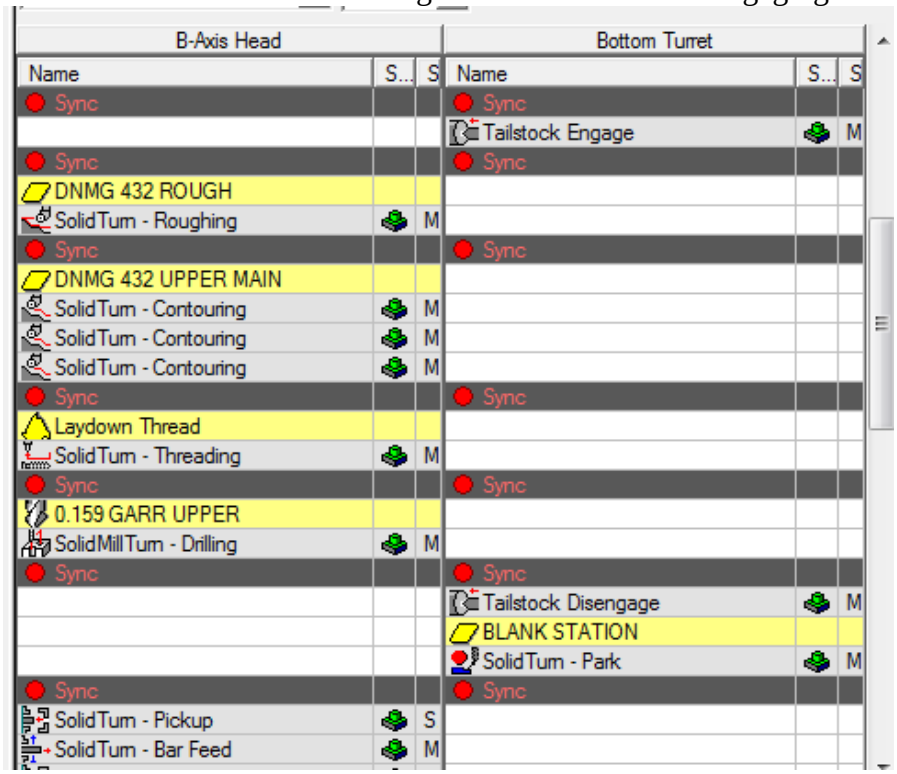
It will be necessary to create a tool that you can use as the tail sock that will resemble the center you will use. You can even use a custom tool if necessary, but often a drill works well
The spindle speed settings will not be used but the feedrate will be used in the same manner the feedrate is used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

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

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



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



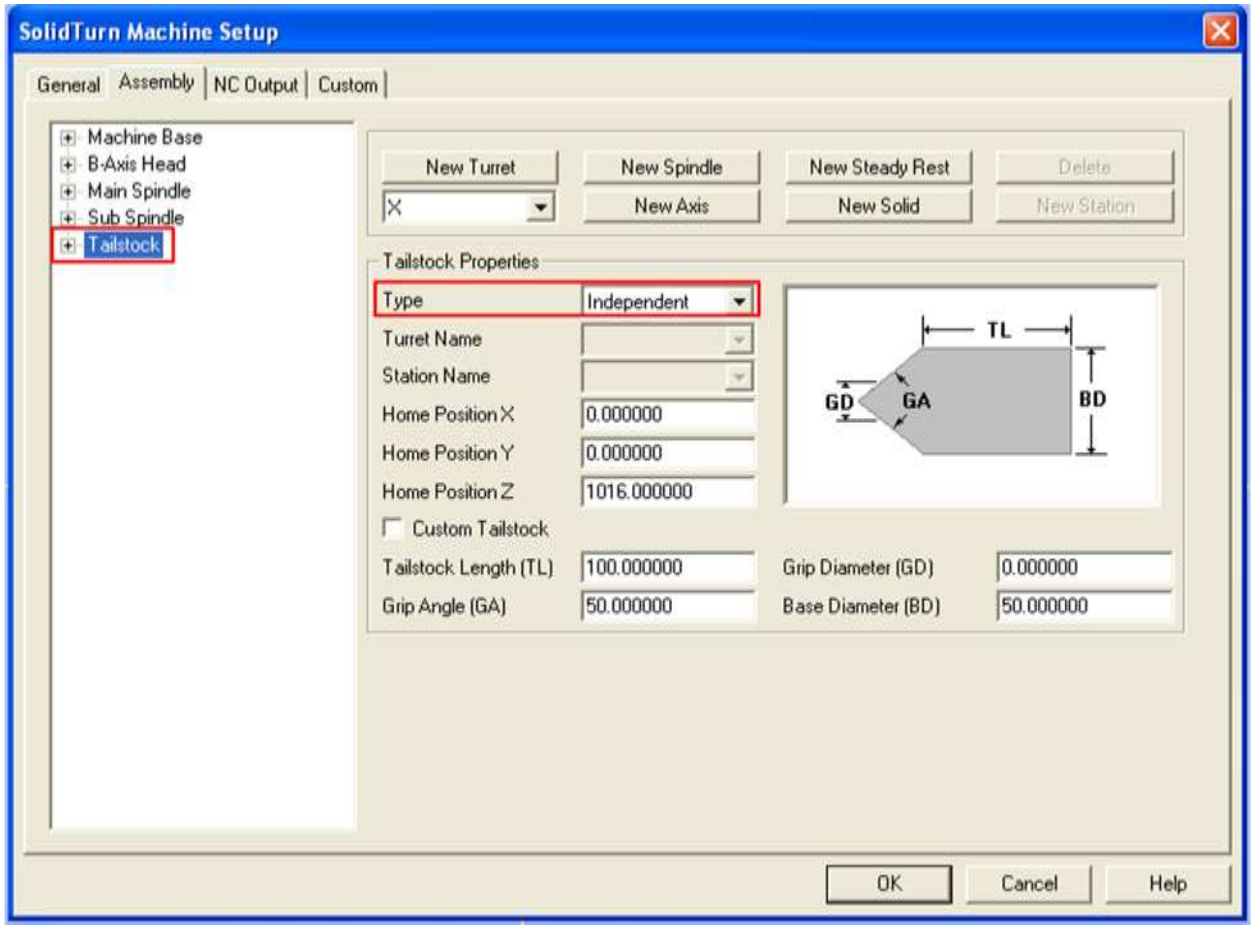
19.3. Independent Tail Stock

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

19.3.1. Defining Tailstock in Machine Setup

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

Machine Setup Assembly Page

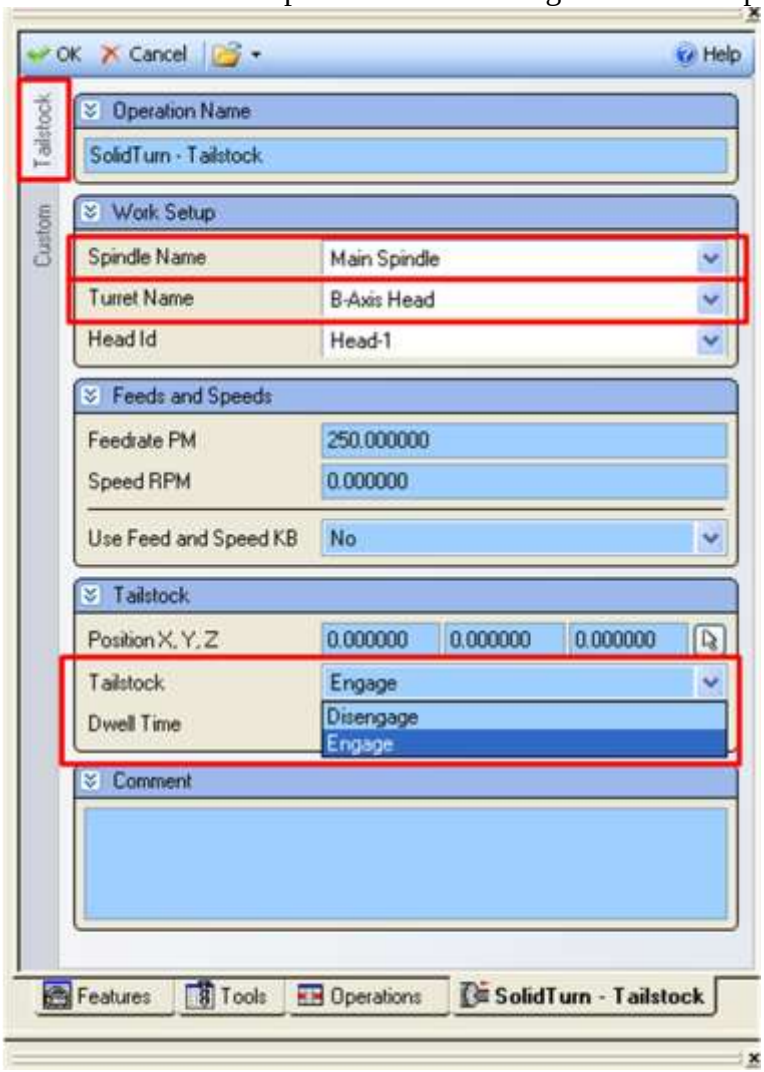


19.3.2. Creating Tailstock Operation

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

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

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



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

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

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

19.4. Live Center on Sub Spindle

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

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

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

20. Steady Rest

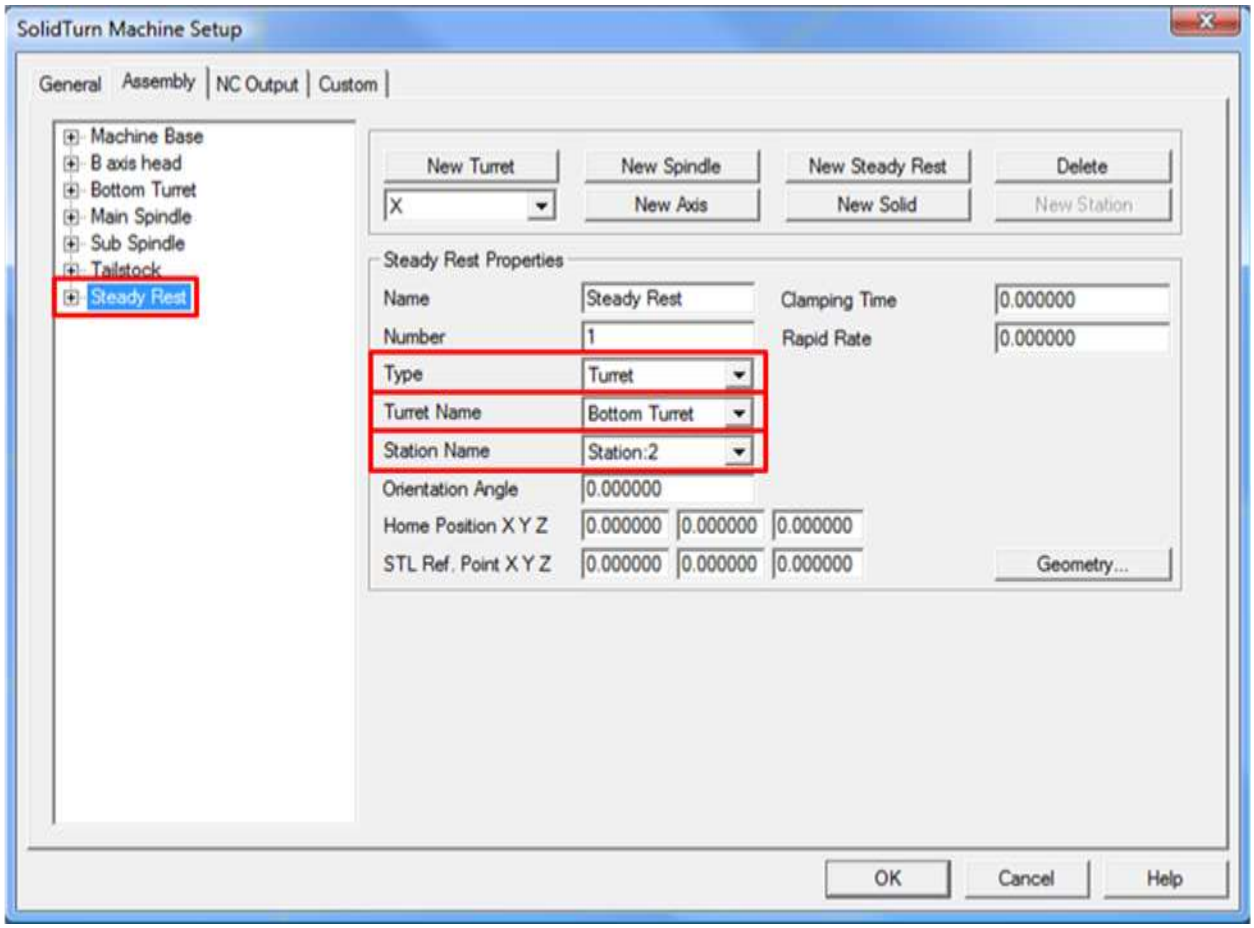
20.1. Lower Turret Steady Rest

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

20.1.1. Defining Steady Rest in Machine Setup

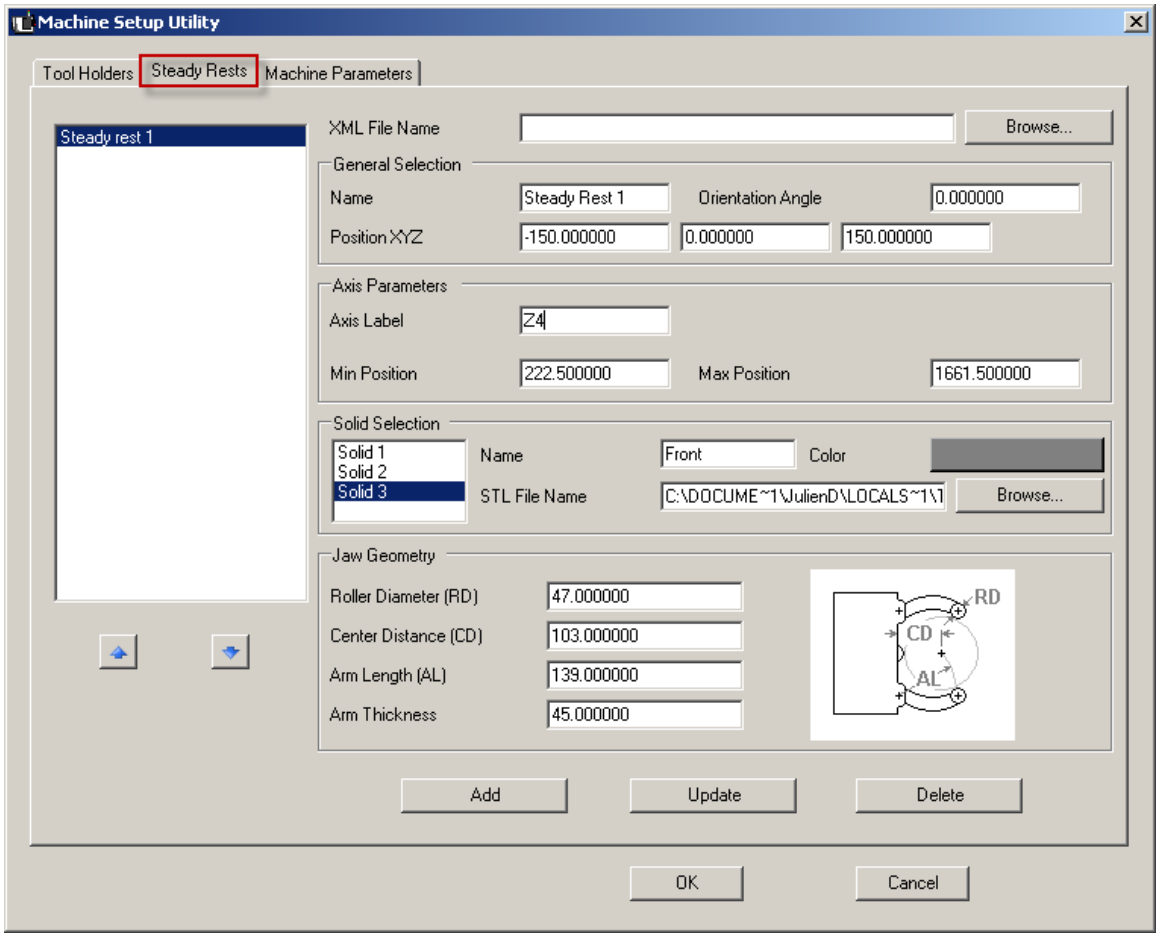
Open the Machine Setup page and navigate to the Assembly tab. Click on Steady Rest button to create a new Steady Rest. A new Steady Rest will be created on the left side of the page.

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



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

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

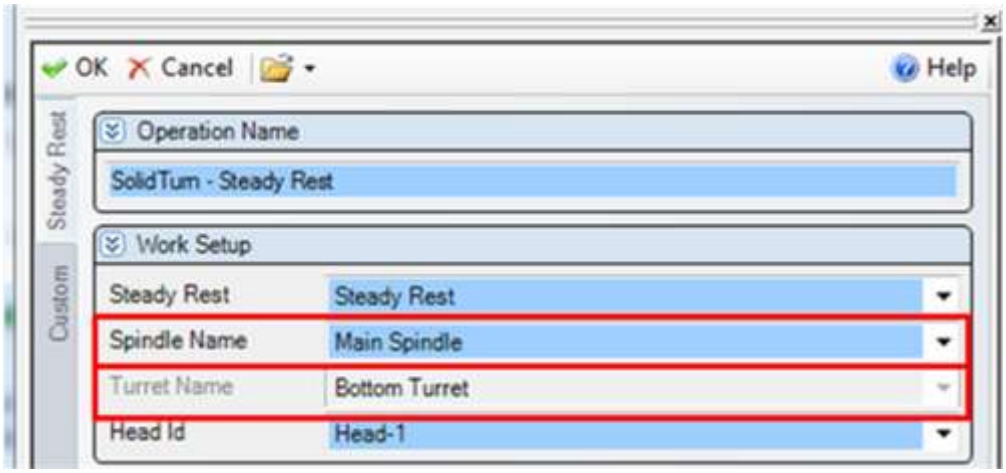


Note: Please refer to your Machine Setup Utility User Manual to have detailed instructions on how to create the steady rest with this utility.

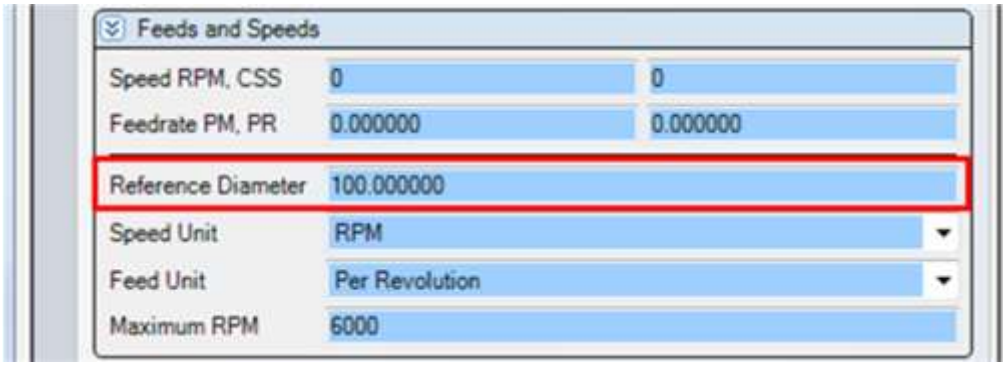
20.1.3. Creating Steady Rest Operation

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

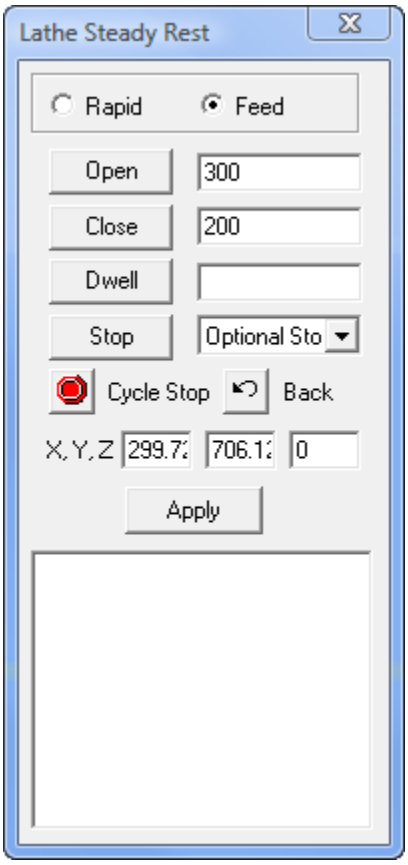
In the Steady Rest tab, select the spindle you would like the steady rest to be located on by selecting either the Main or Sub Spindle in Spindle Name. Then select Bottom Turret for Turret Name.



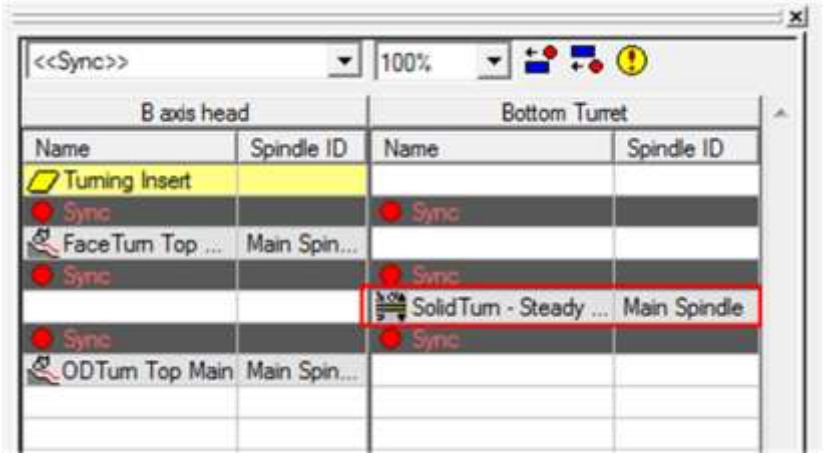
Input the Reference Diameter based on the Diameter of your part at the steady rest location.



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



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



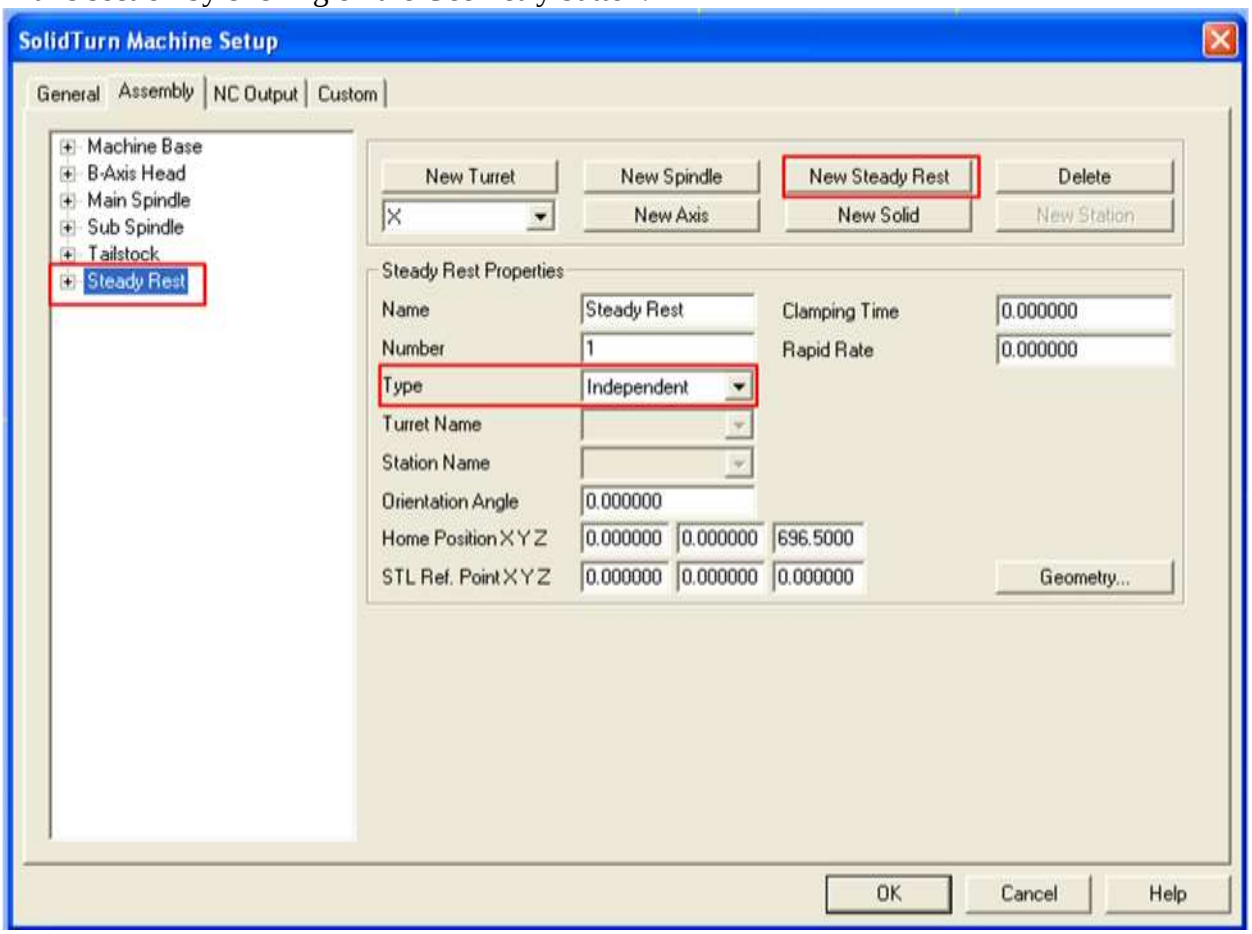
20.2. Independent Steady Rest

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

20.2.1. Defining Steady Rest in Machine Setup

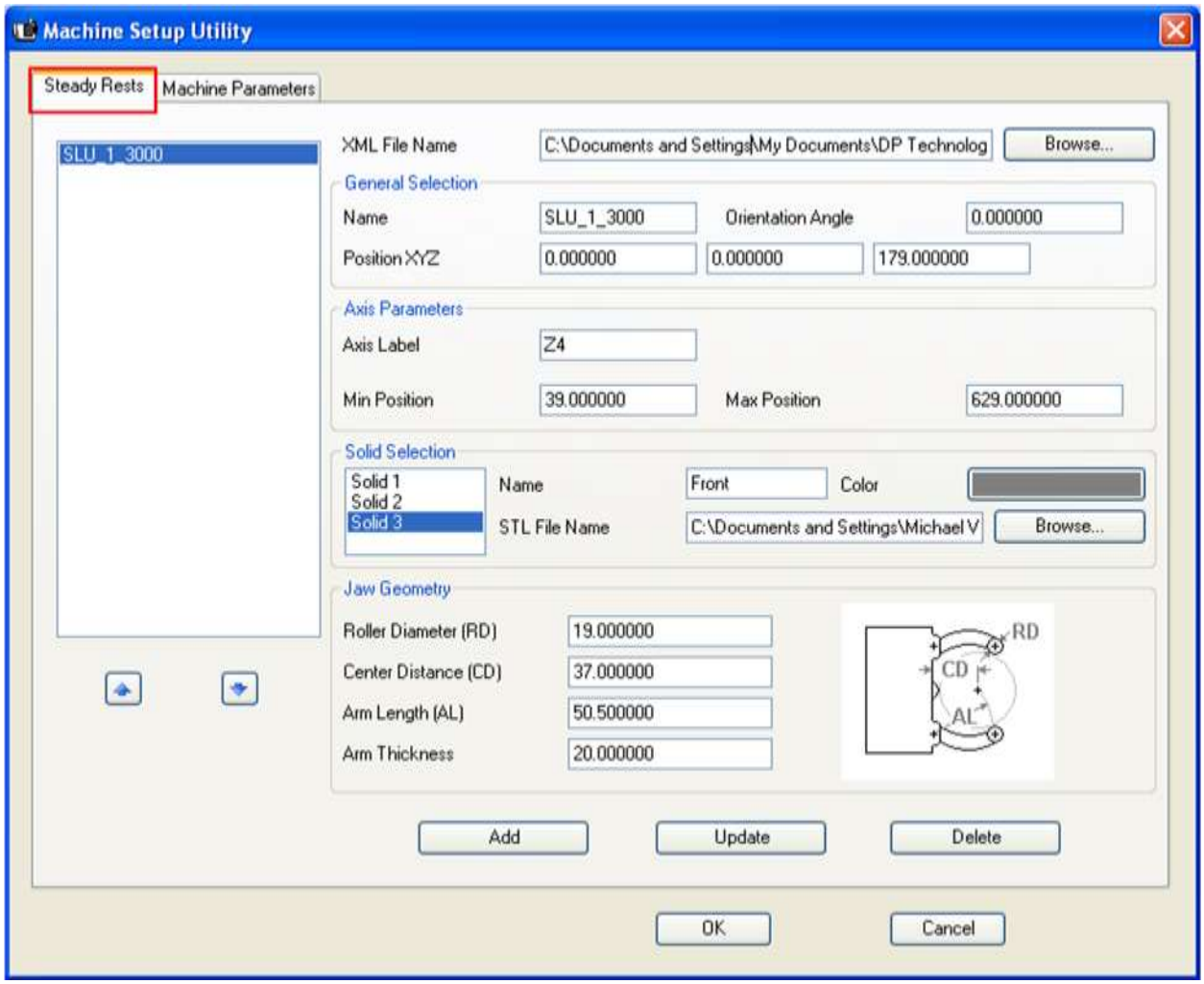
Open the **Machine Setup** page and navigate to the **Assembly** tab. Click on **Steady Rest** button to create a new Steady Rest. A new Steady Rest will be created on the left side of the page.

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



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

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



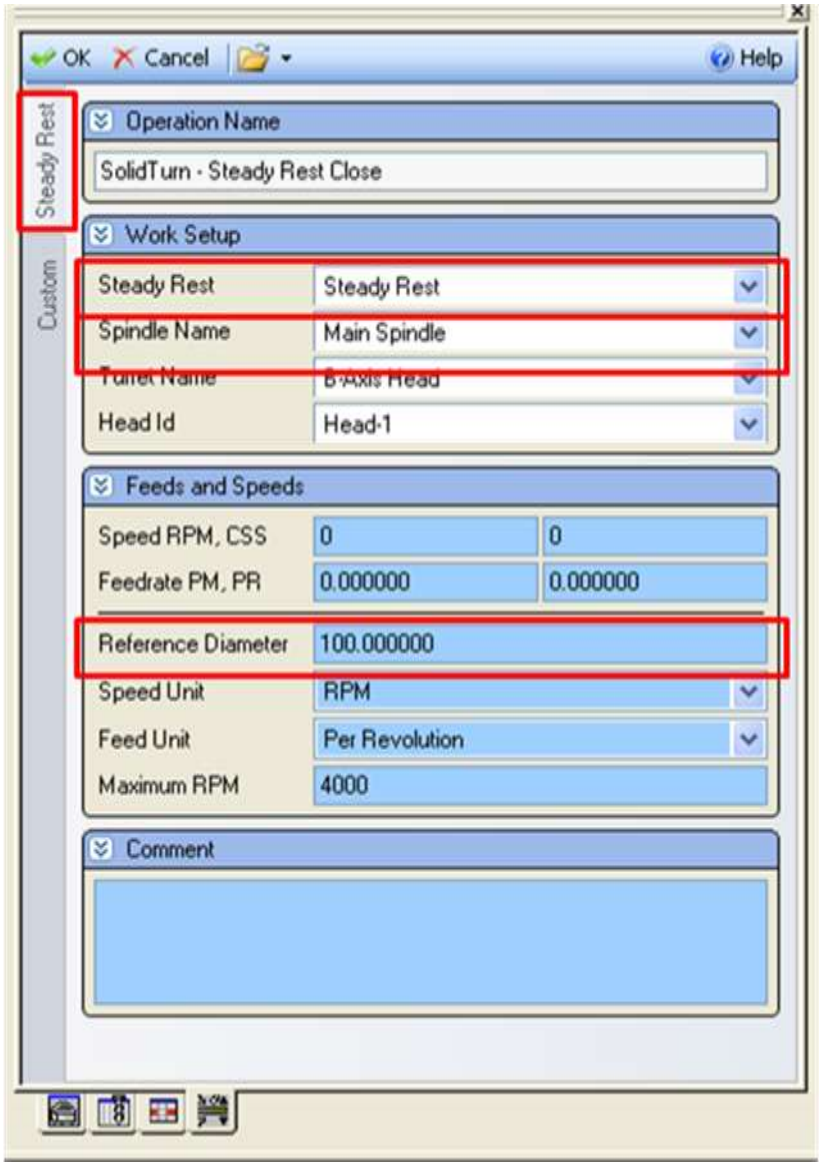
Note: Please refer to your Machine Setup Utility User Manual to have detailed instructions on how to create the steady rest with this utility.

20.2.3. Creating Steady Rest Operation

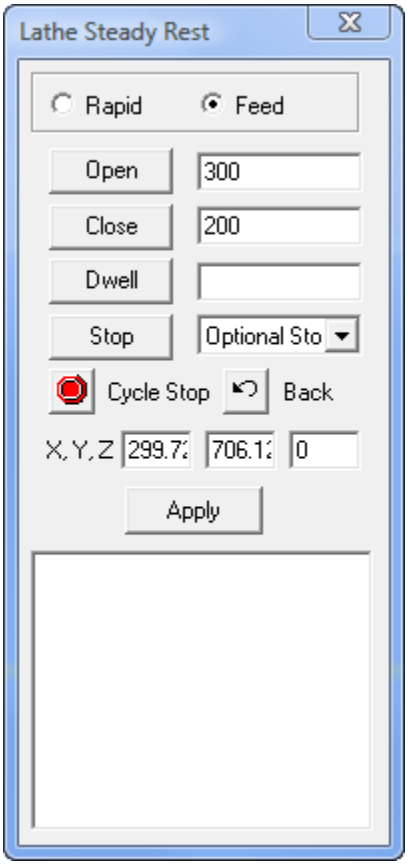
First navigate to the Solid Turn (1) tool bar and then select Steady Rest (2)

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

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

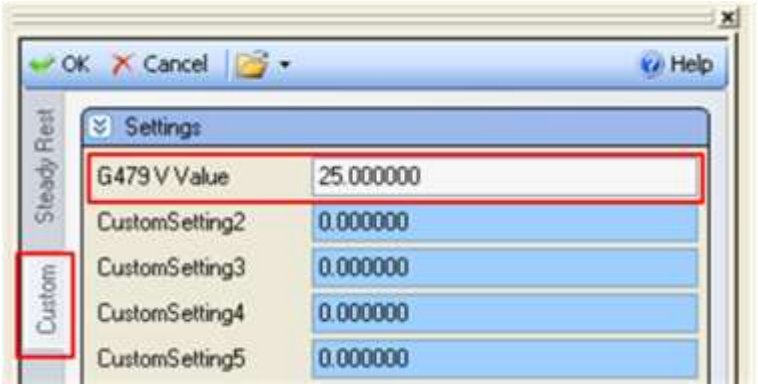


When you click OK, a new window will pop up called “Lathe Steady Rest.” This window will primarily be used to create an accurate simulation in ESPRIT. The important aspect of this with regard to the NC code is the order of operations. If you want the steady rest to open, move to a new location, and then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



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

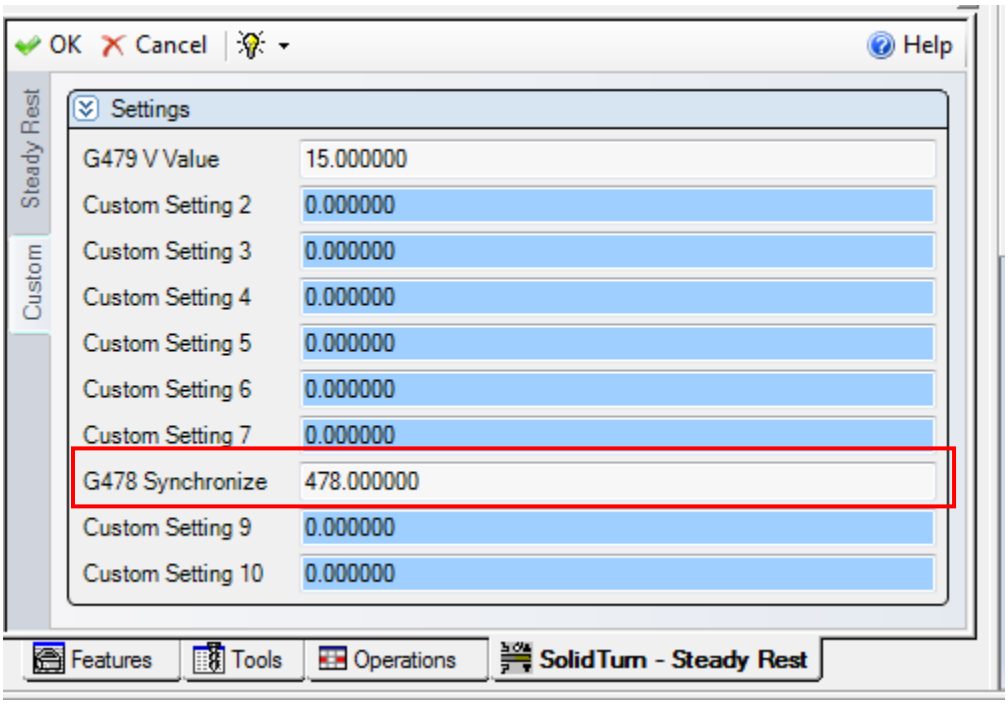
Custom Setting G479 V_ input:



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

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

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



.

21. Special considerations when Programming an NT1000

21.1. Introduction

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

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

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

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

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

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

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

MORI SEIKI

出力日時 2012/05/16 7:26:21
Page 1/ 1

管理文書

管理No CE216-526A-03

設計通知	設計通知No	親番	承認	承認	承認
	MD165125	10H00078		開発部	開発部
多品一葉	役務申請No			2012/05/15	2012/05/15
				山中 孝	島ノ江 洋司

題目/TOPIC	NTX1000/2000：標準マクロエグゼキュータ改版	□マニュアル改訂有
	STANDARD MACRO EXECUTOR MODIFIED:NTX1000/2000	

部署	担当者	変更前設計通知No
電装NT課 NTX設計チーム	桐山 景一	

採用時期	配付先
2009年07月機より	千葉/組立/ 千葉/千品/ 千葉/立加/ その他//千葉CAM課 上原MA その他//北海道MAPPS TM

機種	機番、採用時期
NTX1000 NTX2000	

初品チェック	依頼伝票No
□素材 □加工 □組立 □部品	仕事分類 設計変更

内容/CONTENTS

NTX1000/2000 31iB対応標準マクロエグゼを改版致します。(I98755A17→18)
変更内容：NTX1000におけるミーリング工具＆3次元座標変換中＆サブ主軸側加工＆D2369．0＝1のときの工具補正指令を修正します。2012年5月25日出荷機より対応です。

Standard macro executor for NTX1000/2000 31iB
Contents: Modification of tool offset command for milling cutting process on 3-dimension mode & sub-spindle side & D2369.0=1 for NTX1000.
Applied: From 25 May. 2012 for NTX1000

原因/CAUSE

NTX1000におけるミーリング工具＆3次元座標変換中＆サブ主軸側加工＆D2369．0＝1のときの工具補正指令を修正します。

Modification of tool offset command for milling cutting process on 3-dimension mode & sub-spindle side & D2369.0=1. for NTX1000.

作業指示/PROCESS DIRECTIVE

手配及び発送指示/LOGISTICS DISPATCHING

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

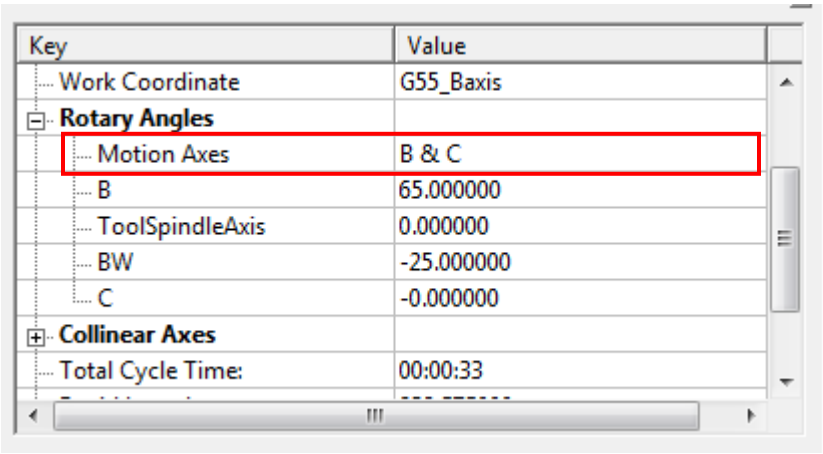
Since the sub spindle (A axis) can be moved to different locations when machining on the NT1000 the post is setup so that program zero for the sub spindle MUST be set while the A axis is at the G330 position (A0). The G474 command is used to shift the Z values by the amount the A axis is moved from A0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

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

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

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

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



The options are as follows:

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

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

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

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

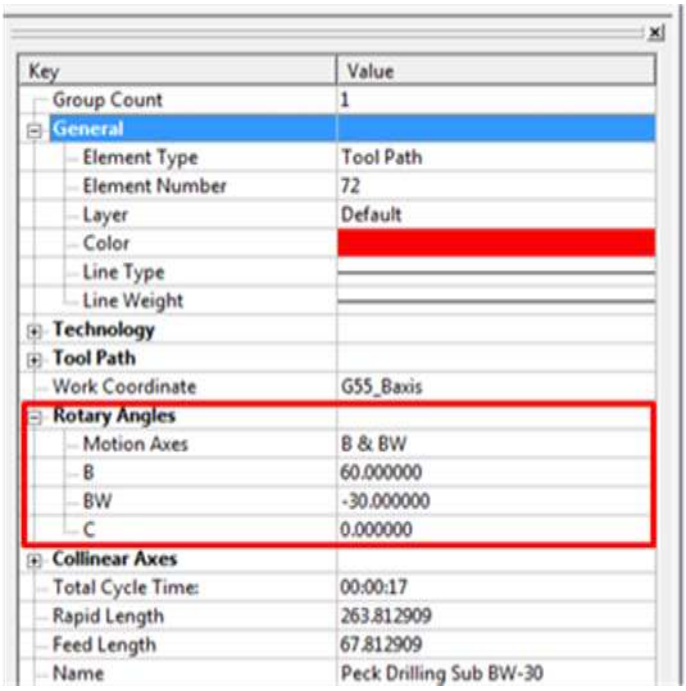
When the Motion axis setup is changed you will also need to

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

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

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

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

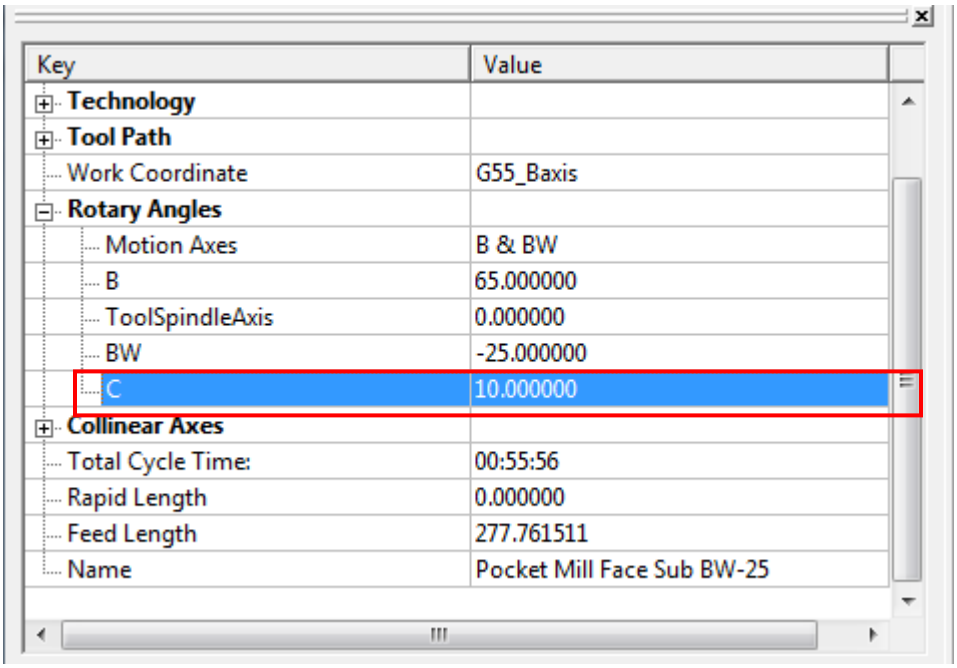


Once the operation grouped, all the user needs to do is expand the Rotary Angle section and change the B or BW angle (ESPRIT will automatically calculate the complementary angle) under Rotary Angles.

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

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

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



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

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

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

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

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

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

21.8. Setting A-axis Position with the AutoPark Addin

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

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

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

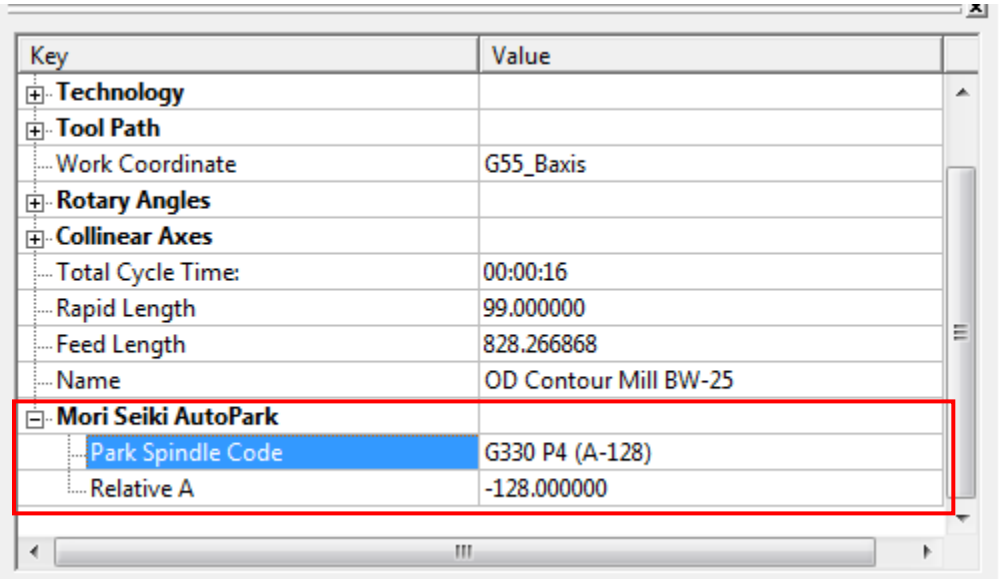
Setting the A axis position:

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

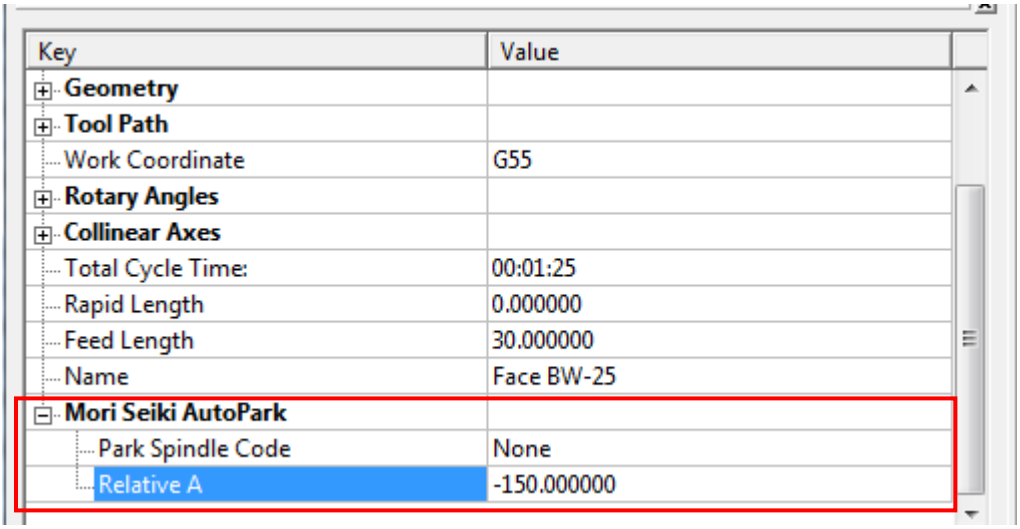
Notes:

- If you select G330 the Relative A field will be set to 0 as this is A0
- If you select G330 P4 the A axis position will be set to the appropriate position for your machine.
  If you change the Relative A axis field to a position other than your machines G330 or G330 P4 position after you park the A axis to either G330 or G30P4 the park spindle code will be set to None.
- If you have run the AutoPark addin on a given sub spindle operation, the A-axis position field (Custom setting 9) from legacy files will be ignored.
- Note the A axis position would always be a value between 0 and the machines negative travel limit.

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



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



NOTES:

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

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

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

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

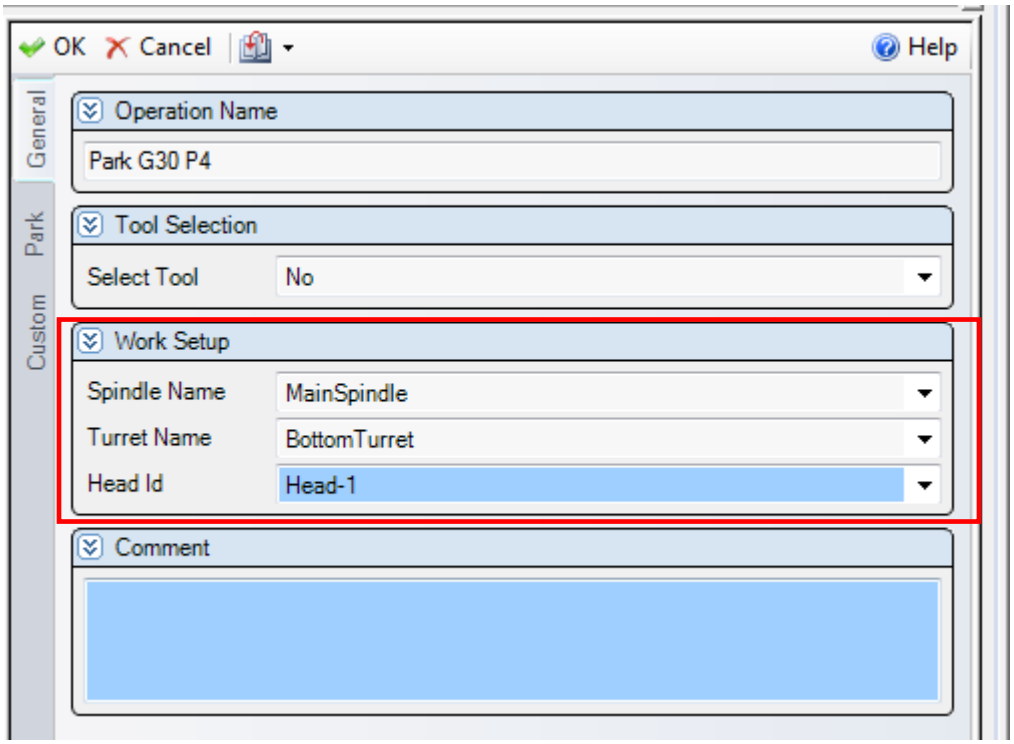
21.9. Parking the Lower Turret

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

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

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

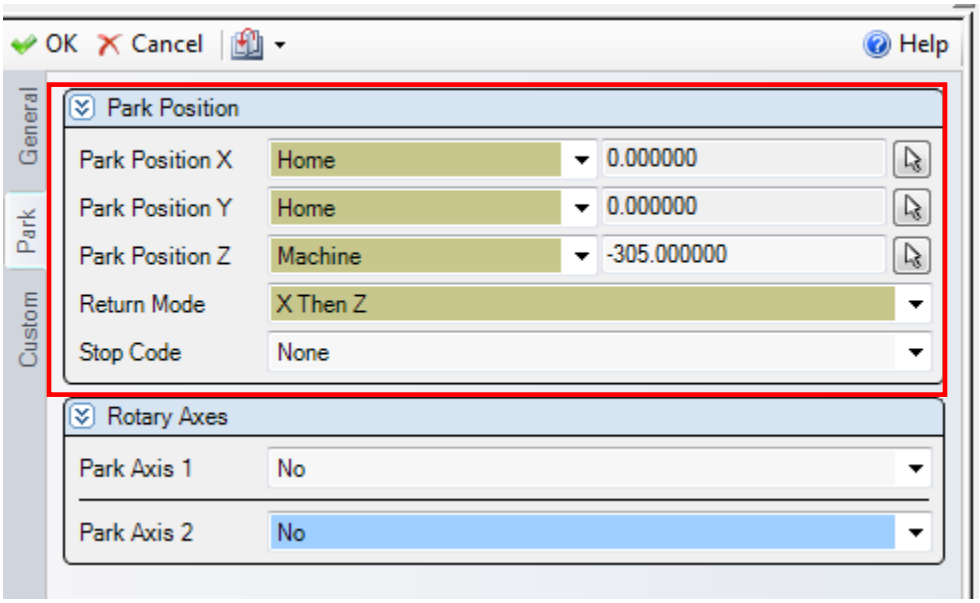
General Tab:



Notes:

1. The tool selection can be setup however you need it to be set to or the select tool option can be set to no.
2. The Spindle must be set to Main Spindle

Park Tab should be set as follows:



Notes:

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

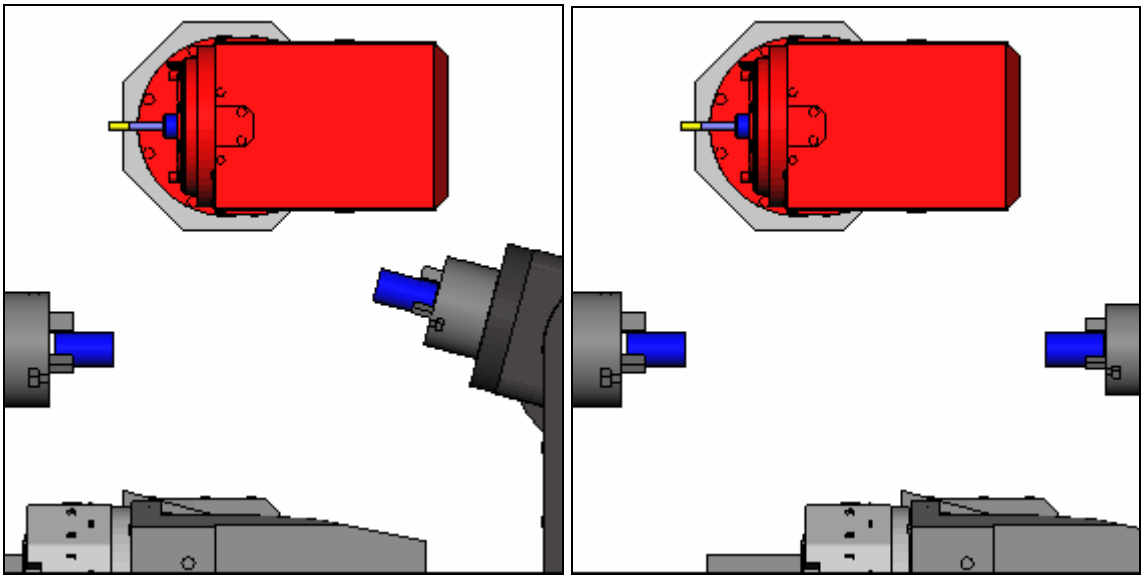
21.10. Parking the A and BW axis Prior to Tool change

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

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

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

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



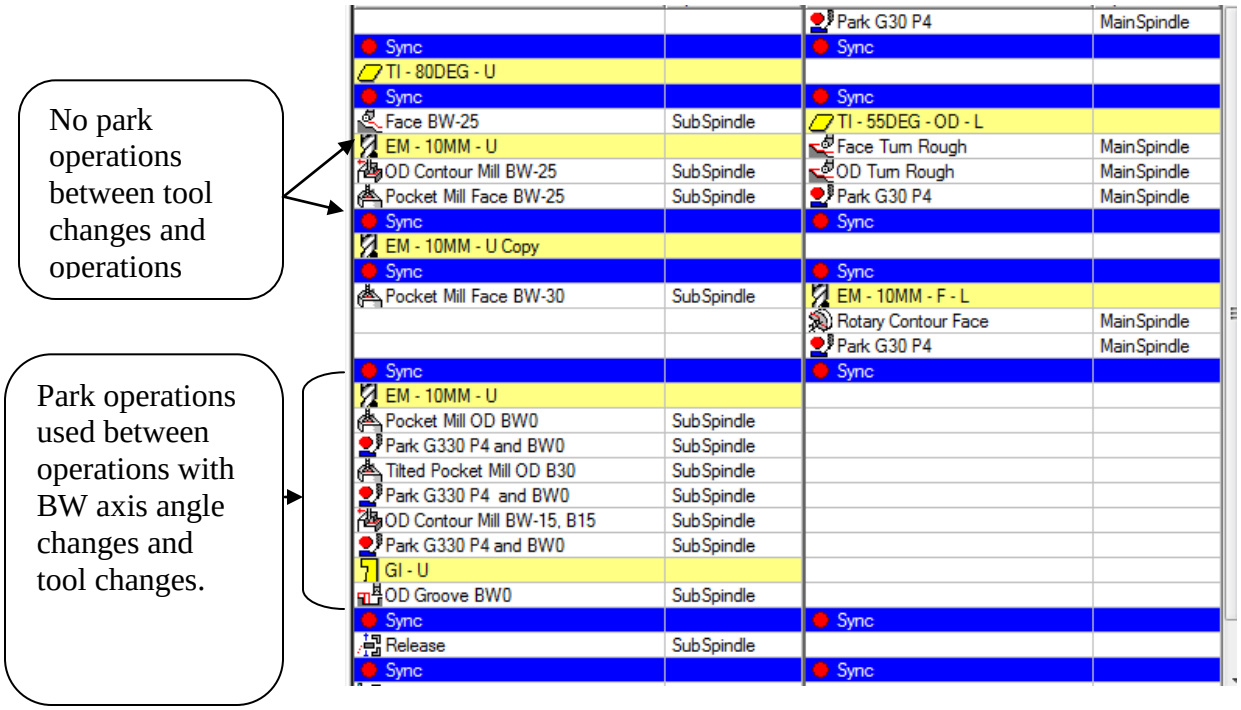
In both of these examples the tool change can be made but there is more clearance when the BW and A axis are at zero.

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

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

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

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



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

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

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

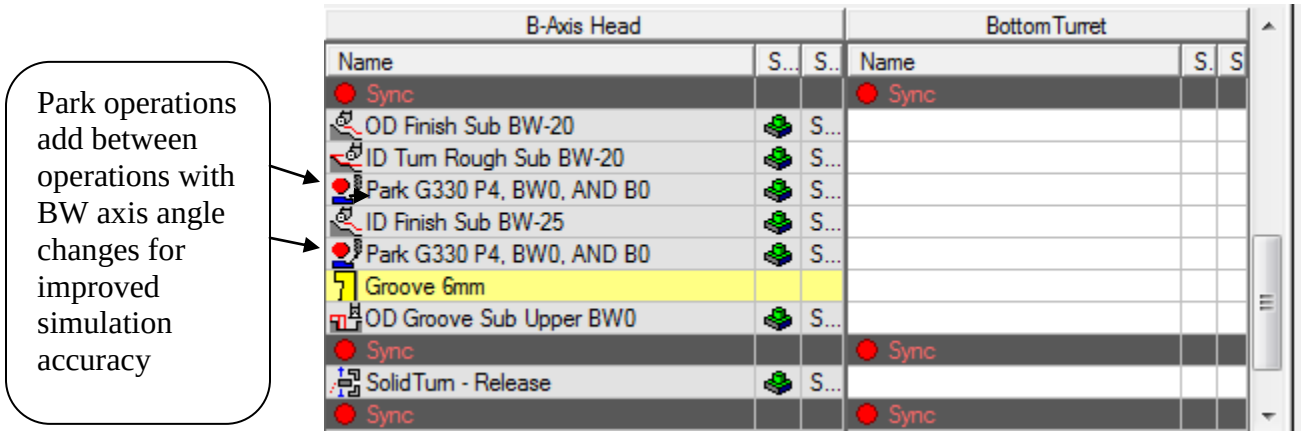
- X axis is moved home
- Y and Z axis are moved home
- The B axis is moved to B0
- The A axis is moved to G440 P4
- The BW axis is moved to the new angle for the next operation
- The B axis is moved to the angle of the next operation
- The A axis is moved to the location set in the next operation.

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

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

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

Refer to the Park Operation section for additional information



21.12. Parking the BW axis Prior to part transfer

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

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

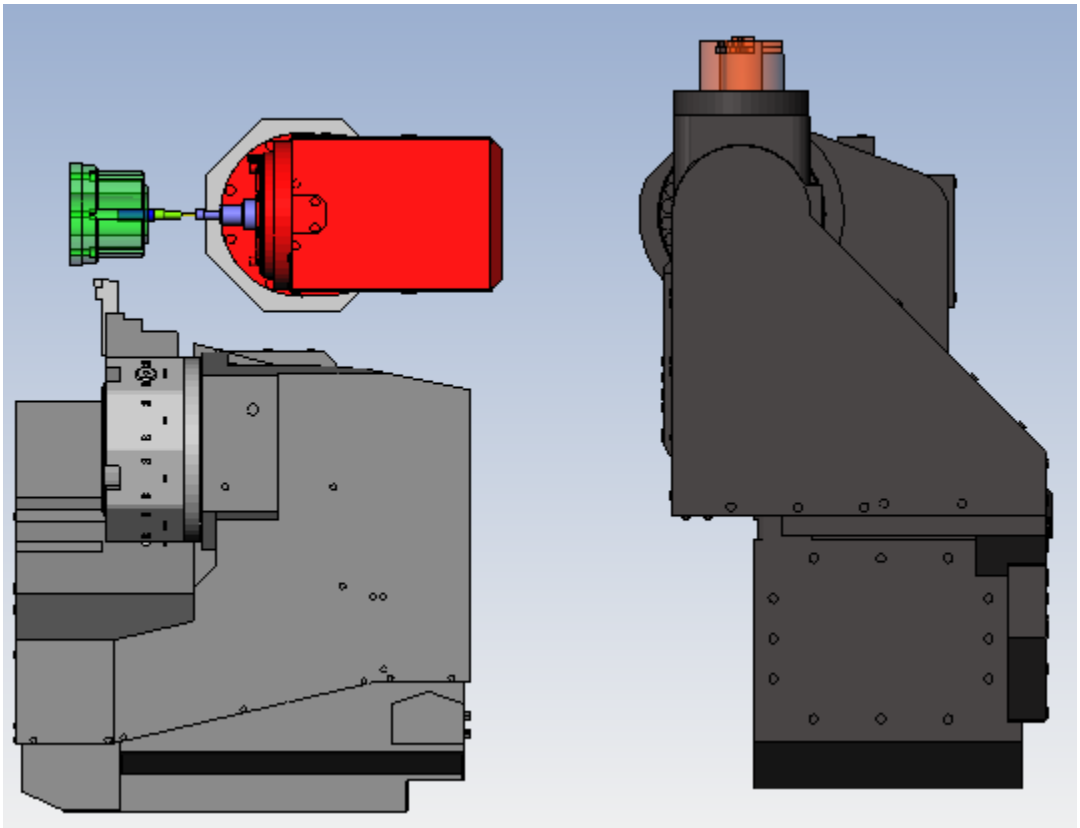
If the transfer does not include a part eject and the BW axis is not at BW0 setup a park operation on the upper head as follows:

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

Refer to the Park Operation section for additional information

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

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



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

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

Refer to the Park Operation section for additional information

21.14. Part Release from Sub Spindle

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

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

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

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

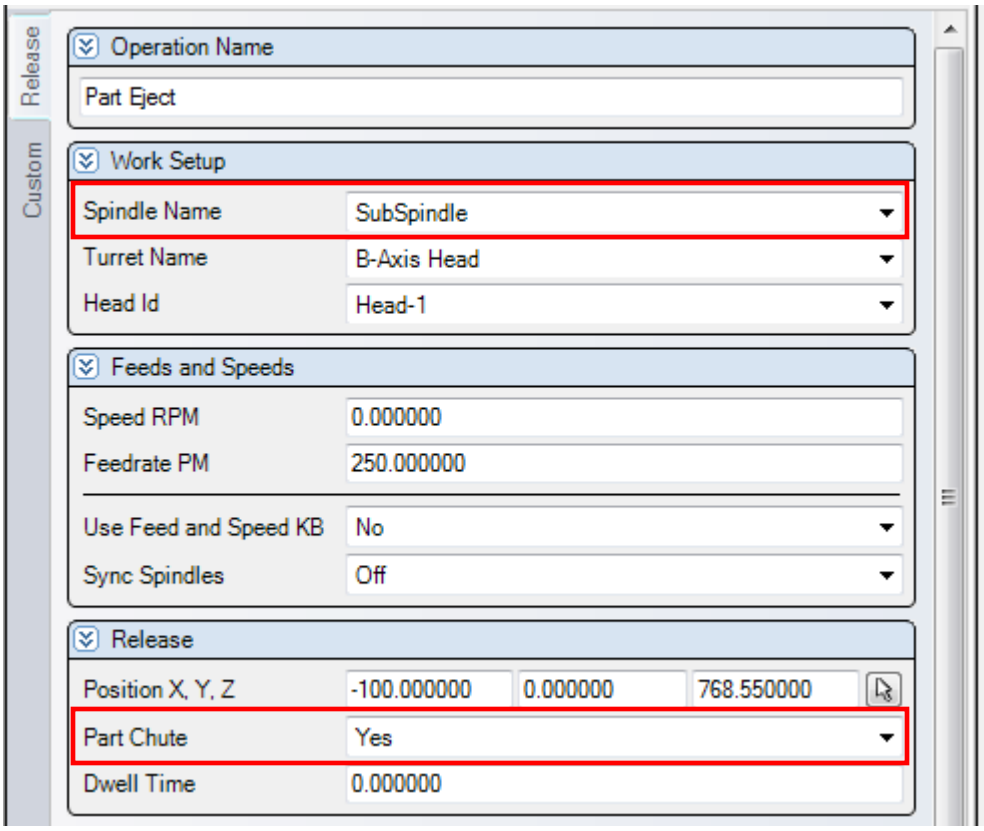
A-axis Position (Custom setting 9) = A axis location for part release
Smart Loader Shutter (M37) (Custom setting 10) =37 and M37\M38 is output to open\close the work unloading conveyer shutter
Notes:

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

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

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

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



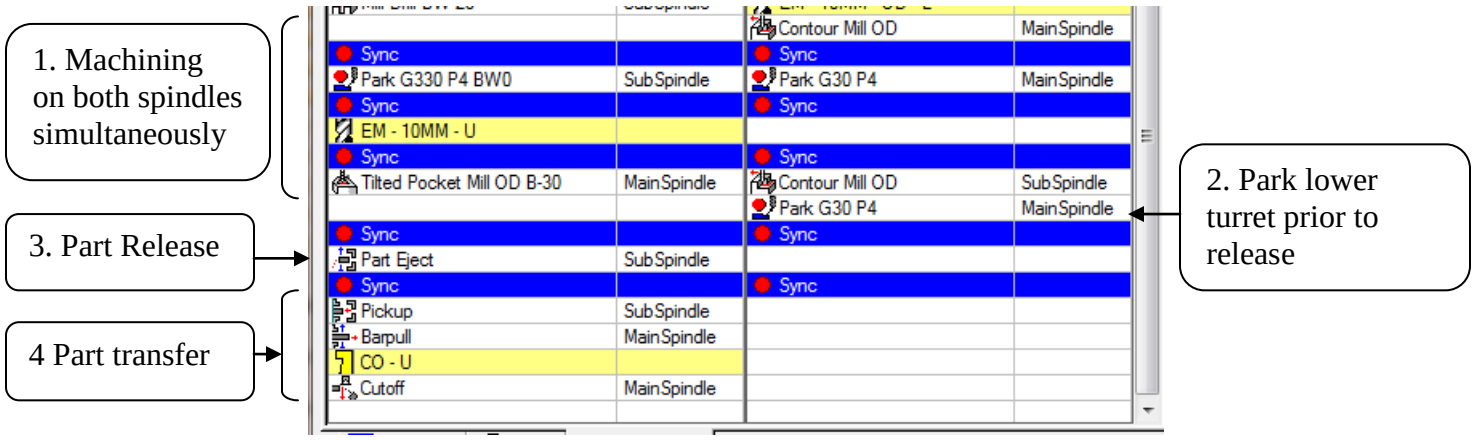
If part chute option is set to yes the NC code will output the following:

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

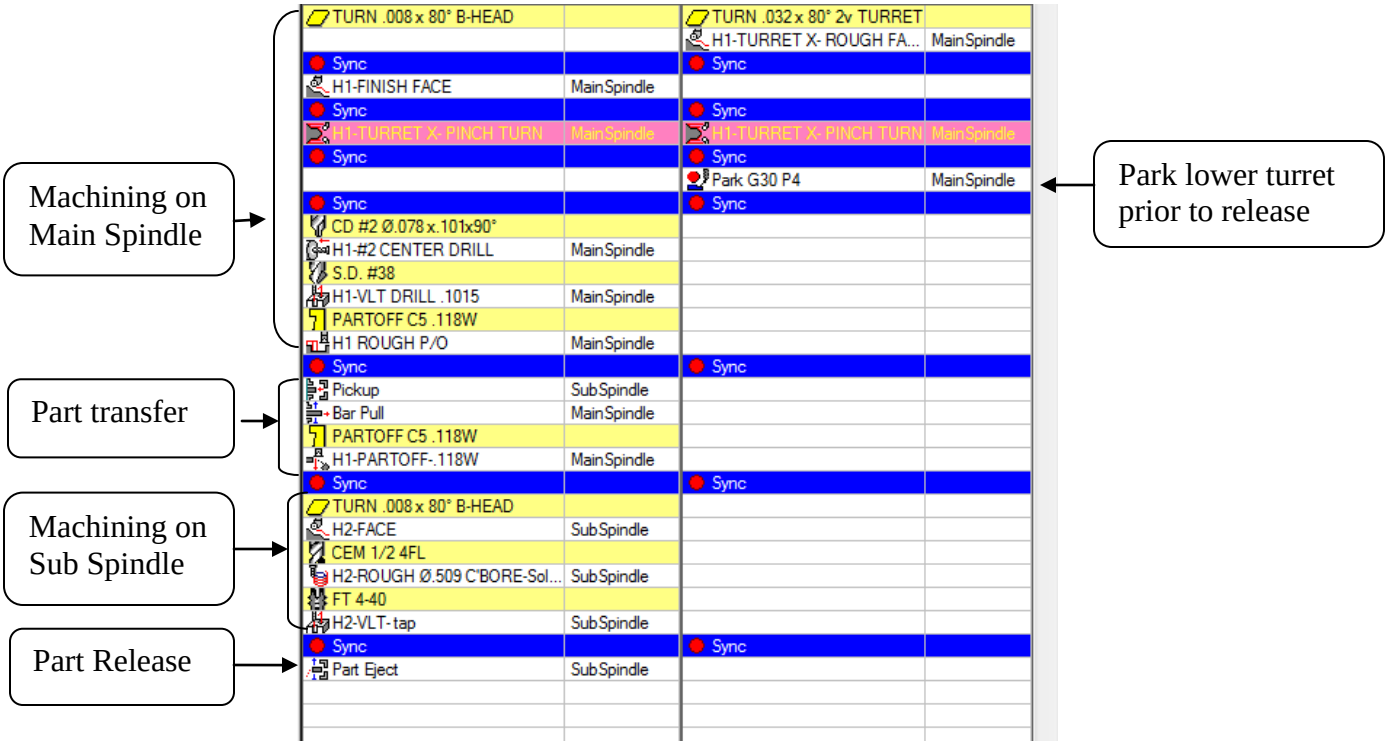
If the part chute option is set to No the post will output an M0 to allow the operator to remove the part from the machine manually

21.15. Location of part release within the ESPRIT file:

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



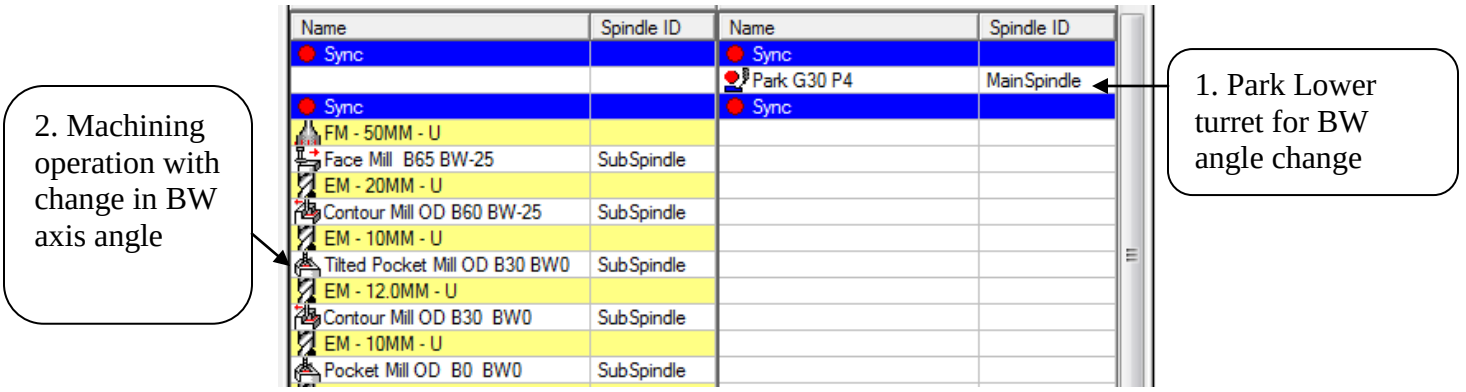
If the part transfer is in the middle of the operations therefore machining only the part one spindle at a time (for example: machining in main spindle, transfer to sub, machine in sub spindle) the part release is typically the last operation in the file as show below:



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

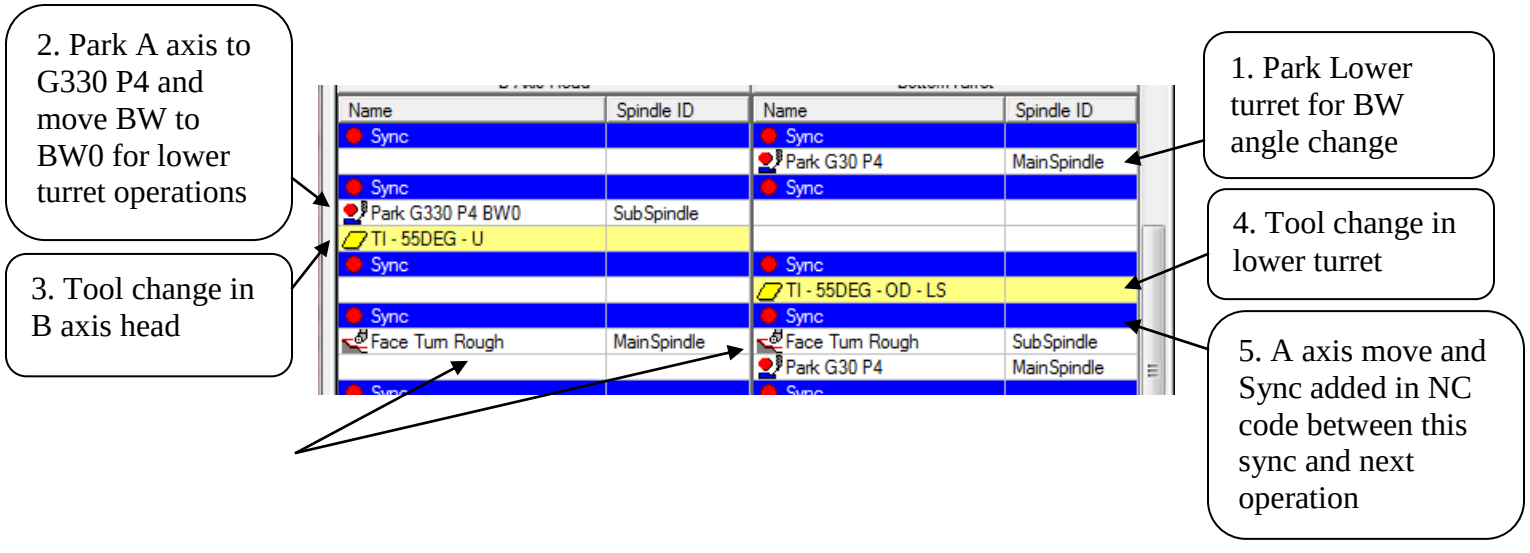
Due to certain machine requirements on the location of the turret heads during movement of the BW rotary axis and to avoid unwanted machine collisions, it is necessary to program specific syncs in ESPRIT to allow the post processor to correctly determine the needed NC code output during these operations.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

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



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

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

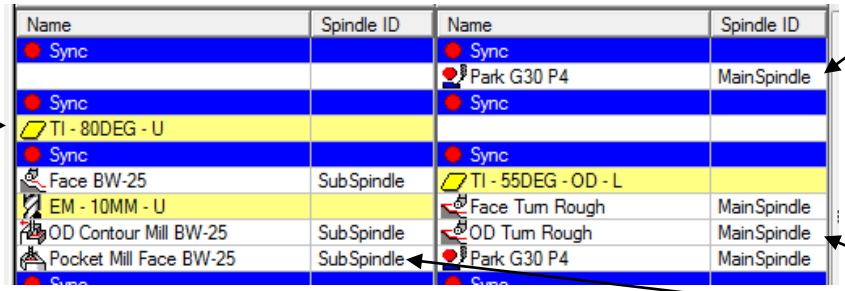


6. Machining operations

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

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

2. Tool change in B axis head, BW and A axis are moved during tool change section



1. Park Lower turret for BW angle change

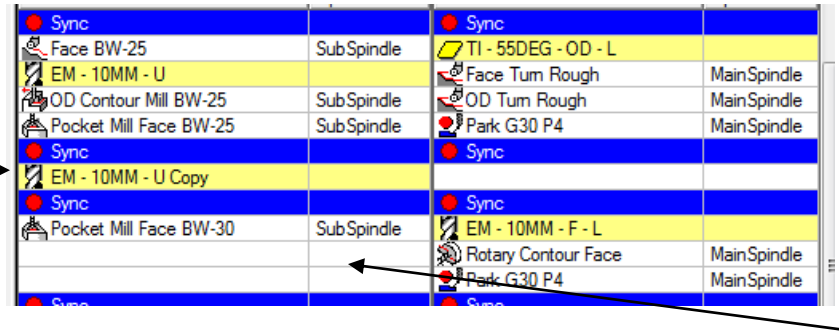
3. Machining operations

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

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

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

2. Tool change in B axis head, BW and A axis are moved during tool change section of NC code

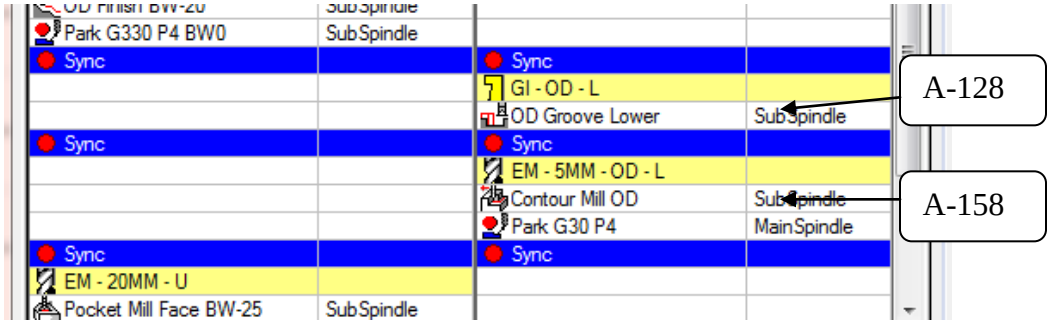


1. Park Lower turret for BW angle change

3. Machining operations

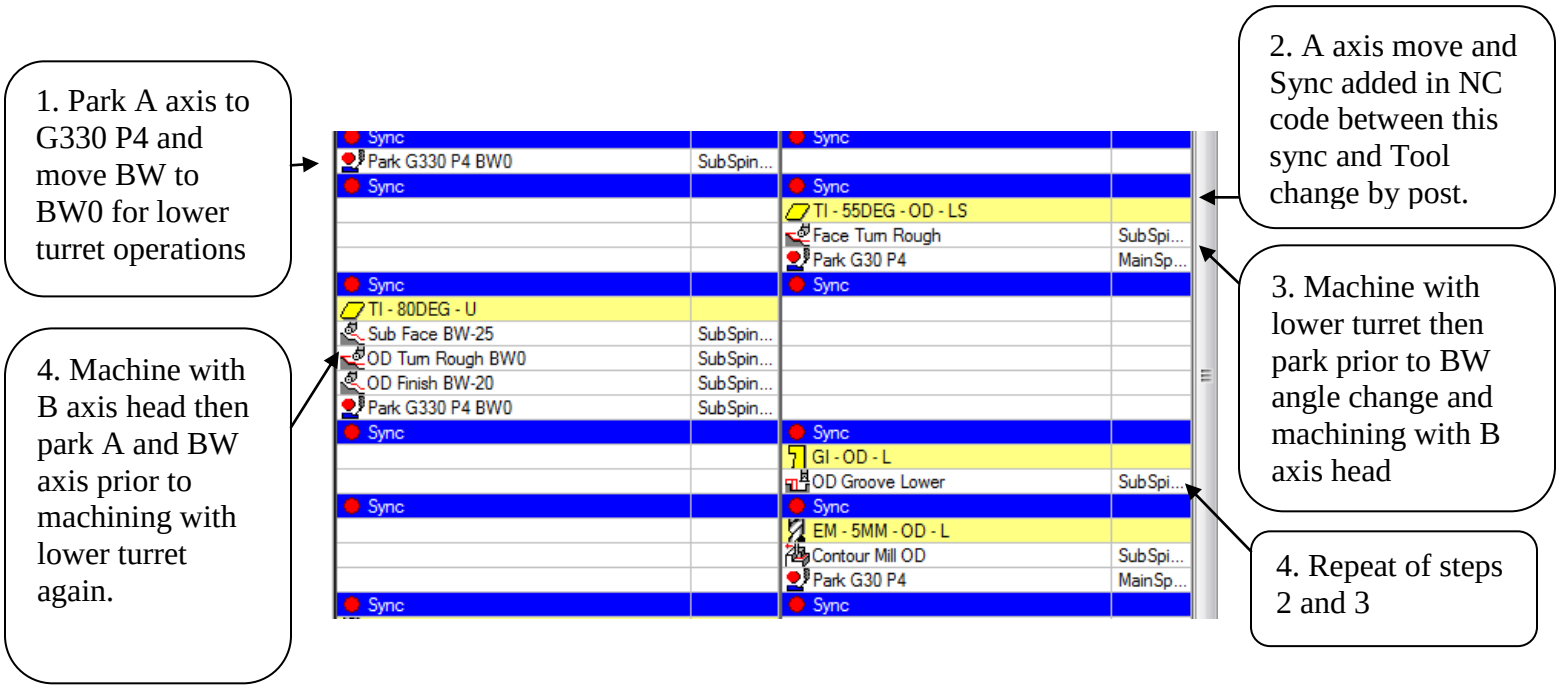
21.16.5. Changing A axis position between two lower turret operations in a row:

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



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

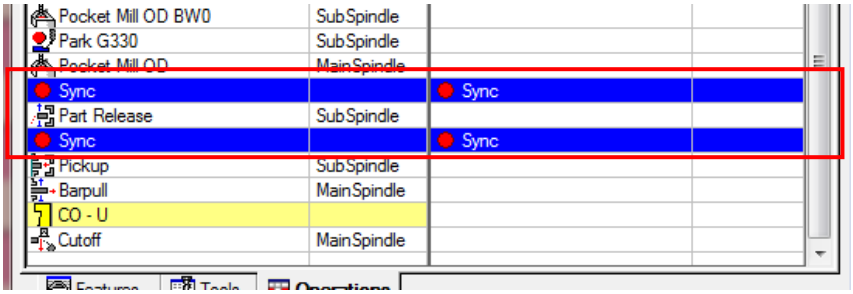
Machining with only one head\turret at a time is much the same as the larger NT machines with the exception of necessary park operations.



21.16.7. Part release on sub spindle

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

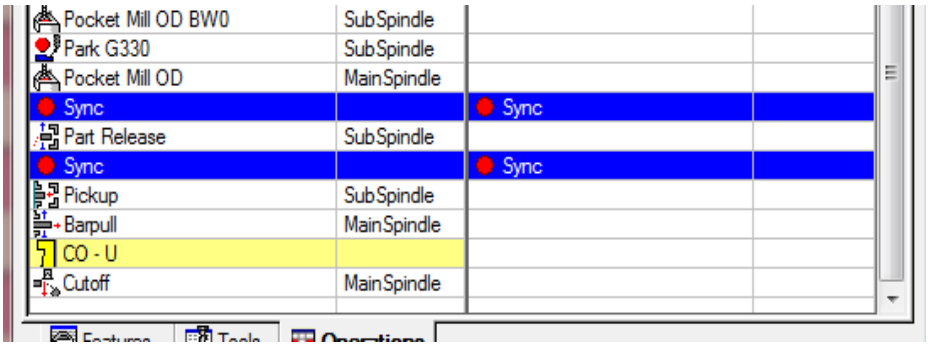
Since the BW axis is rotated by the Release operation the B axis must be moved to a position between 0 and 120 degrees. If needed this can be done with a park operation.



21.16.8. Part Transfer

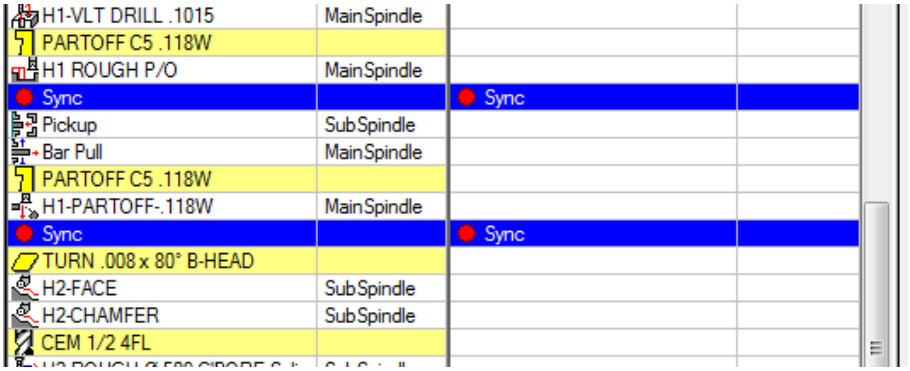
If a part transfer is the last thing in the operation list, a sync must be before the transfer

Part transfer as last operations in file



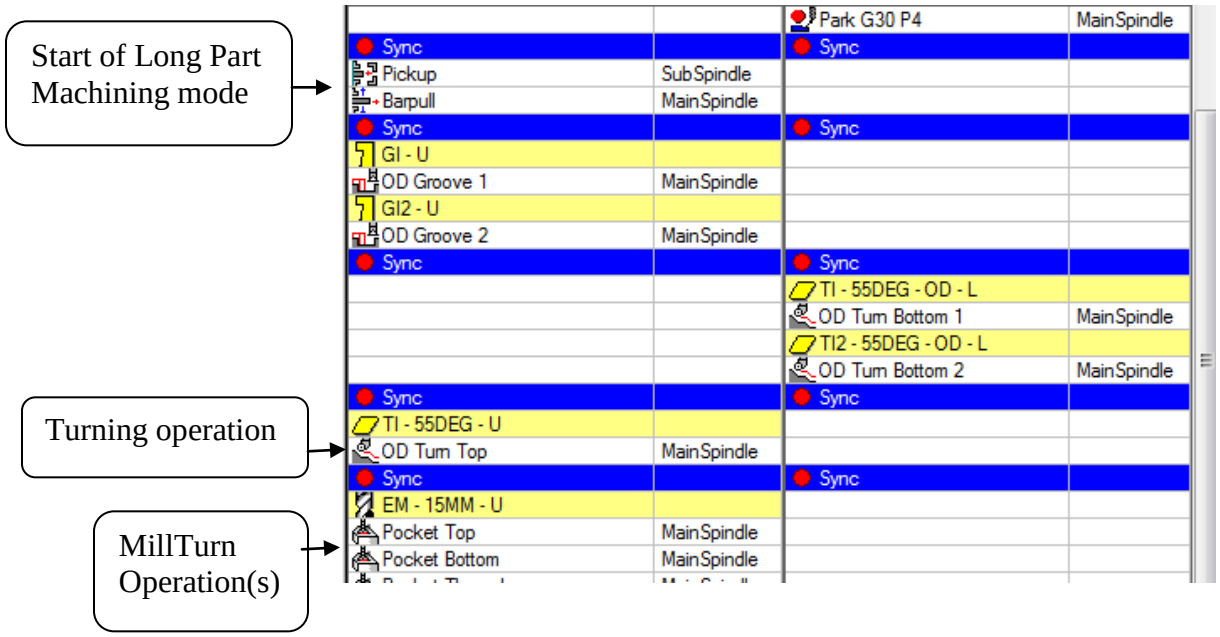
If the part transfer is in the middle of the operations in a given file a sync must be on both sides of the transfer section

Part transfer in the middle of the operations



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

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



22. Custom Settings Index

22.1. Operations Page

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

22.2. Tools Page

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