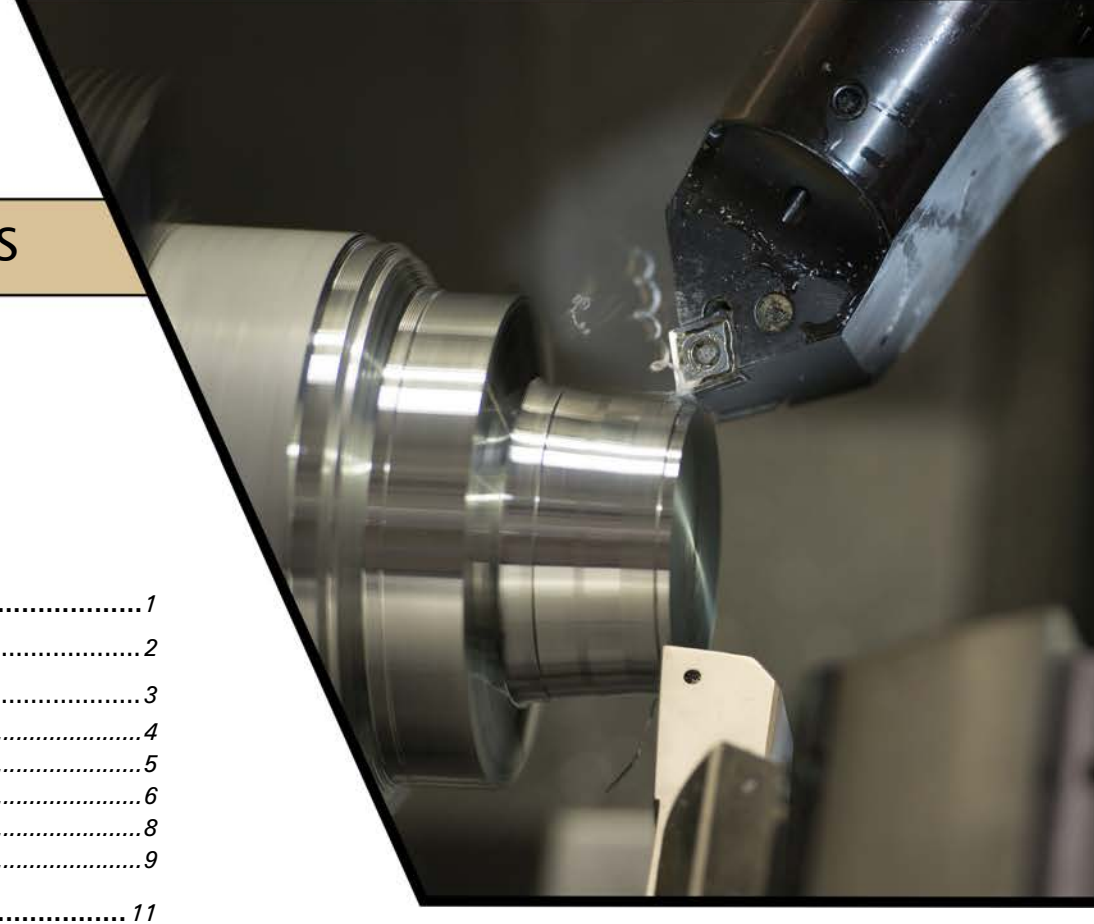
MILL-TURN APPLICATION GUIDES

Generic Fanuc Lathe



revision date: May 25, 2018

Contents

Chapter 1: Working with .machine files *1*

A: Installing your .machine file *2*

B: Customizing your .machine file *3*

Default values for .machine file settings *4*
Configuring the Code Expert editor *5*
Customizing operation defaults and tool libraries *6*
Configuring tool table output *8*
Setting the toolpath directory *9*

Chapter 2: Working with tools and spindles *11*

A: Setting up tools *12*

Initial tool setup *13*
Setting up for the left or right spindle *16*

B: Spindle control functions *17*

Selecting the spindle winding range (M61, M62) *17*
C-axis clamp (M89/M189) *18*

Chapter 3: Working with toolpaths *21*

A: Toolpath reference positions *22*

Setting the start point and end point for an operation *23*
Setting the type of approach/retract motion *25*
Creating custom reference positions *27*
Reference positions and reference points *30*

B: Machining modes and other options *31*

Polar (G12.1) and cylindrical (G7.1) interpolation *32*
Using coolant *35*
Output expanded Mcode comments *37*

Mastercam

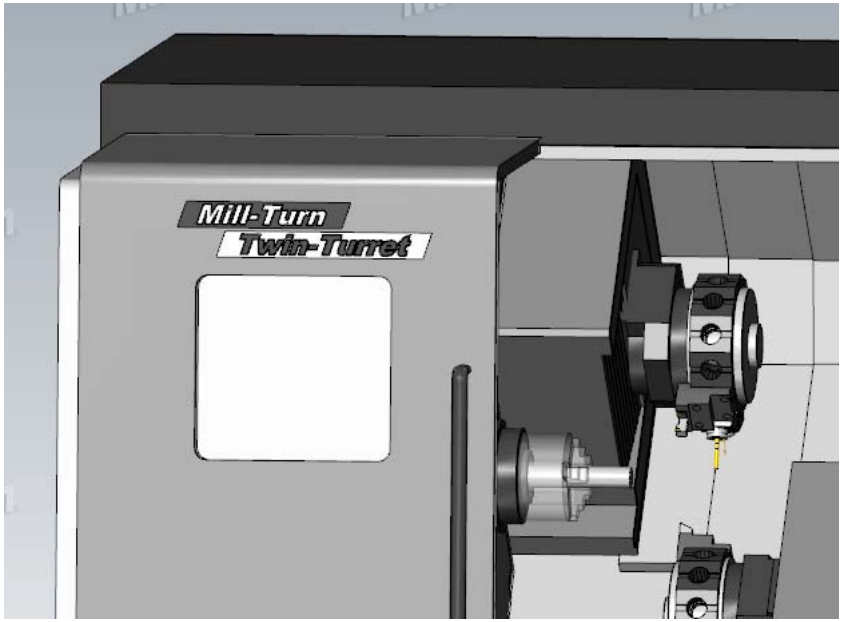
Mill-Turn Application Guide—Generic Fanuc Lathe
Copyright © 2018 CNC Software, Inc.—All rights reserved

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

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 other Mastercam products. 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 file settings**
- **Configuring the Code Expert editor**
- **Configuring tool table output**
- **Setting the toolpath directory**



About this machine—The Generic Fanuc Lathe **.machine** file and post are designed to replicate a typical Fanuc control on a single-stream lathe. This guide covers the following **.machine** files:

- Generic Fanuc Lathe DS (single turret, two spindles)
- Generic Fanuc Lathe TC (single turret, tailstock)
- Generic Fanuc Lathe (single turret, single spindle)

It is intended to be used for:

- training purposes & demonstrations
- working through the exercises in the *Getting Started with Mill-Turn* application guide.

IMPORTANT: This **.machine** file and post are designed to produce sample NC code only! Do **not** attempt to run ANY part program produced by this **.machine** file and post on an actual machine.

Home

Rev. date: May 2018

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

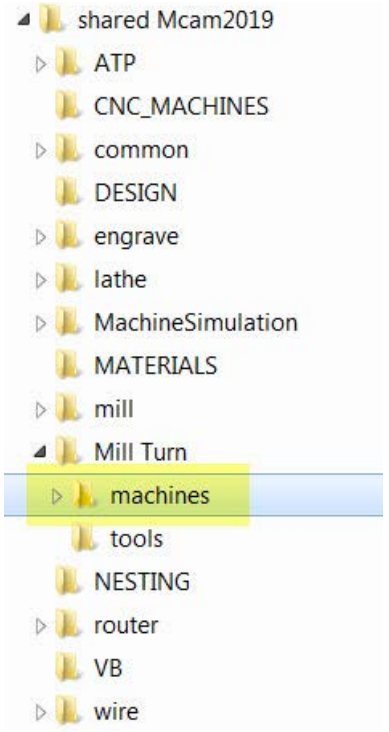
\shared Mcam2019\Mill Turn\MACHINES

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

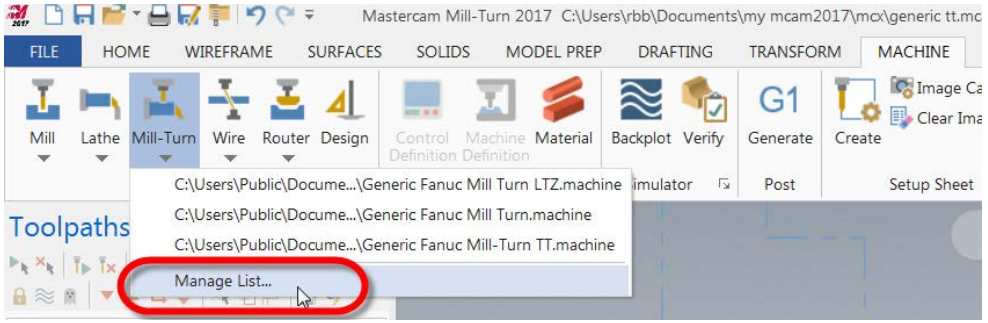
Unlike posts from other Mastercam products, the single **.machine** file includes all the resources that you need to support your Fanuc Lathe application. You do not need to worry about linking the **.machine** file to other files, like posts.

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.

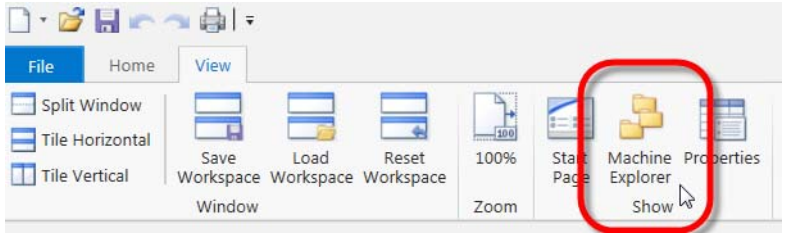


Home

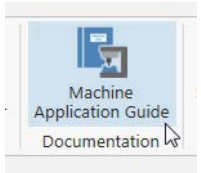
B: Customizing your .machine file

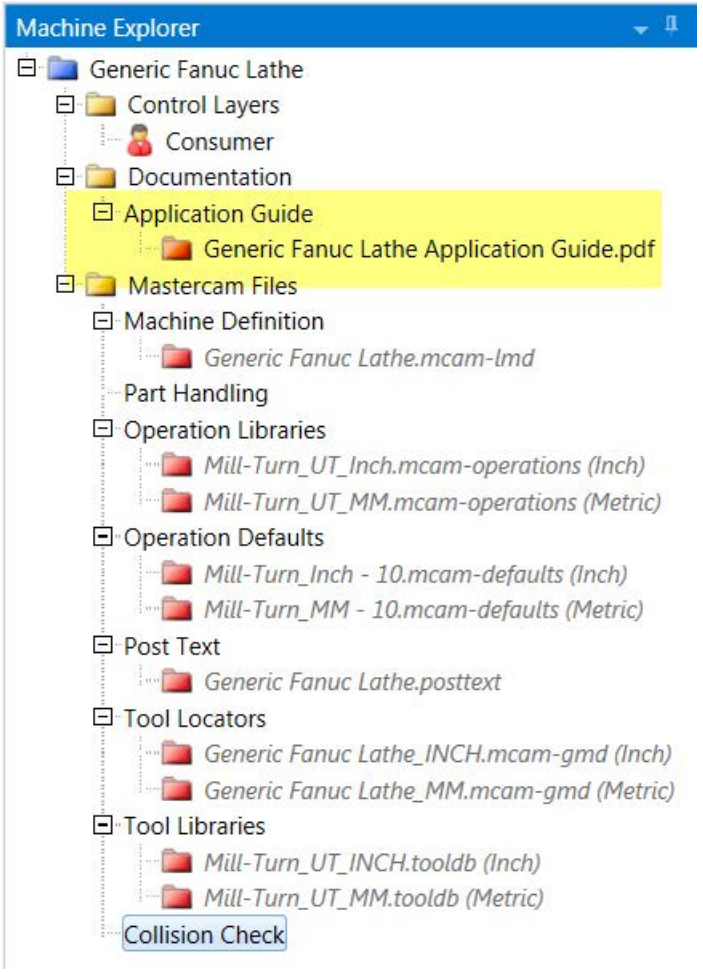
Whenever you load your **.machine** file in Mastercam to begin working on a part, Mastercam also starts Code Expert and loads your **.machine** file there as well. Code Expert is where you can make changes to your **.machine** file, such as editing default settings.

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



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



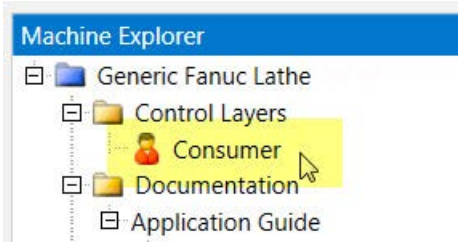


Home

Default values for .machine file settings

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

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



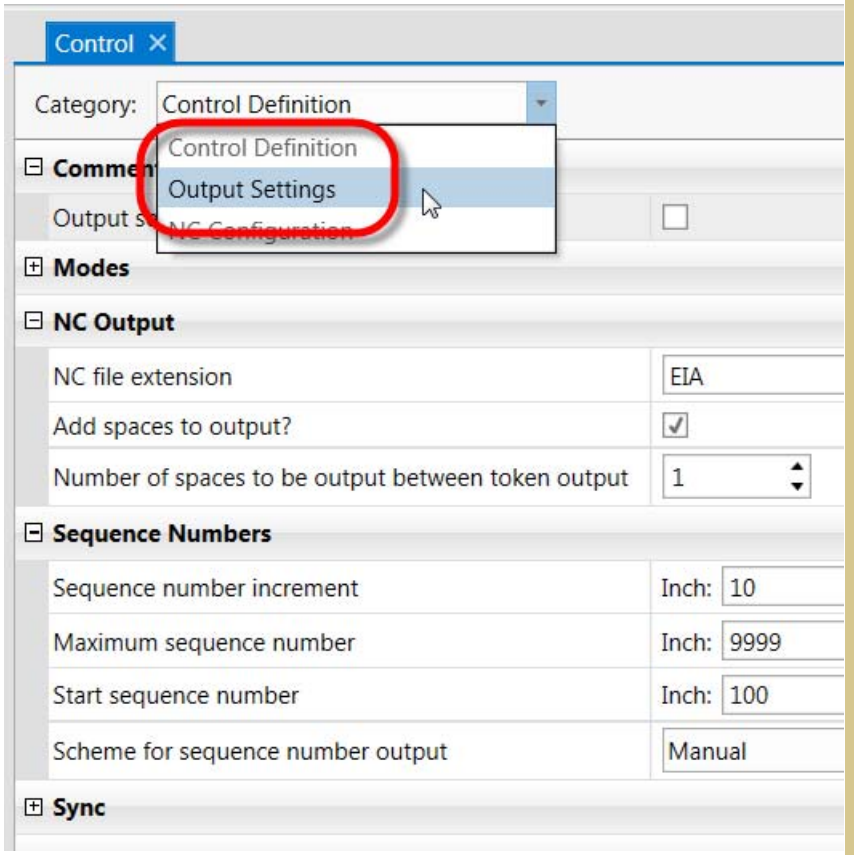
The settings are grouped into several categories. Click the **+** signs to see the individual options. Some of these options (for example, sequence number settings) are very common or generic to most controls; others are specific to your individual machine.

These settings serve a wide variety of functions:

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

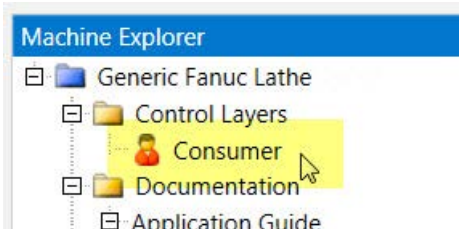
Many of these settings are self-explanatory and you can easily configure them by simply browsing the interface. The settings that are specific to this **.machine** file are described in this guide.

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



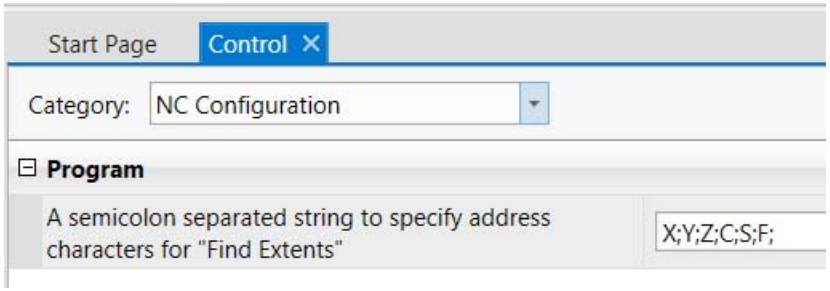
Configuring the Code Expert editor

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

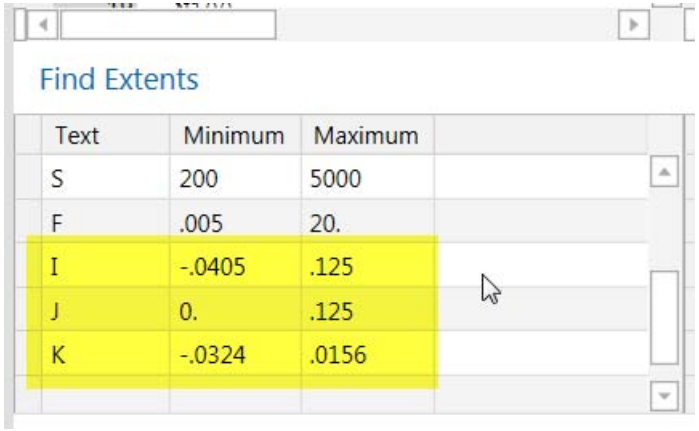


Configuring the Find Extents feature—The **Find Extents** feature in Code Expert scans your NC file and displays the minimum and maximum values for each letter address. By default, this is set to scan **X**, **Y**, **Z**, **C**, **S** and **F**. If you wish, you can edit the set of addresses that are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

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



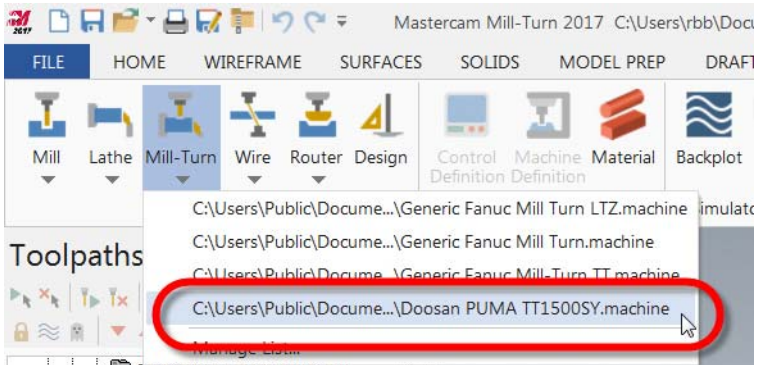
For example, you can choose to add **I;J;K;** to the list. The next time you post, you will see them in the **Find Extents** table.



Customizing operation defaults and tool libraries

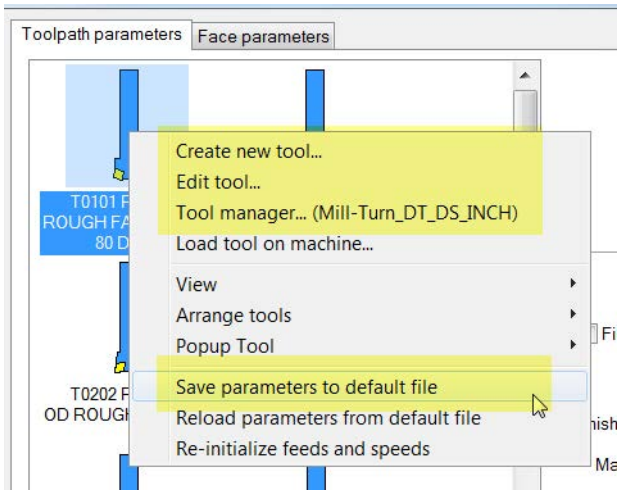
In other Mastercam products, you are used to working with tool libraries and **.defaults** files by simply loading them from your hard disk. The workflow is a little different in Mill-Turn because in Mill-Turn, these files are stored inside the **.machine** file. Follow this general outline.

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:



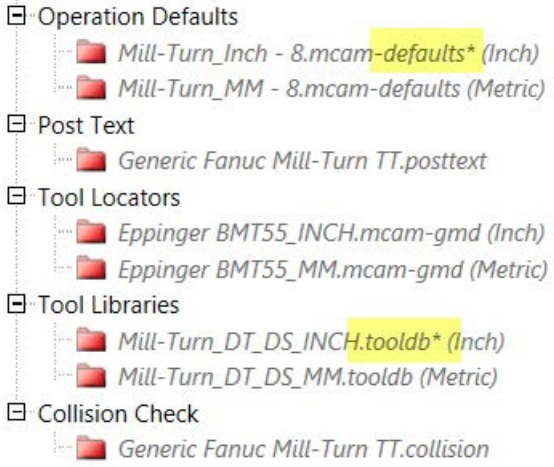
- You can use the **Edit** buttons on the **Machine Group Properties > Files** tab.

- 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.



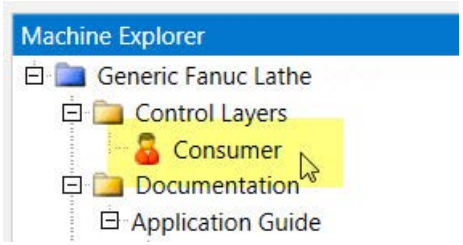
6. Press **[Ctrl+S]** in Code Expert to save the **.machine** file with your changes.

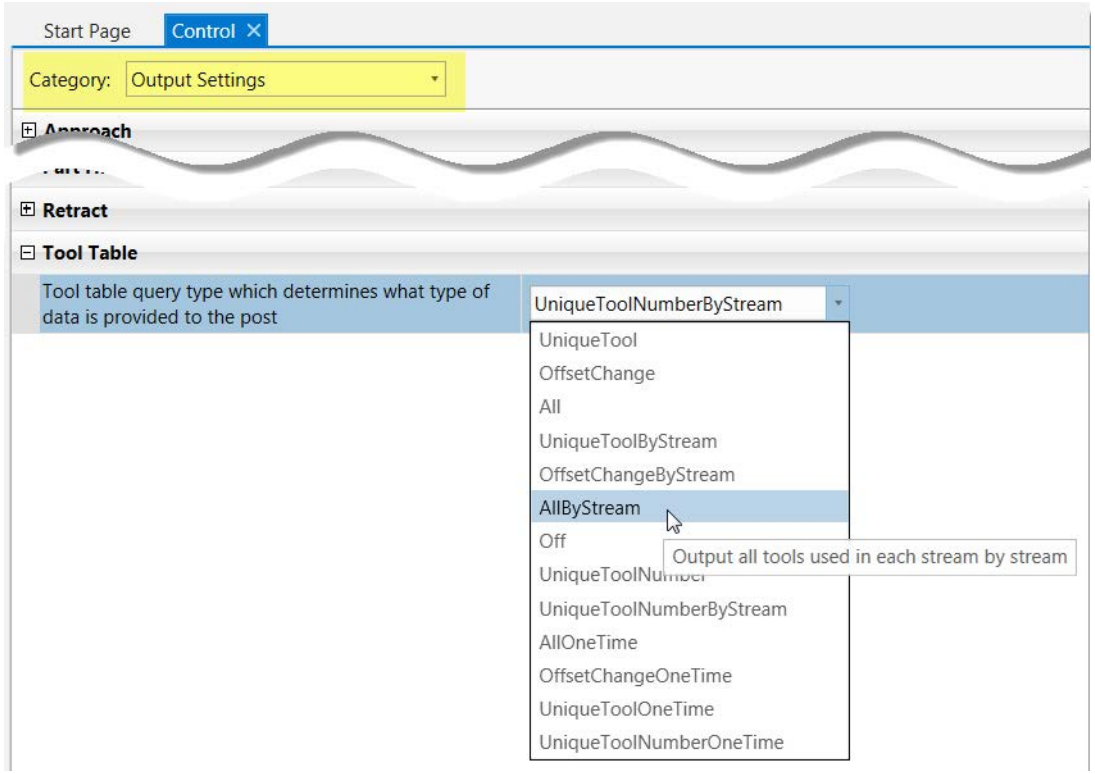
Home

Configuring tool table output

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

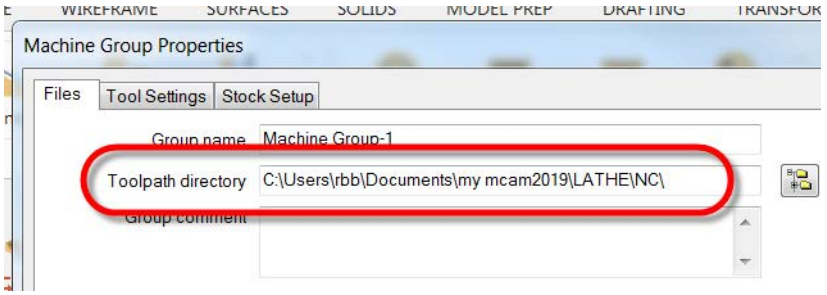




Home

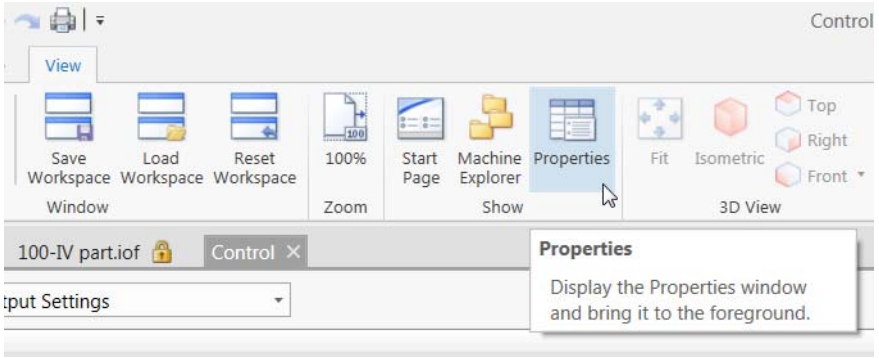
Setting the toolpath directory

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



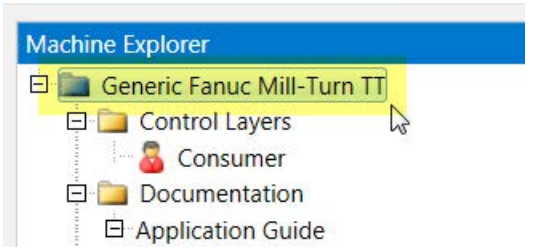
In Mill-Turn, this setting is not used. Instead, it is a property of the **.machine** file. To set it, follow these steps:

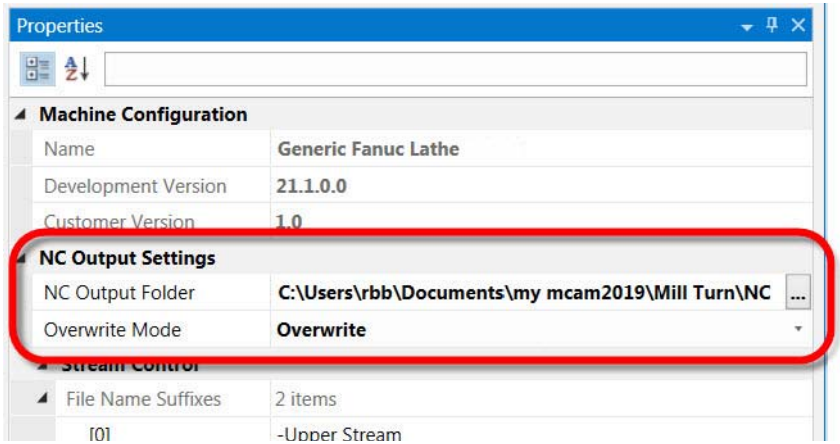
1. Open the **.machine** file in CodeExpert.



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

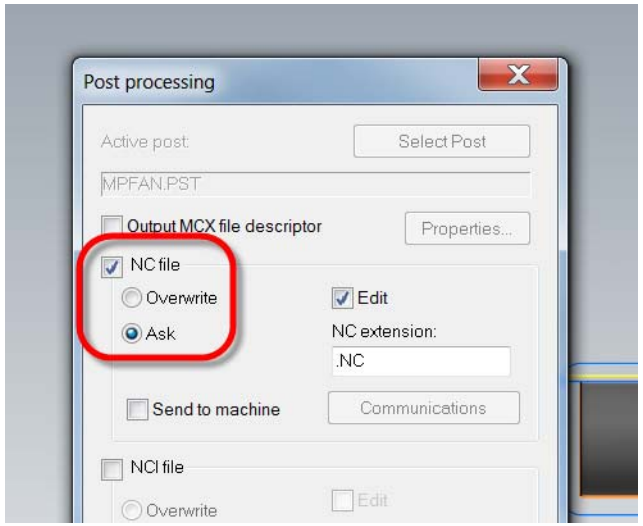
3. Click the machine name in the **Machine Explorer**.





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. Save the **.machine** file when you are done.

Home

Chapter 2: Working with tools and spindles

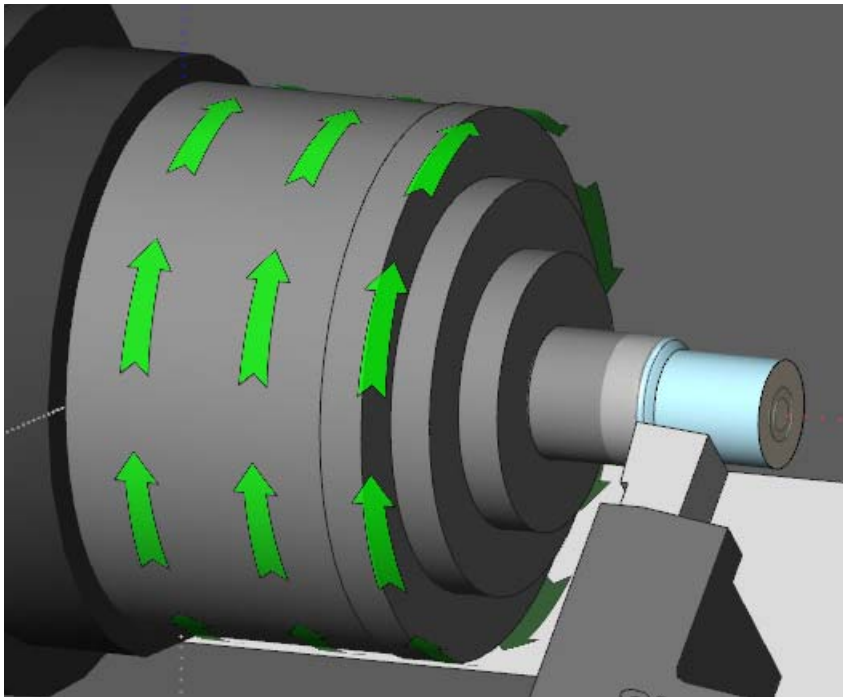
Proper tool use is controlled by:

- **Initial tool setup**
- **Setting up for the left or right spindle**

This chapter explains how these different settings work together to produce the proper tool call output for your Fanuc Lathe machine.

Your Fanuc Lathe **.machine** file also supports a number of spindle control commands, such as spindle syncing, clamping, and more.

- **Selecting the spindle winding range (M61, M62)**
- **C-axis clamp (M89/M189)**



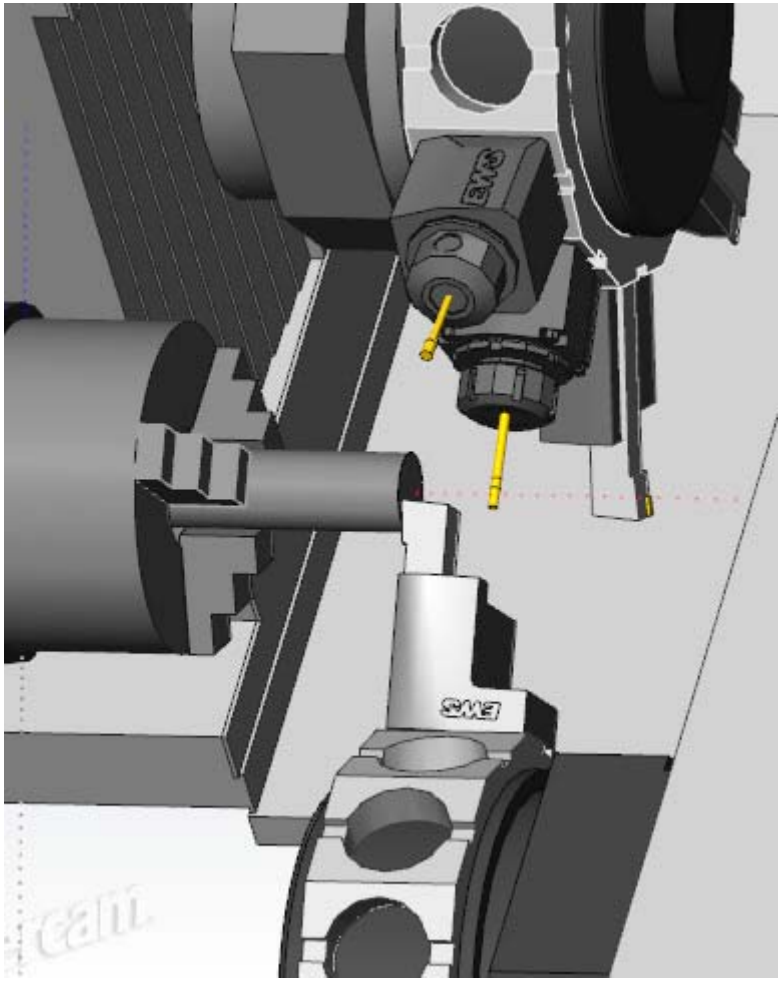
Home

Rev. date:
May 2018

A: Setting up tools

The following sections show you which specific Mastercam settings your post is expecting so that it can output tool calls in the proper format.

- **Initial tool setup**
- **Setting up for the left or right spindle**

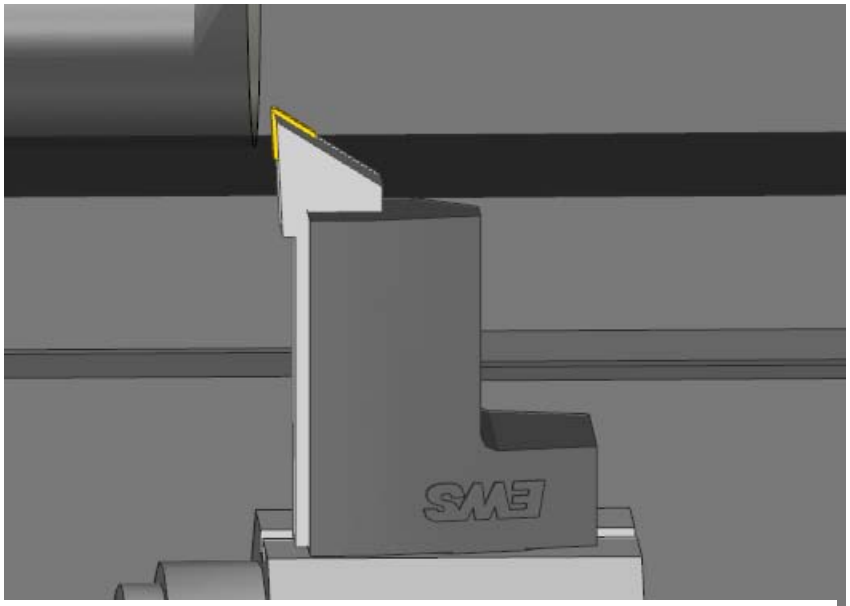


Home

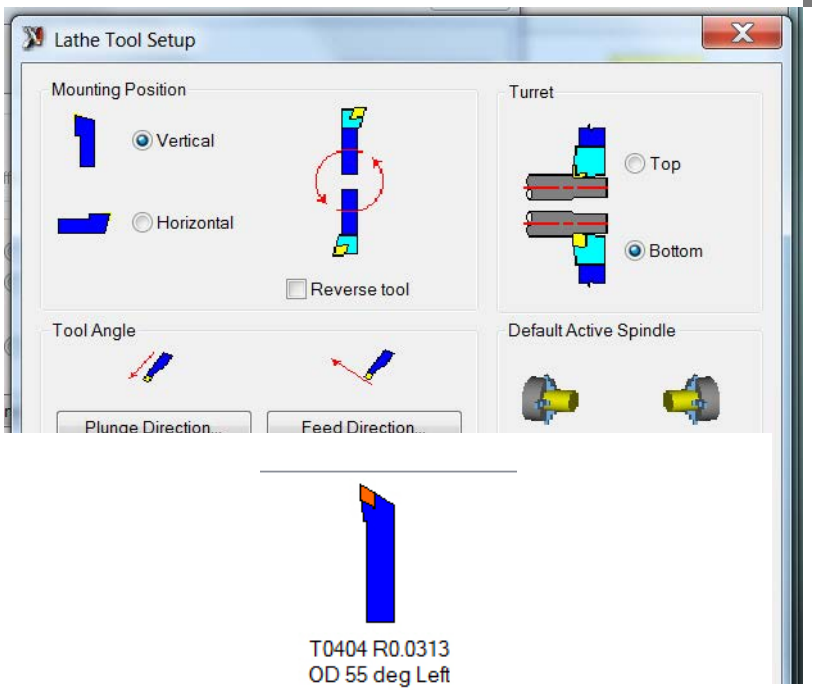
Initial tool setup

Tools for both upper and lower turrets should be defined so that they are oriented proper for the intended spindle. They should be defined in the proper vertical or horizontal orientation.

Consider the toolpath shown at right. It uses a vertical, left-hand, insert-down tool.



The tool definition settings for the tool as pictured above would look like this in Mastercam.

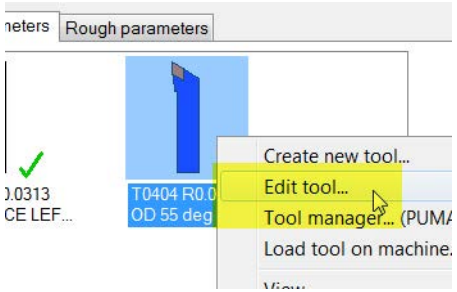


Then when you select the tool in Mastercam, the picture in the tool selection window should look like this.

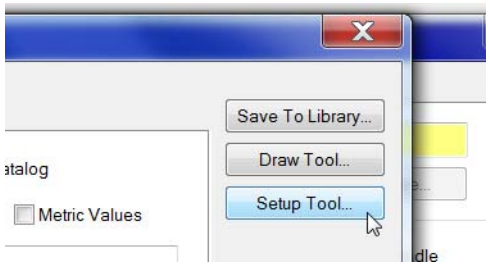
The picture that you see in Mastercam should show the proper orientation for:

- vertical/horizontal mounting
- left-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**.



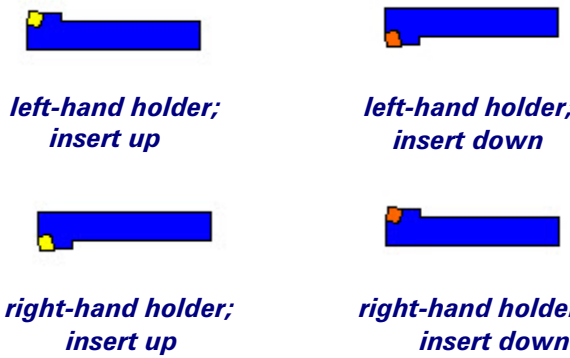
3. Click **Setup Tool** and edit the settings as desired. See “Insert up/down and LH/RH holders” on page 15 to learn more.
4. Decide whether you want to keep the changes for all parts, or just the current part file.
   - Click **Save To Library** if you want the changes to apply to all future parts.
   - Click the **OK** button only if you just want the changes to apply to the current job.

Home

Insert up/down and LH/RH holders

Tools that are mounted for a given spindle 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 it is in its eventual cutting position.

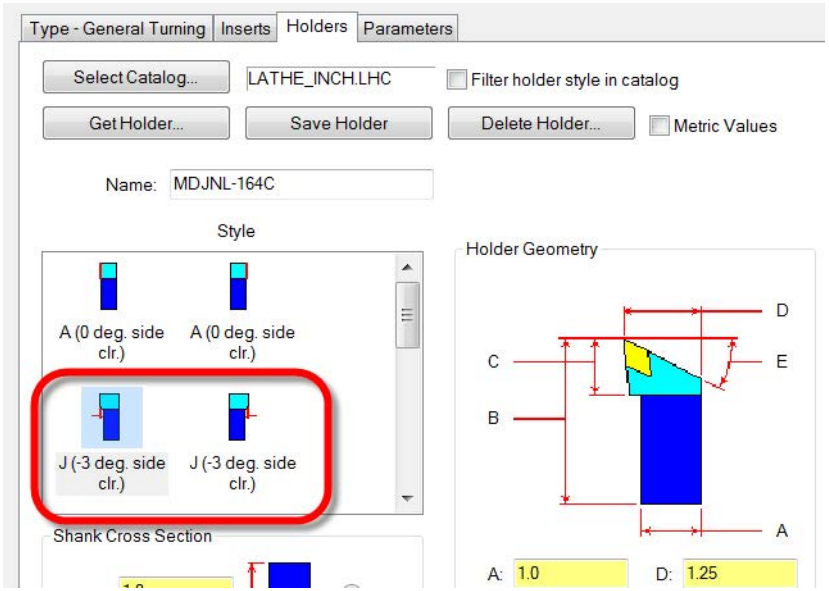
The tool preview window in Mastercam shows you how the tool is oriented. Notice that the insert color changes to show up (yellow) or down (orange).



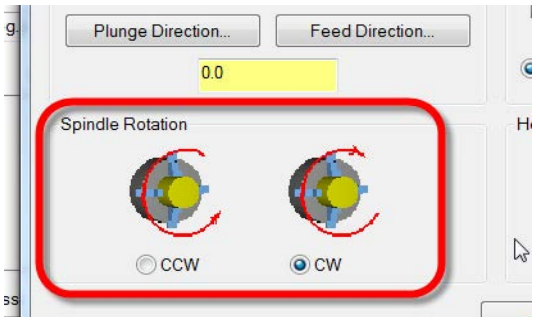
To switch between left- and right-hand holders or insert up/down (while maintaining the spindle orientation), follow these steps:

1. Right-click on a tool and select **Edit tool** to go to the **Tool Definition** dialog box.
2. Select the **Holders** tab.

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



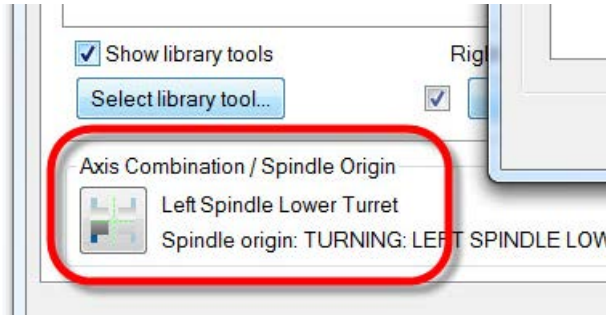
4. Click **Setup**.
5. Change the **Spindle Rotation** direction.



Setting up for the left or right spindle

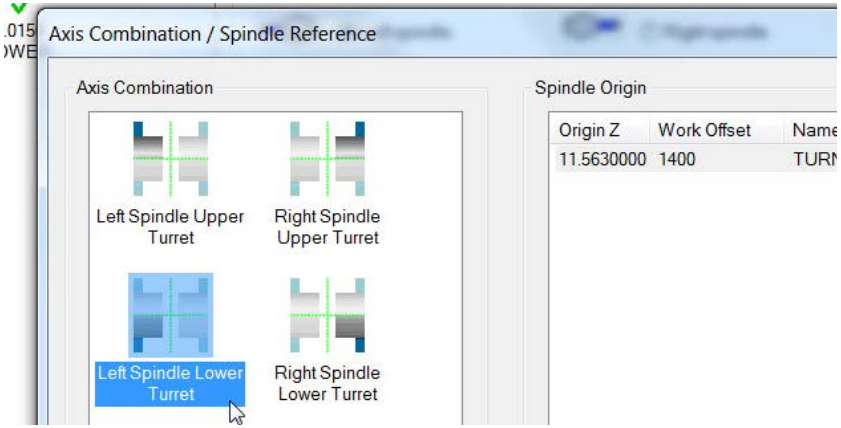
Once the tools have been defined properly, use the axis combination to tell Mastercam which spindle you are using. Follow these steps:

1. Click the **Axis Combination** button. Typically it is better to select this before selecting a tool.



2. Select the desired turret/spindle combination from the **Axis Combination** window. This picture shows the axis combination for using the lower turret on the left spindle.

3. If necessary, select the proper setup from the **Spindle Origin** list. (This is not common.)



4. Select the desired tool after selecting the axis combination. Make sure the picture in Mastercam shows it oriented for the selected spindle.

Home

B: Spindle control functions

Your **.machine** file supports a number of spindle control functions that affect how your spindle/chuck operations are output.

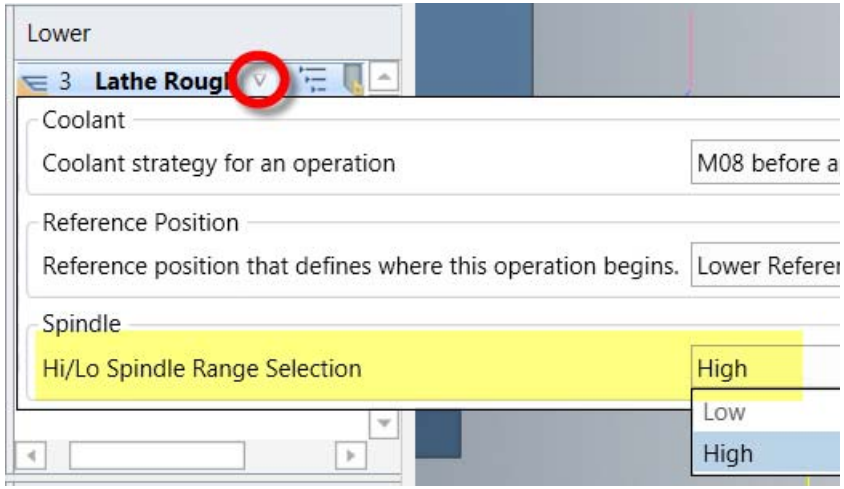
- **Selecting the spindle winding range (M61, M62)**
- **C-axis clamp (M89/M189)**

Selecting the spindle winding range (M61, M62)

Your Fanuc Lathe **.machine** file lets you select the spindle winding range (Low or High) for each spindle to the desired torque.

- Low/High range for left spindle = `M61/M62`
- Low/High range for right spindle = `M161/M162`

Do this by selecting the **Hi/Lo Spindle Range Selection** option in the Sync Manager. Select **Low** for greater torque.



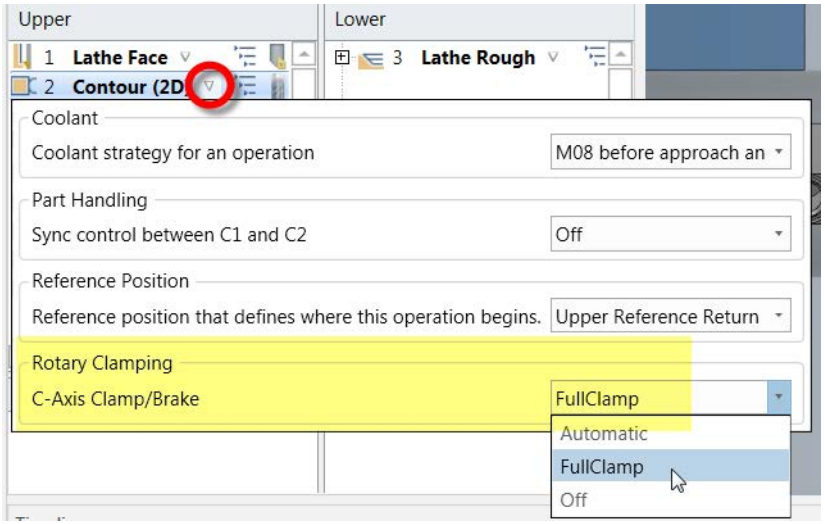
Home

C-axis clamp (M89/M189)

The C-axis high-pressure clamp mode is available for Mill toolpaths. This is an `M89` code for C1 (left spindle) and `M189` for C2 (right spindle).

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

- **Automatic**—Mastercam will automatically decide whether to clamp the C-axis based on the application and toolpath type.
- **FullClamp**—Force `M89/M189` output.
- **Off**—Suppress `M89/M189` output for the selected operation.

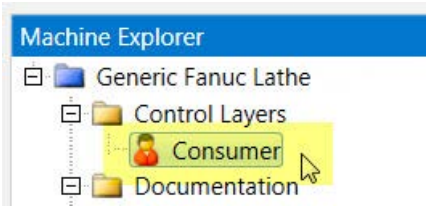


Default settings for M89/M189

You can choose which `M89` mode will be the default

Follow these steps:

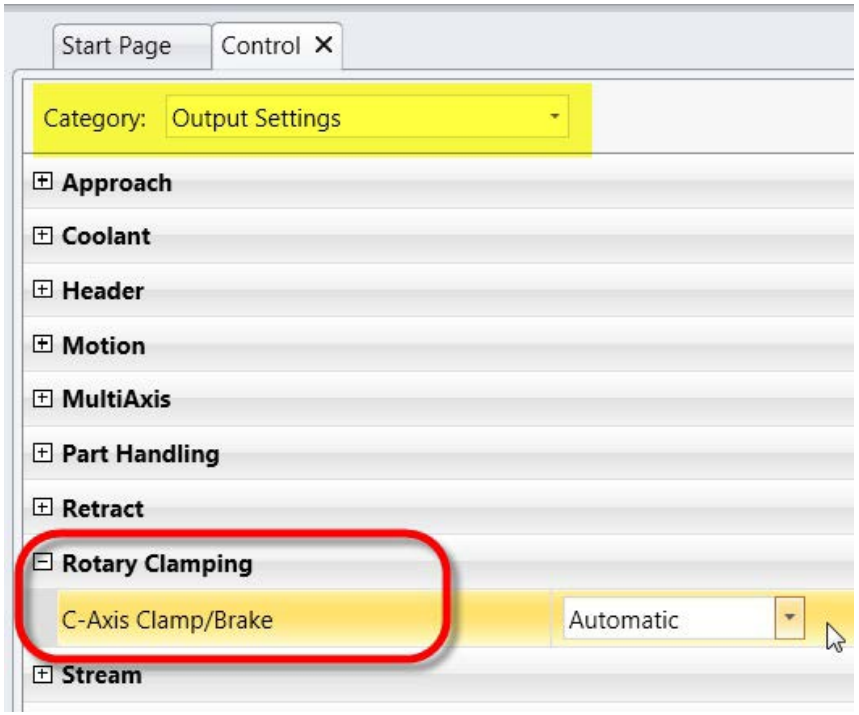
1. Open the Fanuc Lathe **.machine** file in Code Expert.



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

Home

3. Select **Category: Output Settings**.
4. Go to the **Rotary Clamping** section.
5. Select the desired **C-axis Clamp/Brake** mode:
   - **Automatic**—Mastercam will automatically decide whether to clamp the C-axis based on the application and toolpath type.
   - **FullClamp**—Force `M89/M189` output.
   - **Off**—Suppress `M89/M189` output for the selected operation.
6. Save the **.machine** file.



Home

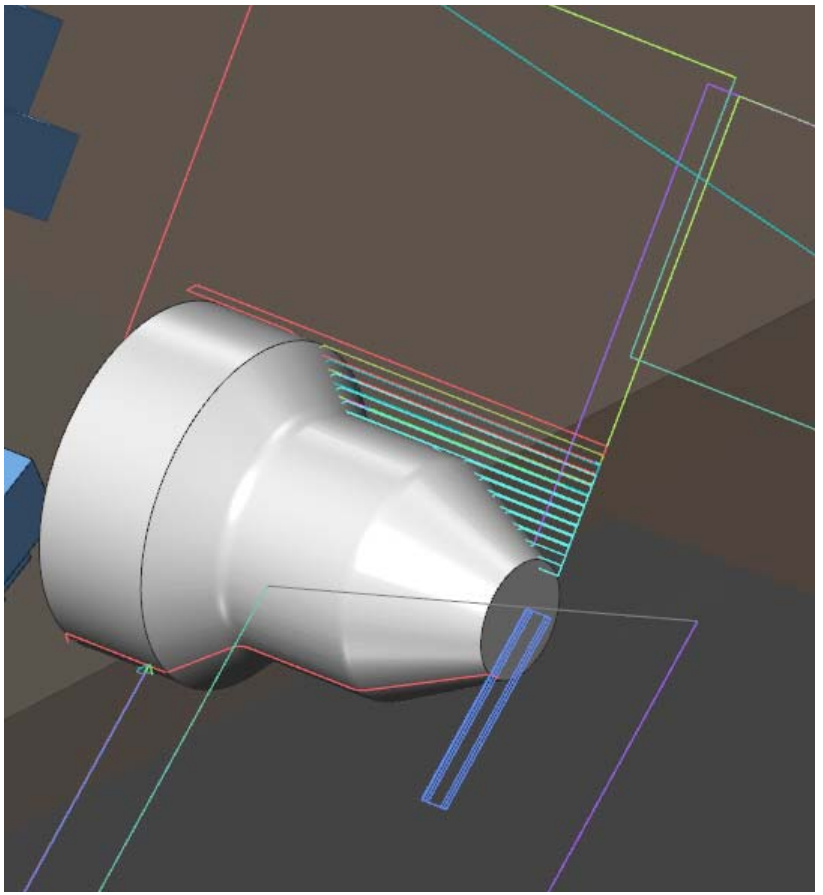
Home

Chapter 3: Working with toolpaths

Your Fanuc Lathe **.machine** file includes a number of settings that let you apply Fanuc Lathe-specific features to the operations that you create in Mastercam.

This chapter includes the following sections:

- **Toolpath reference positions**
  - Setting the start point and end point for an operation
  - Setting the type of approach/retract motion
  - Reference positions and reference points
- **Machining modes and other options**
  - Polar (G12.1) and cylindrical (G7.1) interpolation
  - Using coolant
  - Output expanded Mcode comments



Home

Rev. date:
May 2018

A: Toolpath reference positions

Use Sync Manager controls to determine the start and end points of each operation. 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**
  - **Setting the start point for an operation**
  - **Setting the end point**
  - **Selecting a reference position of None**
  - **Reference positions and null tool changes**
- **Setting the type of approach/retract motion**
- **Creating custom reference positions**
- **Reference positions and reference points**

Home

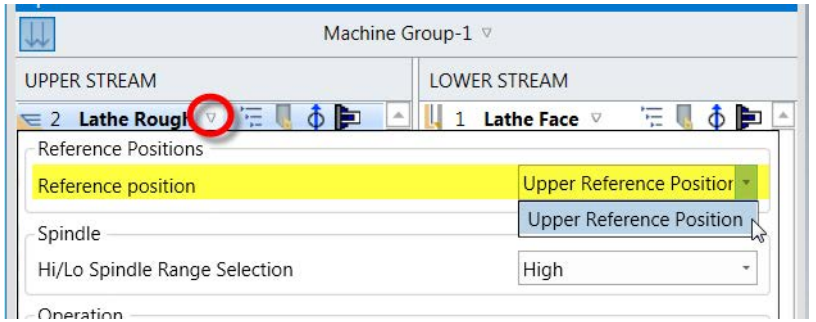
Setting the start point and end point for an operation

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

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

Setting the start point for an operation

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



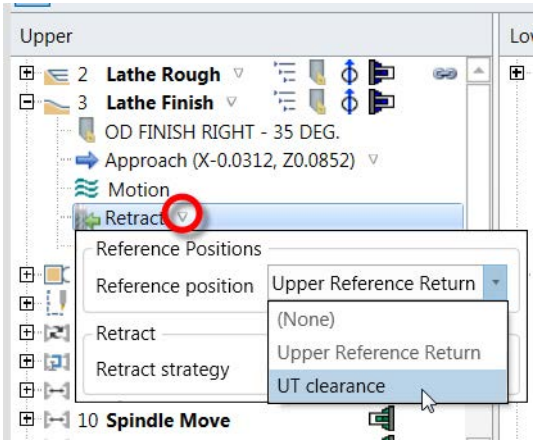
The selected reference position is typically output before the tool change. For example, the highlighted code shows the **Upper Reference Return** reference position from the previous picture

```
18   N5
19   (OPERATION # 1)
20   G28 U0.
21   G28 W0.
22   (T01001   |   ROUGH FACE LEFT - 80 DEG
23   M34
24   G54
25   T01001 ( ROUGH FACE LEFT - 80 DEG. )
26   M262
27   G97 S395 M04 P11
```

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 **Setting the type of approach/retract motion** on 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   G00 Z.25
119   G369
120   G53 U14. V0.
121   G53 W6.
122   G28 B0.
123   T03000
124   M05 P12
125   M01
```

Selecting a reference position of None

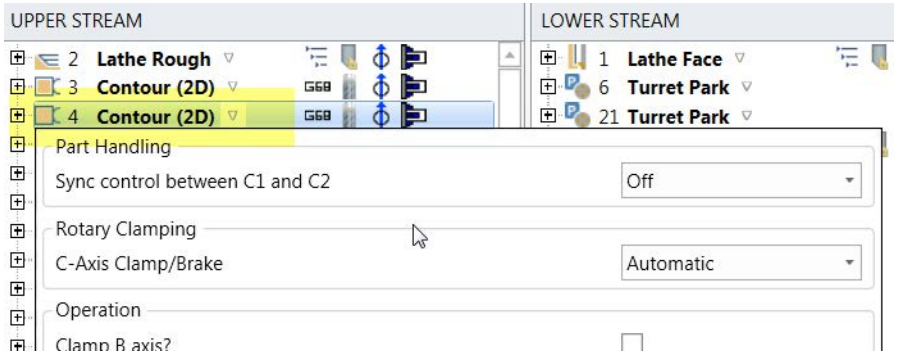
Selecting **None** for a reference position means that there will simply be no output where the reference position is typically output. For example, the code at right shows what happens if **None** is selected as the reference position for operation #2.

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

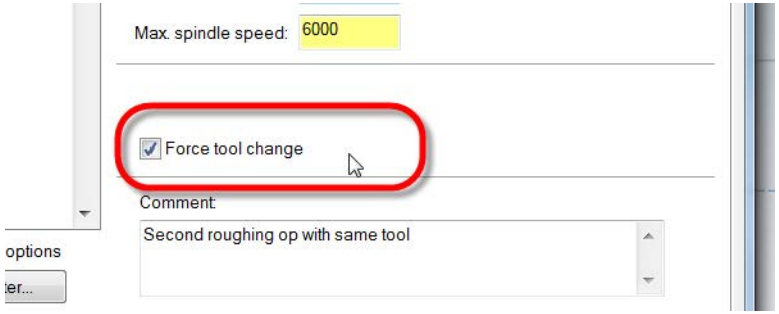
```
81   T02000
82   M05 P11
83   M01
84
85   N2
86   (OPERATION # 3)
87   (T03   |   3/16 FLAT ENDMILL   | DIA. -   0.
88   M35
89   G54
90   M06 T03003 (  3/16 FLAT ENDMILL )
91   T04000
92   G00 G28 W0.
93   G28 B0.
94   M101
```

Reference positions and null tool changes

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



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

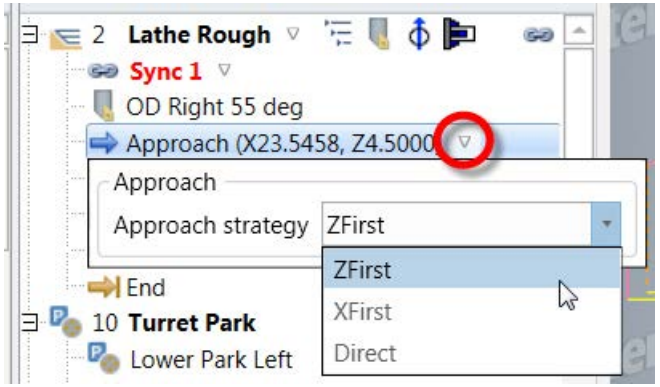


Setting the type of approach/retract motion

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

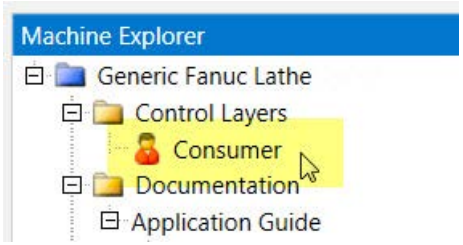
- X-first
- Z-first
- Direct (interpolated) move

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

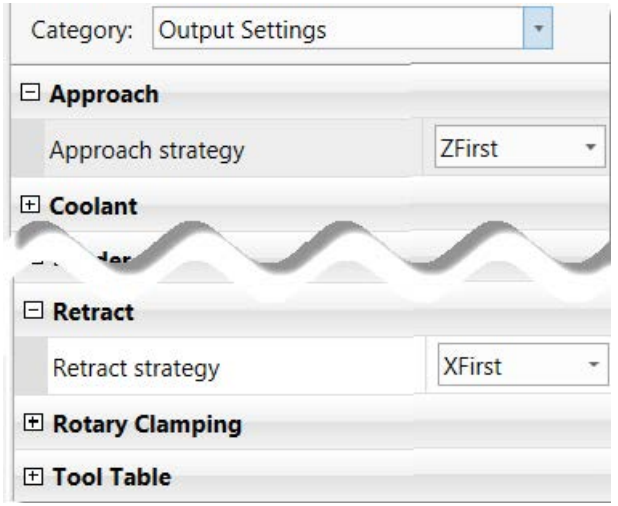


You can also change the default selection in the **.machine** file.

1. Double-click the **Consumer** layer.
2. Go to the **Output Settings** category.



3. Open the **Approach** and **Retract** group.
4. Select the desired strategy for each.
5. Press **Ctrl+S** to save your settings.



Creating custom reference positions

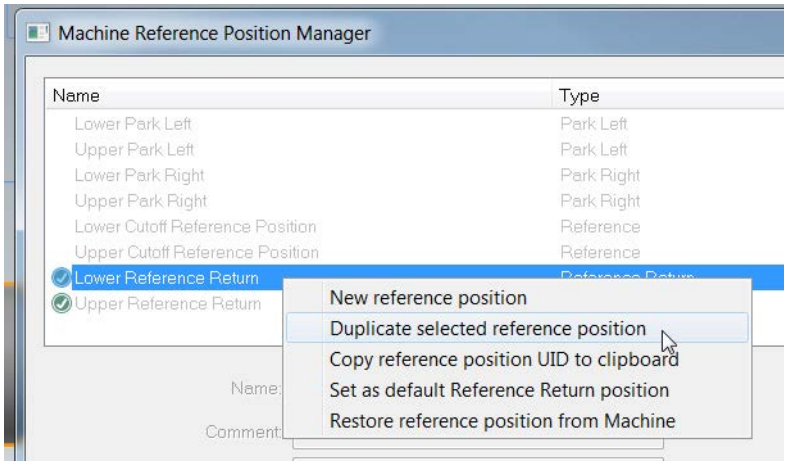
There will be times when you need reference positions that have not been defined in your **.machine** file. These might be necessary to accommodate special part fixturing or tooling, or a part with unusual dimensions. In these cases, you can define your own custom reference positions. These are saved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

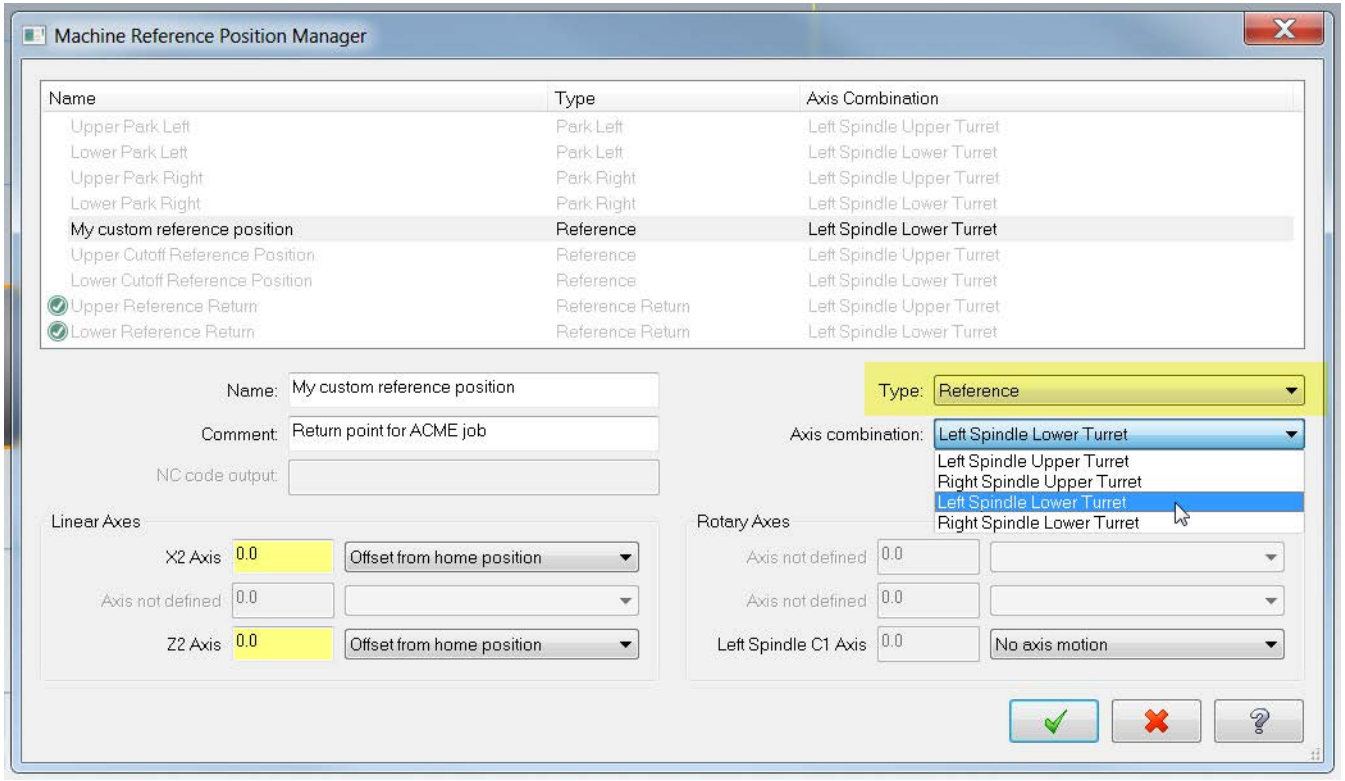
1. Click the button on the Toolpath Manager toolbar.

   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.



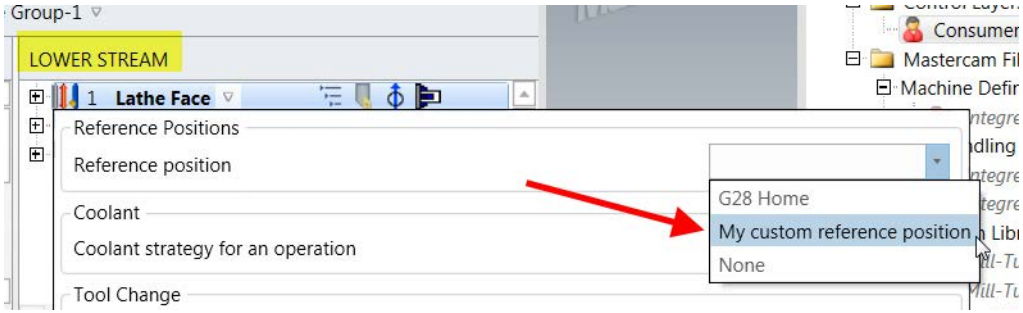
3. Enter a **Name** and edit the other properties.
   - You can see in the list that there are several different types of reference positions; however, you can only create new ones of type **Reference**. You can duplicate a different reference position, but the **Type** will be set to **Reference**.



- 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

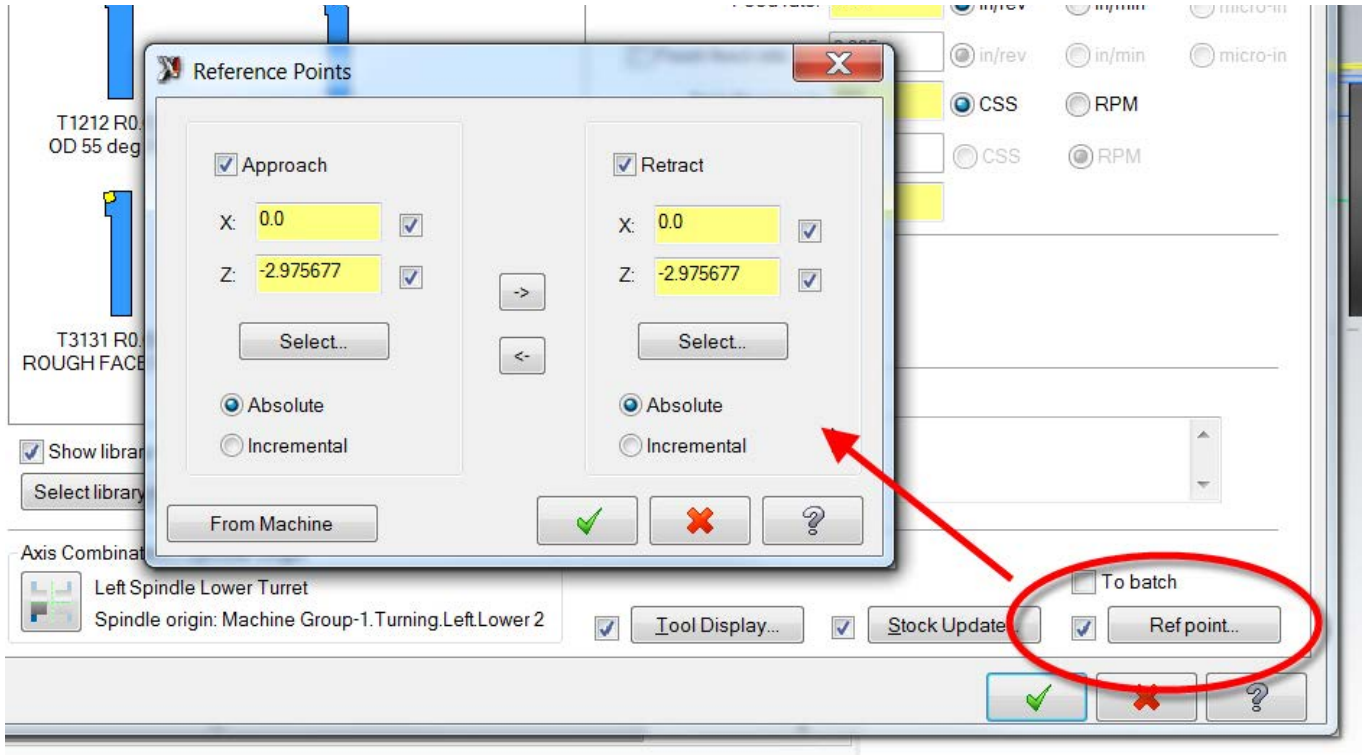
position should be available. Make sure you look in the proper stream.



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, this defines 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 to make sure the tool fully and properly retracts from the ID before moving to the reference position. Do not make the mistake of confusing reference *points* and reference *positions*.



Home

B: Machining modes and other options

Your Fanuc Lathe **.machine** file supports the following milling cycles:

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

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

This section also contains information about how the Fanuc Lathe **.machine** file supports some other options:

- expanded comment output
- coolant

The following sections describe how to set defaults for each mode and how to enable or suppress them for individual operations.

- **Polar (G12.1) and cylindrical (G7.1) interpolation**
- **Using coolant**
- **Output expanded Mcode comments**

Home

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

Your Fanuc Lathe **.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.

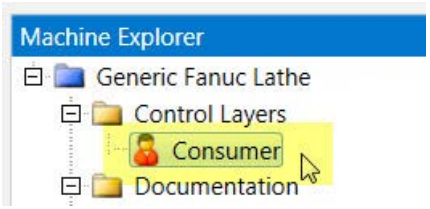
Default settings for G7.1 and G12.1

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

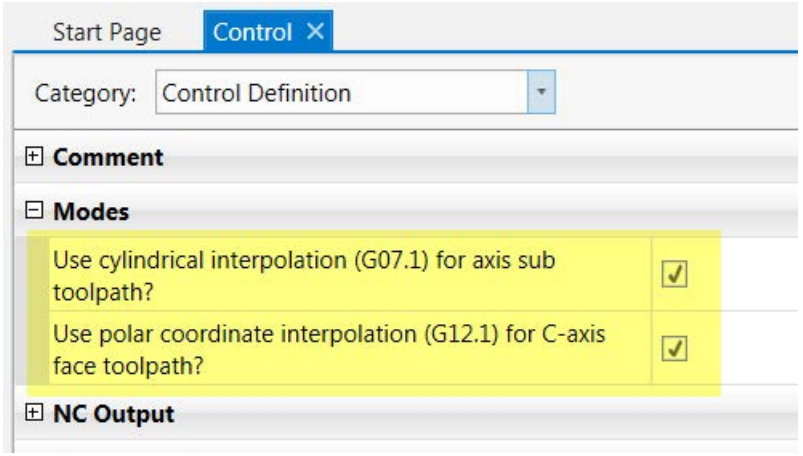
- Choose whether the default output mode will be the G7.1/ G12.1 cycle or longhand output.
- Choose whether the X-axis output will be in diameter or radius values.

Follow these steps:

1. Open the Fanuc Lathe **.machine** file in Code Expert.



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

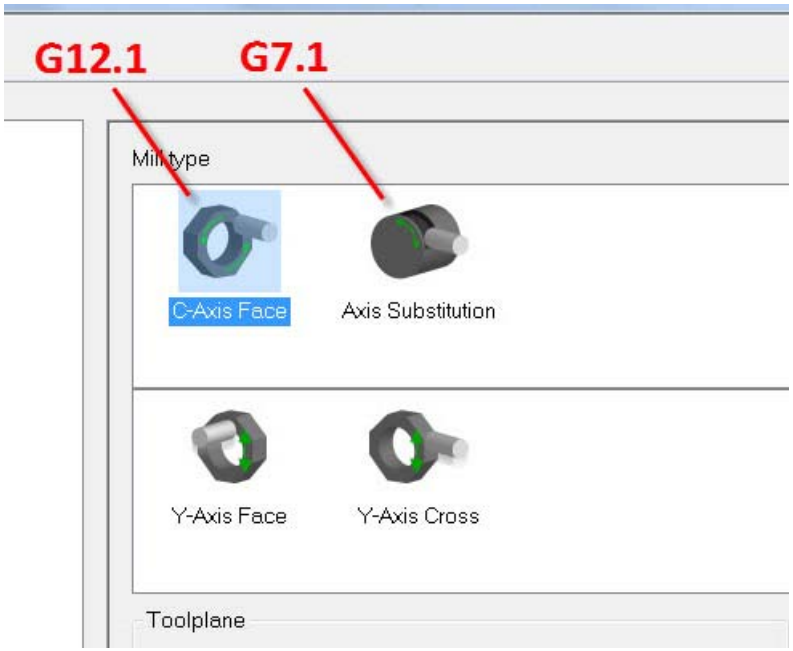


3. Select **Category: Control Definition**.
4. Go to the **Modes** section.
5. For each cycle, choose the default output mode. If you do not select the checkbox, you will get longhand output.
6. Save the **.machine** file when you are done.

Programming G7.1/G12.1 for each operation

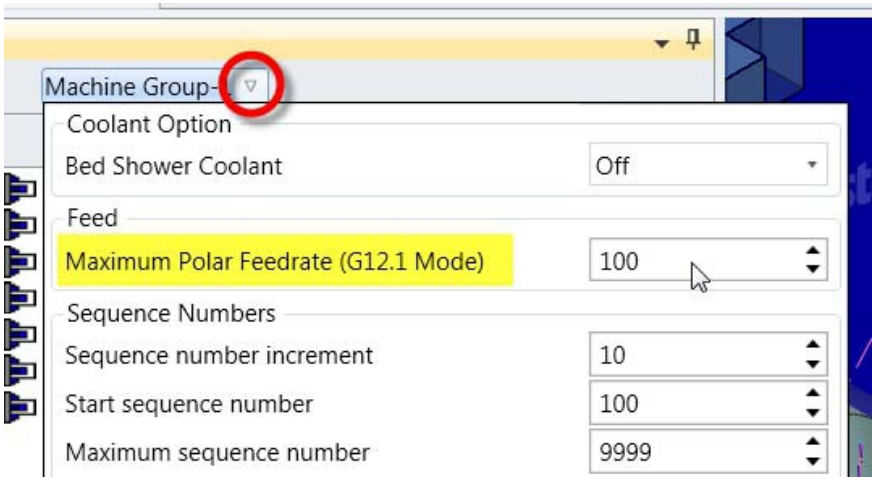
Follow this general workflow for programming these cycles for each operation.

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.



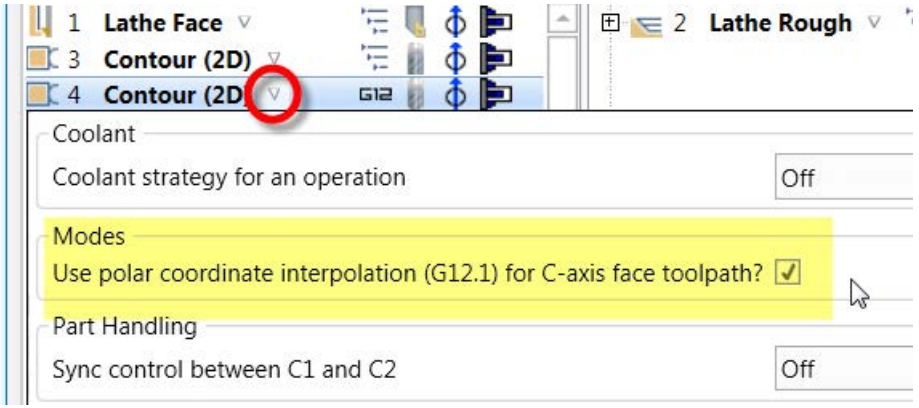
2. For either type of operation, the **Toolplane** is automatically selected. These planes were created for you when you completed the **Job Setup** process. Each plane automatically sets the proper tool orientation for your part.

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



Home

4. For each operation, choose whether or not to output the G7.1/G12.1, or longhand output. Click the little triangle next to the operation and select the cycle option if desires, or leave it unchecked for longhand output.
5. Press **[Ctrl+S]** to save the new setting back to your part file.

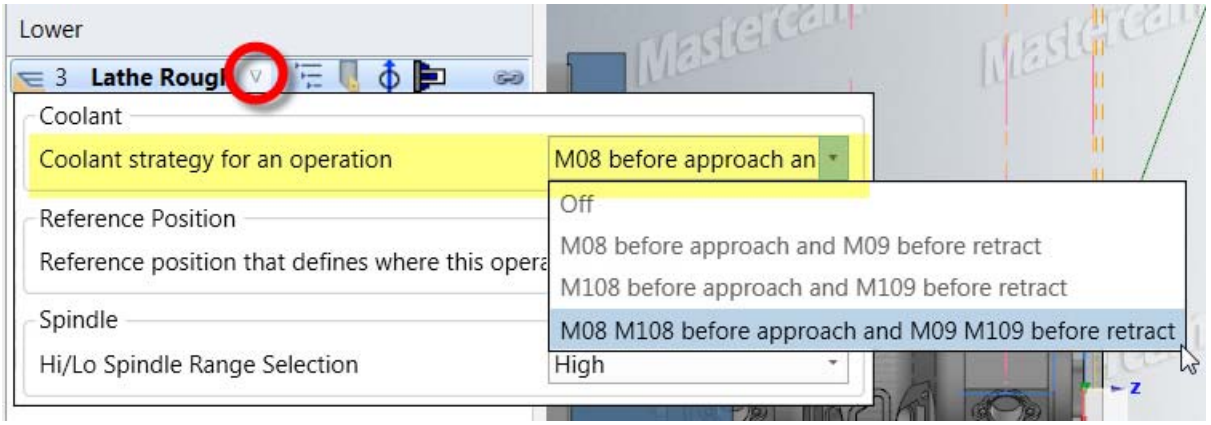


Home

Using coolant

Your Fanuc Lathe **.machine** file supports the following coolant options:

- Flood coolant (`M08`)
- Thru-spindle coolant for the milling spindle (`M108`)
- Bed shower coolant (`M07`)



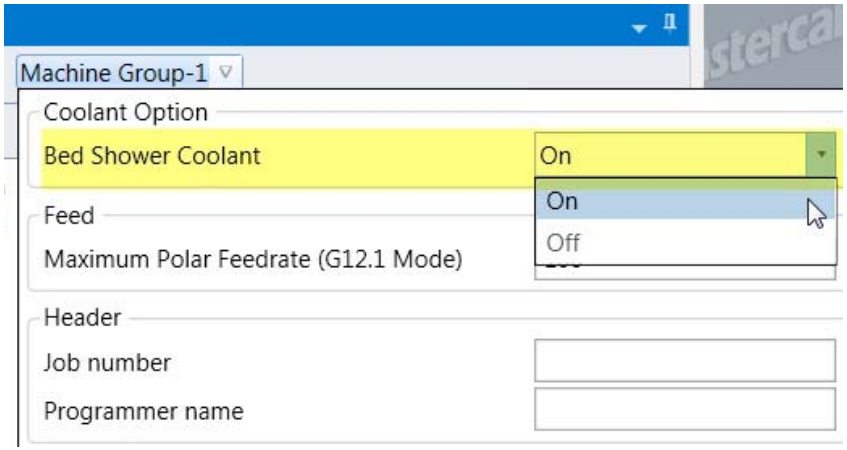
If you are familiar with other Mastercam products, you are used to selecting coolant options inside Mastercam as part of the toolpath parameters. In Mill-Turn, coolant selection is done in the Sync Manager; the **Coolant** button is no longer present inside Mastercam.

For each operation, click the small triangle next to the operation name and select the desired coolant option. Each strategy will automatically turn off the coolant before the retract move. Use the **Off** option only when you want to force the coolant off. It is not necessary to use this option to routinely turn off coolant for each operation.

Home

Bed shower coolant (M07)

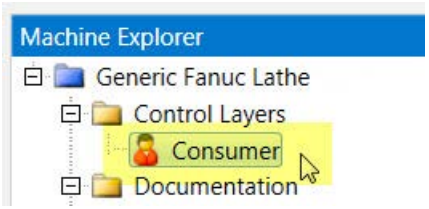
Bed shower coolant (M07) is controlled separately from the other coolant options. It is typically turned on at the beginning of the machining job. Click the small triangle next to the machine group name to turn it on. It is automatically turned off at the end of the stream.



Setting the default coolant option

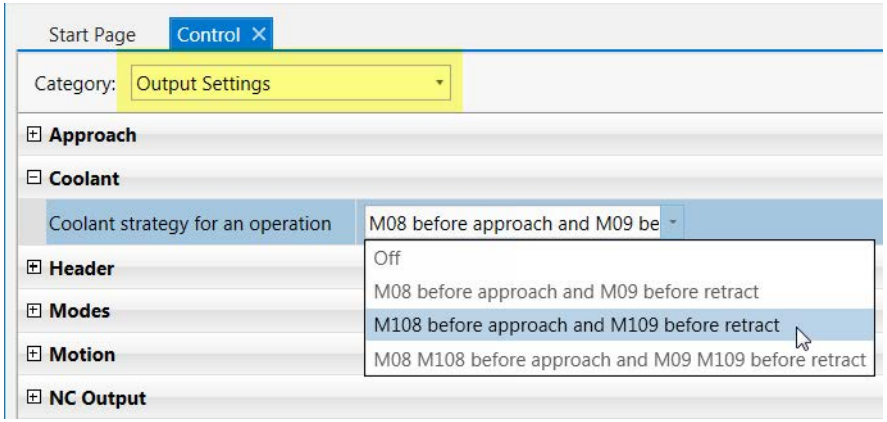
You can select a default coolant strategy and save it in the **.machine** file.

1. Open the Fanuc Lathe **.machine** file in Code Expert.



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

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



Output expanded Mcode comments

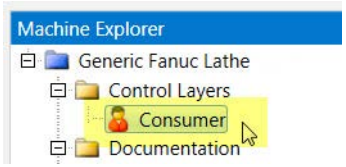
Your **.machine** file includes an option that lets you include expanded comments when certain Mcodes are output:

```
T06006 ( OD ROUGH LEFT - 80 DEG. )
G00 G28 W0.
M161 (LEFT SPINDLE LOW GEAR)
G97 S417 M04 P21
G00 Z-.075
X1.8325
G50 S5000 P21
G96 S200 P21
```

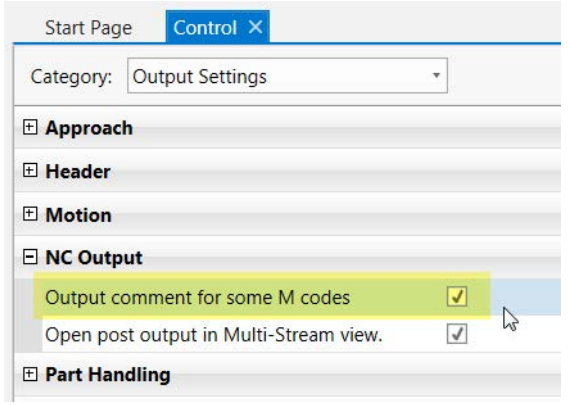
```
T06006 ( OD ROUGH LEFT - 80 DEG. )
G00 G28 W0.
M161
G97 S417 M04 P21
G00 Z-.075
X1.8325
G50 S5000 P21
G96 S200 P21
```

You can disable these extra comments if you wish. Follow these steps:

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



3. Select **Category: Output Settings**.
4. Go to the **NC Output** section.
5. Set the **Output comment for some M codes** option as desired.
6. Press **Ctrl+S** before posting to save your setting.



Home