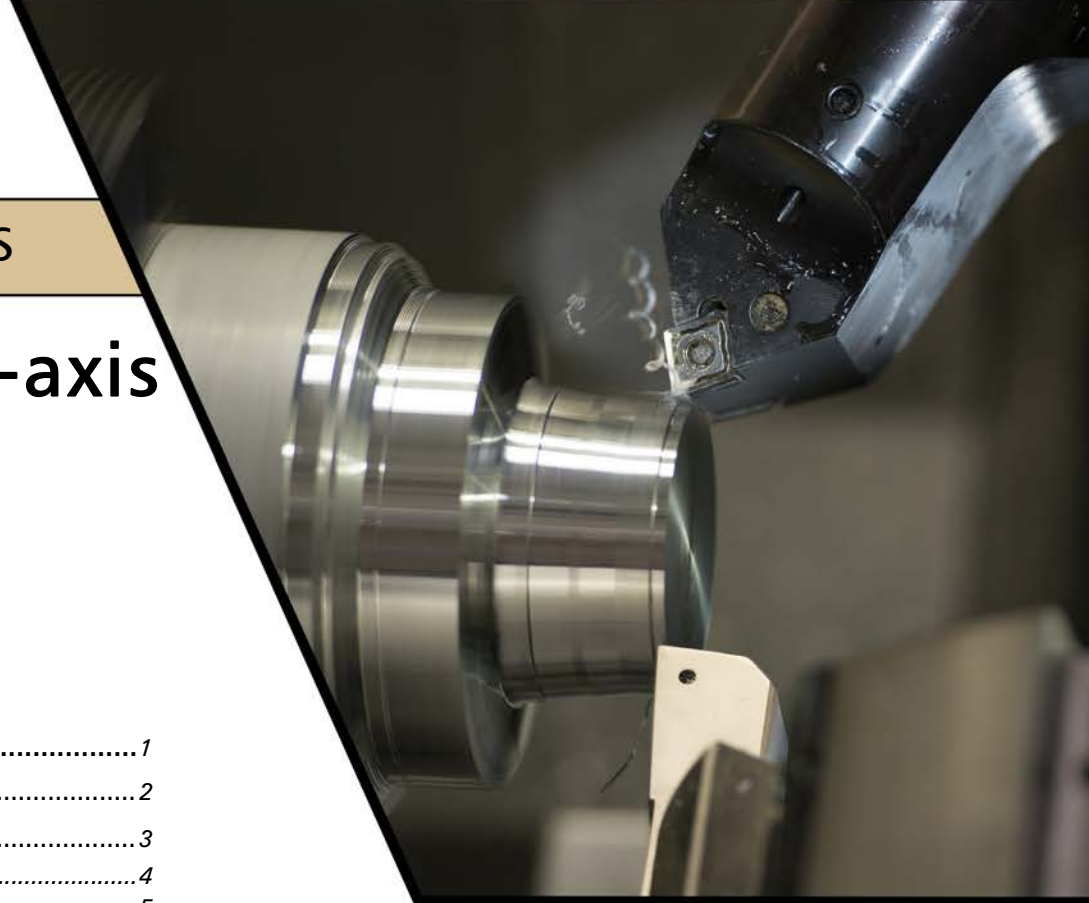


Generic Fanuc Mill-Turn B-axis

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	7
Configuring tool table output.....	9
Setting the toolpath directory and stream names	10
Chapter 2: Working with tools and spindles	13
A: Setting up tools for the B-axis head	14
Initial tool setup.....	15
Selecting the spindle and turret.....	19
Rotating the B-axis head to the proper position.....	20
Using tool locators in the B-axis head.....	22
B: Setting up tools on the lower turret—turret rotates around X	25
Initial tool setup.....	26
Setting up for the left or right spindle.....	28
C: Setting up tools on the lower turret—turret rotates around Z.....	29
Initial tool setup.....	30
Setting up for the left or right spindle.....	32
D: Setting up tools for pinch turning	33

revision date: May 25, 2018



Chapter 3: Working with toolpaths	35
A: Reference positions	36
Setting the start point and end point for an operation.....	37
Setting the type of approach/retract motion.....	39
Creating custom reference positions.....	41
Reference positions and reference points	44
B: Mill machining modes	45
Polar (G12.1) and cylindrical (G7.1) interpolation.....	46
G68.5 coordinate conversion cycle.....	49
C-axis clamping and braking.....	52
Working with coolant.....	54
C: Multiaxis toolpath settings	56
Rotary start position.....	56
Pole handling.....	58
G43.4 (tool tip point control).....	59
Default settings for multiaxis toolpaths.....	60

Chapter I: 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.

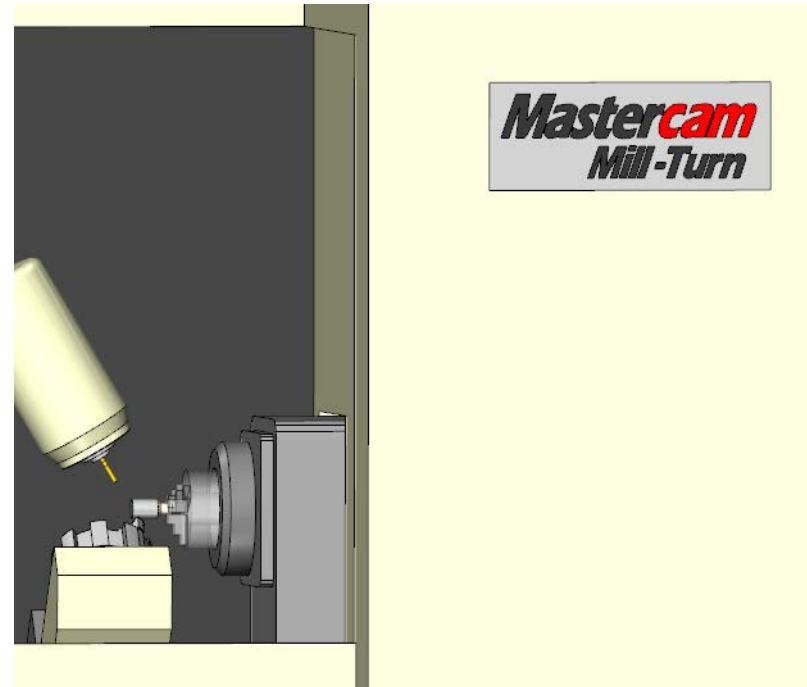
- ❖ **Installing your .machine file**
- ❖ **Default values for .machine file settings**
- ❖ **Configuring the Code Expert editor**
- ❖ **Customizing operation defaults and tool libraries**
- ❖ **Configuring tool table output**
- ❖ **Setting the toolpath directory and stream names**

The Fanuc B-axis **.machine** file simulates a typical Fanuc control on a mill-turn machine that is equipped with both a B-axis head and lower turret. It is intended for:

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

This guide supports both the **Generic Fanuc Mill Turn.machine** and the **Generic Fanuc Mill Turn LTZ.machine**. The difference between the two machines is working with lower-turret tools. The **Working with tools and spindles** chapter has separate sections for each machine.

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



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

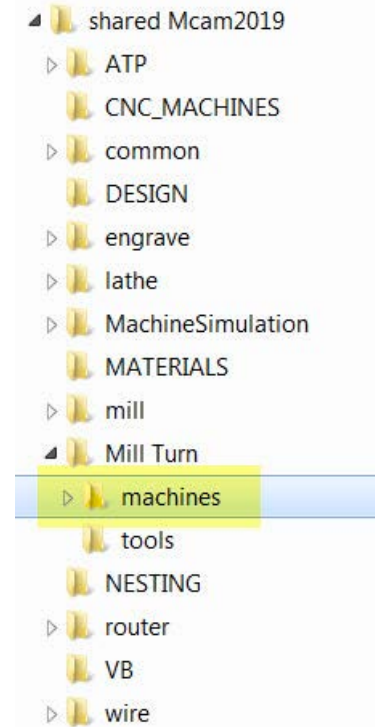
\shared Mcam2019\Mill Turn\MACHINES

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

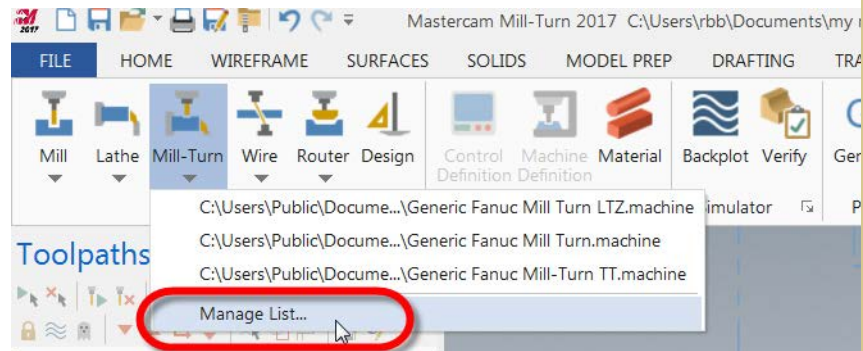
Unlike other Mastercam products, the single **.machine** file includes all the resources that you need to support your Fanuc application. You do not need to worry about linking the **.machine** file to other files, like posts.

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.



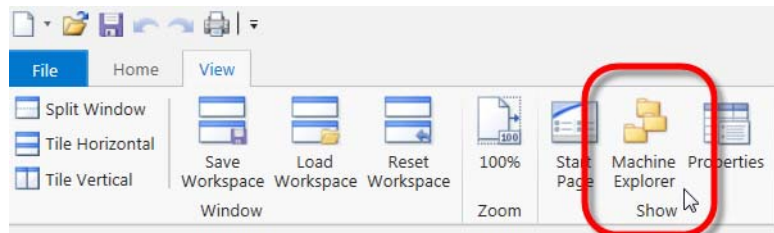
Adding the .machine file to the menu—To make your **.machine** file available on the machine list in Mastercam, select **Machine > Mill-Turn > Manage list**. Then navigate to the folder with your **.machine** file and select it.



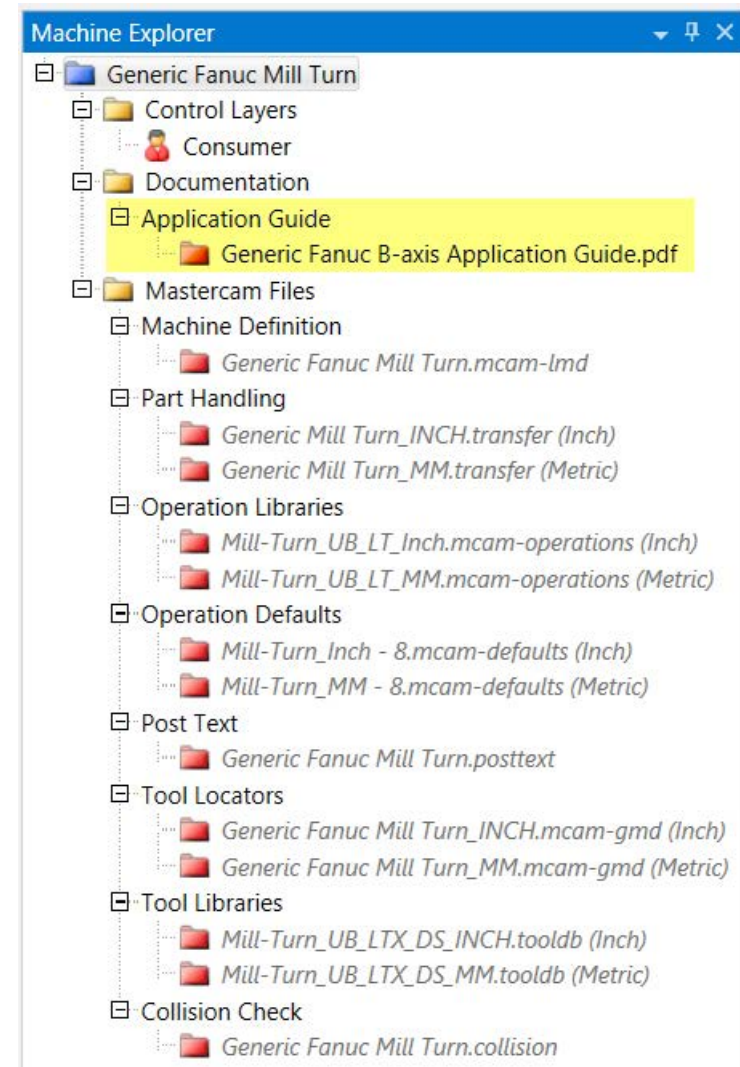
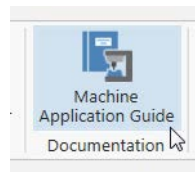
B: Customizing your .machine file

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

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



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



Default values for .machine file settings

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

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

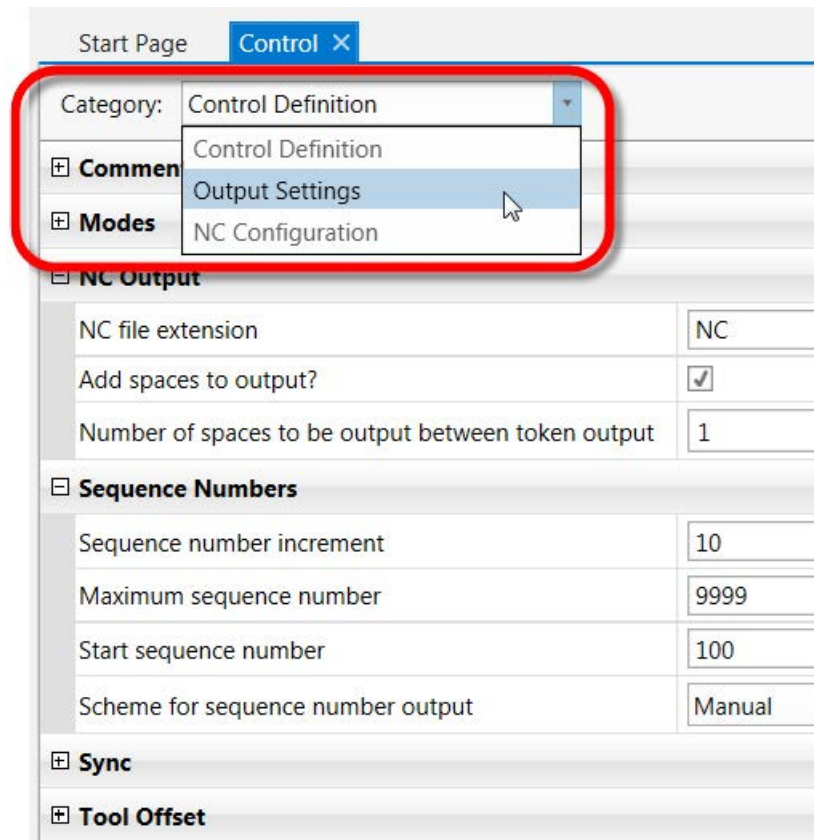
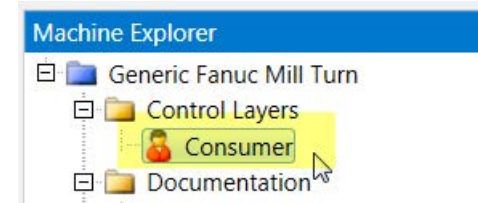
The settings are grouped into several categories. Click the **+** signs to see the individual options. Some of these options (for example, sequence number settings) are very common or generic to most controls; others are specific to your individual machine.

These settings serve a wide variety of functions:

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

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

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

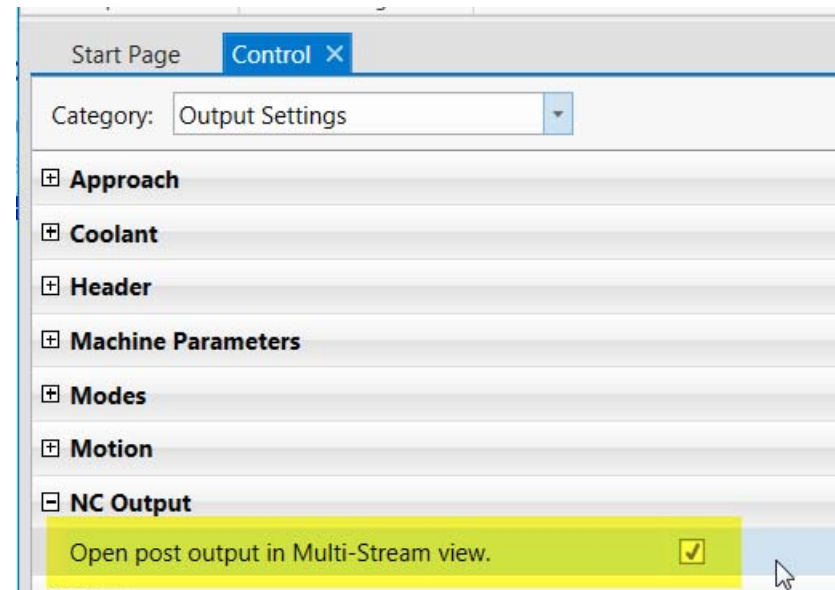
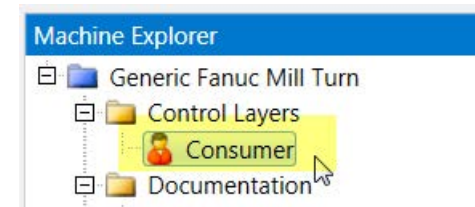


Configuring the Code Expert editor

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

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

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



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

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

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

Start Page Control X

Category: NC Configuration

☒ **Program**

A semicolon separated string to specify address characters for "Find Extents"

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



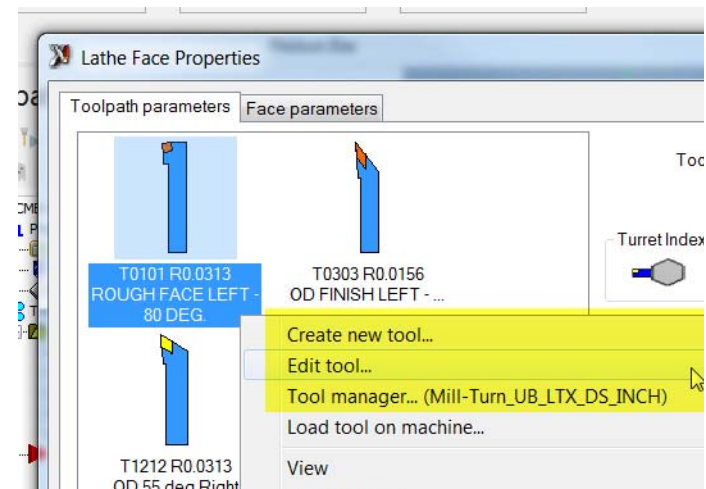
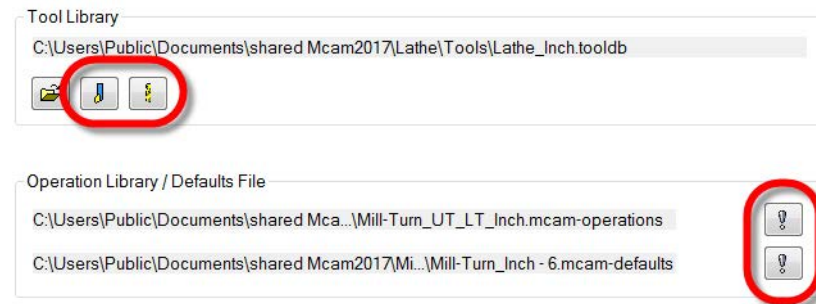
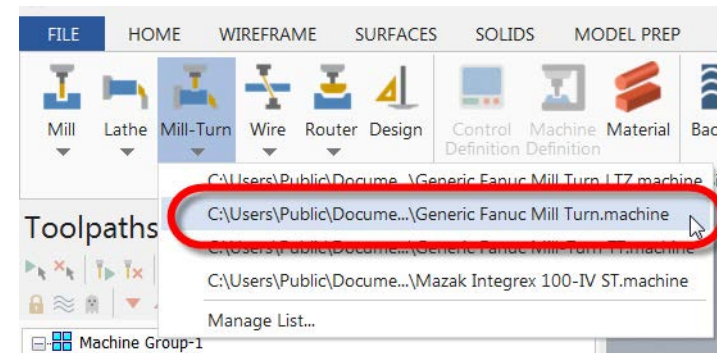
Customizing operation defaults and tool libraries

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.

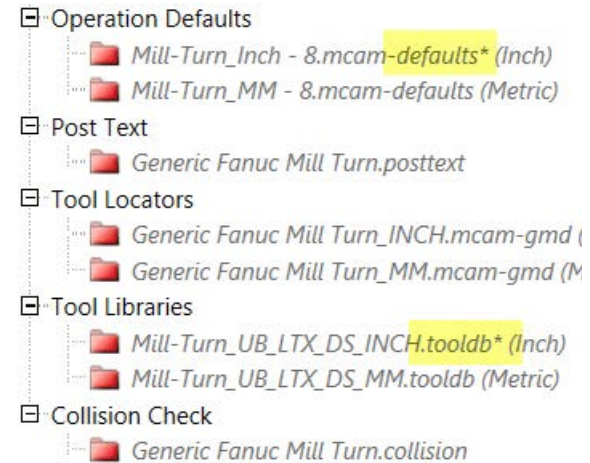
1. Start up Mastercam.
2. Load the desired .machine file or a part that uses the .machine file.
3. Use Mastercam's regular tools for editing these libraries:

- 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.



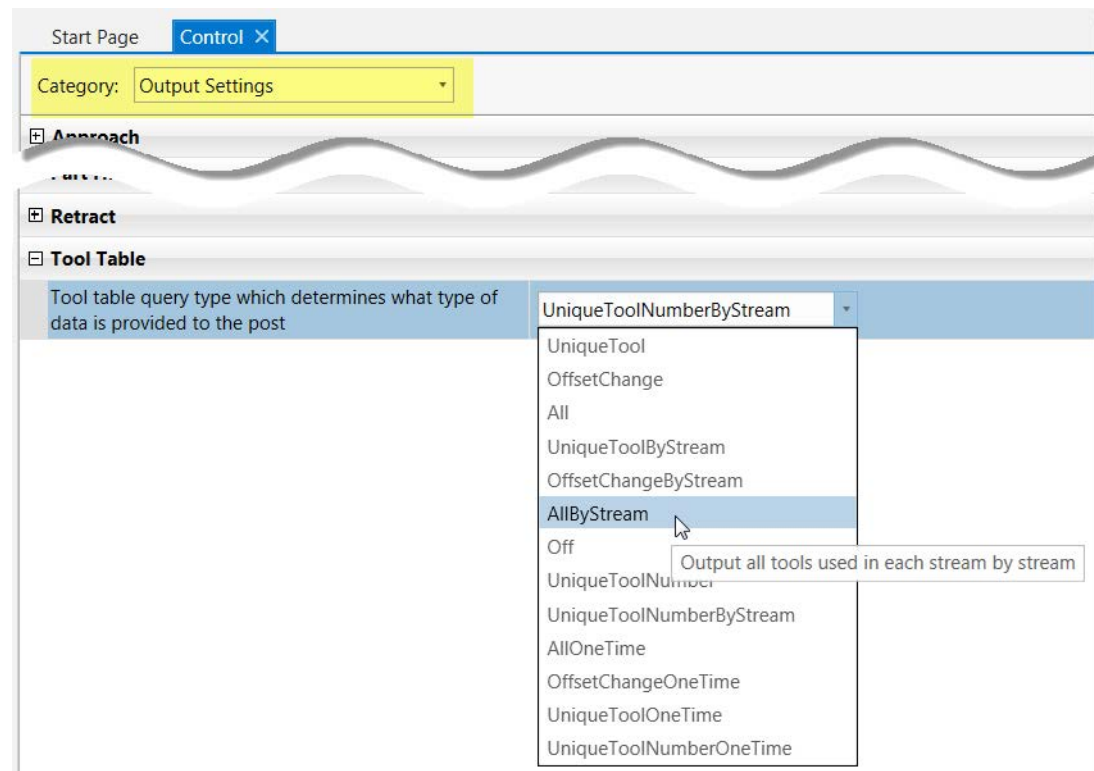
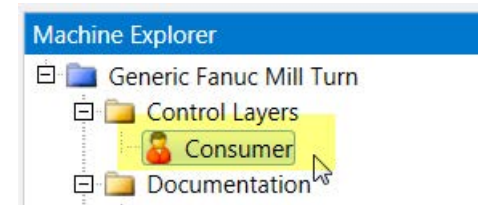
4. Save the changes in the locations that Mastercam prompts you with.
5. When you finish making changes, go back to Code Expert and look at the Machine Explorer. Any files that you changed should be marked “dirty” with an asterisk.
6. Press **[Ctrl+S]** in Code Expert to save the **.machine** file with your changes.



Configuring tool table output

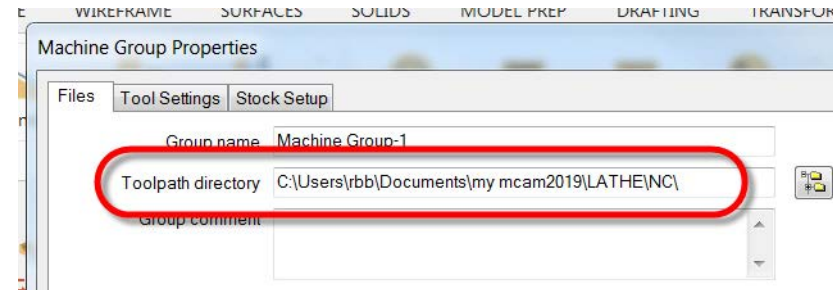
As a user, you can configure your tool table output yourself without needing to do any programming or edits to your post. Your **.machine** file includes 12 different tool table strategies; simply select the desired one. Follow these steps.

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.



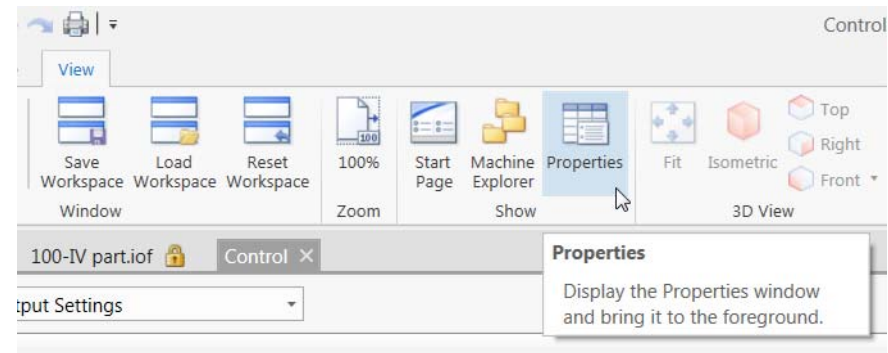
Setting the toolpath directory and stream names

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

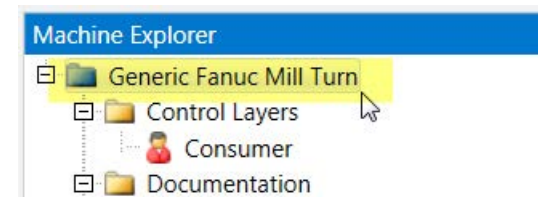


In Mill-Turn, this setting is not used. Instead, it is a property of the **.machine** file. To set it, follow these steps:

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



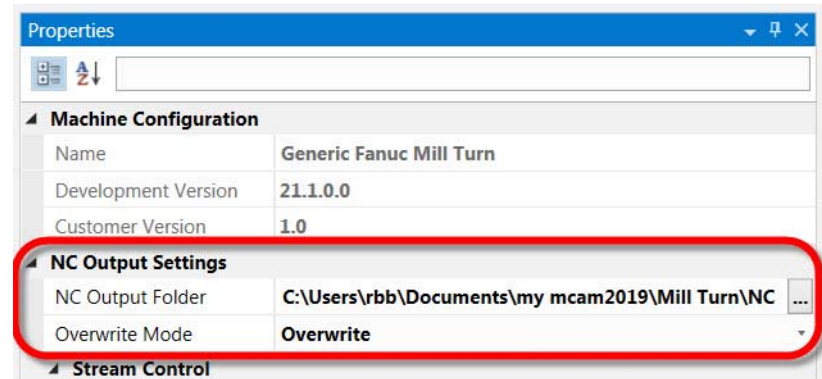
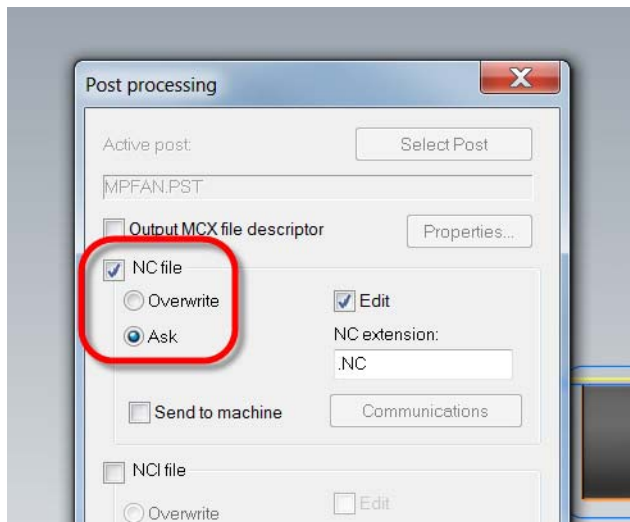
3. Click the machine name in the **Machine Explorer**.





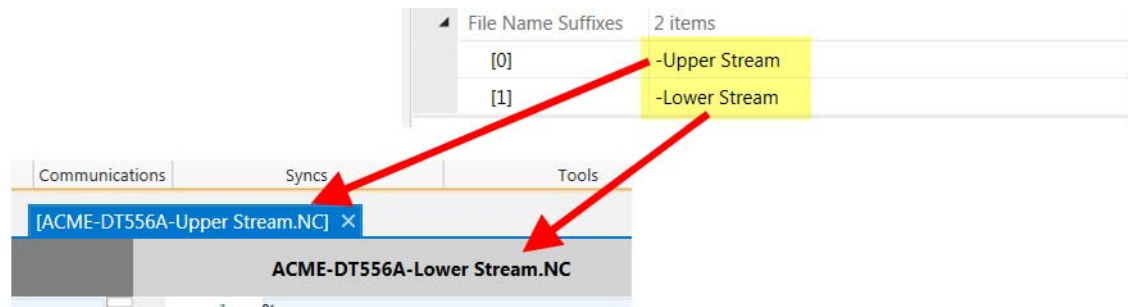
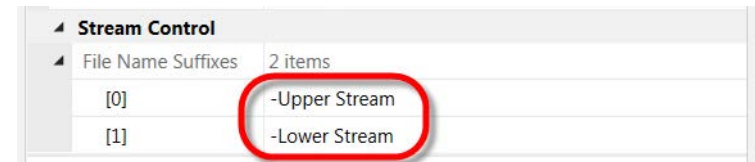
4. Select the desired **NC Output Folder**. Mastercam will write your NC files for this machine to this folder.
5. You can also choose whether Mastercam will automatically overwrite NC files with the same name, or prompt you to enter a different name. Select the desired **Overwrite Mode** to control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



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



Chapter 2: Working with tools and spindles

Tool change output in your NC file is conditioned by:

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

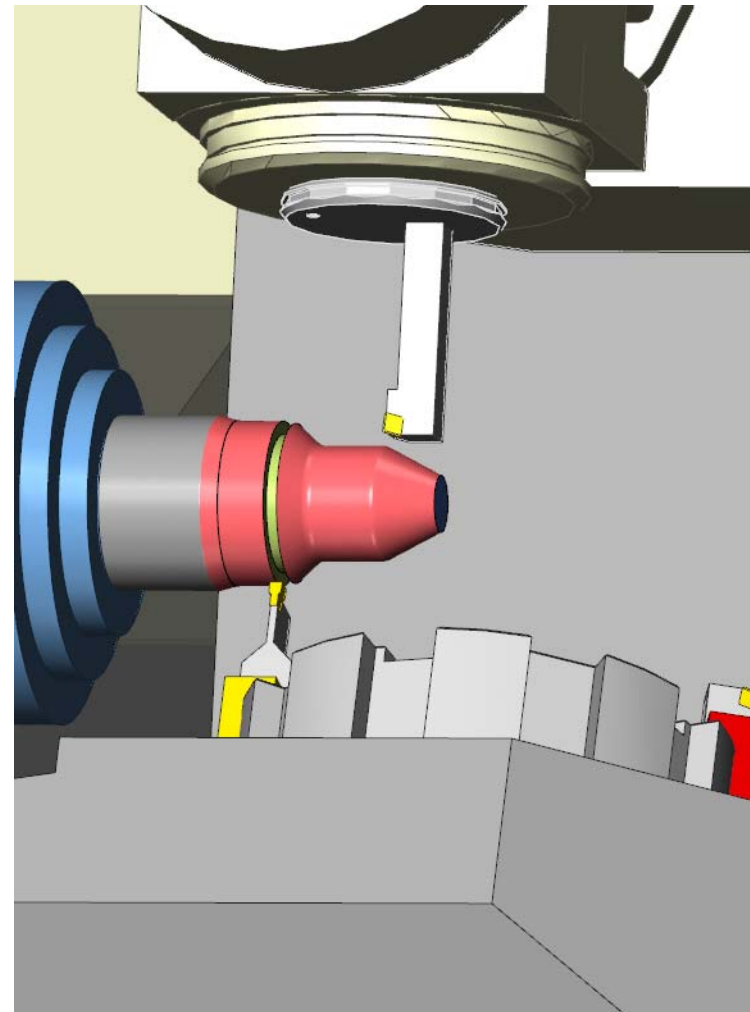
This chapter explains how these different settings work together to produce the proper NC output for your Fanuc machine.

- ❖ **Setting up tools for the B-axis head**
- ❖ **Setting up tools on the lower turret—turret rotates around X**
- ❖ **Setting up tools on the lower turret—turret rotates around Z**

Note that Mastercam Mill-Turn includes two different types of generic Fanuc B-axis machines. The difference is the orientation of the lower turret. Each is described in a separate section as noted above.

Finally, another section discusses how to coordinate tools when performing pinch turning or balanced turning operations:

- ❖ **Setting up tools for pinch turning**



A: Setting up tools for the B-axis head

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.

Your Fanuc B-axis **.machine** file also supports the use of tool locators for tools that are used in the B-axis head. See the following topic to learn more:

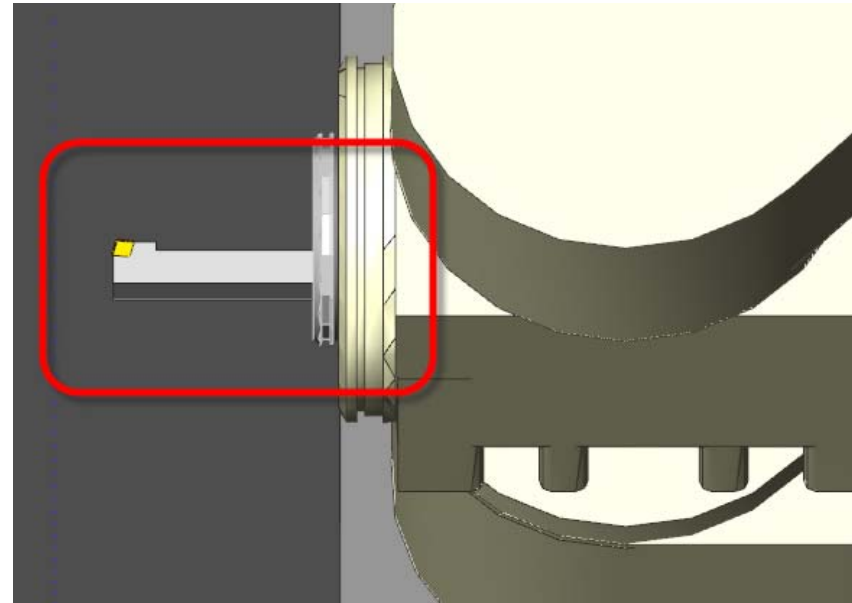
- ❖ **Using tool locators in the B-axis head**



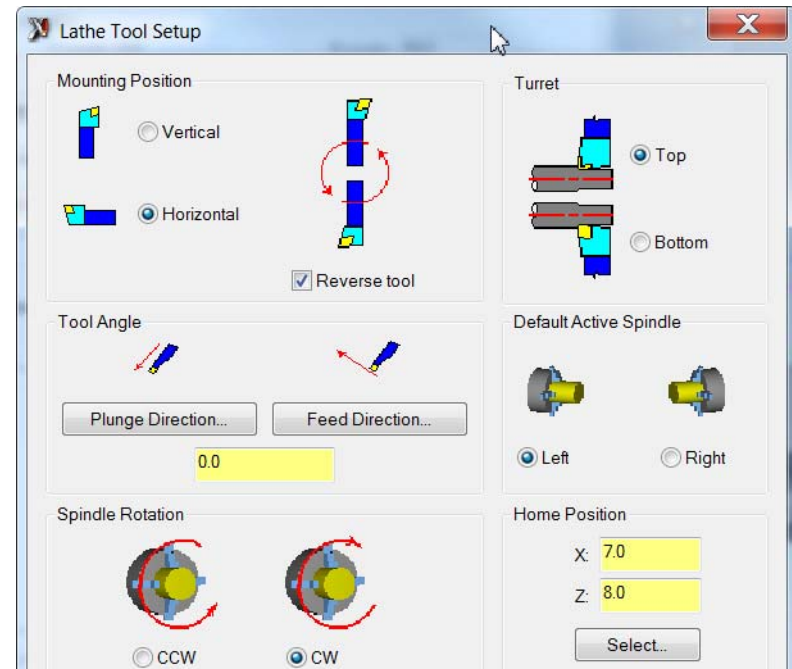
Initial tool setup

For tools that will be used in the B-axis head, 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 Fanuc, this means the tool axis is horizontal, pointed toward the left spindle as shown in the picture.

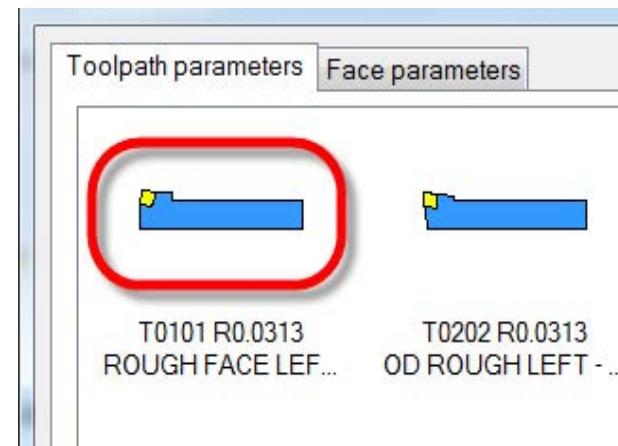


These settings are stored in the Mastercam tool definition. This is what the settings should look like in Mastercam, assuming that the tool is mounted insert-up at the tool change as shown in the above picture:.



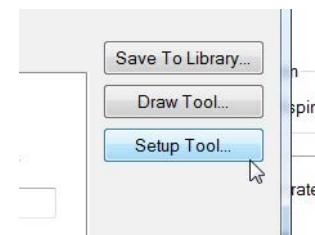
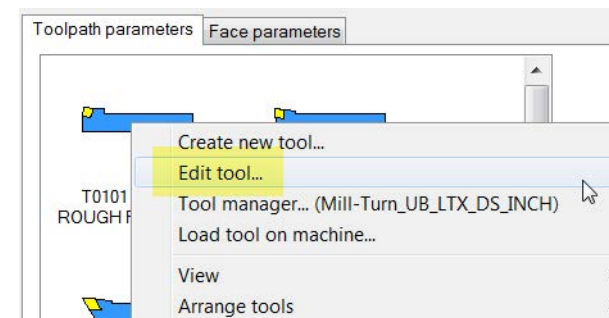
When you select the tool in Mastercam, the picture in the tool selection window should be oriented like the picture.

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 20 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:

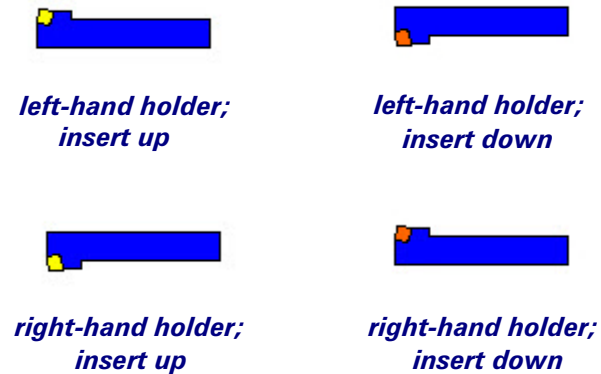
1. Right-click the picture of the tool in the tool selection window.
2. Choose **Edit tool**.
3. Click **Setup Tool**.



Insert up/down and LH/RH holders

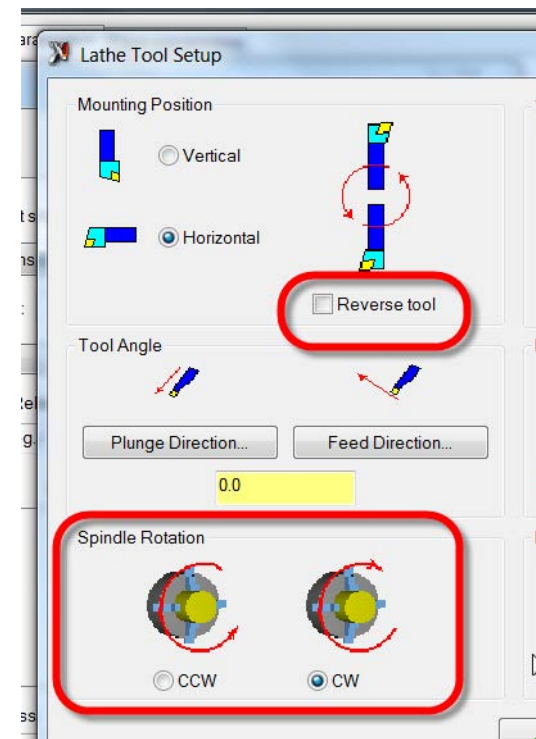
When the tool is mounted at the tool change position, it can still be in any of four different orientations, depending on whether it is mounted insert up or insert down, and whether you are using a left-hand or right-hand holder. Any of these is acceptable with Mill-Turn; the proper choice depends on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



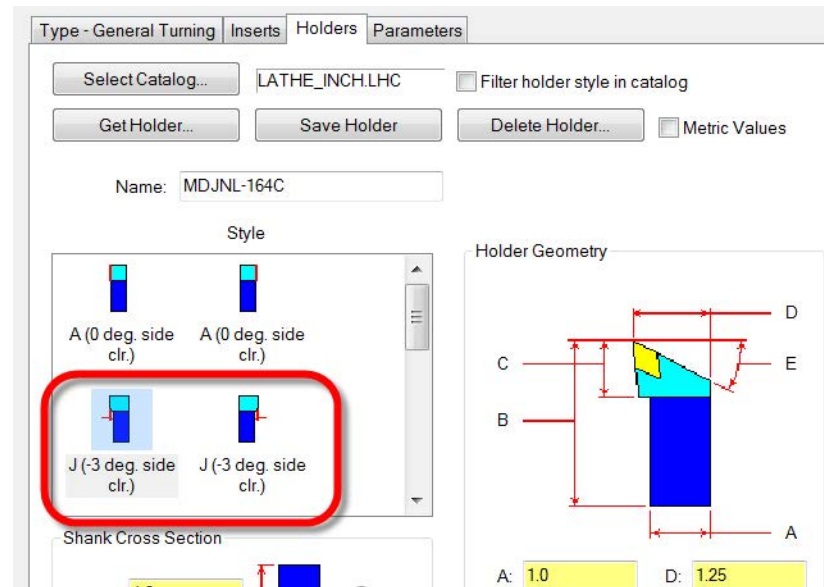
To change the tool from insert-up to insert-down:

1. Go to the **Tool Setup** dialog for the tool.
2. Change the **Spindle Rotation** direction.
3. Toggle the **Reverse tool** setting.



To switch between left- and right-hand holders:

1. 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. Toggle the **Reverse tool** setting.



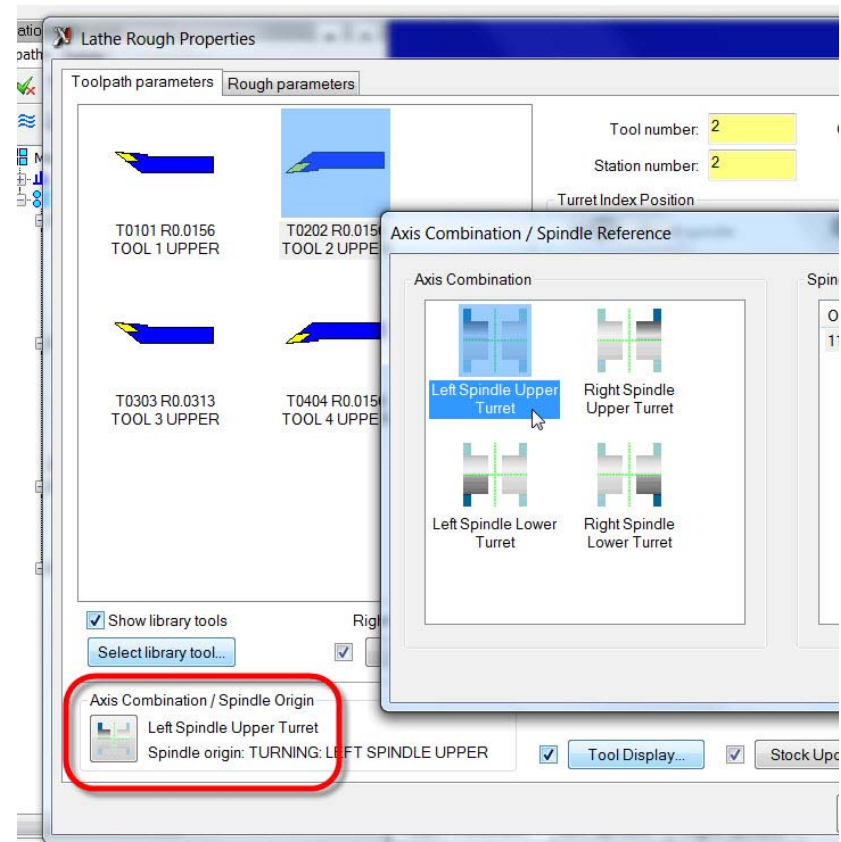
Selecting the spindle and turret

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. The following example shows an ST series machine so you can see how multiple axis combos work.

Follow these steps:

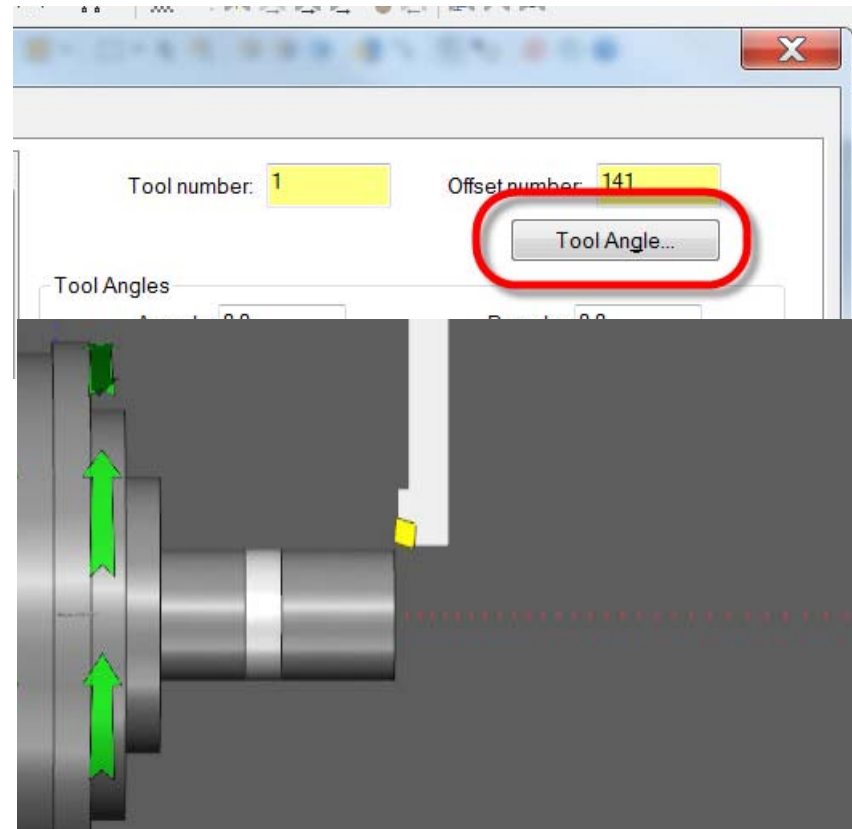
1. Click the **Axis Combination** button.
2. Select the desired turret/spindle combination. This picture shows the axis combination for using the B-axis head on the left spindle.
3. Select the desired tool. Make sure the picture shows it oriented properly for the tool change position for the left spindle.



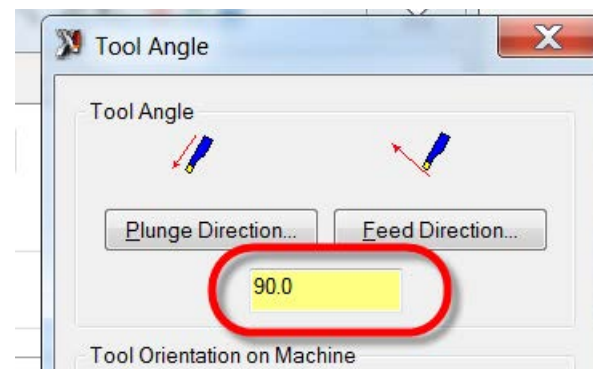
Rotating the B-axis head to the proper position

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:



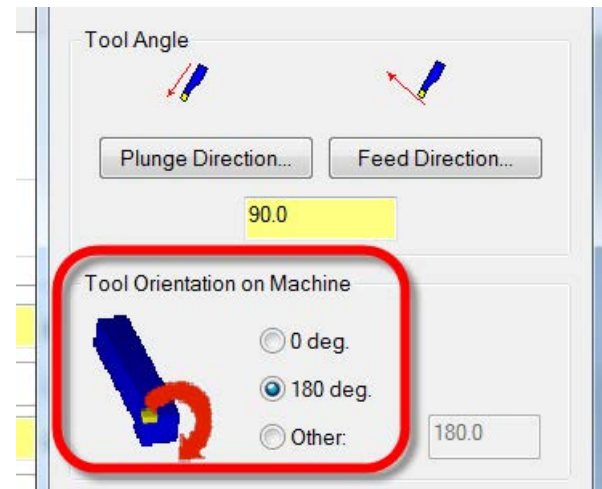
you would enter a **Tool Angle** of 90



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

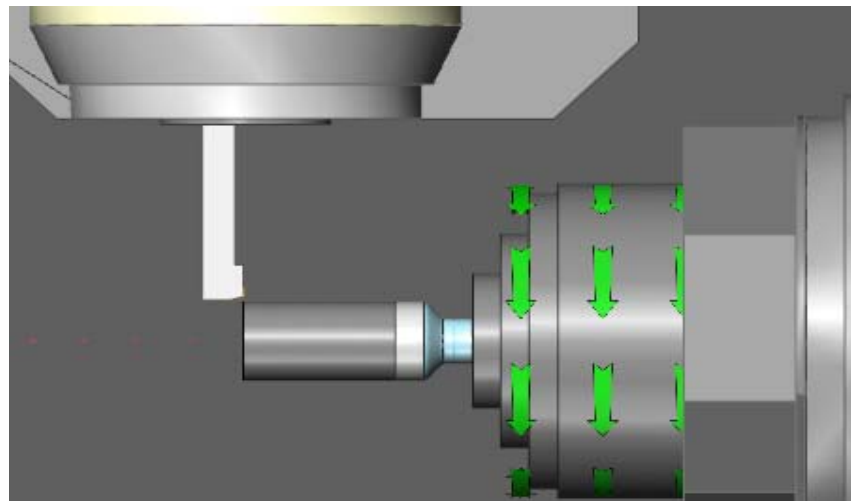
```
(T001 | ROUGH FACE LEFT - 80 DEG.  
M34  
T001 T002 M6  
M107  
G0 B90  
M106
```

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:

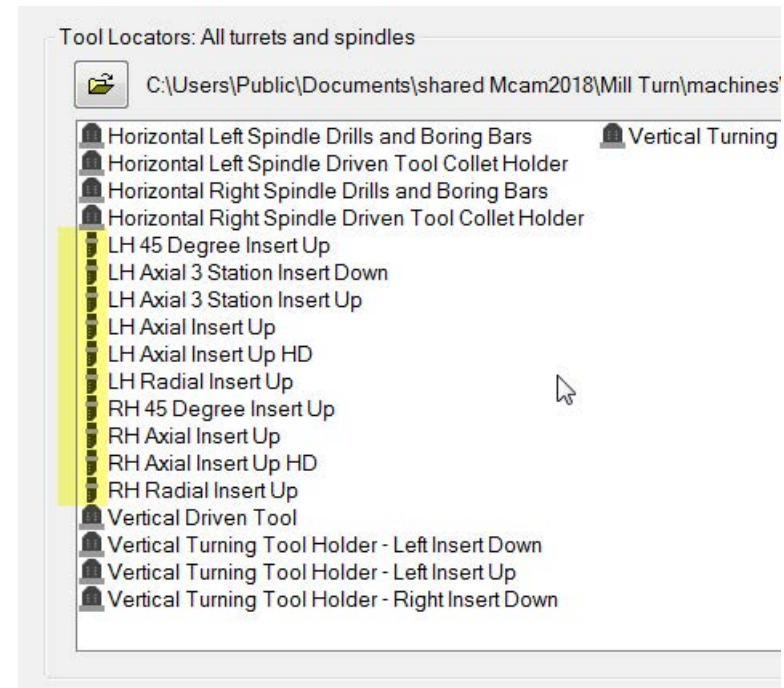
This lets you use the **Tool orientation** setting to orient the tool for cutting on either spindle. Compared to the previous picture, you can see that the insert is now facing towards the right spindle.



Using tool locators in the B-axis head

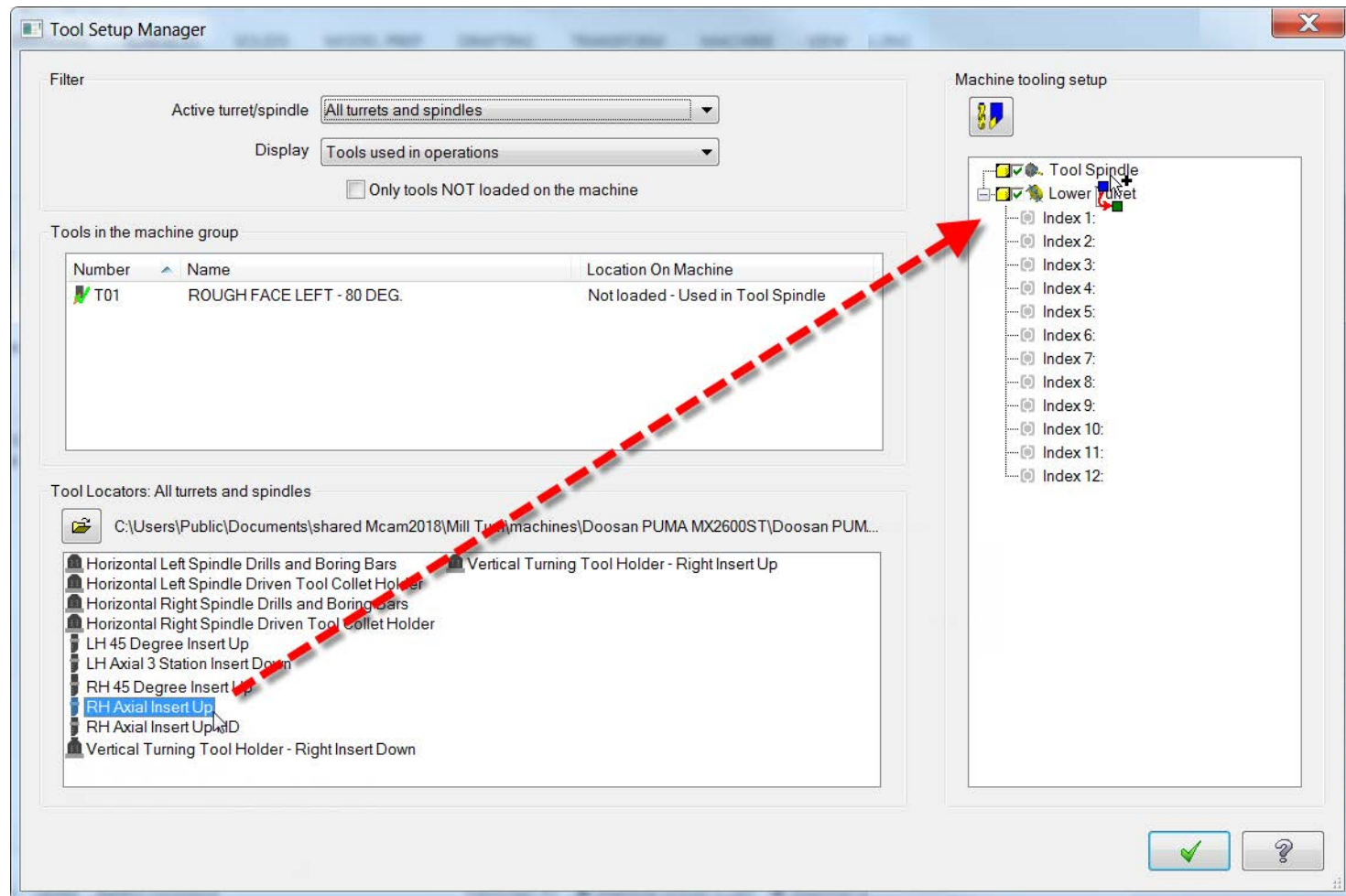
Mastercam 2018 introduced support for tool locators in tool spindle components. When you use a lathe or “stick” tool in a tool spindle, Mastercam 2017 (and earlier versions) mount the tool directly in the tool spindle component and assume that the tip of the insert is on the spindle center line. Using a tool locator lets you model a holder block so that the tool is positioned more realistically and simulated more precisely.

When you are using the **Tool Setup Manager**, you can identify spindle tool locators by looking at the icon in the tool locator window. Tool locators for turret use and spindle use have different icons. The highlighted tool locators are for tool spindle use.



It is important for you to understand that—unlike tool locators for turrets—Mastercam does not automatically use tool locators when loading tools in a tool spindle or B-axis head. The default

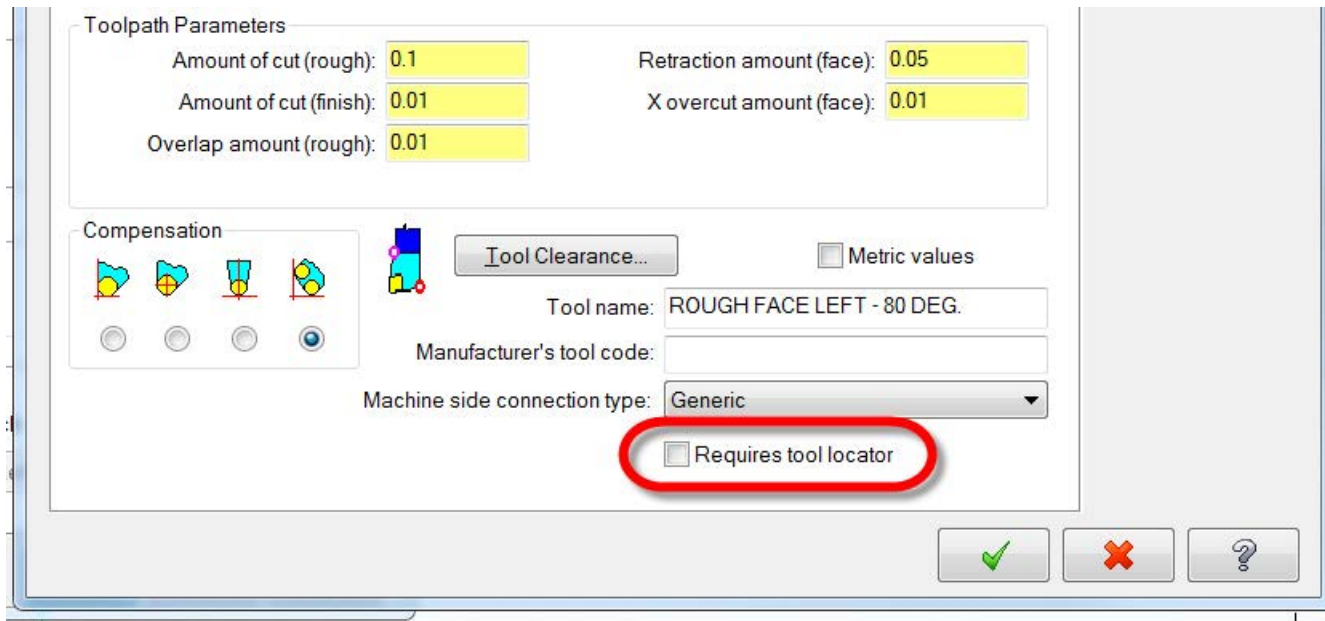
behavior is to mount the tool directly in the spindle, even if a tool locator is available. You need to use the **Tool Setup Manager** to manually add the desired tool locator to the tool spindle component.



You can choose to change this behavior so that Mastercam will automatically look for a tool locator when loading the tool in the spindle.



This is done on a tool-by-tool basis. If the following option is selected in the tool definition, Mastercam will automatically search for a tool locator when loading the tool in a tool spindle:



- If an appropriate tool locator is not found, Mastercam will still load the tool, but it will be loaded directly in the spindle.
- This option has no effect at all on turret use—only tool spindle or B-axis head use.

This option is turned OFF by default for all tool libraries. You need to explicitly enable it for the tools that you choose.

After selecting this option, save the tool definition back to the tool library used in the **.machine** file. See “Customizing operation defaults and tool libraries” on page 7 to learn more.

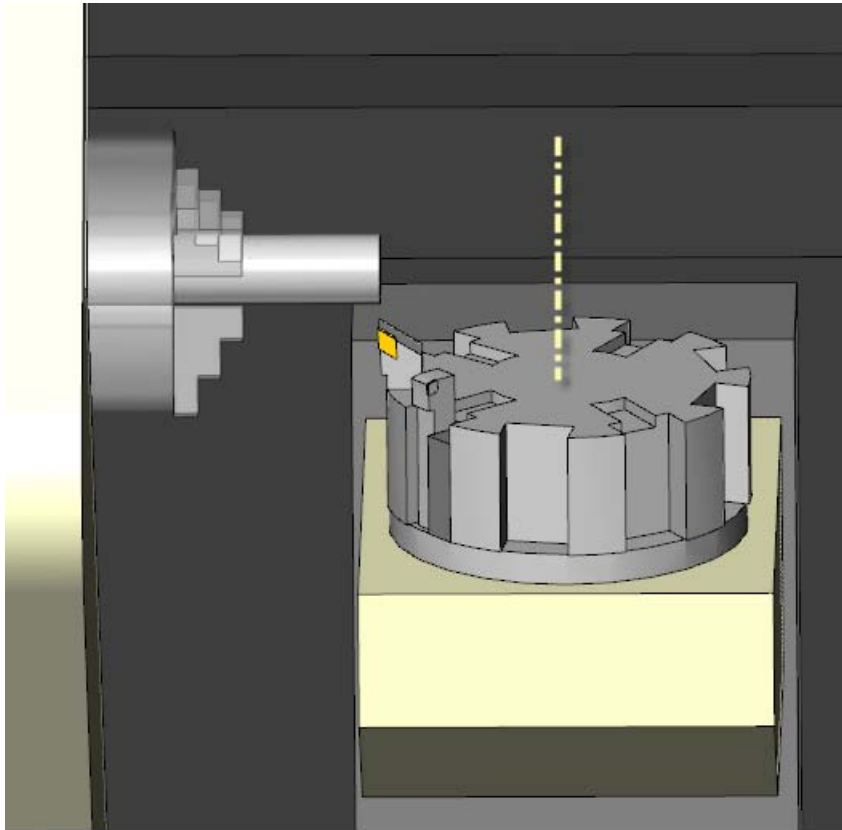


B: Setting up tools on the lower turret— turret rotates around X

This section describes lower-turret tools for the **Generic Fanuc Mill Turn.machine**. On such machines, the lower turret rotates around the X axis, as shown in the picture at right.

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

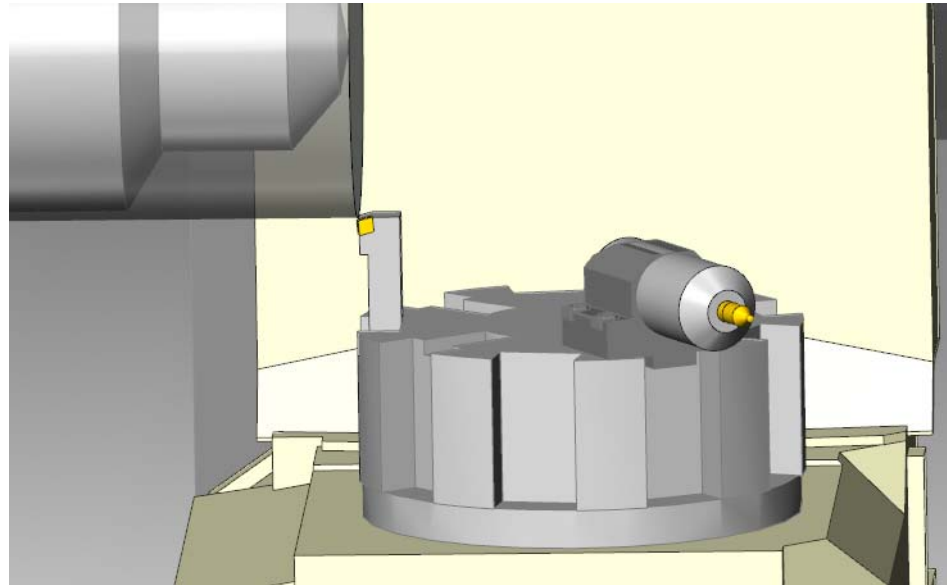
- ❖ **Initial tool setup**
- ❖ **Setting up for the left or right spindle**



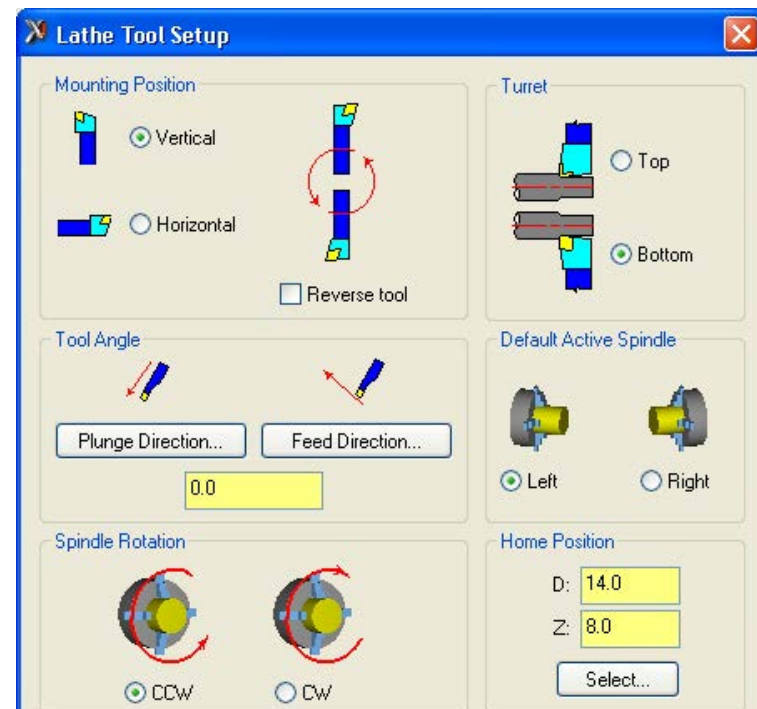
Initial tool setup

By default, tool definitions should be created as if the tool is being used on the left spindle.

Tools should be defined so that they are in the tool change position for the left spindle. They should be defined in the proper vertical or horizontal orientation. For example, a vertical, right-hand, insert-up tool should look like this.



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



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:

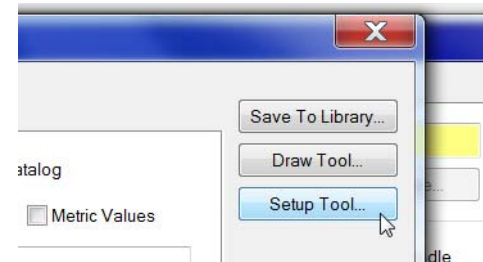
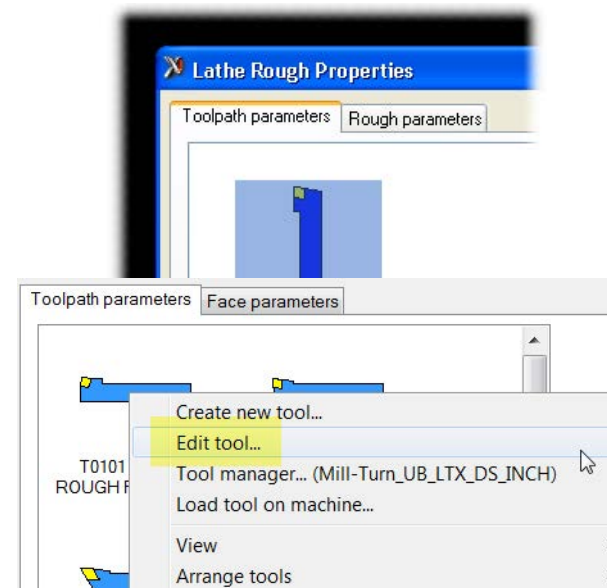
- vertical/horizontal mounting
- left-hand/right-hand holder
- insert up/insert down

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**.

Regardless of whether you will be machining on the left or right spindle, the picture should look like it is set up at the tool change position for the left spindle. **Setting up for the left or right spindle** on page 32 shows you how to set up for the right spindle.



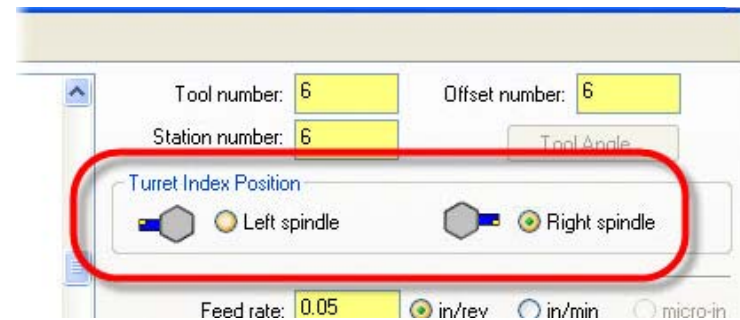
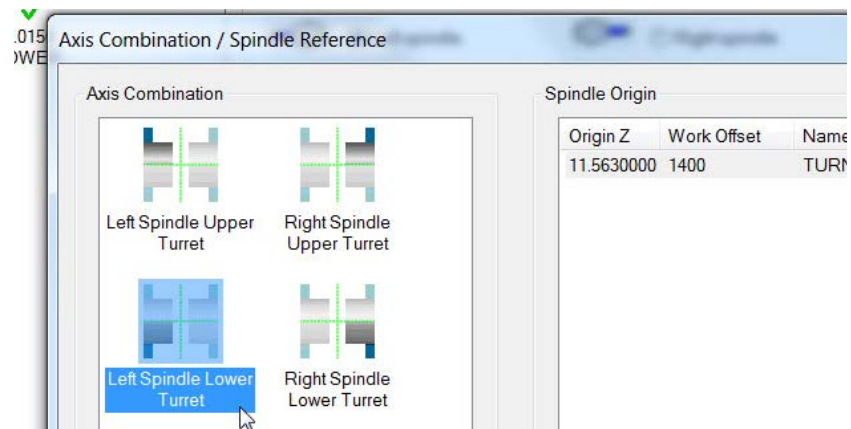
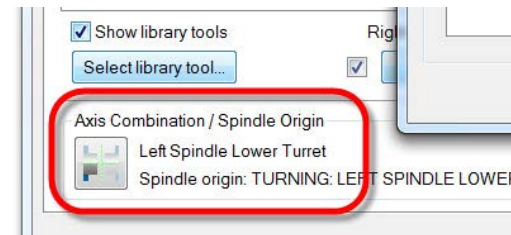
Setting up for the left or right spindle

There are two parts to setting up a toolpath using the lower-turret:

- First, select an axis combination that references the lower turret and desired spindle.
- Second, select the desired spindle with the **Turret Index Position**.

Follow these steps:

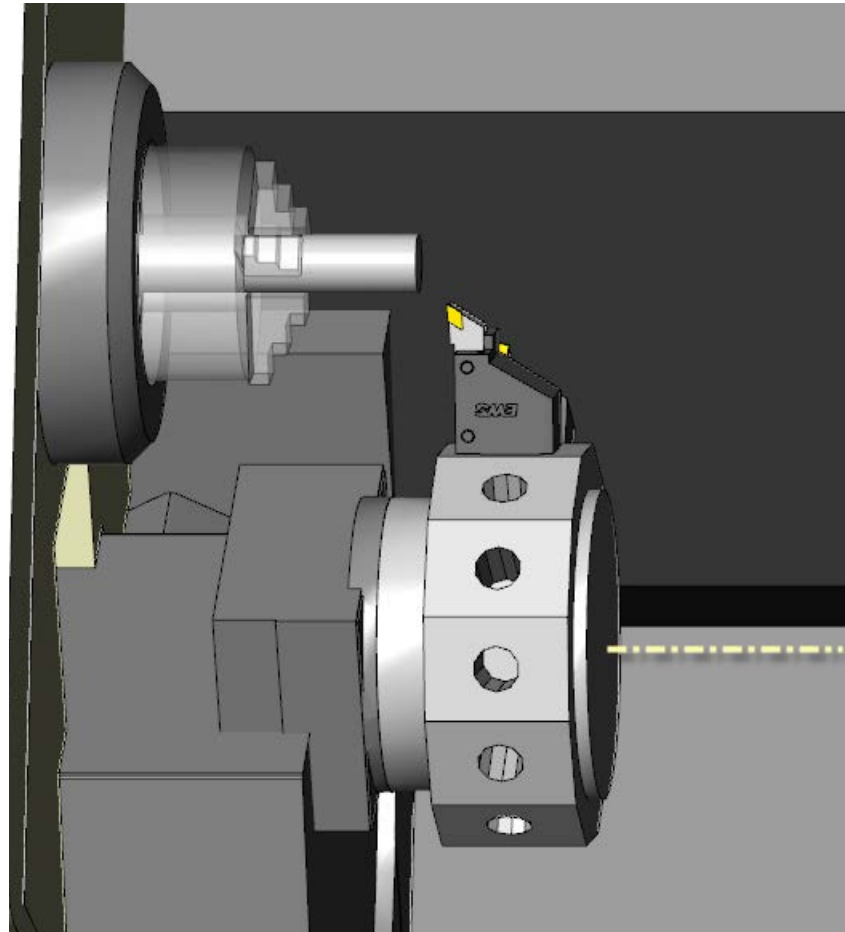
- Click the **Axis Combination** button.
- Select the desired turret/spindle combination. This picture shows the axis combination for using the lower turret on the left spindle.
- If necessary, select the proper setup from the **Spindle Origin** list.
- Select the desired tool. Make sure the picture shows it oriented as described in **Initial tool setup** on page 30.
- Use the **Turret Index Position** to index the tool for the proper spindle.



C: Setting up tools on the lower turret— turret rotates around Z

This section describes lower-turret tools for the **Generic Fanuc Mill Turn LTZ.machine**. On such machines, the lower turret rotates around the Z axis, as shown in the picture at right.

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

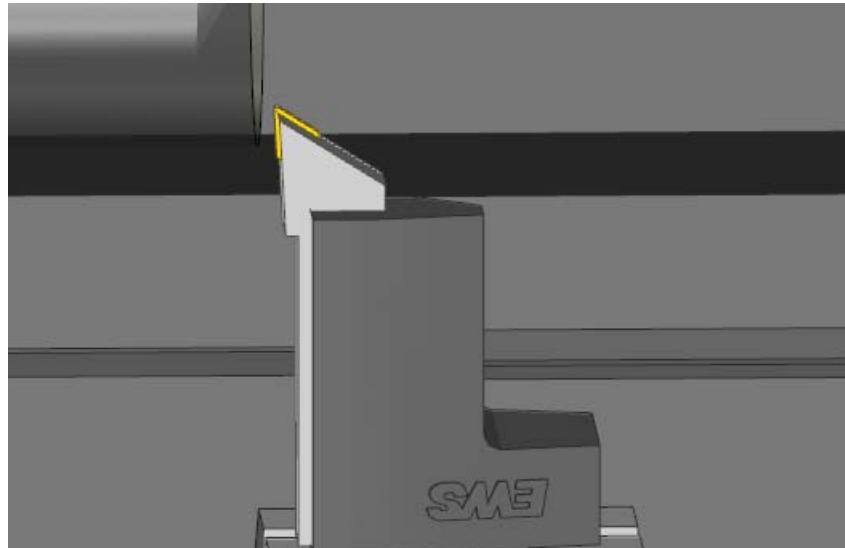


Initial tool setup

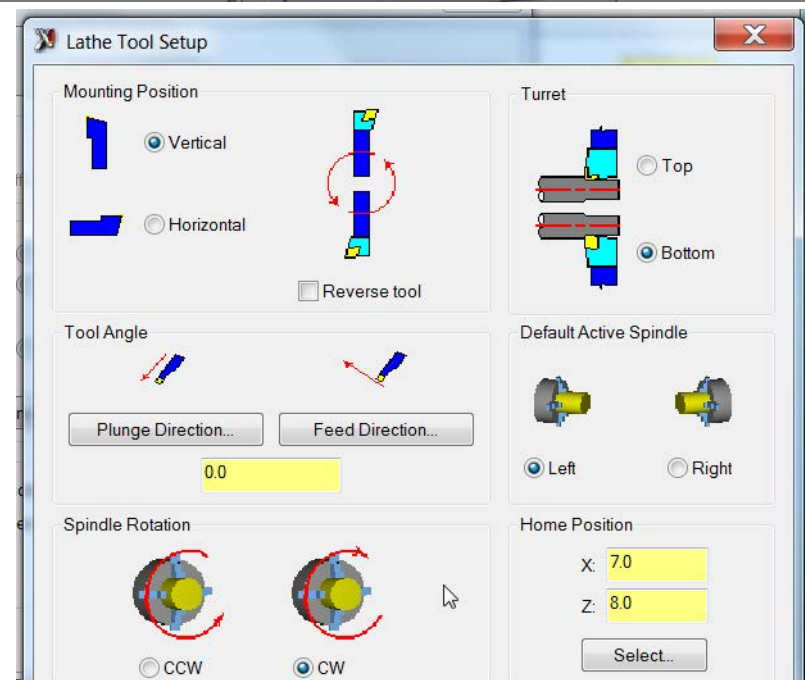
For Z-oriented lower turrets, tools should be defined so that they are oriented proper for the intended spindle. They should be defined in the proper vertical or horizontal orientation.

Note that this is different than tools in the B-axis head. For the B-axis head, the tool needs to be defined in the tool change position.

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



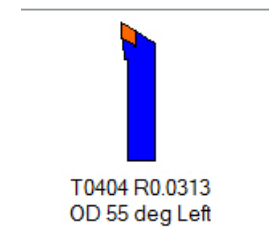
The tool definition settings for the tool as pictured above would look like this in Mastercam.



Then when you select the tool in Mastercam, the picture in the tool selection window should look like this (the orange insert instead of yellow means “insert down”).

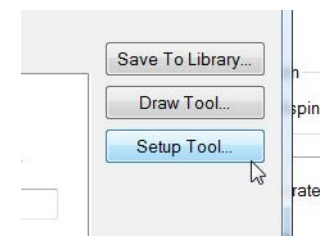
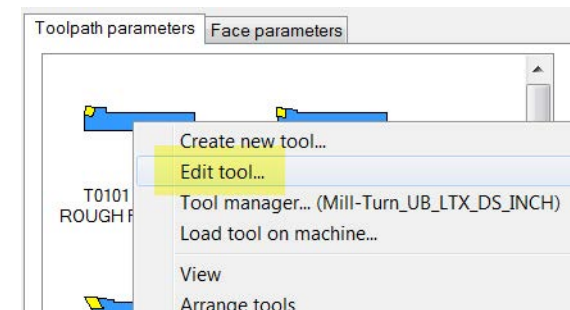
The picture that you see in Mastercam should show the proper orientation for:

- 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 17 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.

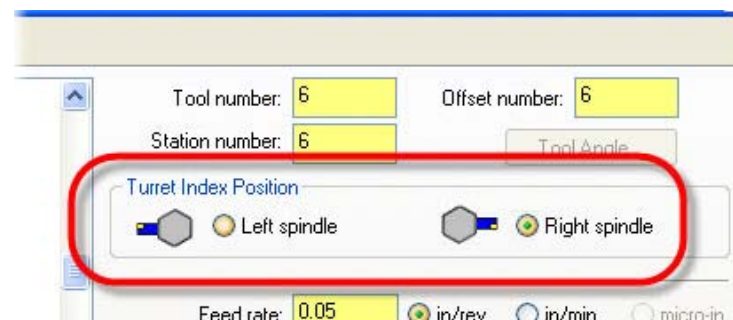
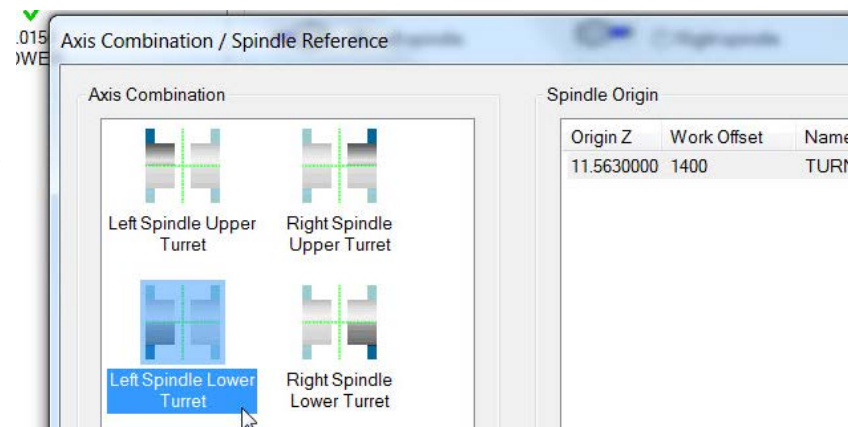
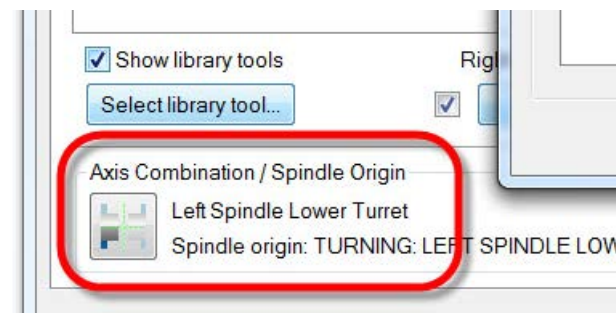


Setting up for the left or right spindle

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.

Note that the **Turret Index Position** option is not available when you are using a lower-turret Z machine. For these machines, the tool definition must point it to the desired spindle.



D: Setting up tools for pinch turning

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.

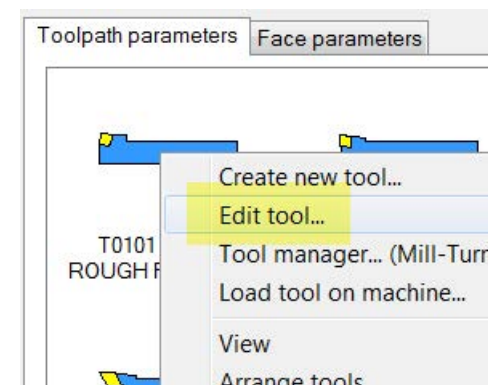
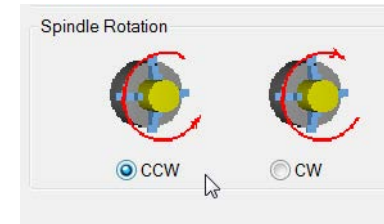
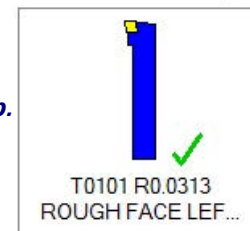
Spindle direction—The spindle direction for the two tools must be the same.

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

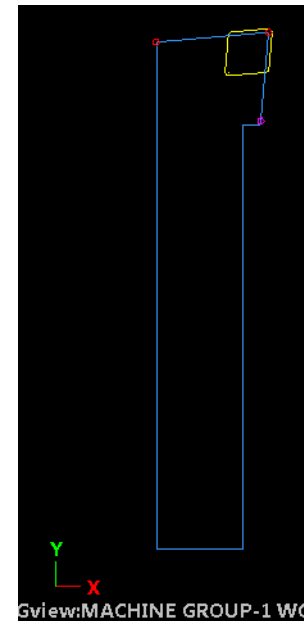
If the upper-turret tool is insert-down...



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

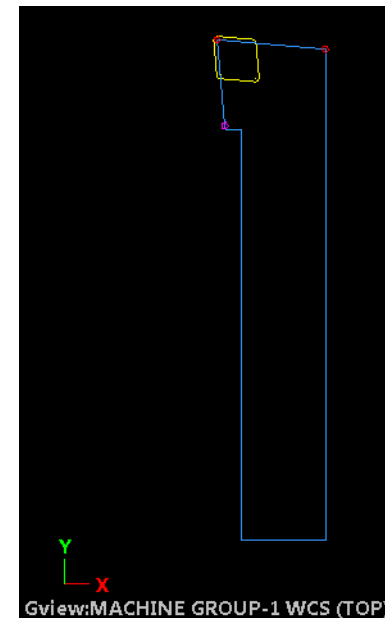


Mastercam displays the tool and holder profile as currently defined. For example, the tool shown here is insert-up with a left-hand holder, but is clearly not defined correctly for work on the left spindle. If you wanted to use this tool in a pinch-turn operation, and it needs to be insert-up to complement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



See “Insert up/down and LH/RH holders” on page 17 to learn more about these settings. When you 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.

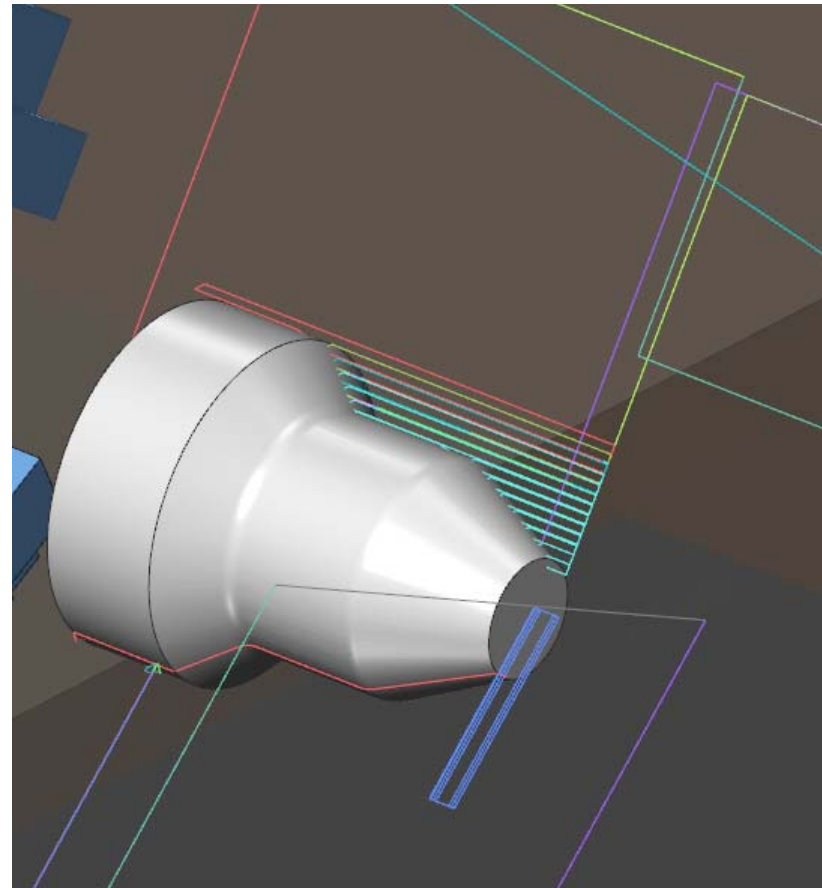


Chapter 3: Working with toolpaths

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

This chapter includes the following sections:

- ❖ **Reference positions**
 - ◆ Setting the start point and end point for an operation
 - ◆ Setting the type of approach/retract motion
 - ◆ Creating custom reference positions
 - ◆ Reference positions and reference points
- ❖ **Mill machining modes**
 - ◆ Polar (G12.1) and cylindrical (G7.1) interpolation
 - ◆ G68.5 coordinate conversion cycle
 - ◆ C-axis clamping and braking
 - ◆ Working with coolant
- ❖ **Multiaxis toolpath settings**
 - ◆ Rotary start position
 - ◆ Pole handling
 - ◆ G43.4 (tool tip point control)
 - ◆ G43.4 (tool tip point control)



A: 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**
- **Reference positions and null tool changes**

- ❖ **Setting the type of approach/retract motion**

- ❖ **Creating custom reference positions**

- ❖ **Reference positions and reference points**



Setting the start point and end point for an operation

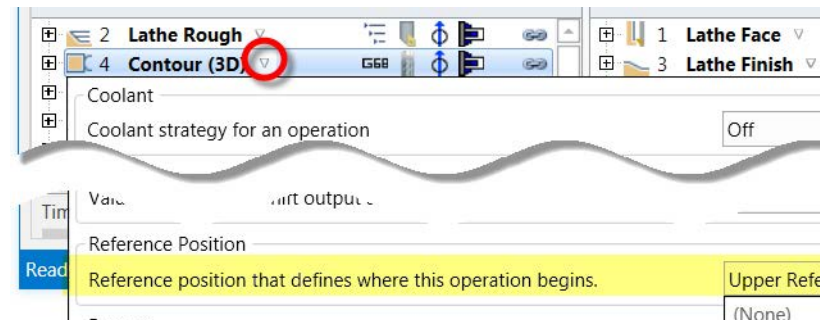
Your **.machine** file includes a set of reference positions that have been defined specifically for your individual machine. Use these to tell Mastercam where you want to start and end each operation. For example, you might—or might not—want to move all the way to the home position between operations.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



```

N110
(OPTION # 4)
(G68.5 ON TILTED PLANE ORIGIN ON SPI
G28 U0. Y0.
G28 W0.
M250
G0 B0.
M251
(T016 | 1/8 FLAT ENDMILL | DIA.

```



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.

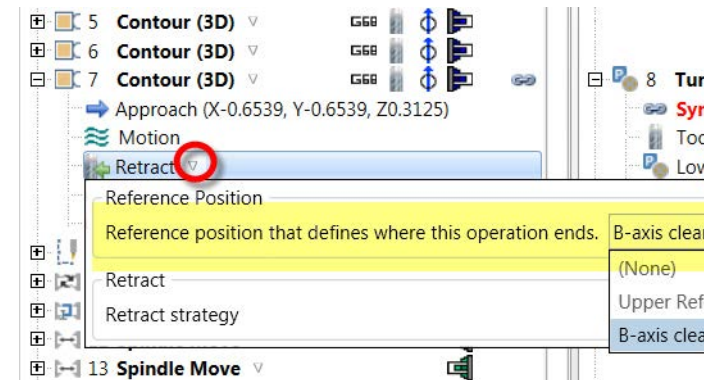
The example shown here displays a user-defined reference position. You can create such positions if, for example, you don't want to retract all the way to the home position between operations. (See **Setting the type of approach/retract motion** on next page.)

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.

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.



```
G0 Z.3125
G69.5
G53 X0. Y0.
G53 Z-9.522 M205
M953
```

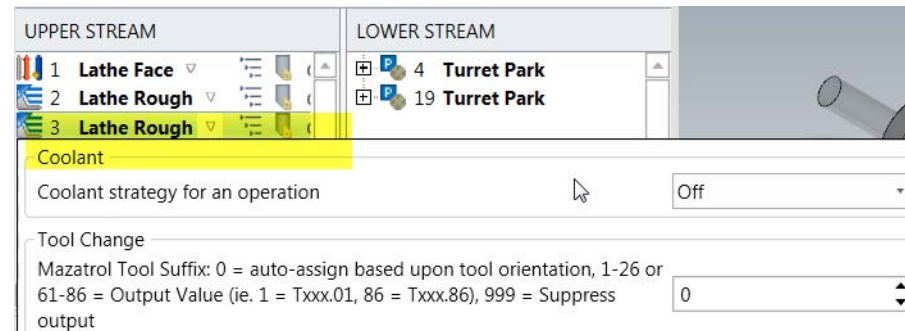
```
N120
(OPERATION # 9)
```

```
64 G0 B0.
65 M251
66 M01
67
68 N110
69 (OPERATION # 2)
70 (T002.01 | OD 55 DEG LEFT | INSERT)
71 G123.1
72 M901
73 M202
74 T002.01 T003 M6
75 M250
76 G0 B90.
77 M251
78 G53.5
```

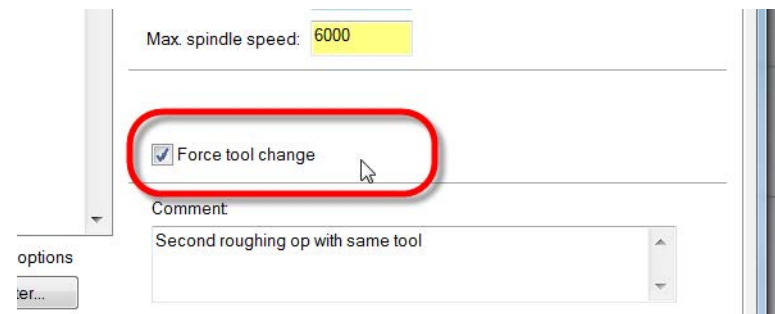


Reference positions and null tool changes

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.



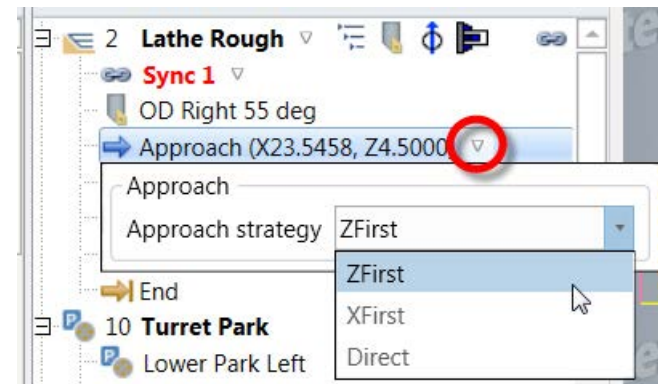
Setting the type of approach/retract motion

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

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

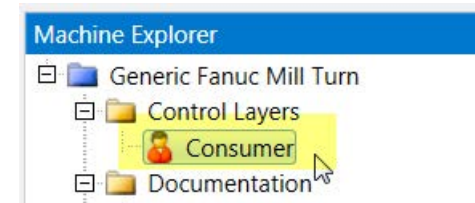


Click the small triangle next to the **Approach** or **Retract** node in the tree.

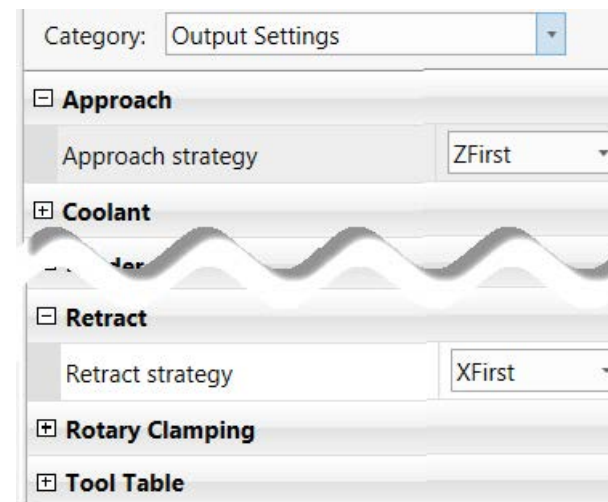


You can also change the default selection in the **.machine** file.

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.

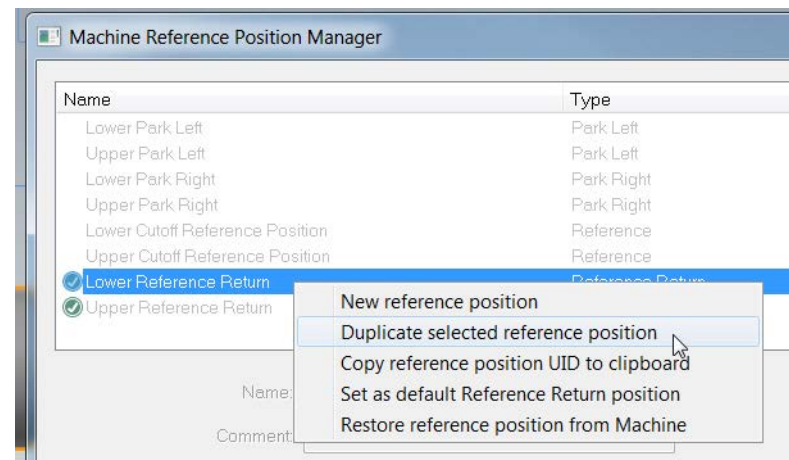


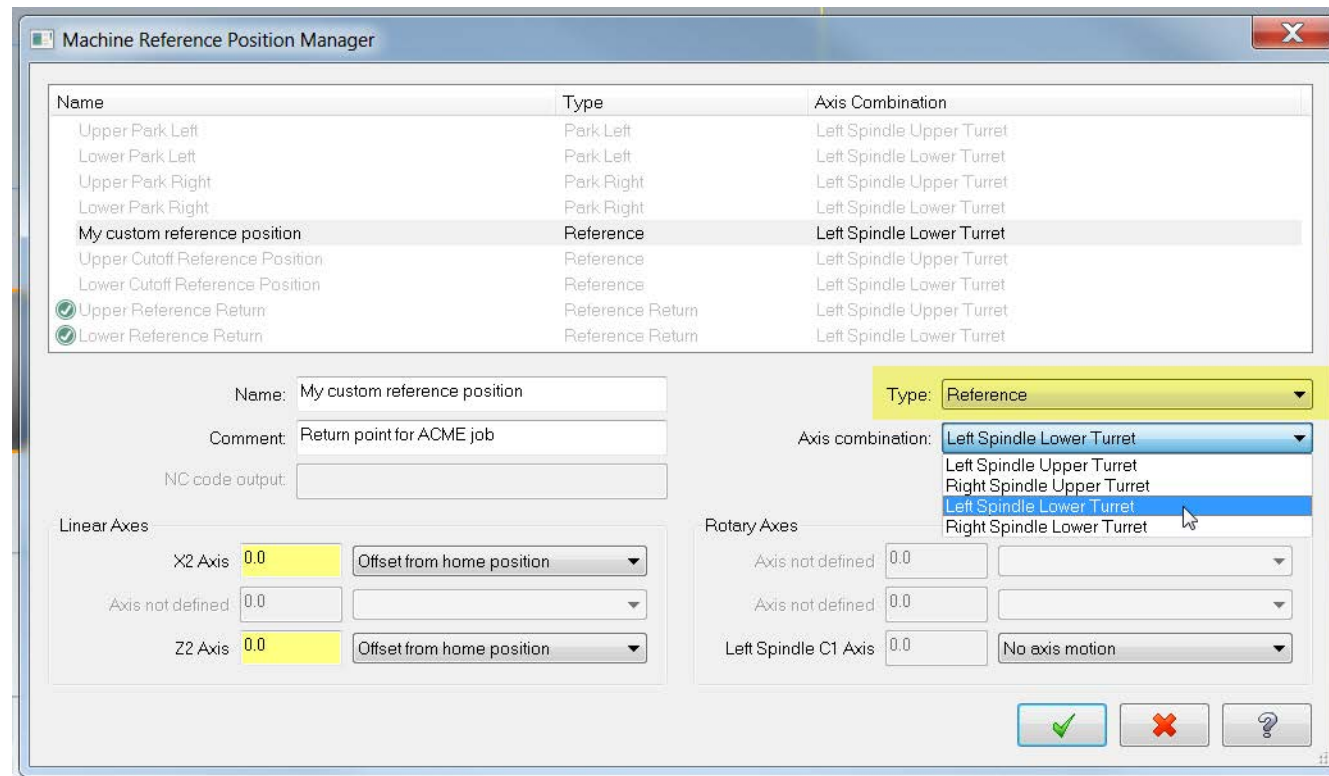
Creating custom reference positions

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**.

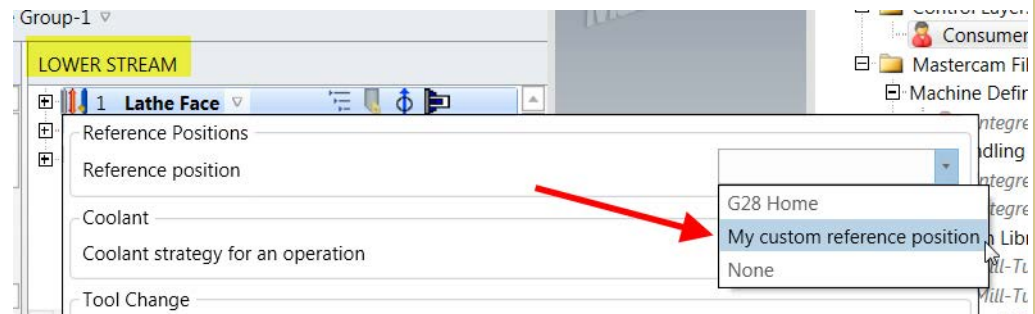




- Each reference position is associated with a specific axis combination. Mastercam uses this to filter the reference points by spindle, turret, and/or stream. For example, when you select reference points in the Sync Manager for a lower stream operation, only reference points valid in that stream will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

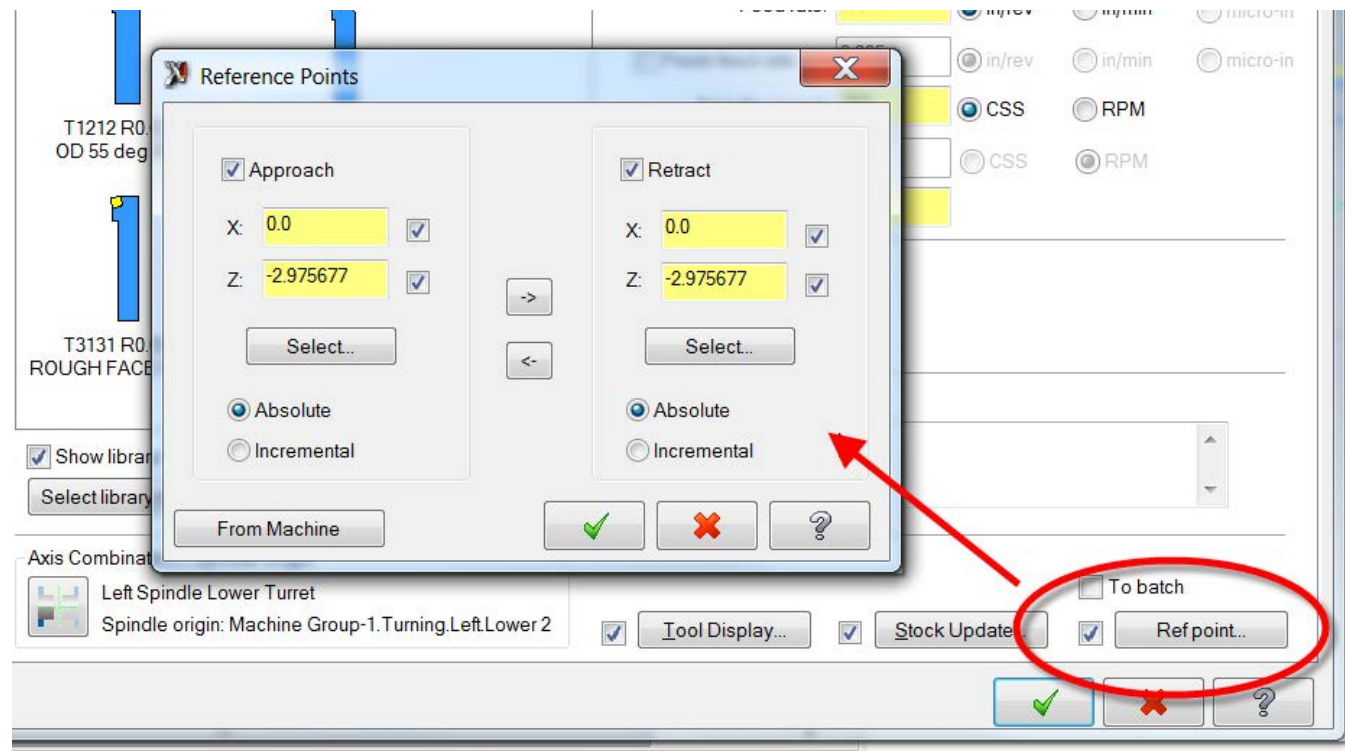


position should be available. Make sure you look in the proper stream.



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*.

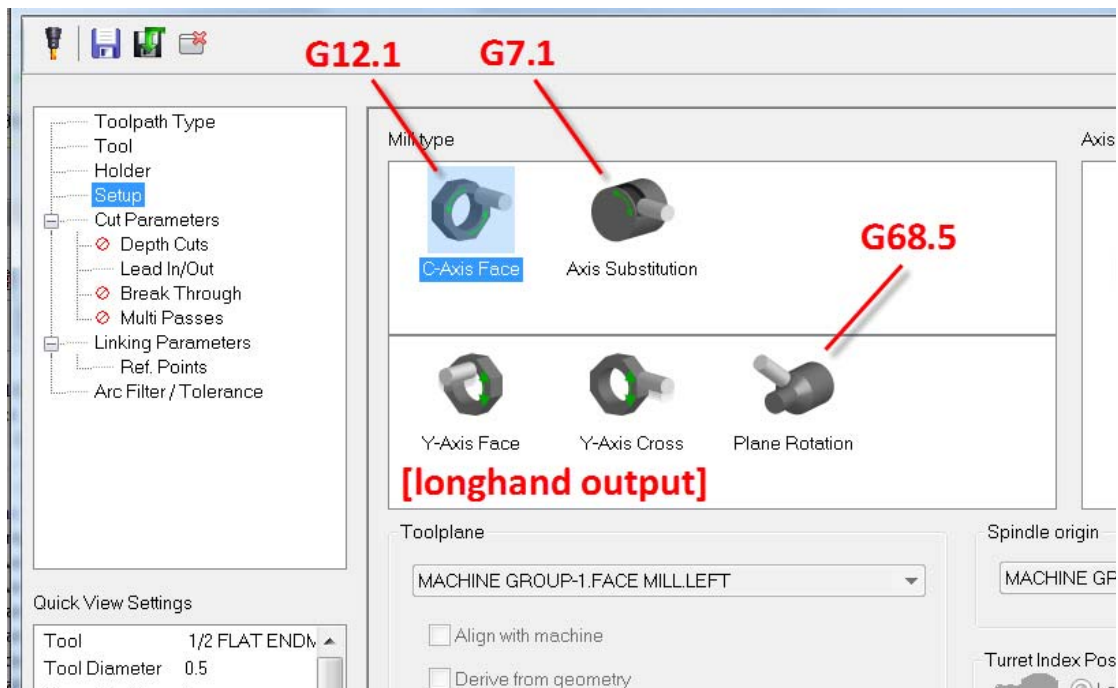


B: Mill machining modes

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

- ❖ **Polar (G12.1) and cylindrical (G7.1) interpolation**
- ❖ **G68.5 coordinate conversion cycle**

The cycles are automatically keyed to the milling setup types on the **Setup** page for mill operations.



The following additional features are also configured with Sync Manager options:

- ❖ **C-axis clamping and braking**
- ❖ **Working with coolant**

Note that the workflow for working with coolant is completely different in Mill-Turn compared to other Mastercam products; see **Working with coolant** to learn more.

See also “Multiaxis toolpath settings” on page 56 for additional milling options.



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

Your Fanuc **.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.

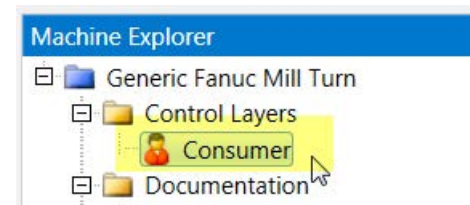
Default settings for G7.1 and G12.1

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

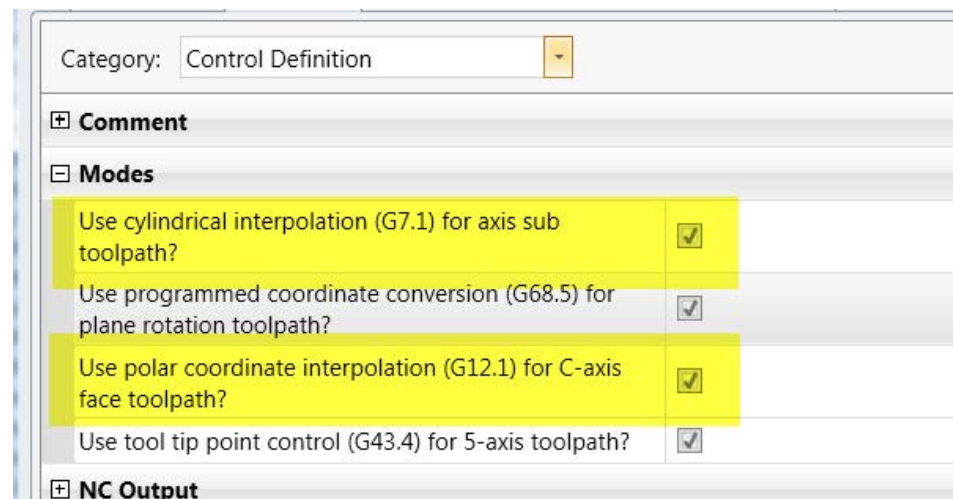
- Choose whether the default output mode will be the G7. 1/ G12. 1 cycle or longhand output.
- Choose whether the X-axis output will be in diameter or radius values.

Follow these steps:

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



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



6. Select **Category: Output Settings**.
7. Go to the **Modes** section.
8. For each cycle, choose whether to output the X-axis coordinates in diameter or radius values.
9. Save the **.machine** file when you are done.

Category: Output Settings

☐ Approach

☐ Coolant

☐ Header

☐ Machine Parameters

☐ Modes

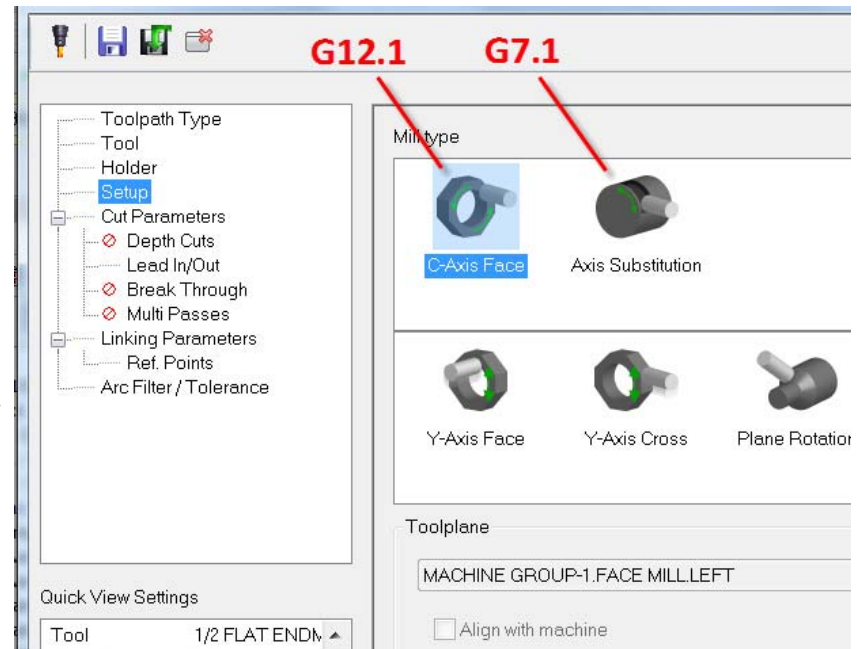
HSAMode	Both
Output G12.1 polar interpolation X values in diameter?	<input type="checkbox"/>
Output G7.1 cylindrical interpolation X values in diameter?	<input type="checkbox"/>



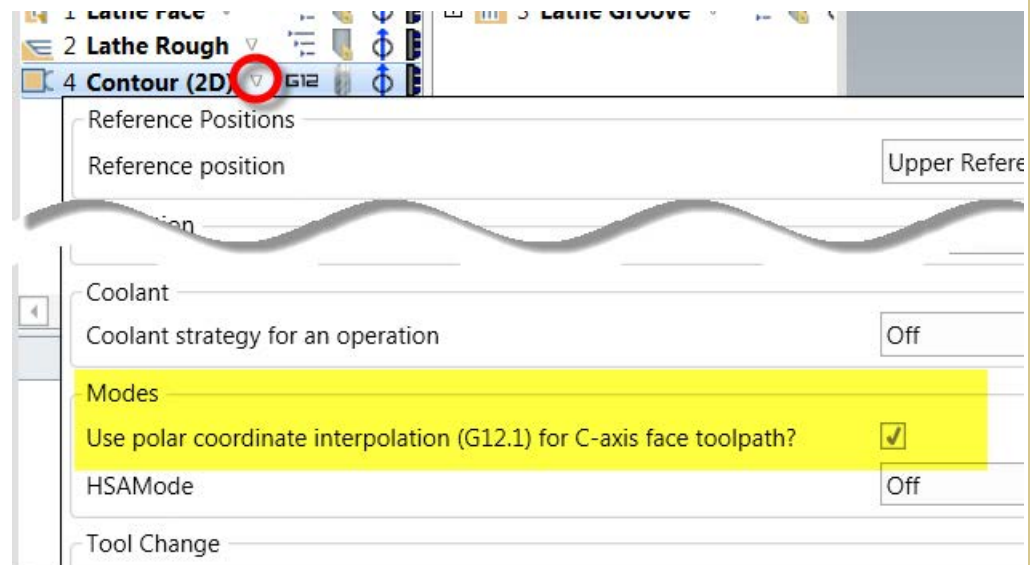
Programming G7.1/G12.1 for each operation

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.



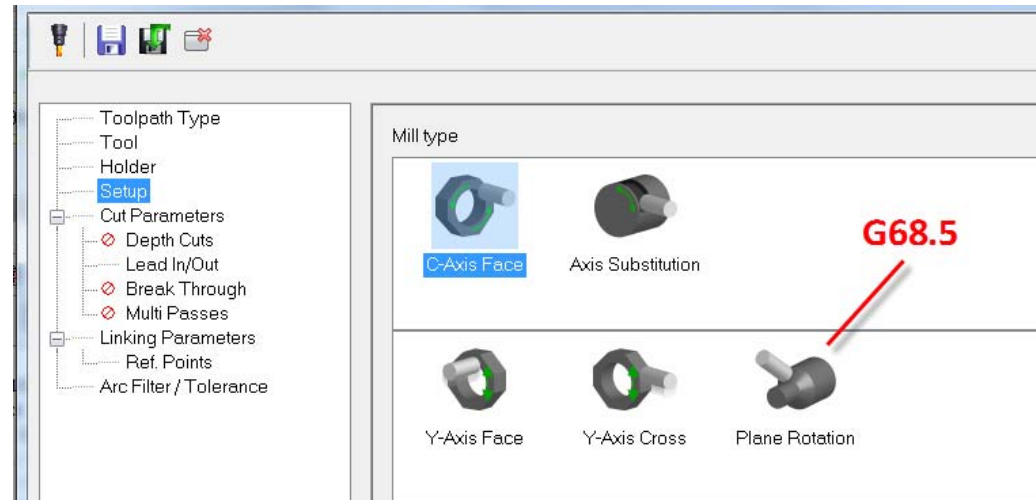
3. After you load the part in the Sync Manager, you can choose whether or not to output the G7.1/G12.1, or longhand output. Click the little triangle next to the operation and select the desired option.
4. For each operation, the default value that you see comes from the **.machine** file. If you override it, make sure you press **[Ctrl+S]** to save the new setting.



G68.5 coordinate conversion cycle

Your Fanuc **.machine** file supports G68.5 coordinate conversion (tilted-plane) machining cycles. Follow this general workflow.

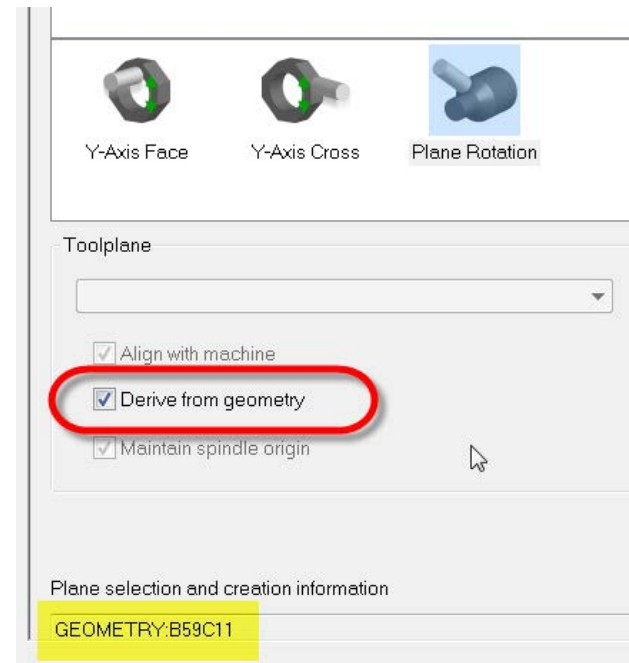
1. The G68.5 mode is automatically enabled when you select **Plane Rotation** on the **Setup** page for milling operations.



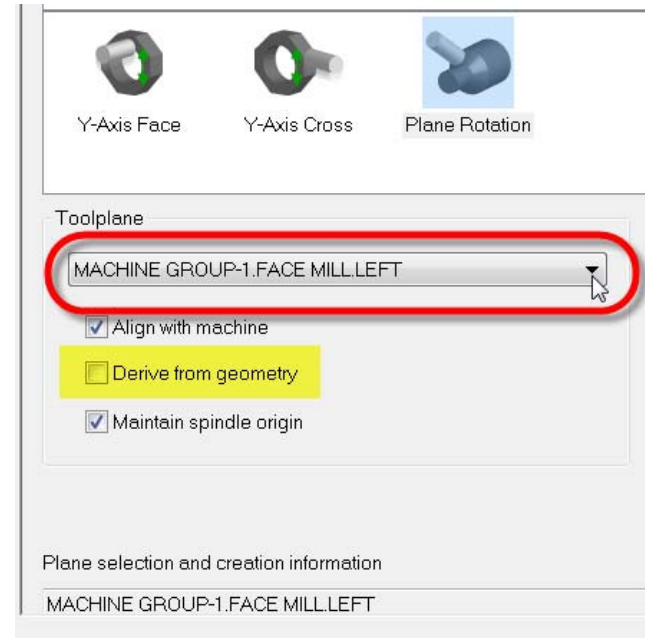
2. Next you need to tell Mastercam how to establish the toolplane.
 - For most applications, you will want Mastercam to automatically create a toolplane based on the selected geometry.
 - Select the **Derive from geometry** option to do this.

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.

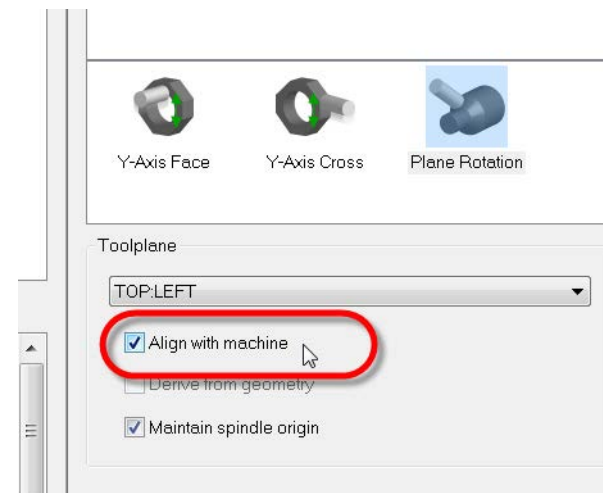
Note: The **Derive from geometry** option will only be available if the selected geometry defines a plane. For example, if the chained geometry is only a single line, the option will be grayed out because you cannot specify a plane based on only a single line.



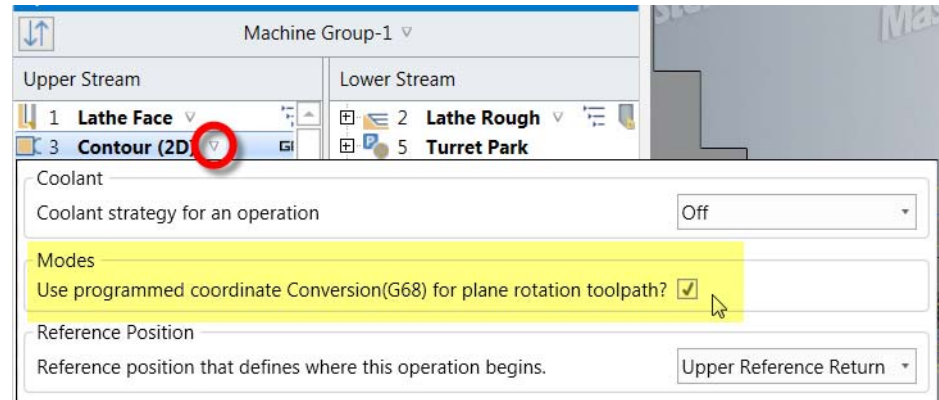
3. You can also use a predefined toolplane instead of creating a new one from the geometry. To do this, clear the **Derive from geometry** option, and select the **Toolplane** from the list.
 - Do this if you want to face mill or cross mill but you want to force G68.5 instead of using G17/G18/G19 plane selection.



4. Make sure the **Align with machine** option is selected.



5. After you load the part in the Sync Manager, you can choose whether to output the G68. 5, or use longhand output. Click the little triangle next to the operation and select the **Use programmed coordinate conversion...** option to output the G68. 5.



C-axis clamping and braking

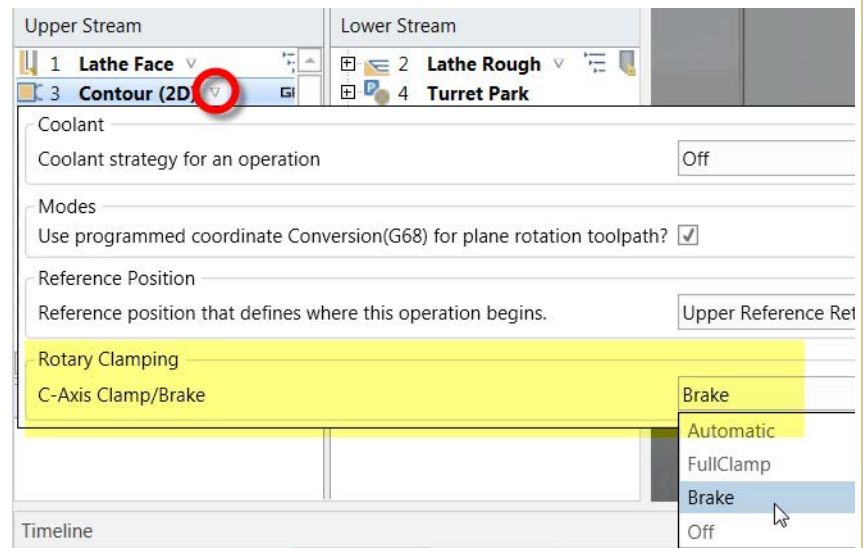
Your **.machine** file lets you explicitly turn on C-axis high-pressure clamping mode (M89/M189) or braking (M88/M188).

M89/M189 and M88/M188

The C-axis high-pressure clamp mode is available for Mill toolpaths. Choose either the full clamp or spindle brake function.

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 or brake the C-axis based on the application and toolpath type.
- **FullClamp**—Force an M89 code for C1 (left spindle) or M189 for C2 (right spindle).
- **Brake**—Force an M88 code for C1 (left spindle) or M188 for C2 (right spindle).
- **Off**—Force M90/M190 for the selected operation.

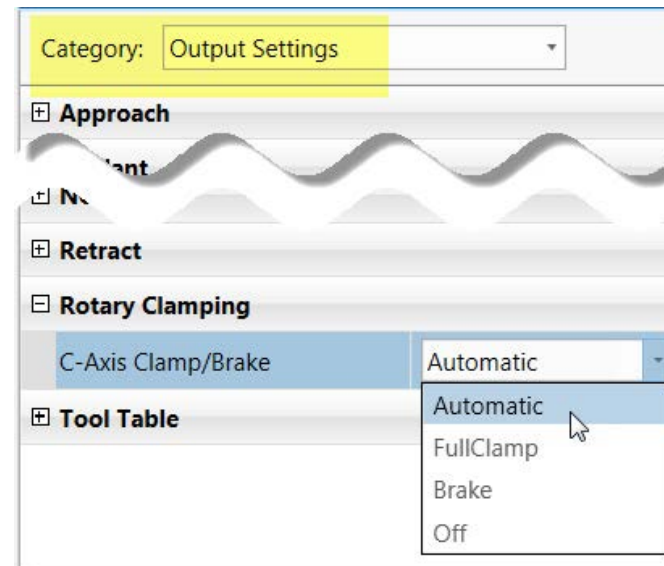
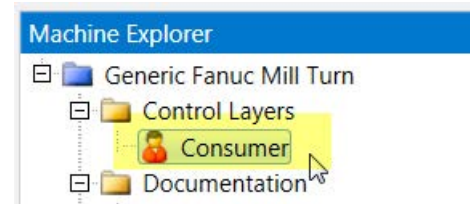


Default settings

You can choose which mode will be the default

Follow these steps:

1. Open the Fanuc **.machine** file in Code Expert.
2. Double-click the **Consumer** layer.
3. Select **Category: Output Settings**.
4. Go to the **Rotary Clamping** section.
5. Select the desired **C-axis Clamp/Brake** mode:
 - **Automatic**—Mastercam will automatically decide whether to clamp or brake the C-axis based on the application and toolpath type.
 - **FullClamp**—Force an M89 code for C1 (left spindle) or M189 for C2 (right spindle).
 - **Brake**—Force an M88 code for C1 (left spindle) or M188 for C2 (right spindle).
 - **Off**—Force M90/M190 for the selected operation.
6. Save the **.machine** file.



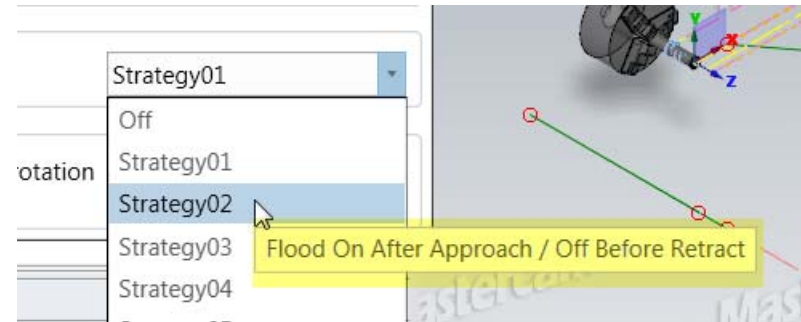
Working with coolant

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

- Flood coolant for B-axis head and lower turret(M08)
- Thru-spindle coolant for the milling spindle (M126)

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.



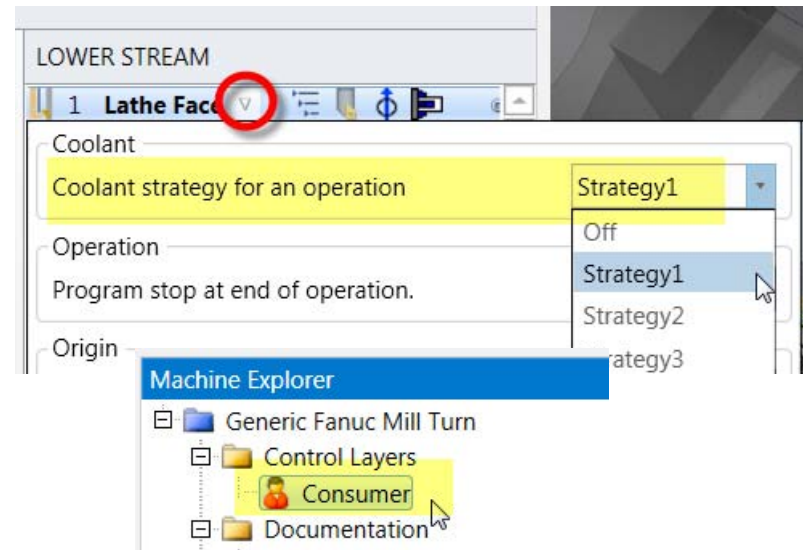
To select a strategy, click the small triangle next to the operation name and select the desired coolant strategy. Do this for each operation.

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, the appropriate coolant-off commands will be output automatically and you do not need to turn them off yourself.

Setting the default coolant option

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

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



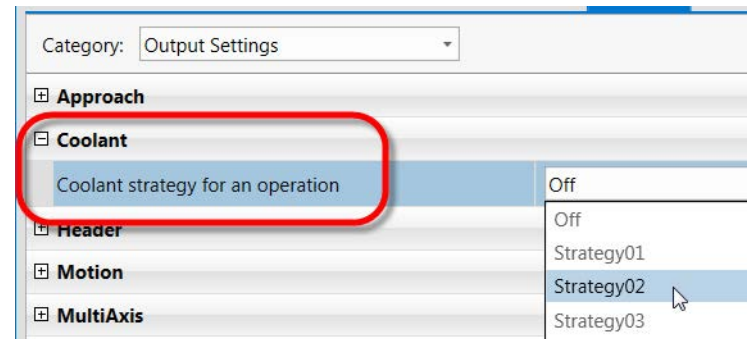
2. Double-click the **Consumer** layer.

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

4. Go to the **Coolant** section.

5. Set the desired default strategy.

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



Coolant options and null tool changes

Operations with null tool changes require special treatment to ensure that the coolant OFF codes are output properly. If you have a sequence of two or more operations with null tool changes in between, all of the operations should have the same coolant strategy selected in the Sync Manager. Mastercam will output the coolant ON code at the beginning of the first operation, and the coolant OFF code at the end of the final operation. You cannot change coolant for individual operations in the sequence.



C: Multiaxis toolpath settings

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

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

Rotary start position

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

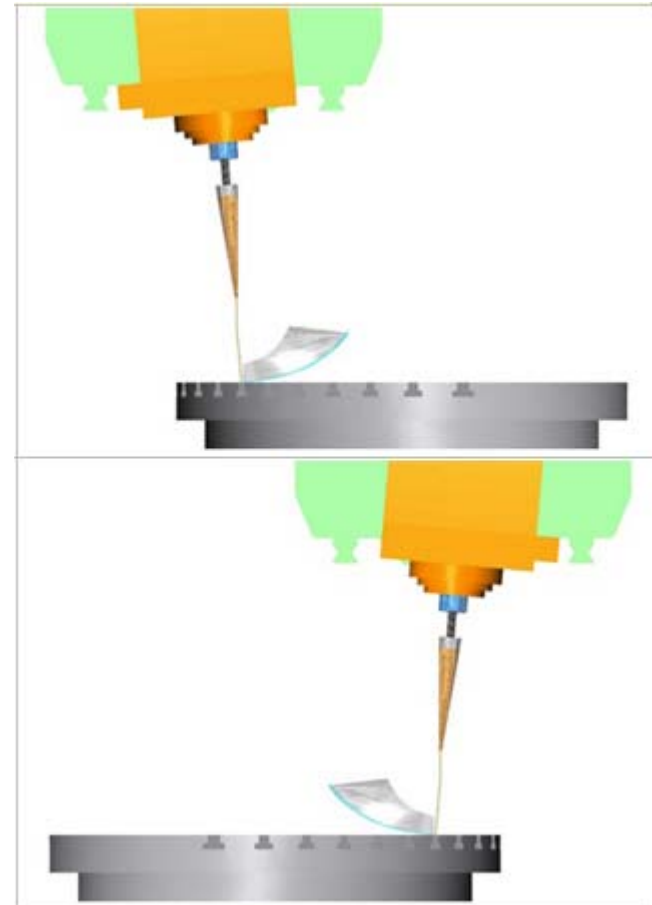
Typically, two logically equivalent solutions are possible for the starting rotary position of a toolpath. The difference between the two solutions is that the second location is separated by 180 degrees in the C-axis, and the sign of the B-axis is reversed. (The picture at right shows this for a generic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

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

Your Fanuc B-axis **.machine** file includes two options that are used by the post to select the proper starting position:

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

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



Preferred start angle type—Select the axis in which the desired start angle will be measured: either the **Primary** or **Secondary** rotary axis. For the Fanuc B-axis, the primary rotary axis is C, and the secondary rotary axis is B.

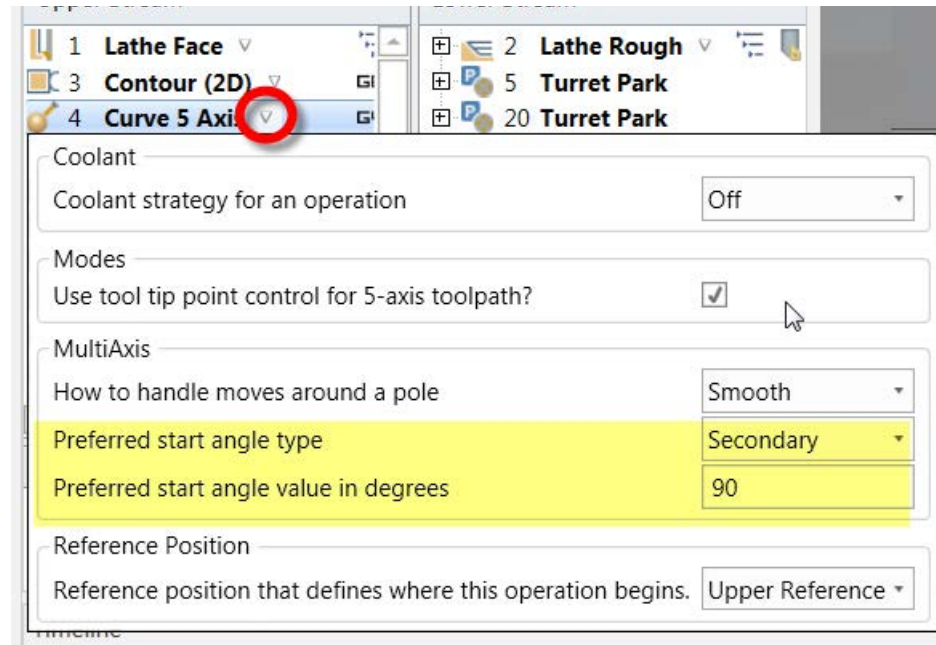
Preferred start angle value—Enter the desired start angle. Mastercam will select the solution closest to this angle. For example, if you enter a value of **90**, and the two possible solutions are –B80 and B80, Mastercam will select B80 for the start position.

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

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

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

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



Pole handling

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

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

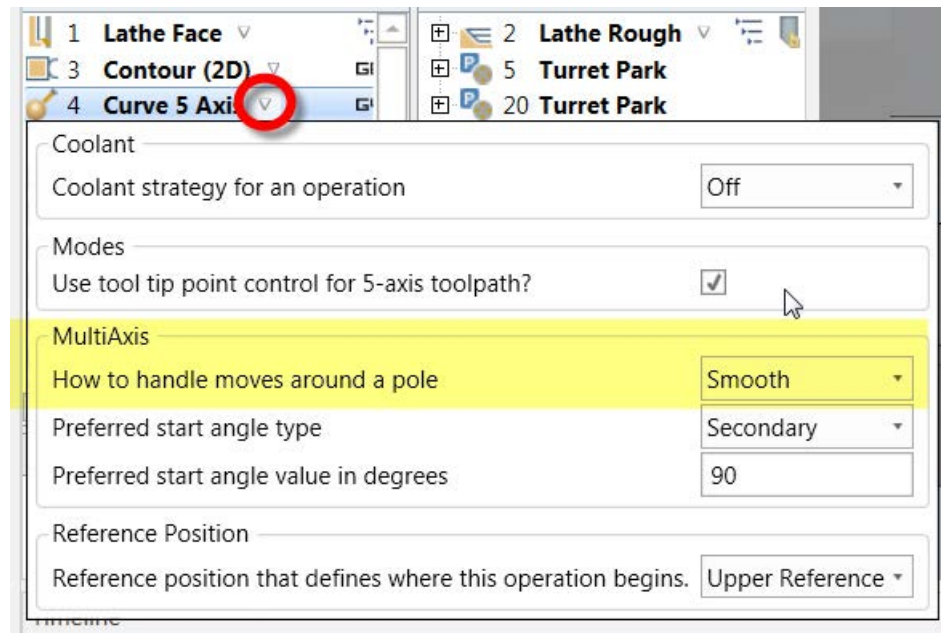
For each operation, you can select the preferred strategy in the Sync Manager.

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

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

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

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

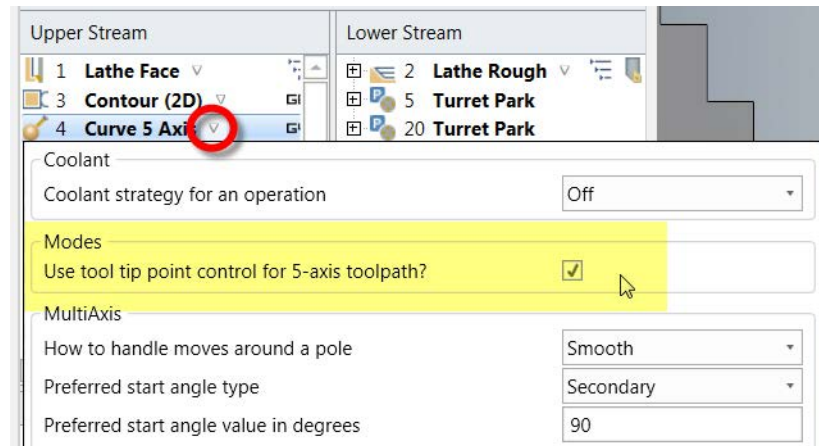


G43.4 (tool tip point control)

Your Fanuc **.machine** file supports the use of G43.4 mode for tool tip point control programming.

After you load the part in the Sync Manager, you can choose whether or not to use the G43.4 mode for each operation.

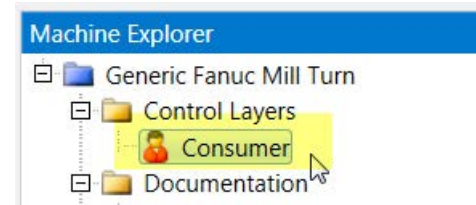
Click the little triangle next to the operation and select the **Use tooltip point control...** option to output the G43.4. This option is available for all multiaxis toolpaths.



Default settings for multiaxis toolpaths

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

1. Open the Fanuc **.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.



Category: Control Definition

Comment

Modes

Use cylindrical interpolation (G07) for axis sub toolpath?	<input checked="" type="checkbox"/>
Use programmed coordinate Conversion(G68) for plane rotation toolpath?	<input checked="" type="checkbox"/>
Use polar coordinate Interpolation G12 for C-axis face toolpath?	<input checked="" type="checkbox"/>
Use tool tip point control for 5-axis toolpath?	<input checked="" type="checkbox"/>

NC Output

Sequence Numbers



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

Control X ACME-MU844A.iof

Category: Output Settings

☒ **Approach**

☒ **Coolant**

☒ **Machine Parameters**

☒ **Motion**

☒ **MultiAxis**

Preferred start angle value in degrees	Inch: 90	Metric:
Strategy to be applied when moves around a pole	Smooth	
Preferred start angle type	Secondary	

☒ **Operation**



