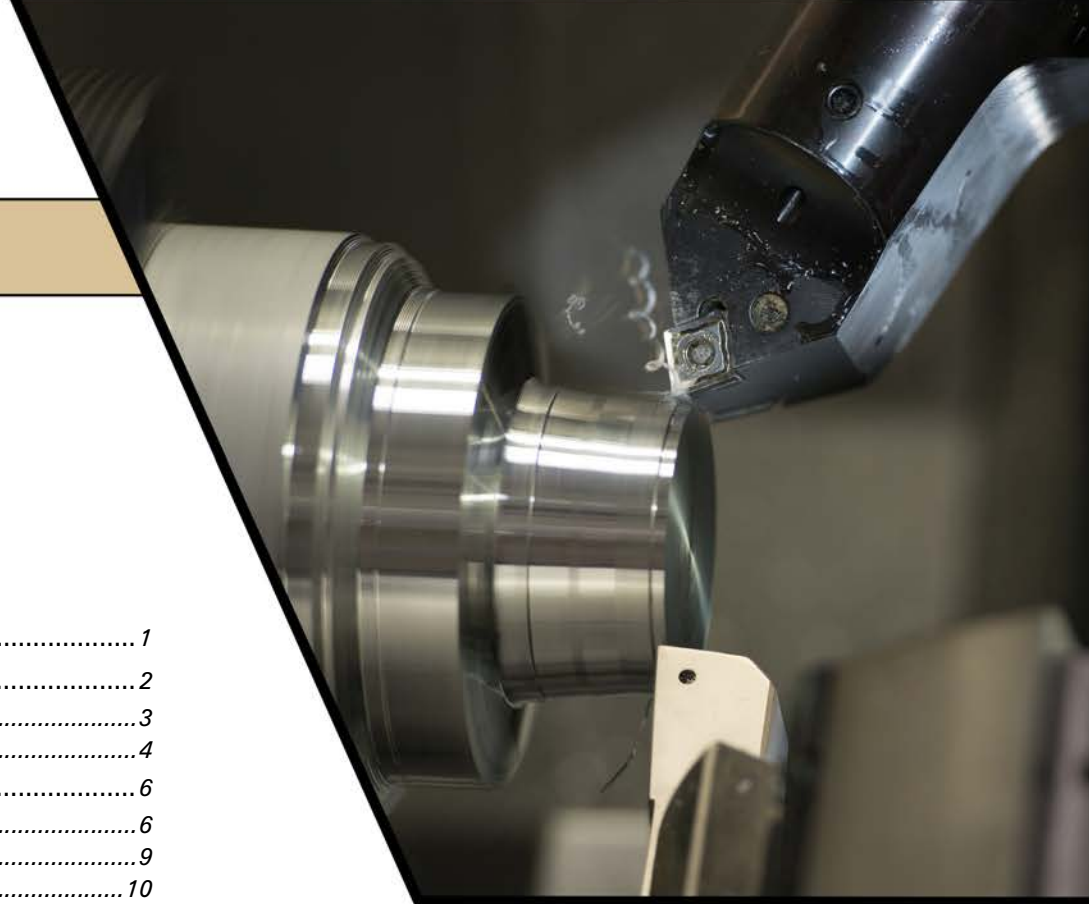


DMG Mori NT/NTX/NZX

Contents

A: Introduction.....	1
B: Customizing your .machine file.....	2
Default values for .machine file settings	3
Configuring the Code Expert editor.....	4
C: Organizing NC output	6
Setting the toolpath directory and stream names	6
Configuring tool table output.....	9
Suppressing “\” comments.....	10
Configuring stops and optional stops.....	11
Outputting the operation number for each toolpath.....	14
Selecting the G-code system.....	16
Resetting the C-axis automatically.....	16
D: Setting up tools.....	19
Rotating the B-axis head to the proper position.....	20
Setting the tool number base for B-axis tools	24
Using heavy tools.....	25
G361/G362 tool change mode.....	25
Live tool support for the lower turret.....	26
E: Supporting peripheral components.....	27
Using the workpiece push check function (G38).....	27
Chip conveyor (M200)/door control (M220).....	30
Part ejector (M47) and part catcher (M73) support.....	32
Tailstock operations.....	33
F: Approach and retract moves	35
Selecting reference positions	35
Setting the type of approach/retract motion.....	38
Forcing G53 output.....	39

revision date: Dec. 2021



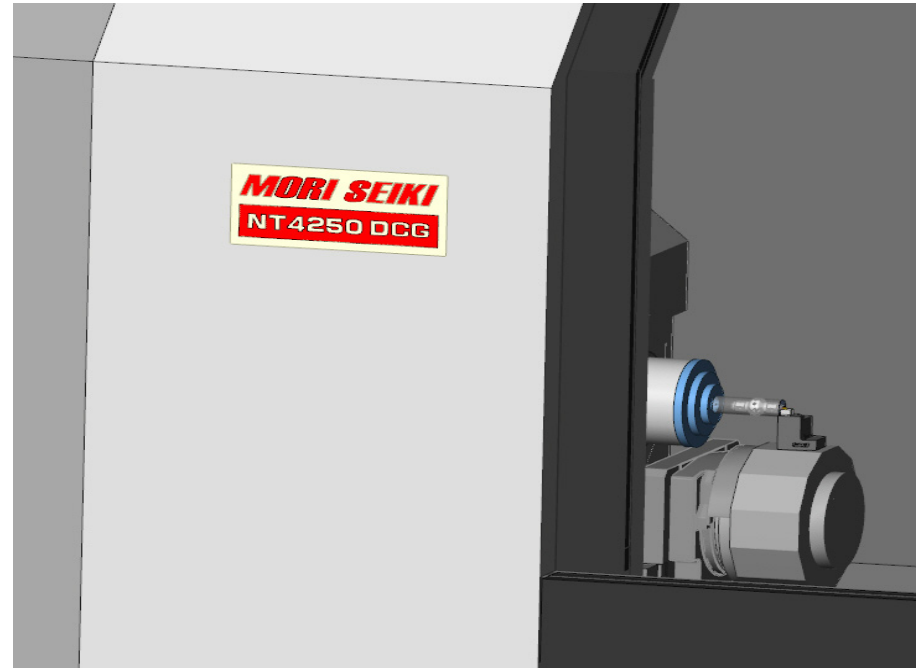
G: Configuring turret park commands	40
H: Milling cycles and modes	43
Polar (G12.1) and cylindrical (G7.1) interpolation.....	44
Tilted-plane machining cycle (G68.1).....	46
Selecting the cutting mode (G332).....	50
Radius/diameter mode (G10.9).....	51
Contour control modes (G05, G08)	52
I: Multiaxis toolpath settings.....	53
Rotary start position.....	53
Pole handling.....	55
Tool tip point control/tool center programming (G43.4)	56
Default settings for multiaxis toolpaths.....	58
J: Coolant options	59

A: Introduction

Your **.machine** file drives your entire Mill-Turn experience. A Mill-Turn **.machine** file is very different from the machine and control definition files that you might be familiar with from other Mastercam products. This application guide explains the specific options and features that are available in your **.machine** file to support your DMG Mori machine. It includes the following sections:

- ❖ Customizing your **.machine** file
- ❖ Organizing NC output
- ❖ Setting up tools
- ❖ Supporting peripheral components
- ❖ Approach and retract moves
- ❖ Milling cycles and modes
- ❖ Multiaxis toolpath settings
- ❖ Coolant options

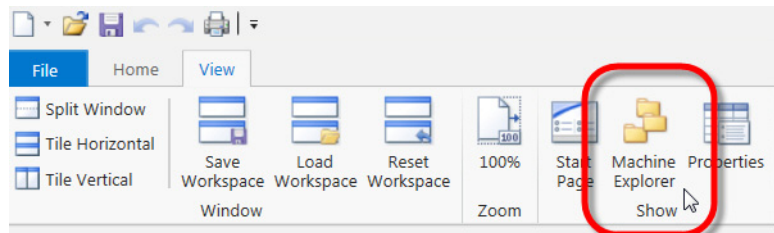
Supported models and controls—This guide applies to the NT, NTX, and NZX models. Note, however, that the **.machine** file does not support the tilted right spindle feature on the NTX models. The DMG Mori **.machine** files and post assume that your machine is equipped with the Fanuc 31i-A5 control.



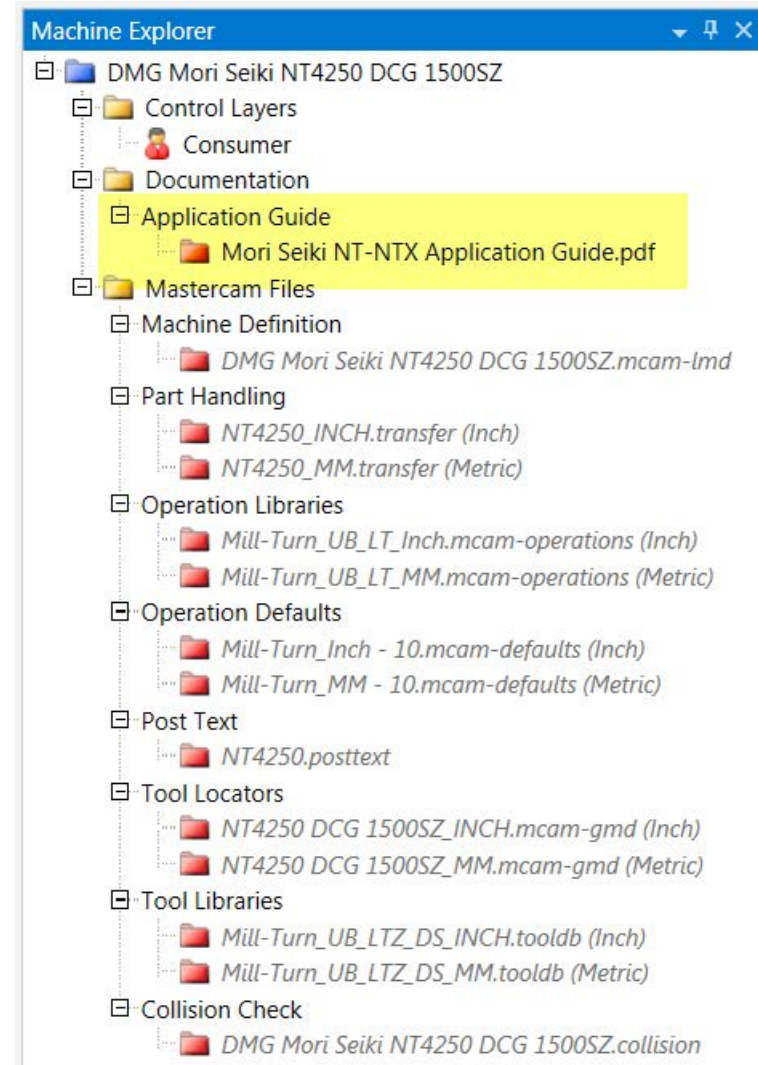
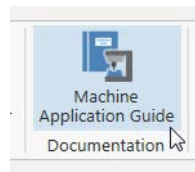
B: Customizing your .machine file

Whenever you load your **.machine** file in Mastercam to begin working on a part, Mastercam also starts Code Expert and loads your **.machine** file there as well. Code Expert is where you can make changes to your **.machine** file, such as editing default settings.

Before you can work with the **.machine** file, the Machine Explorer needs to be visible. Click the **Machine Explorer** button on the **View** tab.



You can also access this application guide directly from Code Expert: click the **Machine Application Guide** button on the **Home** tab.



Default values for .machine file settings

Although your **.machine** file is typically supplied to you ready-to-use by your Reseller, it includes many settings that you can configure yourself according to your preferences and specific application needs. These include sequence and sync numbering, tool offset numbering, use of spaces in your NC file, job/shop info for your NC header, and so on.

Reach these settings by opening your **.machine** file in Code Expert and double-clicking the **Consumer** icon in the Machine Explorer.

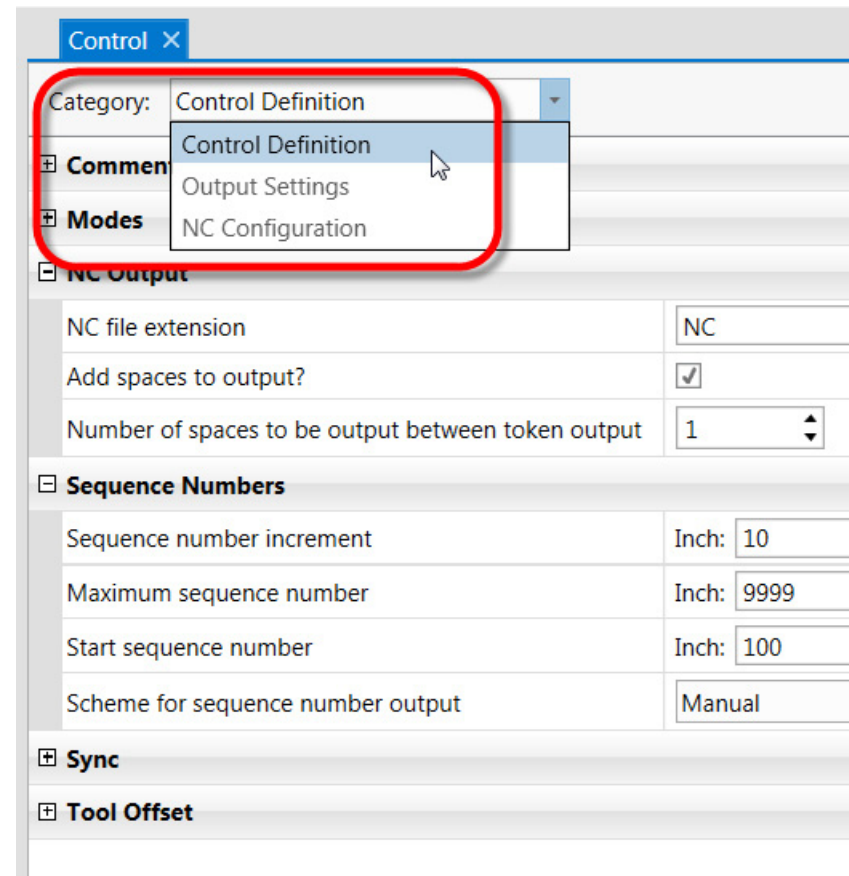
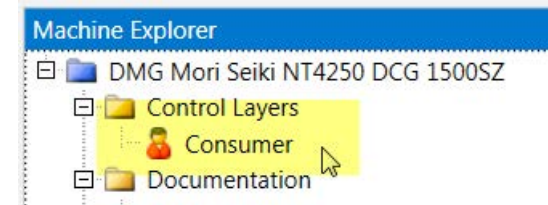
The settings are grouped into several categories. Click the **+** signs to see the individual options. Some of these options (for example, sequence number settings) are very common or generic to most controls; others are specific to your individual machine.

These settings serve a wide variety of functions:

- Mimicking the control definition settings found in other Mastercam products.
- Configuring toolpath modes and cycles.
- Setting default values for Sync Manager and toolpath options.

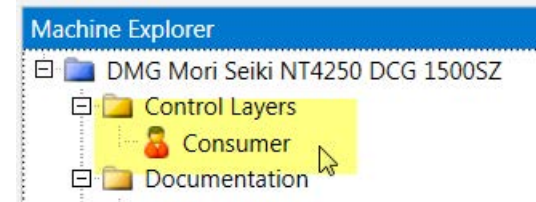
Many of these settings are self-explanatory and you can easily configure them by simply browsing the interface. The settings that are specific to this **.machine** file are described in this guide.

After making any changes, press **[Ctrl+S]** to save your **.machine** file.



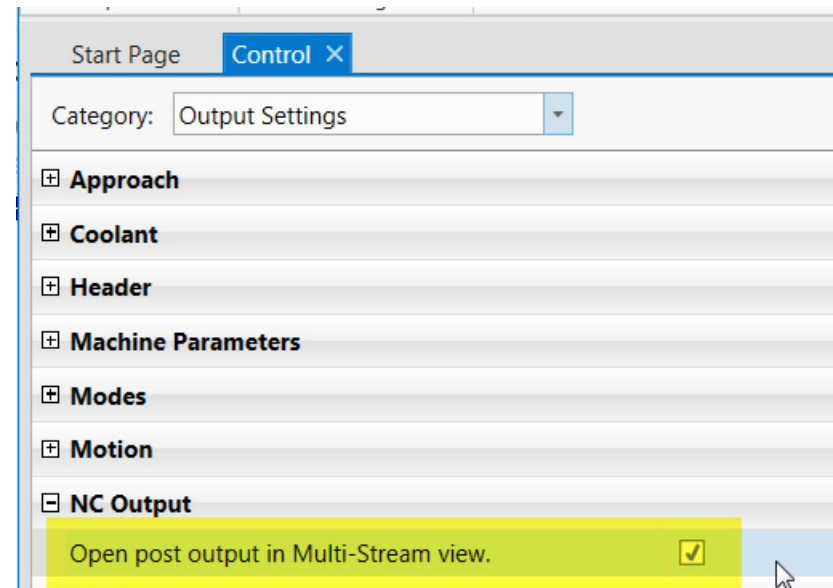
Configuring the Code Expert editor

There are several settings in the **.machine** file that you can use to configure the Code Expert editor. Open the **Consumer** layer to see these options.



Opening in multi-stream view—You can use the Code Expert editor in either single-stream or multi-stream mode. Since the DMG Mori NT/NTX/NZX NC output is typically divided into two streams, you may wish to open files in multi-stream view by default.

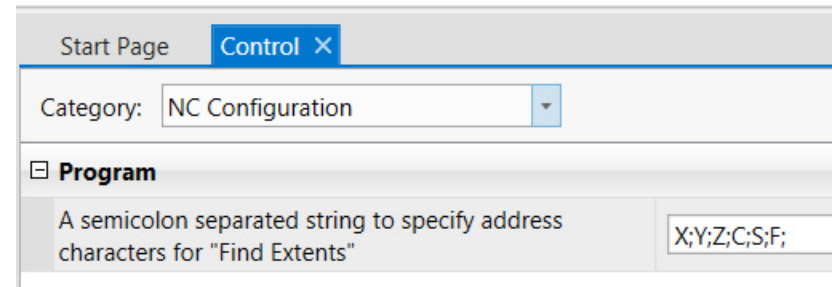
1. Go to the **Output Settings** category.
2. Open the **NC Output** group.
3. Select the **Open post output in Multi Stream view** option.



Configuring the Find Extents feature—The **Find Extents** feature in Code Expert scans your NC file and displays the minimum and maximum values for each letter address. By default, this is set to scan **X, Y, Z, C, S** and **F**. If you wish, you can edit the set of addresses that are scanned.

1. Go to the **NC Configuration** category.
2. Open the **Program** group.
3. Enter the desired letter addresses in the list, separated by semi-colons (;).

For example, you can choose to add **I;J;K;** to the list. The next time you post, you will see them in the **Find Extents** table.

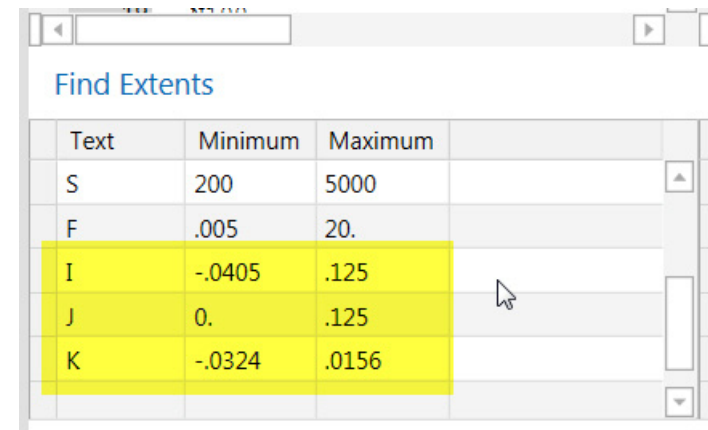


Start Page Control X

Category: NC Configuration

☒ **Program**

A semicolon separated string to specify address characters for "Find Extents" X;Y;Z;C;S;F;



Text	Minimum	Maximum
S	200	5000
F	.005	20.
I	-.0405	.125
J	0.	.125
K	-.0324	.0156



C: Organizing NC output

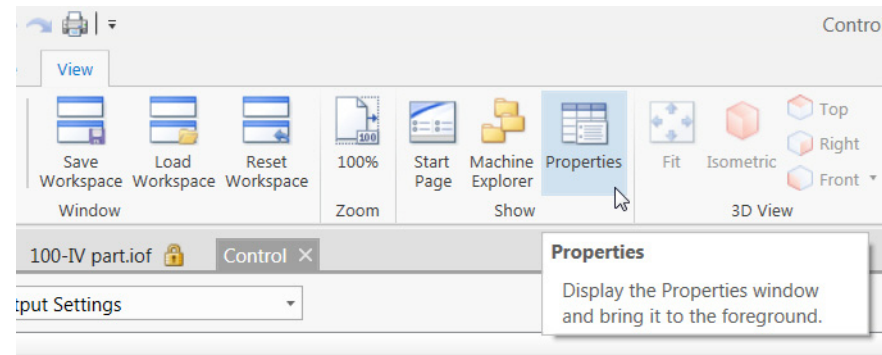
Setting the toolpath directory and stream names

By default, your NC file is written to the following folder:

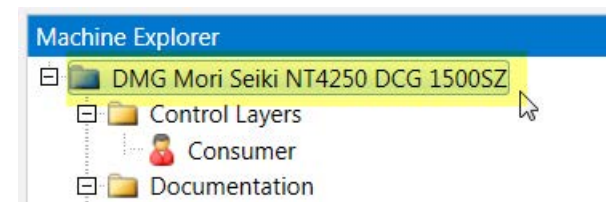
`\my mcam2019\Mill Turn\NC`

If you wish, you can set a different destination. Follow these steps:

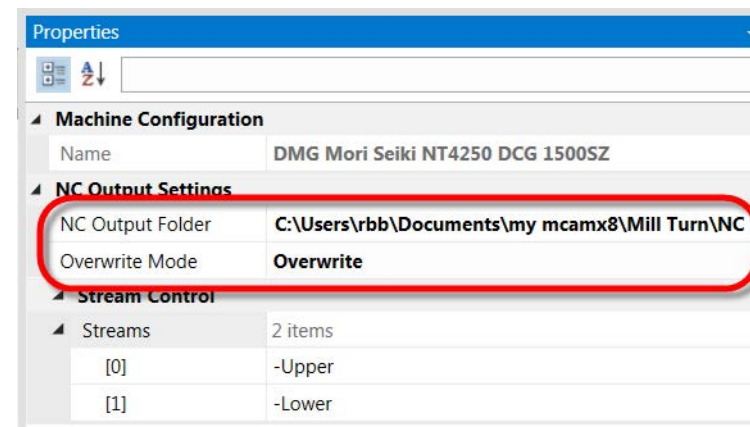
1. Open the **.machine** file in CodeExpert.
2. Make sure that the **Properties** window is displayed.



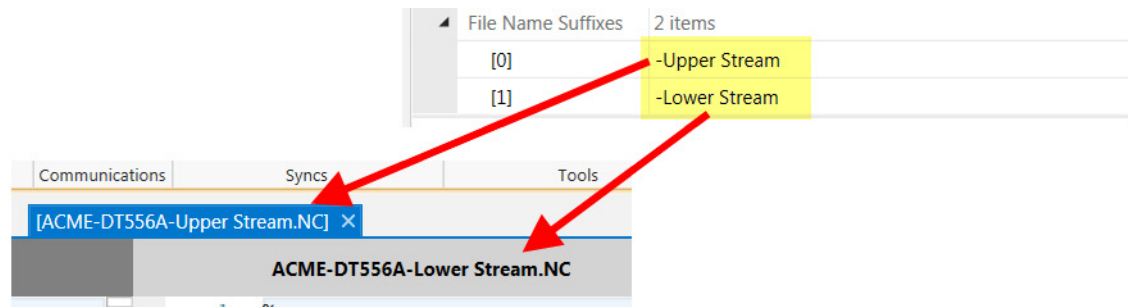
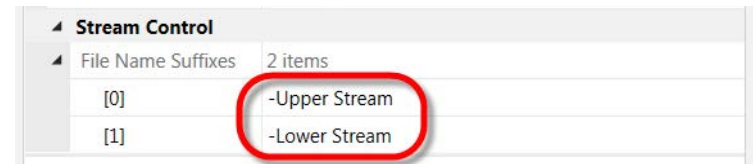
3. Click the machine name in the **Machine Explorer**.



4. Select the desired **NC Output Folder**. Mastercam will write your NC files for this machine to this folder.
5. You can also choose whether Mastercam will automatically overwrite NC files with the same name, or prompt you to enter a different name. Select the desired **Overwrite Mode** to control this.

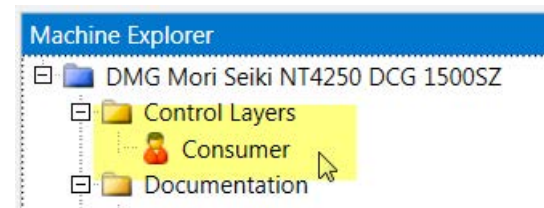


6. Mastercam Mill-Turn also lets you configure the names of the upper and lower streams. These will be automatically added to the NC file names for each stream. You can edit these names if you wish.

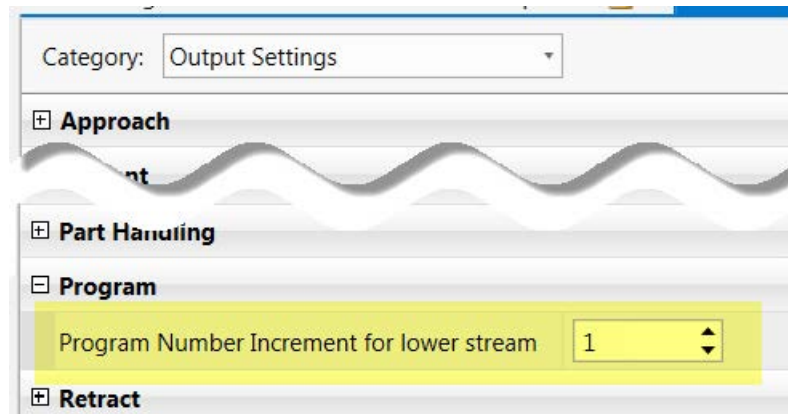


Your **.machine** file also gives you the option to configure the program number for the lower stream.

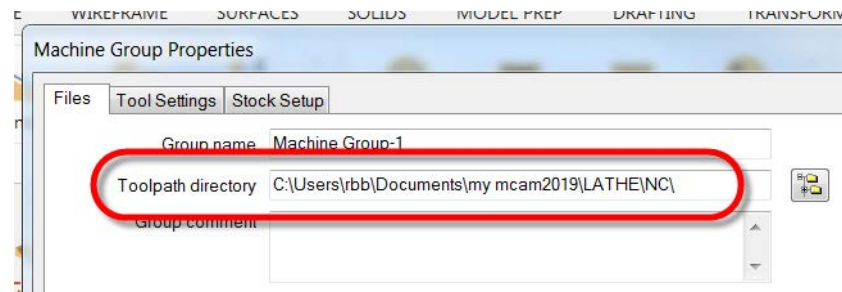
7. Double-click the **Consumer** layer.



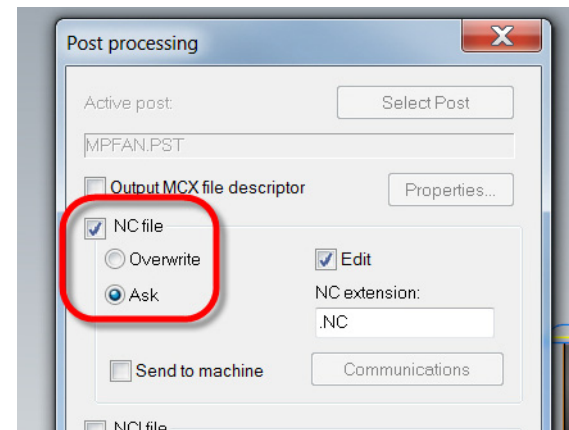
8. Select **Category: Output Settings**.
9. Go to the **Program** section.
10. Enter the desired **Program Number Increment**. Mastercam will add this value to the upper-stream program number to get the program number for the lower stream (0000n).
11. Save the **.machine** file when you are done



If you are familiar with Mastercam, you are probably familiar with the **Toolpath directory** setting in the **Machine Group Properties**. This setting is not used in Mill-Turn.



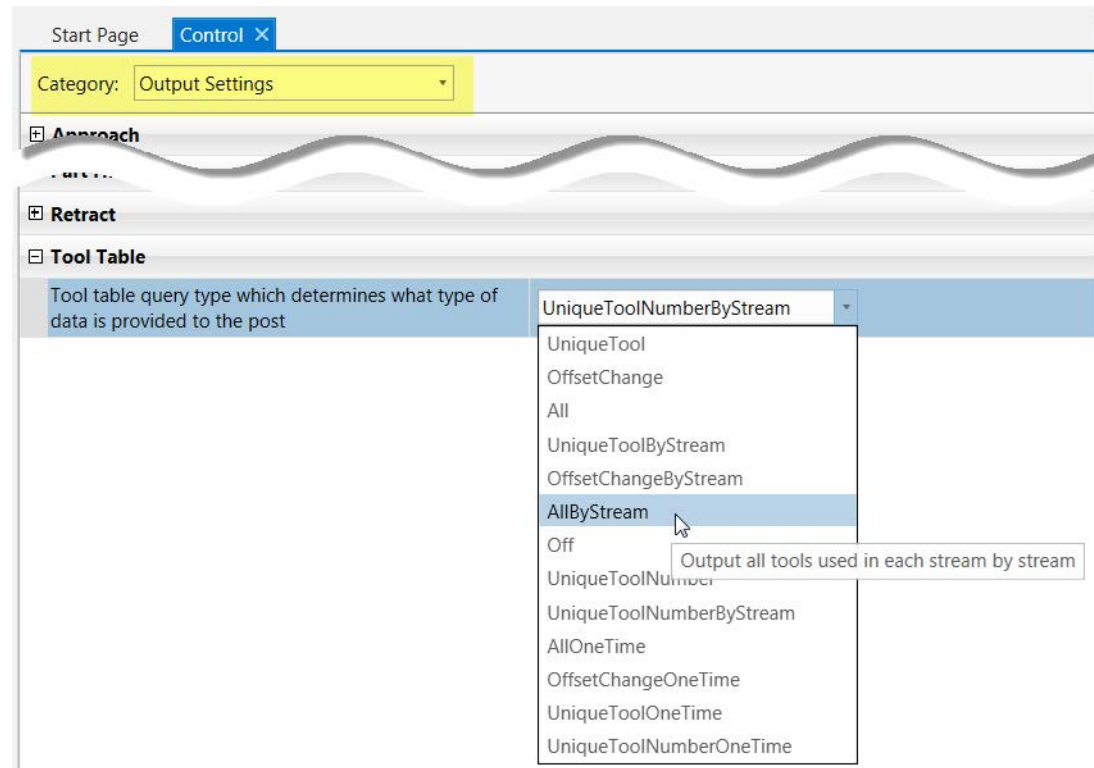
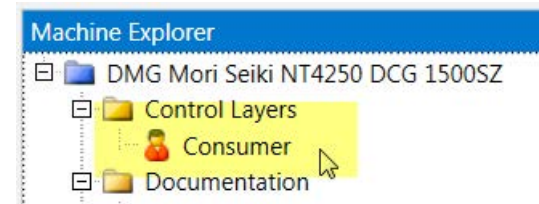
Also, the **Overwrite Mode** setting replaces these settings from the **Posting** dialog box in other Mastercam products.



Configuring tool table output

Your **.machine** file includes 12 different tool table strategies. This means that you can configure your tool table output without needing to do any programming or edits to your post, simply by selecting the desired strategy. Follow these steps:

1. Double-click the **Consumer** layer.
2. Go to the **Output Settings** category.
3. Open the **Tool table** group.
4. Select the desired strategy. Hover over each one to see a description.
5. Press **Ctrl+S** before posting to save your setting.



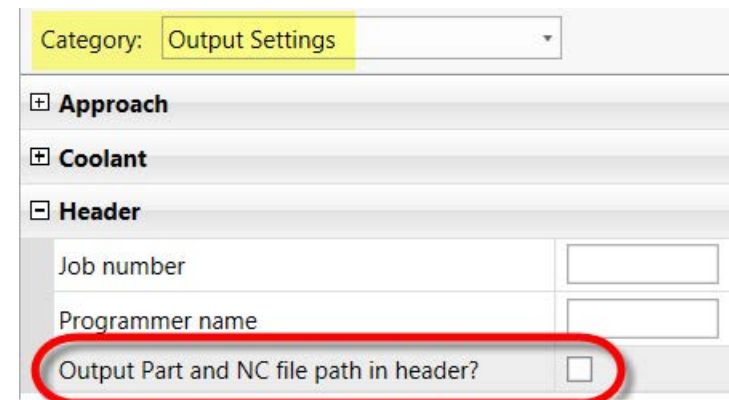
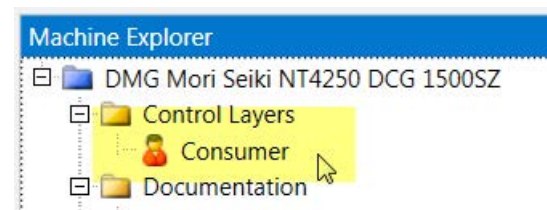
Suppressing “\” comments

Your DMG Mori post is configured by default to output the name and path of the Mastercam part file and the NC file. These strings typically include a “\” backslash character. However, some machines cannot support this character in the NC code.

```
(DMG MORI SEIKI NT MILL TURN MACHINE)
(
  (DATE: WEDNESDAY, 07 FEBRUARY 2018 - TIME: 21:57)
  (MCX FILE - C:\USERS\RBB\DOCUMENTS\MY MCAM2018\PARTS\MORI
  (NC FILE - C:\USERS\RBB\DOCUMENTS\MY MCAM2018\MILL TURN\N
  (JOB NUMBER - )
  (PROGRAMMER - )
  (T1001 | OD ROUGH RIGHT - 80 DEG. | INSERT - CNMG 12 04 (
  (T1002 | 23. BULL ENDMILL 4.4 RAD | DIA. - 23)
  (T1003 | OD CUTOFF RIGHT | INSERT - NONE)
)
```

Your DMG Mori **.machine** file includes an option that you can use to suppress these comments from your NC code. Follow these steps:

1. Double-click the **Consumer** layer.
2. Go to the **Output Settings** category.
3. Open the **Header** group.
4. De-select the **Output Part and NC file path in header?** option.
5. Press **Ctrl+S** before posting to save your setting.



Configuring stops and optional stops

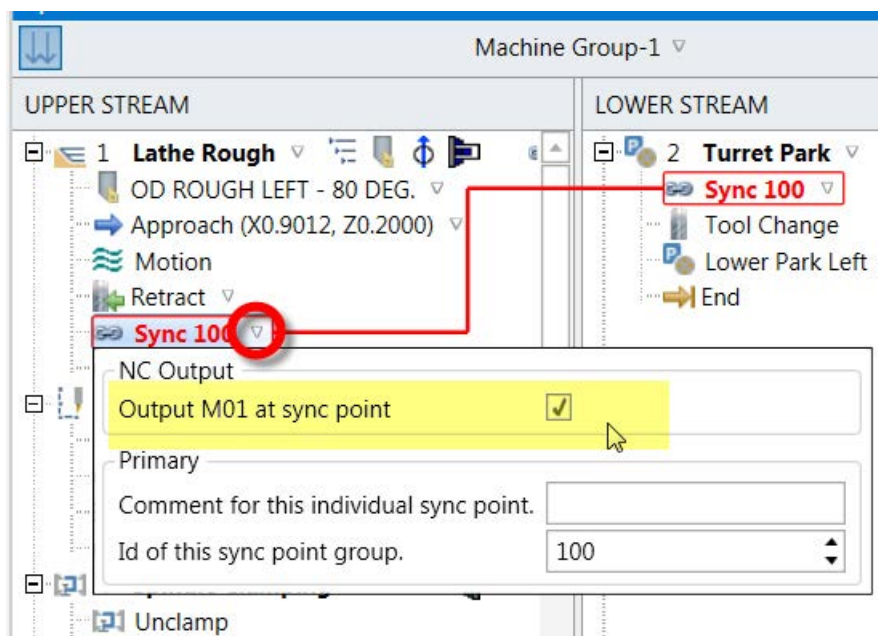
Your **.machine** file gives you several options for pausing the NC program in response to different events. You can program any of the following:

- Output an optional stop (M01) on every wait code or sync.
- Output an optional stop (M01) after every operation.
- Turn off the spindle (M05) after each lathe operation.

Outputting an optional stop with wait codes

Your **.machine** file lets you output an optional stop (M01) immediately following a wait (or sync) code. Follow these steps:

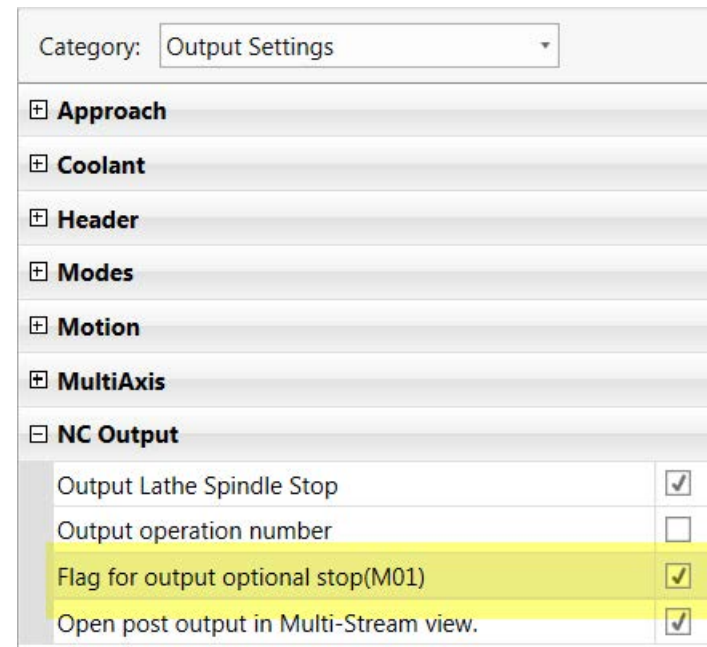
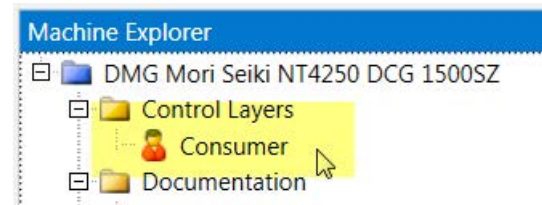
1. In the Sync Manager, locate the sync for which you want to output the M01.
2. Click on the small triangle after the sync command.
3. Select the **Output M01 at sync point** option.
4. Repeat these steps for the sync in the other stream! You must select this option in *both* streams.
5. Save the IOF file to save the changes back to your part file.



Outputting an optional stop after each operation

Your **.machine** file gives you the option of outputting an optional stop (M01) after every each operation. Follow these steps:

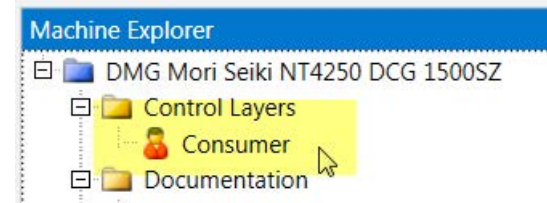
1. Open the DMG Mori NT/NTX/NZX **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.
3. Select **Category: Output Settings**.
4. Go to the **NC Output** section.
5. Select the **Flag for output optional stop** option.
6. Save the **.machine** file.



Stopping the spindle between lathe operations

Your **.machine** file gives you the option of stopping the spindle (M05) after every lathe operation. Follow these steps:

1. Open your **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.



3. Select **Category: Output Settings**.
4. Go to the **NC Output** section.
5. Select the **Output Lathe Spindle Stop** option.
6. Save the **.machine** file.

Category: Output Settings	
⊕ Approach	
⊕ Coolant	
⊕ Header	
⊕ Modes	
⊕ Motion	
⊕ MultiAxis	
⊖ NC Output	
Output Lathe Spindle Stop	<input checked="" type="checkbox"/>
Output operation number	<input checked="" type="checkbox"/>
Flag for output optional stop(M01)	<input type="checkbox"/>
Open post output in Multi-Stream view.	<input checked="" type="checkbox"/>
⊕ Part Handling	



Outputting the operation number for each toolpath

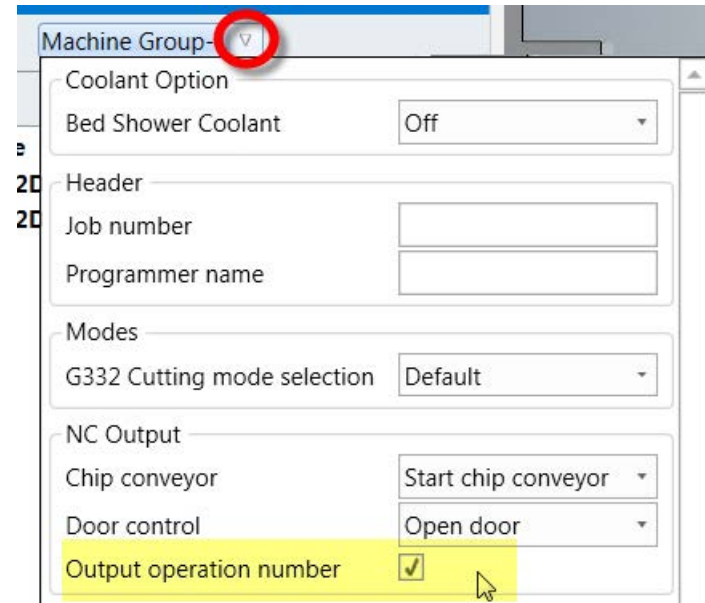
Your **.machine** file gives you the option to output the operation number for each toolpath as a comment in your NC file.

```

11 N130
12 (OPERATION # 2)
13 G10.9 X0
14 (T0101 | OD ROUGH LEFT - 80 DEG. |
15 M46 (TURNING MODE - MAIN SPINDLE)
16 G28 U0.
17 G28 W0.

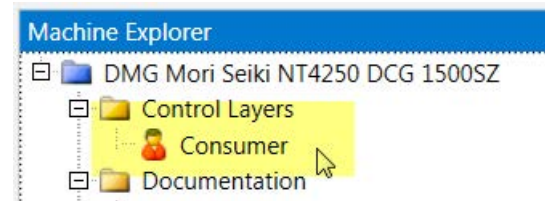
```

Select the **Output operation number** option in the machine group properties section of the Sync Manager. This will enable the comment for all the operations in your part.



You can also control the default setting. Follow these steps:

1. Open the DMG Mori NT/NTX/NZX **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.



3. Select **Category: Output Settings**.
4. Go to the **NC Output** section.
5. Select the **Output operation number** option.
6. Save the **.machine** file.

Category: Output Settings ▼

⊕	Approach	
⊕	Coolant	
⊕	Header	
⊕	Modes	
⊕	Motion	
⊕	MultiAxis	
⊖	NC Output	
	Output Lathe Spindle Stop	<input checked="" type="checkbox"/>
	Output operation number	<input checked="" type="checkbox"/>
	Flag for output optional stop(M01)	<input type="checkbox"/>
	Open post output in Multi-Stream view.	<input checked="" type="checkbox"/>
⊕	Part Handling	

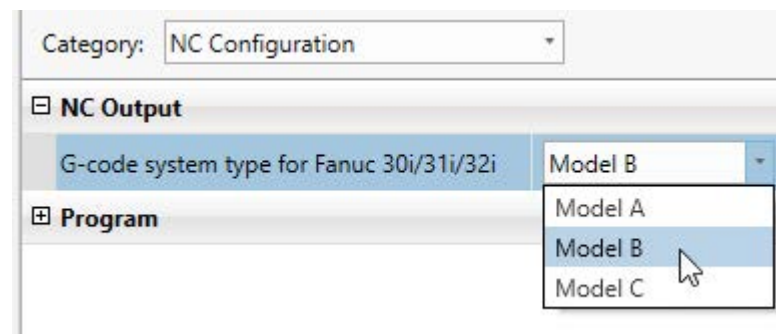
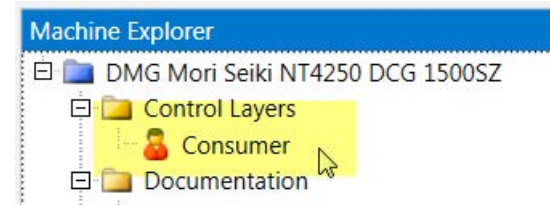


Selecting the G-code system

Fanuc 30/31/32 controls can output three different “flavors” of G-code: A, B, or C. For example, a threading cycle is output as G92 in system A, G78 in system B, and G21 in system C. Your **.machine** file includes a switch that lets you emulate the desired G-code system.

The default setting is **Model B**. Follow these steps if you wish to change this.

1. Double-click the **Consumer** layer.
2. Go to the **NC Configuration** category.
3. Open the **NC Output** group.
4. Select the desired **G-code system type**....
5. Press [**Ctrl+S**] before posting to save your setting.



Resetting the C-axis automatically

Rotary axes that have a signed continuous positioning mode can result in very large angle values. Your **.machine** file lets you reset the C axis at a tool change. This ensures that the C-axis positions for subsequent operations are calculated relative to a base of 0.

Mastercam follows this general outline:

1. Mastercam examines the C-axis angle position as it is output in the **.iof** file. For example, the C-axis position might be 1150 degrees.
2. Mastercam calculates the closest angle position to C0. For example, if the C-axis position is 1150 degrees, the closest



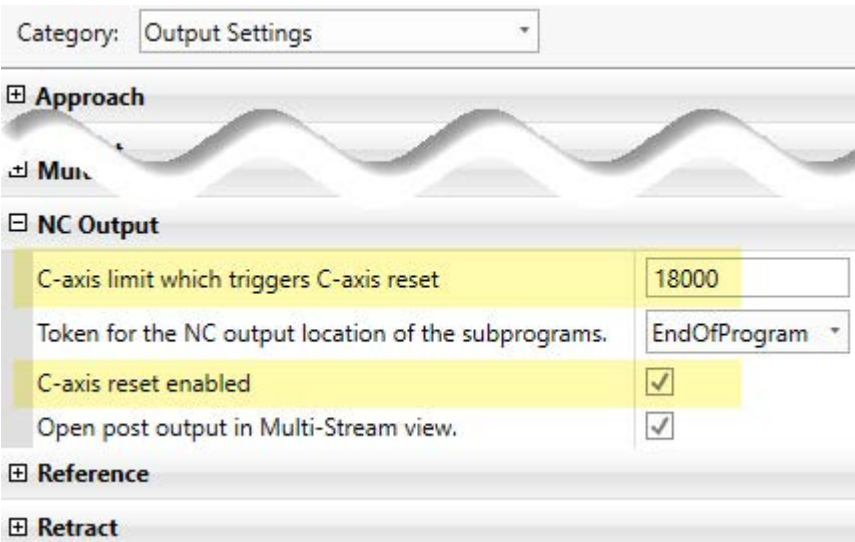
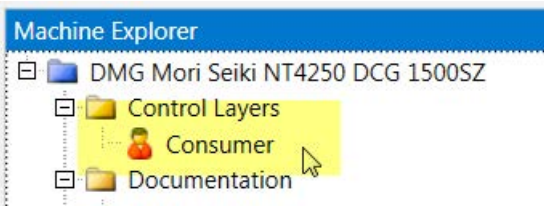
angle position to C0 is 1080 (360×3), so Mastercam stores the value 1080.

3. The next time that the C-axis angle is processed, Mastercam subtracts the stored value from the C-axis value that it reads from the .iof file. For example, if the C-axis position at the end of the approach move of the next operation is C1280, Mastercam subtracts 1080 from it, so that when you output the position you will see C200 in your NC file.



Follow these steps to configure the C-axis reset feature.

1. Double-click the **Consumer** layer.
2. Go to the **Output Settings** category.
3. Open the **NC Output** group.
4. Select **C-axis reset enabled** to activate the feature.
5. Enter the angle that the C axis needs to reach before it is reset.
6. Press [Ctrl+S] before posting to save your setting.

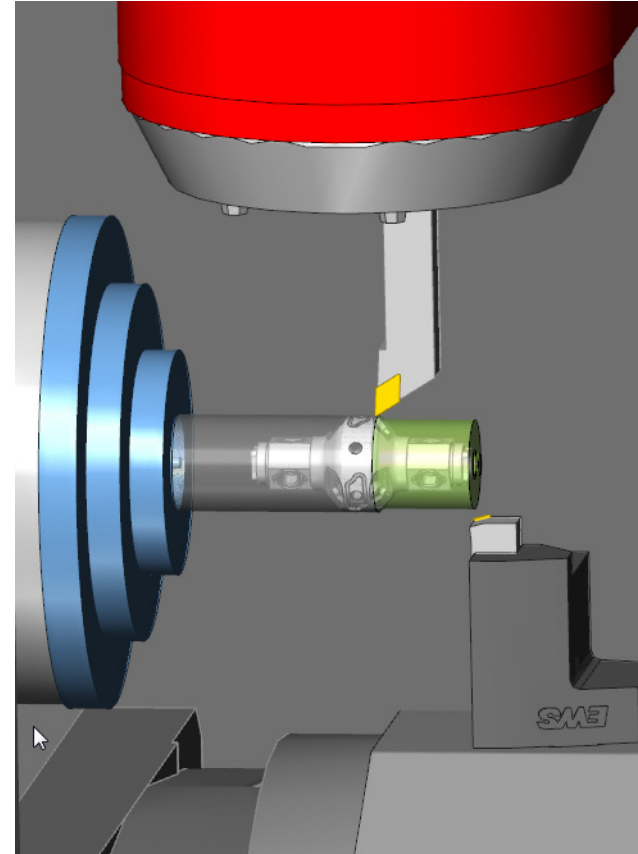




D: Setting up tools

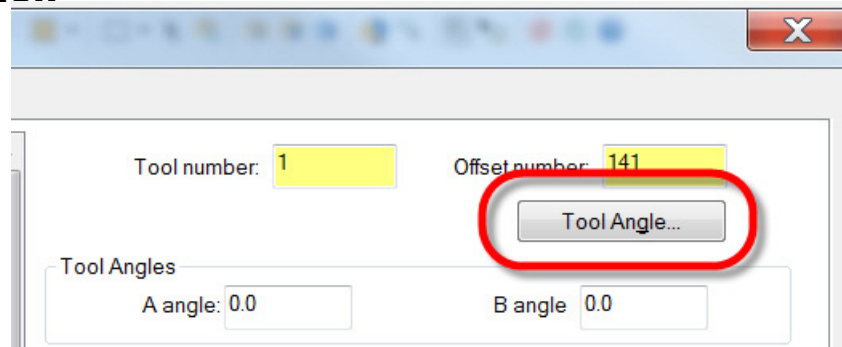
Your DMG Mori NT/NTX/NZX **.machine** file gives you several options for setting up tools that extend Mastercam's tool selection features.

- ❖ Rotating the B-axis head to the proper position
- ❖ Setting the tool number base for B-axis tools
- ❖ Using heavy tools
- ❖ G361/G362 tool change mode
- ❖ Live tool support for the lower turret

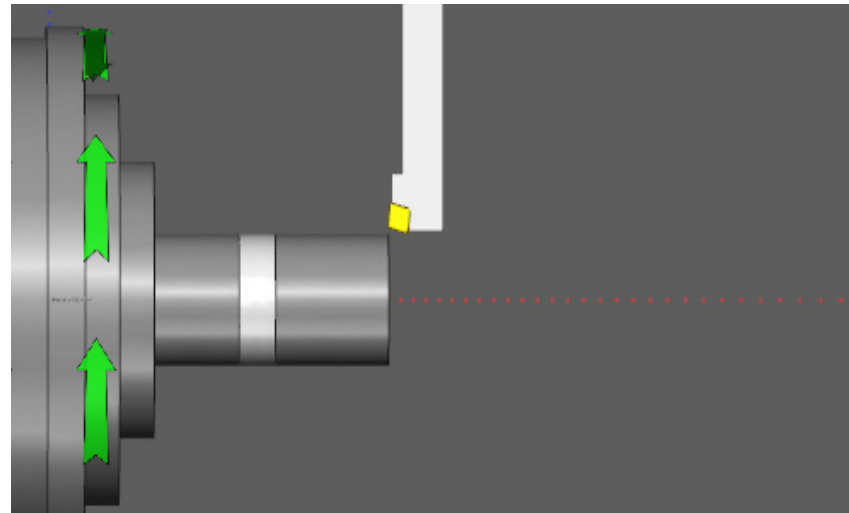


Rotating the B-axis head to the proper position

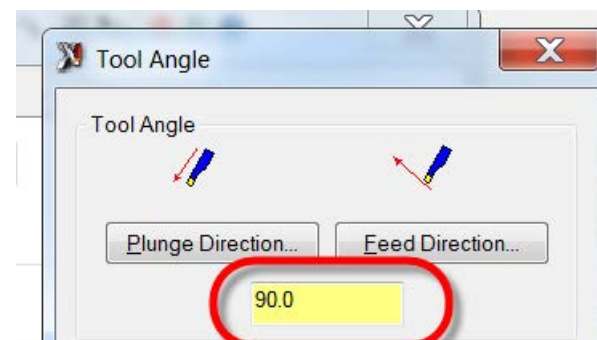
Use the **Tool Angle** button in Mastercam to set both the B-axis angle and the tool orientation so that the tool is in the proper machining position.



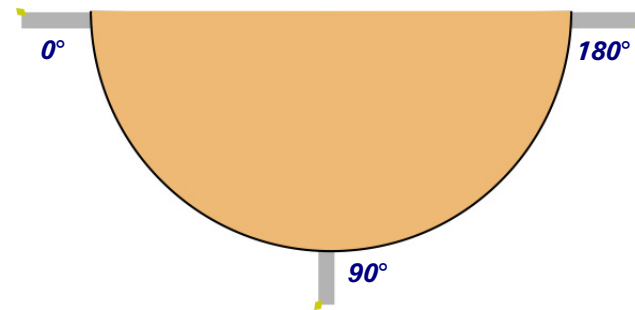
For example, to rotate the B-axis head like in this picture;



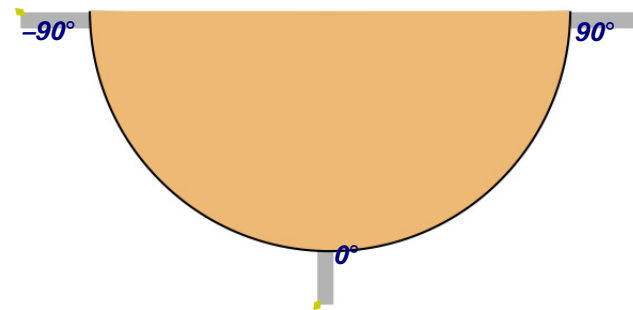
enter a **Tool Angle** of **90**. Use this field to rotate the B-axis head to any angular position. The **Tool Angle** value is typically output in your NC file with the G361 line.



You need to be aware, though, that the Mastercam interface is based on a 0–180 degree scale for tool angle, while the machine uses a –90 thru +90 scale. This means that the values that you enter in the **Tool Angle** field in Mastercam need to be offset by +90 degrees in order for you to get the B value that you expect in your code. For example, if you enter **75** in the **Tool Angle** field, you will see G361 B–15. in your NC code.

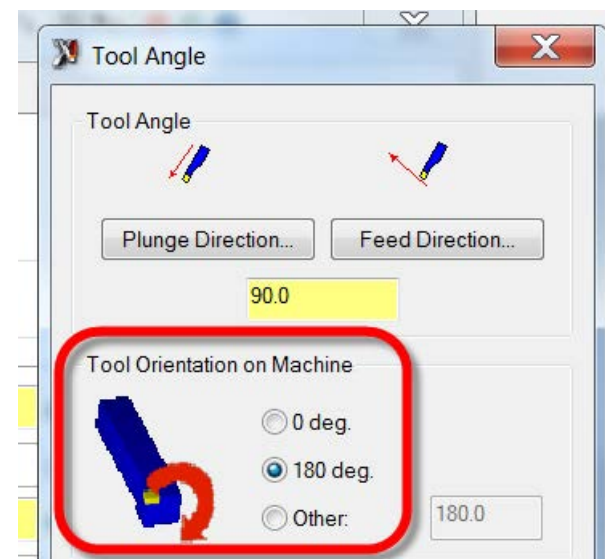


Tool angles in the Mastercam interface



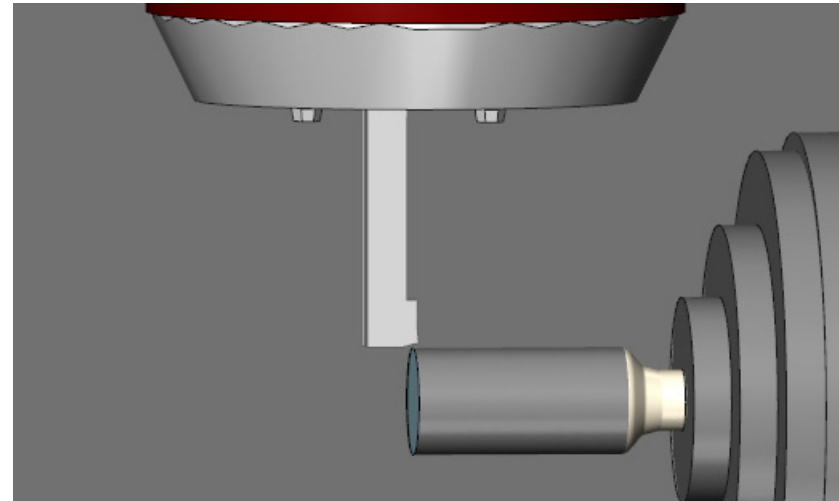
How your DMG Mori uses tool angles

To rotate the tool axis to machine on the right spindle, use the **Tool orientation on Machine** setting.



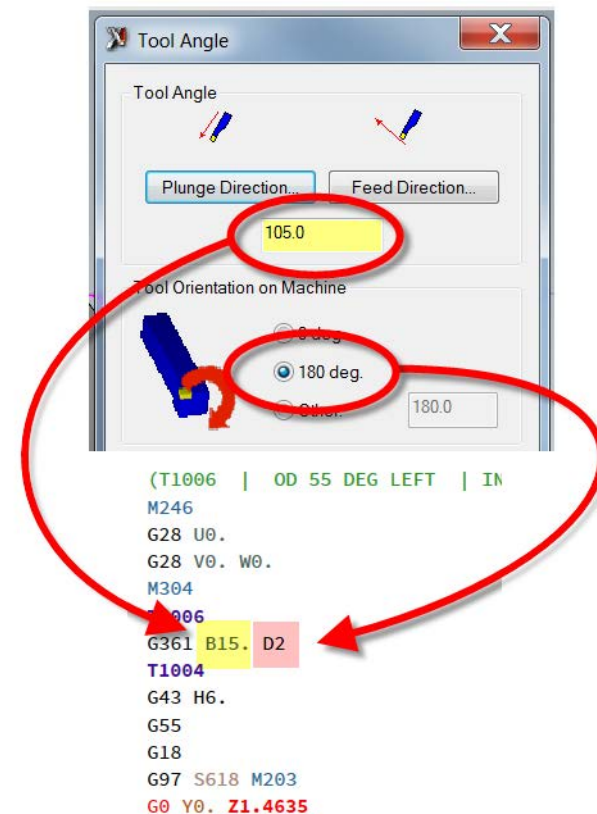
Changing the **Tool orientation** from **0** to **180** rotates the tool axis like in this picture:

This lets you use the **Tool orientation** setting to orient the tool for cutting on either spindle. Compared to the previous picture, you can see that the insert is now facing towards the right spindle.

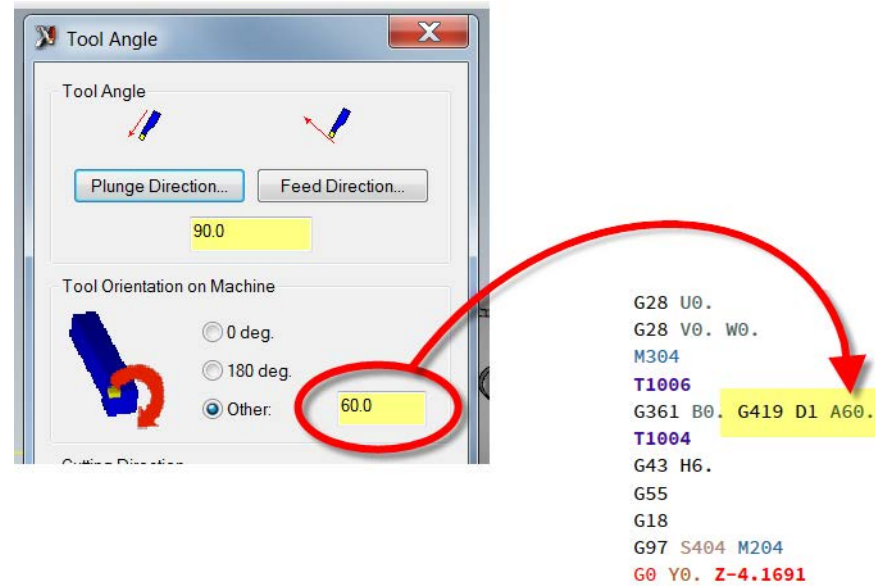


The **Tool orientation** value is typically output as D1 (**0 deg.** option) or D2 (**180 deg.** option) on your G361 line.

The code sample shows how these two settings appear in your output.



Values other than **0 deg.** or **180 deg.** will be output with a G419 D1 A__ code (tool spindle orientation shift) as shown in the picture.

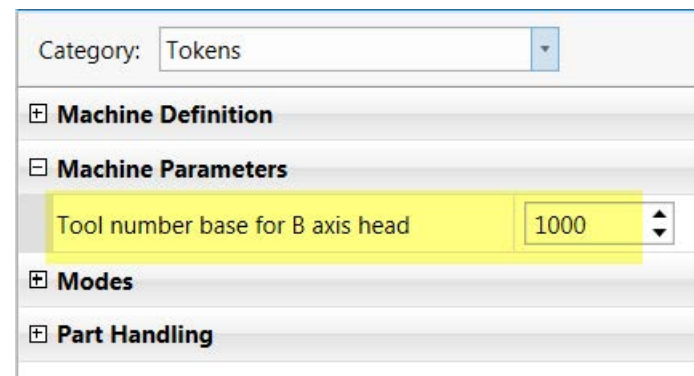
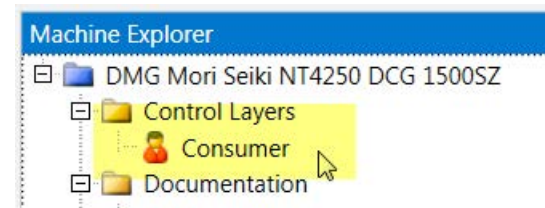


Setting the tool number base for B-axis tools

Your **.machine** file includes an option that lets you control the base value for B-axis tool numbers. For example, the first tool might be numbered 1001 or 2001.

Follow these steps to:

1. Open the DMG Mori NT/NTX/NZX **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.
3. Select the **Tokens** category.
4. Go to the **Machine Parameters** section.
5. Enter the desired **Tool number base** value. The number that you enter here will be added to the first tool, so that a value of 1000 will result in the first tool being numbered 1001.
6. Save the **.machine** file.



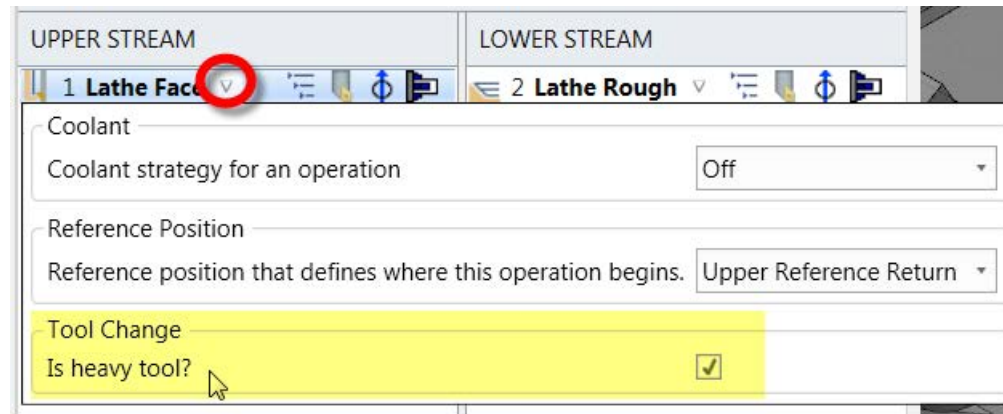
Using heavy tools

Your **.machine** file lets you specify that your selected tool uses a heavy tool setup on your machine. When you select this option, your post will automatically add 9000 to the tool number.

This option is available after you finish programming your toolpaths in Mastercam and are working in the Sync Manager.

Click the little triangle next to the operation name and select the **Is heavy tool?** option.

Note that this option is only valid for **Upper Stream** operations which use tools mounted in the B-axis head.



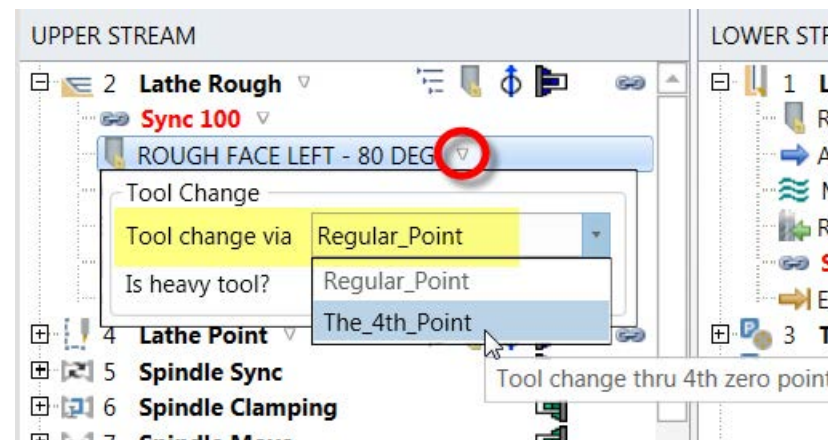
G361/G362 tool change mode

Your DMG Mori **.machine** file includes an option that lets you use G362 mode for upper-stream tool changes instead of G361. G362 tool changes use your machine's fourth zero point instead of the machine zero.

This option is available after you finish programming your toolpaths in Mastercam and are working in the Sync Manager.

Click the little triangle next to the tool name and select the **Tool change via...** option. Select **The_4th_Point** option to output G362 for the tool change, or **Regular_Point** to use G361.

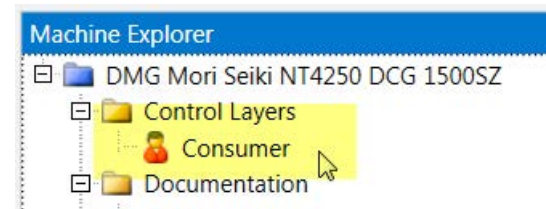
Note that this option is only valid for **Upper Stream** operations which use tools mounted in the B-axis head.



Live tool support for the lower turret

Your DMG Mori NT/NTX/NZX **.machine** file is designed to work with machines whose lower turrets both do and do not support live tooling. If your machine does *not* support live tooling in the lower turret, it is important that you set this option in your **.machine** file. This tells the post to suppress certain Mcodes, such as M45/M245 and M46/M246. Follow these steps:

1. Open the DMG Mori NT/NTX/NZX **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.
3. Select **Category: Tokens**.
4. Go to the **Machine Definition** section.
5. If your machine does *not* support live tooling in the lower turret, clear the **Does lower turret support live tools?** option.
6. Save the **.machine** file.



Category:	Tokens
Machine Definition	
Does lower turret support live tools?	<input type="checkbox"/>
Is there a part catcher?	<input type="checkbox"/>
Has Part Ejector	<input type="checkbox"/>
Machine Parameters	

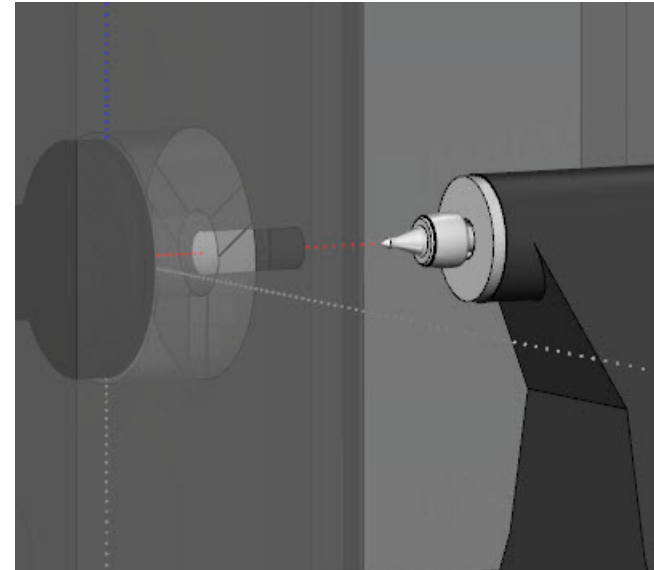


E: Supporting peripheral components

In addition to the standard part handling strategies for pickoff, cutoff, and part transfer operations, your **.machine** file supports a number of other common peripheral functions for DMG Mori NT/NTX/NZX machines.

Beginning with Mastercam 2018, tailstock operations are also supported for **.machine** files that include the proper components.

- ❖ **Using the workpiece push check function (G38)**
- ❖ **Chip conveyor (M200)/door control (M220)**
- ❖ **Part ejector (M47) and part catcher (M73) support**
- ❖ **Tailstock operations**



Using the workpiece push check function (G38)

Your DMG Mori NT/NTX/NZX **.machine** file supports the G38 workpiece push check function for advancing the sub spindle to the grip position.

The workpiece push check function outputs a G38 in the following form:

```
G38 A(J, V)xx Kxx Fxx Qxx
```

where

- the first value is the coordinate position that the subspindle will move to—either an absolute coordinate (A), incremental coordinate (J), or machine coordinate (V). This is calculated automatically by Mastercam based on your other programming information.
- K is the distance that the sub spindle should retract when the part is detected.



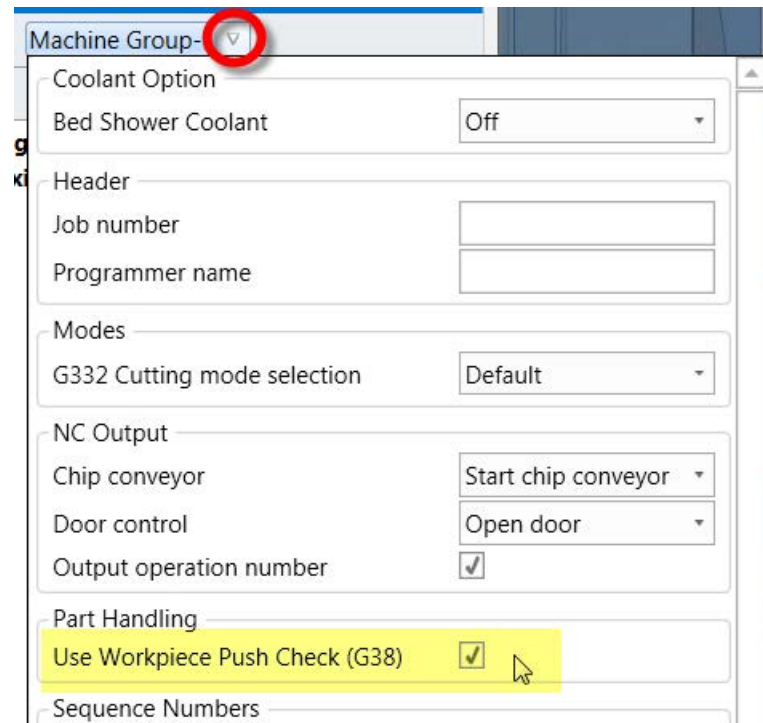
- F is the feedrate of the subspindle. This is preset in the **.transfer** file by your machine builder and is stored in your **.machine** file.
- Q is the tolerance for the part transfer position.

The following sections show you how to configure the workpiece push check function and set your desired values for K and Q.

Turning on workpiece push check mode

Your **.machine** file includes a Sync Manager option that lets you turn on workpiece push check mode for the current part.

1. In the Sync Manager, click the little triangle next to the machine group name.
2. Select the **Use Workpiece Push Check...** option.
3. Save the IOF file before posting.

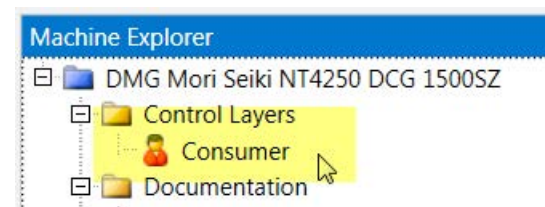


Configuring workpiece push check mode

Follow these steps to configure the workpiece push check function and to tell Mastercam if it will be activated by default for part transfer operations.

Follow these steps:

1. Open the DMG Mori NT/NTX/NZX **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.



3. Select the **Tokens** category.
4. Go to the **Part Handling** section.
5. Enter the desired Q and K values in the appropriate fields.
 - Enter the tolerance value Q in the **Grip Length Tolerance** field.
 - Enter the retract distance K in the **Sub-spindle retract...** field.

Category: Tokens

- Machine Definition
- Machine Parameters
- Modes
- Part Handling

Grip Length Tolerance	Inch: 0.079	Metric: 2
Sub-spindle retract amount when push of part detected	Inch: 0.001	Metric: 0.02

6. Select the **Output Settings** category.
7. Go to the **Part Handling** section.
8. Select the **Use Workpiece Push Check...** option to have workpiece push check mode enabled by default for part transfer operations.
9. Save the **.machine** file.

Category: Output Settings

- Approach
- Coolant
- Part Handling

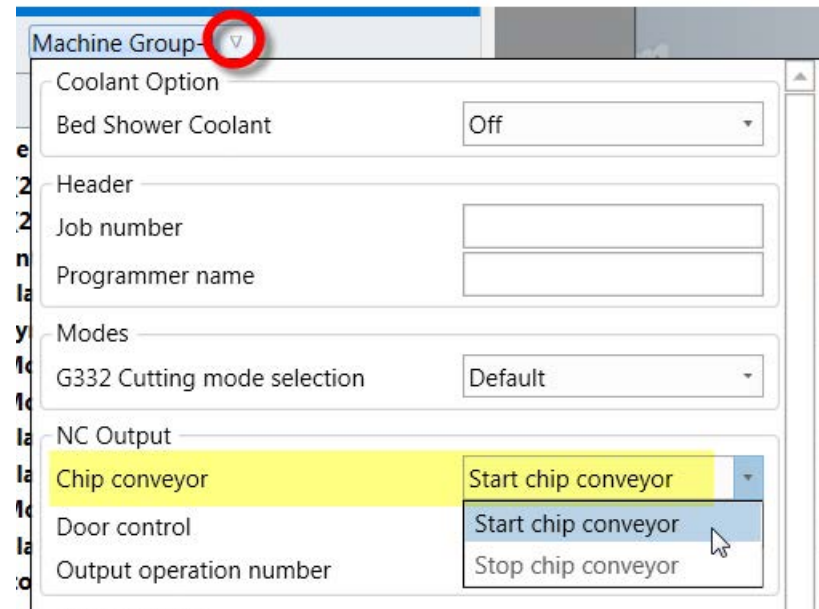
Use Workpiece Push Check (G38)	<input checked="" type="checkbox"/>
--------------------------------	-------------------------------------
- Program



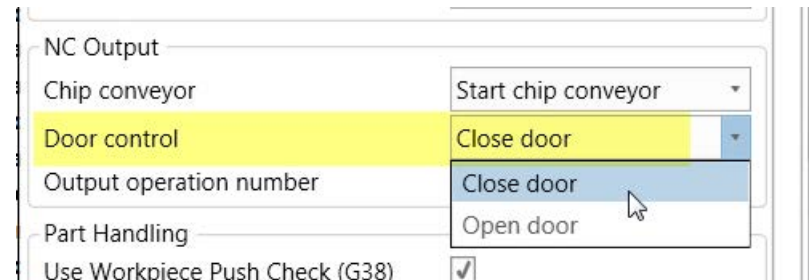
Chip conveyor (M200)/door control (M220)

Typically, most machines are set up so that when the NC program begins running, the door closes first and then the chip conveyor starts up. However, this can be changed with settings on your control. Your **.machine** file includes settings that you can use so that your NC program includes the proper M200/M201 (chip conveyor) or M220/M221 (door open/close) commands to coordinate with the settings on your control.

- To include the chip conveyor commands in your NC program, select the **Start chip conveyor** option from the **Machine Group** options in the Sync Manager. This will result in M200 at the start of your program, and M201 at the end.
- Select the **Stop chip conveyor** option to leave out the chip conveyor commands from your program.

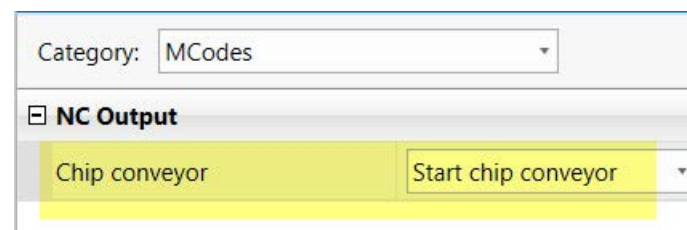
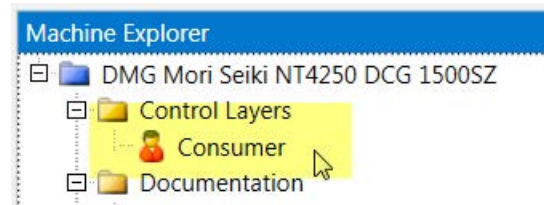


- To include the door open and close commands in your NC program, select the **Close door** option from the **Machine Group** options in the Sync Manager. This will result in M221 (close the door) at the start of your program, and M220 at the end (open the door).
- Select the **Open** option to leave out the door open/close commands from your program.



You can also change the default chip conveyor option in your **.machine** file. Follow these steps::

1. Open the DMG Mori NT/NTX/NZX **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.
3. Select the **Mcodes** category.
4. Select the desired **Chip conveyor** option.
5. Save the **.machine** file.

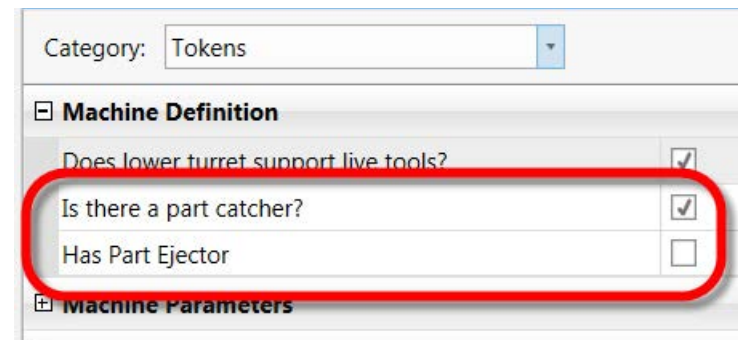
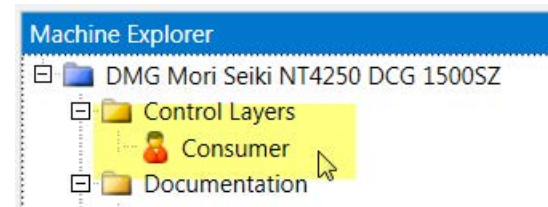


Part ejector (M47) and part catcher (M73) support

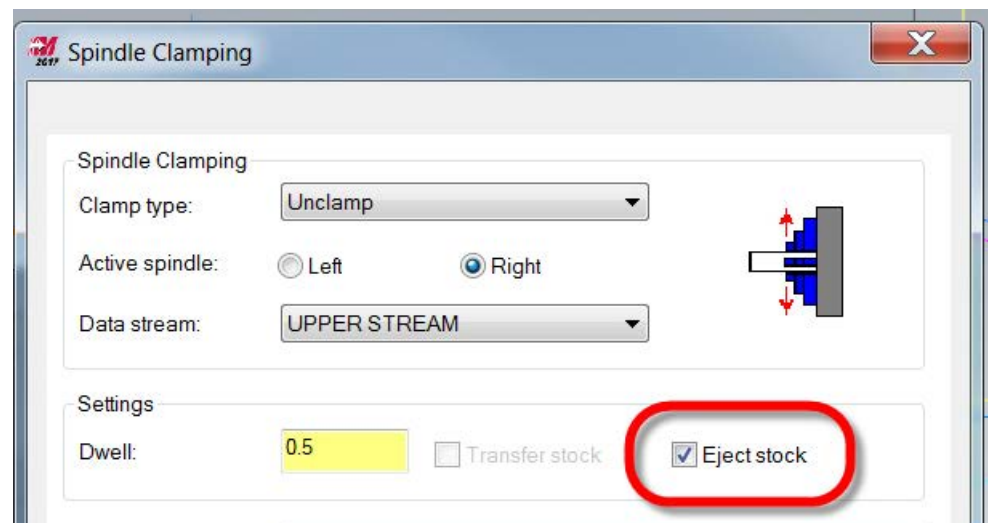
Your **.machine** file includes two options that let you enable support for part catcher (M73/M74) and part ejector (M47) peripheral components.

Follow these steps to turn on or turn off support for these devices:

1. Open the DMG Mori NT/NTX/NZX **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.
3. Select the **Tokens** category.
4. Go to the **Machine Definition** section.
5. If your machine has either a part catcher or part ejector installed, select the proper option.
6. Save the **.machine** file.



The M47 is triggered by the **Eject stock** option for spindle clamping operations inside Mastercam.



Tailstock operations

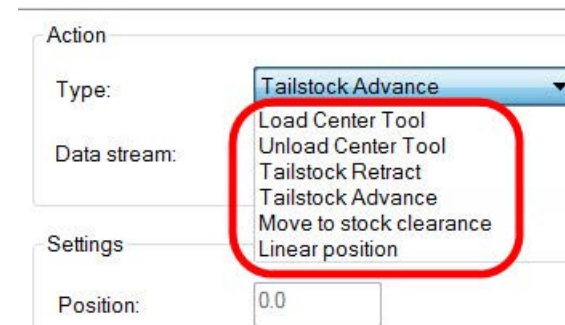
Tailstock output for the DMG Mori machines consists of a G325 command that configures the tailstock operating mode plus individual Mcodes to move the tailstock.

The DMG Mori NT/NTX/NZX .machine file supports the following tailstock events:

- Move to stock clearance. This will output the G325 line.
- Tailstock advance. This outputs the following Mcodes:
 - ♦ M625/M626 for workpiece selection
 - ♦ M434/M435 for thrust selection
 - ♦ M25 tailstock advance
- Load center tool. This will typically output an M01 so that the operator can load the tool.
- Tailstock retract. This will output the M26.

The following tailstock events are not supported:

- Linear position
- Unload center
- Quill advance/retract



The screenshot shows a software interface with a section titled 'Action'. Inside this section, there is a 'Type:' label followed by a dropdown menu. The dropdown menu is currently set to 'Tailstock Advance'. Below the dropdown, there is a 'Data stream:' label followed by a text input field. To the right of the dropdown menu, there is a list of options: 'Load Center Tool', 'Unload Center Tool', 'Tailstock Retract', 'Tailstock Advance', 'Move to stock clearance', and 'Linear position'. A red circle is drawn around this list of options. Below the 'Data stream:' field, there is a 'Settings' section with a 'Position:' label followed by a text input field containing the value '0.0'.



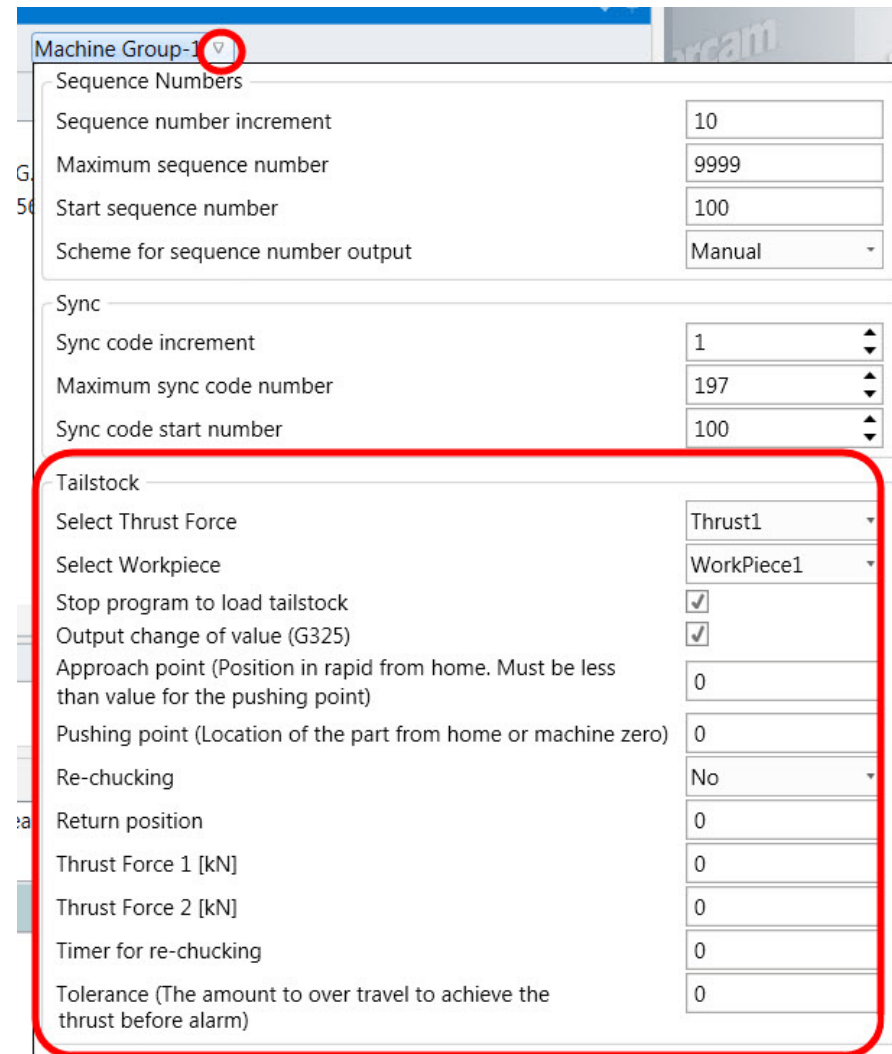
Use machine group options in the Sync Manager to configure the G325 line. The G325 is output with the following parameters

G325 W_ T_ U_ A_ B_ C_ Q_ R_ S_

- W is the workpiece selection, W1 or W2. Use the **Select Workpiece** option to set this.
- Use the **Select Thrust Force** option to select T0 (**Thrust1**) or U0 (**Thrust2**) output.
 - ◆ Use the **Thrust Force 1** and **Thrust Force 2** fields to enter the actual force values for each option. Enter the value in kiloNewtons (kN). (If you know the value in foot-pounds, multiply by 4.448 to convert to kN.)
- Use the **Pushing point** field to set the A value.
- Use the **Approach point** field to set the B value.
- Use the **Return position** field to set the C value.
- Use the **Tolerance** field to set the Q value.
- Use the **Re-chucking** option to set the R value. Select **Yes** to re-chuck the part after the tailstock advance (R1).
- Use the **Timer for re-chucking** field to set the S value. Enter 0 if the **Re-chucking** option is set to **No**.

The **Stop program to load tailstock** option results in an M01 being output for **Load center tool** tailstock operations. If this option is not selected, there will be no NC output for a **Load center tool** tailstock operation.

De-select the **Output change of value (G325)** option to suppress output of the G325 line. The individual Mcodes will still be output, however.



Machine Group-1	
Sequence Numbers	
Sequence number increment	10
Maximum sequence number	9999
Start sequence number	100
Scheme for sequence number output	Manual
Sync	
Sync code increment	1
Maximum sync code number	197
Sync code start number	100
Tailstock	
Select Thrust Force	Thrust1
Select Workpiece	WorkPiece1
Stop program to load tailstock	<input checked="" type="checkbox"/>
Output change of value (G325)	<input checked="" type="checkbox"/>
Approach point (Position in rapid from home. Must be less than value for the pushing point)	0
Pushing point (Location of the part from home or machine zero)	0
Re-chucking	No
Return position	0
Thrust Force 1 [kN]	0
Thrust Force 2 [kN]	0
Timer for re-chucking	0
Tolerance (The amount to over travel to achieve the thrust before alarm)	0



F: Approach and retract moves

Your DMG Mori NT/NTX/NZX **.machine** file gives you several options for configuring approach and retract moves:

- You can select the start point for the approach move and the end point of the retract. These are specified with reference positions.
- You can choose the type of motion for each: dogleg (Z-first or X-first) or a direct move.

Selecting reference positions

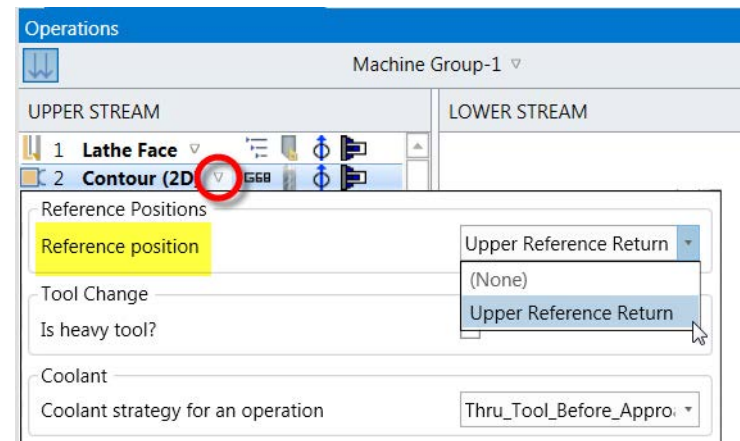
Your **.machine** file includes a set of reference positions that have been defined specifically for your individual machine. Use these to tell Mastercam where you want to start and end each operation. For example, you might—or might not—want to move all the way to the home position between operations. Sync Manager reference positions let you determine exactly where you want each turret to go between operations.

- Select specific positions for the start and end of each operation in the Sync Manager.
- You can also define additional, new reference positions. Do this in the Job Setup inside Mastercam. Do this if your particular part setup requires different reference locations than are already defined in your **.machine** file—for example, to accommodate special fixturing, an unusual part shape, etc.



Setting the start point for an operation

To set the start point for an operation, select the desired **Reference position** in the Sync Manager. Click the small triangle next to the operation name in the Sync Manager, and select the location from the list.



Upper Reference Return and **Lower Reference Return** positions are typically output with the G28 line.

```

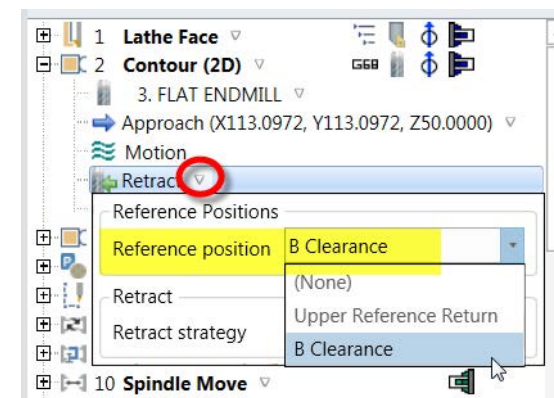
N120
(OPTION # 3)
(T1001 | 3. FLAT ENDMILL | DIA. - 3)
G28 U0.
G28 V0. W0.
G28 H0.
M303
M45
T1001
G361 B0 D0

```

Setting the end point

To set the end point for an operation, select the desired **Reference position** from the operation's **Retract** branch. You can choose to make the end position of one operation the same location as the start point of the next operation by choosing the same reference position.

The example shown here displays a user-defined reference position. You can create such positions if, for example, you don't want to retract all the way to the home position between operations.



The highlighted lines in this code sample show how this might appear in your NC program. You can see that instead of a G28 move to the home position, there is a move to the user-defined reference position.

Selecting a reference position of None

Selecting **None** for a reference position means that there will simply be no output where the reference position is typically output. For example, the code at right shows what happens if **None** is selected as the reference position for operation #2.

Note that you cannot select **None** for the start point of the first toolpath.

Reference positions and null tool changes

If you have consecutive operations in the same stream that use the same tool and tool orientation, Mastercam will typically not output a tool change between the operations. When this happens, Mastercam will not display the reference position option for the **Retract** of the first operation, or for the start of the second operation. You can see in this picture that the **Reference position** option is not available for the start of this operation.

In these instances, you can force the **Reference position** option to be available by selecting the **Force tool change** option inside Mastercam. This might be useful if you have defined custom reference positions that you want to use as clearance positions between such operation.

```

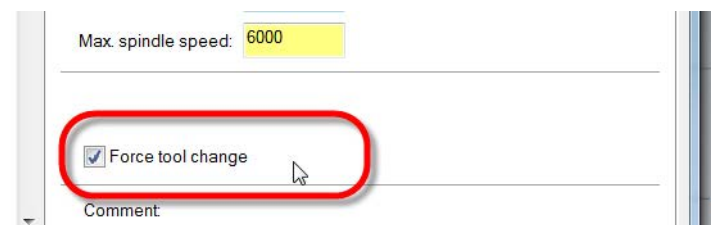
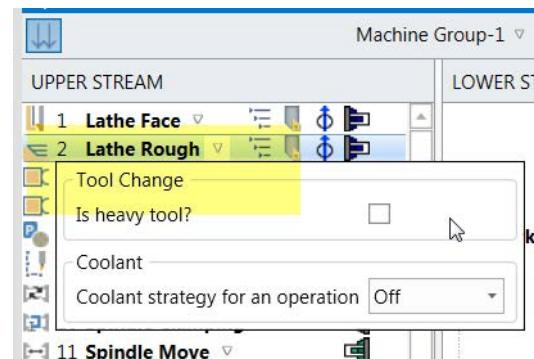
118 X-.2376 Z-2. I.0156 R0.
119 G1 X1.4846
120 G2 X1.5158 Z-1.9852 I0. R.0156
121 G1 X1.7222 Z-.0164
122 X1.8636 Z.0543
123 G0 G53 X-25.3071 Y0.
124 G53 Z-2.6614 M05
125 M01

```

```

54 (Operation # 2)
55 (2 | OD 55 deg Left | )
56 NP002
57 M321
58 MT=101
59 TL=020202 BT=0 BA=90. G52 M602
60 NT002
61 G18
62 G97 S491 M04 M41
63 G95 G00 Z.61
64 X1.5567

```

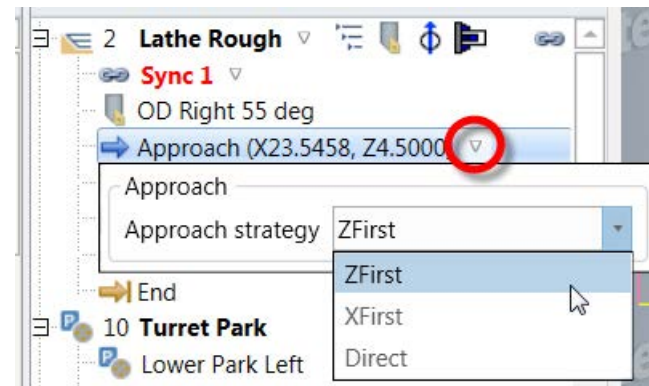


Setting the type of approach/retract motion

For each approach and retract move, you can select the following motion:

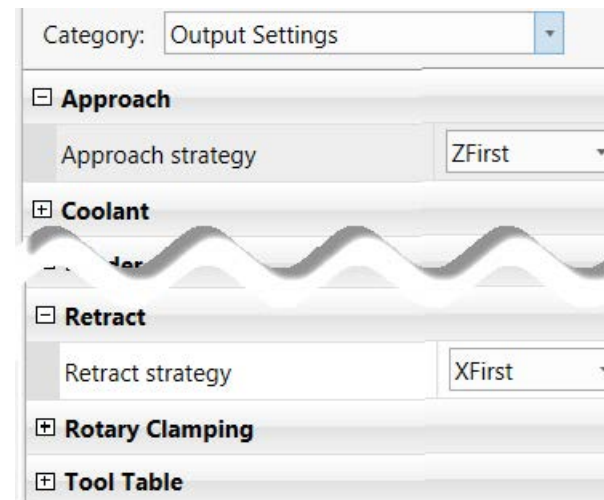
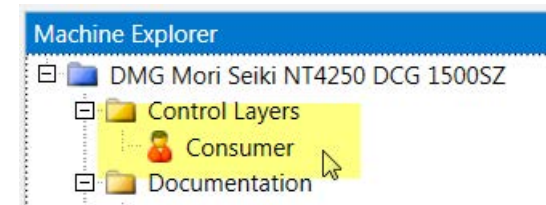
- X-first
- Z-first
- Direct (interpolated) move

Click the small triangle next to the **Approach** or **Retract** node in the tree.



You can also change the default selection in the **.machine** file.

1. Double-click the **Consumer** layer.
2. Go to the **Output Settings** category.
3. Open the **Approach** and **Retract** group.
4. Select the desired strategy for each.
5. Press **Ctrl+S** to save your settings.

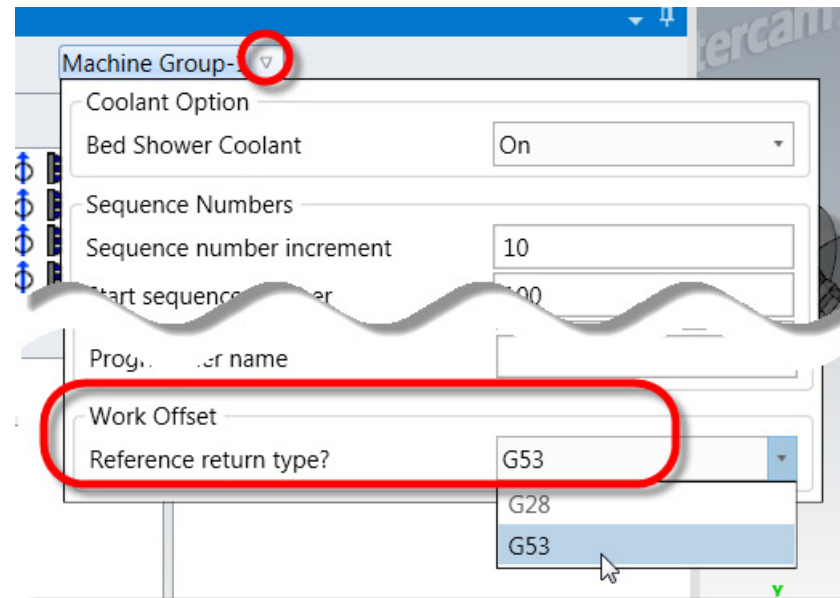


Forcing G53 output

If you like, you can choose to force out a G53 instead of G28 for your home position moves. Follow these steps.

1. Click the small triangle next to your machine group name in the Sync Manager.
2. Select the desired **G53** or **G28** option.
3. Press **[Ctrl+S]** to save the setting back to your Mastercam part file.

This option affects all the operations in your part.



G: Configuring turret park commands

Your DMG Mori NT/NTX/NZX **.machine** file gives you a number of options that you can use to configure your turret park output.

- G28:

```
(OPERATION # 11)
(PARK UPPER TURRET OVER LEFT SPINDLE)
M01 (PARK UPPER HEAD TO LEFT - CHECK FOR INTERFERENCE)
G00 G28 U0. V0.
G53 Z7.63
```

- G53. Additional options let you fine-tune the X and Z coordinates or use a control parameter for the position.
- G30. Additional options let you select from several different output patterns.

Follow these steps:

1. Double-click the **Consumer** layer.
2. Go to the **Output Settings** category.
3. Open the **Reference** group.
4. Select the desired **Output Pattern** for each position.

If you select G28 or one of the G30 patterns, you can ignore the other settings in this section. Continue with the next step to configure the X and Z coordinates for G53 output.

5. Each DMG Mori NT/NTX/NZX **.machine** file includes default Z values for the turret park positions. You can override these with your own values or parameter.
 - Enter the appropriate value or code for each turret park position in the **Z Code** fields. You can enter any string, including a control parameter or coordinate position.
 - If you enter a coordinate position, you must include **Z** with your value. Mastercam will output the string on the G53 line exactly as you enter it here.



- If this is a coordinate value, enter it in whichever units (inch or metric) you use most often. If you create a part using the other units, you will need to re-enter these values in the proper units. The post will not scale these values if you switch units in your part.
6. Your machine tool can be configured so that the machine origin is in either of two places:
- on the spindle center line at the face of the spindle
 - the home position of the X and Z axes

In your Mastercam machine definition, the machine origin is located at the center line of the spindle face. This means that for your G53 output, you need to specify how you want the post to calculate the X and Z coordinates.

- a. For the X coordinate, select the **G53 X0 Position** option. You can choose X0 to be either on the center line or at the maximum X position.

The following picture compares the two options. The **At the maximum X** option is on the left.

```
(OPERATION # 9)
(PARK UPPER TURRET OVER LEFT SPINDLE)
M01 (PARK UPPER HEAD TO LEFT - CHECK FOR INTERFERENCE)
⇒ G00 G53 X0. Y0.
G53 Z5.5118
```

Z Code For Upper Turret Park Right For G53	
G53 X0 Position	At the maximum X
G53 Z0 Position	On the center line
Lower Turret Park Left Output Pattern	At the maximum X
G53	

```
(OPERATION # 9)
(PARK UPPER TURRET OVER LEFT SPINDLE)
M01 (PARK UPPER HEAD TO LEFT - CHECK FOR INTERFERENCE)
⇒ G00 G53 X32.2047 Y0.
G53 Z5.5118
```

- b. For the Z coordinate, select the **G53 Z0 Position** option. You can choose Z0 to be either at the left spindle face or at the Z-axis home position.

Z Code For Upper Turret Park Right For G53	
G53 X0 Position	At the maximum X
G53 Z0 Position	On the left
Lower Turret Park Left Output Pattern	On the left
Upper Turret Park Left Output Pattern	Same as Z-home
G53	



The following picture compares the two options. The **Same as Z-home** option is on the left.



7. Press [**Ctrl+S**] to save your settings.

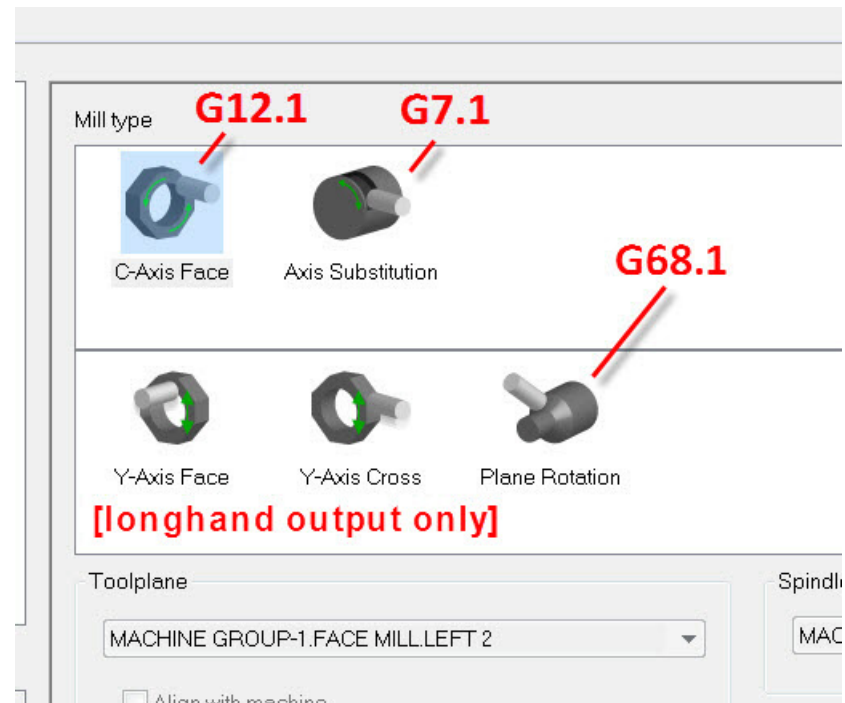


H: Milling cycles and modes

Your DMG Mori NT/NTX/NZX **.machine** file supports the following milling cycles:

- ❖ **Polar (G12.1) and cylindrical (G7.1) interpolation**
- ❖ **Tilted-plane machining cycle (G68.1)**

The cycles are automatically keyed to the milling setup types on the **Setup** page for mill operations. Just select the proper application icon when programming the toolpath in Mastercam, and the appropriate cycle will be activated in your post



This section also contains information about configuring other DMG Mori NT/NTX/NZX machining modes:

- ❖ **Selecting the cutting mode (G332)**
- ❖ **Radius/diameter mode (G10.9)**
- ❖ **Contour control modes (G05, G08)**

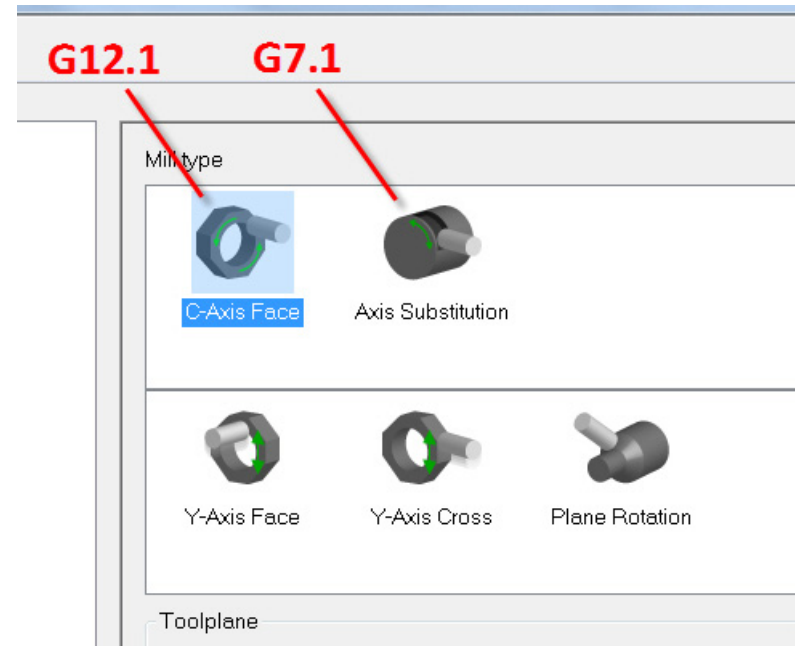


Polar (G12.1) and cylindrical (G7.1) interpolation

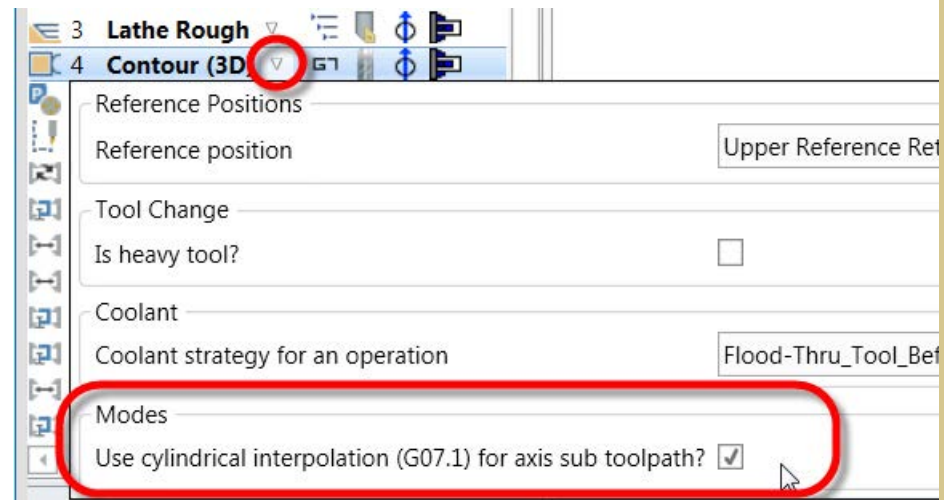
Your DMG Mori NT/NTX/NZX .machine file supports both polar (G12.1) and cylindrical (G7.1) interpolation cycles.

Follow this general workflow for programming these cycles.

1. The cycles are keyed to specific setup types and are automatically enabled when you select **C-axis Face** or **Axis Substitution** on the **Setup** page for milling operations.
2. For either type of operation, the **Toolplane** is automatically selected. These planes were created for you when you completed the **Job Setup** process. Each plane automatically sets the proper tool orientation for your part.



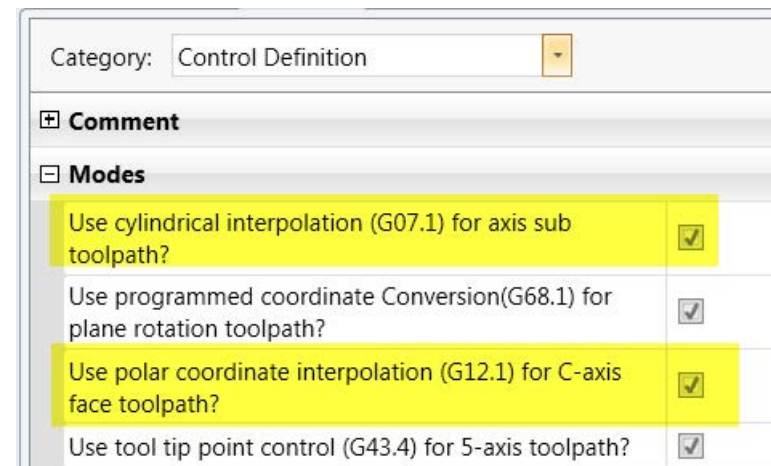
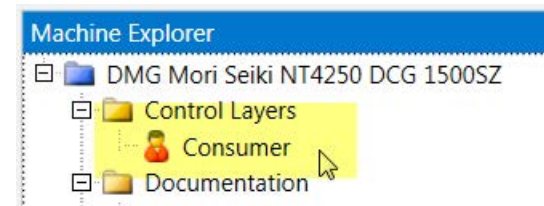
3. For each operation, choose whether or not to output the G7.1/G12.1, or longhand output. Click the little triangle next to the operation and select the desired option.
4. Press **[Ctrl+S]** to save your setting.



Default settings for G7.1 and G12.1

Choose whether the default output mode will be the G7.1/G12.1 cycle or longhand output. Follow these steps:

1. Open the DMG Mori NT/NTX/NZX **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.
3. Select **Category: Control Definition**.
4. Go to the **Modes** section.
5. For each cycle, choose the default output mode.
If you do not select the checkbox, you will get longhand output.
6. Save the **.machine** file when you are done.



Tilted-plane machining cycle (G68.1)

Your DMG Mori NT/NTX/NZX **.machine** file supports G68.1 coordinate conversion (tilted-plane) machining cycle.

- Configure the default output settings in the **.machine** file.
- Enable or disable the cycle for each operation.

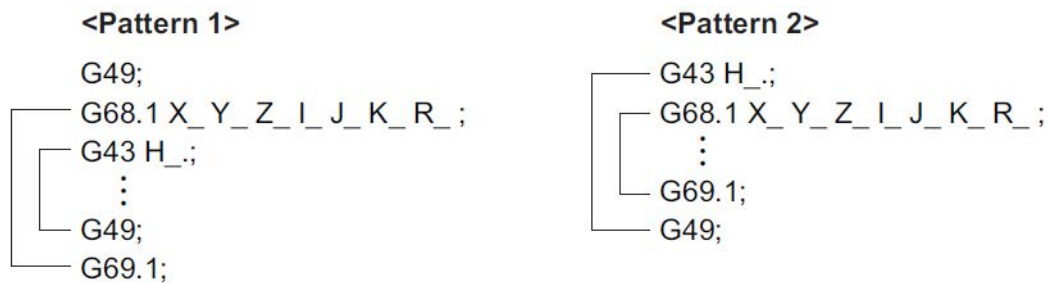
You can also choose to output multiaxis operations with a G68.1 command indexed to a fixed B-axis position. See “Using G68.1 instead of TCP” on page 57.

Default output settings for G68.1

In the **.machine** file, make the following default selections:

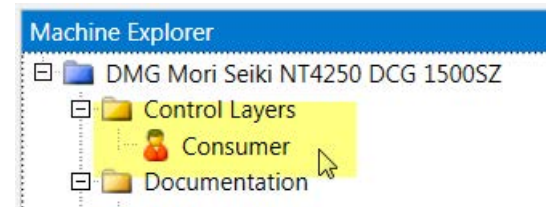
- whether to output the G68.1 cycle or use long-hand output.
- choice of output Pattern 1 or Pattern 2.

Output patterns 1 or 2 control how the G68.1 on/off commands will be output relative to the length offset G43/G49 commands. With Pattern 1, the offset commands are output inside the G68.1/G69.1, while with Pattern 2 they are outside the cycle:

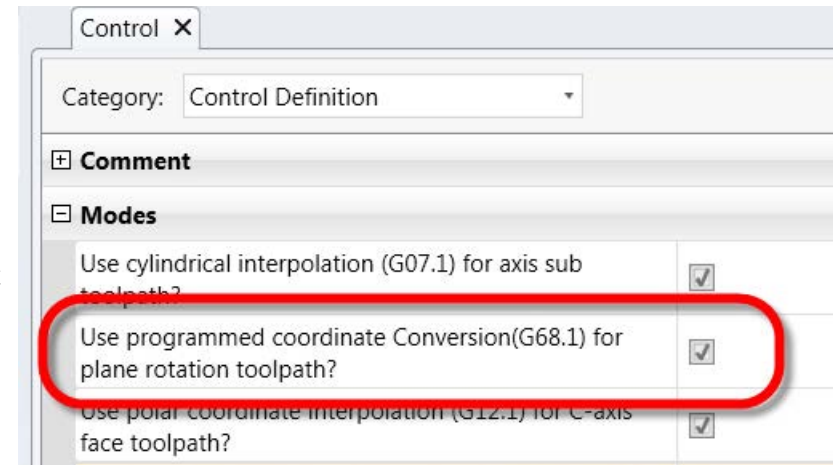


Follow these steps:

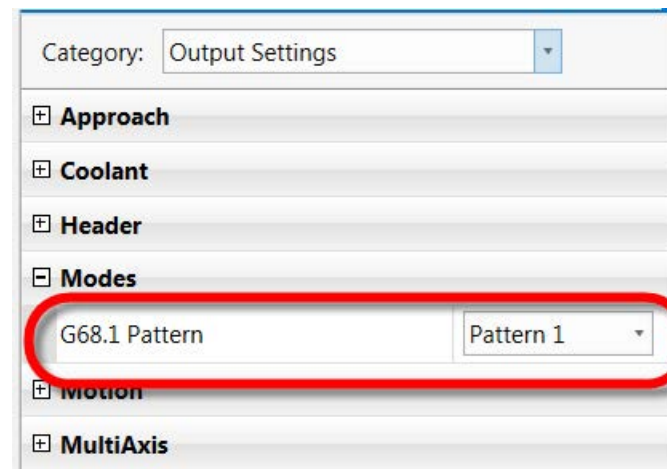
1. Open the DMG Mori NT/NTX/NZX **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.



3. Select **Category: Control Definition**.
4. Go to the **Modes** section.
5. Select the **Use programmed coordinate conversion...** option to output the G68.1 by default. If you do not select the checkbox, you will get longhand output.



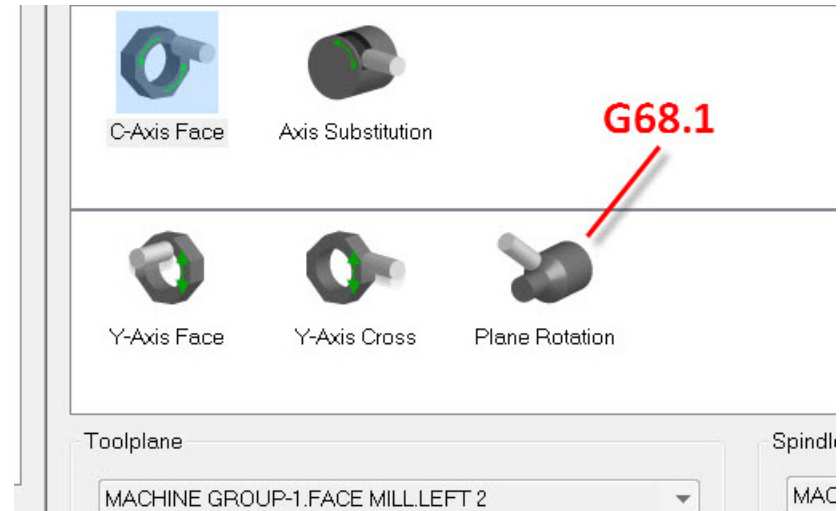
6. Select **Category: Output Settings**.
7. Go to the **Modes** section.
8. Select the desired **G68.1 Pattern**.
9. Save the **.machine** file.



Programming G68.1 for each operation

Follow this general workflow for programming G68.1 cycles for each operation.

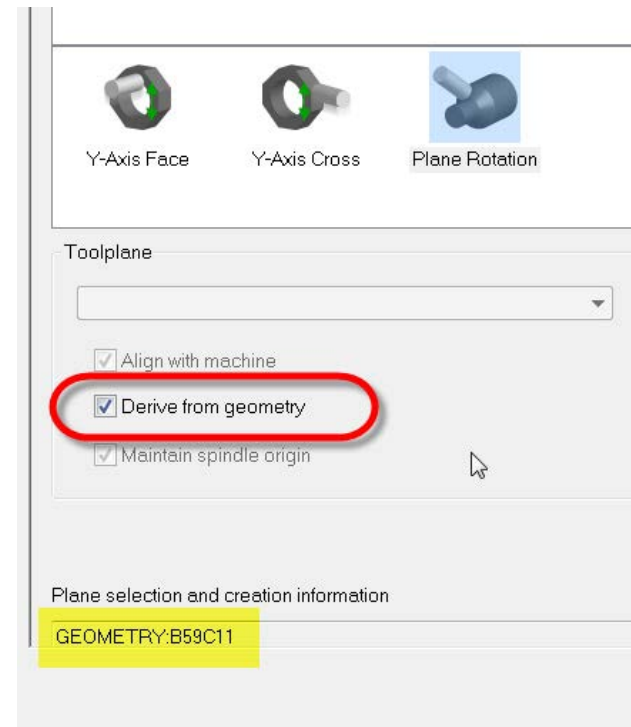
1. Select **Plane Rotation** on the **Setup** page for milling operations. This is keyed to the G68.1 cycle.



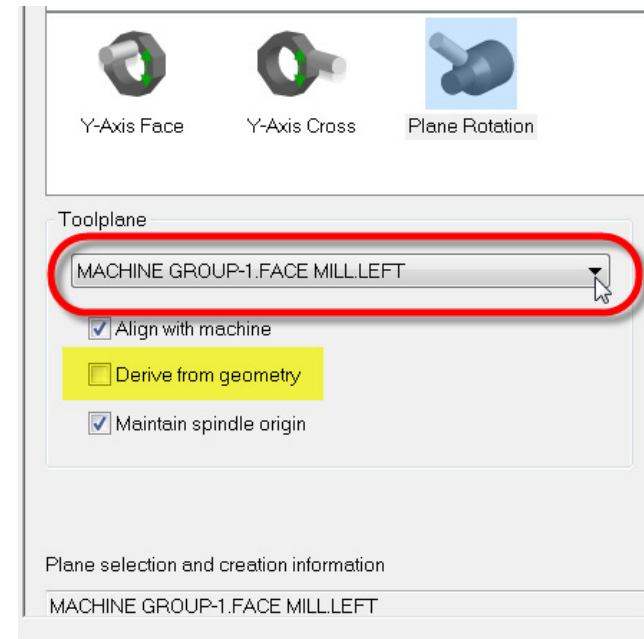
2. Tell Mastercam how to establish the toolplane.
 - For most applications, you will want Mastercam to automatically create a toolplane based on the selected geometry.
 - Select the **Derive from geometry** option to do this.

The name for the new plane is displayed at the bottom of the dialog box; Mastercam tries to create a name that describes its rotation.

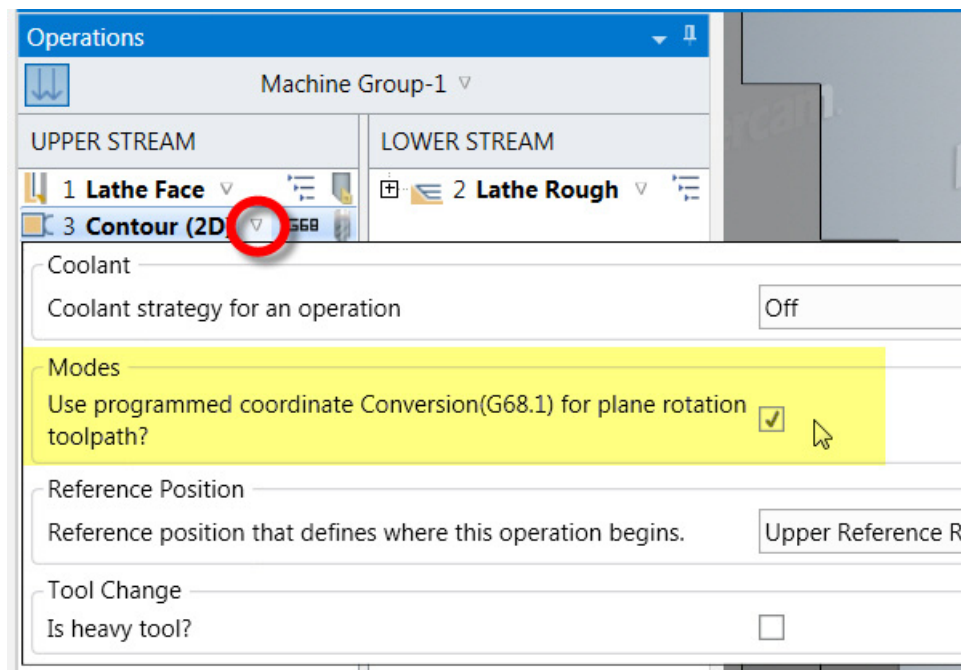
NOTE: The **Derive from geometry** option will only be available if the selected geometry defines a plane. For example, if the chained geometry is only a single line, the option will be grayed out because you cannot specify a plane based on only a single line.



3. You can also use a predefined toolplane instead of creating a new one from the geometry. To do this, clear the **Derive from geometry** option, and select the **Toolplane** from the list.
 - Do this if you want to face mill or cross mill but you want to force G68.1 instead of using G17/G18/G19 plane selection.



4. After you load the part in the Sync Manager, you can choose whether to output the G68.1, or use longhand output. Click the little triangle next to the operation and select the **Use programmed coordinate conversion...** option to output the G68.1.
5. For each operation, the default setting comes from the **.machine** file. If you change it, make sure you press **[Ctrl+S]** to save the new setting back to your Mastercam part file.



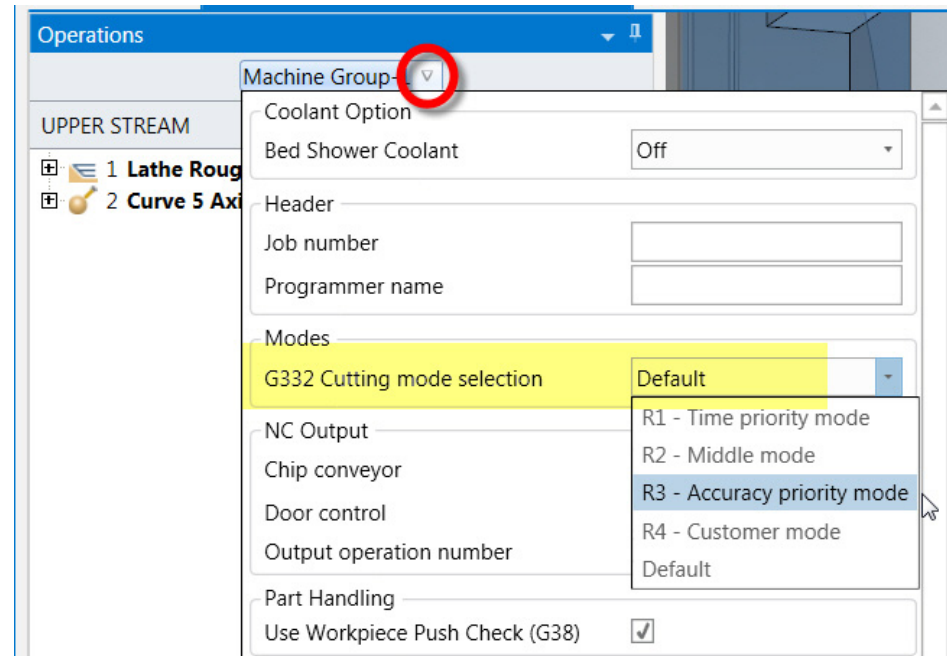
Selecting the cutting mode (G332)

Select the G332 cutting precision mode that best matches your machining application. Select from the following values:

- **R1** = Time priority mode. This results in the shortest machining time, but roughest part.
- **R2** = Middle mode. This is a semi-roughing mode that results in shorter machining time than R3, but greater accuracy than R1.
- **R3** = Accuracy priority mode. This results in the most accurate toolpath and best surface finish, but longest machining time. This is the recommended setting for most applications.
- **R4** = Custom mode.
- **Default** = no G332 will be output in the NC file.

The G332 command is output near the top of your NC file and applies to the entire part. This means that the option is available as part of the **Machine Group** options in the Sync Manager.

Click the small red triangle next to the machine group name and select the desired **G332 Cutting mode**.

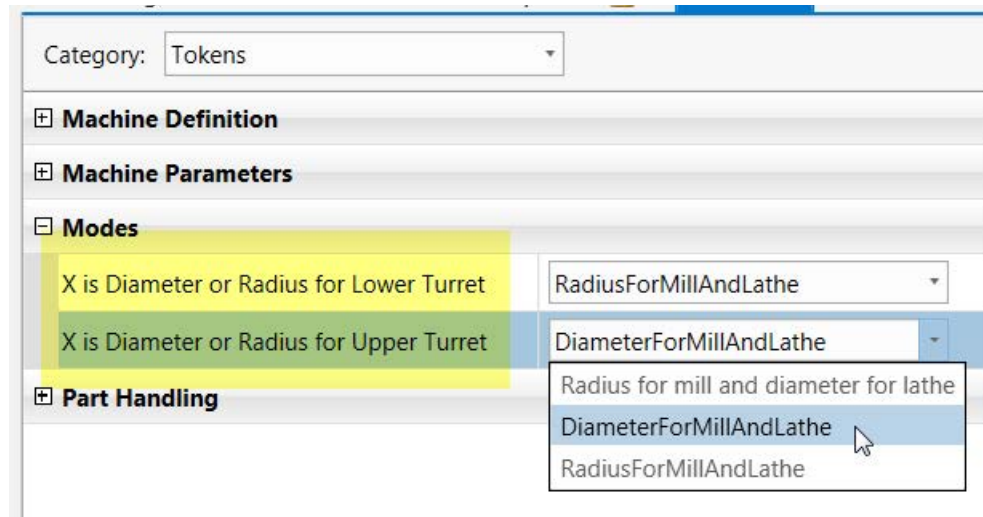
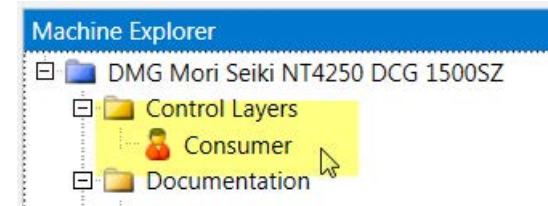


Radius/diameter mode (G10.9)

Your DMG Mori NT/NTX/NZX **.machine** file lets you select diameter (G10.9 X1) or radius (G10.9 X0) mode for X-axis output. You can set this individually for each turret. In addition, you can filter the output choices for mill or lathe operations.

Follow these steps:

1. Open the **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.
3. Select **Category: Tokens**.
4. Go to the **Modes** section.
5. For each turret, select the desired output mode.
 - Diameter mode for both mill and lathe operations.
 - Radius mode for both mill and lathe operations.
 - Radius mode for mill and diameter mode for lathe.
6. Save the **.machine** file.



Contour control modes (G05, G08)

Your DMG Mori **.machine** file supports a number of contour control options to improve accuracy for high-speed machining.

These options include:

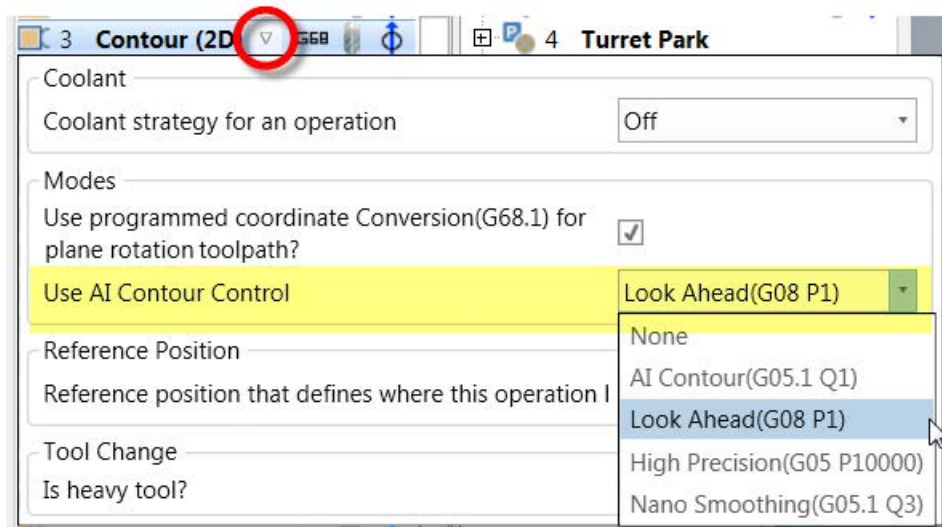
- AI contour control (G05.1 Q1)
- AI nano contour control (G05.1 Q3)
- High-precision contour control using RISC processing (G05 P10000)
- Look-ahead control function (G08 P1)

These options are available for mill and multiaxis toolpaths. Select the desired mode as shown.

- The default option is **None**.
- You need to select the desired mode for each operation. Each mode is automatically cancelled after each operation.

When using AI contour control mode or AI nano contour control mode, you should make sure that G332 is enabled and the proper cutting mode is selected; see “Selecting the cutting mode (G332)” on page 50 to learn more.

Note that these options are not present on the NZX machines—only the NT and NTX machines.



I: Multiaxis toolpath settings

Your DMG Mori **.machine** file includes a number of options that you can use to configure how your post handles multiaxis toolpaths. You can edit the default settings in the **.machine** file, and you can also set them individually for each operation.

- ❖ **Rotary start position**
- ❖ **Pole handling**
- ❖ **Tool tip point control/tool center programming (G43.4)**
- ❖ **Default settings for multiaxis toolpaths**

Rotary start position

Select the rotary start position to help Mastercam decide between two possible rotary orientations at the start of the toolpath

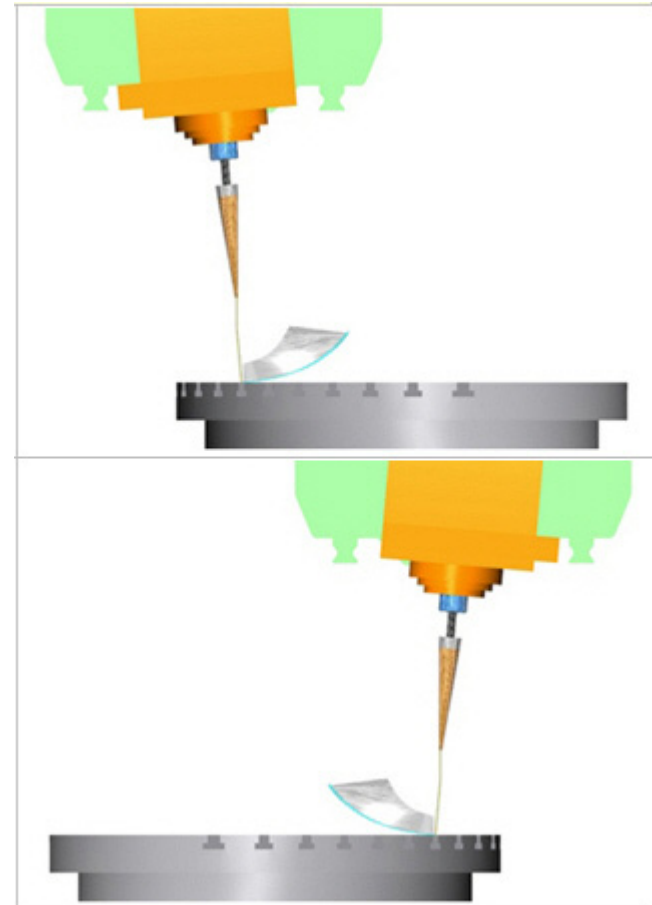
Typically, two logically equivalent solutions are possible for the starting rotary position of a toolpath. The difference between the two solutions is that the second location is separated by 180 degrees in the C-axis, and the sign of the B-axis is reversed. (The picture at right shows this for a generic multi-axis machine.)

However, from an applications perspective, one location is usually better than the other: for example, one might be a shorter distance from the end of the previous toolpath, or might involve rotation angles that are not mechanically possible.

Your DMG Mori **.machine** file includes two options that are used by the post to select the proper starting position:

- A desired starting angle (rotary position)
- The axis in which the angle is measured. This can be either the primary or secondary rotary axis.

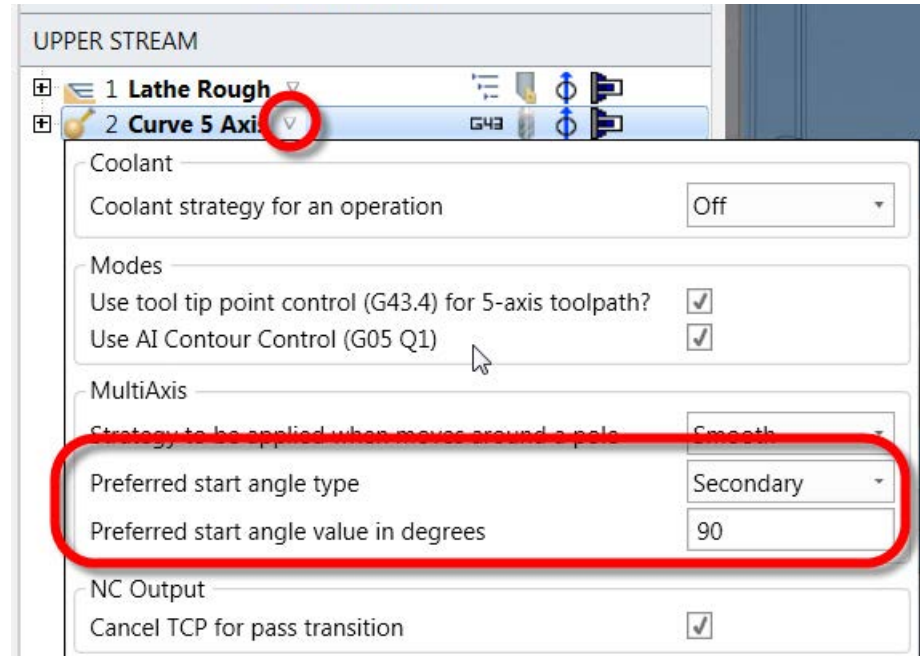
Mastercam will automatically select the solution that is closest to this angle.



Preferred start angle type—Select the axis in which the desired start angle will be measured: either the **Primary** or **Secondary** rotary axis. For the DMG Mori NT/NTX/NZX, the primary rotary axis is C, and the secondary rotary axis is B.

Preferred start angle value—Enter the desired start angle. Mastercam will select the solution closest to this angle. For example, if you enter a value of **90**, and the two possible solutions are –B80 and B80, Mastercam will select B80 for the start position.

You can enter either positive or negative angles in this field; however, be aware that Mastercam determines “closeness” in terms of the angle’s position on a unit circle. For example, if you specify **–270** here ($90 - 360$), and possible solutions are –B80 and B80, then the post will take B80 since –270 is closer to 80 than to –80 in the unit circle.



Note—Mastercam will automatically discard solutions that are outside the limits.

For your changes to be effective, you must follow these steps.

1. In Mastercam, select all the operations and post them. This will create a new IOF file that opens in the Sync Manager.
2. In the Sync Manager, select the desired start angle options.
3. Press **Ctrl+S** to save the changes back to Mastercam.
4. Go back to Mastercam and repeat step 1. This will create a new IOF file based on your new start angle settings.
5. In the Sync Manager, post your operations to generate the new NC code.



Pole handling

A pole (or “singularity”) occurs when a machine has a C-axis that is rotating around Z, and the tool orientation is almost parallel to the Z-axis. When this happens, and the tool is sufficiently parallel to the Z-axis, any C value will satisfy the conversion of the tool orientation vector to rotational values: for example, C0 B0 or C10 B0 or C300 B0. In these cases, only the B-axis value is important—the C value is arbitrary.

Pole handling should handle those cases when the tool axis and the spindle axis are colinear. That means that both directions are parallel and a mathematical singularity is reached.

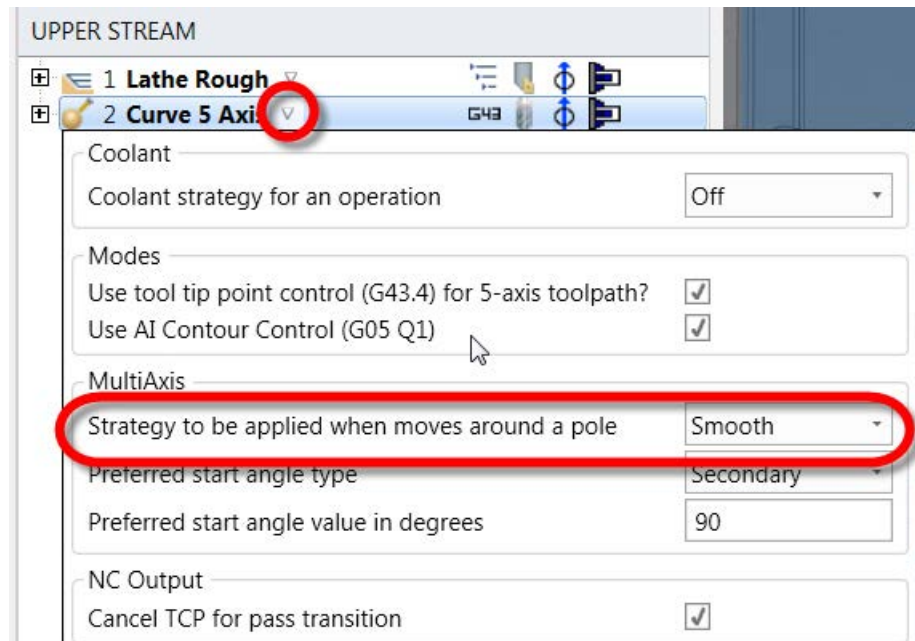
For each operation, you can select the preferred strategy in the Sync Manager.

Freeze—Freeze the angle around the pole. While the pole condition is active, the C-axis position will be frozen at its last position before the pole condition occurred.

RotAngle—Use a rotation angle around the pole. If the undetermined angle is in the spindle, your post will output an angle move only if it is necessary to stay within linear axis limits. This works the same as **Force rotation** except that this mode will be activated only when necessary to preserve limits.

Force—Force rotation. If the undetermined angle is in the spindle, Mastercam will substitute rotary angles for linear motions, rotating the spindle with rotary moves while X and Y are constant.

Linear or Smooth interpolation—If the part contains sections with only 3-axis moves that separated by sections with rotary moves, your post will interpolate tool angle positions between the sections to avoid “jumps.” The jumps occur because each 3-axis section has an undetermined rotary position. **Linear** interpolation will result in a linear transition between the sections, while **Smooth** interpolation will gradually increase/decrease the angle moves between the sections.



Tool tip point control/tool center programming (G43.4)

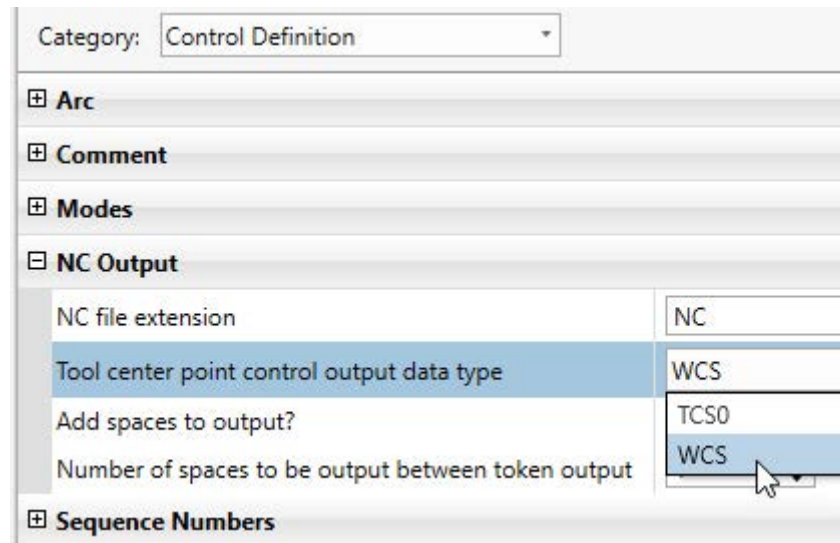
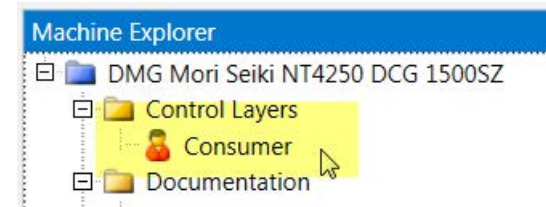
Your **.machine** file supports the use of G43.4 mode for tool center programming (TCP) with multiaxis toolpaths. TCP is turned on automatically for multiaxis toolpaths. Your **.machine** file contains several options for configuring how TCP mode is applied.

Selecting the TCP data type

You can choose the data type or output mode for TCP output. This setting is saved in your the **.machine** file and is applied to all of your parts. Follow these steps:

1. Open the **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.
3. Select **Category: Control Definition**.
4. Go to the **NC Output** section.
5. Select the proper **Tool center point control output data type** for your control. This should match the setting on your control.
 - Select **TCS0** to output table coordinate system data fixed to C0.
 - Select **WCS** to use the workpiece coordinate system. This is the most common option.
6. Press **[Ctrl+S]** to save the **.machine** file

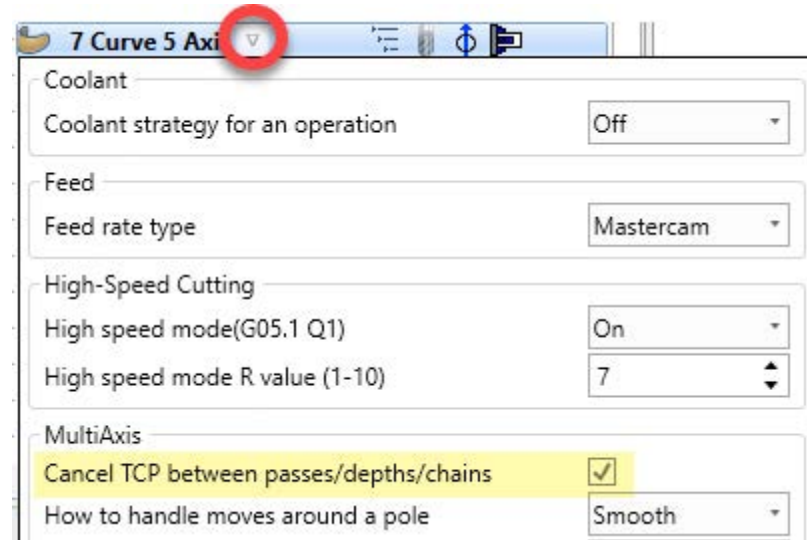
Note—After changing this setting, you must return to Mastercam and reload the machine, regenerate your operations, and repost them to the Sync Manager for the change to be effective..



Canceling TCP between passes

Your **.machine** file includes an option to cancel TCP for transition moves between cutting passes. For example, if your toolpath requires a large C-axis move between passes, it might be safer to turn off TCP while the tool is moving from one pass to the next.

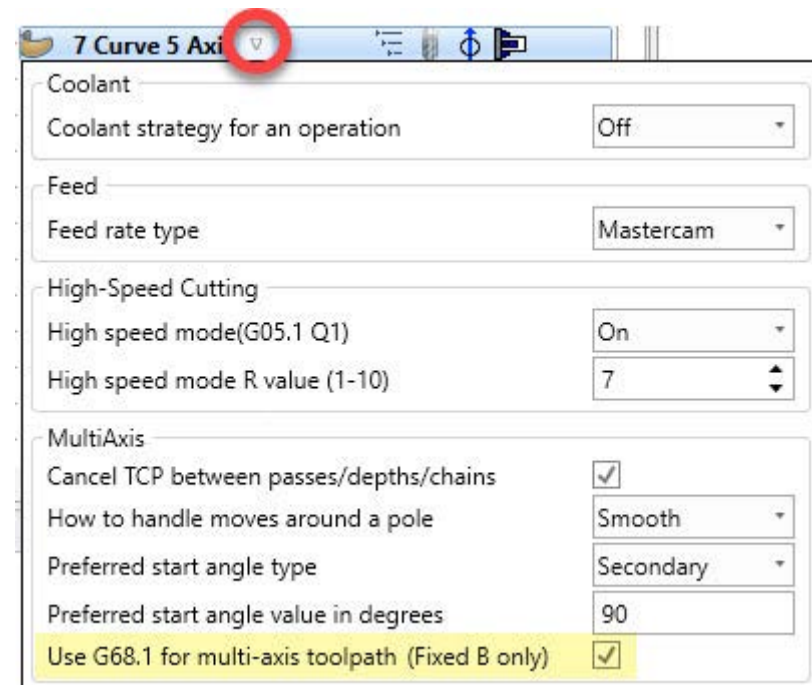
Set this option individually for each multiaxis operation as desired. Select **Cancel TCP between passes/depths/chains** as shown.



Using G68.1 instead of TCP

Your **.machine** file includes an option that lets you index the B-axis to a fixed angular position with a G68.1 command, instead of using TCP.

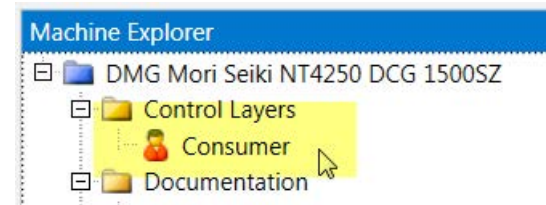
Set this option individually for each multiaxis operation as desired. Select the **Use G68.1 for multi-axis toolpath...** option as shown.



Default settings for multiaxis toolpaths

For each multiaxis option, you can save default values in your **.machine** file. Follow these steps:

1. Open the **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.



3. Select **Category: Output Settings**.
4. Go to the **MultiAxis** section.
5. Set the desired values for the start angle and pole handling options.
6. Save the **.machine** file.

Category: Output Settings	
⊕ Approach	
⊕ Coolant	
⊕ Header	
⊕ Modes	
⊕ Motion	
⊖ MultiAxis	
Preferred start angle value in degrees	<input type="text" value="90"/>
Strategy to be applied when moves around a pole	Smooth
Preferred start angle type	Secondary
⊕ NC Output	
⊕ Part Handling	



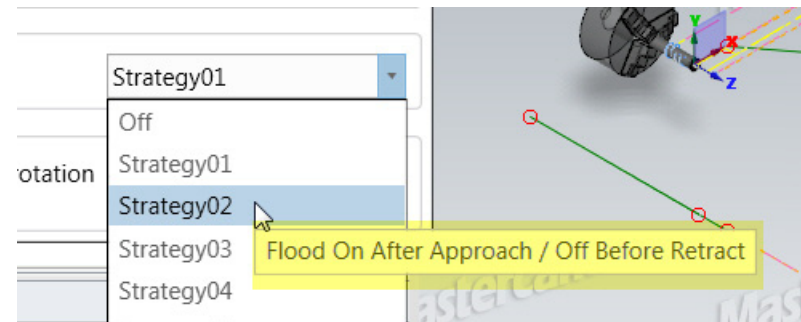
J: Coolant options

Your DMG Mori NT/NTX/NZX **.machine** file supports the following coolant options:

- Flood coolant for B-axis head and lower turret(M08)
- Thru-spindle coolant for the milling spindle (M484)
- Bed shower coolant (M382)

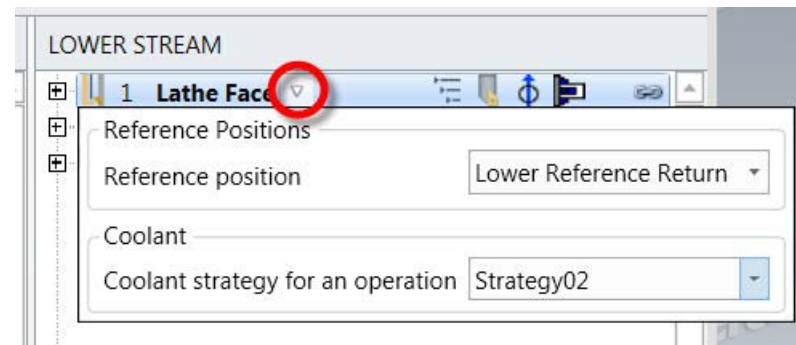
If you are familiar with other Mastercam products, you are used to selecting coolant options inside Mastercam as part of the toolpath parameters. In Mill-Turn, coolant selection is done in the Sync Manager instead. The **Coolant** button is no longer present inside Mastercam.

Most coolant options are selected through defined *strategies* that turn coolant options on/off at different points in the toolpath cycle. Hover over a strategy to see a description of exactly what it does.



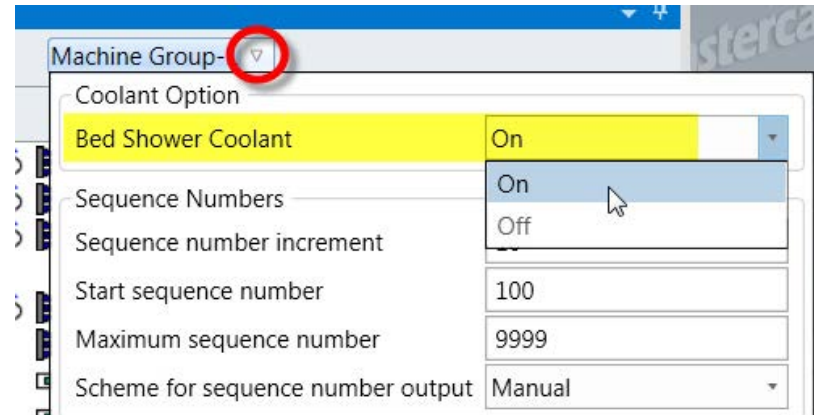
To select a strategy, click the small triangle next to the operation name and select the desired coolant strategy. Do this for each operation.

Select **Off** only if you want to force all the coolant options off for that operation. Typically, if you select a strategy to turn coolant on, it will include the appropriate coolant-off commands and you do not need to turn them off yourself.



Bed shower coolant (M382)

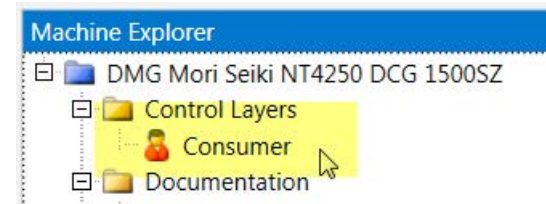
Bed shower coolant (M382) is controlled separately from the other coolant options. It is typically turned on at the beginning of the machining job. Click the small triangle next to the machine group name to turn it on. It is automatically turned off at the end of the stream.



Setting the default coolant option

You can select a default coolant strategy and save it in the **.machine** file.

1. Open the DMG Mori NT/NTX/NZX **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.



3. Select **Category: Output Settings**.
4. Go to the **Coolant** section.
5. Set the desired default strategy.
6. Save the **.machine** file.

