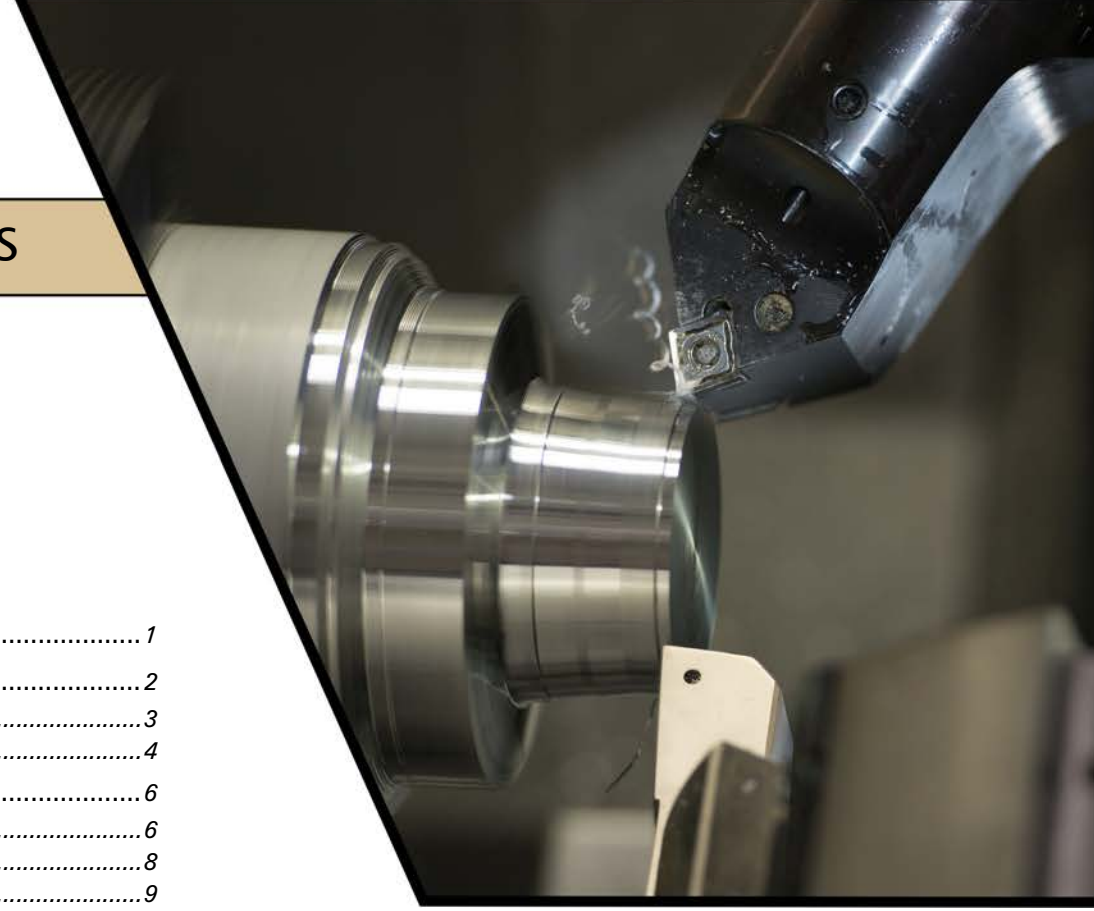
MILL-TURN APPLICATION GUIDES

Nakamura-Tome WT/WY



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 8
- Pre-staging the first tool 9
- Setting the program number prefix 10
- Outputting process table comments 11
- Syncing the last operation in each stream 11

D: Configuring part-handling operations 13
- Configuring the spindle-spindle distance 14
- Pickoff sequences 16
- Cutoff operations 18
- Coordinating spindle alignment angles 22
- Configuring spindle sync (G496) output 23
- Using push mode (G131) 25
- Syncing part handling operations 27
- Selecting G53 or G330 output for turret parks 28
- Activating the torque skip function (G359) 29

E: Approach and retract moves 31
- Selecting reference positions 31
- Setting the type of approach/retract motion 34
- Setting G28 or G53 output 35

revision date: June 15, 2018

Mastercam

F: Milling cycles and functions *36*
Polar (G12.1) and cylindrical (G7.1) interpolation..... *37*
Clamping the spindle (M86/M87, M32/M33)..... *41*
G: Balanced turning and pinch turning *42*
Numbering syncs for pinch turn operations..... *43*
H: Using coolant..... *44*

Mill-Turn Application Guide—Nakamura-Tome WT/WY
Copyright © 2018 CNC Software, Inc.—All rights reserved

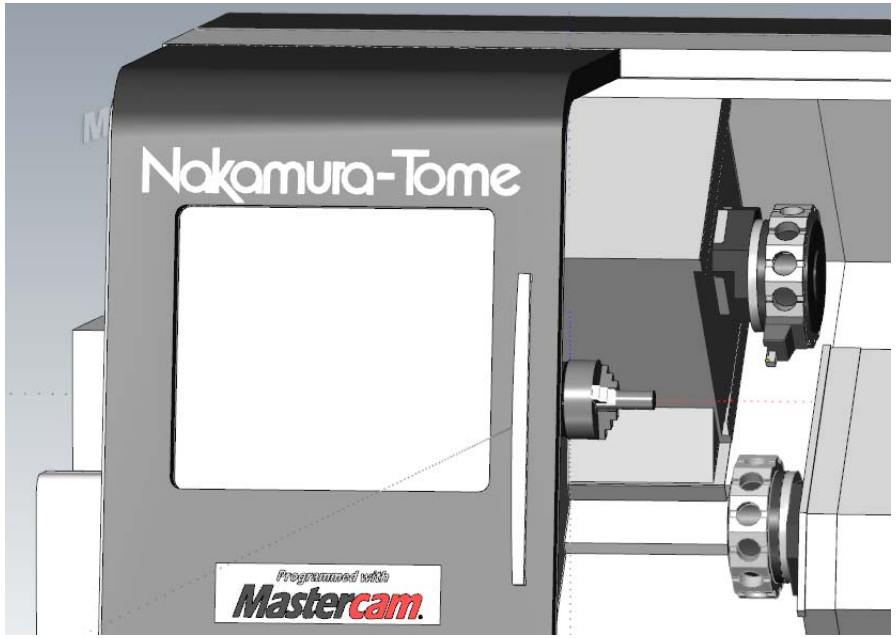
Terms of Use—Use of this document is subject to the Mastercam End User License Agreement.
A copy of the Mastercam End User License Agreement is included with the Mastercam product package.
The Mastercam End User License Agreement can also be found at: **http://www.mastercam.com/en-us/Company-Info/Legal/LicenseAgreement/**

A: Introduction

The **.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 chapter gives you some basic information about working with **.machine** files. It includes the following topics:

- **Customizing your .machine file**
- **Organizing NC output**
- **Configuring part-handling operations**
- **Approach and retract moves**
- **Milling cycles and functions**
- **Balanced turning and pinch turning**
- **Using coolant**

Supported models and controls—This application guide supports both the WT and WY series of machines. The Nakamura-Tome WT/WY **.machine** file and post are designed to support the Fanuc 31-TB control.



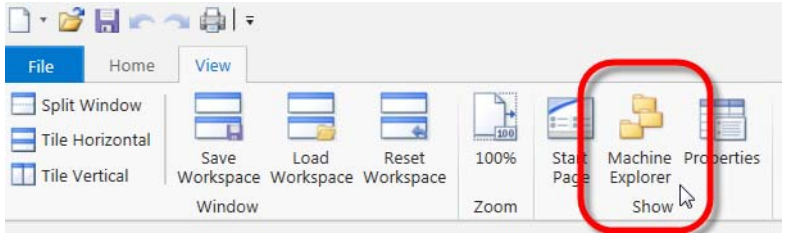
Home

Rev. date:
June 2018

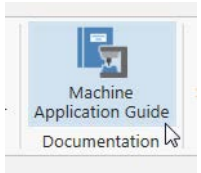
B: Customizing your .machine file

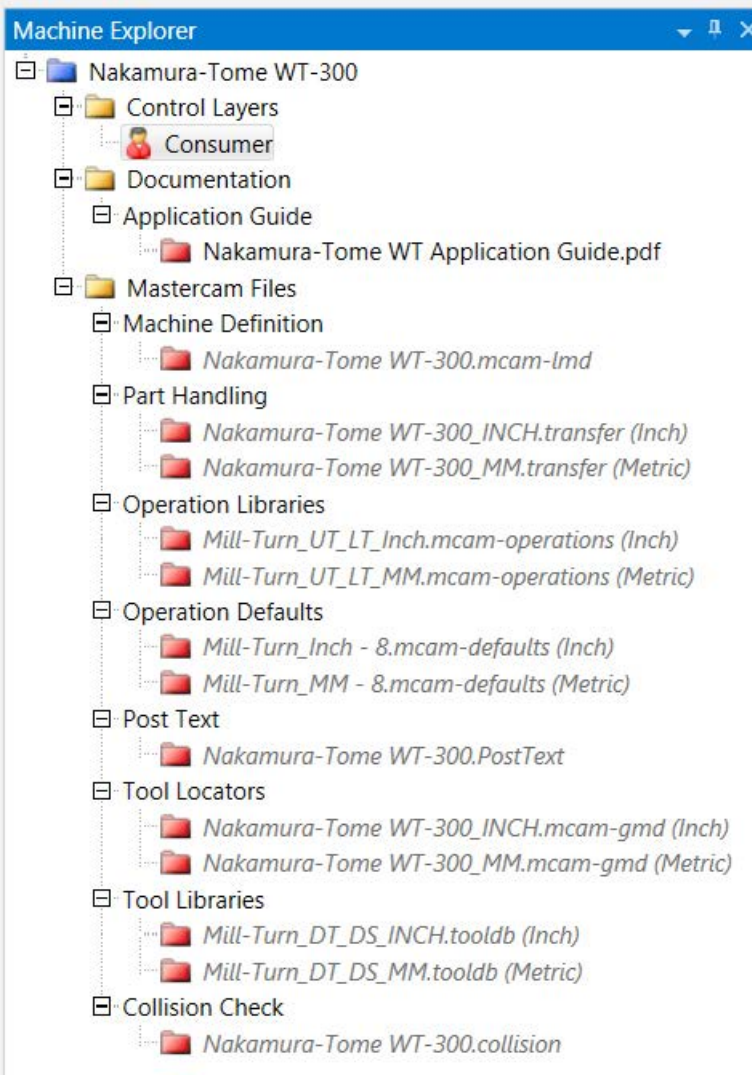
Whenever you load your **.machine** file in Mastercam to begin working on a part, Mastercam also starts Code Expert and loads your **.machine** file there as well. Code Expert is where you can make changes to your **.machine** file, such as editing default settings.

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.



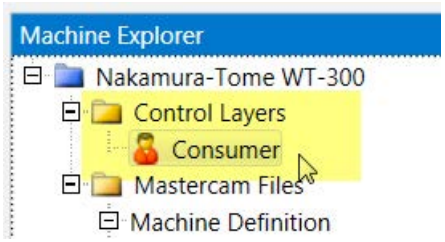


Home

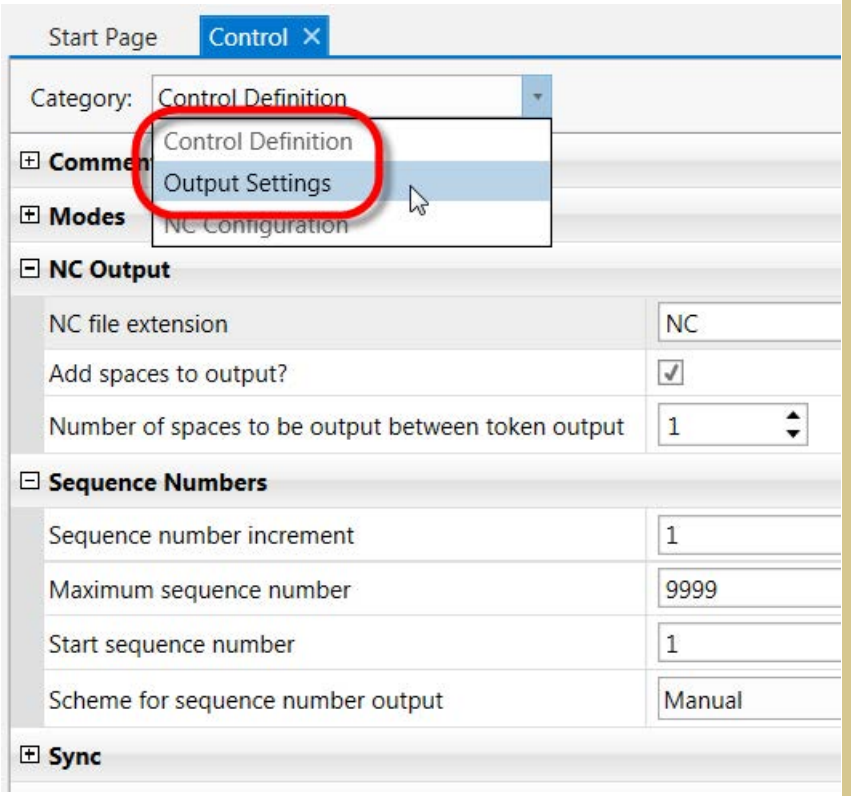
Default values for .machine file settings

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.

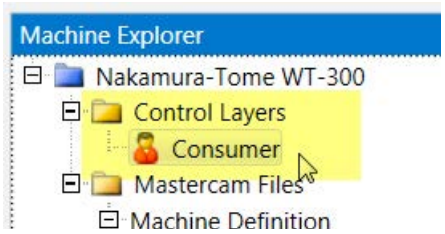
Many of these settings are self-explanatory and you can easily configure them by simply browsing the interface. The settings that are specific to this **.machine** file are described in this guide.

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

Home

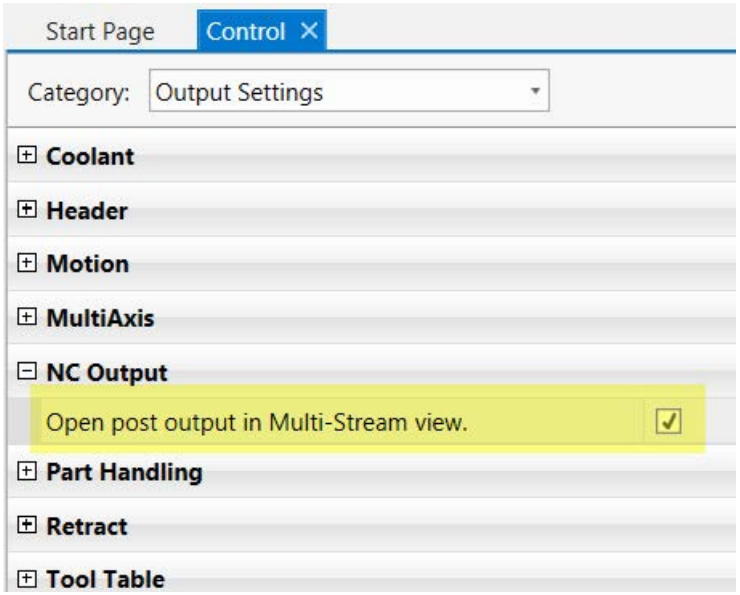
Configuring the Code Expert editor

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 Nakamura-Tome WT/WY NC output is typically divided into two streams, you may wish to open files in multi-stream view by default.

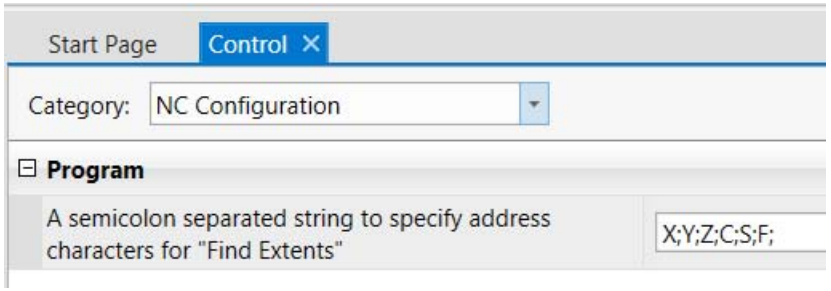
1. Go to the **Output Settings** category.
2. Open the **NC Output** group.
3. Select the **Open post output in Multi Stream view** option.



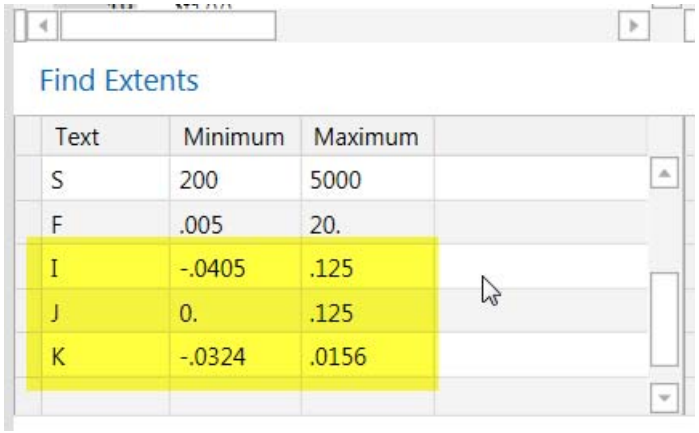
Home

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.



C: Organizing NC output

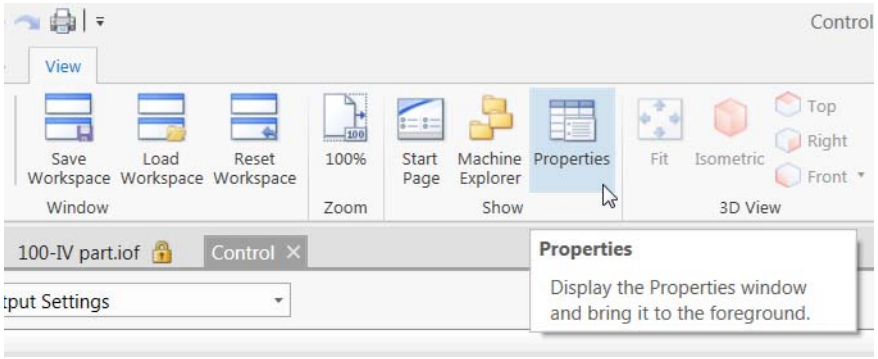
Setting the toolpath directory and stream names

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

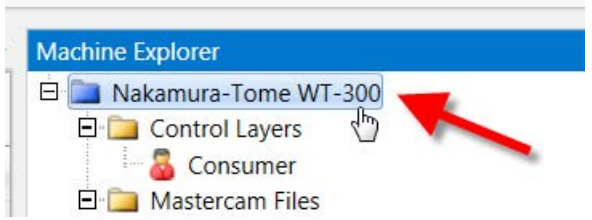
\my mcam2019\Mill Turn\NC

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

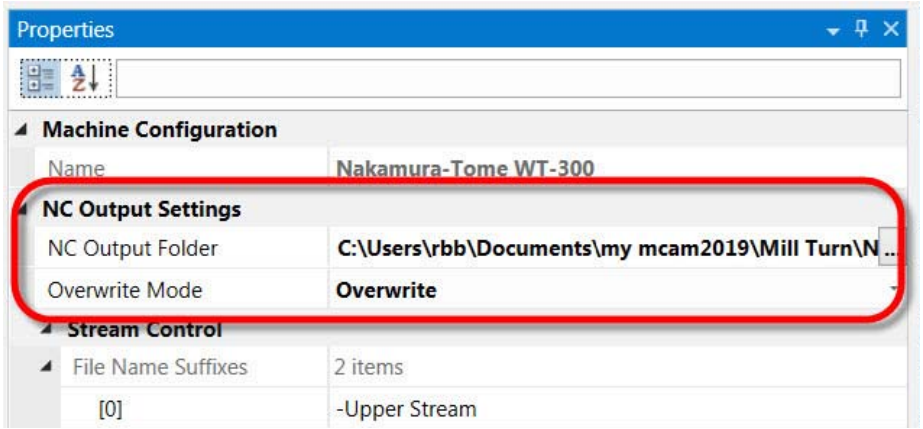
1. Open the **.machine** file in Code Expert.
2. Make sure that the **Properties** window is displayed.



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

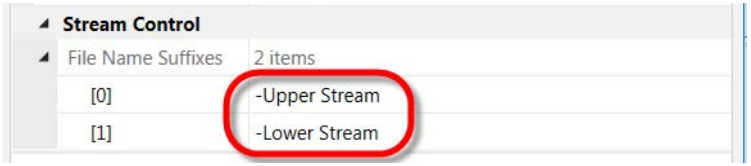


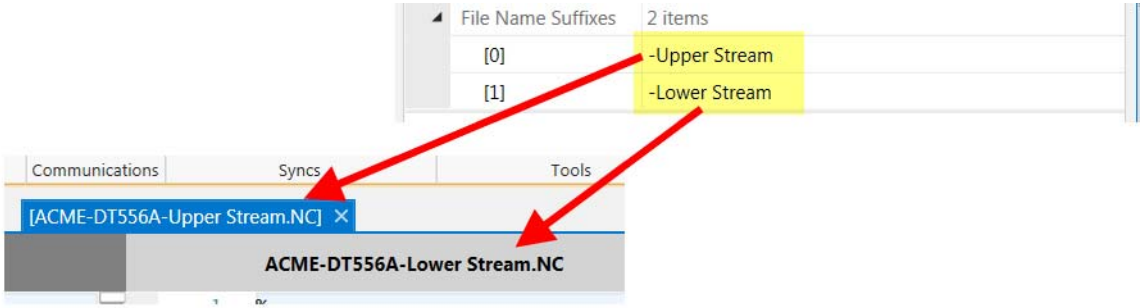
4. Select the desired **NC Output Folder**. Mastercam will write your NC files for this machine to this folder.
5. You can also choose whether Mastercam will automatically overwrite NC files with the same name, or prompt you to enter a different name. Select the desired **Overwrite Mode** to control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



Home

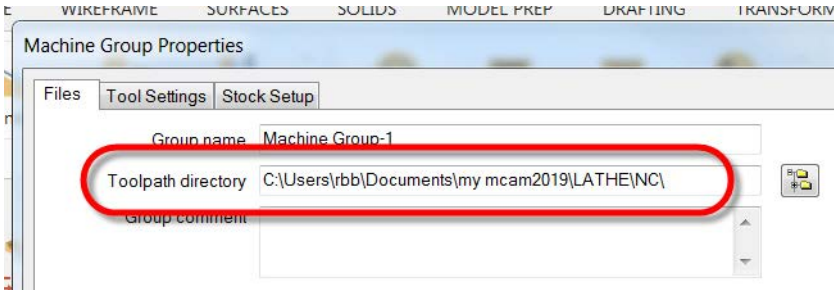
6. Mastercam Mill-Turn also lets you configure the names of the upper and lower streams. These will be automatically added to the NC file names for each stream. You can edit these names if you wish.



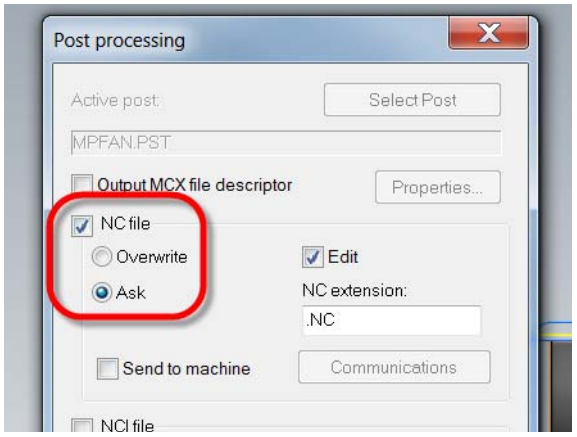


7. Press **Ctrl+S** to save the **.machine** file.

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 **Post processing** dialog box in other Mastercam products.

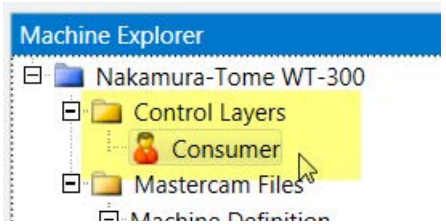


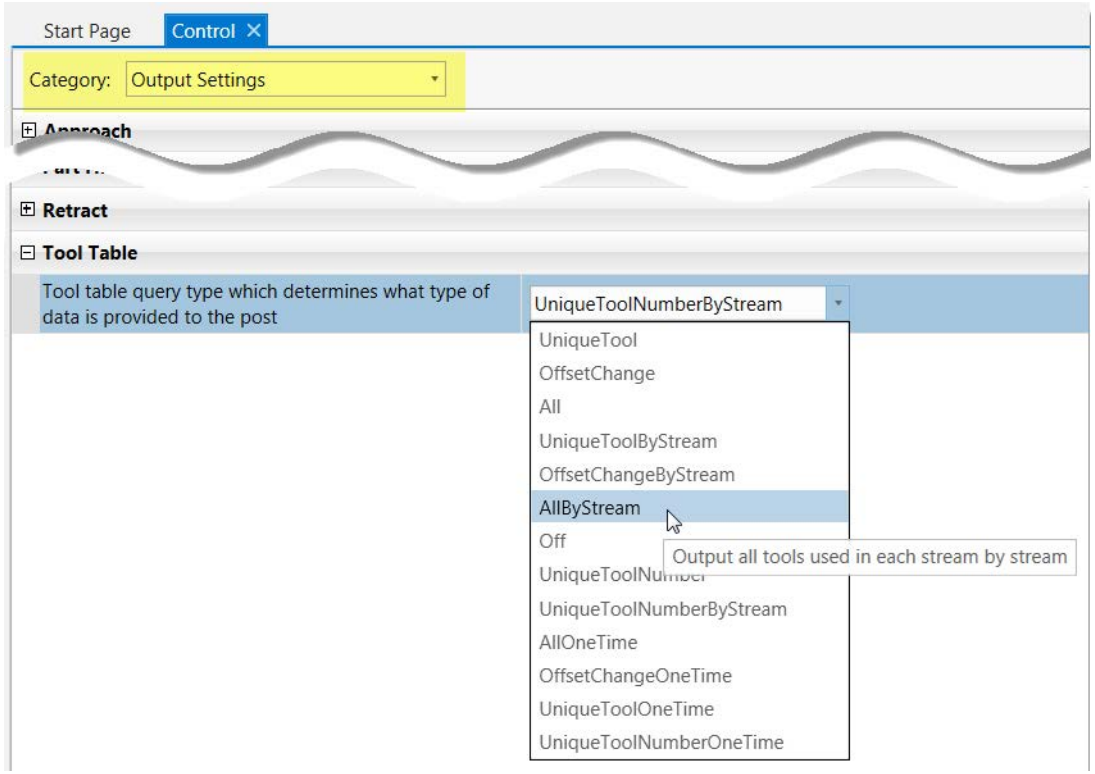
Home

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.





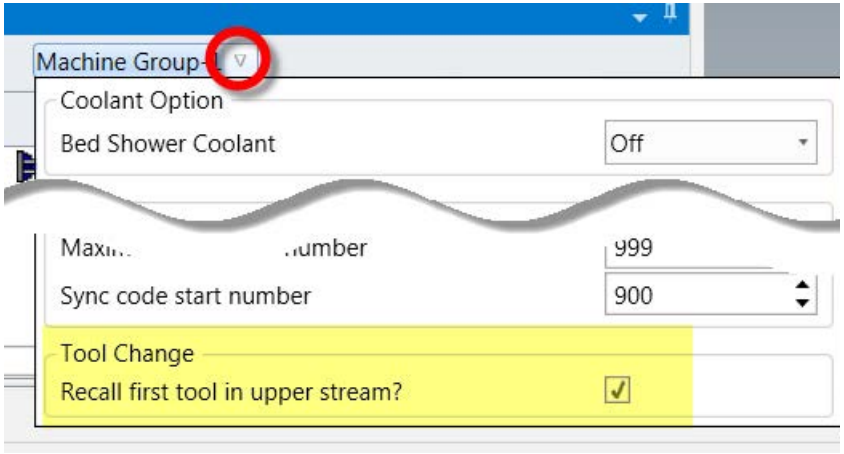
Home

Pre-staging the first tool

Your Nakamura-Tome **.machine** file includes an option that outputs a tool call at the end of your program for the first tool used in the upper turret. This can be a convenient way to pre-stage the first tool for the next part.

```
(OPERATION # 13)
(SEND UPPER TURRET HOME)
G28 U0. V0. W0.
M87
M41
M01
T0101 ( 1/8 FLAT ENDMILL)
M30
%
```

To activate this feature, simply select the **Recall first tool in upper stream?** option from the **Machine Group** options in the Sync Manager.



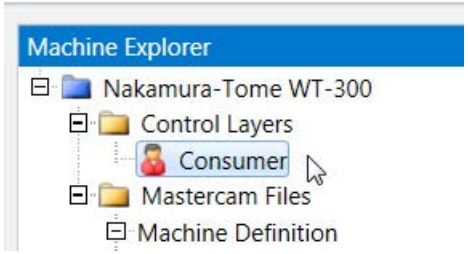
Home

Setting the program number prefix

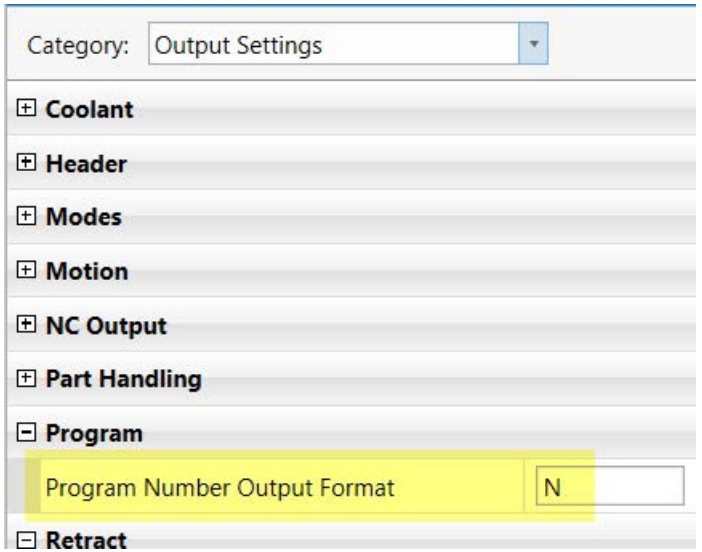
By default, the Nakamura-Tome WT/WY **.machine** file is set-up to use N for the program number. You can change this to a different letter if you wish—typically, O.

1. Open the Nakamura-Tome WT/WY **.machine** file in Code Expert.

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



3. Select **Category: Output Settings**.
4. Go to the **Program** section.
5. Enter the desired letter in the **Program Number Output Format** field.
6. Save the **.machine** file.

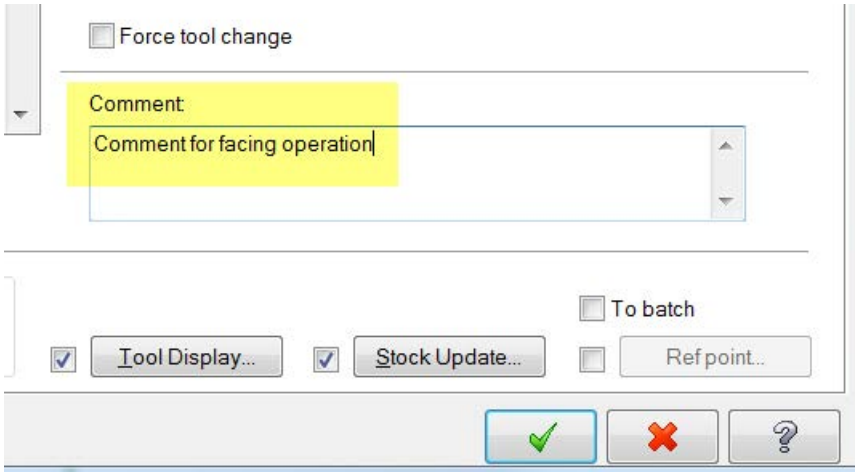


Home

Outputting process table comments

Your **.machine** file is set up to take operation comments from your Mastercam toolpaths and output them on `M41/M441/M91/M491` lines. These comments will then be included in the process table on your control.

Enter the comment text in the **Comment** box when you are creating your toolpaths in Mastercam.



It will be output as shown here:

```
G1998 S1 (LEFT SPINDLE)

N6 M41 (COMMENT FOR FACING OPERATION)
M428
(OPERATION # 1)
G28 U0.
G28 W0.
(T0101  |  ROUGH FACE LEFT - 80 DEG.
T0101 (ROUGH FACE LEFT - 80 DEG.)
```

Syncing the last operation in each stream

If your part program ends with both turrets working on the same spindle, it is important to create a sync between the ends of the last operation in each stream. This will ensure that one stream does not turn off the spindle while the other stream is still active.

Home

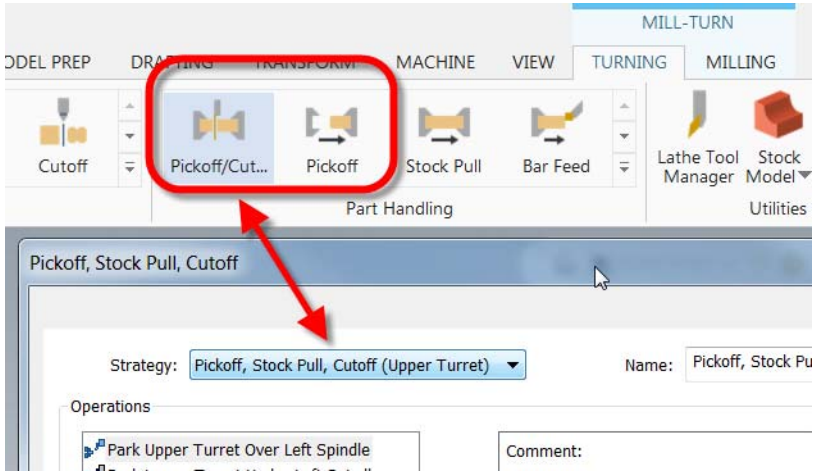
D: Configuring part-handling operations

Your **.machine** file includes customized support for part handling applications. These are sequences of individual component commands—spindle moves and other spindle actions, turret parks, bar pulls, and cut-off moves—that work together to create a complete part transfer application so you can easily program toolpaths on both spindles. Your Nakamura-Tome WT/WY **.machine** file has been configured to support two applications:

- simple pickoff from main to subspindle
- a complete pickoff–bar pull–cutoff transfer

Use the following workflow to accomplish these successfully:

- **Configuring the spindle-spindle distance**. Before using your **.machine** file, you need to set the spindle—spindle distance between the faces of the main and subspindle. Either validate the default distance or enter the proper distance for your specific machine.
- Inside Mastercam, click **Pickoff** or **Pickoff/Cutoff** from the **Turning** ribbon bar to select the part handling application. See the following sections to learn more:
  - **Pickoff sequences**
  - **Cutoff operations**
  - **Coordinating spindle alignment angles**
  - **Configuring spindle sync (G496) output**
- Turn on optional operating modes:
  - **Selecting G53 or G330 output for turret parks**
  - **Using push mode (G131)**
  - **Activating the torque skip function (G359)**
- **Syncing part handling operations**. Before you run Simulation or post your part from the Sync Manager, you need to create a number of syncs between the individual part handling operations in each stream.



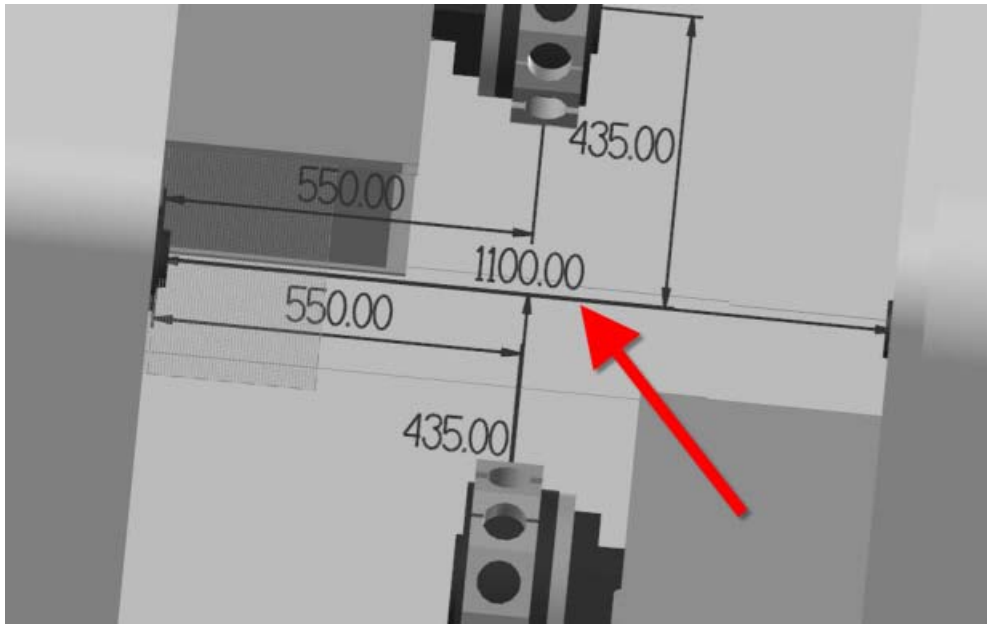
Home

Configuring the spindle-spindle distance

The distance between the faces of the main spindle and the subspindle is very important to the coordinate positions that will be output in the part handling operations. It is crucial that you validate the default value against the actual machine tool on your shop floor and adjust it if necessary.

Mastercam will automatically take into account the width of the chuck and jaws from your LMD and Job Setup information, so make sure you measure this distance from the face of the spindles.

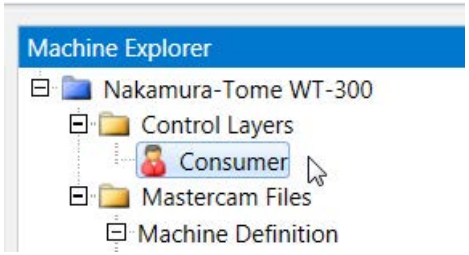
Typically, you will set this once when you receive your **.machine** file and not need to change it again.



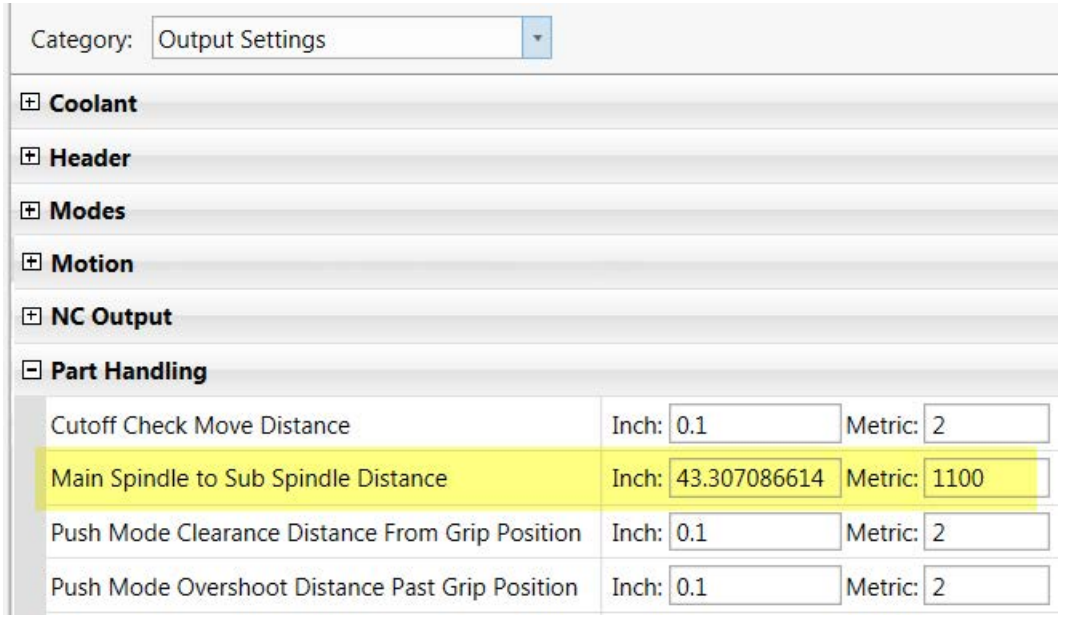
Follow these steps:

1. Open the Nakamura-Tome WT/WY **.machine** file in Code Expert.

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



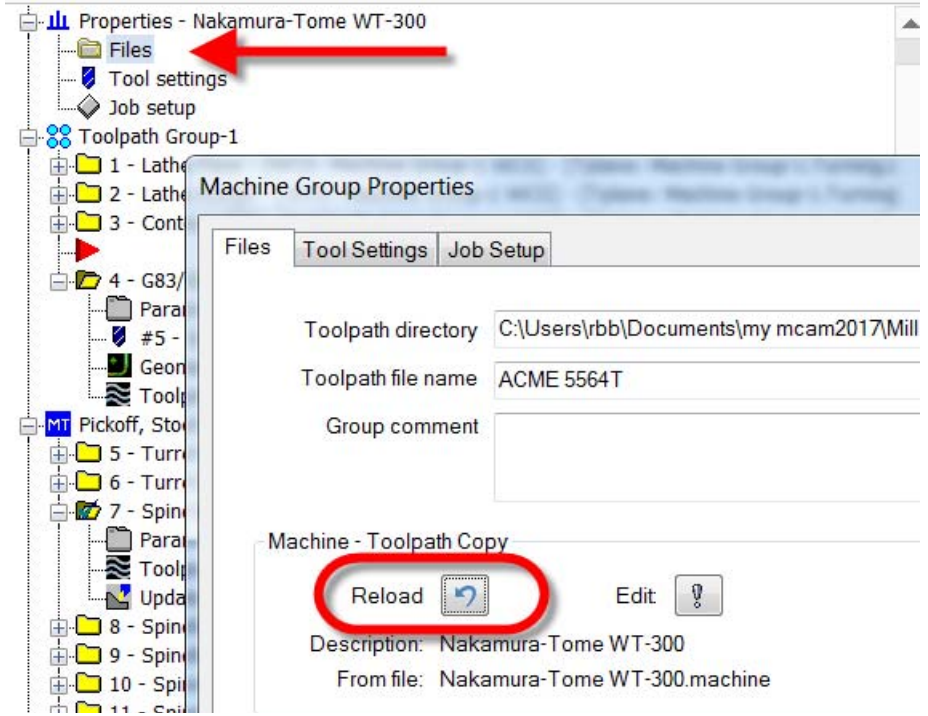
3. Select **Category: Output Settings**.



4. Go to the **Part Handling** section.
5. Enter the **Main Spindle to Sub Spindle Distance**.
6. Enter both inch and metric values.
7. Save the **.machine** file.

IMPORTANT: It is important to realize that if you change this value for any reason, you will need to reload the **.machine** file into any existing parts if you want to re-post them with the new value. Follow these steps:

1. Go to the Toolpath Manager inside Mastercam.
2. Click the **Files** entry in the **Machine Group Properties**.
3. Click the **Reload** button.
4. Click the **G1** button to re-generate the IOF file.
5. Post the file from the Sync Manager.

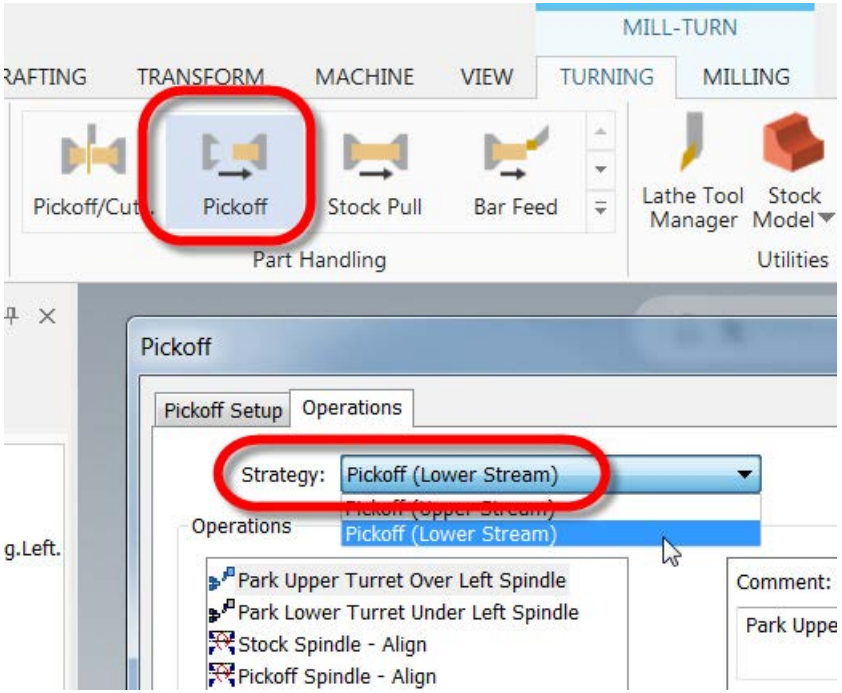


Pickoff sequences

Your Nakamura-Tome WT/WY.**machine** file includes a pre-built application scenario to model a simple part transfer or pickoff operation—in other words, a part transfer that does not involve a cutoff operation or bar pull. Typically, this is used when you are machining a single part instead of bar stock.

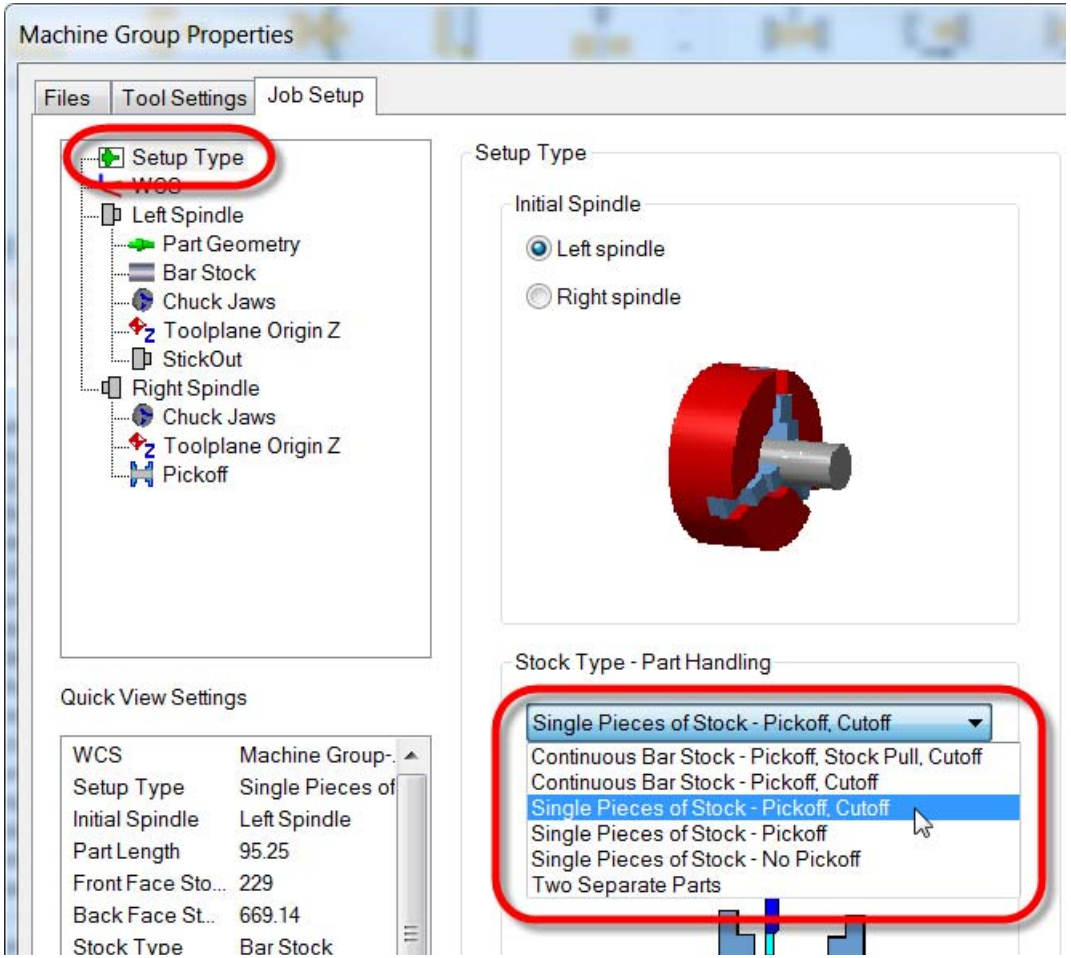
Follow this general outline:

1. In Mastercam, select **Pickoff** from the **Turning** ribbon bar.
2. Select the **Pickoff (Lower Stream)** strategy. The Upper Stream strategy is typically not useful for the Nakamura-Tome WT/WY machine, since the operations should be output in the lower stream.
3. Make sure that the angles for the spindle alignment operations are properly coordinated; see “Coordinating spindle alignment angles” on page 22.
4. Create the necessary syncs between the individual part handling operations; see “Syncing part handling operations” on page 27.



Home

Note that the **Pickoff** function is only available when the appropriate **Stock Type** has been selected in the **Job Setup**. For example, if you select one of the bar stock options, you will not be able to select the **Pickoff** function from the ribbon bar.



Home

Cutoff operations

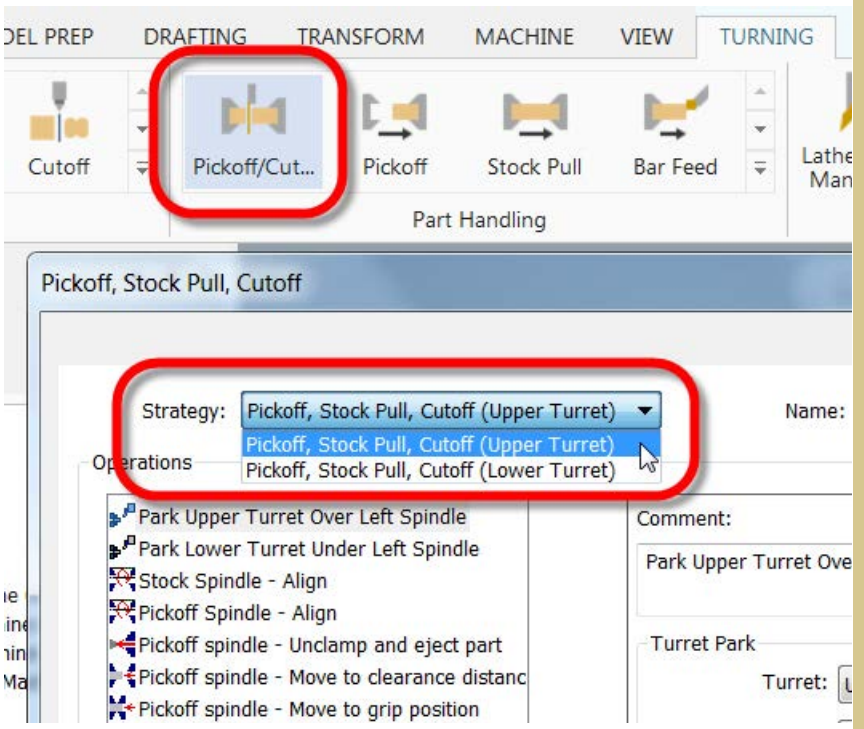
Your Nakamura-Tome WT/WY.**machine** file includes two pre-built application scenarios for modeling a complete pickoff–bar pull–cutoff sequence. One scenario places the cutoff operation in the upper stream, the other in the lower stream.

Follow this general outline. Complete these steps in Mastercam:

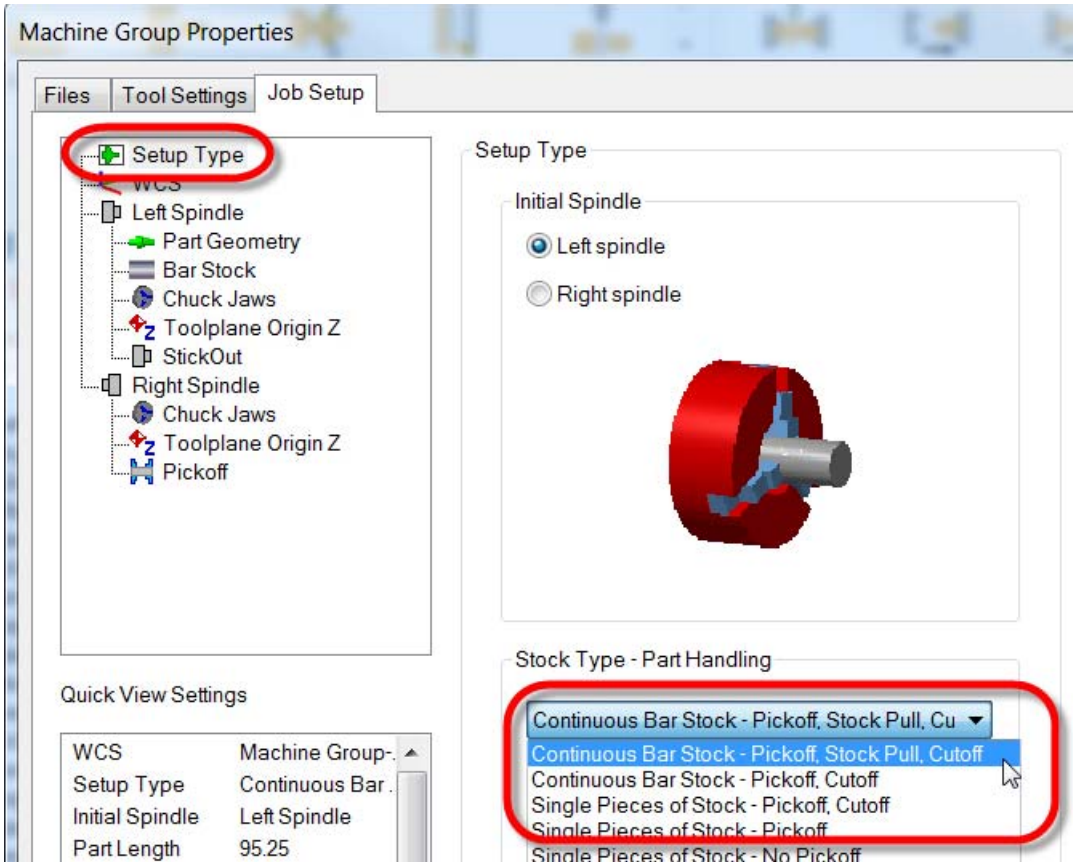
1. In Mastercam, select **Pickoff/Cutoff** from the **Turning** ribbon bar.
2. Choose whether the cutoff will be in the upper or lower stream by selecting the proper option from the **Strategy** list.
3. Make sure that the angles for the spindle alignment operations are properly coordinated; see “Coordinating spindle alignment angles” on page 22.

Complete these steps in the Sync Manager:

4. You can choose whether to use G131 push mode for the spindle clamp. See “Using push mode (G131)” on page 25.
5. Find the lathe point operation that pre-positions the cutoff tool and set its **Reference position** to **None**. See “Setting “Reference position” to “None” when pre-positioning the cutoff tool” on page 19.
6. Choose whether to use or disable cutoff check mode. See “Using cutoff check mode (G300)” on page 20.
7. Create the necessary syncs between the individual part handling operations; see “Syncing part handling operations” on page 27.



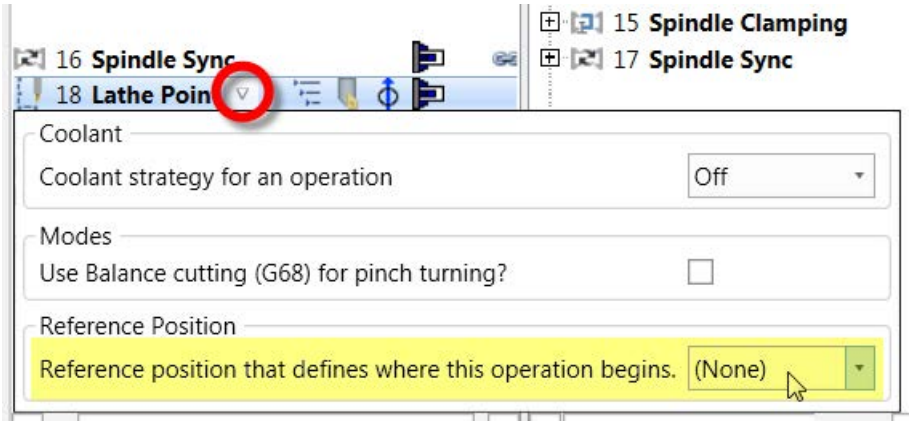
Note that to use one of the cutoff scenarios, you should select the **Pickoff, Stock Pull, Cutoff** option as the **Stock Type** in the **Job Setup**. For example, if you select one of the **Single pieces** options, you will not be able to select the **Pickoff/Cutoff** function from the ribbon bar.

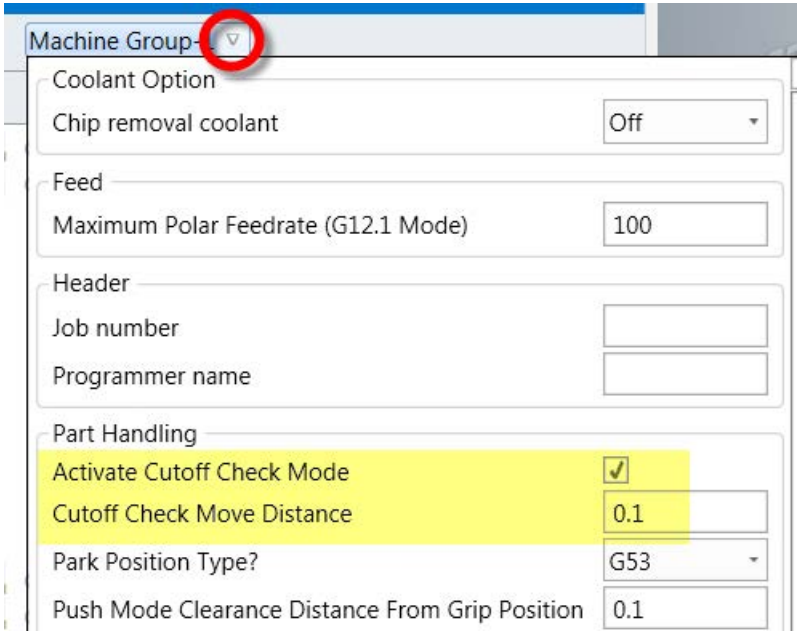


Setting "Reference position" to "None" when pre-positioning the cutoff tool

In order for your part to simulate correctly, it is important that you disable the reference position for the lathe point operation that is used to pre-position the cutoff tool.

1. Locate the Lathe Point operation in the Sync Manager. It should be in the same stream as the cutoff operation.
2. Click the small triangle next to the operation name.
3. Set the **Reference Position** to **None**.





4. Save the file.

Using cutoff check mode (G300)

Your **.machine** file includes the option to activate the `G300` cutoff check mode for the spindle retract after the cutoff operation.

To activate `G300`, select the **Activate Cutoff Check Mode** option from the **Machine Group** options in the Sync Manager.

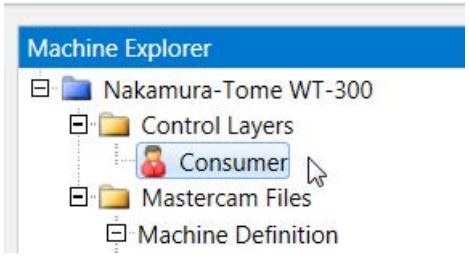
Then enter the **Cutoff Check Move Distance**. Your post will use this value to compute the proper `B` coordinate that is output on the `G300` line.

```
(OPERATION # 21)
(PICKOFF SPINDLE - RETRACT)
M104
G300 B-20.3437
G00 G28 B0. (MOVE RIGHT SPINDLE HOME)
M105
```

You can change the default settings for this option and save them in your **.machine** file:

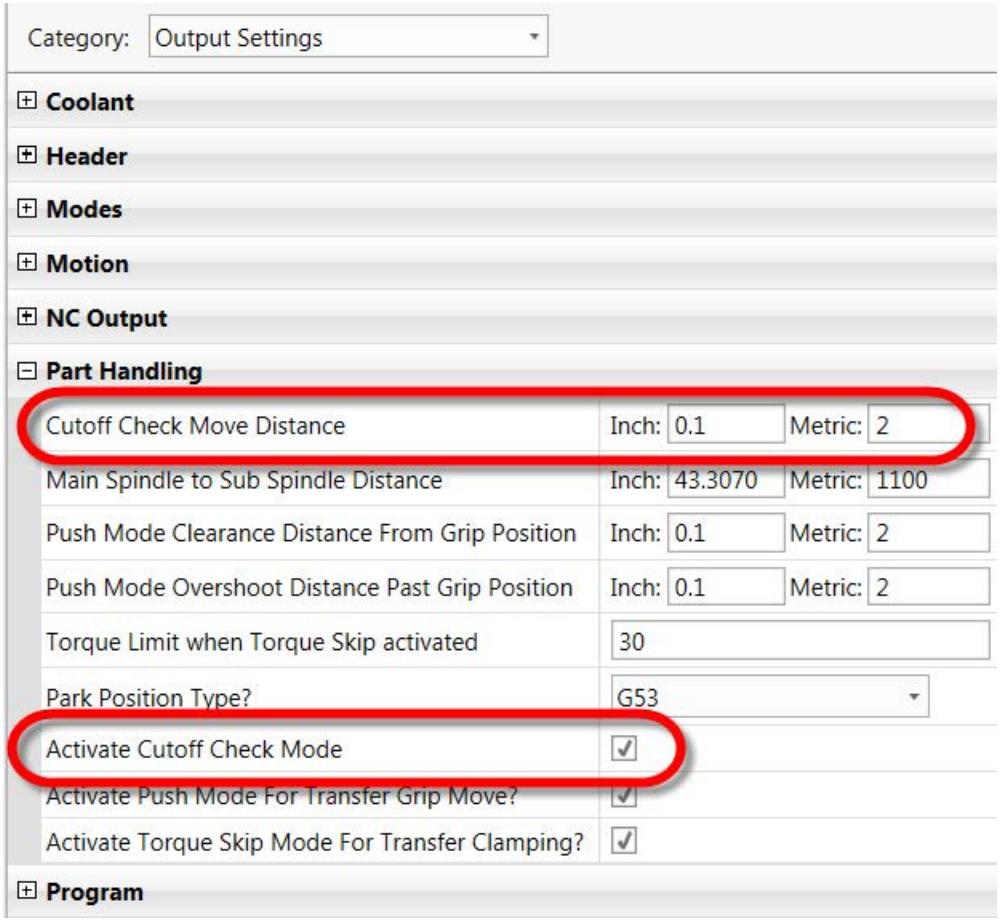
1. Open the Nakamura-Tome WT/WY **.machine** file in Code Expert.

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



3. Select **Category: Output Settings**.

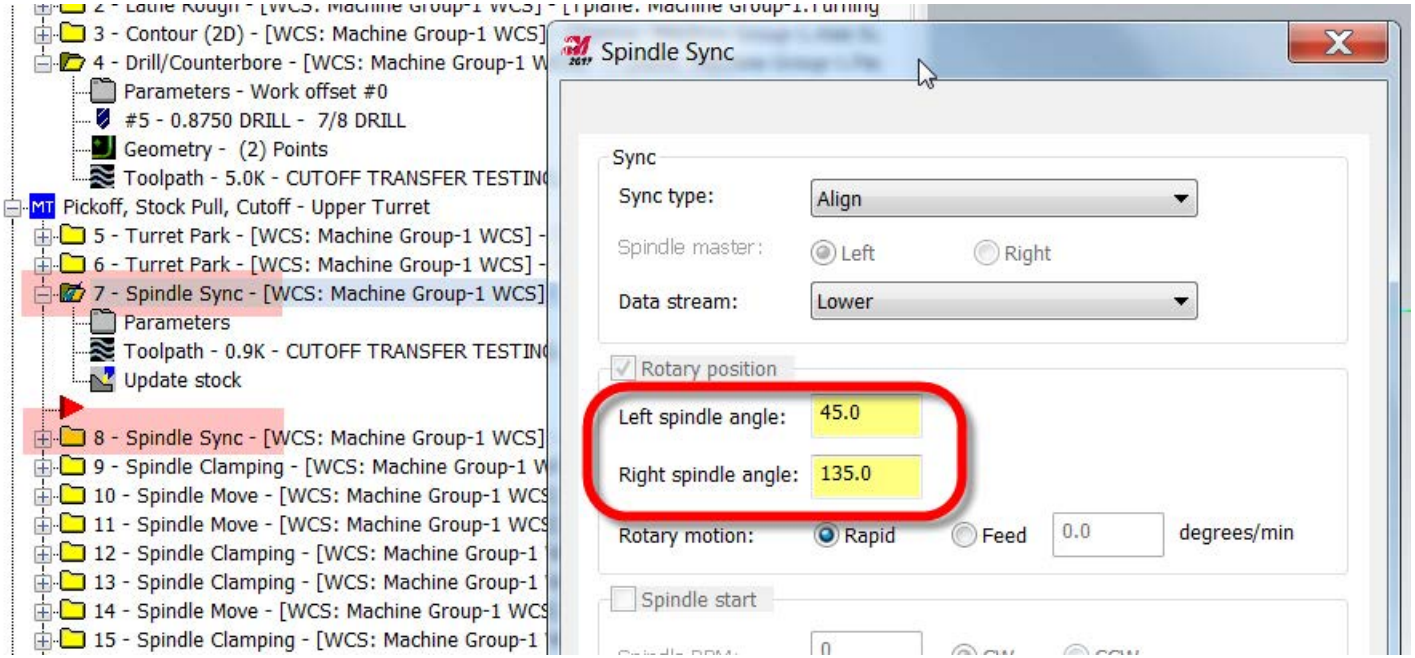
4. Go to the **Part Handling** section.
5. Set the options as desired. Enter both inch and metric values.
6. Press **[Ctrl+S]** to save the **.machine** file.



Home

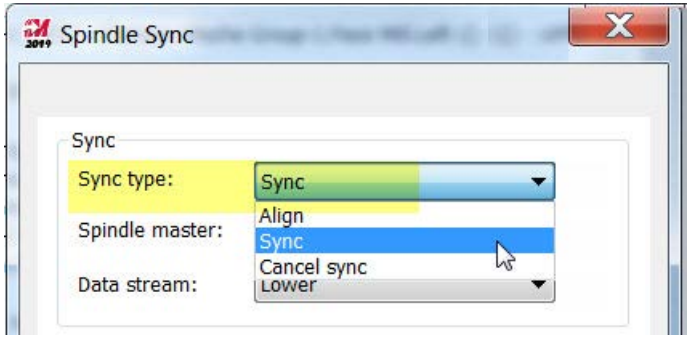
Coordinating spindle alignment angles

The spindle sync operations that are created by Mastercam let you rotate either spindle to whatever C-axis position that you wish. However, please note that these operations are created in pairs: one for upper stream output, and one for the lower stream. If you enter your own values for these angles (the default value is 0), it is important that you enter the same values in each operation so that the upper stream and lower stream output will match.



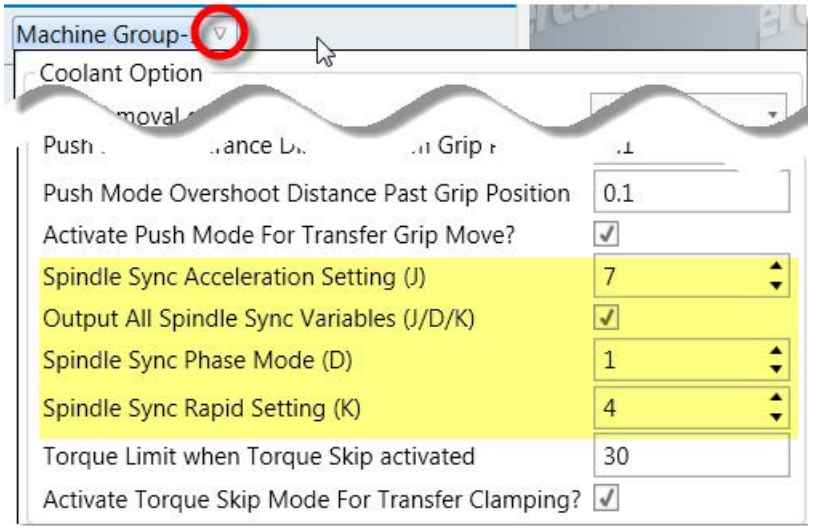
Configuring spindle sync (G496) output

Your Nakamura-Tome post is configured to output `G496` for spindle sync operations, not `M96`. This is effective when the **Sync type** is set to **Sync**, not **Align**.



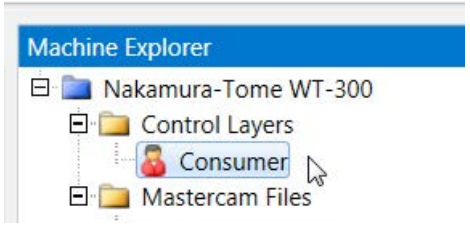
Your **.machine** file includes options that let you customize the values for the `J`, `D`, and `K` spindle sync variables.

1. Click the small triangle next to the machine group name.
2. Select the option to **Output All Spindle Sync Variables**.
3. Enter the desired values for the the `J`, `D`, and `K` values.
4. Press **[Ctrl+S]** to save the settings back to your part file.

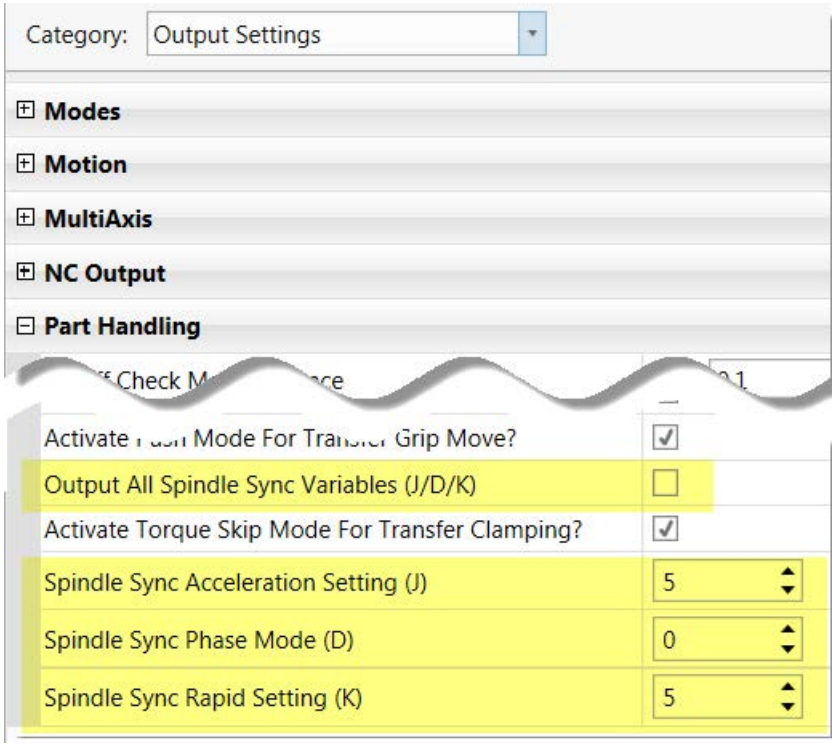


You can also change the default settings for these options:

1. Open the Nakamura-Tome WT/WY **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.



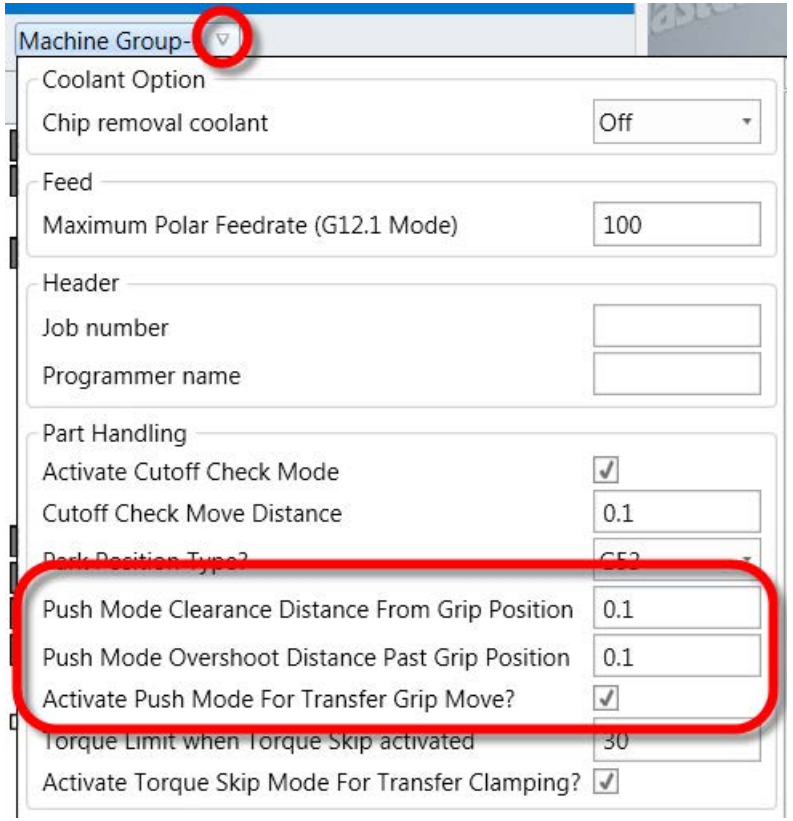
3. Select **Category: Output Settings**.
4. Go to the **Part Handling** section.
5. Enter the desired values.
6. Press **[Ctrl+S]** to save the **.machine** file.



Using push mode (G131)

Your Nakamura-Tome WT/WY.**machine** file includes support for the G131 push mode for spindle clamping operations.

1. In the Sync Manager, click the small triangle next to the **Machine Group** options.
2. Select the **Activate Push Mode…** option.
3. If desired, enter new values for the **Clearance Distance** when approaching the part and the **Overshoot Distance** when clamping it.
4. Press **[Ctrl+S]** to save the settings back to your part file.



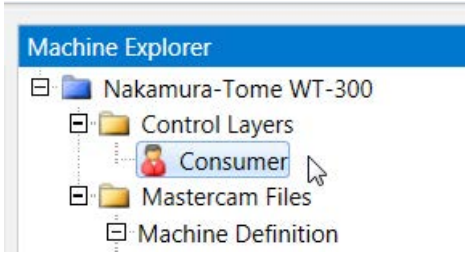
Home

Default settings for push mode

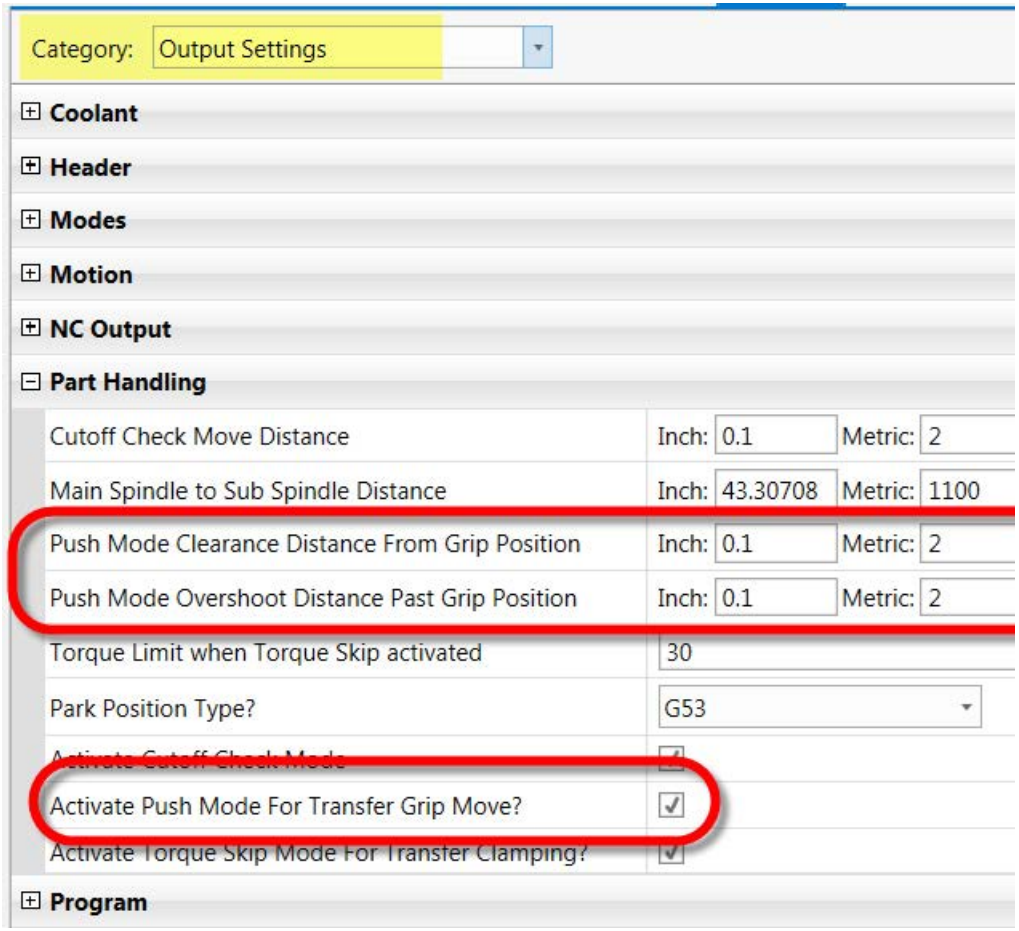
If you wish, you can change the default settings for the push mode options. Follow these steps:

1. Open the Nakamura-Tome WT/WY **.machine** file in Code Expert.

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



3. Select **Category: Output Settings**.
4. Go to the **Part Handling** section.
5. Select the **Activate Push Mode…** option to have G131 turned on by default.
6. Enter the desired default values for the clearance and overshoot distances.
7. Press **[Ctrl+S]** to save the **.machine** file.



Syncing part handling operations

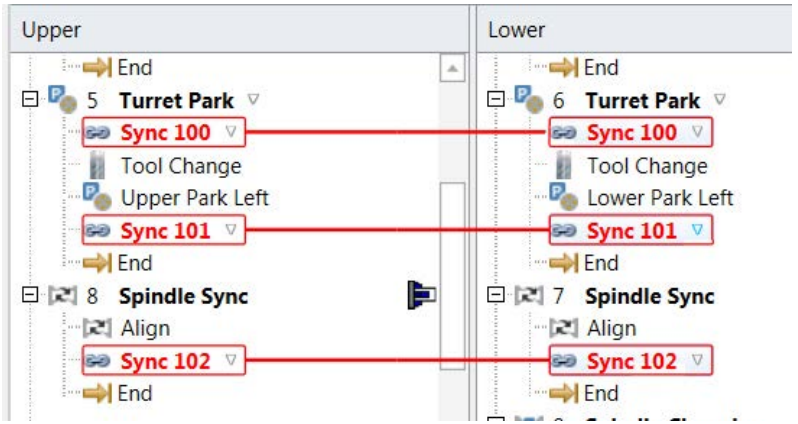
After creating a pickoff or part transfer sequence in Mastercam, your part will not be usable until you create a number of syncs in the Sync Manager.

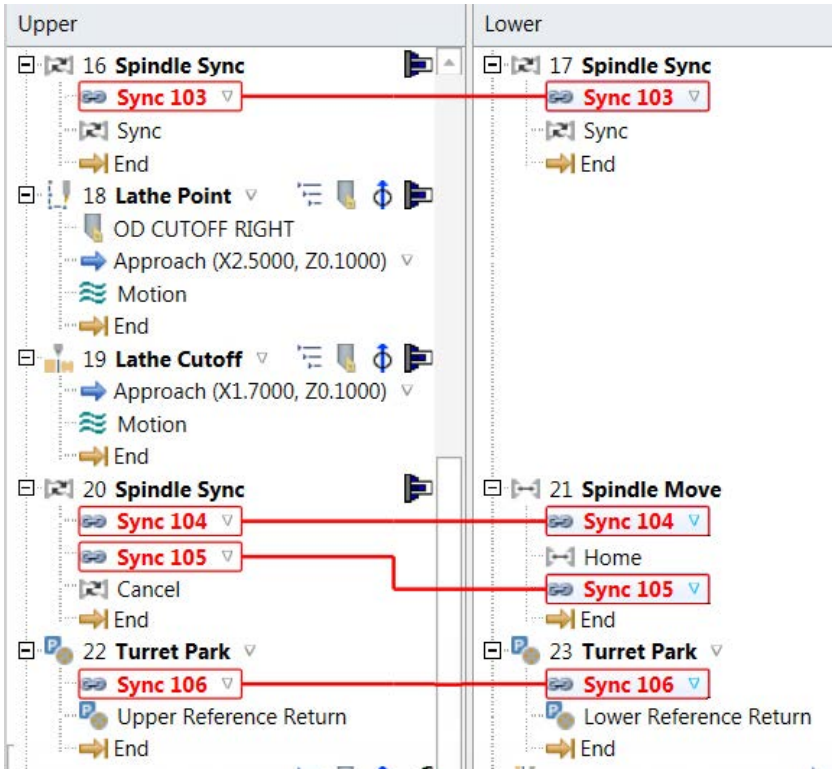
The example shown here is typical for a pickoff/bar pull/cutoff part handling application. You can modify this as needed. Use the Sync Manager display and Simulation to see how each turret and spindle move.

- Sync the beginning and end of the turret park operations (syncs 100 and 101).
- Sync the ends of the first spindle alignment operation in each stream (sync 102).
- Find the Lathe Point operation that is used to pre-position the cutoff tool. Create a sync between the spindle alignment operations that precede it (sync 103).
- Find the Lathe Cutoff operation. Create syncs between the beginning (sync 104) and end (sync 105) of the spindle operations that follow it.
- Finally, sync the beginning of the final turret park operations (sync 106).

When you are done, make sure you press **[Ctrl+S]** to save the syncs back to your Mastercam part file.

When creating the syncs, please pay attention to whether you are placing the sync at the beginning or end of the operation.



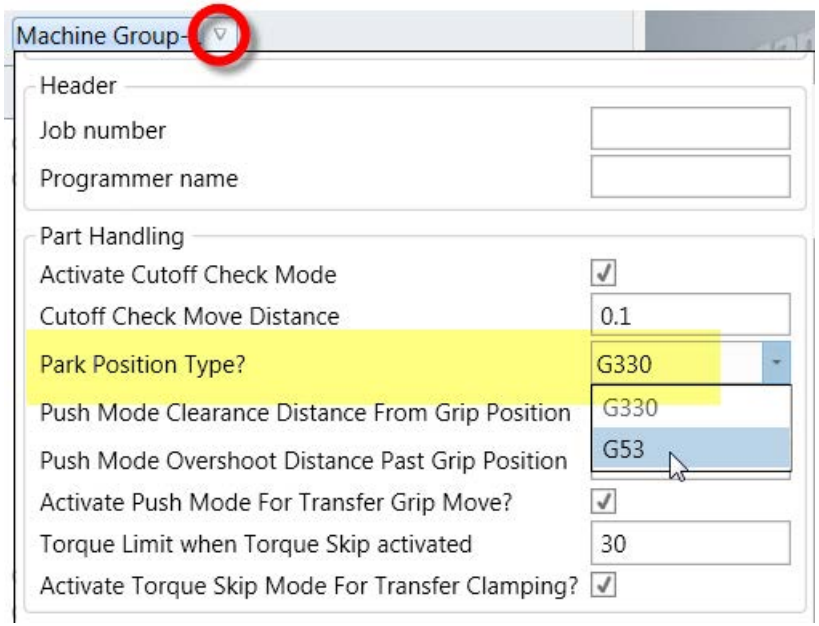


Home

Selecting G53 or G330 output for turret parks

When you are parking the turrets at the beginning of a part handling scenario, you can choose whether to output the park position as either a `G53` position or a `G330` reference point.

This option is available from the **Machine Group** options in the Sync Manager. Go to the **Part Handling** section and select the desired **Park Position Type**.



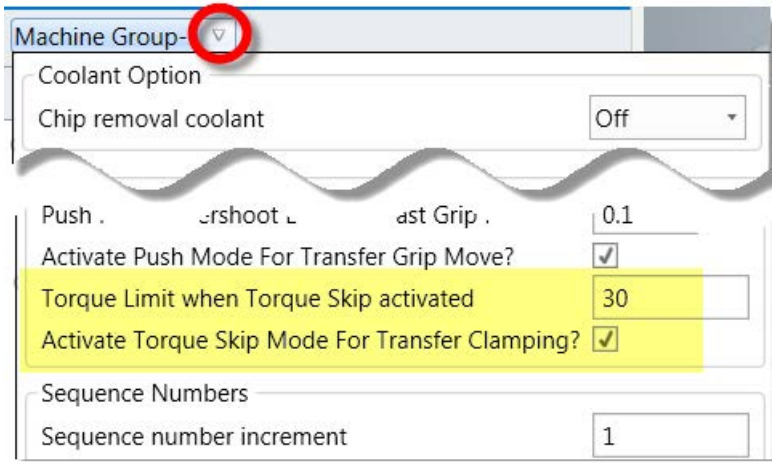
Home

Activating the torque skip function (G359)

Your Nakamura-Tome WT/WY **.machine** file supports the `G359` torque skip function. This is used for clamping operations; it will be enabled before the clamp and disabled after.

Turning on torque skip mode

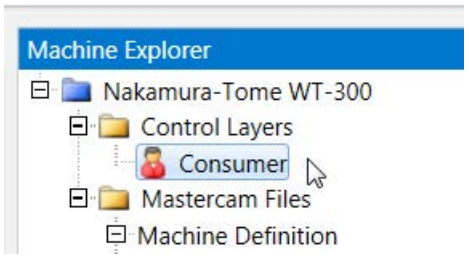
Follow these steps.



1. In the Sync Manager, click the little triangle next to the machine group name.
2. Select the **Activate Torque Skip Mode** option.
3. Enter the **Torque Limit**. This will be output as `G359 B___` in your NC code.
4. Press **[Ctrl+S]** to save the settings back to your part file.

Setting the default options

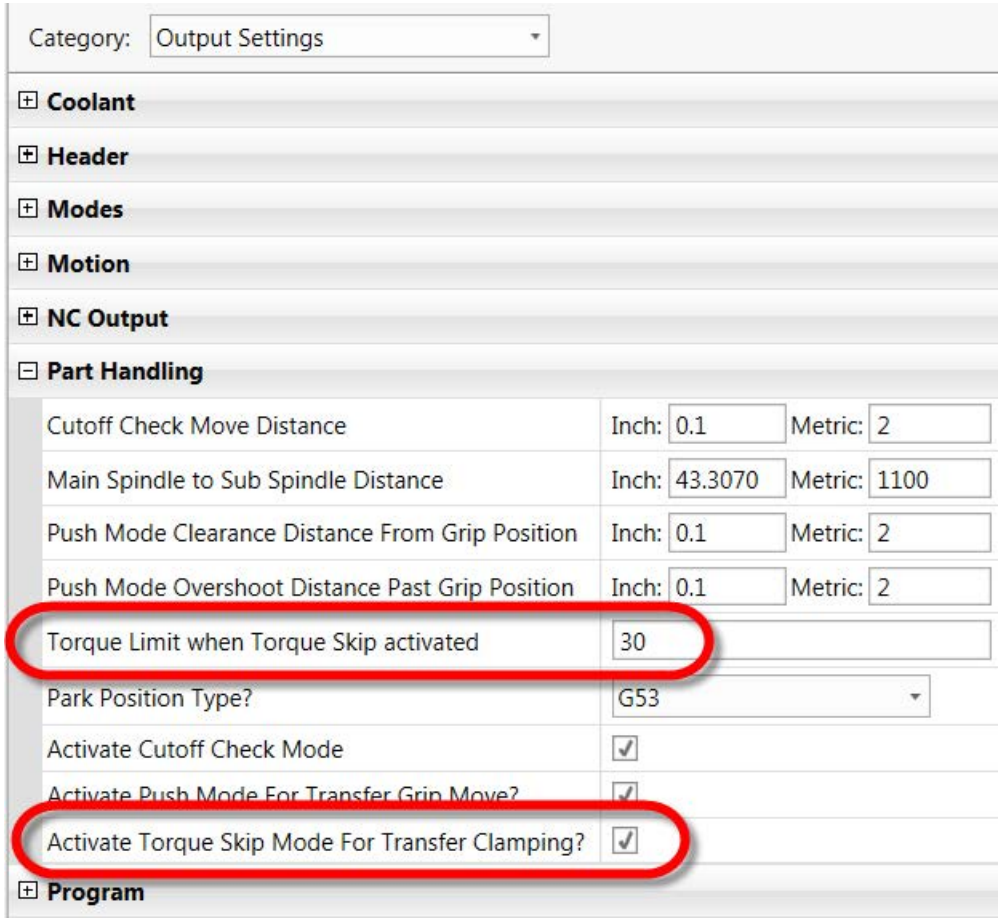
You can choose to enable or disable torque skip mode by default. Follow these steps:



1. Open the Nakamura-Tome WT/WY **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.

3. Select **Category: Output Settings**.

4. Go to the **Part Handling** section.
5. Set the options as desired.
6. Press **[Ctrl+S]** to save the **.machine** file.



Home

E: Approach and retract moves

Your Nakamura-Tome WT/WY **.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.

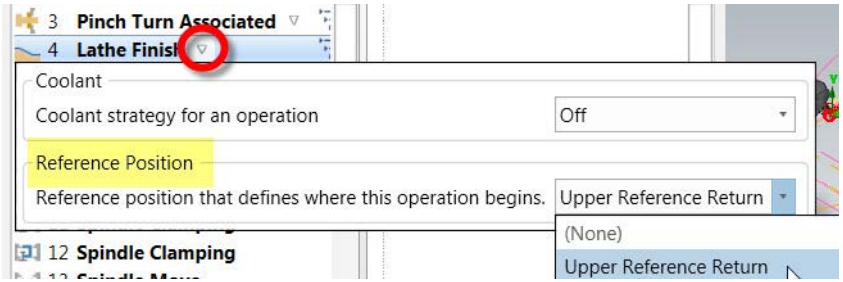
- Select specific positions for the start and end of each operation in the Sync Manager.
- You can also define additional, new reference positions. Do this in the Job Setup inside Mastercam. Do this if your particular part setup requires different reference locations than are already defined in your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

Home

Rev. date:
June 2018

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.



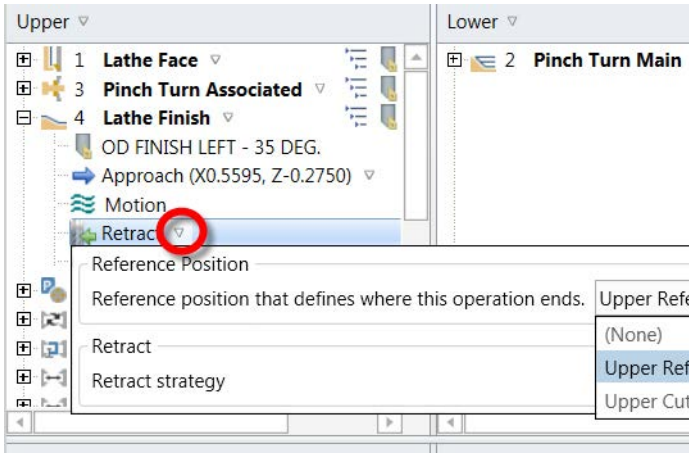
The selected reference position is typically output before the tool change. For example, the highlighted code shows the **Upper Reference Return** reference position from the previous picture

```
122  N3
123  (OPERATION # 4)
124  G28 U0. V0.
125  G28 W0.
126  (T0404  |  OD FINISH LEFT - 35 DE
127  M41
128  T0404 (OD FINISH LEFT - 35 DEG.)
129  G54
```

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.

```
118   G00 Z.25
119   G369
120   G53 U14. V0.
121   G53 W6.
122   G28 B0.
123   T03000
124   M05 P12
125   M01
```

Selecting a reference position of None

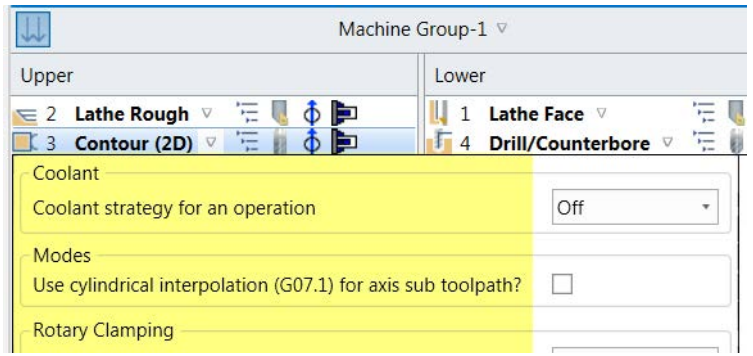
Selecting **None** for a reference position means that there will simply be no output where the reference position is typically output. For example, the code at right shows what happens if **None** is selected as the reference position for operation #2.

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

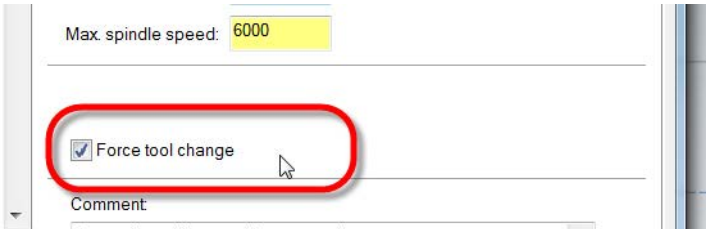
```
81    T02000
82    M05 P11
83    M01
84
85    N2
86    (OPERATION # 3)
87    (T03  |   3/16 FLAT ENDMILL  | DIA. -  0.
88    M35
89    G54
90    M06 T03003 (  3/16 FLAT ENDMILL )
```

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.

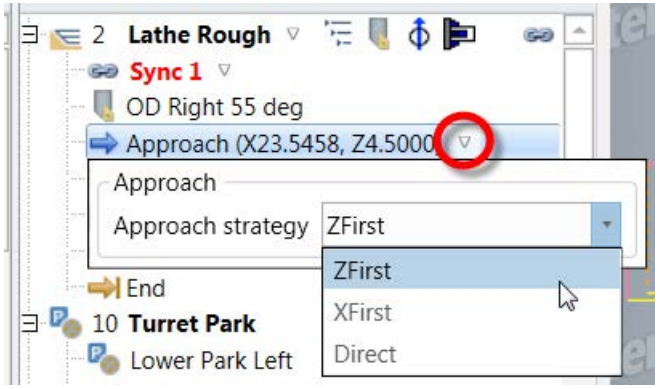


Setting the type of approach/retract motion

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

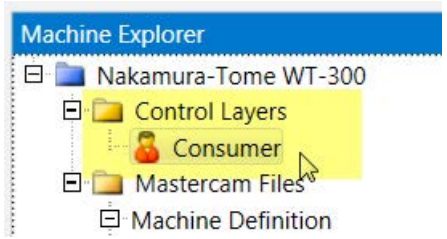
- X-first
- Z-first
- Direct (interpolated) move

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

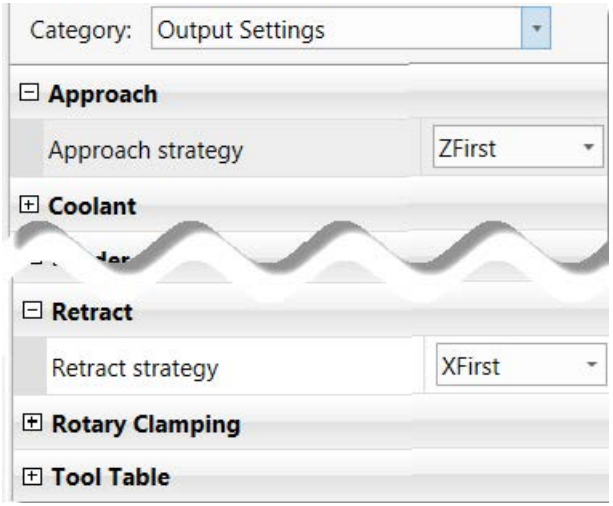


You can also change the default selection in the **.machine** file.

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.

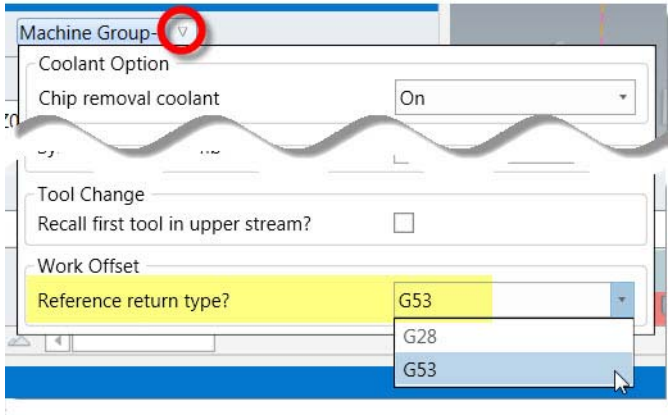


Home

Setting G28 or G53 output

You can choose to output the home position move as a G28 or G53 move. Click the small triangle next to the machine group name in the Sync Manager, and select the desired **Reference return type**.

For most applications, either G53 or G28 setting will work fine when you use the standard **Upper Reference Return** or **Lower Reference Return** positions. However, if you are using custom reference positions, outputting those coordinates with a G28 might not be appropriate.



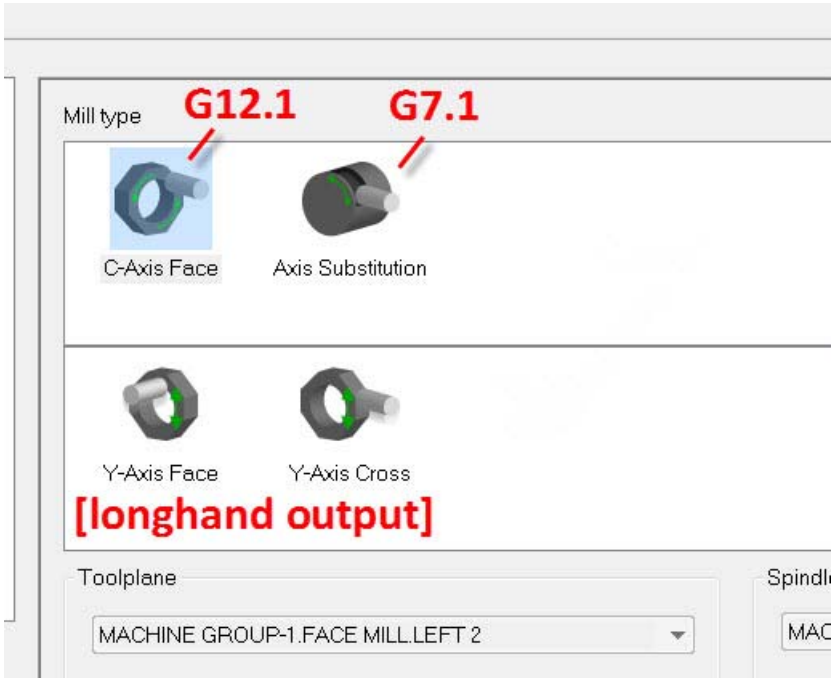
Home

F: Milling cycles and functions

Your Nakamura-Tome WT/WY **.machine** file supports the following milling cycles:

- G7.1 (polar interpolation)
- G12.1 (cylindrical interpolation)

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



A separate section describes **Clamping the spindle (M86/M87, M32/M33)**.

Home

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

Your Nakamura-Tome WT/WY **.machine** file supports both polar (G12.1) and cylindrical (G7.1) interpolation cycles. This topic explains how to:

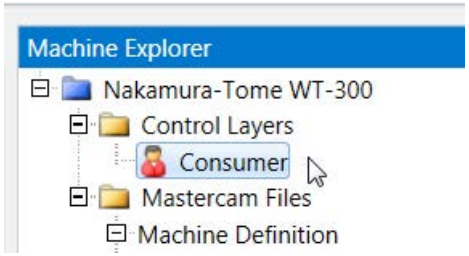
- configure default settings for these operations.
- select the cycle for each individual operation.

Default settings for G7.1 and G12.1

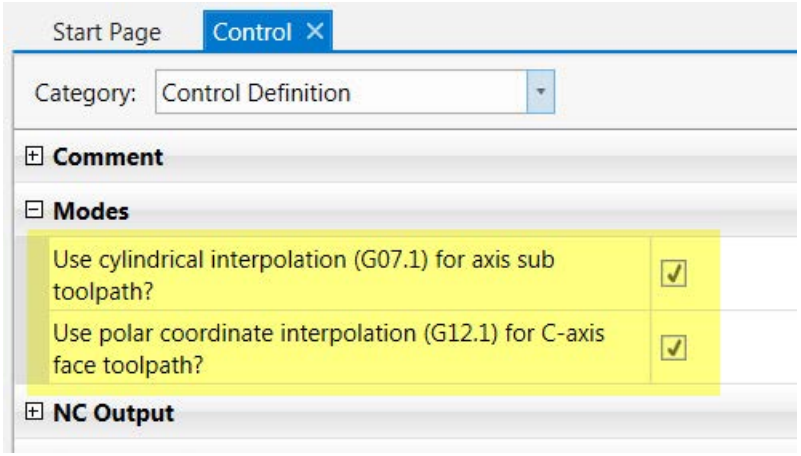
Follow these steps:

1. Open the Nakamura-Tome WT/WY **.machine** file in Code Expert.

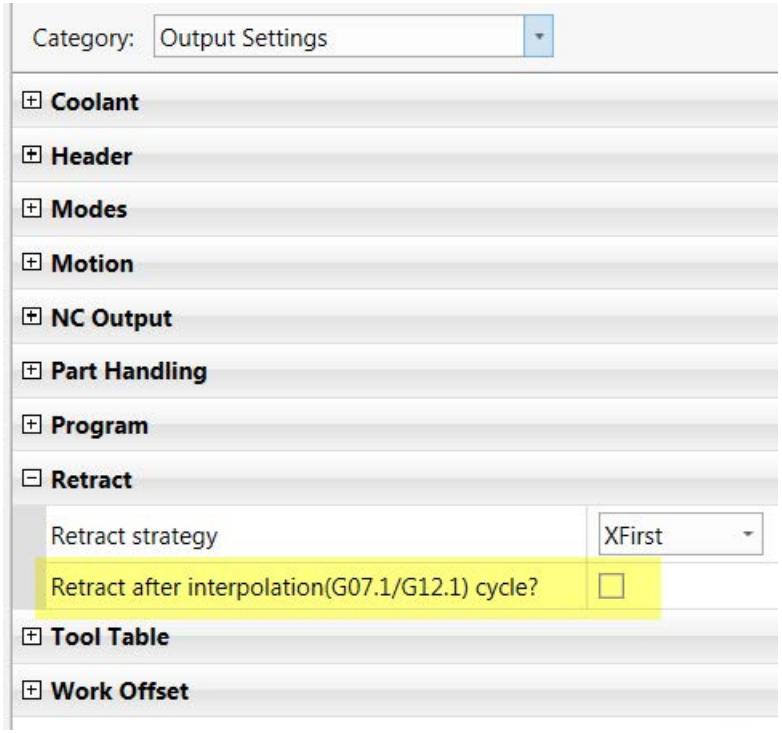
2. Double-click the **Consumer** layer.



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. Select **Category: Control Definition**.
7. Go to the **Retract** section.
8. You can choose to have the post add a home position retract after the cycle has completed. Select the **Retract after…** option to do this.
9. Save the **.machine** file.



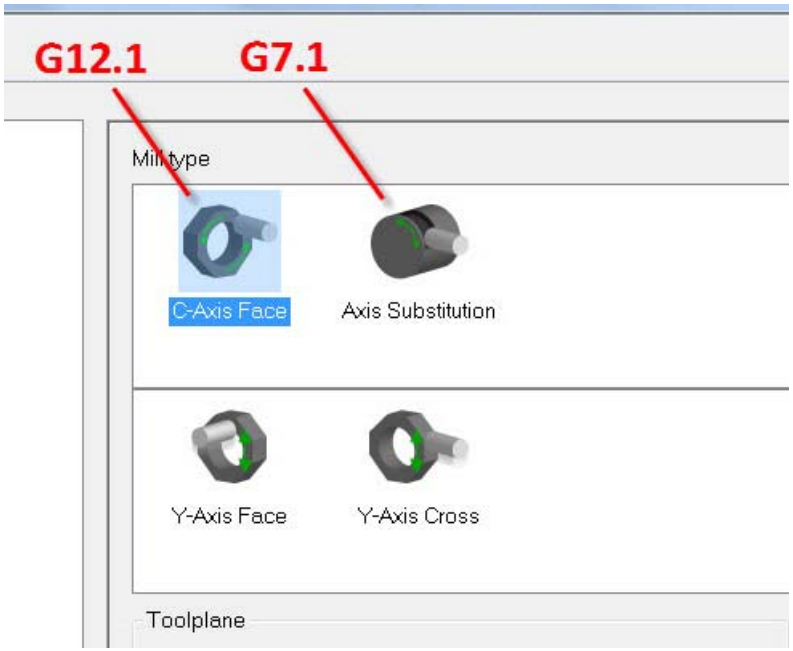
Home

Programming G7.1/G12.1 for each operation

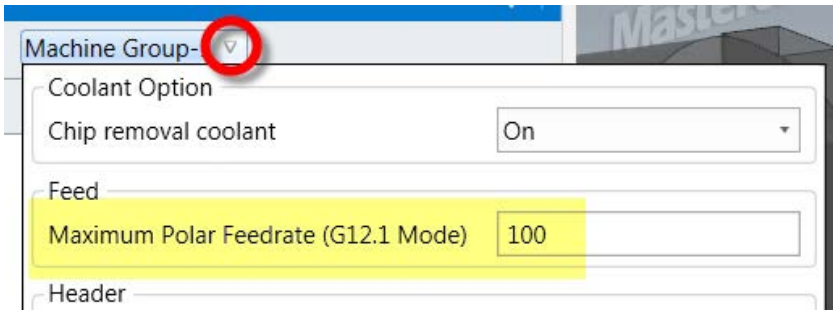
Follow this general workflow for programming these cycles for each operation.

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.



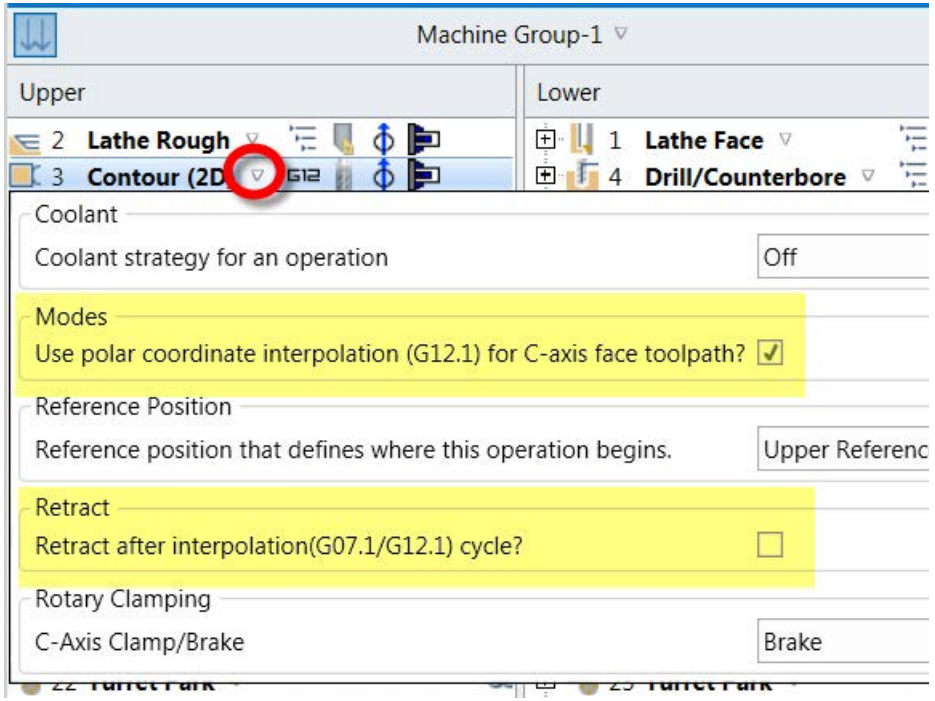
3. After you load the part in the Sync Manager, set the maximum polar feedrate. This is a machine group option that applies to the entire part. Click the small triangle next to the group name and enter the desired value.



4. 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 cycle option if desires, or leave it unchecked for longhand output.

5. You can choose to have the post add a home position retract after the cycle has completed. Select the **Retract after…** option to do this.

6. Press **[Ctrl+S]** to save the settings back to your part file.



Home

Clamping the spindle (M86/M87, M32/M33)

The C-axis high-pressure clamp mode is available for Mill toolpaths.

Set this option in the Sync Manager. Click the small triangle next to the toolpath name and select the desired **C-axis Clamp/Brake** mode:

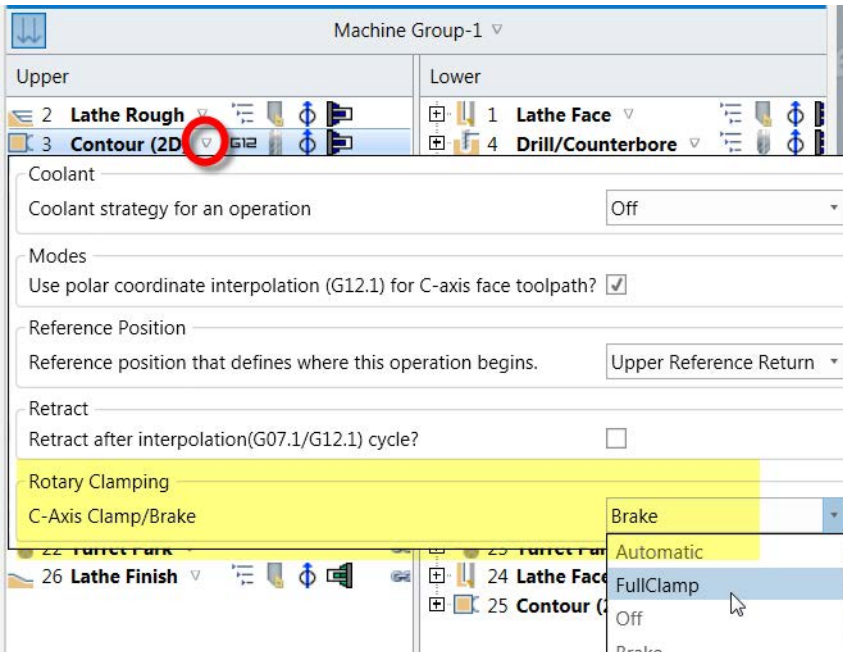
- **Automatic**—Mastercam will automatically decide whether to clamp the C-axis based on the application and toolpath type.
- **FullClamp**—Force C-axis clamp.
- **Off**—Suppress C-axis clamp for the selected operation.

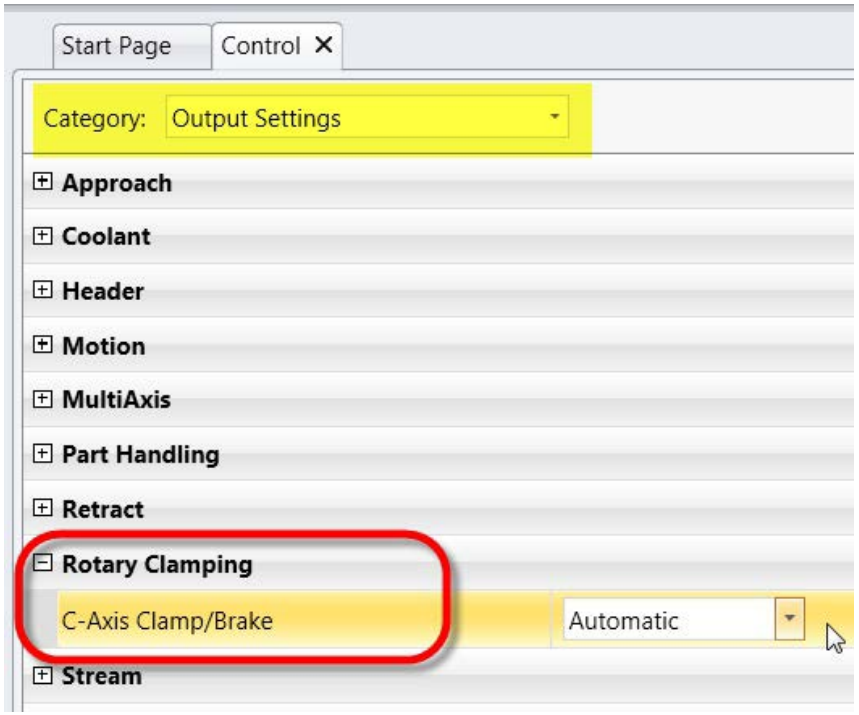
Default settings

You can choose which C-axis clamp mode will be the default

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 **Rotary Clamping** section.
5. Select the desired **C-axis Clamp/Brake** mode:
   - **Automatic**—Mastercam will automatically decide whether to clamp the C-axis based on the application and toolpath type.
   - **FullClamp**—Force C-axis clamp output.
   - **Off**—Suppress C-axis clamp output for the selected operation.
6. Save the **.machine** file.





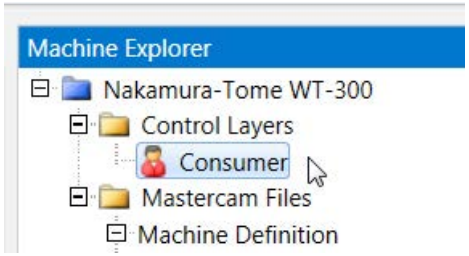
Home

G: Balanced turning and pinch turning

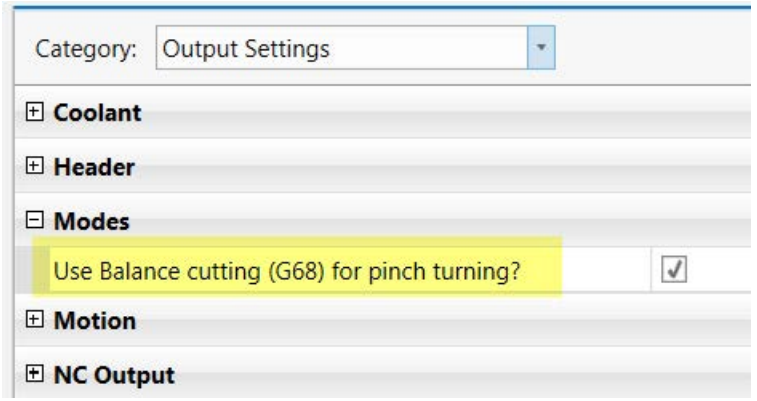
Your Nakamura-Tome WT/WY .machine file supports G68 mode for balanced turning. Use Mastercam's **Pinch Turn** toolpath to create pinch turning and balanced turning operations.

You can choose whether or not to use G68 mode for your balanced turning operations. To output the G68 code, follow these steps:

1. Open the Nakamura-Tome WT/WY **.machine** file in Code Expert.



2. Double-click the **Consumer** layer.
3. Select **Category: Output Settings**.
4. Go to the **Modes** section.
5. Select the **Use Balance cutting (G68)…** option.
6. Save the **.machine** file when you are done.



If you do not select this option, you will still be able to post out balanced turning operations, but they will not have a G68 code.

Home

Numbering syncs for pinch turn operations

Mastercam automatically generates and numbers the proper wait codes when you generate a pinch turn operation.

However, when you create your own syncs in the Sync Manager between other operations, these will be numbered higher than the ones created for the pinch turn operation, even if the operations occur earlier. Choose the **Renumber** command from the ribbon bar to fix this.

This will result in all of your syncs being numbered consecutively—your own plus the pinch-turning syncs created by Mastercam.

```
66   G00 X1.8325 Z.075
67   G50 S4500
68   S200
69   M101 (WAIT)
70   G68
71   G99 G01 Z-.025 F.01
72   Z-1.8009
73   G03 X2.02 Z-2.0149 R.2912
74   G01 Z-2.5476
75   X2.1614 Z-2.4769
76   G00 Z.075
77   X1.4575
78   M102 (WAIT)
79   G01 Z-.025
80   Z-1.6431
81   X1.665 Z-1.7302
82   X1.8064 Z-1.6595
83   G00 Z.075
84   X1.0825
85   M103 (WAIT)
86   G01 Z-.025
87   Z-.3671
```

Home

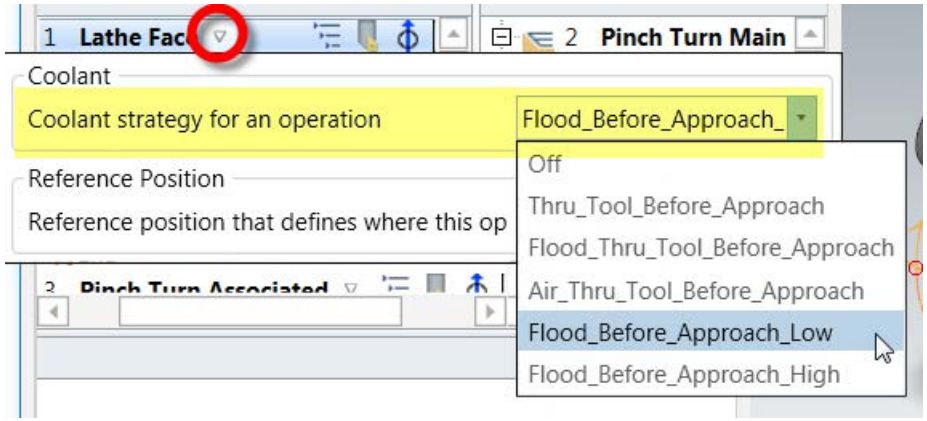
H: Using coolant

Your Nakamura-Tome WT/WY **.machine** file supports the following coolant options:

- Low-pressure flood coolant (`M07`)
- High-pressure flood coolant (`M08`)
- Air blast (`M20`)
- Thru-tool air blast for the milling spindle (`M740`)
- Chip removal air blast (`M706`)

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; the **Coolant** button is no longer present inside Mastercam.

For each operation, click the small triangle next to the operation name and select the desired coolant option:



Thru Tool Before Approach—Turn on thru-tool air blast (`M740`) before the approach move, and automatically turn it off before the retract.

Flood Thru Tool Before Approach—Turn on both flood coolant and the thru-tool air blast (`M07` and `M740`) before the approach move. Coolant is automatically turned off before the retract.

Air Thru Tool Before Approach—Use air blast (`M20`) before the approach move, and automatically turn it off before the retract.

Flood Before Approach Low—Turn on low-pressure flood coolant (`M07`) before the approach move. Coolant is automatically turned off before the retract.

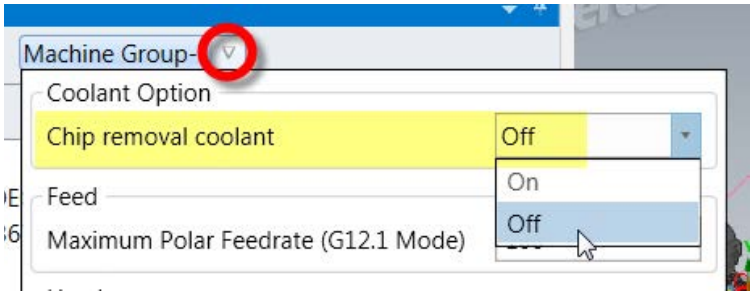
Flood Before Approach High—Turn on high-pressure flood coolant (`M08`) before the approach move. Coolant is automatically turned off before the retract.

Home

Off—Force coolant off. Note that it is not necessary to use this option to turn off coolant per operation when it has been turned on with one of the above strategies.

Chip removal air blast (M706)

The chip removal air blast (`M706`) is controlled separately from the other coolant options. It is 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.

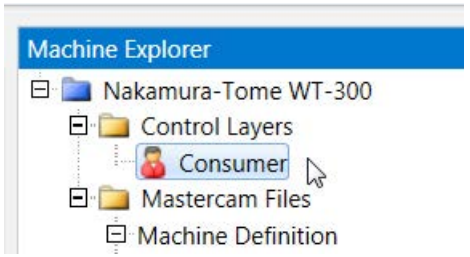


Setting the default coolant option

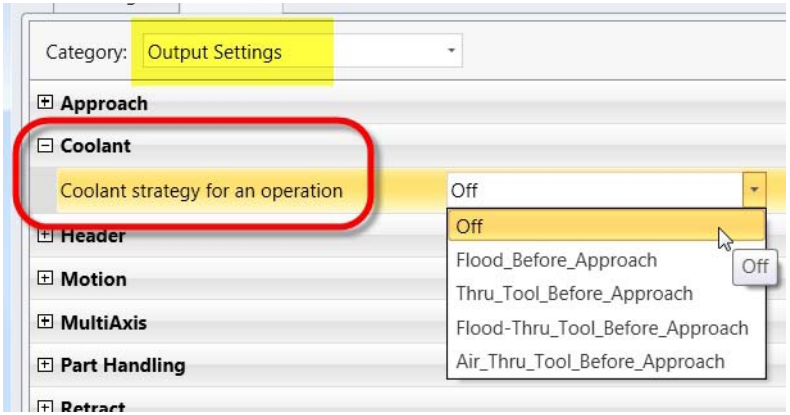
You can select a default coolant strategy and save it in the **.machine** file.

1. Open the Nakamura-Tome WT/WY **.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.



Home