Nakamura-Tome AS Application Guide

revision date: November 2018

Contents

Mastercam

1. Introduction

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

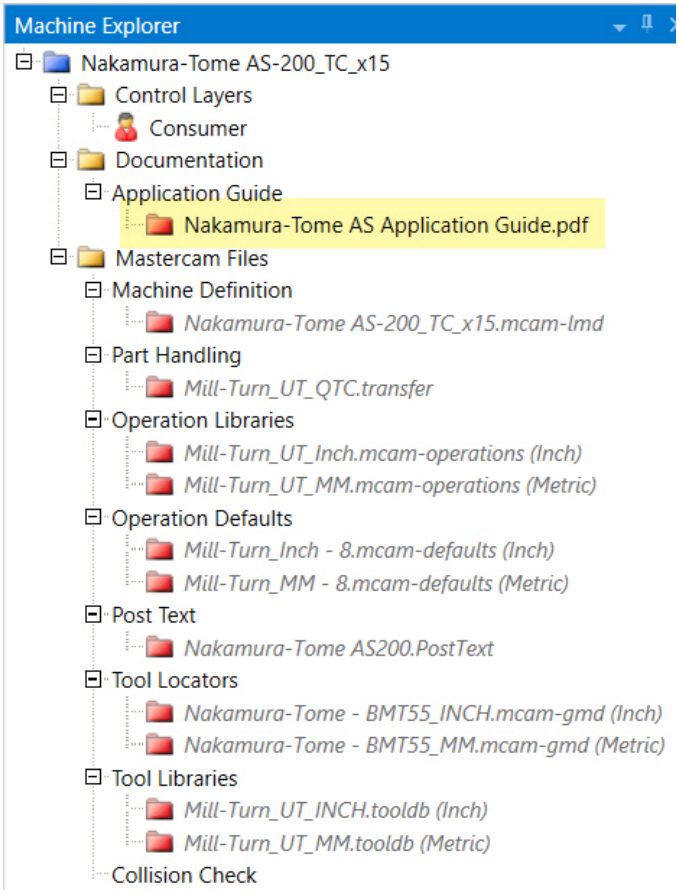
- **Customizing your .machine file**
- **Organizing NC output**
- **Approach and retract moves**
- **Milling cycles and modes**
- **Coolant**

Supported models and controls—This guide applies to all Nakamura-Tome AS models.

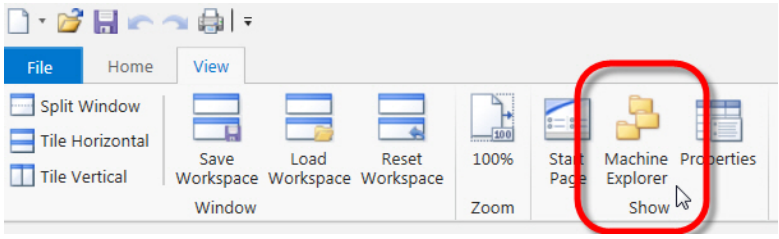
2. Customizing your .machine file

Contents

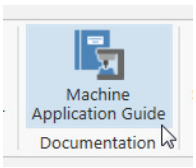
Whenever you load your **.machine** file in Mastercam, Mastercam also starts Code Expert and loads your **.machine** file there as well. Code Expert is where you can make changes to your **.machine** file, such as editing default settings.



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.

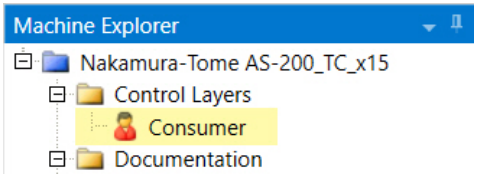


A. Default values for .machine file settings

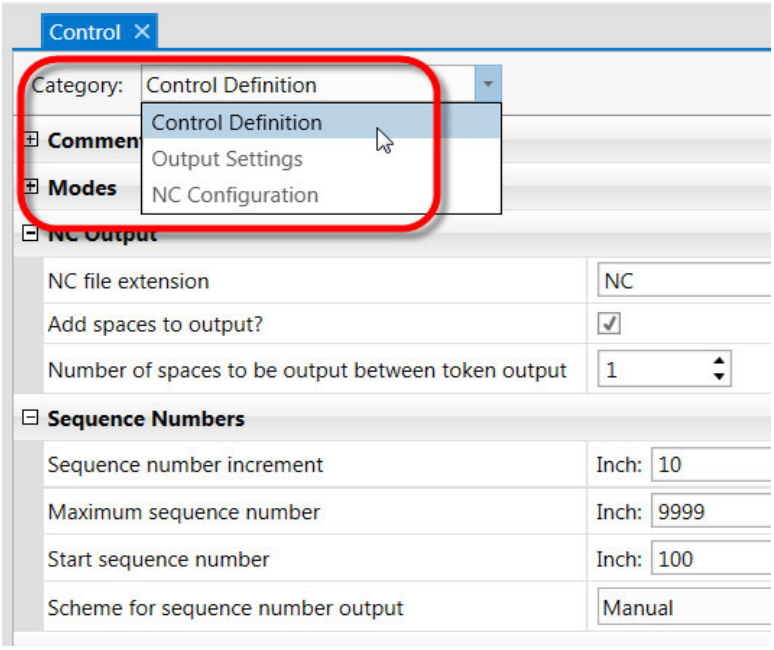
Contents

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 numbers, 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 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.

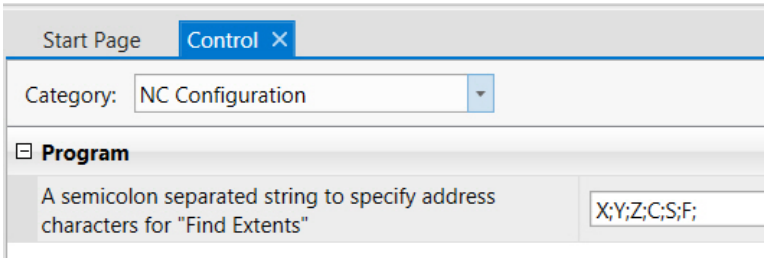
Many of these settings are self-explanatory and you can easily configure them by simply browsing the interface. Settings that are specific to this **.machine** file are described in this guide.

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

B. Configuring the Code Expert editor

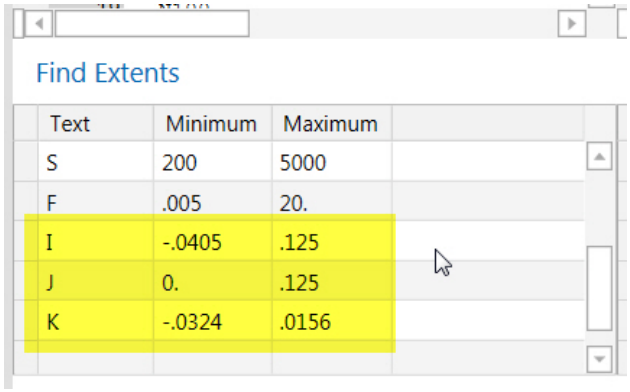
Contents

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 (**;**).
4. Press [**Ctrl+S**] to save the **.machine** file.

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



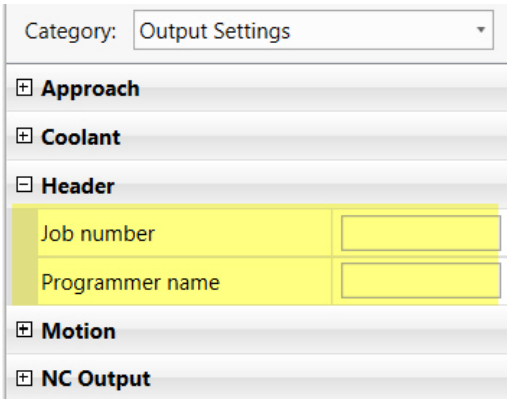
C. Programmer and job number

Contents

Your Nakamura-Tome AS post outputs the job number and programmer name in the header of your NC file:

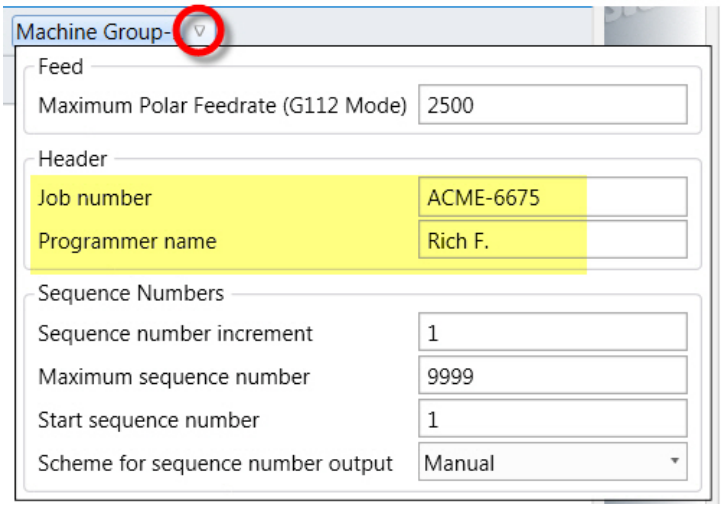
```
%
O1234 ( ACME-6675.NC )
(DATE:  TUESDAY, 26 JUNE 2018  - TIME:  13:49)
(MCX FILE -  ACME-6675.mcam)
(JOB NUMBER -  ACME-6675)
(PROGRAMMER -  RICH F.)
(T0202  |  OD 55 DEG RIGHT  | INSERT -  DNMG-432)
(T0303  |  SPOT TOOL .75 DIA.  | INSERT -  NONE)
```

You can save the default values in your **.machine** file.



1. Go to the **Output Settings** category.
2. Open the **Header** group.
3. Enter the desired values.
4. Press [**Ctrl+S**] to save the **.machine** file.

You can also enter custom values for each part. Click the small triangle next to the machine group name and enter the desired values.



3. Organizing NC output

Contents

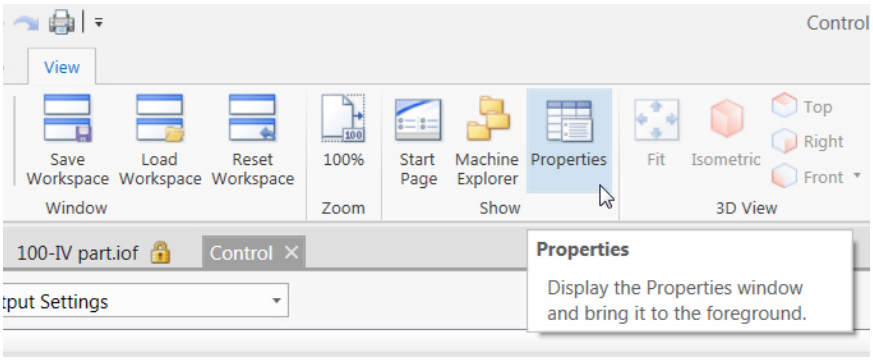
A. Setting the toolpath directory

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

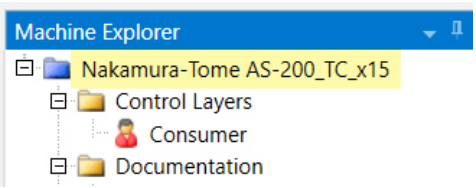
\my mcam2019\Mill Turn\NC

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

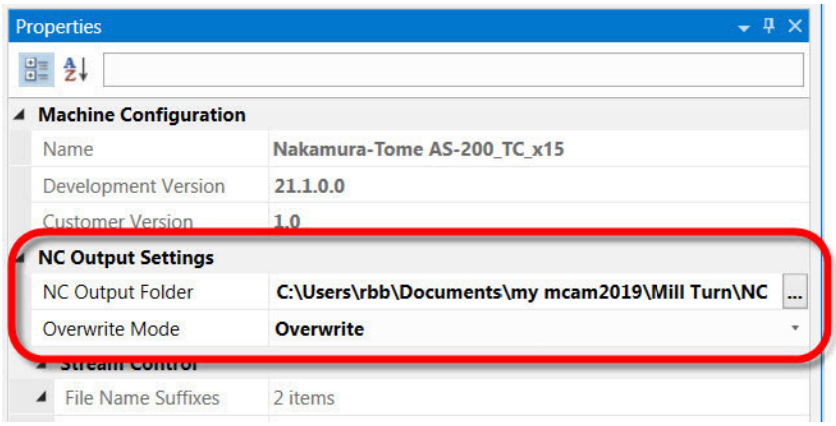
1. Open the **.machine** file in Code Expert.
2. Make sure that the **Properties** window is displayed.



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



4. Select the desired **NC Output Folder**. Mastercam will write your NC files to this folder.

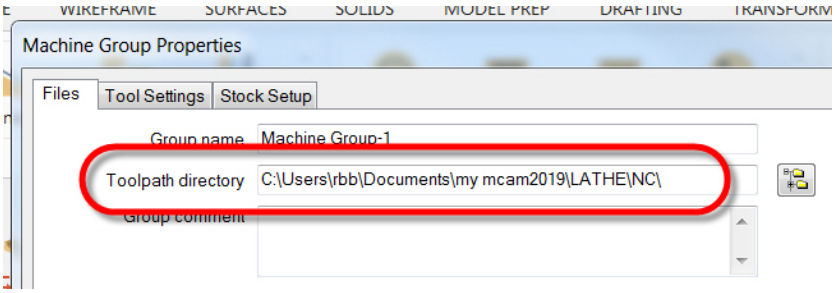


5. You can also choose whether Mastercam will automatically overwrite NC files with the same name, or prompt you to enter a different name. Select the desired **Overwrite Mode** to control this.

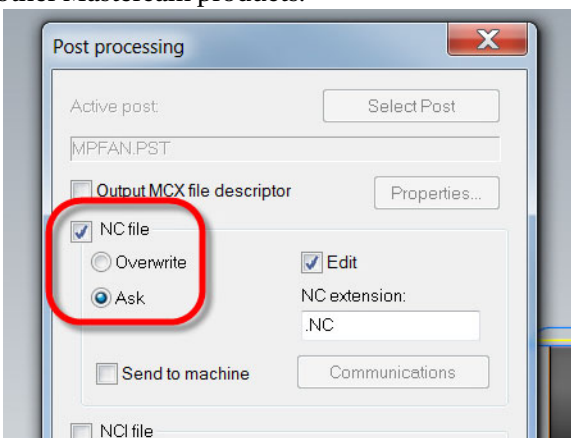
6. Save the **.machine** file when you are done

Contents

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



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



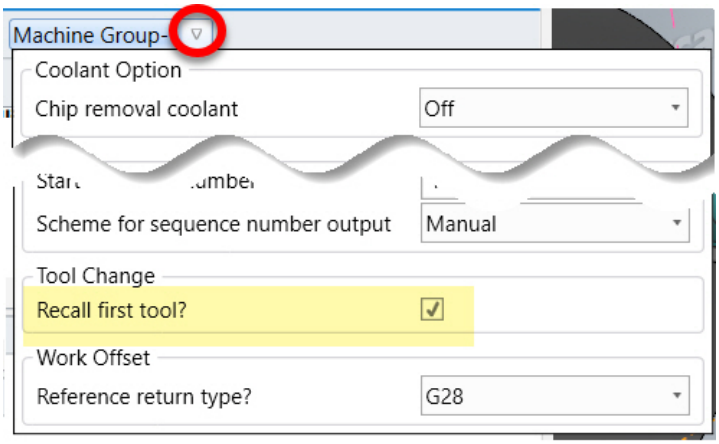
B. Pre-staging the first tool

Contents

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

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

To activate this feature, select the **Recall first tool** option from the machine group options in the Sync Manager.



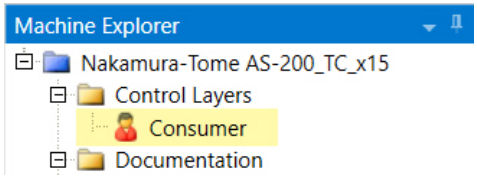
C. Configuring rapid moves

Your **.machine** file lets you choose what type of motion to use for rapid moves:

- Each axis moves at its maximum feed rate.
- All axes arrive at the programmed point at the same time.
- An interpolated move at the maximum feed rate.

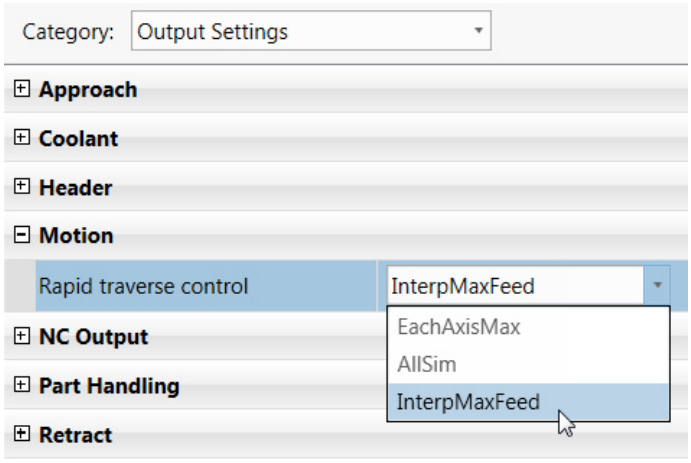
Follow these steps:

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



2. Go to the **Output Settings** category.
3. Open the **Motion** group.

4. Select the desired strategy.



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

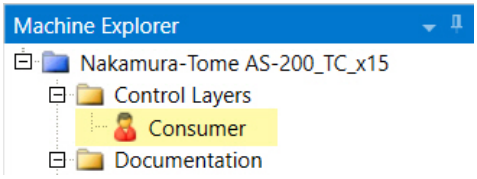
Contents

D. Setting the program number prefix

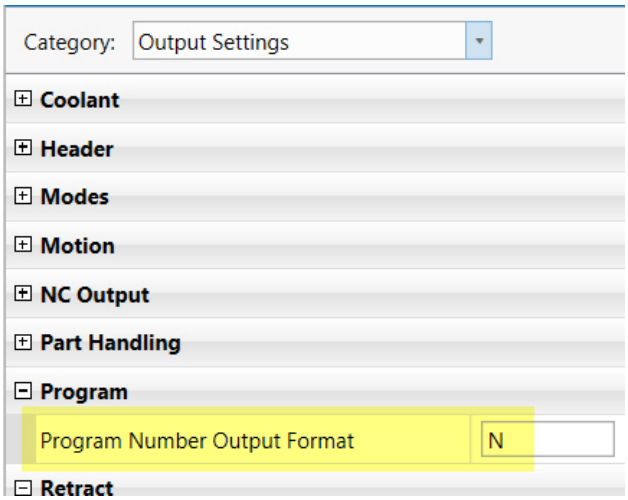
Contents

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

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



2. Double-click the **Consumer** layer.
3. Select **Category: Output Settings**.
4. Go to the **Program** section.
5. Enter the desired letter in the **Program Number Output Format** field.



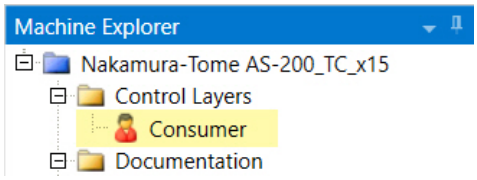
6. Press [**Ctrl+S**] to save the **.machine** file.

E. Sequence numbering

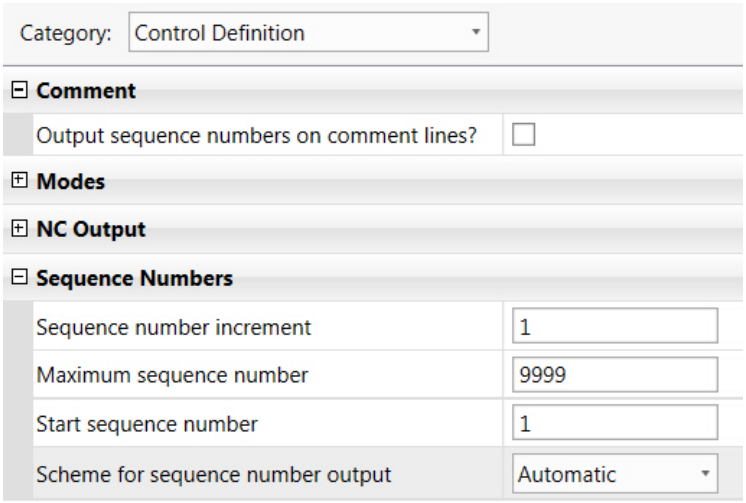
Contents

Follow these steps to set sequence number options:

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



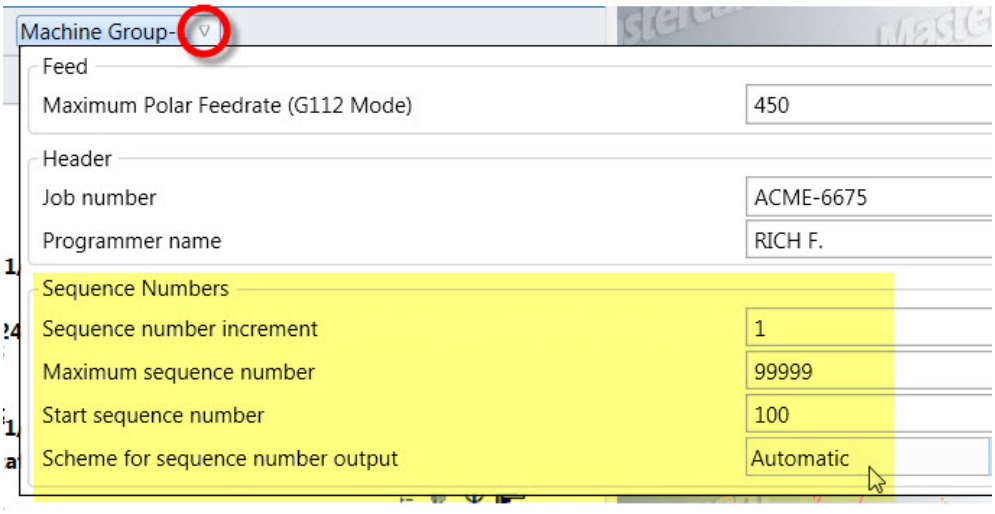
2. Go to the **Control Definition** category.
3. Open the **Sequence Numbers** group.



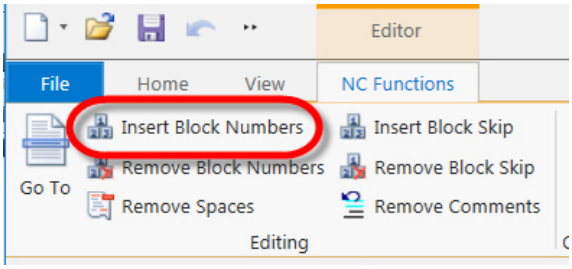
4. To turn on sequence numbering, select **Automatic**.
5. Enter your desired settings for starting number, maximum number, and increment.
6. Open the **Comment** group.
7. Select the check box to tell Mastercam to include sequence numbers on comment lines.
8. Press [**Ctrl+S**] before posting to save your settings.

Contents

You can change most of these settings for each part. Click the small triangle next to the machine group name and enter the desired values:



If you select the **Manual** option for sequence number output, use the **Insert Block Numbers** command in Code Expert to add sequence numbers after you post.



4. Approach and retract moves

Contents

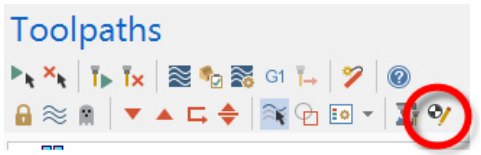
Your Nakamura-Tome AS **.machine** file gives you several options for configuring approach and retract moves:

- You can select the start point for the approach move and the end point of the retract. These are specified with reference positions.
- You can choose the type of motion for each: dogleg (Z-first or X-first) or a direct move.
- You can choose to output the home position move as a `G28` or `G53` move.

A. Selecting reference positions

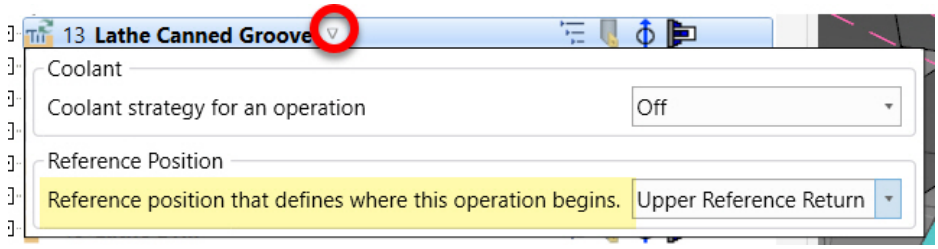
Your **.machine** file includes a set of reference positions that have been defined specifically for your individual machine. Use these to tell Mastercam where you want to start and end each operation. For example, you might not want to move all the way to the home position between operations. Sync Manager reference positions let you determine exactly where you want the 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 if your particular part setup requires reference locations different than those already defined in your **.machine** file—for example, to accommodate special fixturing, an unusual part shape, etc. Click the **Edit reference positions** button in Mastercam to create them.



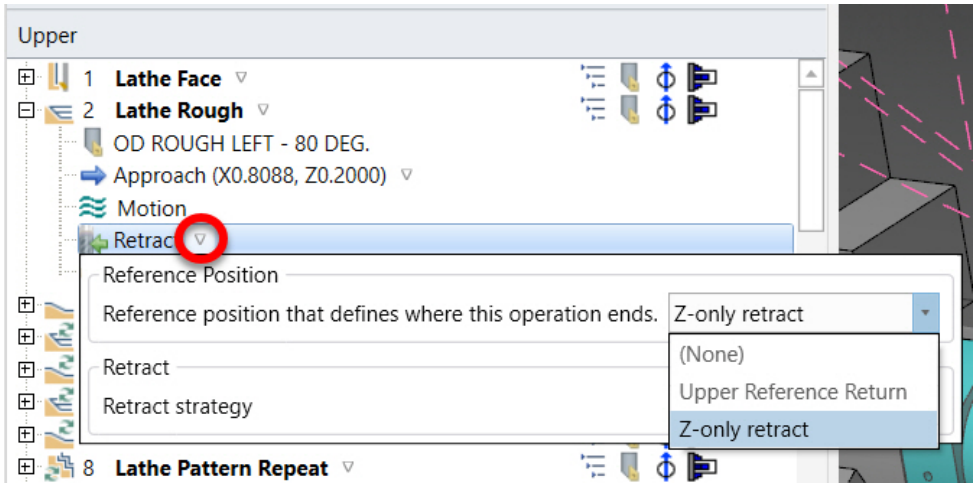
Setting the start point for an operation

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.



Setting the end point

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



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

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

```
Z-.4688
X1.2725 Z-.6206
X1.4139 Z-.5499
G00 G53
G53 Z3.495
T0300
M01
```

Contents

Selecting a reference position of None

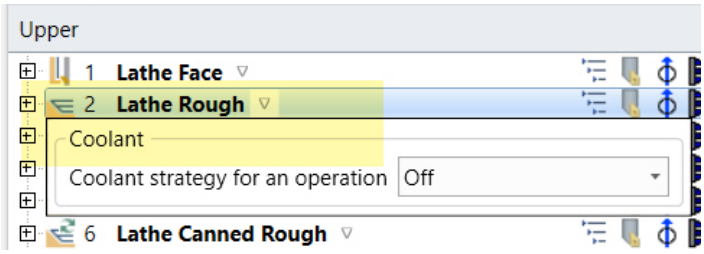
Selecting **None** for a reference position means that there will be no output where the reference position is normally output.

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

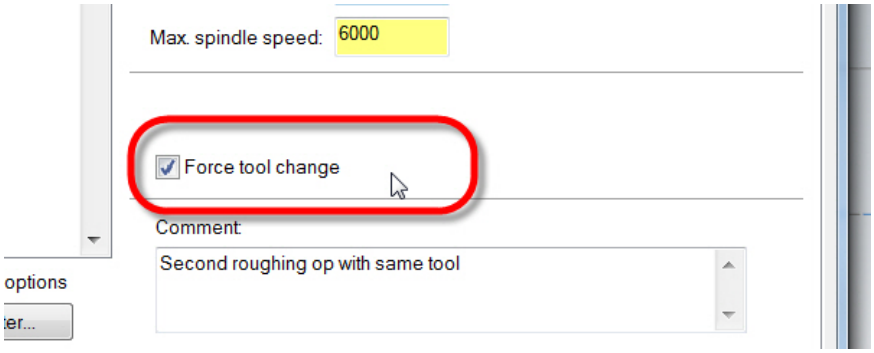
Reference positions and null tool changes

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

You can see in this picture that the **Reference position** option is not available for the start of Operation 2. That is because Operation 2 uses the same tool as Operation 1.



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



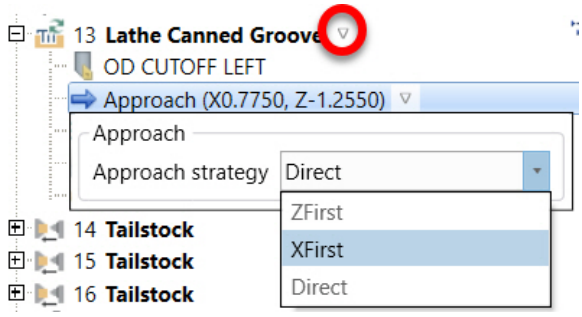
B. Setting the type of approach/retract motion

Contents

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

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

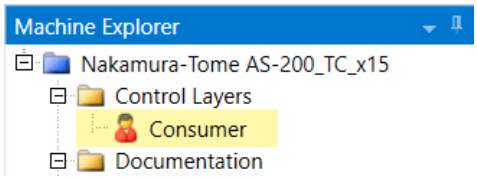
Click the small triangle next to the **Approach** or **Retract** node in the tree.



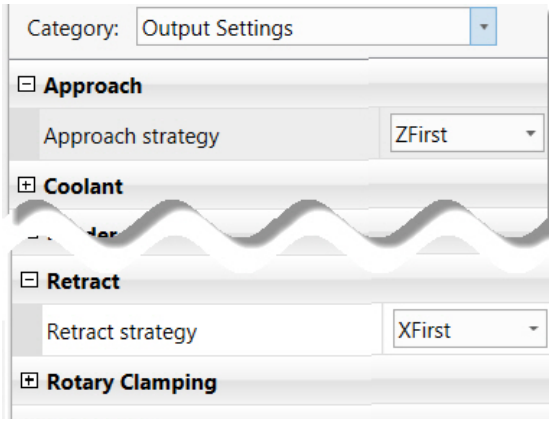
IMPORTANT: The recommended best practice for tailstock models is to select **Z First** for approach moves, and **X First** for retract moves. Selecting the other options results in a higher risk of collisions with the tailstock.

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.

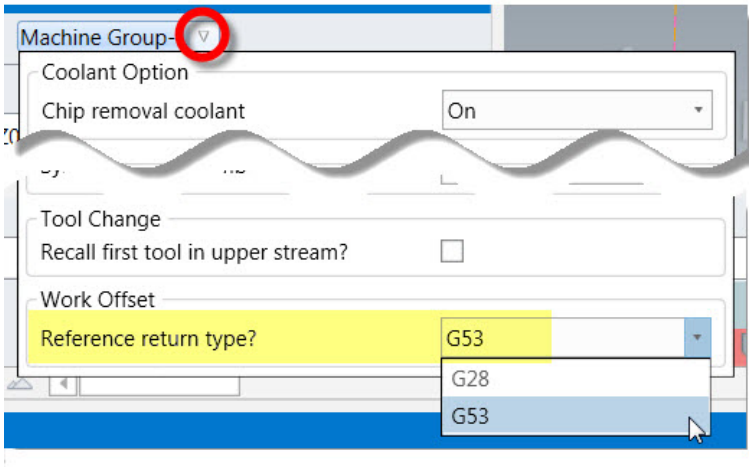


4. Select the desired strategy for each.
5. Press [**Ctrl+S**] to save your settings.

C. Setting G28 or G53 output

Contents

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



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

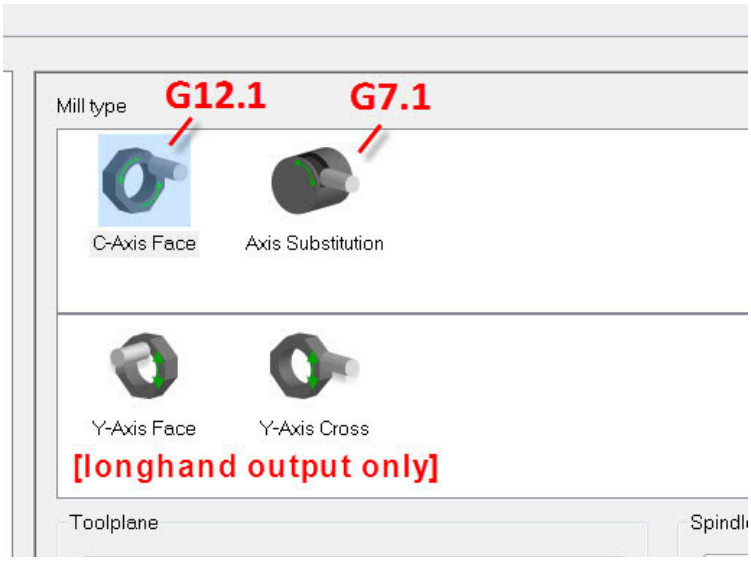
5. Milling cycles and modes

Contents

Your Nakamura-Tome AS **.machine** file supports the following milling cycles:

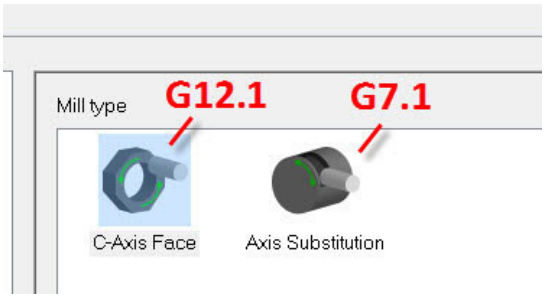
- `G7.1` (polar interpolation)
- `G12.1` (cylindrical interpolation)

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



Follow this general workflow for programming these cycles.

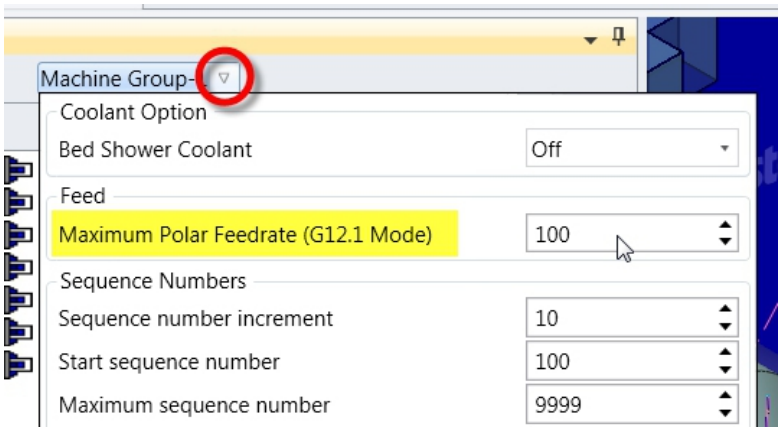
1. Select **C-axis Face** or **Axis Substitution** on the **Setup** page for milling operations.



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.

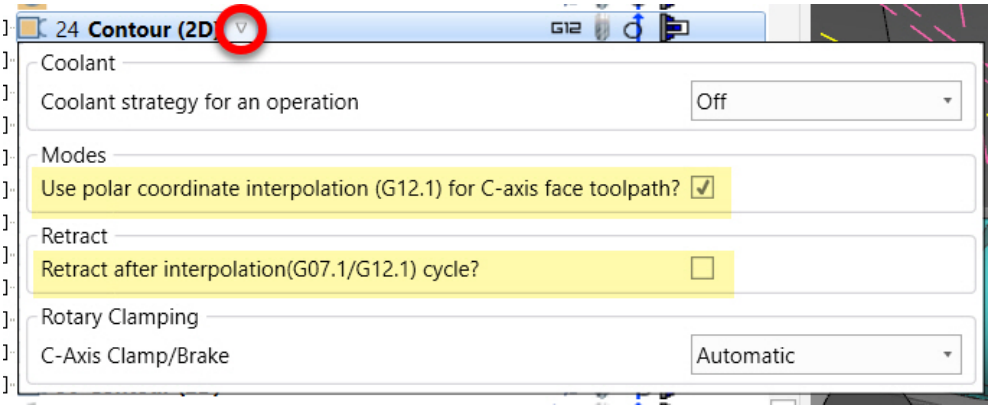
Contents

2. After you load the part in the Sync Manager, set the **Maximum Polar Feedrate**. This is a machine group option that applies to the entire part. Click the small triangle next to the group name and enter the desired value.



3. For each operation, choose whether or not to output the `G7.1/G12.1` cycle command or longhand output.

   - Click the small triangle next to the operation and select the cycle option if desired.
   - Leave it unchecked for longhand output.



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

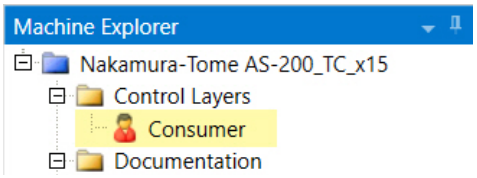
5. Press [**Ctrl+S**] to save the settings back to your part file.

A. Default settings for G7.1 and G12.1

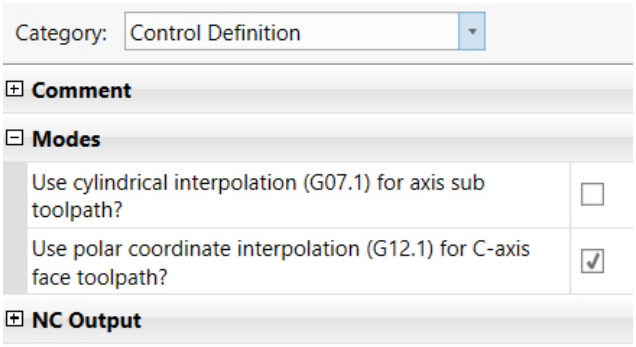
Contents

Follow these steps:

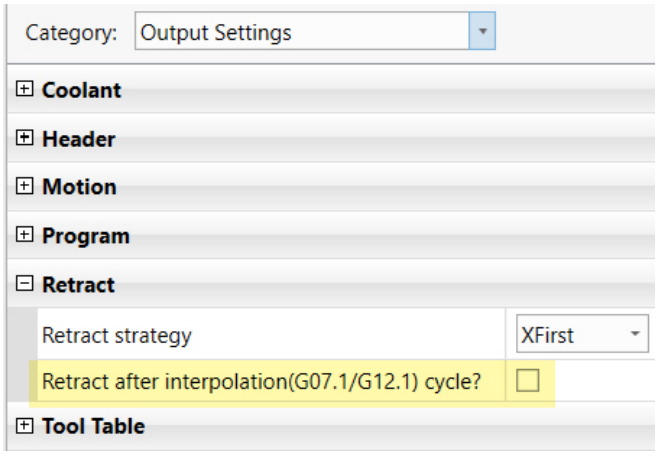
1. Open the **.machine** file in Code Expert.



2. Double-click the **Consumer** layer.
3. Select **Category: Control Definition**.
4. Go to the **Modes** section.



5. For each cycle, choose the default output mode. If you do not select the checkbox, the result will be longhand output.
6. Select **Category: Output Settings**.
7. Go to the **Retract** section.



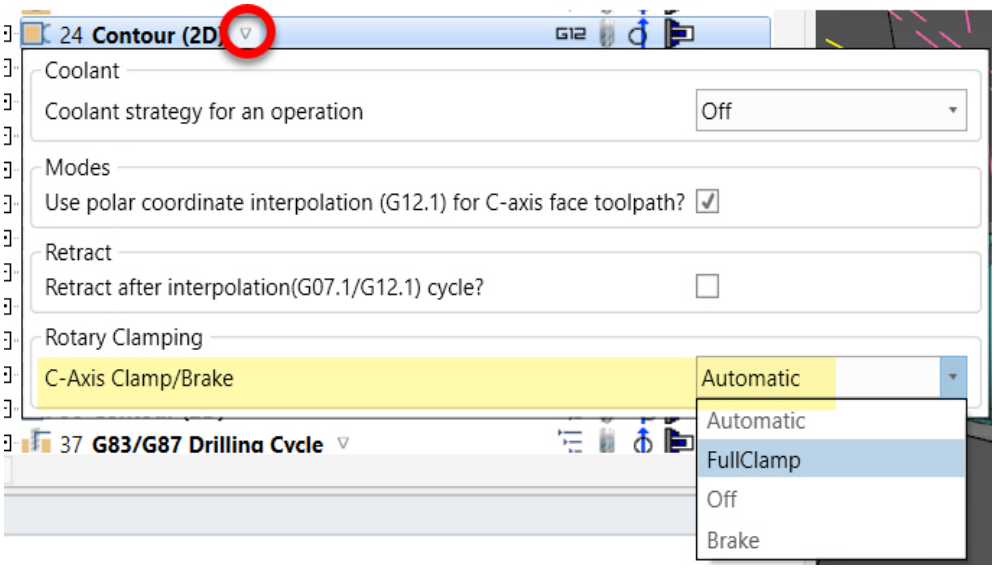
8. Select the **Retract after interpolation** option to have the post add a home position retract after the cycle has completed.
9. Press [**Ctrl+S**] to save the **.machine** file.

B. Rotary axis clamping mode

Contents

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

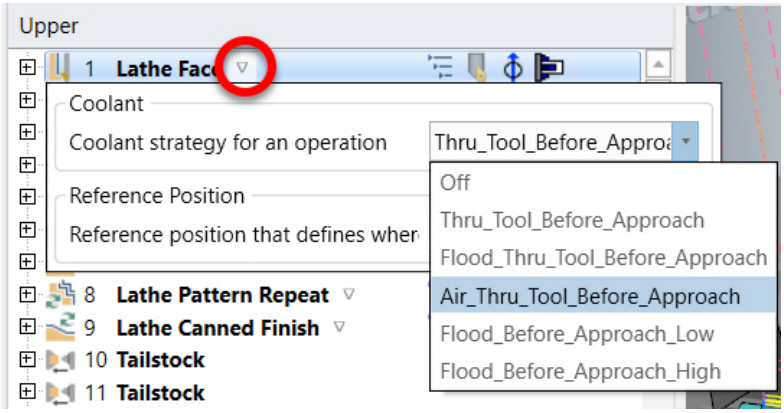
Set this option in the Sync Manager. Click the small triangle next to the toolpath name and select the desired **C-axis Clamp/Brake** mode:



6. Coolant

Contents

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 in Mastercam.



Your Nakamura-Tome AS .machine file supports the following coolant options:

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

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

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

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

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

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

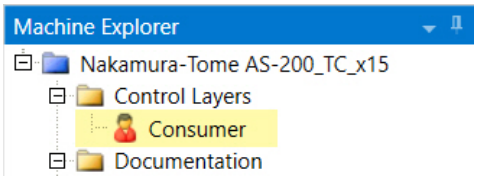
Flood Before Approach High—Turn on high-pressure flood coolant (M08) before the approach move. Coolant is automatically turned off before the retract.

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

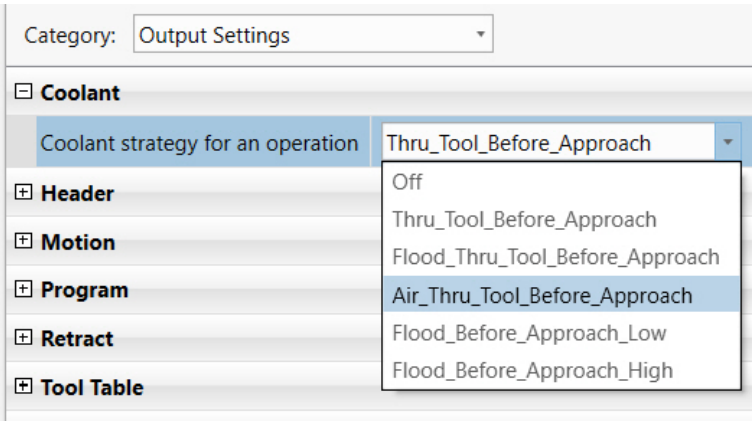
Contents

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

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



2. Double-click the **Consumer** layer.
3. Select **Category: Output Settings**.
4. Go to the **Coolant** section.
5. Select the desired default strategy.



6. Press [**Ctrl+S**] to save the **.machine** file.

A. Chip removal air blast (M706)

The chip removal air blast (`M706`) is controlled separately from the other coolant options. It is turned on at the beginning of the machining job. Click the small triangle next to the machine group name to turn it on. It is automatically turned off at the end of the program.