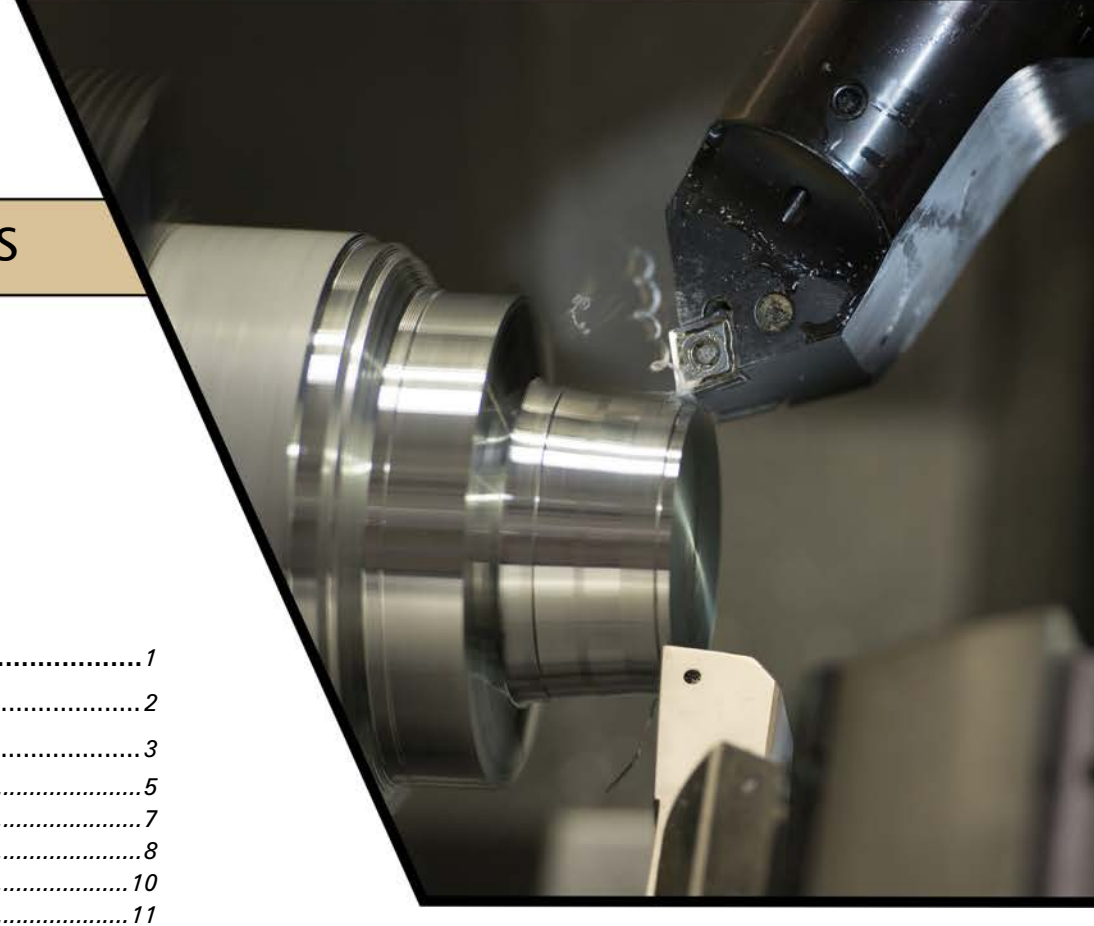
MILL-TURN APPLICATION GUIDES

DMG Mori NT/NTX



Contents

Chapter 1: Working with .machine files *1*

A: Installing your .machine file *2*

B: Customizing your .machine file *3*

Customizing operation defaults and tool libraries *5*
Default values for .machine settings *7*
Configuring the Code Expert editor *8*
Configuring tool table output *10*
Setting the toolpath directory and stream name/number *11*

C: Managing machine licenses *14*

Verifying that you have the proper license *15*
Reactivating disabled licenses *17*

Chapter 2: Working with tools and spindles *19*

A: Setting up tools for the B-axis head *20*

Initial tool setup *21*
Selecting the spindle and turret *25*
Rotating the B-axis head to the proper position *26*
Setting the tool number base for B-axis tools *30*
Using heavy tools *31*
G361/G362 tool change mode *32*

B: Setting up tools on the lower turret *33*

Live tool support *35*

C: Part handling operations and peripheral components *36*

Using the workpiece push check function (G38) *37*
Chip conveyor (M200)/door control (M220) *40*
Part ejector (M47) and part catcher (M73) support *42*

revision date: Aug. 15, 2016

Mastercam

Chapter 3: Working with toolpaths *45*

A: DMG Mori NT/NTX machining modes *46*

Polar (G12.1) and cylindrical (G7.1) interpolation *47*
Tilted-plane machining cycle (G68.1) *49*
Selecting the cutting mode (G332) *53*
Radius/diameter mode (G10.9) *54*

B: Balanced turning operations *55*

Setting up tools for pinch turn operations *56*
Numbering syncs for pinch turn operations *59*

C: Multiaxis toolpath settings *60*

Rotary start position *61*
Pole handling *64*
Using tool center programming (G43.4) *66*
Using AI contour control (G05.1) *68*

D: Reference points and home positions *69*

Setting the start point and end point for an operation *70*
Creating custom reference positions *75*

E: Coolant and other toolpath options *79*

Using coolant *80*
Configuring stops and optional stops *82*
Outputting the operation number for each toolpath *85*

Copyright © 2016 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: **www.mastercam.com/legal/licenseagreement/**

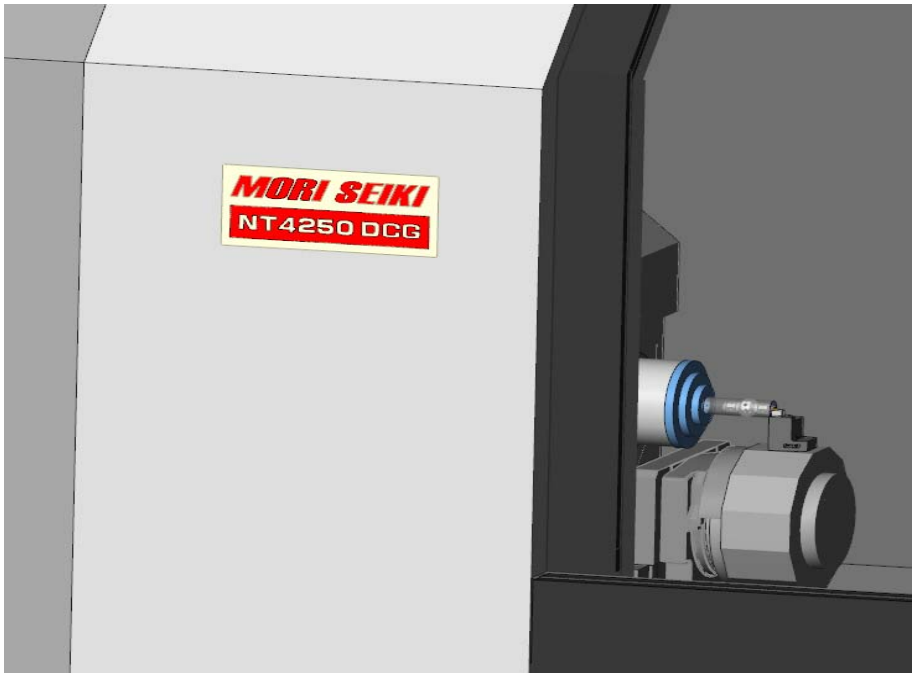
Chapter 1: Working with .machine files

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

- **Installing your .machine file**
- **Customizing operation defaults and tool libraries**
- **Default values for .machine settings**
- **Configuring the Code Expert editor**
- **Configuring tool table output**
- **Setting the toolpath directory and stream name/ number**
- **Managing machine licenses**

This book is intended to support Mastercam 2017. Please understand that you cannot use an X8 or X9 **.machine** file in 2017, and vice versa. If necessary, use Mastercam's migration utility to create a 2017 **.machine** file.

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



Home

A: Installing your .machine file

Your **.machine** file is packaged in a .zip file. Simply unzip it to your desired location.

The default location for **.machine** files is the

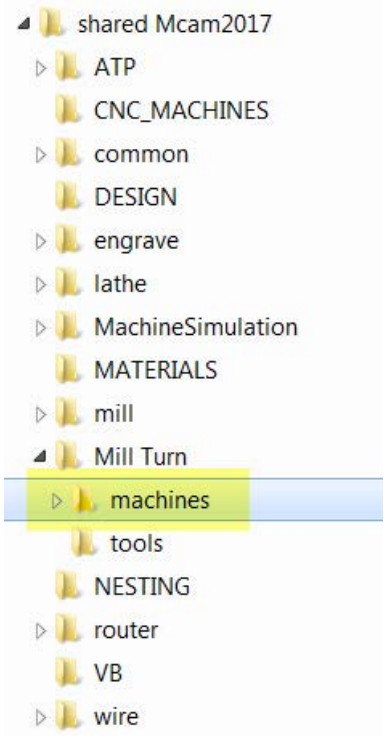
\shared Mcam2017\Mill Turn\MACHINES

folder. However, you can place it anywhere you wish.

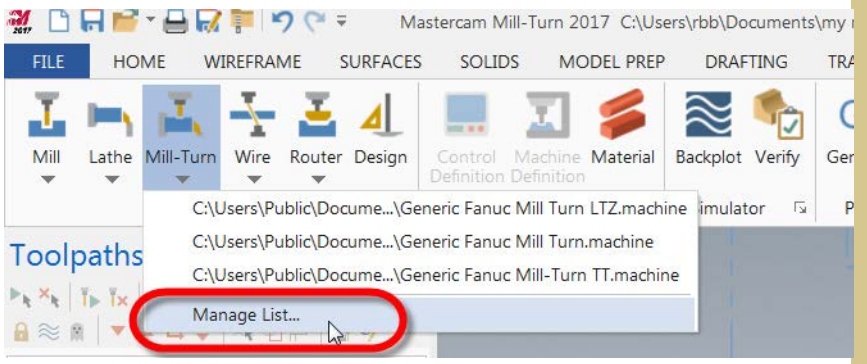
Unlike legacy Mastercam, the single **.machine** file includes all the resources that you need to support your DMG Mori NT/NTX application. You do not need to worry about linking the **.machine** file to other files, like posts.

Running Mill-Turn from a network location—To work with your **.machine** file from a network location, simply copy it to the desired location on your network. Since the **.machine** file does not point to any other files, there is no difference between a network location or a local drive.

Note, however, that Mastercam will look for the **.machine** file every time you load a part that uses it, so it does need to be in a location that your workstation is regularly connected to.



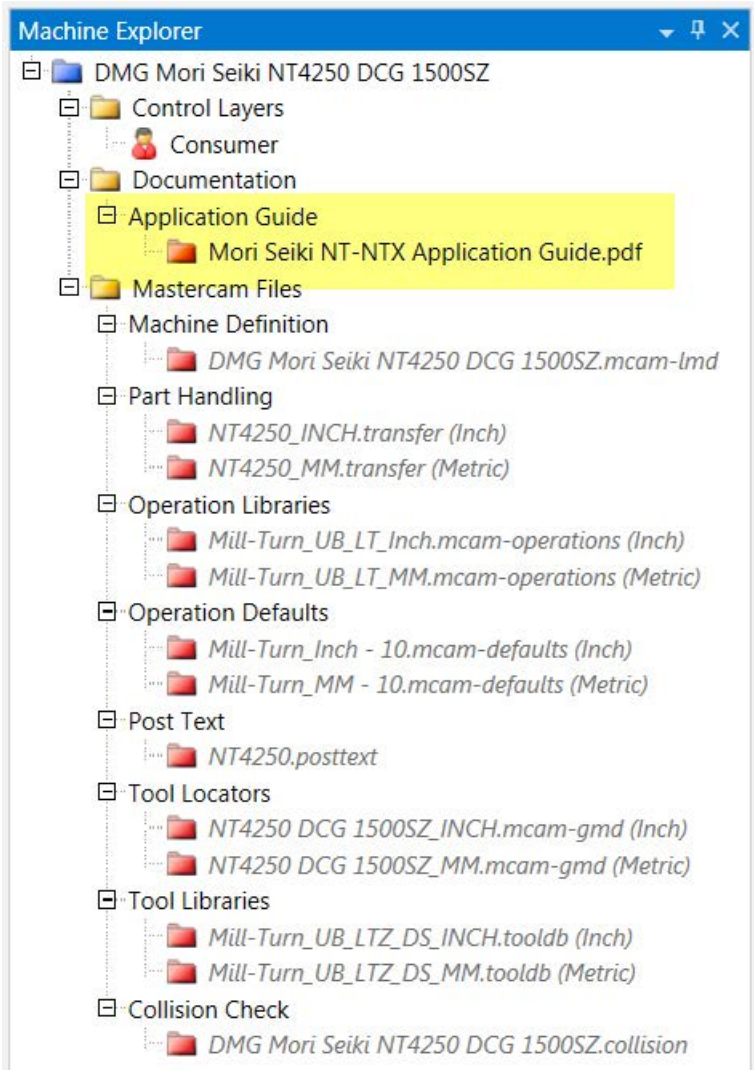
Adding the .machine file to the menu—To make your **.machine** file available on the machine list in Mastercam, select **Machine > Mill-Turn > Manage list**. Then navigate to the folder with your **.machine** file and select it.



B: Customizing your .machine file

Whenever you load your **.machine** file in Mastercam to begin working on a DMG Mori NT/NTX part, Mastercam also starts up the Code Expert and loads your **.machine** file there as well. You can see the Code Expert icon appear on your task bar, right next to Mastercam.

When you switch to Code Expert and click on the Machine Explorer, you can see all the different components of your **.machine** file.



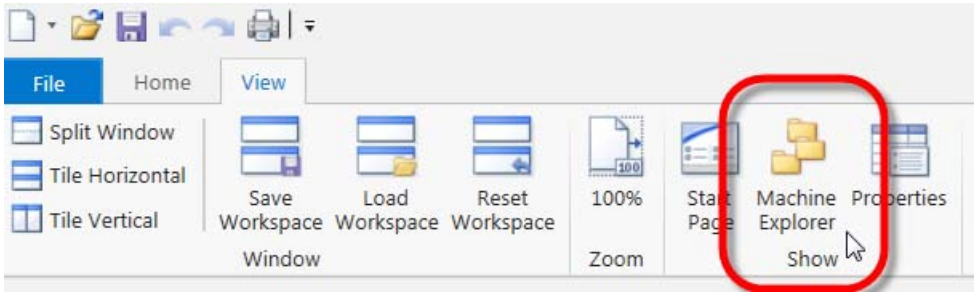
- The **Control Layers** section lists the resources used by the post. The **Consumer** layer is designed specifically for machine defaults and other settings that you are allowed to edit. You will do this often in the other procedures in this book.
- The **Mastercam Files** section lists the support files that are encapsulated inside the your **.machine** file. Users of other Mastercam products are familiar with these files being stored in many locations throughout your Mastercam installation. In Mill-Turn, they are all brought together inside the **.machine** file.

  They are grayed out because you cannot edit them in Code Expert. However, you can edit common files like your operation defaults, operation libraries, and tool libraries inside Mastercam. You just need to follow a slightly different workflow than you are used to. Follow the guidelines in the next section.
- Beginning in Mastercam 2017, you can access this application guide directly from Code Expert: just double-click the **.pdf** file under **Application Guide**.

Home

Opening the Machine Explorer—Before you can work with the **.machine** file, the Machine Explorer needs to be visible.

Simply go to the **View** menu inside Code Expert and click the **Machine Explorer** button.



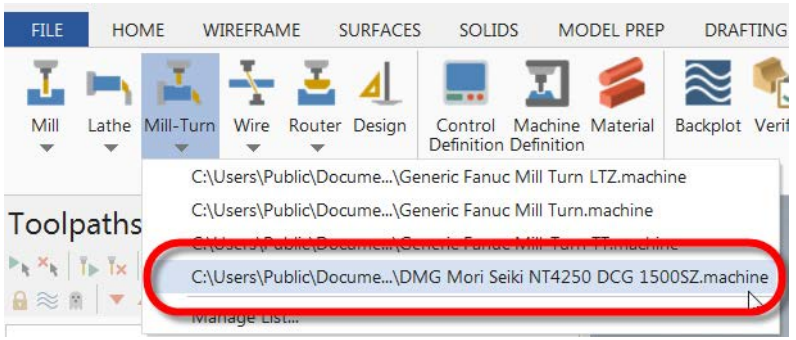
Home

Customizing operation defaults and tool libraries

Load the .machine file in Mastercam to edit tool libraries and defaults.

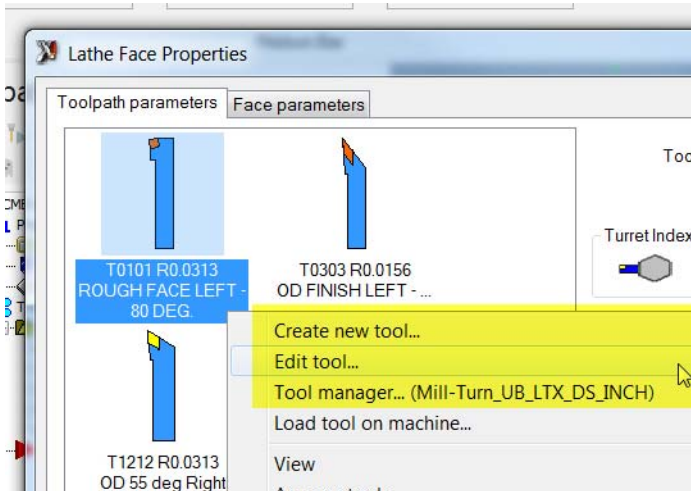
In legacy Mastercam you are used to working with tool libraries and **.defaults** files by simply loading them from your hard disk. The workflow is a little different in Mill-Turn because in Mill-Turn, these files are stored inside the **.machine** file. Follow this general outline.

1. Start up Mastercam.
2. Load the desired **.machine** file or a part that uses the **.machine** file.
3. Use Mastercam's regular tools for editing these libraries:



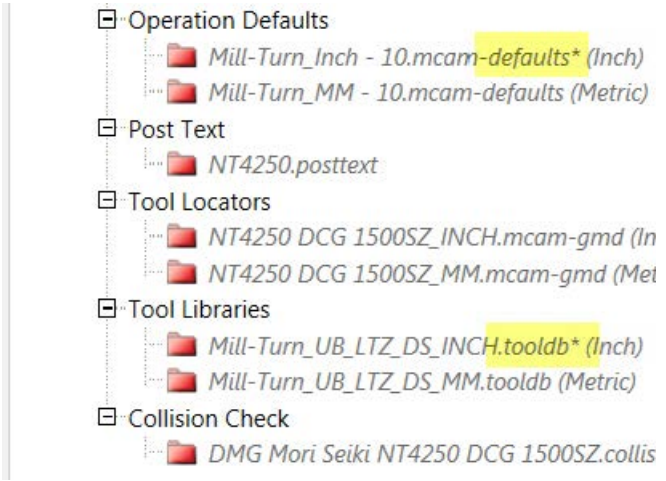
- You can use the **Edit** buttons on the **Machine Group Properties > Files** tab.

- You can also use the controls in the **Toolpath parameters** page for any operation.



Home

4. Save the changes in the locations that Mastercam prompts you with.
5. When you finish making changes, go back to Code Expert and look at the Machine Explorer. Any files that you changed should be marked “dirty” with an asterisk.

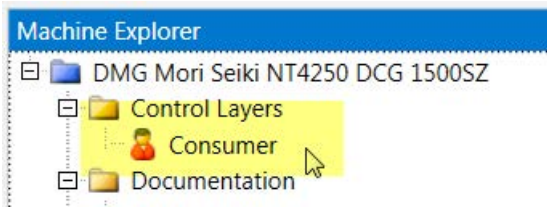


6. Press **[Ctrl+S]** in Code Expert to save the **.machine** file with your changes.

Home

Default values for .machine settings

The .machine file includes a number of configurable settings and defaults, similar to the control definition in legacy Mastercam.

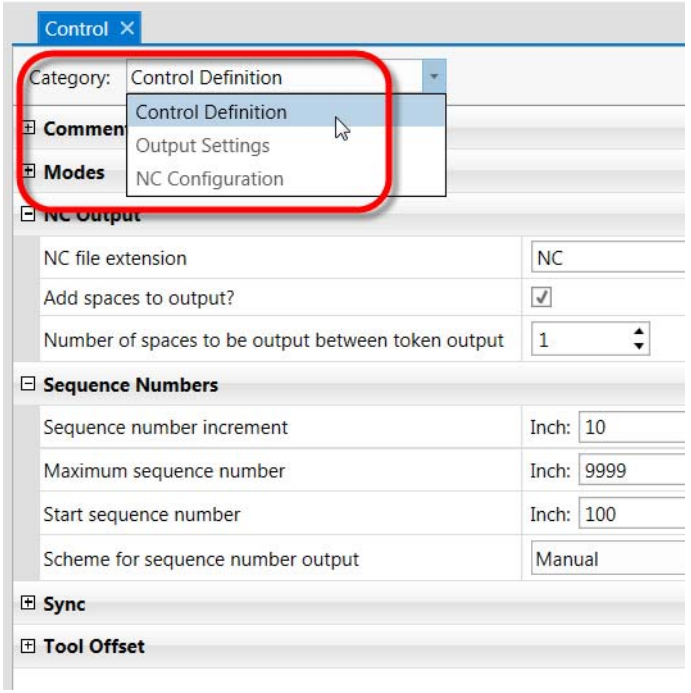


Although typically your **.machine** file will be supplied to you ready-to-use by your Reseller, it includes many settings that you can configure yourself according to your preferences and specific application needs. These include sequence and sync numbering, tool offset numbering, use of spaces in your 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.

Most of the settings are grouped into two categories:

- **Control Definition**
- **Output Settings**



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 very specific to individual machines. These settings serve a wide variety of functions:

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

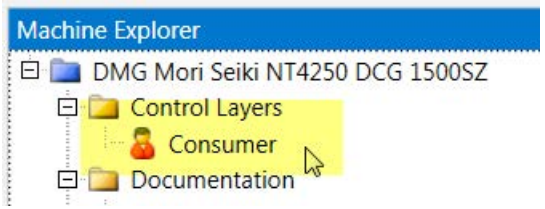
Most of these settings are self-explanatory and you can easily configure them to meet your needs by simply browsing the interface. The settings that are specific to this **.machine** file are described fully in this manual.

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

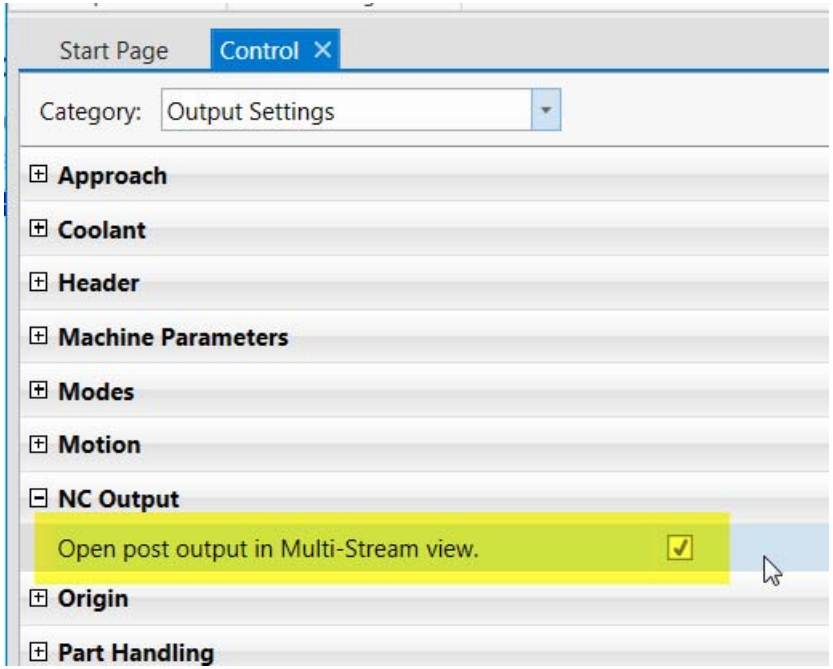
Home

Configuring the Code Expert editor

Several settings in your .machine file help configure the Code Expert editor.



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



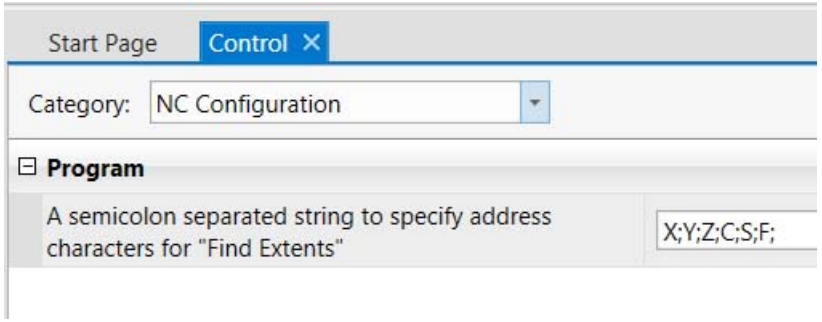
Opening in multi-stream view—You can use the Code Expert editor in either single-stream or multi-stream mode. Since the DMG Mori NT/NTX NC output is typically only a single stream, you may wish to clear this option by default.

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

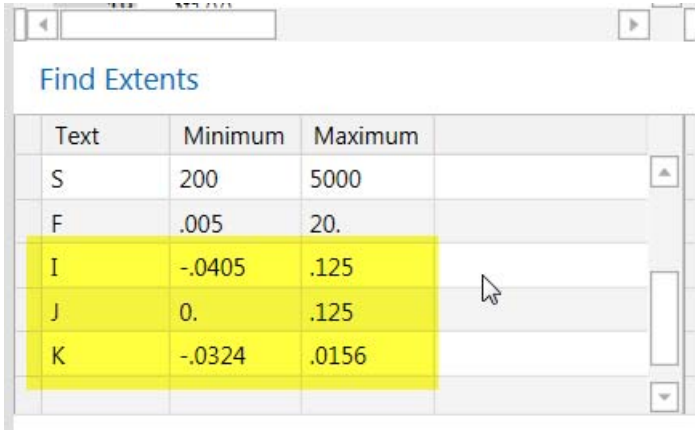
Home

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**, **B**, **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.



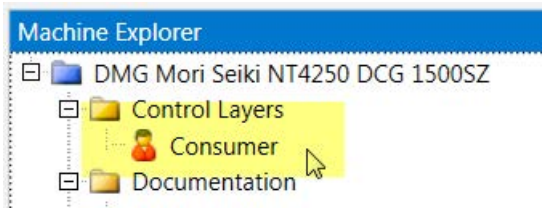
Configuring tool table output

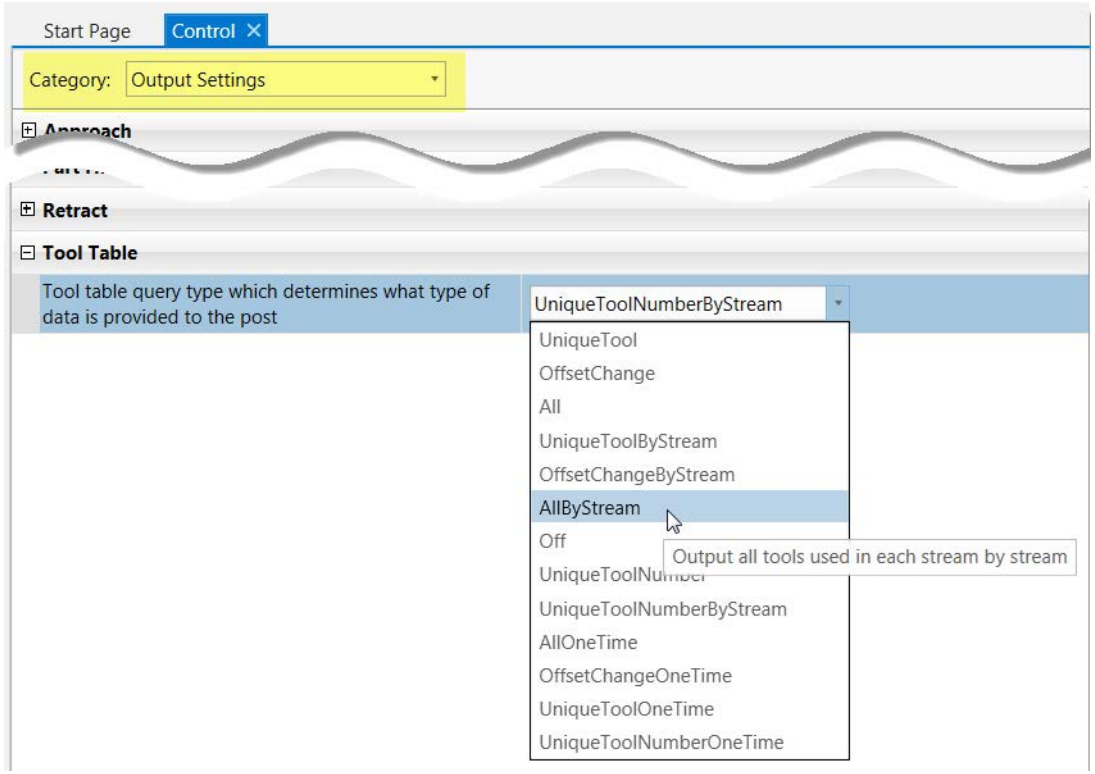
Use pre-defined strategies to get the tool table output you want.

As a user, you can configure your tool table output yourself without needing to do any programming or edits to your post. Your **.machine** file includes 12 different tool table strategies; simply select the desired one. 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.

Press **Ctrl+S** before posting to save your setting.

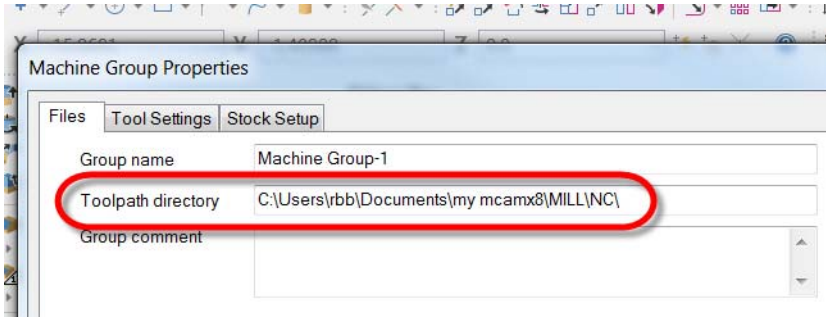




Setting the toolpath directory and stream name/number

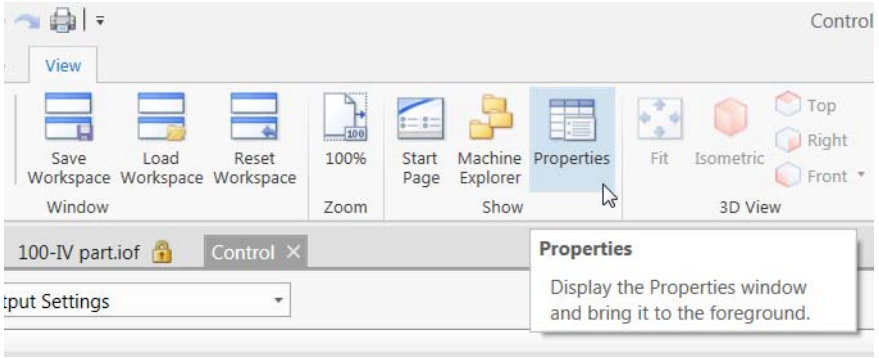
Several common posting options from regular Mastercam are replaced by machine options in Mill-Turn.

If you are familiar with Mastercam, you are probably familiar with the **Toolpath directory** setting in the **Machine Group Properties**.



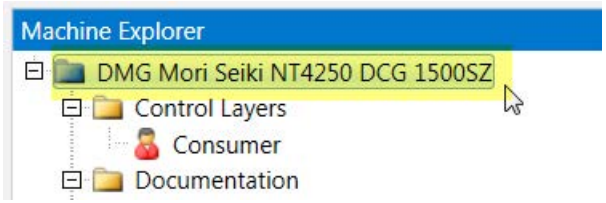
In Mill-Turn, this setting is not used. Instead, it is a property of the **.machine** file. To set it, follow these steps:

1. Open the **.machine** file in CodeExpert.



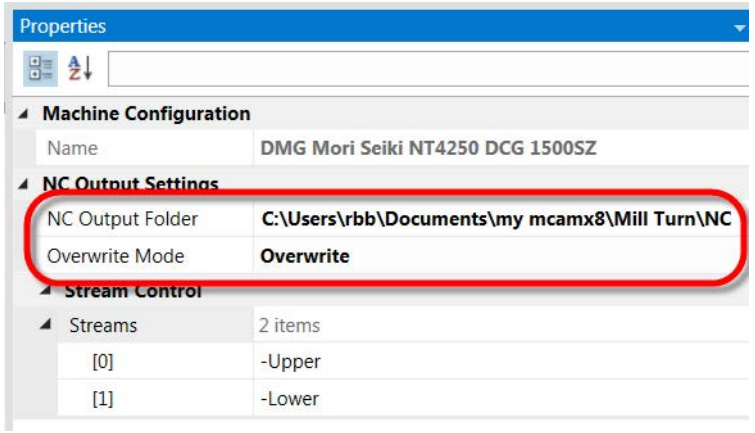
2. Make sure that the **Properties** window is displayed.

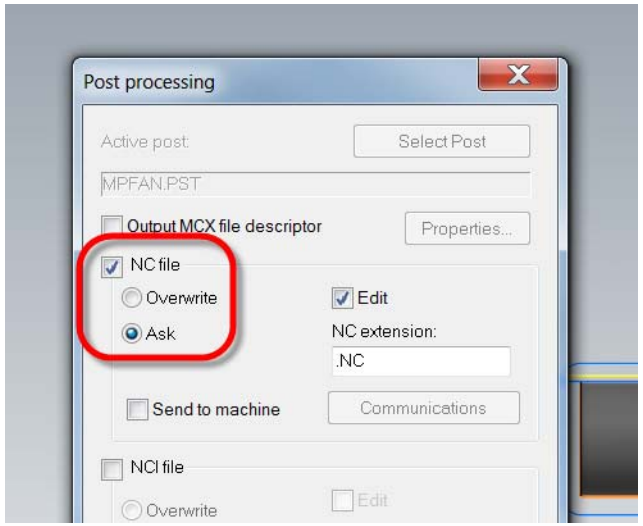
3. Click the machine name in the **Machine Explorer**.



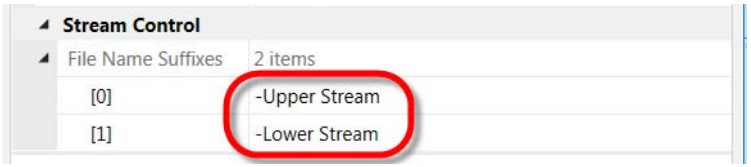
4. Select the desired **NC Output Folder**. Mastercam will write your NC files for this machine to this folder.
5. You can also choose whether Mastercam will automatically overwrite NC files with the same name, or prompt you to enter a different name. Select the desired **Overwrite Mode** to control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

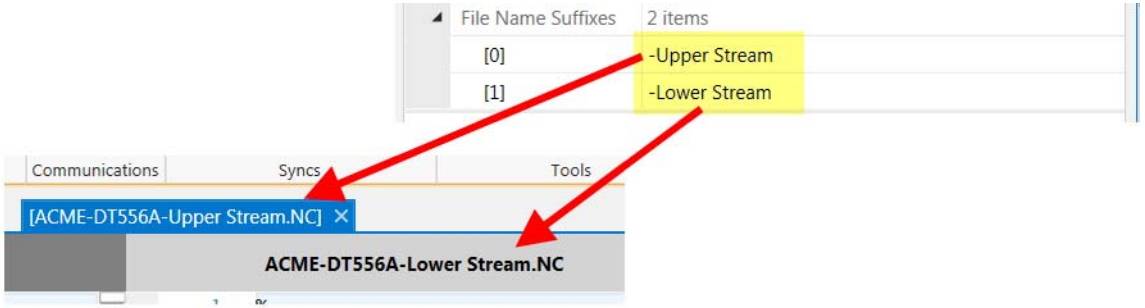
   This **Overwrite Mode** setting replaces the following setting from the **Posting** dialog box in regular Mastercam:





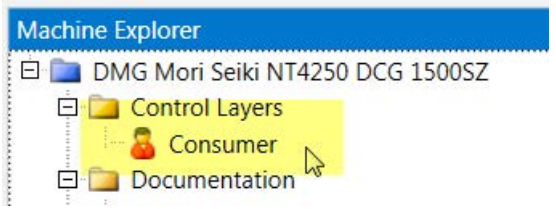
6. Mastercam Mill-Turn also lets you configure the names of the upper and lower streams. These will be automatically added to the NC file names for each stream. You can edit these names if you wish.



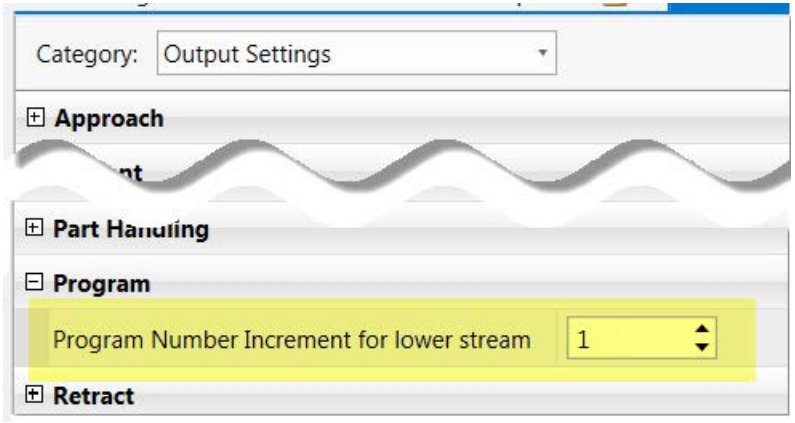


Your **.machine** file also gives you the option to configure the program number for the lower stream.

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 (`O0000n`).
11. Save the **.machine** file when you are done.

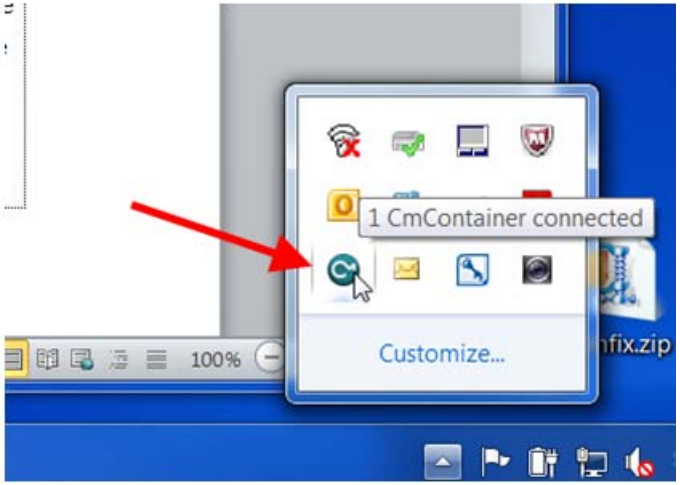


C: Managing machine licenses

Before you can use this Mill-Turn machine, Mastercam requires that you have a license for it.

- If you are familiar with other Mastercam products, you are probably familiar with HASPs—a small device that plugs into your USB port that Mastercam uses to make sure that you are an authorized user.
- Licenses for Mill-Turn machines are different. Mastercam uses a product called CodeMeter to administer these licenses.

You can access CodeMeter from your System Tray. Use it to inspect or manage your Mill-Turn licenses.



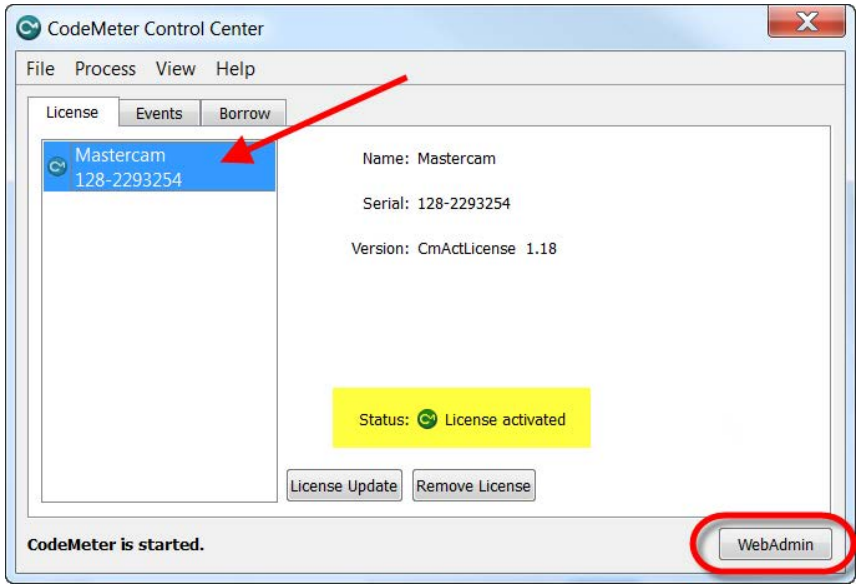
Home

Verifying that you have the proper license

Beginning with X8, a specific license is required for each machine.

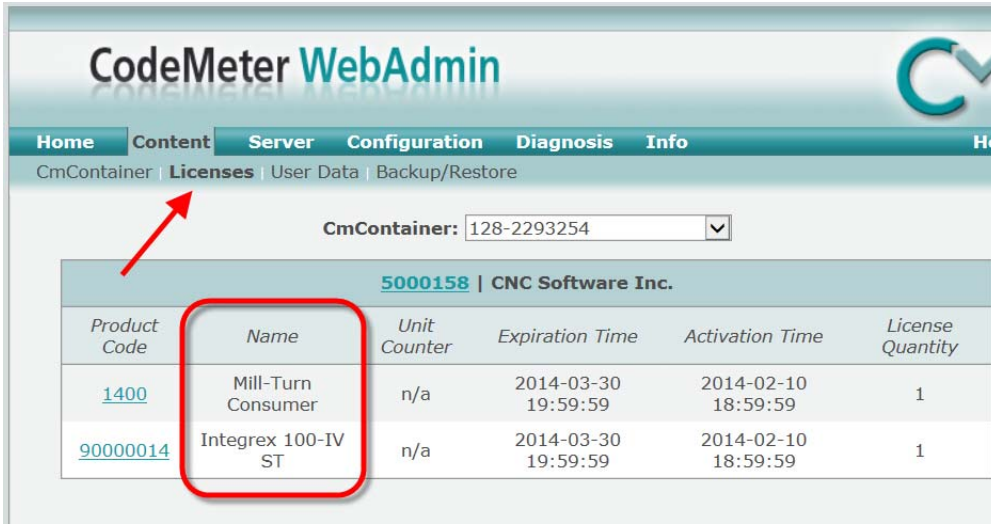
Follow these steps to see what licenses are available on your workstation and verify their status.

1. Start CodeMeter by clicking on it from the System Tray.
   - When CodeMeter opens, you should see a single item in the **Licenses** tab. This is a *container* that stores all the Mill-Turn licenses installed on your system.
   - The **Status** should tell you that they are **activated**.
2. Click the **WebAdmin** button to inspect the individual licenses.



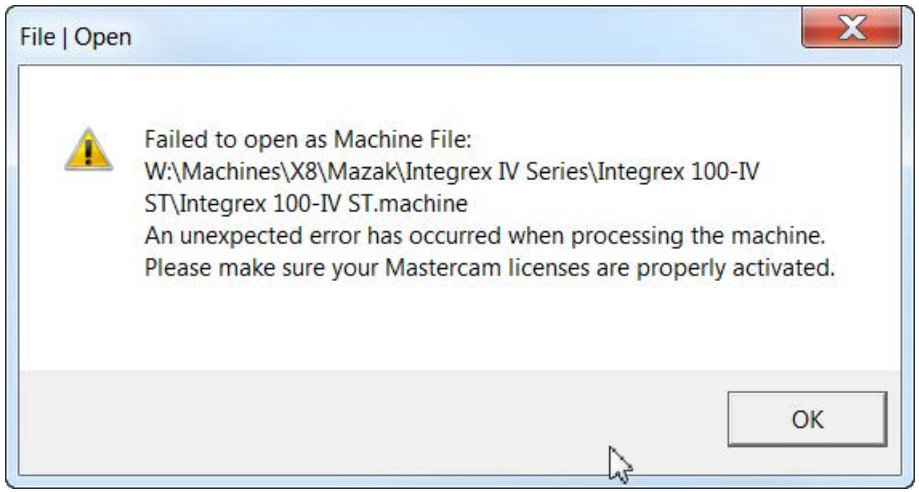
3. When the **WebAdmin** console starts up, go to the **Licenses** tab to see exactly what licenses are installed.

   This example shows two installed licenses: the base license that every Mill-Turn user must have, and one license for a machine. If you had licenses for more machines, they would be listed here.



Home

If you try to access a machine for which you do not have a license—or open a part file that uses such a machine—Mastercam displays the following error message:



Follow the steps in the next section or call your Reseller for assistance.

Home

Reactivating disabled licenses

If you lose the connection to your HASP, your machine license might be disabled.

If you feel that you have gotten the **Failed to open as machine file** message in error and you are sure the license has been installed on your system, the license may have become disabled.

To run properly, the CodeMeter application needs to maintain a constant connection with your HASP. If this connection is broken, your CodeMeter licenses cannot be activated and you will not be able to run Mill-Turn.

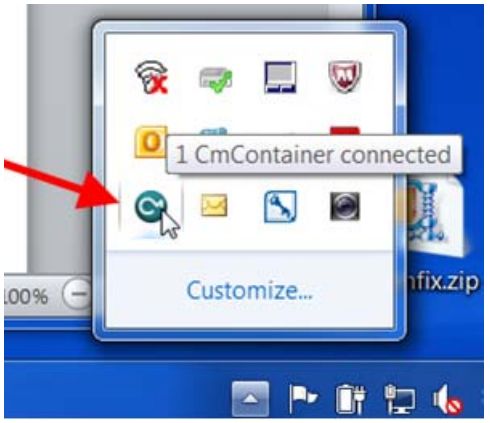
Some common reasons why this might happen include:

- You unplug your HASP before shutting down your workstation.
- You run on a NetHASP and the network connection is broken.
- Your computer has gone into a “sleep” or hibernation mode and CodeMeter can no longer detect the HASP.

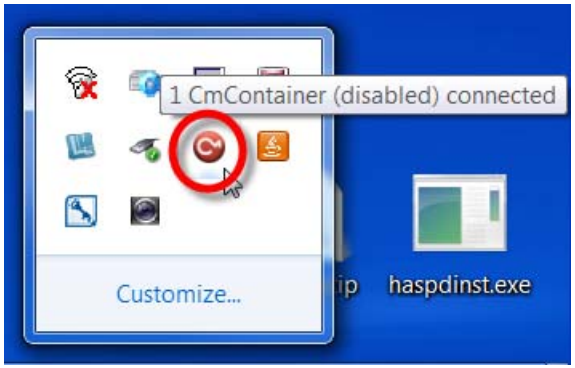
If your CodeMeter licenses are not valid for any reason, the icon in the System Tray changes color, from blue-green to red.

However, when this happens, you can easily repair the license with CodeMeter. Follow these steps:

1. Make sure that your workstation is properly connected to the HASP.
2. Start CodeMeter by clicking its icon in the System Tray.

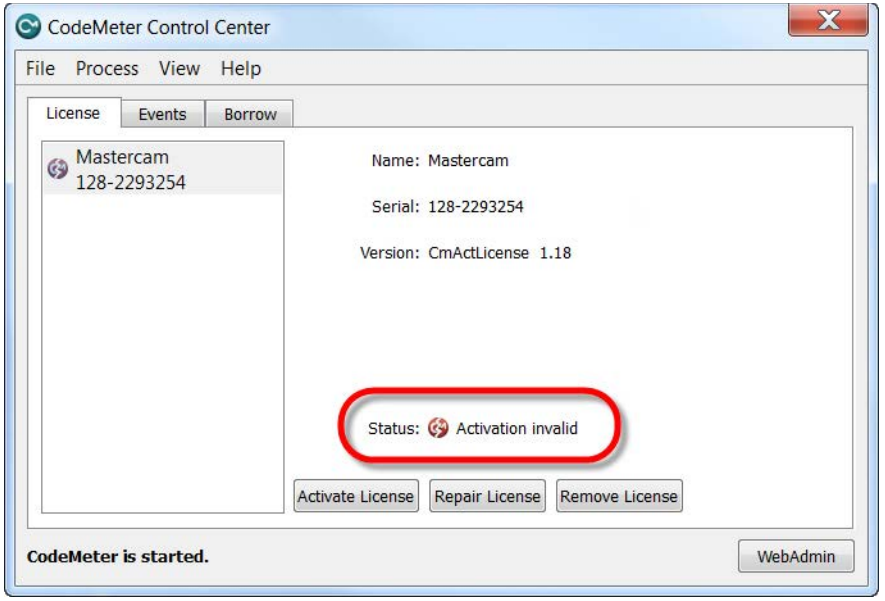


Blue-green icon = status OK



Red icon = disabled

Home



3. When CodeMeter starts, make sure your Mastercam license container is visible.

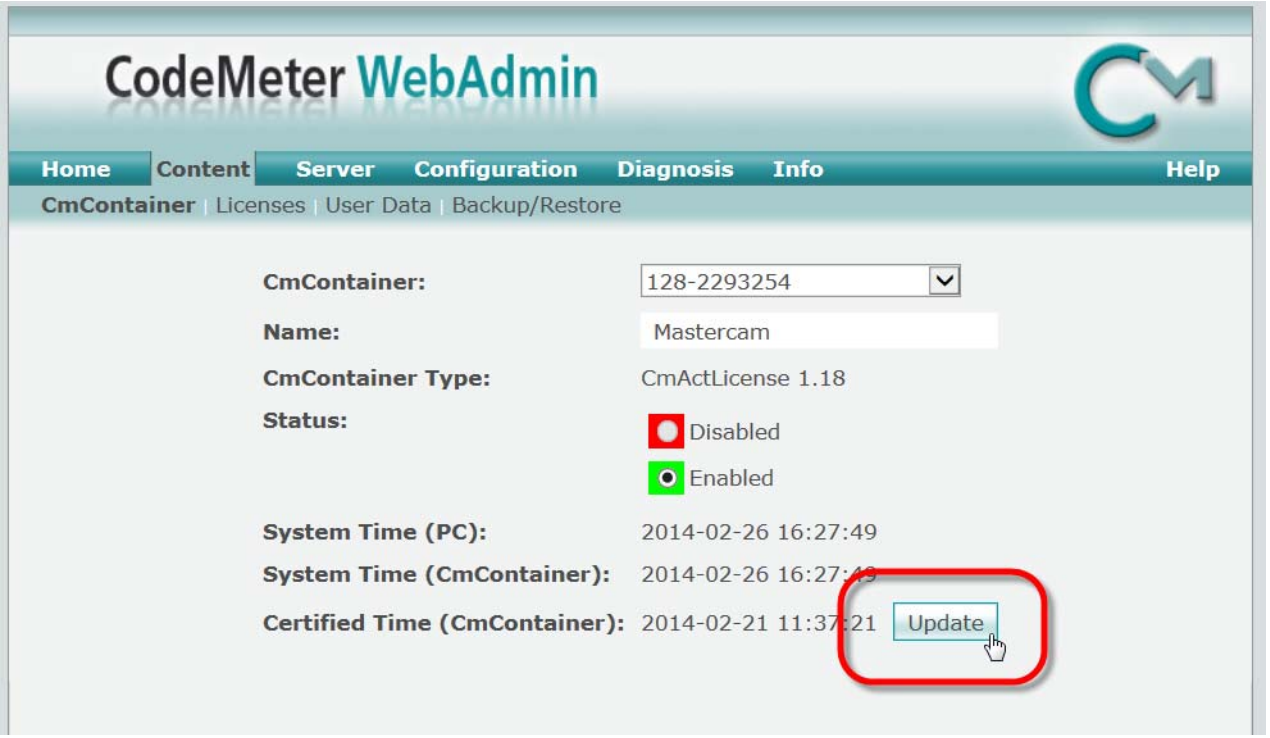
   CodeMeter should tell you that the license is invalid.

4. Click the **Repair License** button.

5. Click the **WebAdmin** button.

6. Click **Update**.

You should be able to run Mill-Turn normally at this point. Please contact your Reseller if you still have problems.



Chapter 2: Working with tools and spindles

Tool change and tool code output in your NC file is affected by:

- the tool definition (how the tool is set up in the tool library).
- toolpath settings (tool angle and orientation plus tool/offset number).

In addition, your **.machine** file includes a number of settings that are specific to DMG Mori NT/NTX machines.

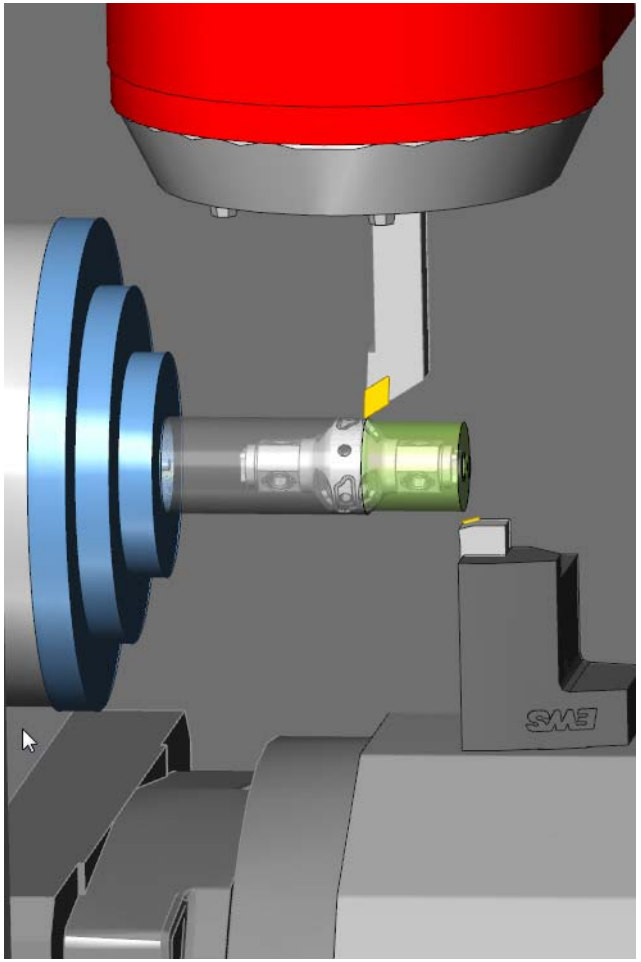
This chapter explains how these different settings work together to produce the proper NC output for your machine.

- **Setting up tools for the B-axis head**
- **Setting up tools on the lower turret**

See also “Setting up tools for pinch turn operations” on page 56 in the next chapter.

This chapter also includes a section about part handling/spindle pickoff operations and peripheral components.

- **Part handling operations and peripheral components**



Home

A: Setting up tools for the B-axis head

Use the tool definition to set the initial position, and the Tool Angle controls to orient it for each operation.

There are three main parts to making sure your NC code positions your tool properly:

- **Initial tool setup**
- **Selecting the spindle and turret**
- **Rotating the B-axis head to the proper position**

The **Initial tool setup** is typically only done once when a new tool is added to your library, while the other two are performed for each operation.

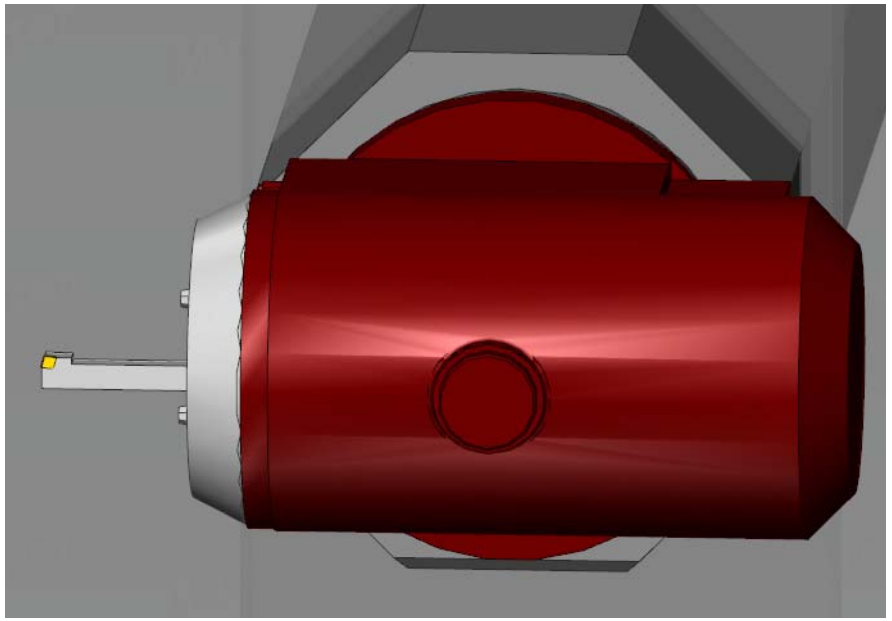
More options for B-axis tool support are available in the Sync Manager:

- **Setting the tool number base for B-axis tools**
- **Using heavy tools**
- **G361/G362 tool change mode**

Home

Initial tool setup

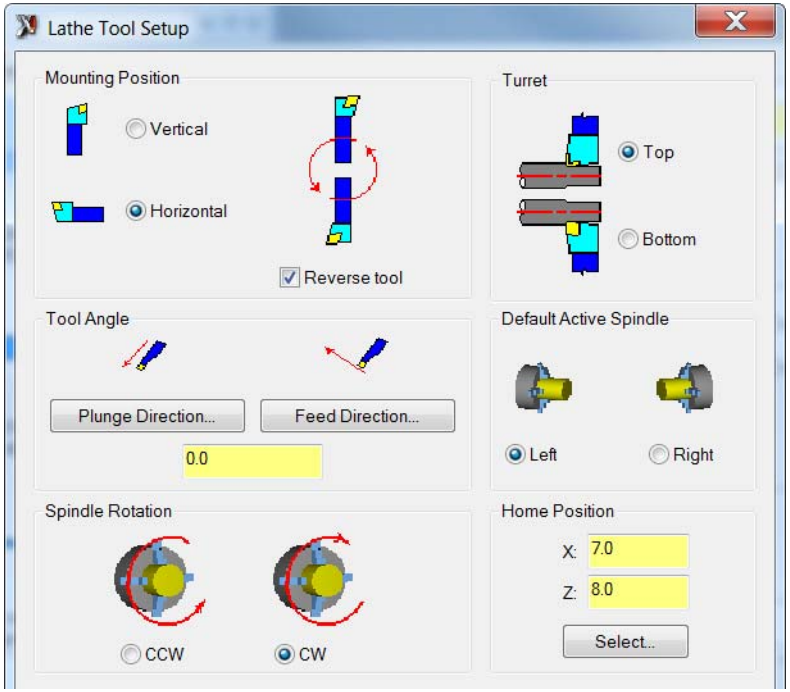
Tool definition settings for B-axis tools.



By convention, the tool definition should reflect the tool change position and orientation.

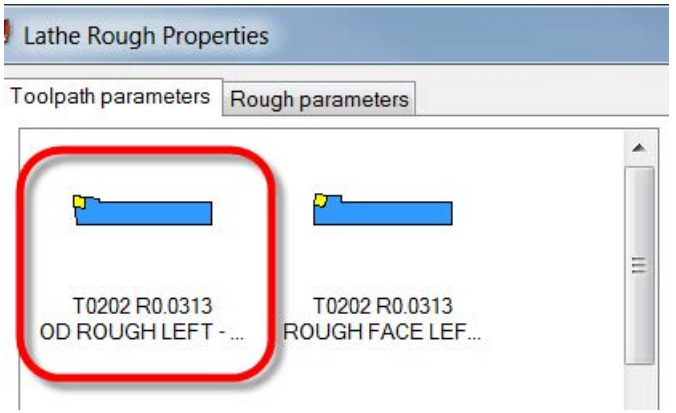
- Tools should be defined so that their initial orientation matches the B-axis tool change position.
- For the DMG Mori NT/NTX, this means the tool axis is horizontal, pointed toward the left spindle as shown in the picture.

These settings are stored in the Mastercam tool definition. This is what the settings should look like in Mastercam, assuming that the tool in mounted insert-up at the tool change as shown in the above picture:.



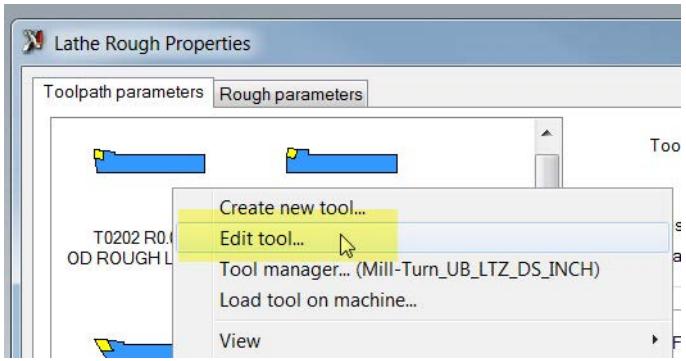
When you select the tool in Mastercam, the picture in the tool selection window should be oriented like the picture.

The picture should look like this even if you are creating an operation for the right spindle. **Rotating the B-axis head to the proper position** on page 26 shows you how to rotate the tool for right-spindle machining. When you first select the tool, it should look like it does at the tool change regardless of which spindle you are working on.

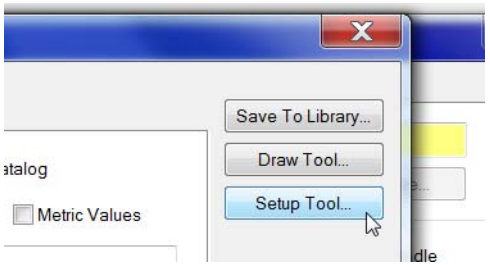


To review or edit these settings, follow these steps:

1. Right-click the picture of the tool in the tool selection window.
2. Choose **Edit tool**.



3. Click **Setup Tool**.



Home

Insert up/down and LH/RH holders

When the tool is mounted at the tool change position, it can still be in any of four different orientations, depending on whether it is mounted insert up or insert down, and whether you are using a left-hand or right-hand holder. Any of these is acceptable with Mill-Turn; the proper choice depends on how you will need the tool oriented when you rotate it into its eventual cutting position.

The tool preview window in Mastercam shows you how the tool is oriented. Notice that the insert color changes to show up (yellow) or down (orange).

left-hand holder; insert up

left-hand holder; insert down

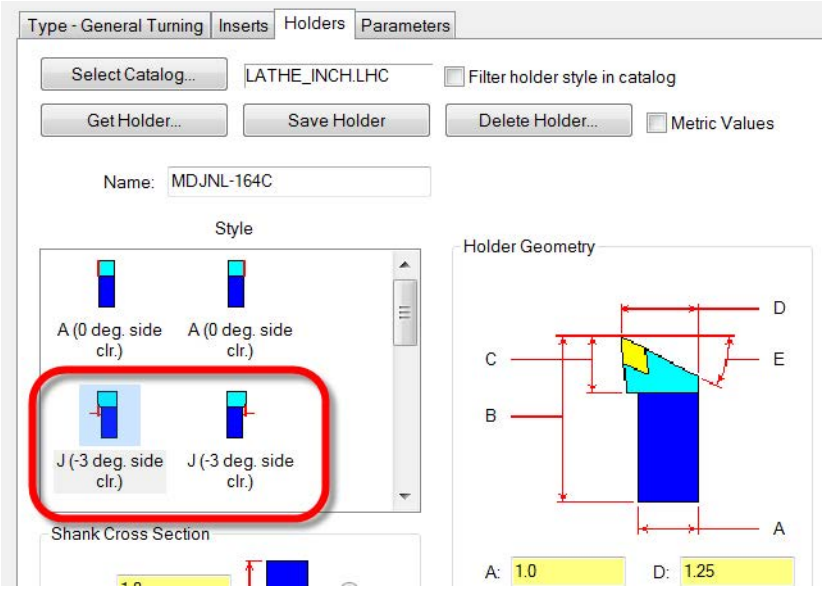
right-hand holder; insert up

right-hand holder; insert down

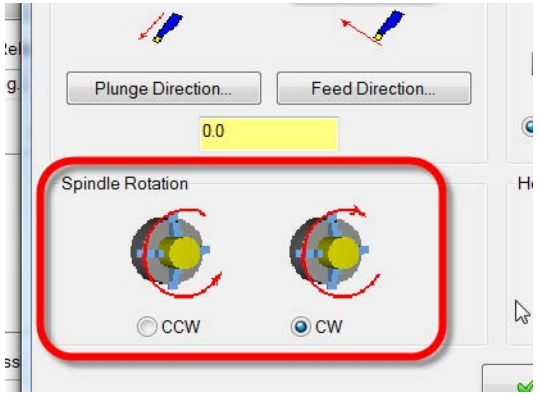
To change the tool from insert-up to insert-down, you need to change the direction of the holder (from left-hand holder to right-hand, or vice versa), then change the spindle direction:

1. Go to the **Tool Definition** dialog box.
2. Select the **Holders** tab.

   The holders are arranged in left/right pairs.
3. Switch to the desired left- or right-hand holder.



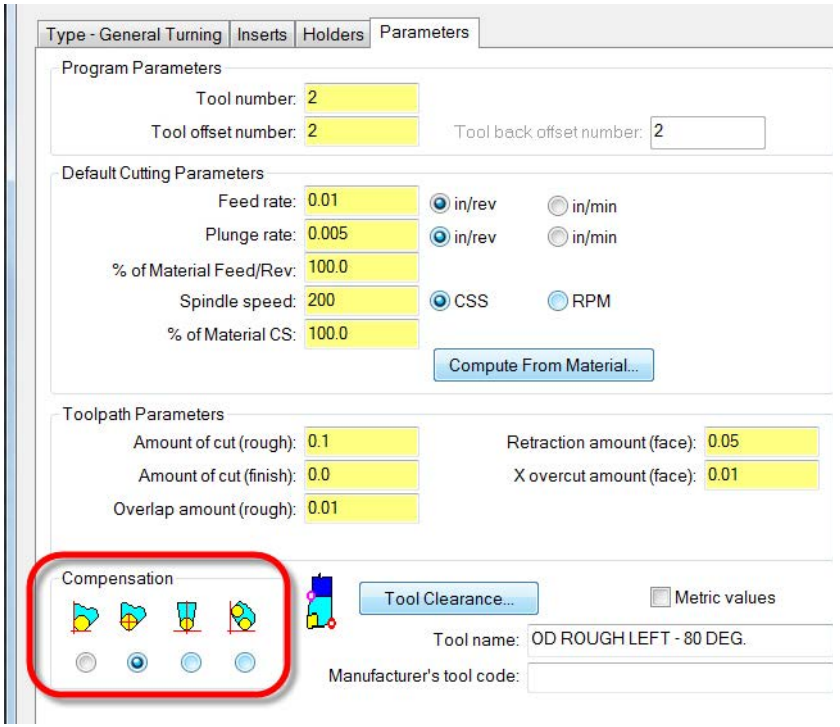
4. Click **Setup**.
5. Change the **Spindle Rotation** direction.



Specifying the tool's compensation point

If you are using the tool at an angle other then vertical or horizontal, and your compensation toolpath does not appear to be calculated correctly, try editing the tool so that the compensation point is set to the center of the tool as shown here.

Note that if you use this compensation method, you will need to touch off your tools differently at the machine.



Selecting the spindle and turret

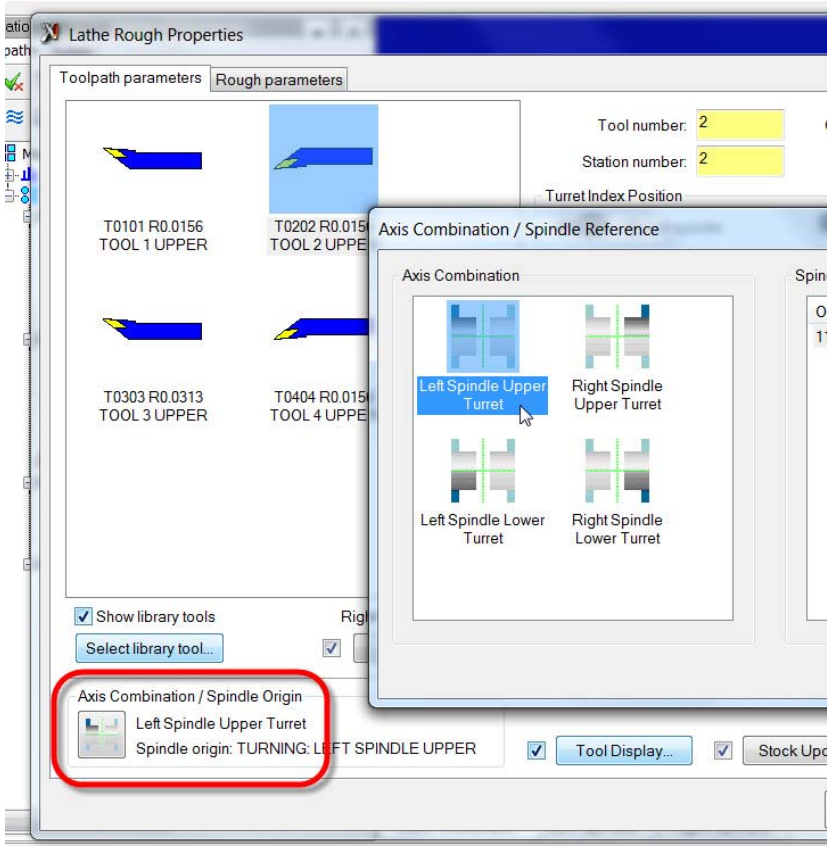
Use axis combinations to set the working spindle and turret.

In Mastercam, you select an axis combination to tell Mastercam which spindle and turret you will be using.

There is a specific axis combination for each possible combination of B-axis head, left/right spindle, and turret. Even if you are using a machine that has only a B-axis head and left spindle, Mastercam still defines at least one axis combination.

Follow these steps:

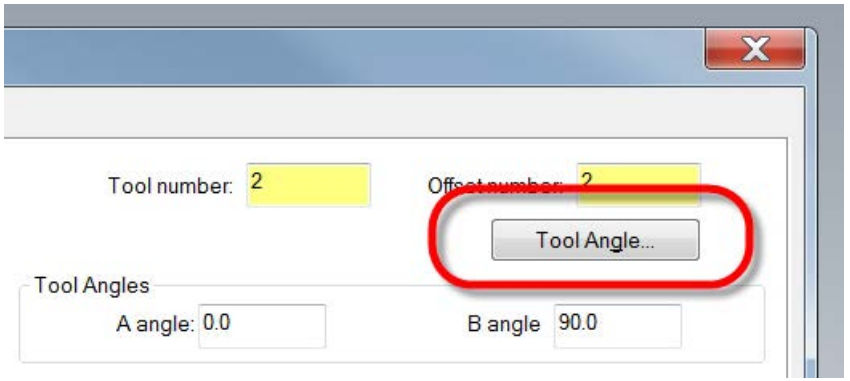
1. Click the **Axis Combination** button.
2. Select the desired turret/spindle combination. This picture shows the axis combination for using the B-axis head on the left spindle.
3. Select the desired tool. Make sure the picture shows it oriented properly for the tool change position for the left spindle.



Home

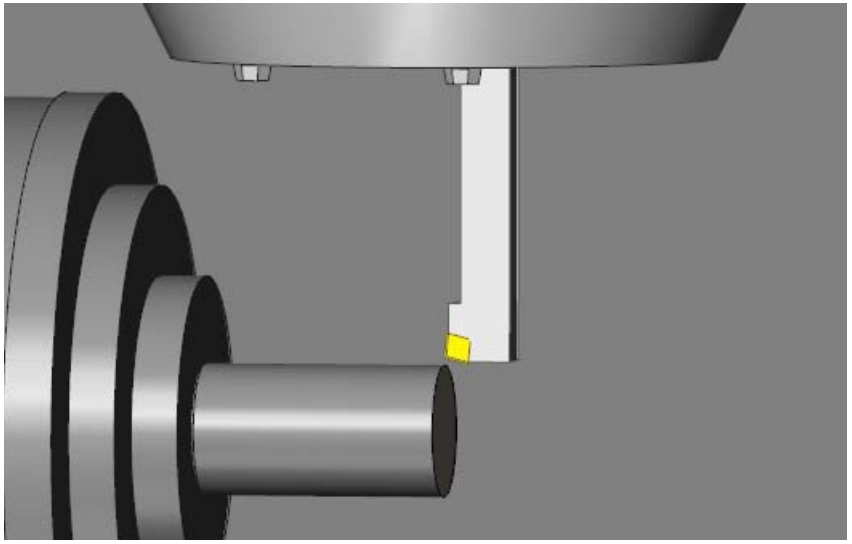
Rotating the B-axis head to the proper position

Click the Tool Angle button to set the B-axis rotation.

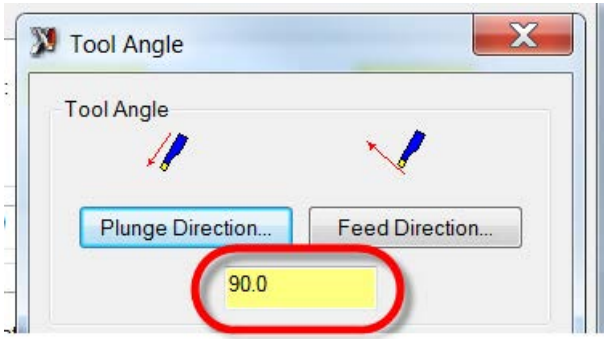


Both B-axis rotation and tool orientation are set with the **Tool Angle** button. Use it to rotate the tool into the proper machining position.

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

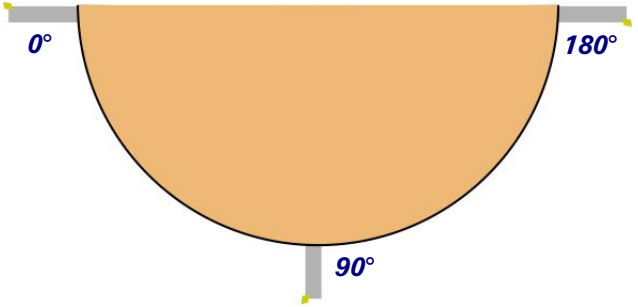


you would enter a **Tool Angle** of **90**

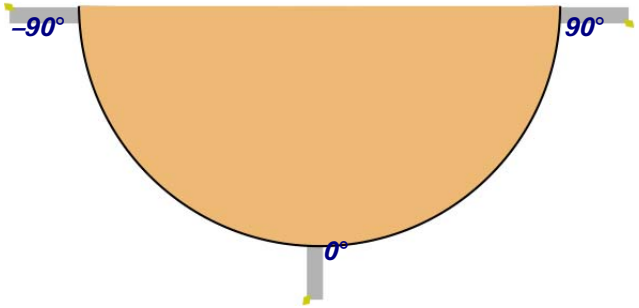


Use this field to rotate the B-axis head to any angular position. The **Tool Angle** value is typically output in your NC file with the `G361` line.

Home

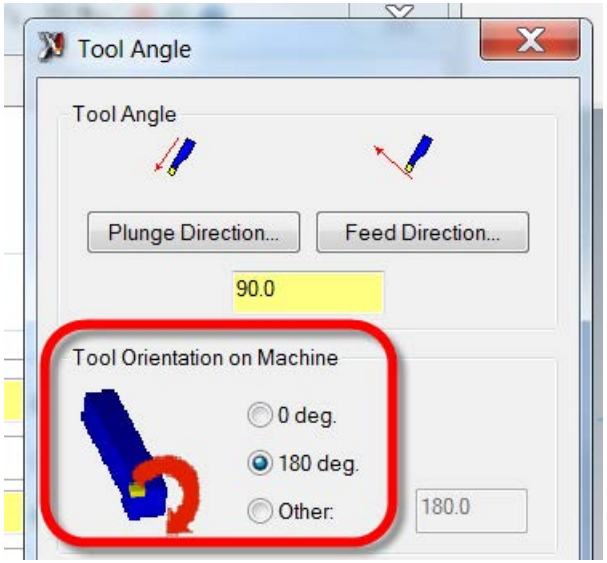


Tool angles in the Mastercam interface

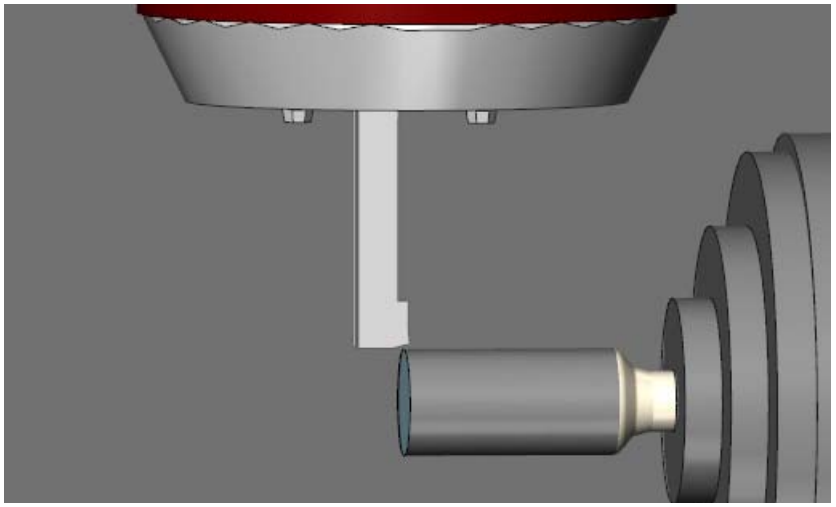


How your DMG Mori uses tool angles

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.



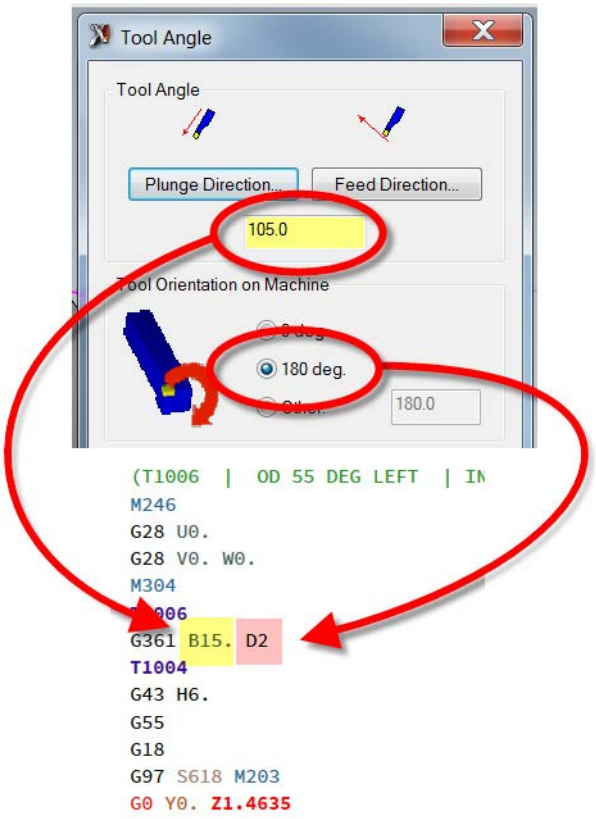
To rotate the tool axis to machine on the right spindle, use the **Tool orientation on Machine** setting.



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

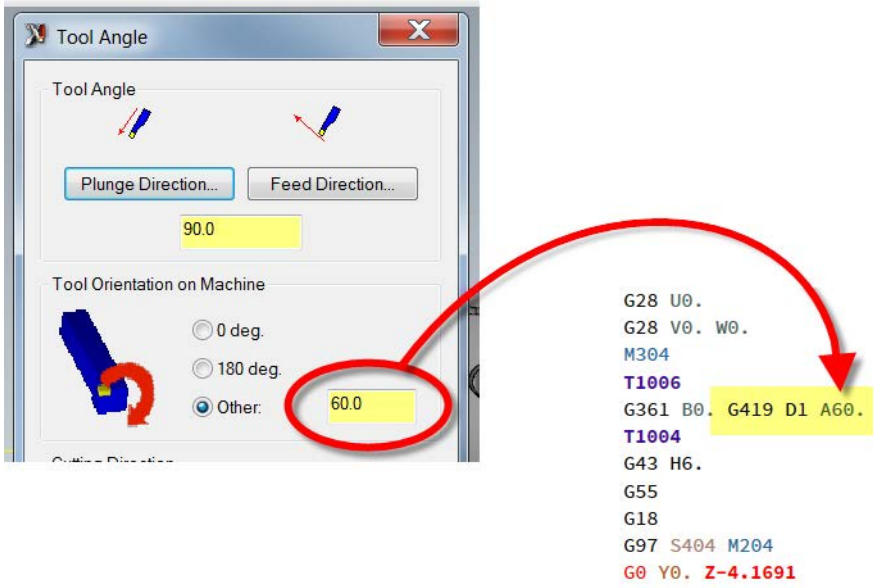
This lets you use the **Tool orientation** setting to orient the tool for cutting on either spindle. Compared to the previous picture, you can see that the insert is now facing towards the right spindle.

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.



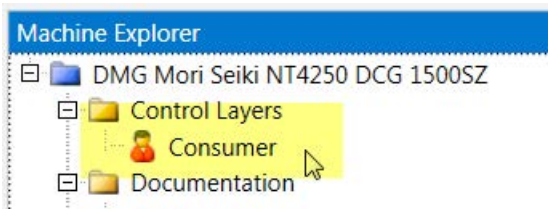
Setting the tool number base for B-axis tools

Set the starting point for tool numbers in the .machine file.

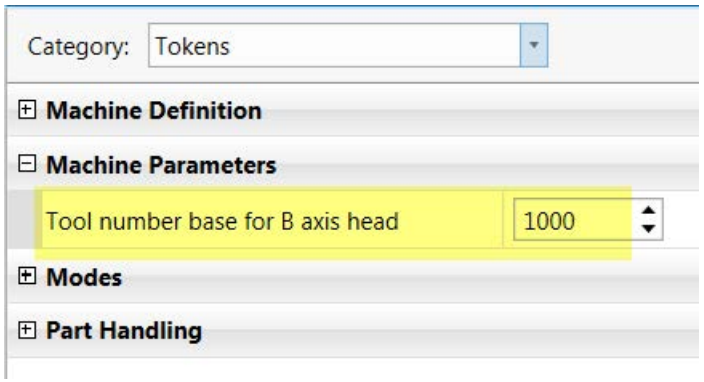
Your **.machine** file includes an option that lets you control the base value for B-axis tool numbers. For example, the first tool might be numbered 1001 or 2001.

Follow these steps to:

1. Open the DMG Mori NT/NTX **.machine** file in the 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

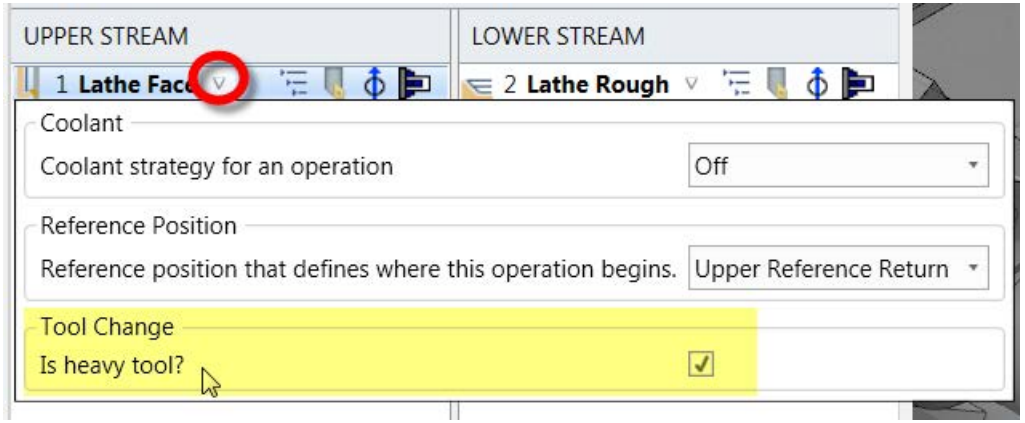
Mill-Turn supports Mori's heavy tool setups.

Your DMG Mori **.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.



Home

G361/G362 tool change mode

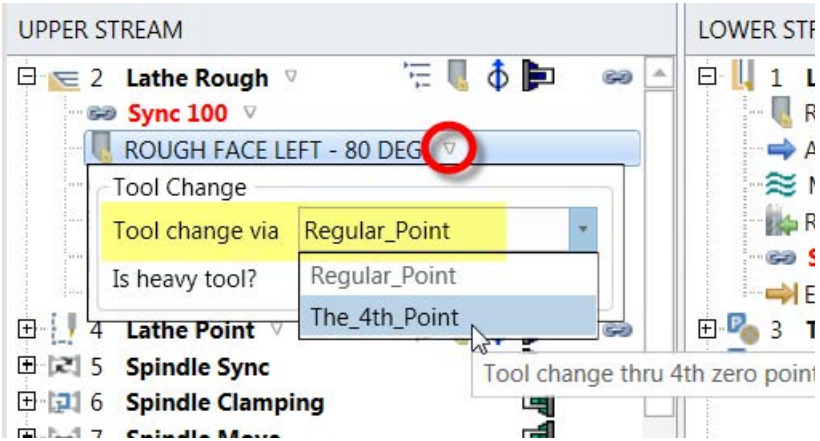
Mill-Turn supports G362 mode for tool change via fourth zero point.

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`.

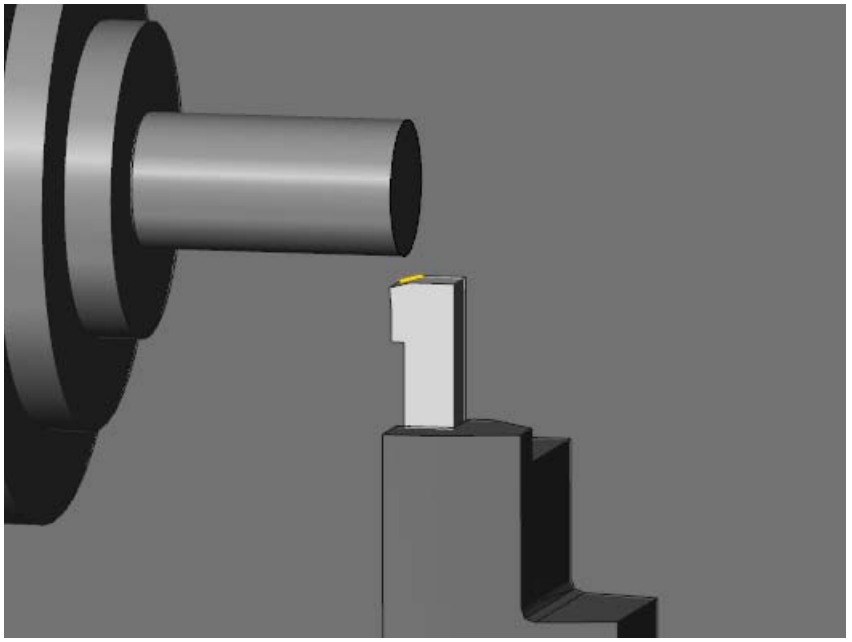
Note that this option is only valid for **Upper Stream** operations which use tools mounted in the B-axis head.



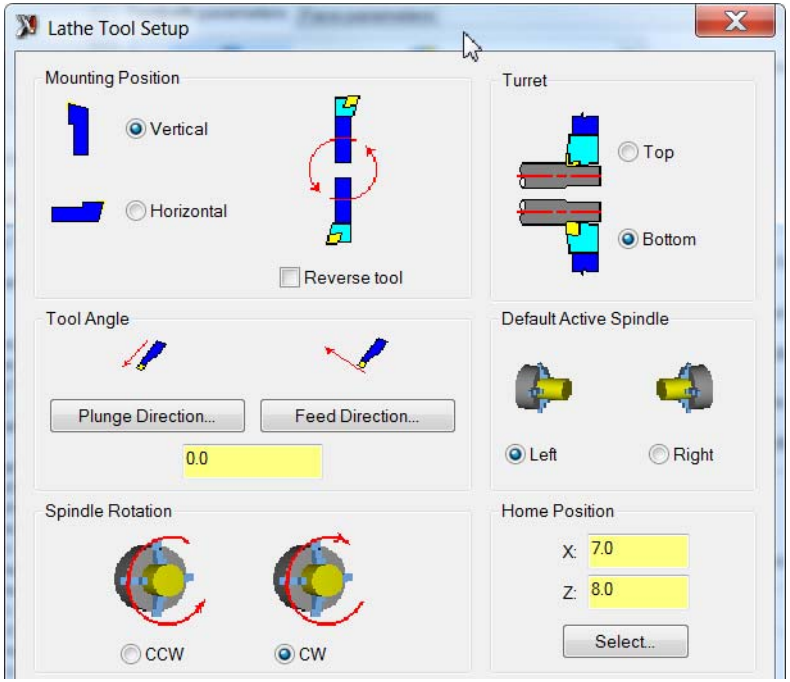
B: Setting up tools on the lower turret

Define tools in the orientation in which they will be used.

Tool definitions for lower-turret tools should be created in the orientation in which the tool will be used. This is different than tools for the B-axis head, which need to be defined in the tool change position.



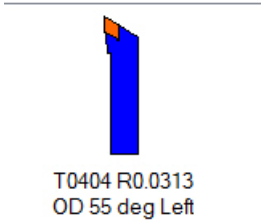
Tools should be defined so that they are facing in the proper direction for the intended spindle. They should also be defined in the proper vertical or horizontal orientation. For example, a vertical, right-hand, insert-down tool should look like this.



The tool setup settings for the tool as pictured above should look like this in Mastercam.

Home

Then when you select the tool in Mastercam, the picture in the tool selection window should look like this.

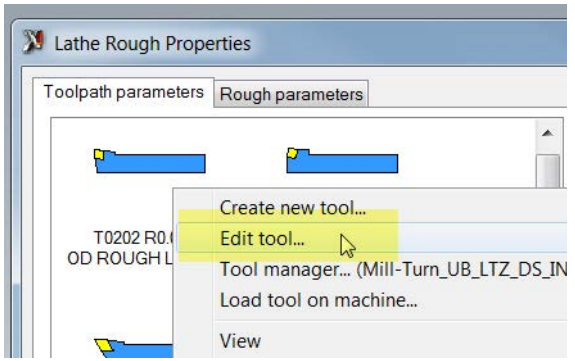


The picture that you see in Mastercam should show the proper orientation for:

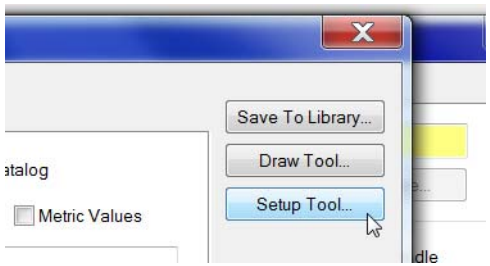
- vertical/horizontal mounting
- left/right spindle
- left-hand/right-hand holder
- insert up/insert down

To review or edit these settings, follow these steps:

1. Right-click the picture of the tool.
2. Choose **Edit tool**.



3. Click **Setup Tool**.



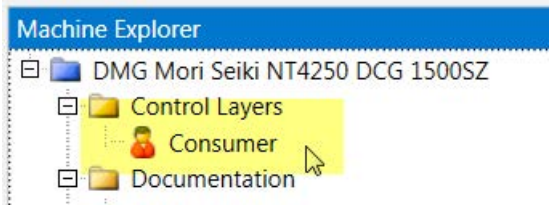
Live tool support

Tell Mastercam if your lower turret does not support live tooling.

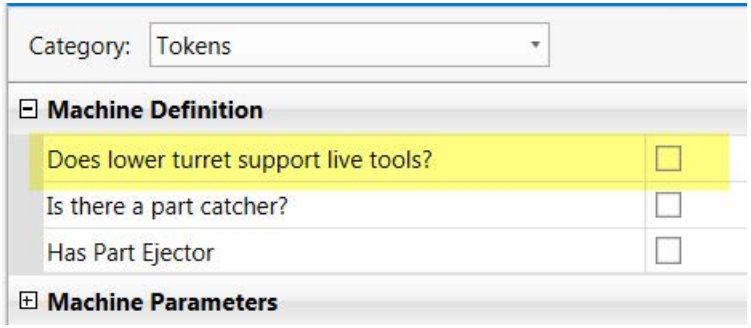
Your DMG Mori NT/NTX .machine file is designed to work with machines whose lower turrets both do and do not support live tooling. If your machine does *not* support live tooling in the lower turret, it is important that you set this option properly in your **.machine** file. This tells the post to suppress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

1. Open the DMG Mori NT/NTX **.machine** file in the 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.



C: *Part handling operations and peripheral components*

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

Home

Using the workpiece push check function (G38)

The DMG Mori NT/NTX .machine file supports the G38 workpiece push check function.

Your DMG Mori NT/NTX **.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
```

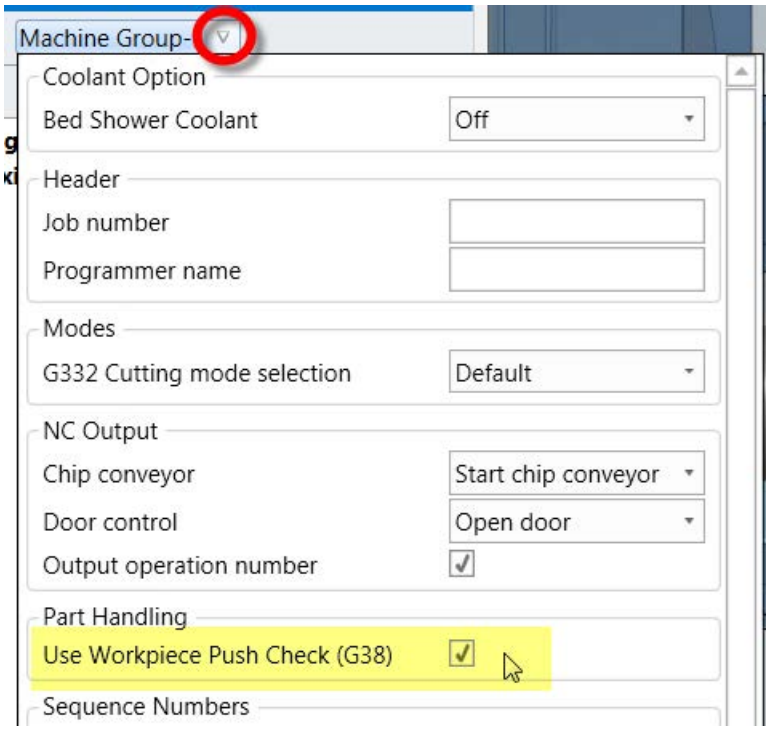
where

- 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.

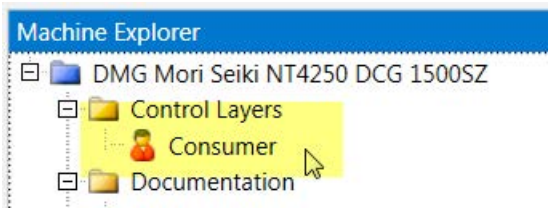


1. In the Sync Manager, click the little triangle next to the machine group name.
2. Select the **Use Workpiece Push Check...** option.
3. Save the IOF file before posting.

Configuring workpiece push check mode

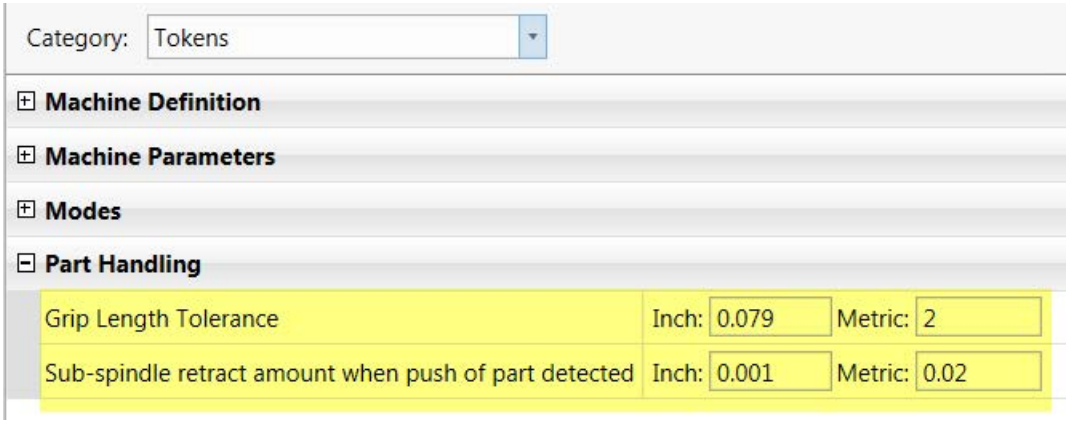
Follow these steps to configure the workpiece push check function and to tell Mastercam if it will be activated by default for part transfer operations.

Follow these steps:

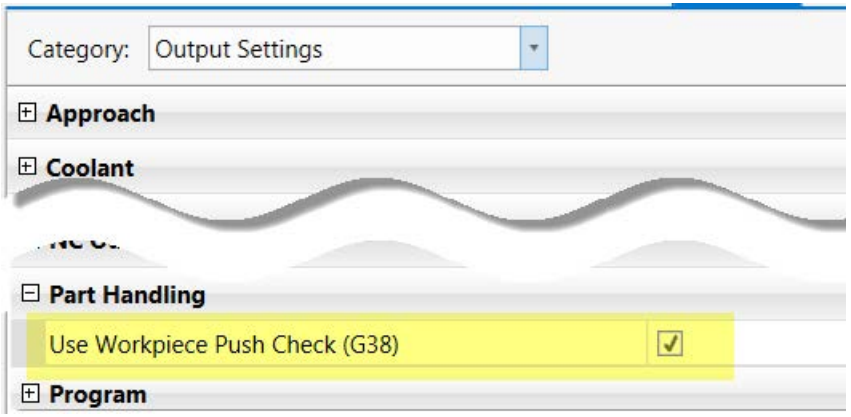


1. Open the DMG Mori NT/NTX **.machine** file in the Code Expert.
2. Double-click the **Consumer** layer.

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.



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.

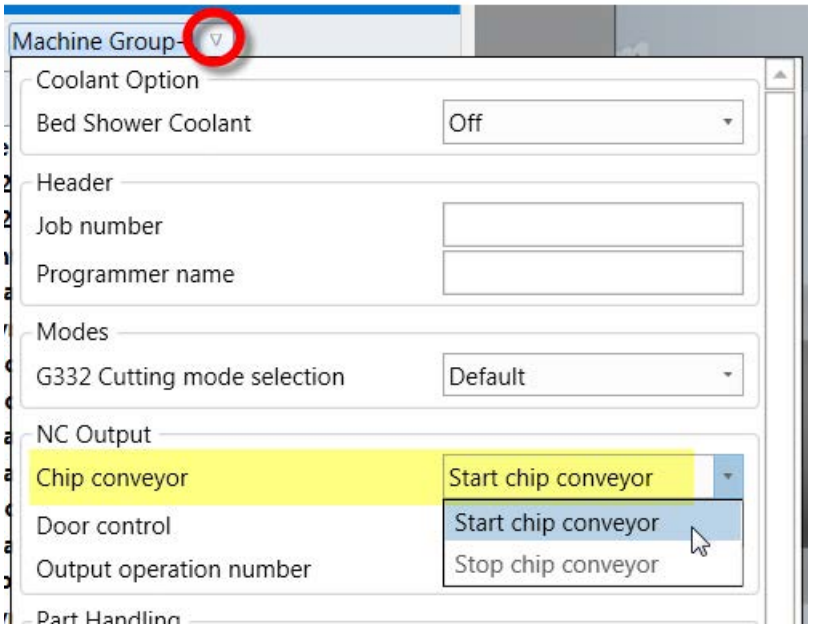


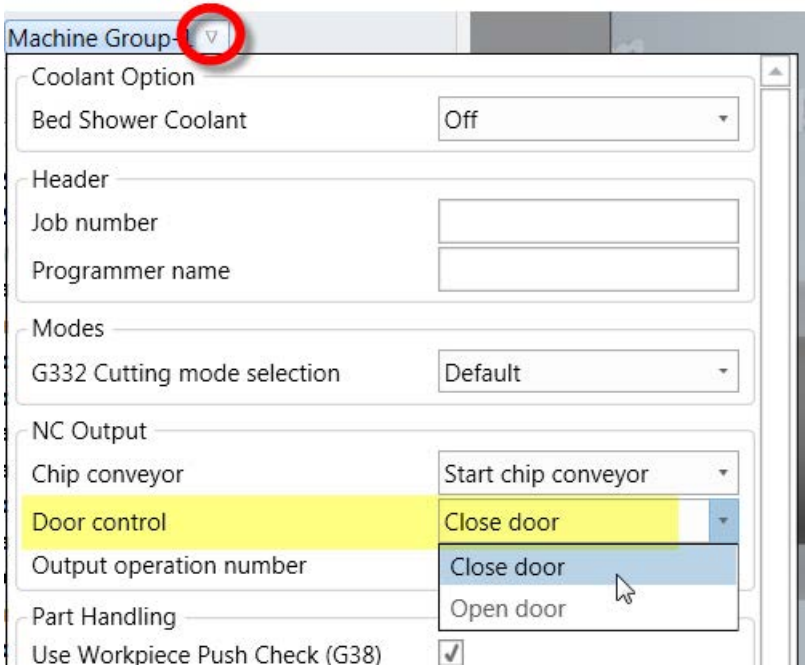
Chip conveyor (M200)/door control (M220)

Use .machine file options so that your post is coordinated with settings on your control.

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.

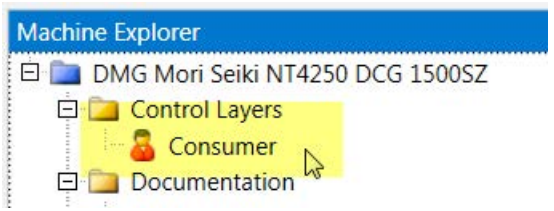




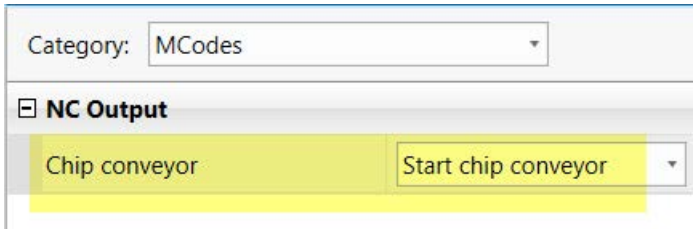
- To include the door open and close commands in your NC program, select the **Close door** option from the **Machine Group** options in the Sync Manager. This will result in `M221` (close the door) at the start of your program, and `M220` at the end (open the door).
- Select the **Open** option to leave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



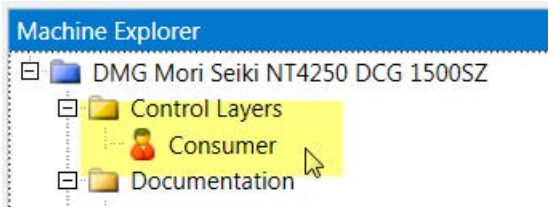
Part ejector (M47) and part catcher (M73) support

Enable support for these components in your .machine file.

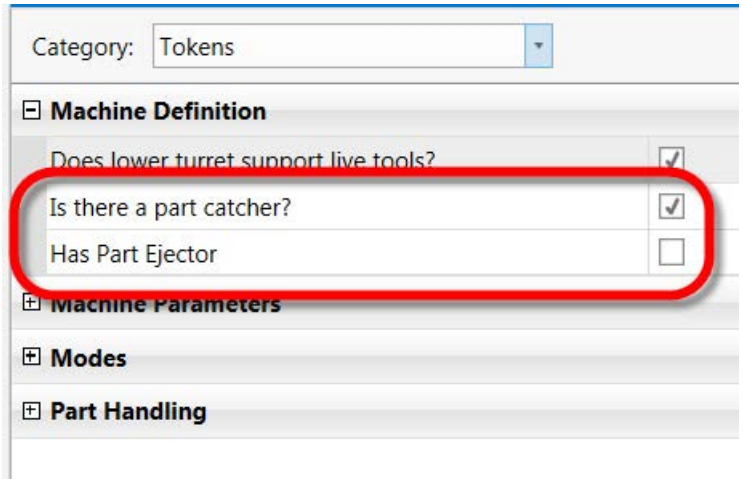
Your **.machine** file includes two options that let you enable support for part catcher (`M73/M74`) and part ejector (`M47`) peripheral components.

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

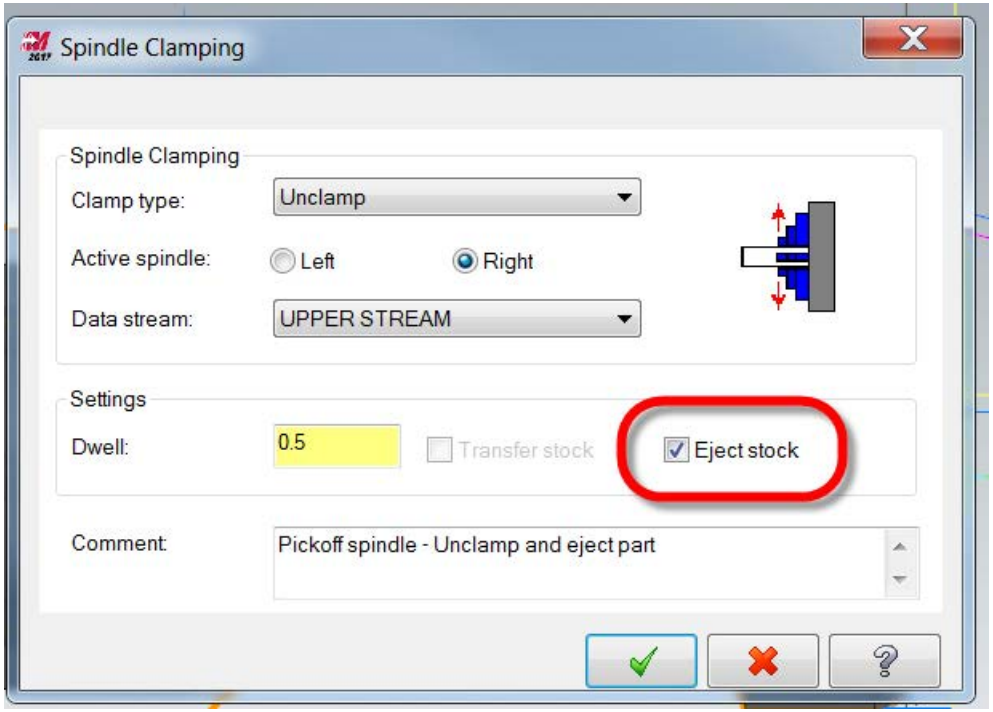
1. Open the DMG Mori NT/NTX **.machine** file in the Code Expert.
2. Double-click the **Consumer** layer.



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.



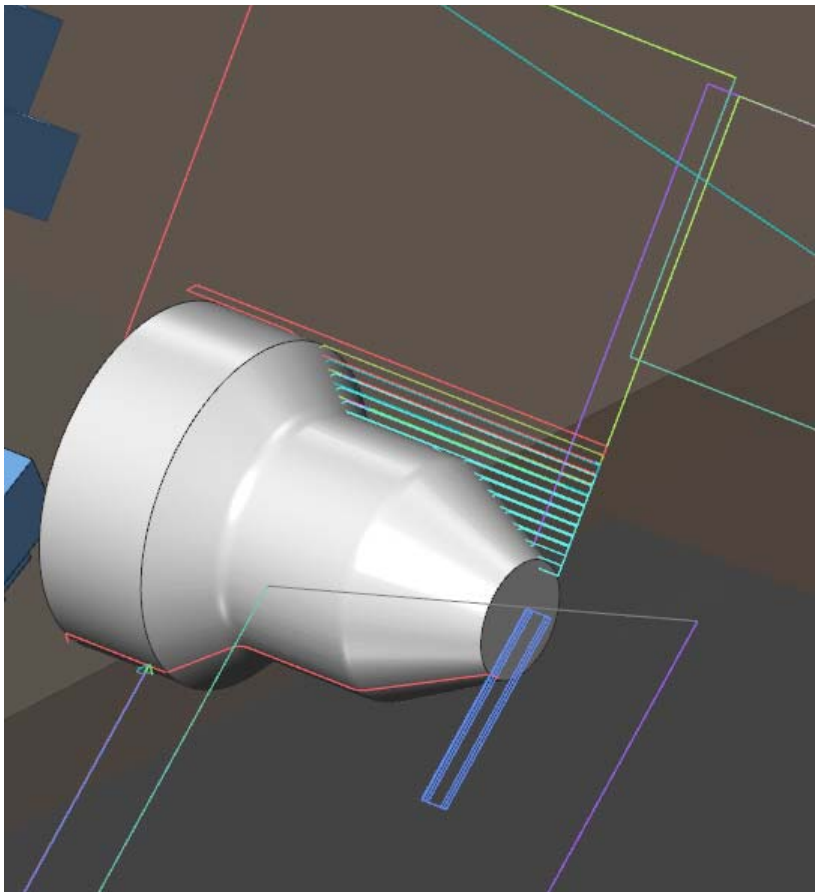
Home

Home

Chapter 3: Working with toolpaths

Your DMG Mori NT/NTX **.machine** file extends Mastercam's toolpath programming capabilities with a variety of options that are specific to the NT/NTX machines.

- **DMG Mori NT/NTX machining modes**
  - Polar (G12.1) and cylindrical (G7.1) interpolation
  - Tilted-plane machining cycle (G68.1)
  - Selecting the cutting mode (G332)
  - Radius/diameter mode (G10.9)
- **Balanced turning operations**
  - Setting up tools for pinch turn operations
  - Numbering syncs for pinch turn operations
- **Multiaxis toolpath settings**
  - Rotary start position
  - Pole handling
  - Using tool center programming (G43.4)
  - Using AI contour control (G05.1)
- **Reference points and home positions**
  - Setting the start point and end point for an operation
  - Creating custom reference positions
- **Coolant and other toolpath options**
  - Using coolant
  - Configuring stops and optional stops
  - Outputting the operation number for each toolpath

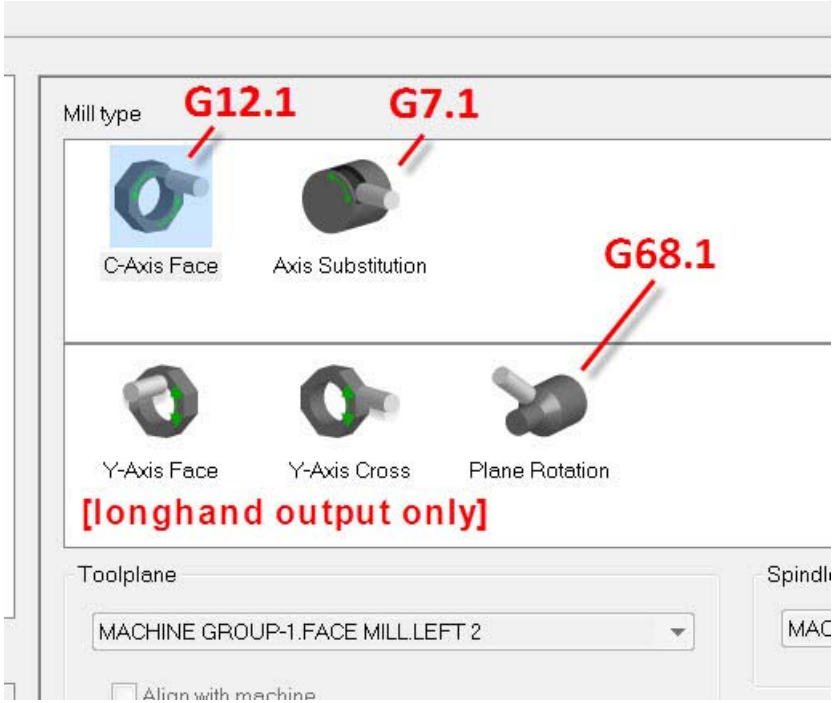


Home

Rev. 3.1
August 2016

A: DMG Mori NT/NTX machining modes

Your DMG Mori NT/NTX .machine file supports several different machining modes for mill operations



Your DMG Mori NT/NTX **.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 operating modes:

- **Selecting the cutting mode (G332)**
- **Radius/diameter mode (G10.9)**

Home

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

Configure the type of output you want for polar and cylindrical interpolation cycles.

Your DMG Mori NT/NTX **.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.

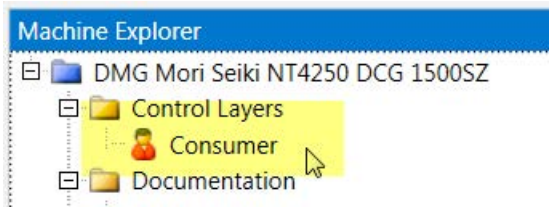
Default settings for G7.1 and G12.1

Two different default settings are available in the **.machine** file.

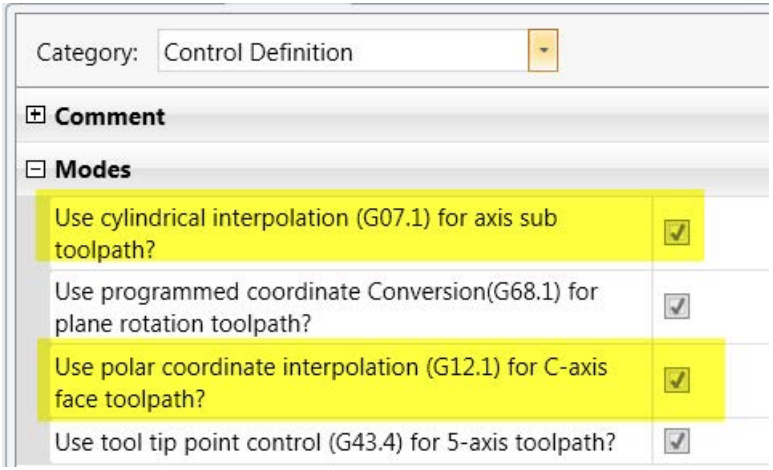
- Choose whether the default output mode will be the G7.1/G12.1 cycle or longhand output.

Follow these steps:

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



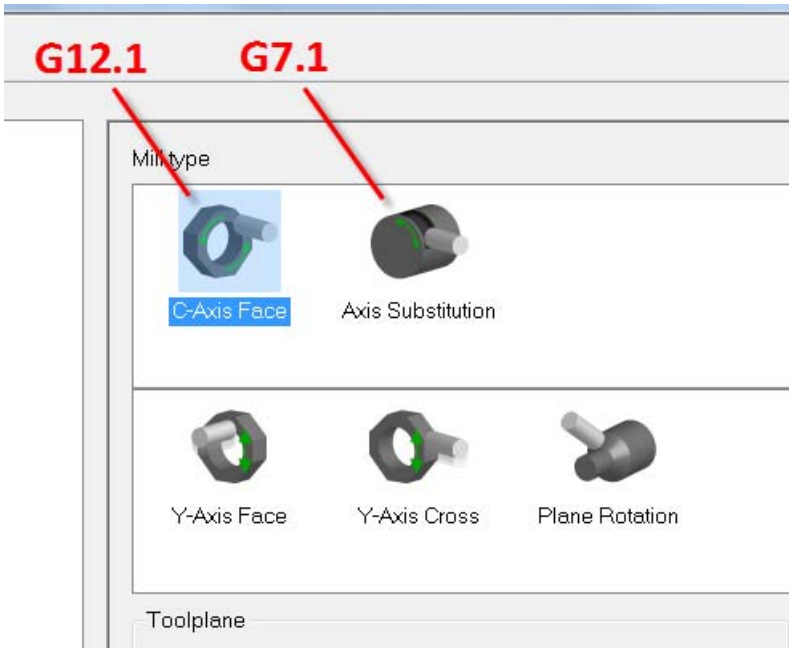
3. Select **Category: Control Definition**.
4. Go to the **Modes** section.
5. For each cycle, choose the default output mode. If you do not select the checkbox, you will get longhand output.
6. Save the **.machine** file when you are done.



Programming G7.1/G12.1 for each operation

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

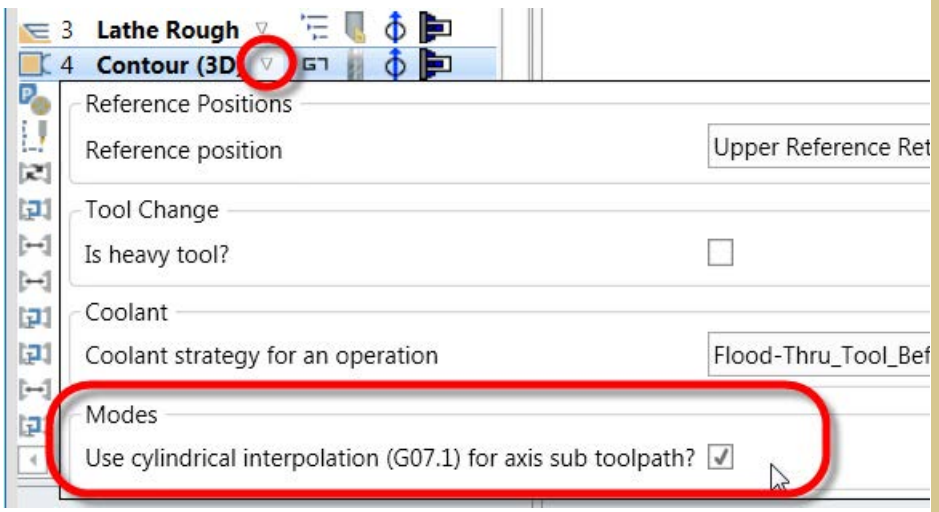
1. The cycles are keyed to specific setup types and are automatically enabled when you select **C-axis Face** or **Axis Substitution** on the **Setup** page for milling operations.



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. 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. For each operation, the default setting comes from the **.machine** file. If you override it, make sure you press **[Ctrl+S]** to save the new setting.



Home

Tilted-plane machining cycle (G68.1)

Configure the type of output you want for G68.1 tilted-plane machining cycles.

Your DMG Mori NT/NTX **.machine** file supports G68.1 coordinate conversion (tilted-plane) machining cycles. This topic explains how to:

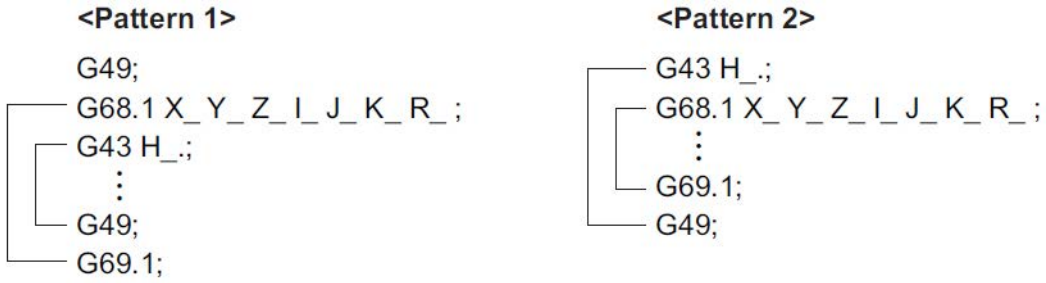
- configure the default output pattern.
- configure each individual operation to output the G68.1 cycle.

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.

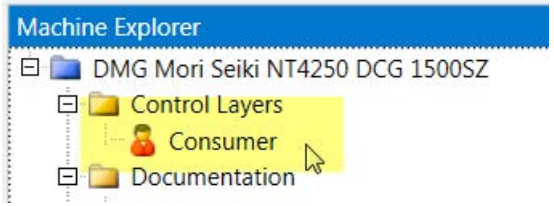
Output patterns 1 or 2 control how the G68.1 on/off commands will be output relative to the length offset G43/G49 commands. With Pattern 1, the offset commands are output inside the G68.1/G69.1, while with Pattern 2 they are outside the cycle:



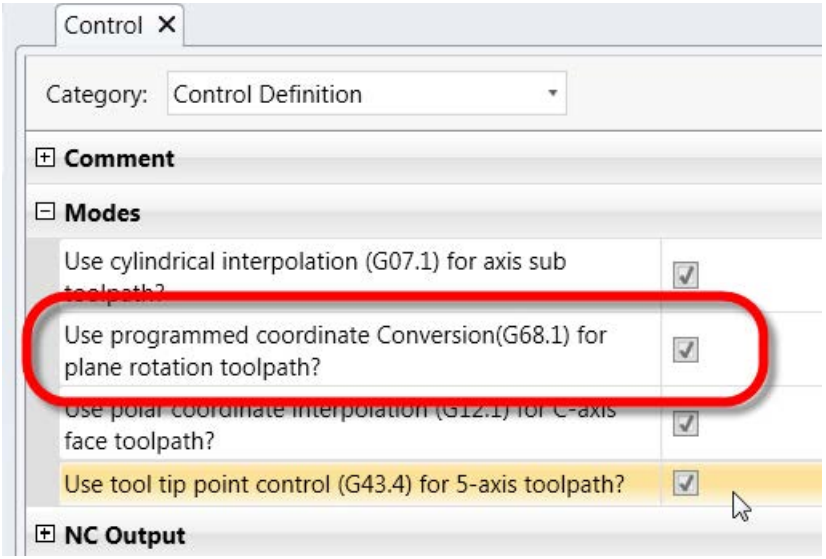
Follow these steps:

1. Open the DMG Mori NT/NTX **.machine** file in the Code Expert.

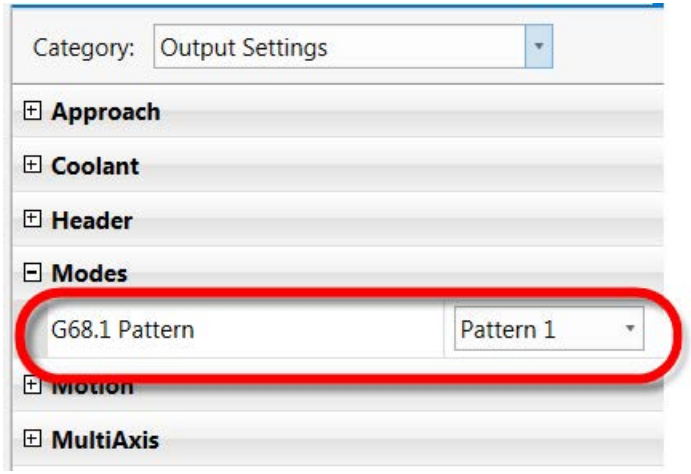
2. Double-click the **Consumer** layer.



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.



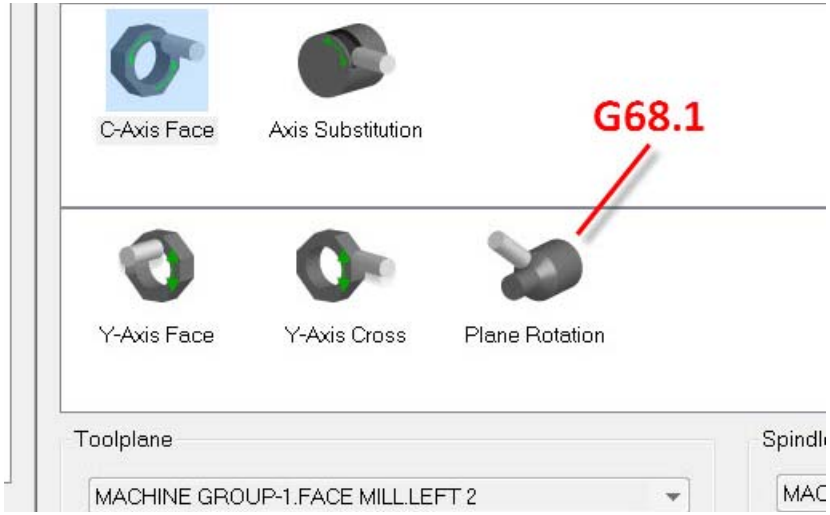
6. Select **Category: Output Settings**.
7. Go to the **Modes** section.
8. Select the desired **G68.1 Pattern**.
9. Save the **.machine** file.



Programming G68.1 for each operation

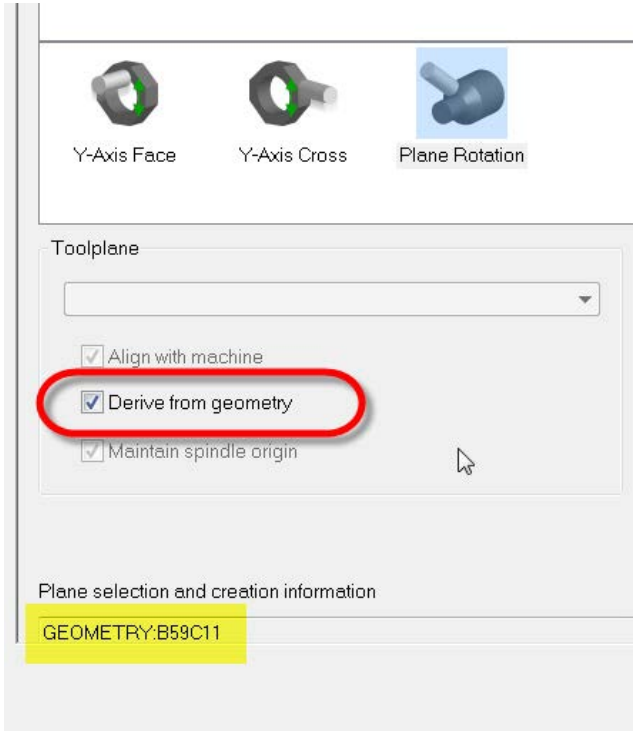
Follow this general workflow for programming G68.1 cycles for each operation.

1. The G68.1 mode is automatically enabled when you select **Plane Rotation** on the **Setup** page for milling operations.

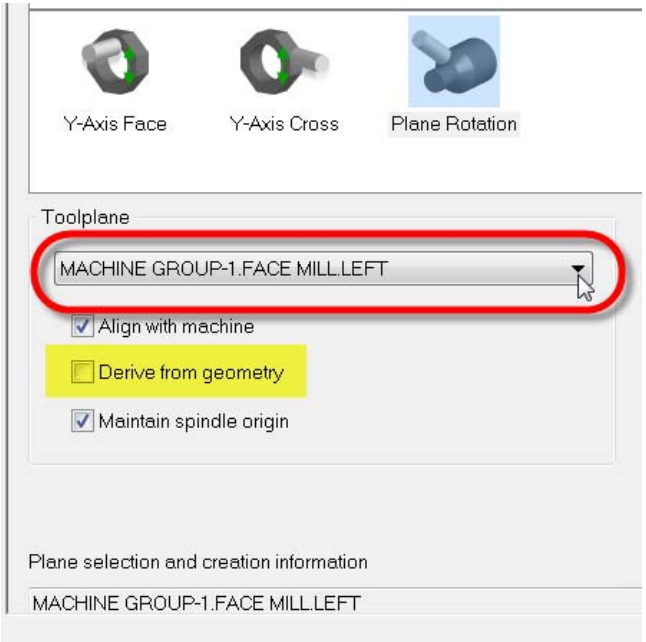


2. Next you need to tell Mastercam how to establish the toolplane.
   - For most applications, you will want Mastercam to automatically create a toolplane based on the selected geometry.
   - Select the **Derive from geometry** option to do this.

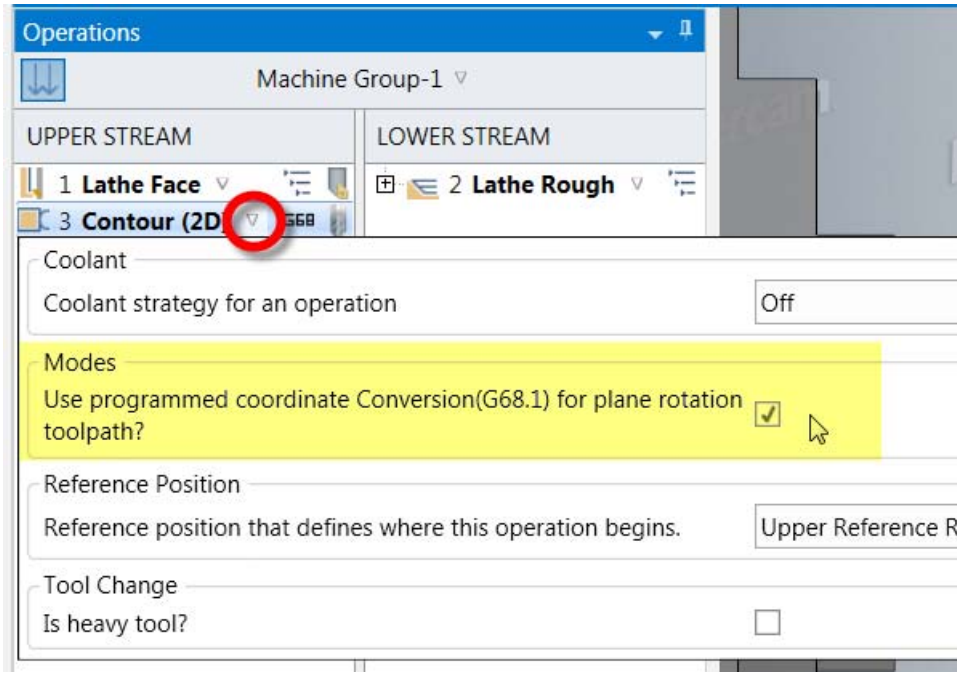
   The name for the new plane is displayed at the bottom of the dialog box; Mastercam tries to create a name that describes its rotation.



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.

Home

Selecting the cutting mode (G332)

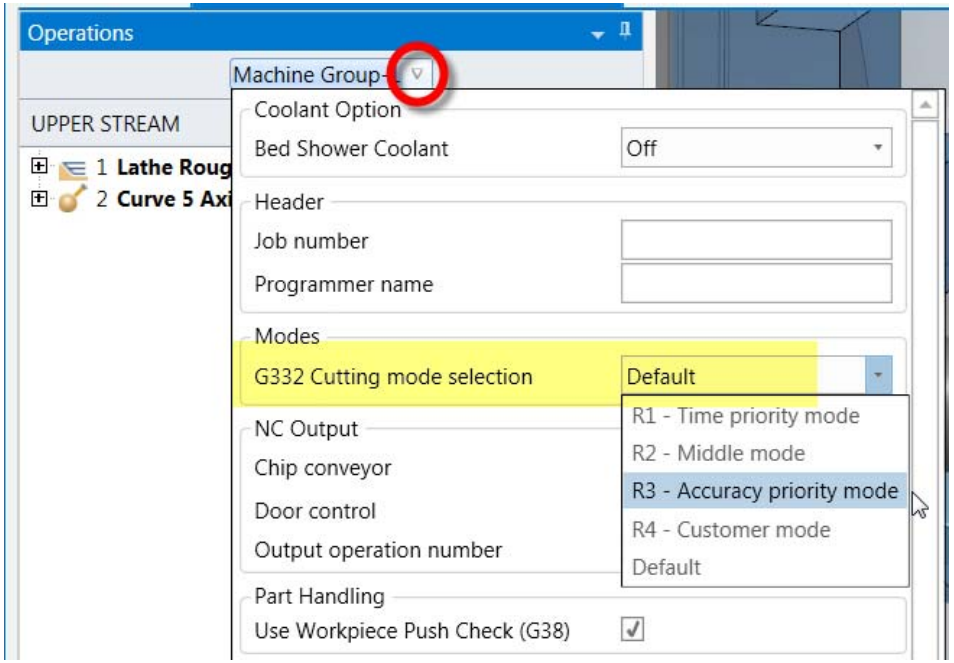
Your .machine file supports G332 R1 through R4 precision cutting modes.

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)

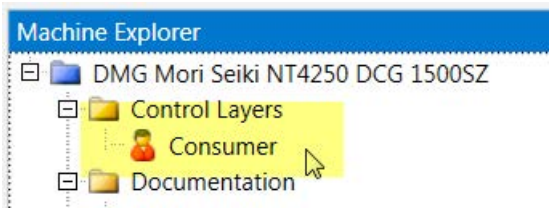
Select diameter or radius output separately for each turret and for mill vs. lathe operations.

Your DMG Mori NT/NTX **.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.

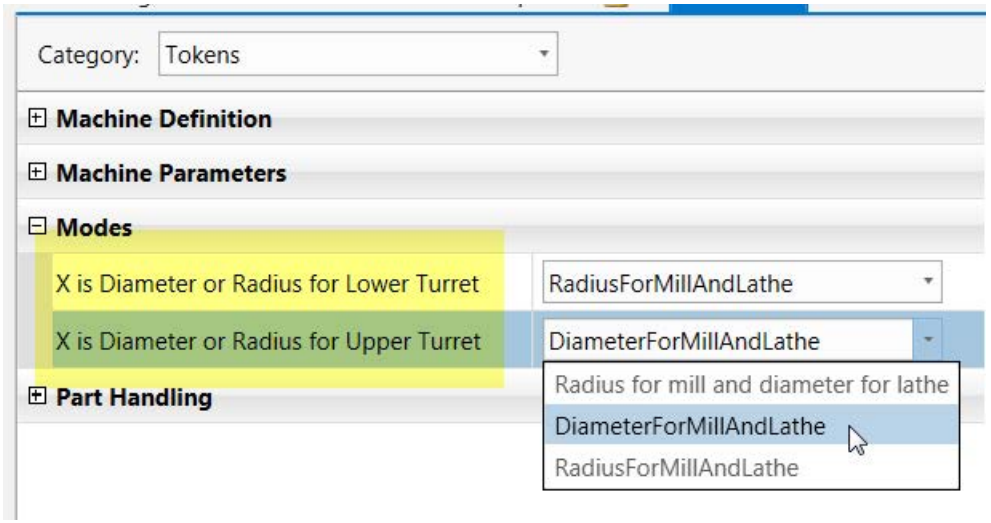
Follow these steps:

1. Open the DMG Mori NT/NTX **.machine** file in the 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.

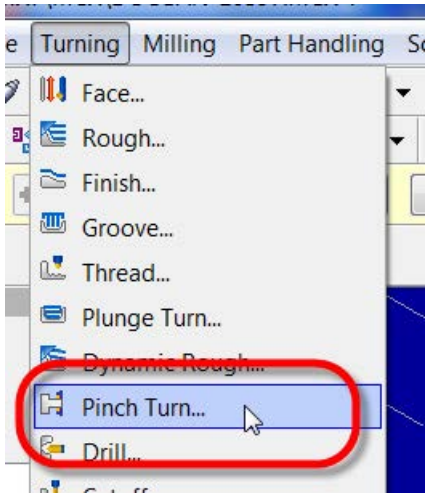


B: Balanced turning operations

Information about programming balanced turning (pinch turn) operations.

Balanced cut mode on your DMG Mori NT/NTX machine (G68 mode) typically corresponds to Mastercam's pinch turn operation.

However, you will not see a G68 in your NC code; instead, the DMG Mori NT/NTX post simulates balanced cut mode with regular motion commands and wait codes distributed between the two streams.



Home

Setting up tools for pinch turn operations

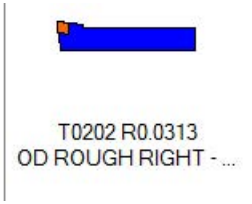
Set up your tools in each stream so that they are complementary.

When creating a pinch turn operation, it is important that the settings for the two tools be properly coordinated. Mastercam will not be able to create the operation if they conflict. Follow these guidelines.

Insert direction—The insert directions for the two tools need to be complementary. For example, if the upper-turret tool is insert-down, then the lower-turret tool needs to be insert-up.

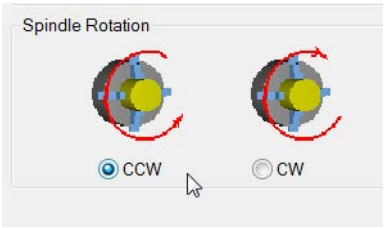
You can tell the insert direction from the picture in the tool window. An orange insert means insert-down; a yellow insert means insert-up.

If the upper-turret tool is insert-down...

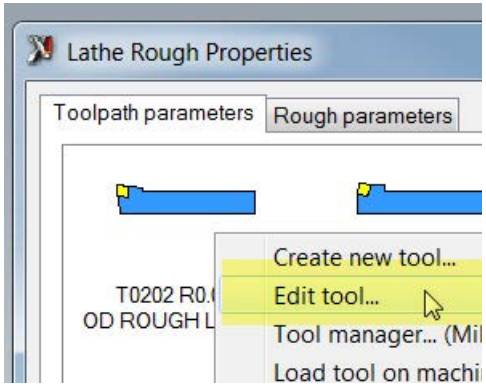


...the lower-turret tool must be insert-up.

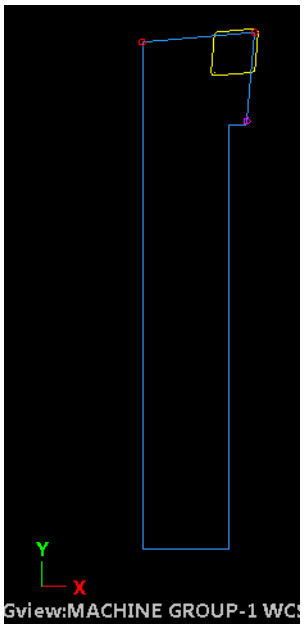
Spindle direction—The spindle direction for the two tools must be the same.



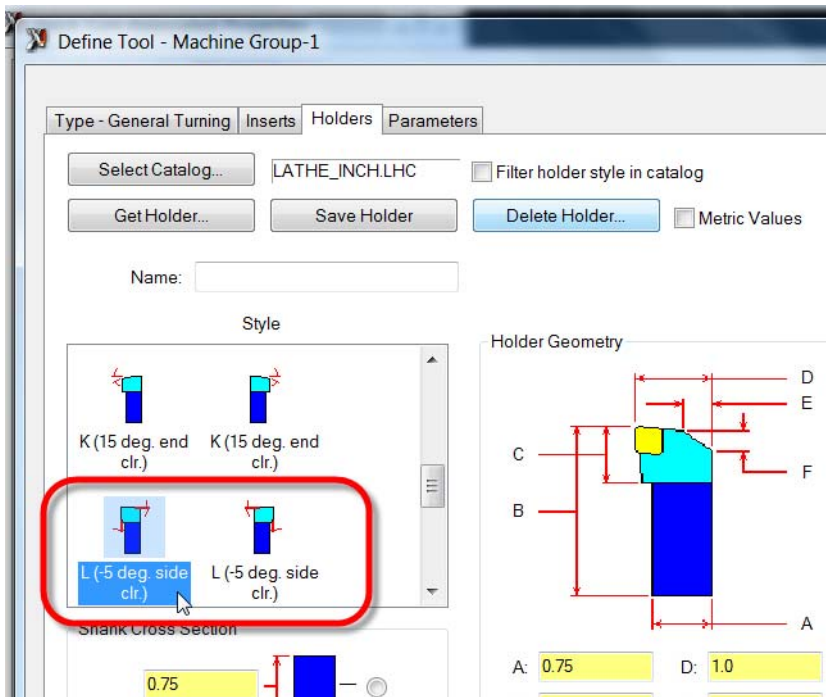
Tool holder—If the above two conditions are met and the toolpath is still not being created properly, you might need to select a different tool holder. Right-click the picture of the tool, and select **Edit tool**. Then click the **Draw tool** button.



Mastercam displays the tool and holder profile as currently defined. For example, the tool shown here is insert-up with a left-hand holder, but is clearly not defined correctly for work on the left spindle. If you wanted to use this tool in a pinch-turn operation, and it needs to be insert-up to complement the upper-turret roughing tool, you need to select a right-hand holder for it.

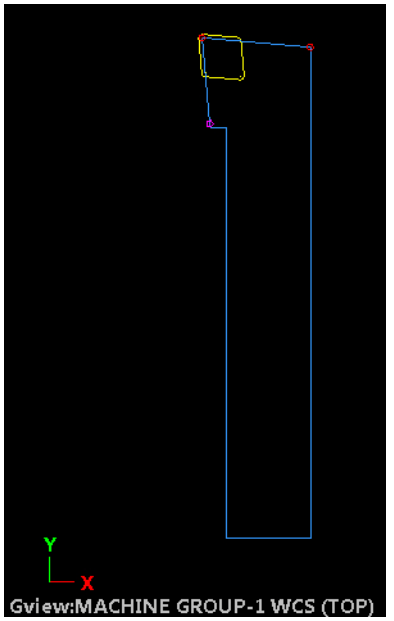


To do this, go to the **Holders** tab in the **Define Tool** dialog box. Most of the holders are listed in left-right pairs as shown in the picture. Select the one that you need.



Click the **Draw tool** button again. You should see the tool oriented for the other spindle as shown here.

Once the **Draw tool** picture is correct, Mastercam should be able to create the pinch turn operation properly.



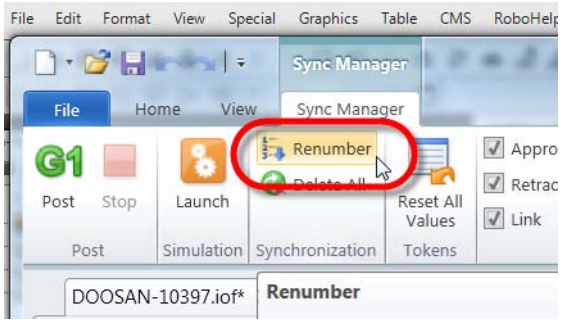
Home

Numbering syncs for pinch turn operations

Use the **Renumber** command to coordinate automatic and manual syncs.

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 after the ones created for the pinch turn operation. This means that after creating your own syncs, choose the **Renumber** command from the ribbon bar.



This will renumber all of your syncs consecutively—your own plus the ones created by Mastercam.

```
31  X1.8221
32  G50 S5000
33  G96 S200
34  M100
35  G99 G1 Z.1 F.01
36  Z-1.7962
37  G3 X2.02 Z-2.0149 R.2912
38  G1 Z-2.5476
39  X2.1614 Z-2.4769
40  G0 Z.2
41  X1.4262
42  M101
43  G1 Z.1
44  Z-1.63
45  X1.6442 Z-1.7214
46  X1.7856 Z-1.6507
47  G0 Z.2
48  X1.0304
49  M102
50  G1 Z.1
51  Z-.365
```

Home

C: Multiaxis toolpath settings

Options for configuring multiaxis (5-axis) toolpath motions.

Your DMG Mori **.machine** file includes a number of options that you can use to configure multiaxis toolpaths. For the most part, the default settings are stored in the **.machine** file, and you can override the default in the Sync Manager for each operation.

The following sections describe how to configure the default settings for each mode as well as for individual operations.

- **Rotary start position**
- **Pole handling**
- **Using tool center programming (G43.4)**
- **Cancelling TCP between passes**

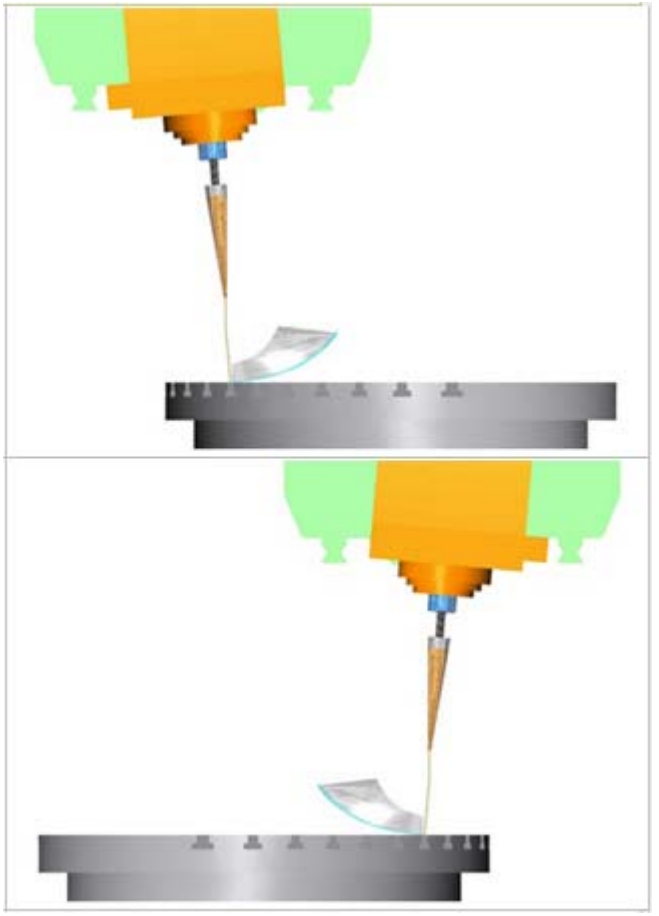
Home

Rotary start position

Help MP.NET 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.)

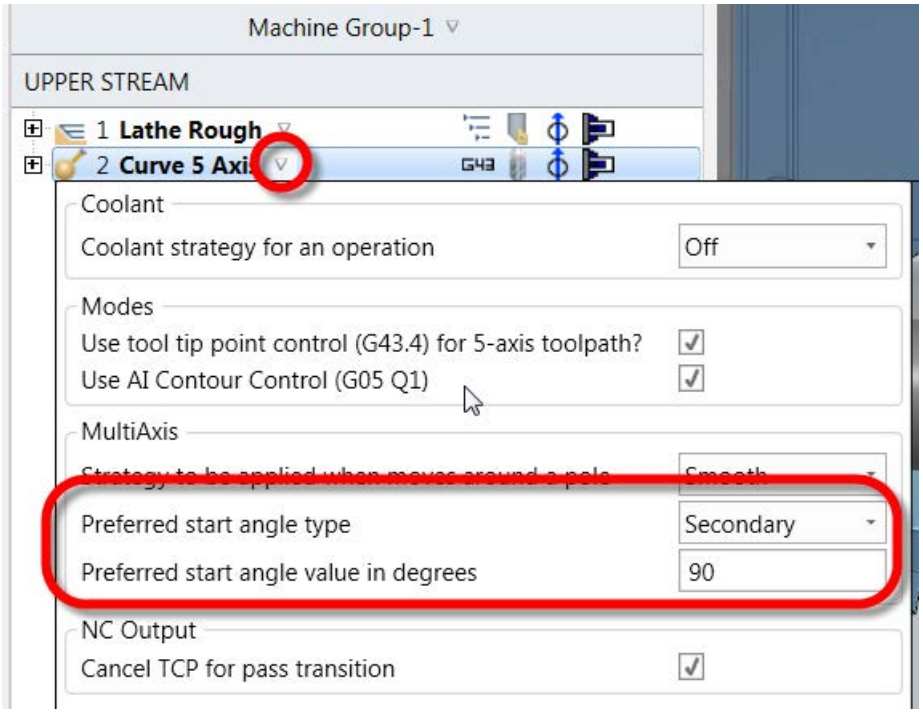
However, from an applications perspective, one location is usually preferable than the other: for example, one might be a shorter distance from the end of the previous toolpath, or might involve rotation angles that are not mechanically possible.



Home

Your DMG Mori **.machine** file includes two options that are used by the post to select the proper starting position. They let you specify a desired starting angle (rotary position), and 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.

These options are designed to be set for each operation. They are exposed to the user in the Sync Manager.



Preferred start angle type—Select the axis in which the desired start angle will be measured. Select either the **Primary** or **Secondary** rotary axis. For the DMG Mori NT/NTX, 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 angle measures in this field; however, be aware that MP.NET 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.

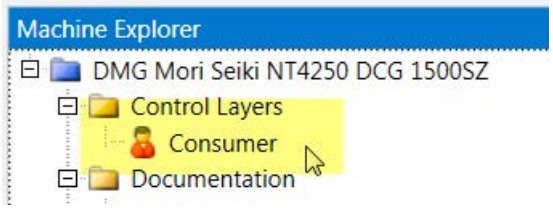
> **Note—**Mastercam will automatically discard solutions that are outside the limits.

Home

Setting the default start angle

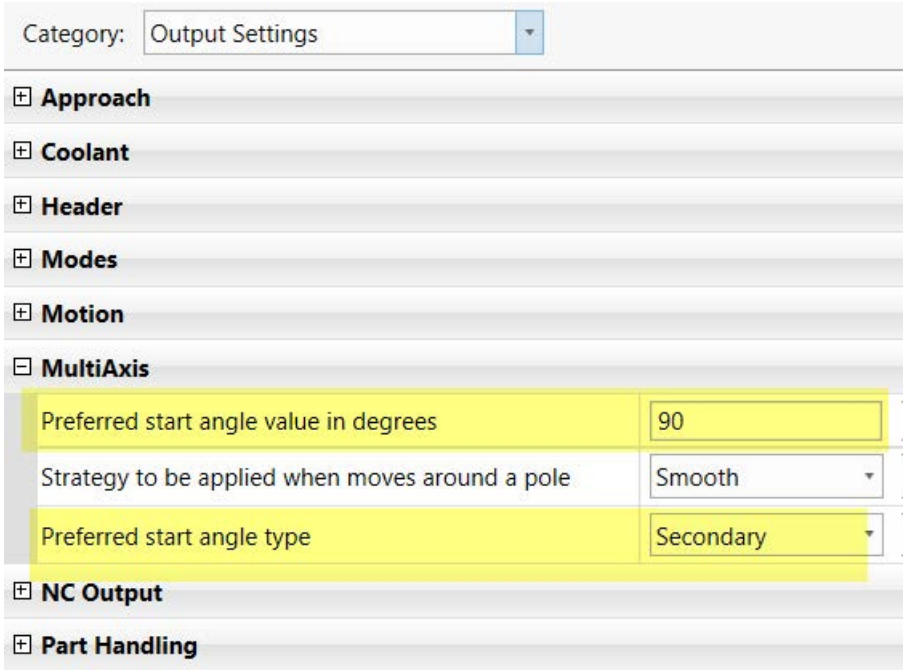
You can set the default start angle position in the **.machine** file.

1. Open the DMG Mori NT/NTX **.machine** file in the 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 two start angle options..
6. Save the **.machine** file.



Home

Home

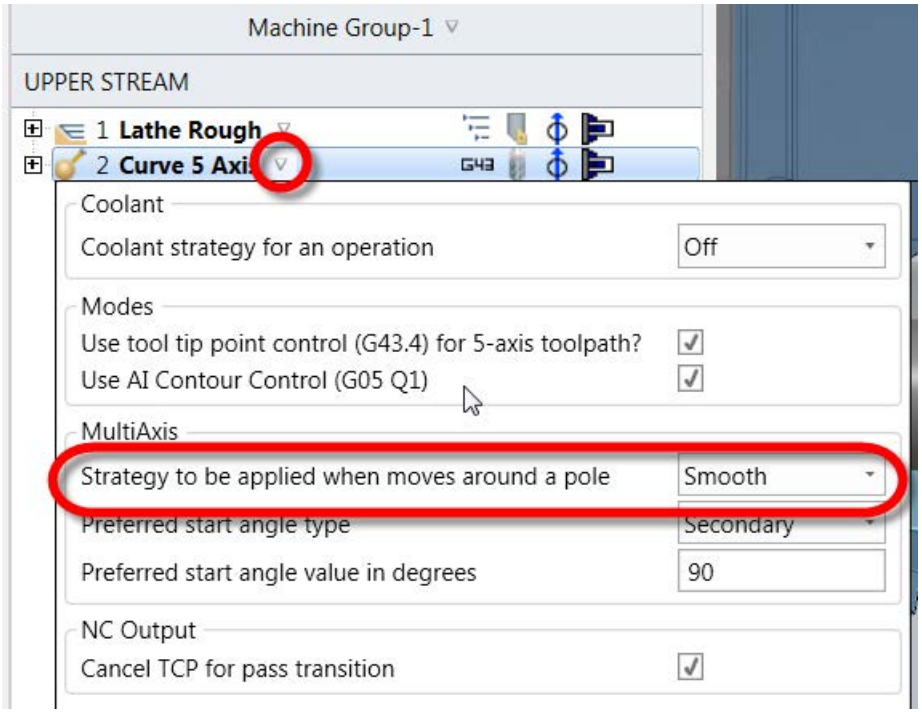
Pole handling

Configuring rotary output when the tool is almost parallel to the Z-axis.

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`, etc. 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.



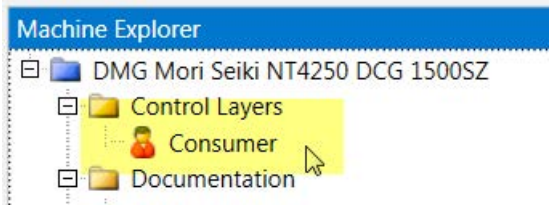
Freeze—Freeze the angle around the pole. While the pole condition is active, the C-axis position will be frozen at its last position before the pole condition occurred.

RotAngle—Use a rotation angle around the pole. If the undetermined angle is in the spindle, MP.NET 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 just 3-axis moves, separated by sections with rotary moves, MP.NET will interpolate tool angle positions

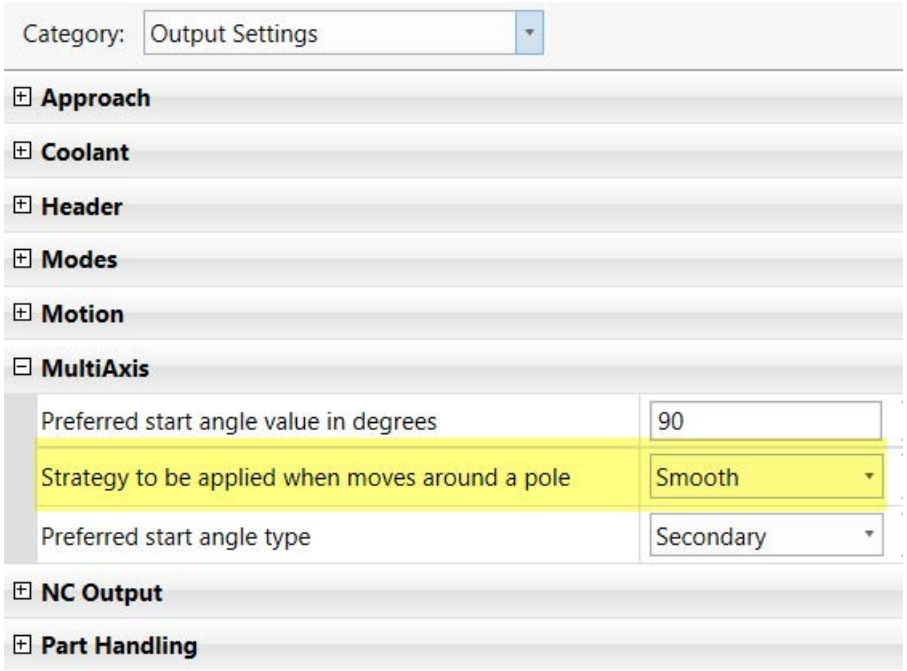
between the sections to avoid “jumps.” The jumps occur because each 3-axis section has a undetermined rotary position. **Linear** interpolation will result in a linear transition between the sections, while **Smooth** interpolation will gradually increase/decrease the angle moves between the sections.



Setting the default pole handling strategy

You can set the default pole handling strategy in the **.machine** file.

1. Open the DMG Mori **.machine** file in the Code Expert.
2. Double-click the **Consumer** layer.
3. Select **Category: Output Settings**.
4. Go to the **MultiAxis** section.
5. Select the desired strategy..
6. Save the **.machine** file.



Using tool center programming (G43.4)

Enabling smooth velocity control (G43.4) wih tool center programming.

Your DMG Mori NT/NTX **.machine** file supports the use of smooth velocity control mode (G43.4) for tool tip point control programming with multiaxis toolpaths. When G43.4 mode is turned on, the post will automatically output a G49 to turn it off. This topic explains how to:

- Turn it on or off for each operation.
- Suppres it for transition passes.
- Configure the default setting.

The default setting for each operation comes from the **.machine** file. If you override it for any operation, make sure you press **[Ctrl+S]** to save it back to your part file.

Enabling/disabling G43.4 for each operation

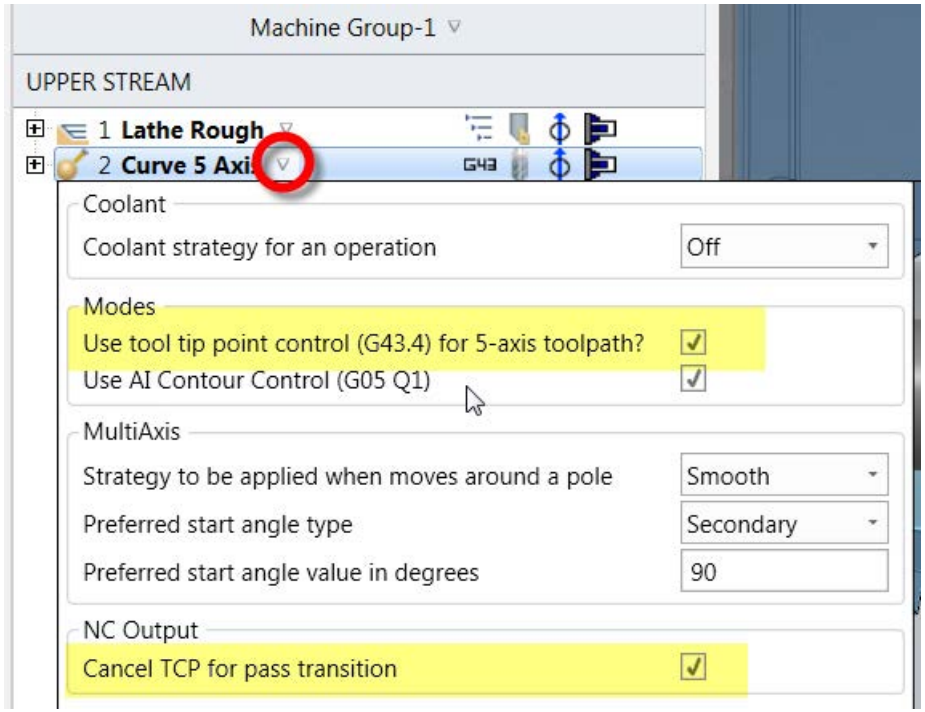
After loading the part in the Sync Manager, you can choose to turn on G43.4 mode for each individual operation.

Click the little triangle next to the operation and select the **Use tool tip point control...** option to turn on G43.4 mode. This option is available for all multiaxis toolpaths.

Cancelling 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.

Select the **Cancel TCP for pass transition** option to turn off G43.4 mode between passes.



Home

Home

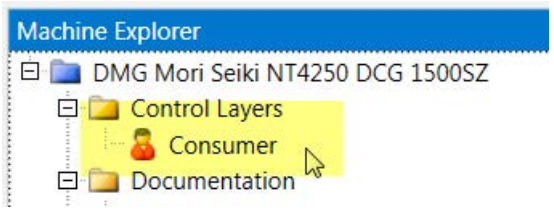
Default settings for G43.4

In the **.machine** file, choose the default setting for G43.4 mode.

Follow these steps:

1. Open the DMG Mori NT/NTX **.machine** file in the Code Expert.

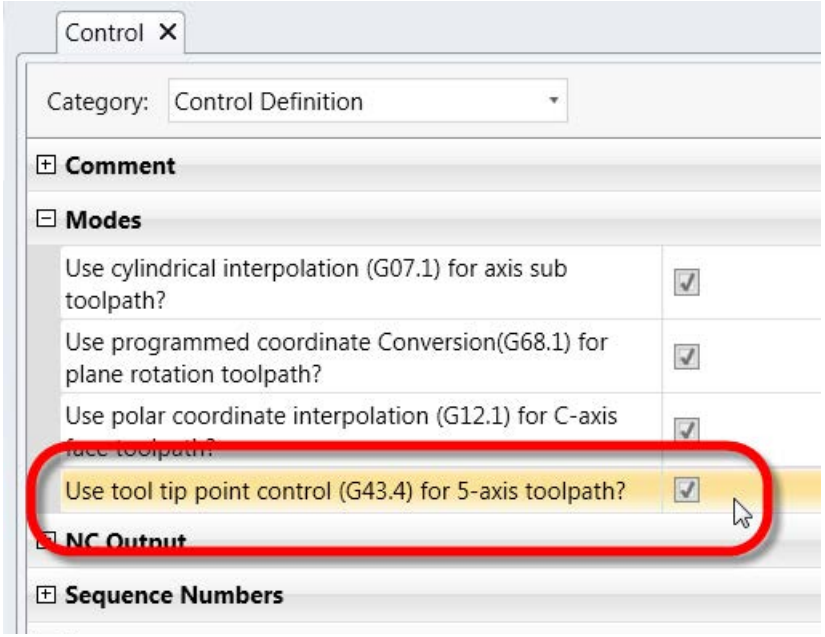
2. Double-click the **Consumer** layer.



3. Select **Category: Control Definition**.

4. Go to the **Modes** section.

5. Select the **Use tool tip point control...** option to output `G43.4` by default.



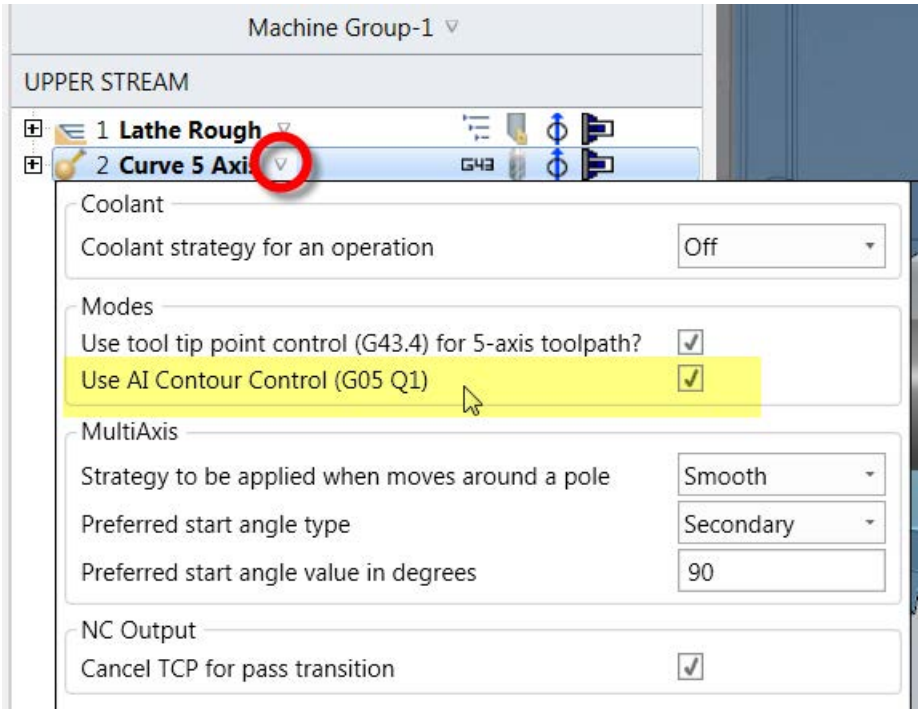
6. Save the **.machine** file.

Using AI contour control (G05.1)

Your .machine file supports G05.1 Q1 mode for AI contour control.

The option to turn on AI contour control mode (`G05.1 Q1`) is available for multiaxis mill toolpaths. Use it for high-speed, high-precision machining. Select the option in the Sync Manager for each operation for which you want to machine in this mode; Mastercam will automatically turn it off after each operation.

When using this mode, you should make sure that G332 is enabled and the proper cutting mode is selected; see “Selecting the cutting mode (G332)” on page 53 to learn more.

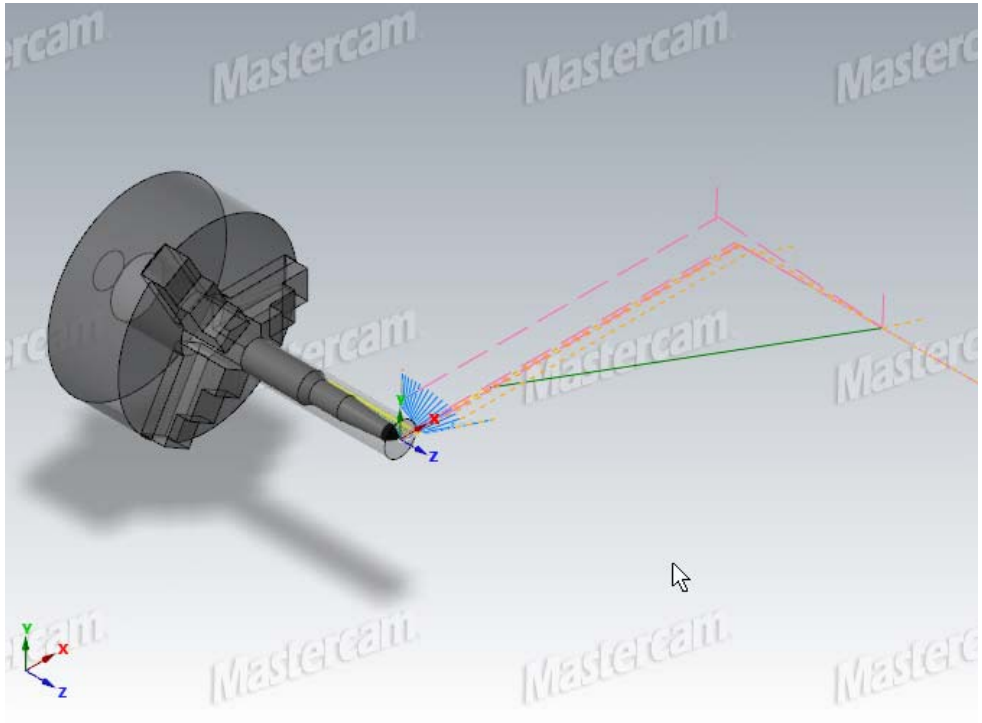


Home

D: Reference points and home positions

For each operation, you can manually determine the start and end points. Mastercam uses a Sync Manager option called *reference positions* that lets you choose this. You can also create new, custom reference positions for specific parts or part setups.

- **Setting the start point and end point for an operation**
- **Creating custom reference positions**



Home

Setting the start point and end point for an operation

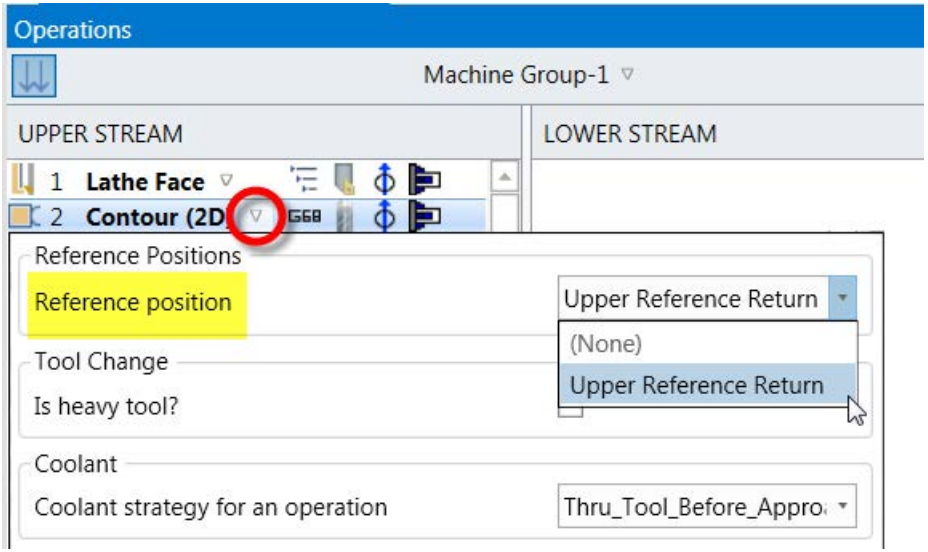
Use Sync Manager options to determine where each operation begins and ends.

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 the spindle to go between operations.

- Select specific positions for the start and end of each operation in the Sync Manager.
- You can also define additional custom 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

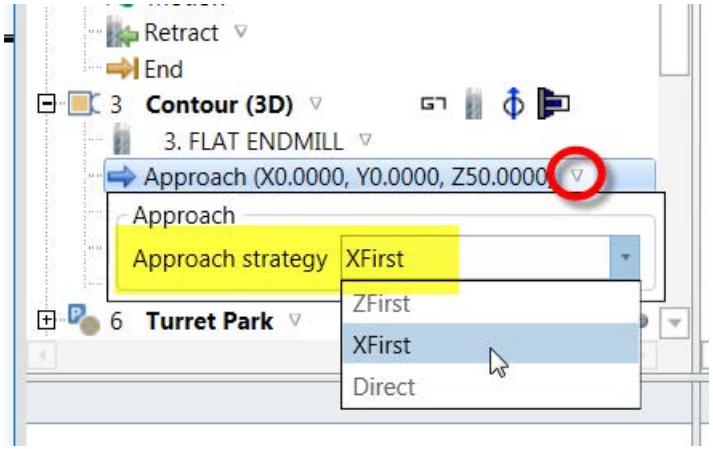
To set the start point for an operation, select the desired **Reference position** in the Sync Manager. Click the small triangle next to the operation name in the Sync Manager, and select the location from the list.



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

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

After selecting the beginning reference position, you can select the approach strategy. This defines the type of motion from the beginning reference position to the first point of the toolpath.



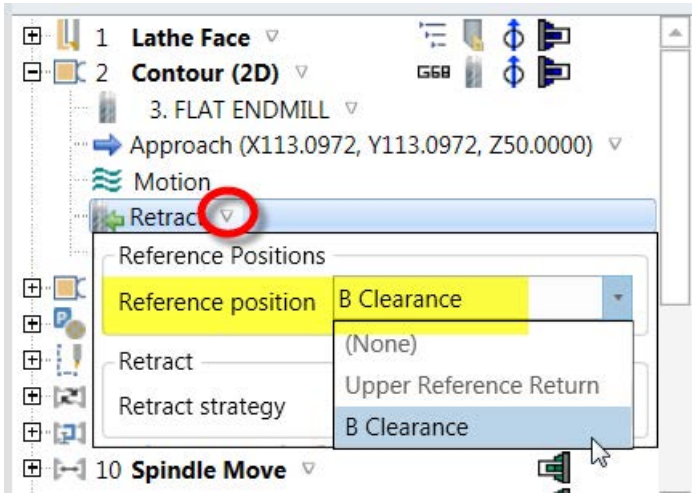
For example, **XFirst** would result in the code shown here:

```
G54
M594
M08 M684
G0 G18 X68.
Z0. Y0.
X28.
G18 W0. H0.
G98
G7.1 C18.
```

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. (See **Creating custom reference positions** on the next page.)



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   X-.2376 Z-2. I.0156 K0.
119   G1 X1.4846
120   G2 X1.5158 Z-1.9852 I0. K.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
126
127   N120
128   (OPERATION # 4)
```

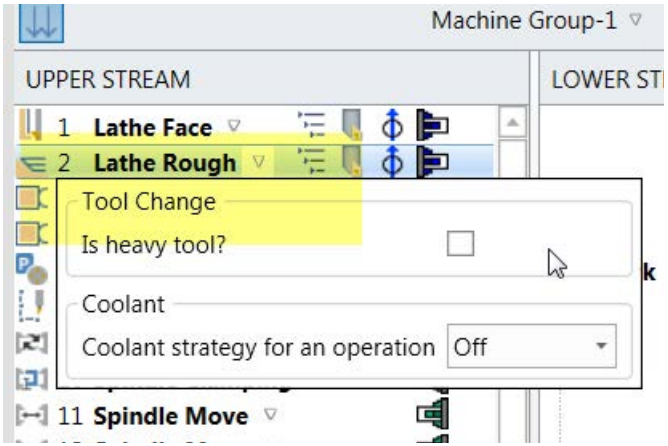
Selecting a reference position of None

Selecting **None** for a reference position means that there will simply be no output where the reference position is typically output.

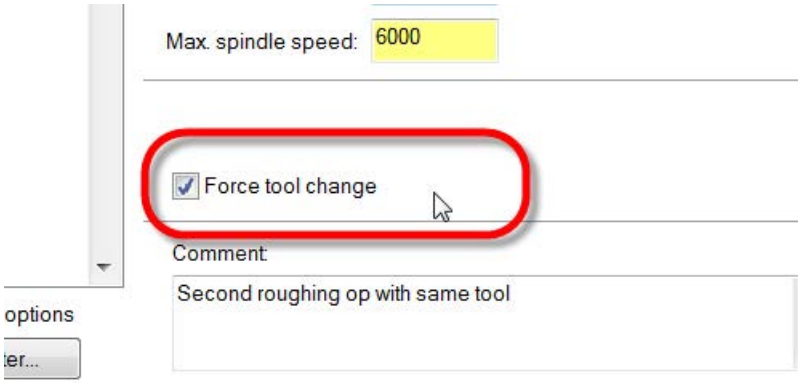
Home

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. In the picture at right, operations 1 and 2 use the same tool. You can see that the **Reference position** option is not available for operation 2.



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 operations (see next section).



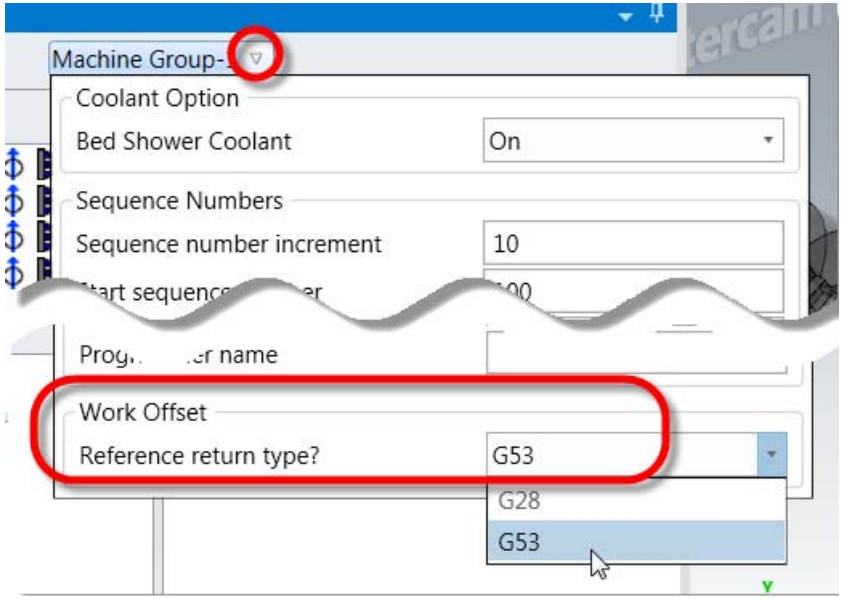
Home

Forcing G53 output

If you like, you can choose to force out a `G53` instead of `G28` for your home position moves. You can select this option in your job setup (machine group) options in the Sync Manager.

1. Click the small triangle next to your machine group name in the Sync Manager.
2. Select the **G53** option if that is how you want your home position moves to be output.

This option affects all the operations in your part.



Home

Creating custom reference positions

Create your own reference positions for specific jobs or parts.

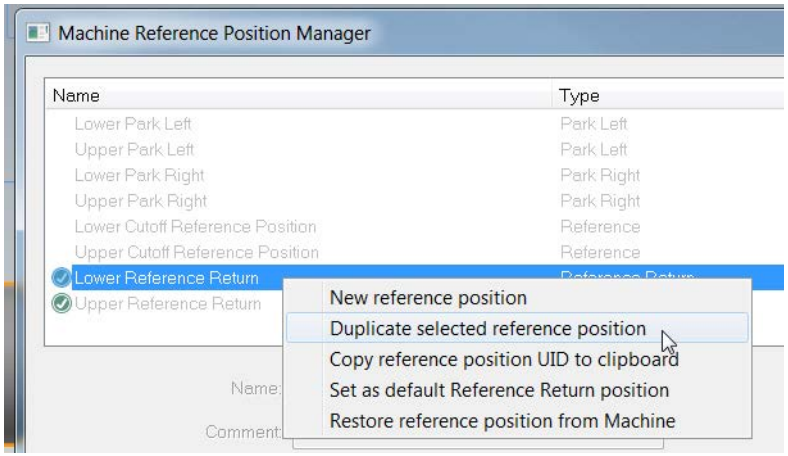
There will be times when you need reference positions that have not been defined in your **.machine** file. These might be necessary to accommodate special part fixturing or tooling, or a part with unusual dimensions. In these cases, you can define your own custom reference positions. These are saved with your part, not in the **.machine** file, and therefore are only available to the current part.

Follow these steps to create a new reference position.

1. Click the button on the Toolpath Manager toolbar.

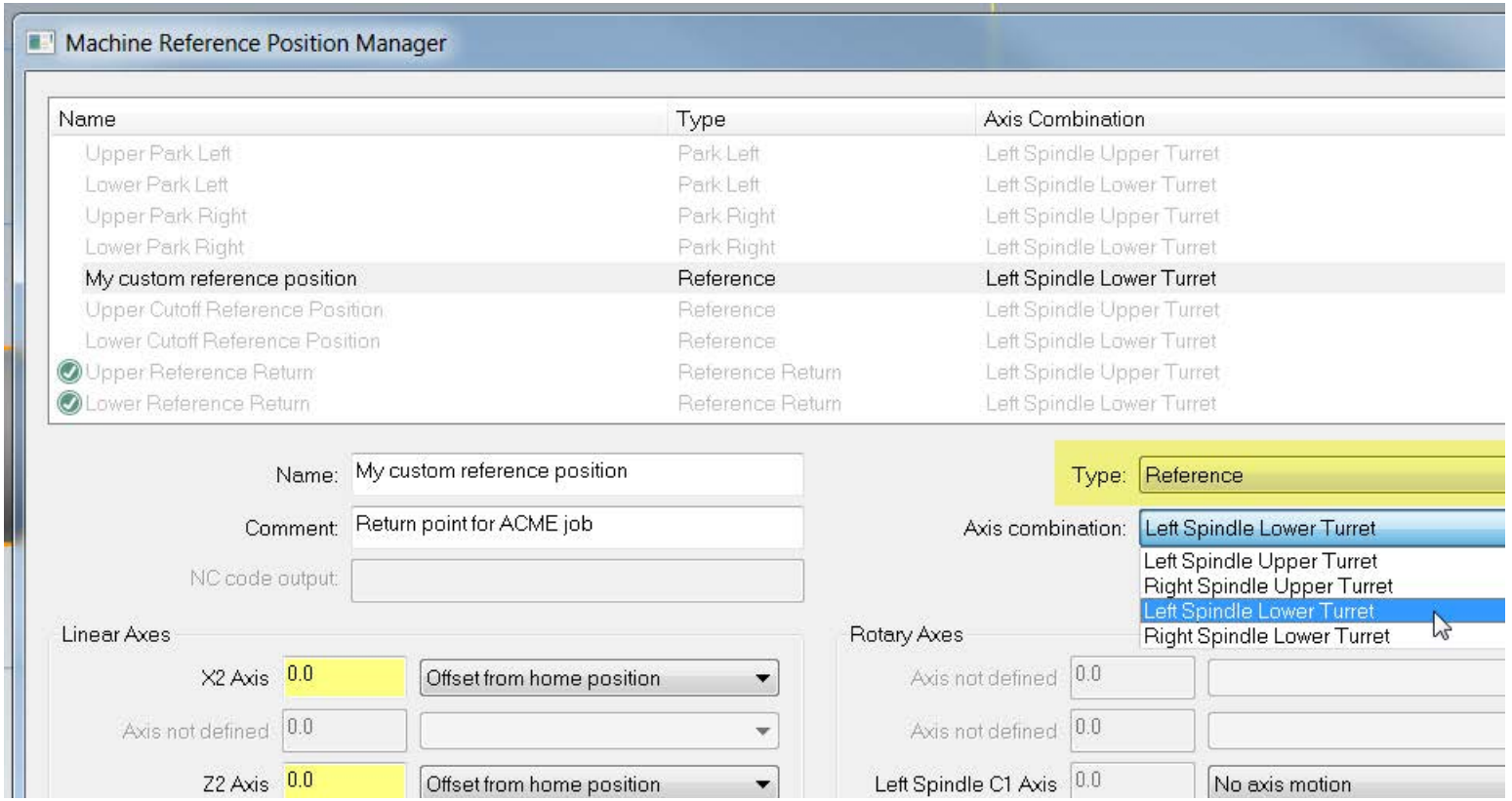
   Mastercam displays the **Machine Reference Positions Manager**, which lists all of the reference positions that have been defined for the current machine. Most of them will be grayed out, meaning that you are not allowed to edit them.

2. Right-click in the list and select **New reference position** or **Duplicate selected reference position** to create a new one.



3. Enter a **Name** and edit the other properties.

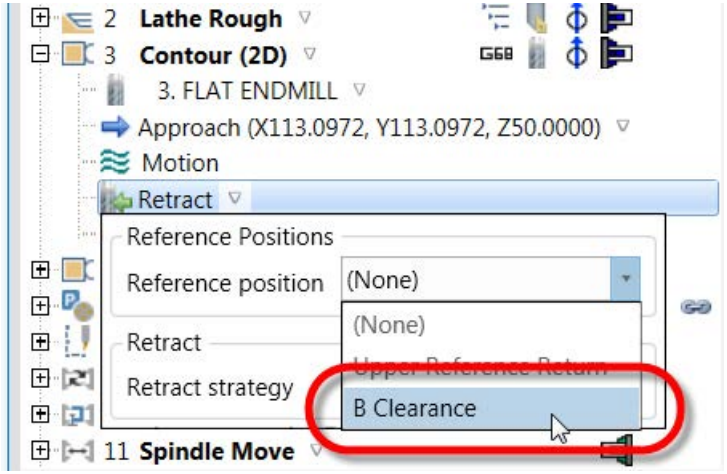
   - You can see in the list that there are several different types of reference positions; however, you can only create new ones of type **Reference**. You can duplicate a different reference position, but the **Type** will be set to **Reference** or **Park** (used for turret park operations).



   - Each reference position is associated with a specific axis combination. Mastercam uses this to filter the reference points by spindle, turret, and/or stream. For example, when you select reference points in the Sync Manager for a lower stream operation, only reference points valid in that stream will be displayed.

- For each reference position, enter the desired axis motion and coordinate. You can choose to move any or all axes. Mastercam only displays axes that are included in the selected axis combination.

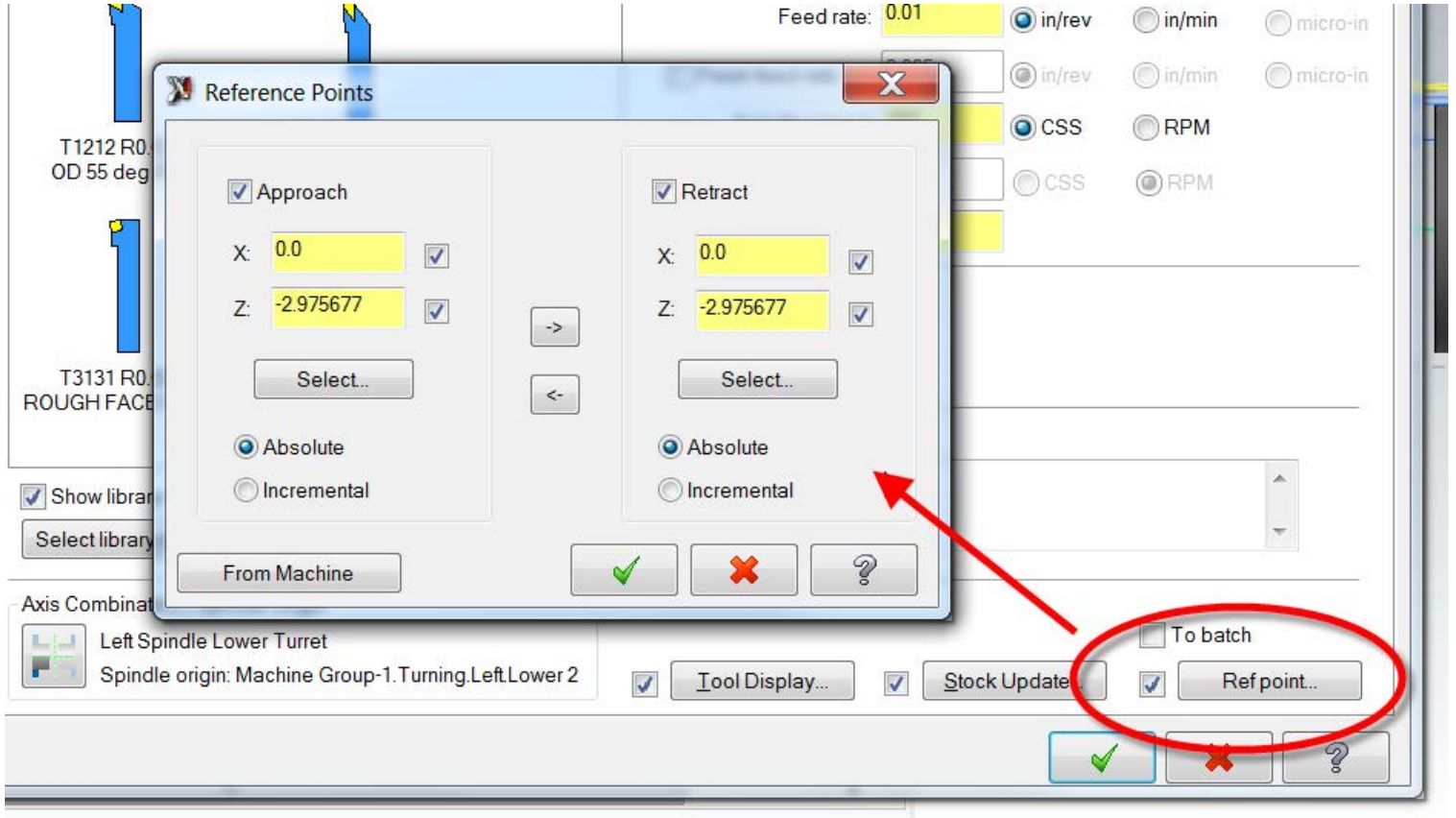
4. Click **OK** when you are done.
5. Click **G1** to post the operations.
6. Go to the Sync Manager. The new reference position should be available. Make sure you look in the proper stream, if your machine tool supports multi-stream output..



Home

Reference positions and reference points

Mastercam toolpaths still include the **Reference Point** feature. This is available in the toolpath parameter pages for each Mill and Lathe toolpath. In Mill-Turn, reference *points* define an intermediate position between the toolpath and the reference *position* that is selected in the Sync Manager. For example, if you are turning an ID operation, you might create toolpath reference *points* (as shown in the picture) to make sure the tool fully retracts from the ID before moving to the reference (or home) position. Do not make the mistake of confusing reference *points* and reference *positions*.



E: Coolant and other toolpath options

This section discusses the workflows for using coolant as well as some other miscellaneous options for your NC programs.

- **Using coolant**
  - Bed shower coolant (M382)
  - Setting the default coolant option
- **Configuring stops and optional stops**
  - Outputting an optional stop with wait codes
  - Outputting an optional stop after each operation
  - Stopping the spindle between lathe operations
- **Outputting the operation number for each toolpath**

Home

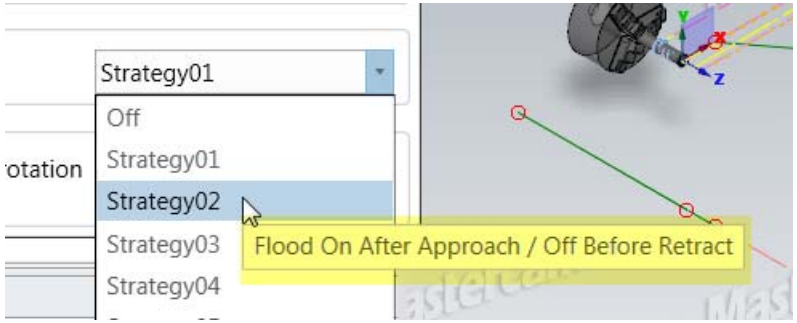
Using coolant

Use Sync Manager options to select pre-configured coolant strategies.

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

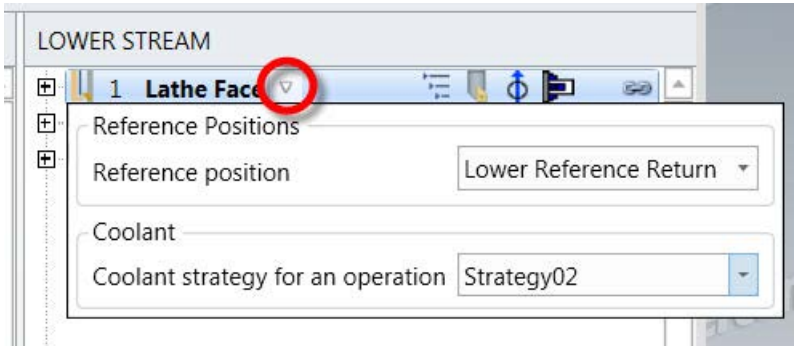
- Flood coolant for B-axis head and lower turret(`M08`)
- Thru-spindle coolant for the milling spindle (`M484`)
- Bed shower coolant (`M382`)

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.

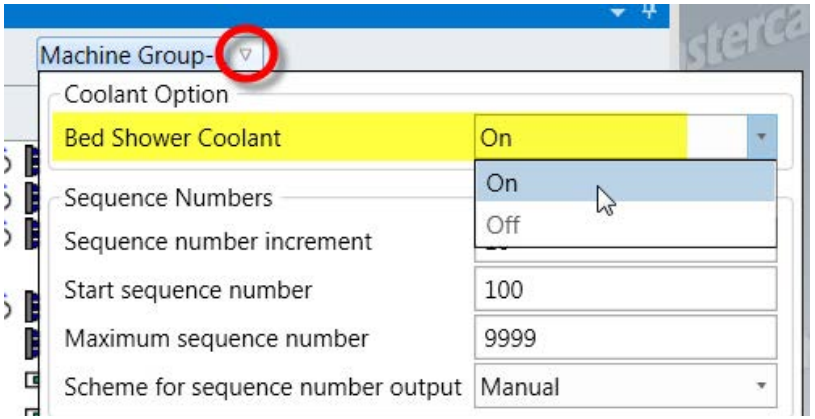
To select a strategy, click the small triangle next to the operation name and select the desired coolant strategy. Do this for each operation.



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.

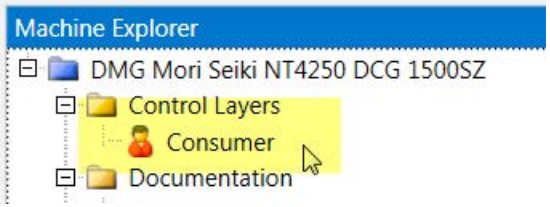
Bed shower coolant (M382)

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.



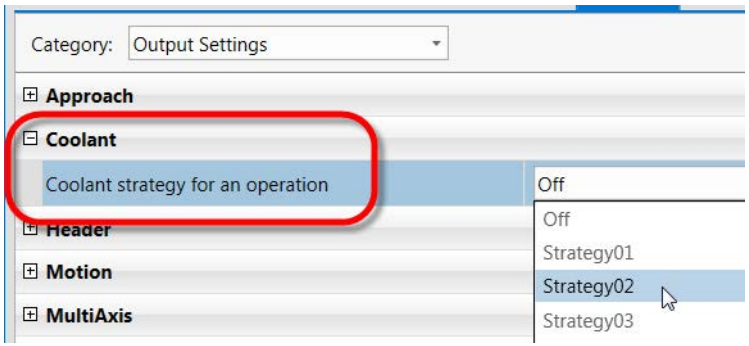
Setting the default coolant option

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



1. Open the DMG Mori NT/NTX **.machine** file in the 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.

Configuring stops and optional stops

You can choose to output an M01 with a wait code (sync), after each operation, or to stop the spindle after each turning operation.

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

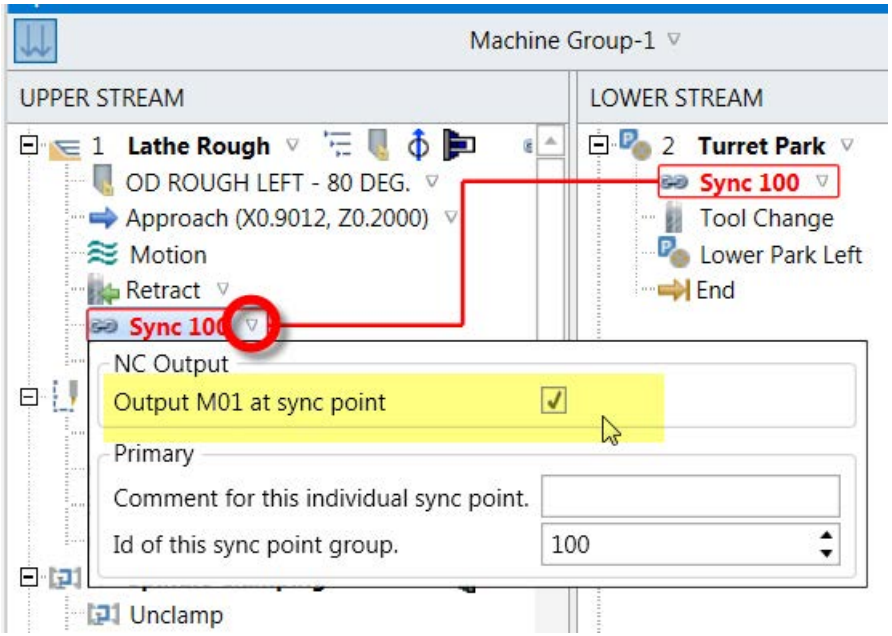
- Output an optional stop (`M01`) on every wait code or sync.
- Output an optional stop (`M01`) after every operation.
- Turn off the spindle (`M05`) after each lathe operation.

See the following sections to learn about these options.

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:

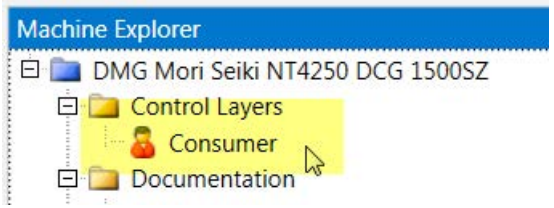
1. In the Sync Manager, locate the sync for which you want to output the `M01`.
2. Click on the small triangle after the sync command.
3. Select the **Output M01 at sync point** option.
4. Repeat these steps for the sync in the other stream! You must select this option in *both* streams.
5. Save the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



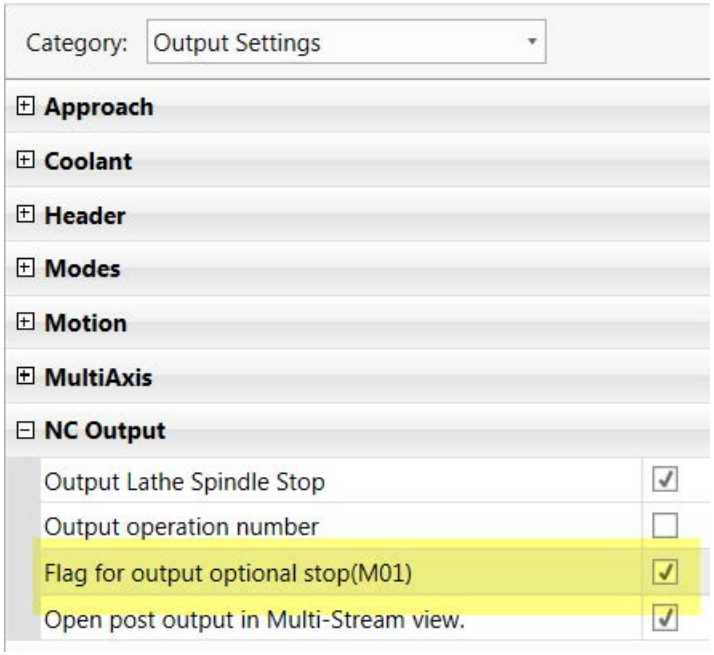
Outputting an optional stop after each operation

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 **.machine** file in the 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.

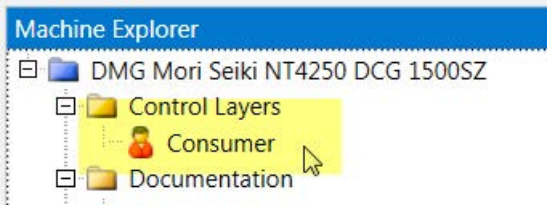


Home

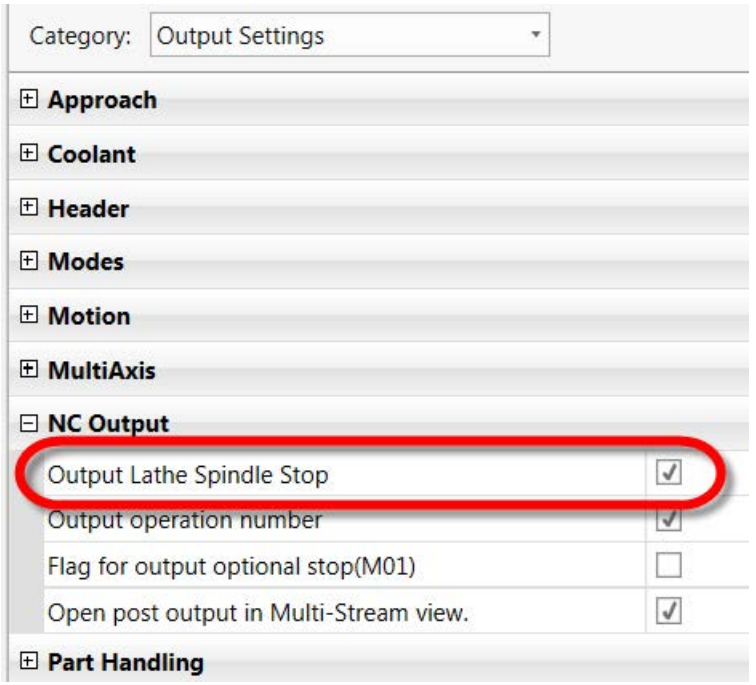
Stopping the spindle between lathe operations

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

1. Open the DMG Mori NT/NTX **.machine** file in the 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.



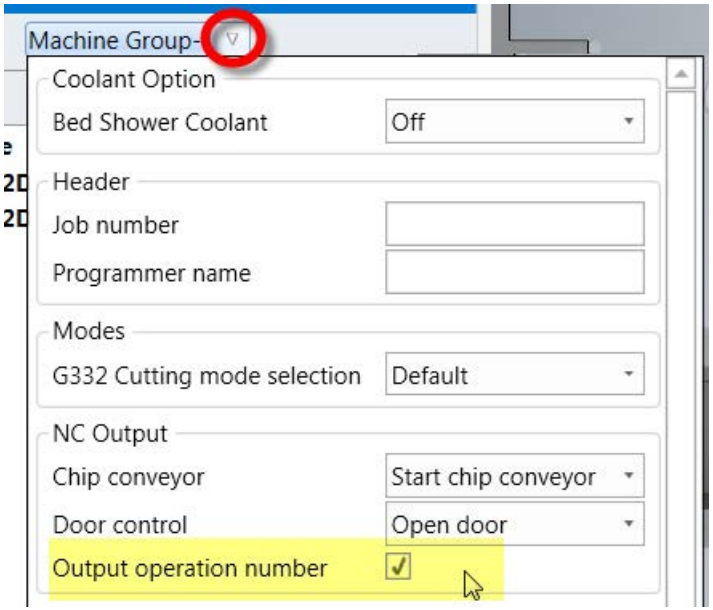
Outputting the operation number for each toolpath

Choose whether or not to output the operation number in your NC file.

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

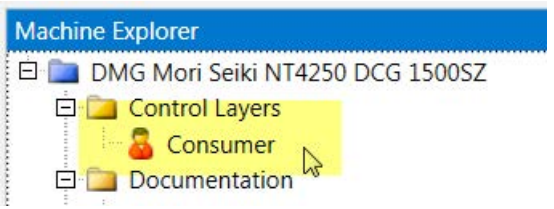
```
11  N130
12  (OPERATION # 2)
13  G10.9 X0
14  (T0101 | OD ROUGH LEFT - 80 DEG. |
15  M46 (TURNING MODE - MAIN SPINDLE)
16  G28 U0.
17  G28 W0.
```

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.

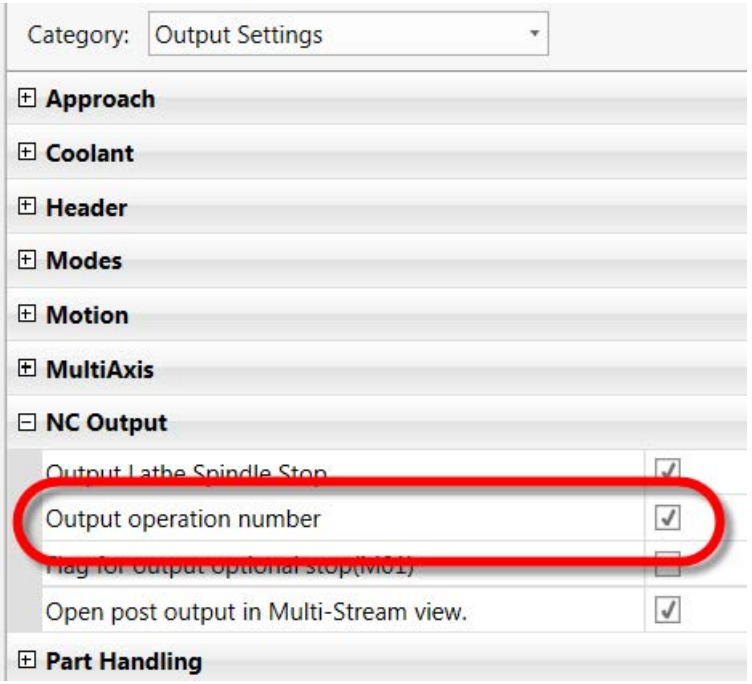


You can also control the default setting. Follow these steps:

1. Open the DMG Mori NT/NTX **.machine** file in the 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.



Home