MASTERCAM MILL-TURN TUTORIAL

June 2017

POWERED BY MASTERCAM'S DYNAMIC MOTION™ TECHNOLOGY

Mastercam®
Be Dynamic.

MASTERCAM MILL-TURN TUTORIAL

June 2017
© 2017 CNC Software, Inc. – All rights reserved.
Software: Mastercam 2018

Terms of Use

Use of this document is subject to the Mastercam End User License Agreement. The Mastercam End User License Agreement can be found at:

http://www.mastercam.com/companyinfo/legal/LicenseAgreement.aspx

Be sure you have the latest information!

Information might have changed or been added since this document was published. The latest version of the document is installed with Mastercam or can be obtained from your local Reseller. A ReadMe file (ReadMe.PDF) – installed with each release – includes the latest information about Mastercam features and enhancements.

Table of Contents

Introduction **9**

General Tutorial Requirements 10

1 Introduction to Mill-Turn **11**

What is Mastercam Mill-Turn? 12

System Requirements 13

Getting Ready to Work 13

Preparing the Files 13

Loading a Missing Machine File 14

How the Tutorial is Organized 14

2 Transitioning to Mill-Turn from Lathe **17**

Running Over a Network 18

Replacement of Parameters that Differ in Mill-Turn 18

Menu Changes 19

Toolpaths Manager Changes 20

Toolbar Functions 20

Machine Group Properties (Files tab) 21

Machine Group Properties (Job Setup tab) 22

Communications 24

3 Work Flow Overview **27**

Load or Create Part Geometry 28
Load the Appropriate Machine 28
Complete Job Setup 28
Setup Type Page 29
WCS Page 30
Part Geometry Page 31
Bar Stock Page 32
Chuck Jaws Page (Left and Right Spindles) 33
Toolplane Origin Z Page (Left and Right Spindles) 33
StickOut Page 34
Pickoff Page 34
Finishing Job Setup 35
Create Toolpath Operations 36
Create Part-Handling Operations 37
Synchronize the Operations 38
Synchronizing Multi-Stream Jobs 38
Customizing Settings 38
Simulate the Operations 39
Post the Operations 40
4 Working with Machine Files 41
Installing Your Machine File 42

Customizing Your Machine File 42
Customizing Operation Defaults and Tool Libraries 44
Editing Machine File Settings and Defaults 45
Opening Code Expert in Multi-Stream View 47
Configuring the Find Extents Feature 48
Setting the Toolpath Directory 49
Managing Machine Licenses 52
Verifying that You Have the Proper License 52
Reactivating Disabled Licenses 55
5 Working with Tools and Spindles 57
Initial Tool Setup 58
Editing Tool Settings 59
Insert Up/Down and LH/RH Holders 60
Changing Insert Orientation 60
Switching Between LH and RH Holders 61
Specifying the Tool's Compensation Point 64
Selecting the spindle and turret 66
Selecting an Axis Combination 66
Setting Up Tools for Pinch Turning 67
6 Reference Positions 69
Exercise 1: Using Reference Positions 70

Exercise 2: Creating Reference Positions 75

7 Configuration and Job Setup 79

Adding a Machine File 80

Exercise 1: Adding Your Machine 80

Setting Up Your Job 81

Exercise 2: Selecting a Machine 81

Completing Job Setup 82

Exercise 3: Setting Setup Type and WCS 82

Exercise 4: Specifying Part Geometry 84

Exercise 5: Specifying Bar Stock Settings 86

Exercise 6: Setting Spindle and Chuck Parameters 87

Exercise 7: Defining the Pickoff Operation 90

8 Turning Toolpaths 93

Programming the Left Spindle 94

Exercise 1: The Facing Toolpath 94

Exercise 2: The Roughing Toolpath 96

Exercise 3: The Finishing Toolpath 99

Exercise 4: Simulating the Toolpaths 101

Exercise 5: Synchronizing the Toolpaths 104

The Part Handling Operations 108

Exercise 6: Creating Part Handling Operations 108

Exercise 7: Simulating the New Operations 110
Programming the Right Spindle 113
Exercise 8: The Facing Toolpath, Right Spindle 113
Exercise 9: The Roughing Toolpath, Right Spindle 117
Exercise 10: The Pinch-Turn Toolpath 120
Exercise 11: Synchronizing the Toolpaths 122
9 Milling Toolpaths 125
Exercise 1: Setting Up the Part 126
Exercise 2: Importing the Milling Operations 126
Exercise 3: Completing the Pocket Operation 129
Exercise 4: Completing the Facing Operation 134
Exercise 5: Completing the Rough Contour Operation 138
Exercise 6: Completing the Finish Contour Operation 139
10 More about Sync Manager 141
Sync Manager's Window 142
Exercise 1: Customizing the Display 143
Exercise 2: Customizing Operation Settings 145
Exercise 3: Using the Timeline 148
Exercise 4: Sync Manager and Mastercam Interactions 151
Exercise 5: Changing Data in Sync Manager 151
Exercise 6: Changing Data in Mastercam 154

11 More about Machine Simulation 157

Machine Simulator's Window 158

Exercise 1: Starting Machine Simulation 159

Exercise 2: Running a Simulation 160

12 Posting 167

The Editor's Window 168

Exercise 1: Posting a Part 169

Exercise 2: Using the Editing Functions 171

Exercise 3: Using the Syncs and Tools Functions 173

Exercise 4: Using the Right-Click Menu 175

Exercise 5: Sending the NC Code to Your Control 176

13 Loading Multi-Station Tool Locators 181

Exercise 1: Loading the Part and Processing the Toolpaths 181

Exercise 2: Loading a Tool Locator and Tool 184

Exercise 3: Loading the Remaining Tools in the Tool Locator 187

Exercise 4: Loading the Second Tool Locator and Live Mill Tools 188

Exercise 5: Simulating the Operations 191

Conclusion 195

Mastercam Resources 195

Contact Us 196

Introduction

This tutorial introduces you to the basics of Mastercam Mill-Turn. Mill-Turn is a large product with several components, each of which is covered in this tutorial. When you have completed the lessons included in this tutorial, you should have the knowledge you need to get started with this versatile product.

Tutorial Goals

- Become familiar with the Mill-Turn components.
- Learn the main differences between Mastercam Lathe and Mastercam Mill-Turn.
- Know the Mill-Turn work flow.
- Explore the contents and use of machine files.
- Understand tool setup and reference positions.
- Setup a Mill-Turn job.
- Create turning and milling toolpaths.
- Use Sync Manager and Machine Simulation effectively.
- Load tools into multi-station tool locators.

WARNING: Screen colors in the tutorial pictures were modified to enhance image quality; they may not match your Mastercam settings or the tutorial results. These color differences do not affect the lesson or your results.

Estimated time to complete this tutorial: 8 hours

General Tutorial Requirements

All Mastercam 2018 tutorials have the following general requirements:

- You must be comfortable using the Windows® operation system.
- The tutorials cannot be used with Mastercam Demo/Home Learning Edition. The Demo/HLE file format (`emcam`) is different from Mastercam (`mcam`), and basic Mastercam functions, such as file conversions and posting, are unavailable.
- Each lesson in the tutorial builds on the mastery of the preceding lesson's skills. We recommend that you complete them in order.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be access from the Mastercam 2018 workstation, either with the tutorial or in any location that you prefer.
- You will need an internet connection to view videos that are referenced in the tutorials. All videos can be found on our YouTube channel: www.youtube.com/user/MastercamTechDocs
- All Mastercam tutorials require you to configure Mastercam to work in a default metric or English configuration. The tutorial provides instructions for loading the appropriate configuration file.

1

Introduction to Mill-Turn

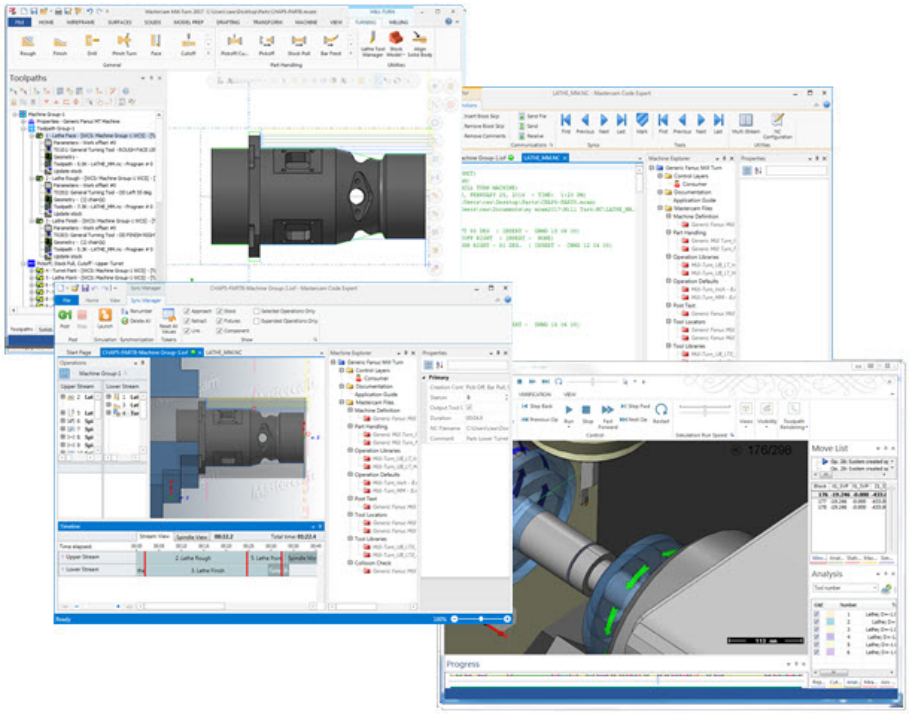
This tutorial presents an overview of Mastercam Mill-Turn and teaches the basics of preparing, simulating, and posting a Mill-Turn part. Along the way, you learn about the applications that make up Mill-Turn, including not only Mastercam, but also Code Expert.

Lesson Goals

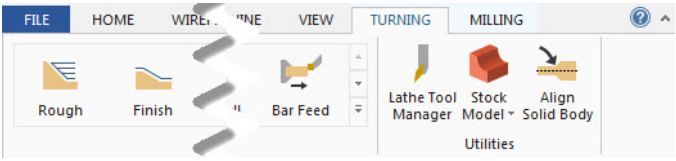
- Understand the applications that make up Mill-Turn.
- Prepare the tutorial files.
- Learn how to load machine files.

What is Mastercam Mill-Turn?

Mastercam Mill-Turn is a collection of functions, toolpaths, and applications. These Mill-Turn elements work together to create synchronized G-code for cutting parts on a multitasking machine.



You begin the Mill-Turn work flow in Mastercam, where you load or create the part to cut. You then load a Mill-Turn machine, which adds Mill-Turn operations to Mastercam's interface (see the following picture). These operations include not only milling and turning toolpaths, such as facing, pocketing, and pinch-turn, but also part-handling operations for programming pickoffs, cutoffs, bar feeds, spindle movement, and more.



When you have designed and toolpathed your part, you continue work in Code Expert, which is home to sync manager, editor, and it is also where you launch MT simulation. In Sync Manager you synchronize operations, in Editor you edit NC files, and in Machine Simulation, you watch your operations in action.

System Requirements

Up-to-date system requirements are available for Mastercam online: www.-mastercam.com/en-us/Support/System-Requirements.

An internet connection and e-mail access is recommended for support and updates. Visit www.mastercam.com for more information.

Getting Ready to Work

Now that you know what Mill-Turn can do, it is time to prepare your materials for the remainder of the tutorial.

Preparing the Files

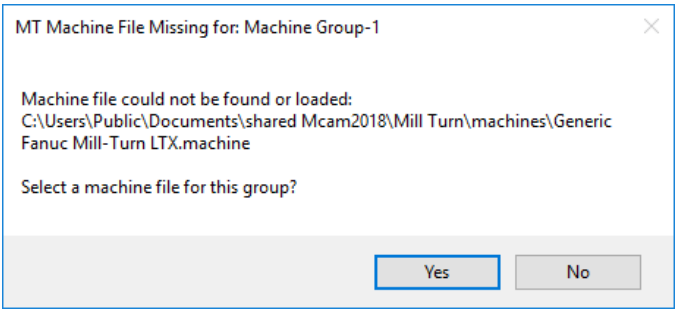
1. Place the part files included with this tutorial in a convenient location on your hard drive.
2. Place a copy of `Generic Fanuc Mill-Turn LTX.machine` (also included with this tutorial) in your `C:\Users\Public\Documents\shared Mcam2018\Mill Turn\machines` directory.

Loading a Missing Machine File

Mill-Turn part files store a reference to their machine file. If the machine file is missing or moved, Mastercam asks you to re-associate the machine with the part. The following procedure shows how to complete this important task. (You do not need to do this now. You only need to do this if your part cannot find its machine file.)

1. Open the part file with Mastercam.

   The MT Machine File Missing dialog box appears, as seen below.



2. Click **Yes**.

   The MT Machine Files dialog box appears.

3. Navigate to the machine file for your part (in the case of this tutorial, the machine file is `Generic Fanuc Mill-Turn LTX.machine`), and select it.

   Code expert loads the machine file associated with the part file.

4. *Optional* : Save the part file under a new name, so that you do not overwrite the original.

How the Tutorial is Organized

This tutorial's lessons introduce the concepts, applications, and work flow that make up Mastercam Mill-Turn. Because the exercises in each lesson are meant to be com-

pleted in order, we highly recommend that you work through this tutorial from the beginning to the end rather than skip from one lesson or tutorial to another.

The following list describes each of this tutorial's lessons:

Lesson 1, Introduction to Mill-Turn introduces the Mill-Turn product and offers guidelines for using this tutorial.

Lesson 2, Transitioning to Mill-Turn *from Lathe* describes the differences between Mastercam Mill-Turn and Mastercam Lathe, including network issues, file compatibility, and user-interface changes.

Lesson 3, Workflow Overview is a general road map for successfully working with Mastercam Mill-Turn. Topics covered include part geometry, Mill-Turn machine solutions, job setup, toolpaths, and more.

Lesson 4, Working with Machine Files, describes how to install, edit, and customize machine files, as well as how to configure Code Expert Editor and manage Mill-Turn licenses.

Lesson 5, Working with Tools and Spindles, explains how to set up tools and specify axis combinations.

Lesson 6, Reference Positions demonstrates how to create and use reference positions to specify where to start and end operations.

Lesson 7, Configuration and Job Setup presents detailed exercises for preparing a Mill-Turn part, including specifying a machine and completing job setup.

Lesson 8, Turning Toolpaths features step-by-step procedures for creating and synchronizing turning toolpaths and part-handling operations on the left and right spindles of your machine.

Lesson 9, Milling Toolpaths features step-by-step procedures for creating and synchronizing milling operations, including facing, pocketing, and contour toolpaths.

Lesson 10, More about Sync Manager provides an overview of Sync Manager's user interface and functionality. This lesson also includes a discussion of Sync Manager's interaction with Mastercam.

Lesson 11, More about Machine Simulation provides an overview of Machine Simulation's user interface and functionality.

Lesson 12, Posting is an overview of Code Expert Editor, including posting, editing, and navigating an NC file. This lesson also presents a procedure for sending a file to a machine control.

Lesson 13, Loading Multi-Station Tool Locators describes how to load tool locators onto a machine's turrets, as well as load tools into the multi-station locators. The complete sample program demonstrates many tool locator and tool combinations.

2

Transitioning to Mill-Turn from Lathe

Before you start work with Mill-Turn, you must understand how it differs from pre-X7 Mastercam. This lesson provides the information you need to begin using Mill-Turn effectively.

Lesson Goals

- Use a machine on a network.
- Update Lathe parameters.
- Understand changes in the menus.
- Review changes in Toolpaths Manager.
- Explore communication settings.

Running Over a Network

A machine solution (represented by a `MACHINE` file) includes the post processor and other files that Mastercam needs to run Mill-Turn. This means that managing Mill-Turn over a network is simplified. Because Mastercam can access the machine file from any network location, you can move the file where you wish. You do not need to worry about the links between the machine solution's components.

When you load a Mill-Turn part, Mastercam looks for the machine file and reloads it. If the machine solution's machine file cannot be found—either because it has been moved or renamed—you are prompted to find and select it. This interaction is different from earlier versions of Mastercam, which do not look for the machine definition when you load your part, but relies on the copy stored in your part file. Mill-Turn requires that your machine file be available whenever you open your part. This ensures that if your machine file (or post) has changed, your part will stay up-to-date.

Replacement of Parameters that Differ in Mill-Turn

Many options on the Lathe **Toolpath parameters** dialog box (highlighted in this picture) are not present in Mill-Turn:

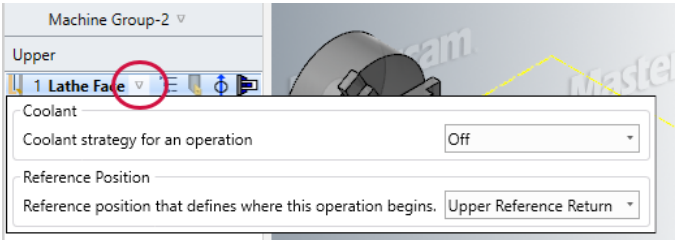
The following picture shows the same dialog box in Mill-Turn:

In Mill-Turn, these functions are all machine-specific features that are configured in the machine file.

- Coolant options are selected in Sync Manager. Instead of selecting individual options, you select strategies that combine several different coolant events. Your machine developer defines these strategies based on options that are available on your machine.

- Mastercam reads home positions from the machine file. You select approach and retract strategies in Sync Manager.
- Misc values and canned text do not exist in Mill-Turn. You access machine-specific options and parameters in Sync Manager.

The following picture shows how coolant appears in Sync Manager. Click the small inverted triangle next to each operation or segment to see what options are available.



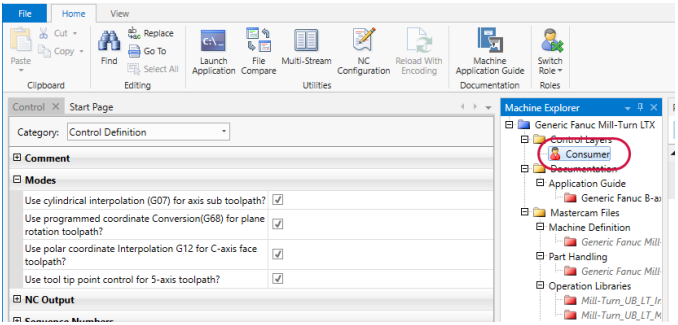
Unlike Lathe, where you are limited to 20 miscellaneous ints and reals, Mill-Turn has no limit on the number of machine-specific options. Plus, the options can be specific to individual operation types or parts of an operation. Also, while the previous misc values are limited to numbers, toolpath options in Mill-Turn can be presented as numbers, checkboxes, list selections, or strings.

Menu Changes

Mill-Turn organizes the single Toolpaths menu from older versions of Lathe into different contextual tabs, as follows:

- The Turning tab contains Lathe operations, including part handling (formerly Misc Ops).
- The Milling tab contains 2D, 3D, and Multiaxis Mill operations.

In addition, the Machine Definition Manager and Control Definition Manager are disabled. You can access many of their settings in Mill-Turn by opening the machine file in Code Expert, and double-clicking **Consumer** in Machine Explorer.



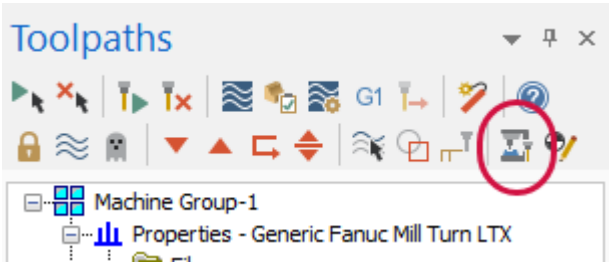
Toolpaths Manager Changes

The Toolpaths Manager for Mill-Turn operations has important differences from that with non-Mill-Turn operations.

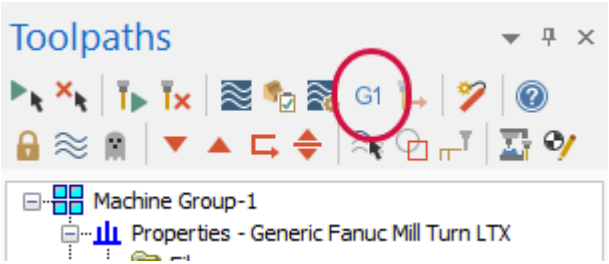
Toolbar Functions

The Tool Setup button on the Toolpaths Manager toolbar is used exclusively by Mill-Turn, whereas the G1 button has a new function in Mill-Turn:

The **Tool Setup** button lets you configure how tools are loaded on your machine.

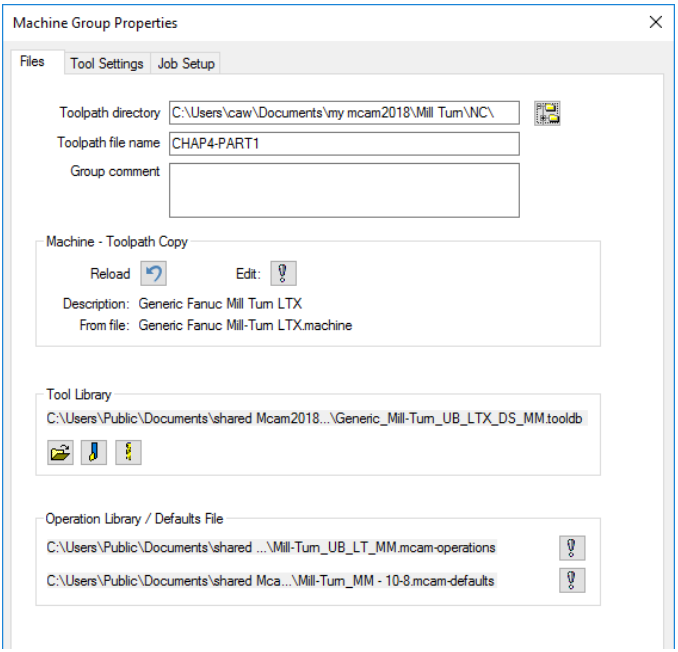


The **G1** button opens Sync Manager (only in Mill-Turn). From Sync Manager, you can create syncs, configure toolpath options, post operations, and launch Simulation.



Machine Group Properties (Files tab)

Some options in the Files tab have been removed for Mill-Turn users, since Mill-Turn stores the individual support files in the machine file.

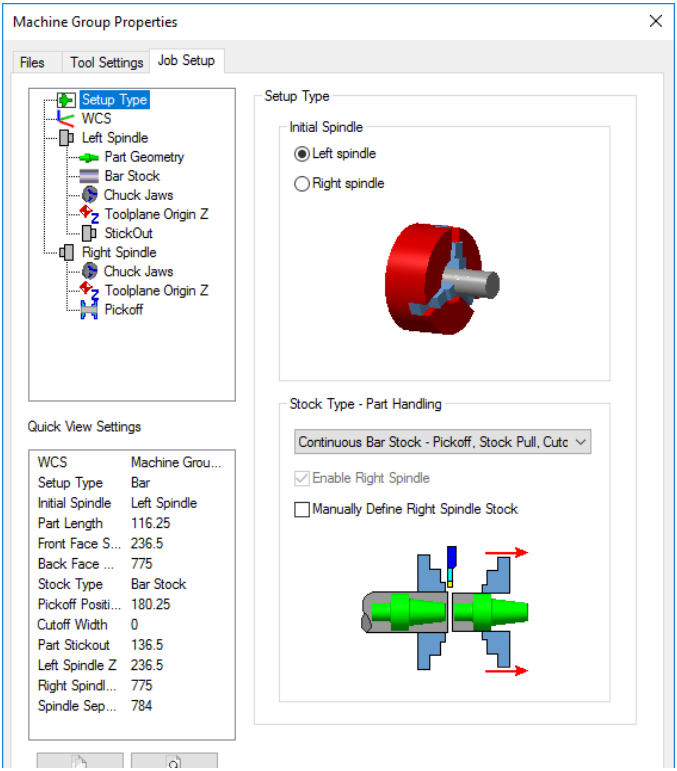


- Although a default tool library is stored in the machine file, you can still select a different one here.
- When creating individual operations, you can always select a tool from any library, just like in Mill and Lathe.
- You can also import operations from any library, just like in Mill and Lathe. The operations library listed on this tab is only a default selection.

While in non-Mill-Turn Mastercam, you can use the **Replace** button to select a different machine definition, in Mill-Turn you cannot do this. You can, however, use the **Edit** button to make changes to the current machine, if necessary.

Machine Group Properties (Job Setup tab)

The **Job Setup** tab is new for Mill-Turn. It replaces the **Stock Setup** tab from Mastercam and is available only in Mill-Turn.



Although the Job Setup tab seems to recreate much of the functionality of the Stock Setup tab in Lathe, it is more closely tied to your work flow.

Follow this general work flow when creating your parts:

1. **Create your part geometry.** The Mill-Turn work flow proceeds more smoothly if you minimize later changes to your part.
2. **Create a solid model of your finished part.** A solid model is not required, but if you have one, the Machine Simulation and Sync Manager functions display it with your toolpaths.

3. **Select your machine file from the Machine tab.** The Job Setup function launches automatically.
4. **Complete the Job Setup pages in the order presented**.

When you complete Job Setup, Mastercam automatically creates resources that help you program your part. The following list describes these resources.

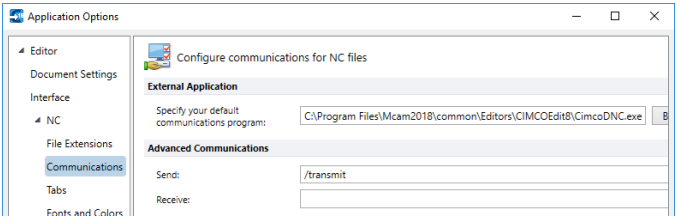
- *A custom WCS*—This is a copy of WCS Top, whose origin is located at the machine zero location from your machine file.
- *New views within the WCS*—These views correspond to standard machining orientations and applications, for programming both left and right spindles. Each view is created with the proper origin and work offset from your Job Setup settings. For example, in Job Setup you can choose to locate Z0 at either the front face or back face of your part; Mastercam then automatically creates tool planes with the desired origin.
- *Viewsheets for each spindle*—Each viewsheet contains a copy of the part geometry properly positioned relative to the chuck jaws.

You learn much more about the individual Job Setup pages when you complete the exercises later in this tutorial.

Communications

In Mill-Turn, you use Code Expert to configure your communications settings.

1. Open Code Expert from your Start menu (**Start, All Programs, Mastercam 2018, Code Expert**). If you already have a Mill-Turn machine loaded, Code Expert will already be open.
2. Select **Options** from the File tab.
3. Select **Editor, NC, Communications**.



4. Set your options as desired. Use **Help** to learn more.

Mill-Turn uses the same XML-based configuration files as previous Mastercam versions, so if you have been using a customized file, you can use it in Mill-Turn. You can also use the same communications program.

3

Work Flow Overview

Mastercam Mill-Turn introduces a streamlined work flow for creating the operations to cut your part. This lesson presents an overview of this work flow. The following lessons provide more detail and hands-on exercises.

The Mill-Turn work flow comprises the following steps. The remainder of this lesson explains each step in more detail.

Lesson Goals

- Load or create part geometry
- Load the appropriate machine
- Complete job setup
- Create toolpath operations
- Create part handling operations
- Synchronize the operations
- Simulate the operations
- Post the operations

Load or Create Part Geometry

Your first step with Mill-Turn is to create your part's geometry. Often, this means simply loading the part's file, although you can use Mastercam's CAD tools to create the part's geometry or to modify an existing part.

Load the Appropriate Machine

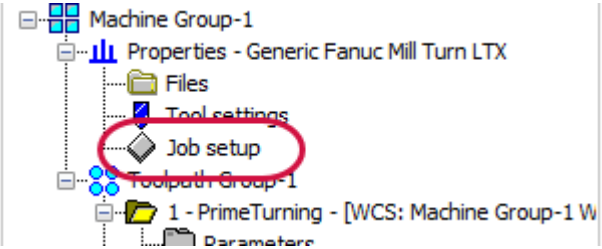
Before you can create Mill-Turn operations on your part, you must load a Mill-Turn machine. Choose **Machine, Mill-Turn**, and then select the machine file you need.

The machine file is a group of files stored as a unit. These files include the machine definition, tool database, and other data that non-Mill-Turn Mastercam stores in separate files.

Complete Job Setup

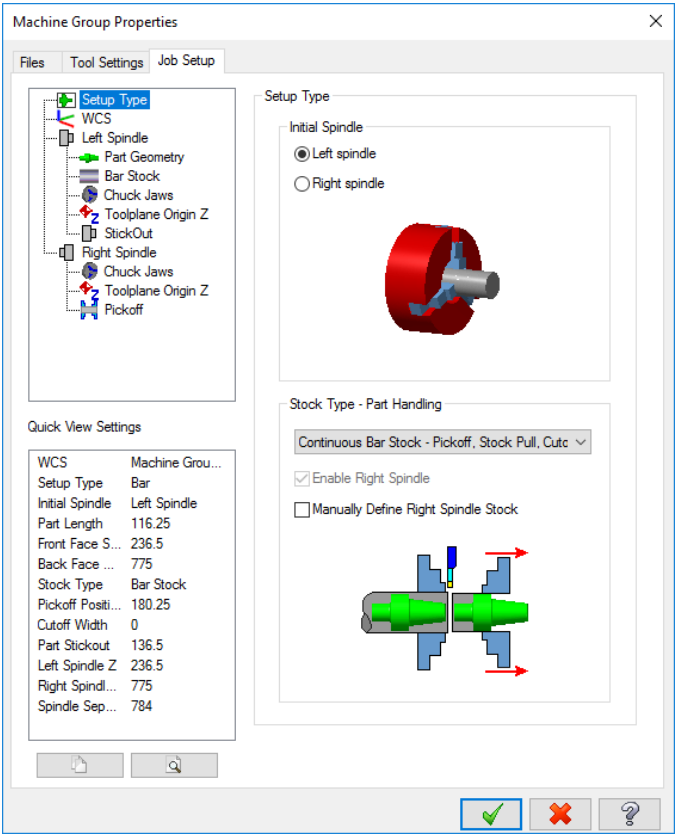
When you choose a Mill-Turn machine, Mastercam not only loads the machine, but also starts Code Expert, in which you complete many Mill-Turn tasks.

After the machine file and Code Expert load, Mastercam displays the Job Setup tab of the Machine Group Properties dialog box. (You can also open Job Setup at any time from the Toolpaths Manager, as shown below.) This tab is where you define the general parameters for your job. Job Setup arranges these parameters in work flow order from top to bottom. The following sections describe each page of parameters.



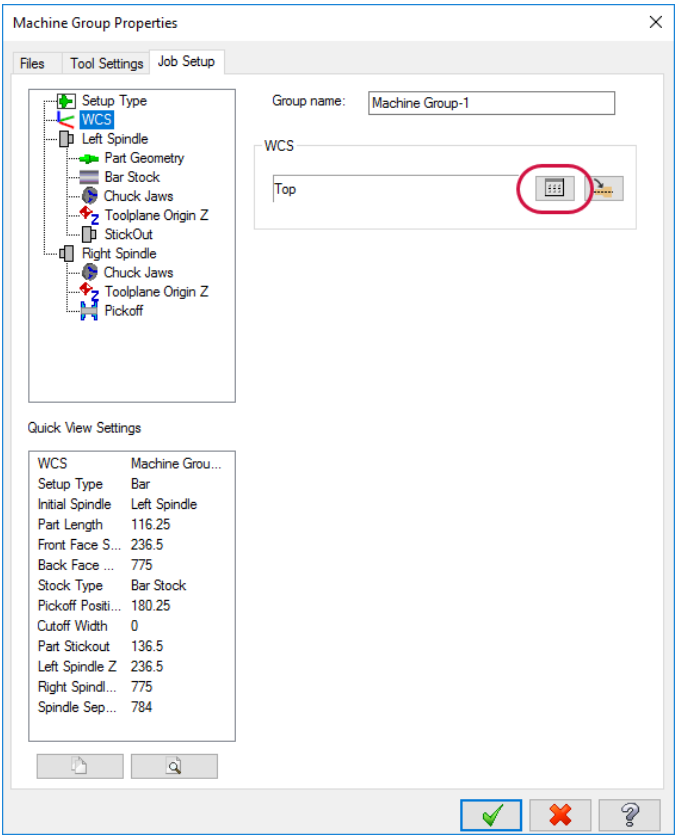
Setup Type Page

Use this page to set the spindle on which your job will begin. Here, you also specify your stock type (either continuous bar or single piece) and how to handle the parts as they are created. Stock type determines which part handling operations—such as pick-offs, cutoffs, and stock pulls—are available.



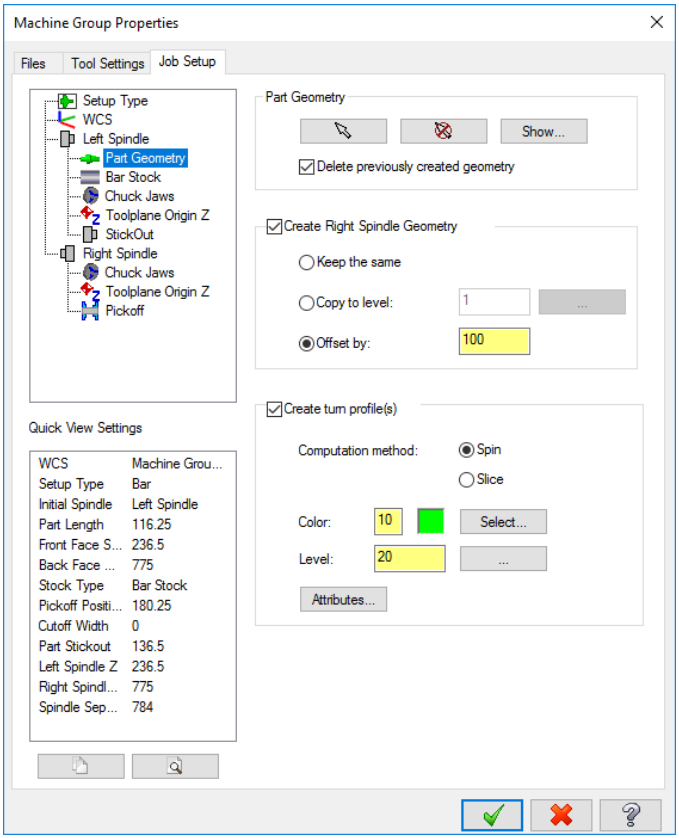
WCS Page

Use this page to define your Work Coordinate System (WCS) by clicking the **Select Named Plane** button (circled in the image below) and choosing the initial WCS from the Plane Selection dialog box. Use the WCS option when importing geometry that is aligned incorrectly for machining, orienting the part on the machine differently from how it is oriented in Mastercam. On this page, you can also change the machine group's name, which appears in the Toolpaths Manager.



Part Geometry Page

Use this page to select your part geometry and tell Mastercam how to create geometry for the other spindle or for a turn profile. You can also select the color and specify the levels on which Mastercam creates the geometry.

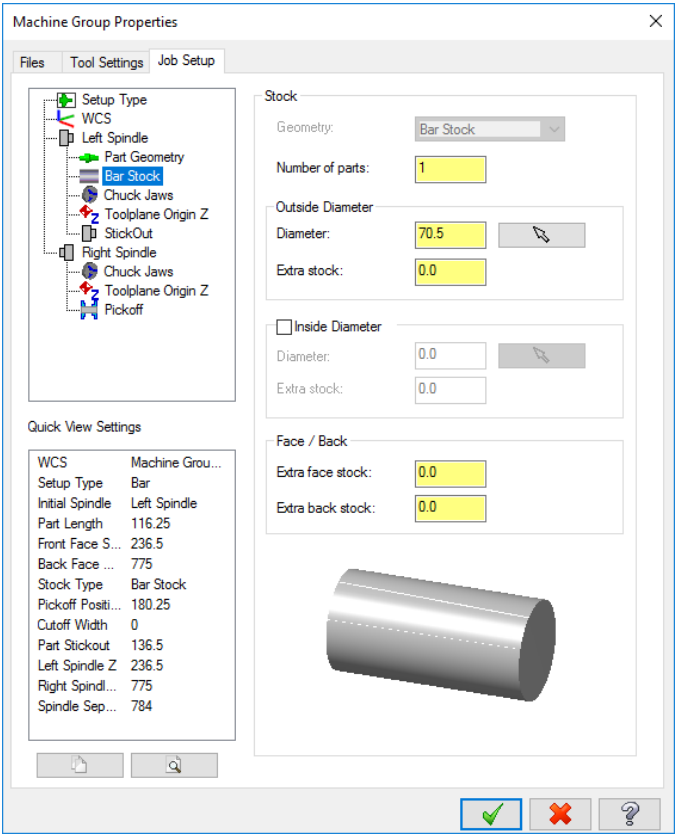


If **Create Right Spindle Geometry** is selected, Mill-Turn creates the part geometry in both spindles. If **Create Right Spindle Geometry** is off, Mill-Turn shows the part geometry in only the starting spindle. Having the part geometry in both spindles lets you program the part on both the starting and opposing spindles.

Bar Stock Page

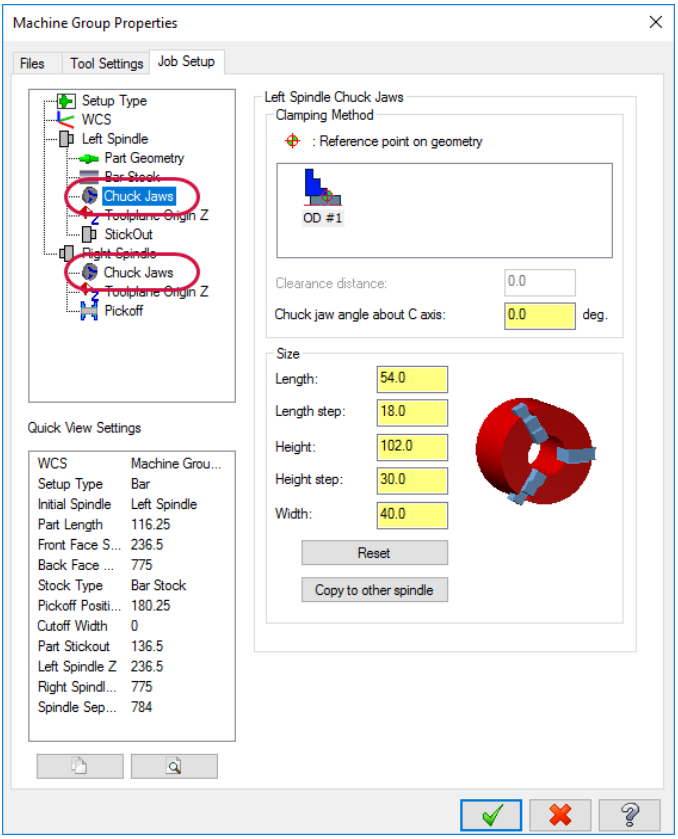
Use this page to select the type of stock and its dimensions. You can also specify the number of parts to create and the amount of material to leave for other operations. Extra face and back stock are commonly used to leave stock for facing operations and affect the part transfer cutoff positions.

Note: Mill-Turn supports different stock types, with the stock page changing depending on your setup type.



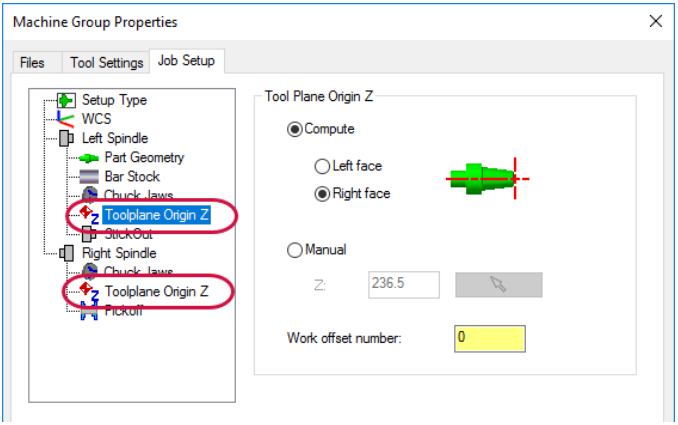
Chuck Jaws Page (Left and Right Spindles)

Use these pages to set chuck parameters for each spindle, including the clamping method, chuck dimensions, and part stickout.



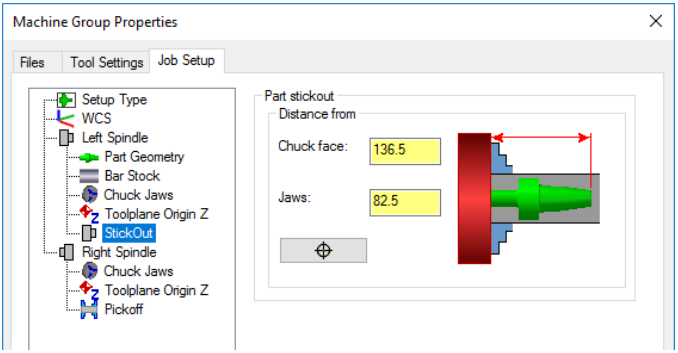
Toolplane Origin Z Page (Left and Right Spindles)

The Toolplane origin sets the zero point for operations on each spindle. Mastercam can calculate the value from the part geometry, or you can enter it yourself.



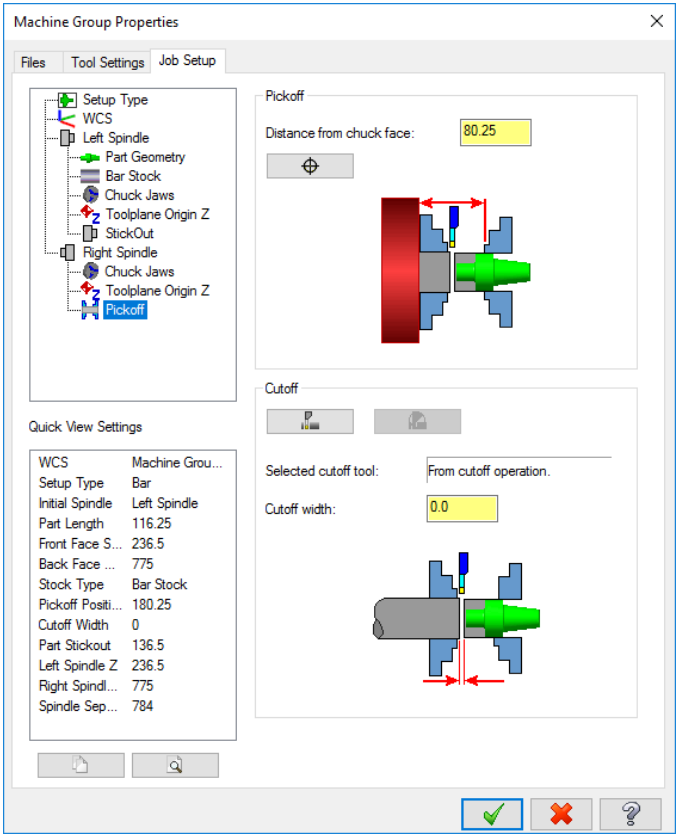
StickOut Page

Use this page to specify how much of the part sticks out from the face or jaws of the spindle. You can enter the values manually or choose a location in the graphics window.



Pickoff Page

Use this page to set pickoff and cutoff parameters. These include the pickoff location, the cutoff tool, and the cutoff width. Note that Mill-Turn uses the cutoff width to adjust the cutoff positions.



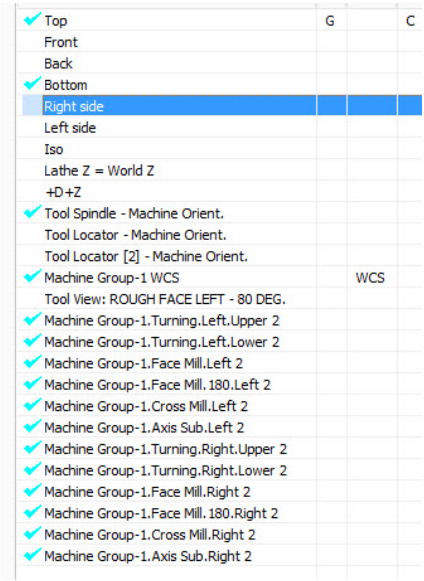
Finishing Job Setup

When you have finished setting up your job, click **OK** in Job Setup to finalize your choices. Mastercam creates your machine group, plus a set of convenient resources:

- The new WCS
- Viewsheets for each spindle
- Default toolplanes

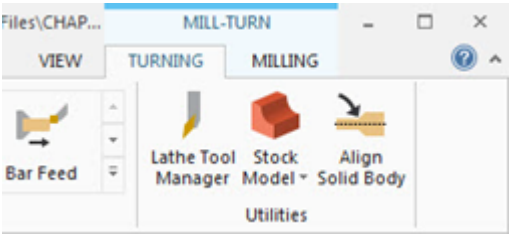
The viewsheets, subspindle geometry, and the revolved part profile make it easy for you to program operations on the subspindle.

Mastercam creates planes that correspond to standard machining orientations for different types of operations in each spindle, with origins keyed to the locations that you selected on the Job Setup pages.



Create Toolpath Operations

With job setup completed, you can create the operations to cut your part. Choose between turning and milling operations, each tab having its own galleries in Mastercam's Mill-Turn tab. (Note that this tab appears only when you have loaded a Mill-Turn machine file.)



Creating an operation means choosing not only a specific toolpath, such as facing or roughing, but also providing information about how your machine will perform the operation. For example, as part of the toolpath operation, you choose the cutting tool and specify the axis combination (the pairing of a spindle and a turret).

Create Part-Handling Operations

Next, you tell Mastercam how to handle the part using pickoff and cutoff operations. You choose these transfer operations from Mastercam's Part Handling functions on the Turning tab, whose available operations depend on the type of stock you have specified. For example, single-piece stock requires only a pickoff, whereas continuous bar stock requires a cutoff and a pickoff. The part handling functions also include such operations as bar feeds, stock pulls, and turret parking.

Note that part-handling operations are not limited to finished parts. They can be used for many reasons, including bar feeding, pickoff ops where the opposing spindle supports the part, and other custom routines written by the machine solution provider.

The Pickoff, Cutoff, and Stock Pull operations are complete strategies with predefined lists of sub-operations, relieving you from having to create these sub-operations one by one, although you can modify them as needed. Choosing one of these strategies adds a group of operations to Mastercam's Toolpaths Manager.

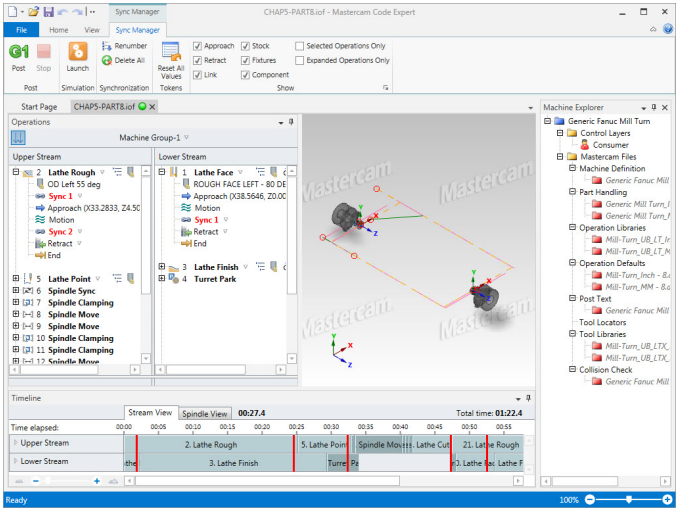
The resulting operations in the part handling group cannot be moved or deleted individually; the entire group must be deleted. This restriction allows the machine solution provider to code the post logic to handle operations in specific fashions required by the machine tool.

Synchronize the Operations

Sync Manager is a visual tool that creates sync points for multi-stream machines. It also helps you create—for both single- and multi-stream machines—custom job setups that ensure valid output.

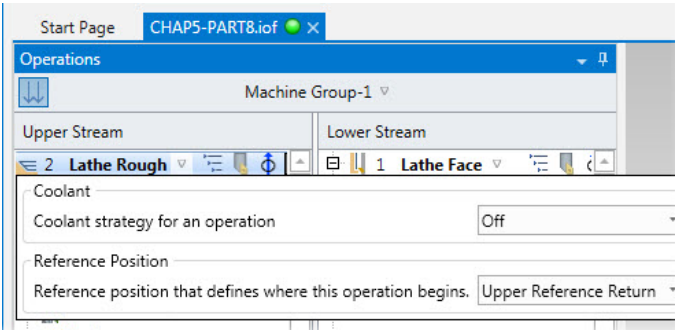
Synchronizing Multi-Stream Jobs

After you create your operations in Mastercam, run Sync Manager (shown in the following picture) from the Toolpaths Manager, and synchronize the operations on your machine. Use your mouse to drag-and-drop operations between streams to ensure that each can complete successfully without collisions or other problems.



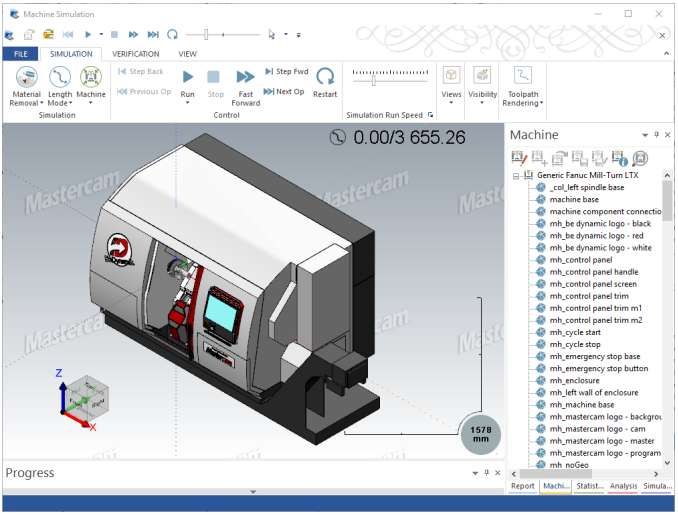
Customizing Settings

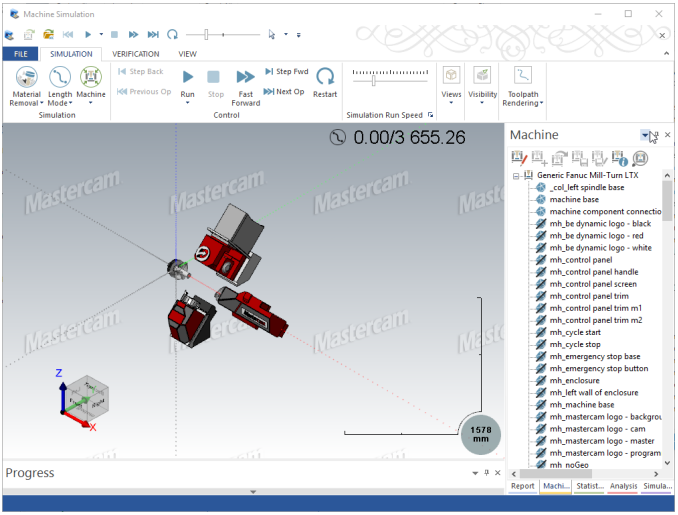
Sync Manager provides access to many settings, including coolant strategies, reference positions, and options traditionally stored as misc values in previous Mastercam versions. You can modify these settings by clicking the arrow associated with the option to change, and typing the required values into the provided fields.



Simulate the Operations

Use Mill-Turn to simulate the machine's motions as it cuts your part. By zooming or setting the visibility of specific components, you can customize the view to your needs. For example, the first picture below shows Machine Simulation's display in isometric view, and the second picture shows the same job with the machine housing turned off.





Post the Operations

When you have validated your job, you post the operations, which creates the G-code that you send to your machine to cut the part. After it is generated, the G-code displays in the editor, from which you can access the communications functions needed to transmit the code to your machine.

4

Working with Machine Files

This lesson provides basic information on working with machine files, which drive your entire Mill-Turn experience. A Mill-Turn machine file (which has the `MACHINE` extension) differs from the machine and control definition files used in non-Mill-Turn Mastercam.

Lesson Goals

- Installing your machine file
- Customizing your machine file
- Customizing operation defaults and tool libraries
- Editing machine file settings and defaults
- Configuring Code Expert Editor
- Setting the toolpath directory
- Managing machine licenses

Installing Your Machine File

Your machine file is packaged in a `ZIP` file with the PDF version of the machine's application guide. Simply unzip the machine file to your desired location. The default location is `\shared Mcam2018\Mill Turn\MACHINES`. However, you can place the machine file anywhere you wish.

Unlike non-Mill-Turn Mastercam machines, a Mill-Turn machine file includes all the resources needed to support your machine application. That is, the machine file does not need to link to other files, such as posts.

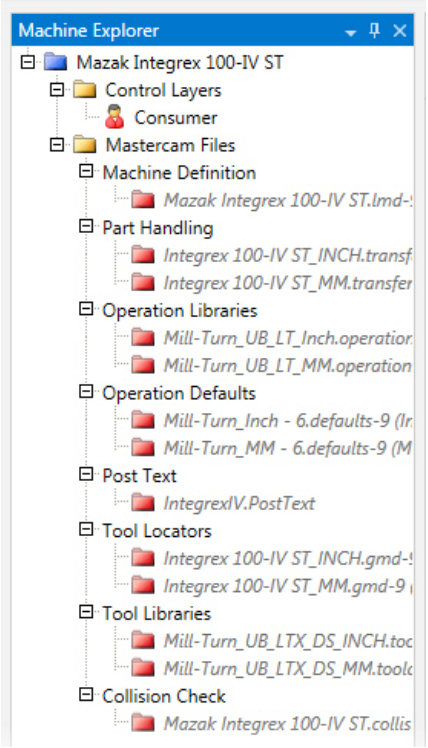
To work with your machine file from a network location, copy it to the desired location on your network. Since the machine file points to no other files, it works the same from a network location or from a local drive.

Note, however, that Mastercam looks for the machine file when you load a part that uses it, so the machine file must be in a location to which your workstation regularly connects.

Note: This guide comes with generic machine files that you can use to try Mill-Turn. You use one of them throughout this tutorial. If you wish, you can experiment with the others.

Customizing Your Machine File

When you load your machine file in Mastercam, Mastercam also loads the machine file into Code Expert. When you switch to the Code Expert window and show Machine Explorer, you can see the components of your machine file, as described below:



The **Control Layers** section lists the resources the post uses. Most of these resources are hidden by the post/machine developer. However, the **Consumer** layer contains machine defaults and other settings that you, as a user, can edit.

The **Mastercam Files** section lists support files that are encapsulated inside the your machine file. In non-Mill-Turn Mastercam, these support files are stored throughout your Mastercam installation; Mill-Turn brings them together inside the machine file.

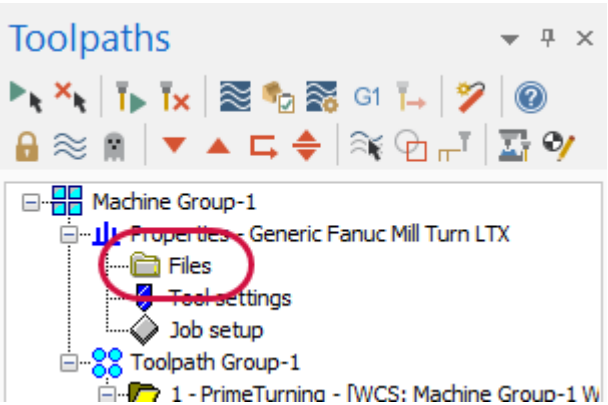
The files in the **Mastercam Files** section are disabled because you cannot edit them in Code Expert. However, you can edit common files such as your operation defaults, operation libraries, and tool libraries inside Mastercam. To do this, follow the guidelines in the next section.

Customizing Operation Defaults and Tool Libraries

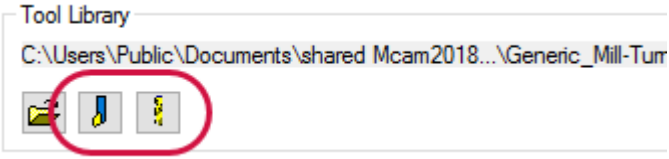
You can follow this procedure as needed to customize operation defaults and tool libraries. (You do not need to do this as part of the tutorial.)

1. Start Mastercam.
2. Load your machine file or a part that uses the machine file.
3. In **Toolpaths Manager**, click the **Files** icon in the Properties section.

   The **Machine Group Properties** dialog box opens with the Files tab selected.



4. In the **Tool Library** section, use the tool editing buttons to make needed tool library changes.

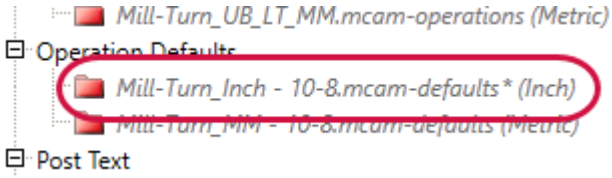


5. In the **Operations Library / Defaults File** section, use the edit buttons to make needed operation and defaults changes.

6. Click the green check to accept your changes and exit the **Machine Group Properties** dialog box.

   Note: You can also make changes using the Toolpaths Parameters page in any operation.

7. Switch to Code Expert and notice that your changed file is marked with an asterisk in Machine Explorer.



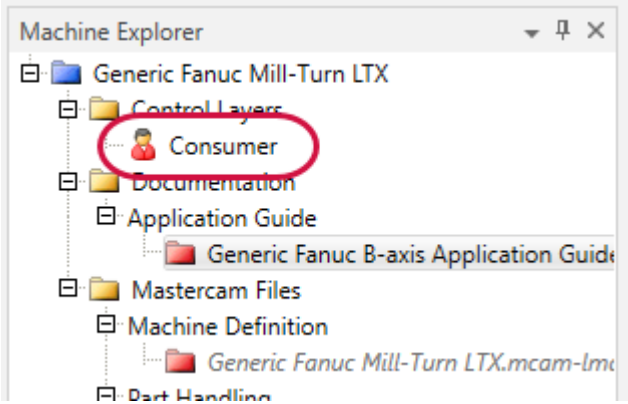
8. In Code Expert, click in **Machine Explorer** to give it the focus, and then press [**Ctrl+S**] to save your machine changes.

Editing Machine File Settings and Defaults

Your Reseller typically supplies your machine file ready to use. However, you can configure some settings according to your preferences and application needs. These settings include, but are not limited to, the following:

- Sequence and sync numbering
- Tool offset numbering
- Use of spaces in your NC file
- Job/shop info for your NC header

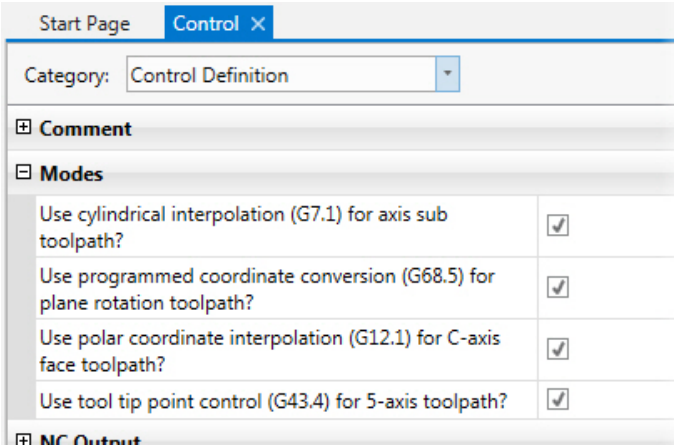
To find these settings, open your machine file in Code Expert and double-click the **Consumer** icon in Machine Explorer.



Most of the settings are grouped into two categories:

- Control Definition
- Output Settings

Select a category from the **Category** drop-down, and then click the + signs to see the individual options. (See the following picture.) Some of these options (for example, sequence number settings) are common to most controls; others are specific to individual machines.



These settings serve a variety of functions:

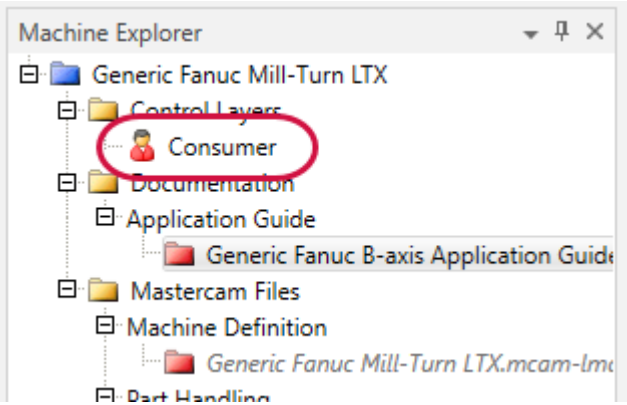
- Mimicking the control definition settings in non-Mill-Turn Mastercam
- Configuring toolpath modes and cycles
- Setting default values for Sync Manager and toolpath options

Most of these settings are self-explanatory, and you can easily configure them by browsing the interface. After making changes, press [**Ctrl+S**] to save your machine file.

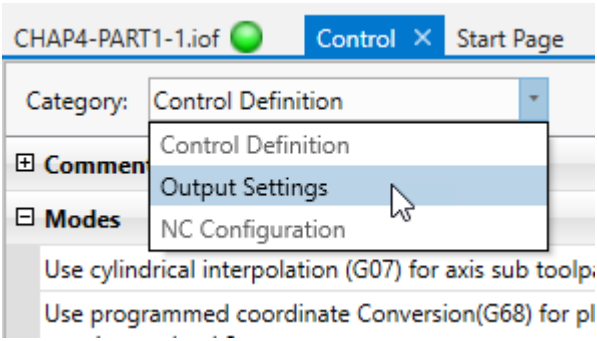
Opening Code Expert in Multi-Stream View

Several settings in the machine file configure Code Expert Editor. Access these options by opening the Consumer layer. For example, you can use Code Expert Editor in either single-stream or multi-stream mode. To make multi-stream mode the default, do the following:

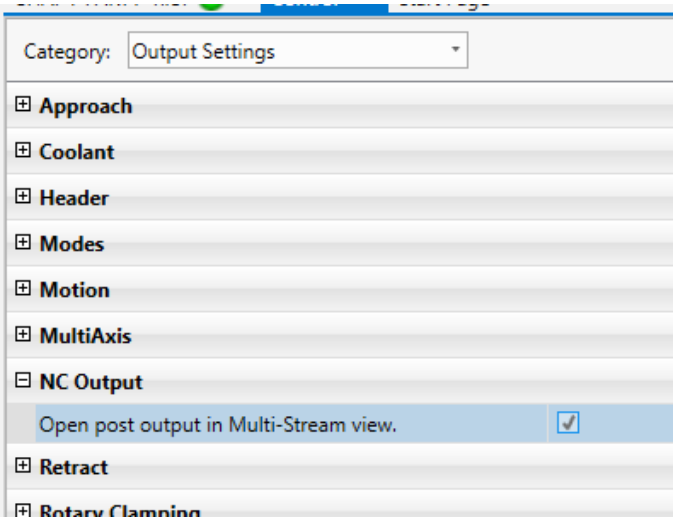
1. Open the **Consumer** layer.



2. Select the **Output Settings** category.



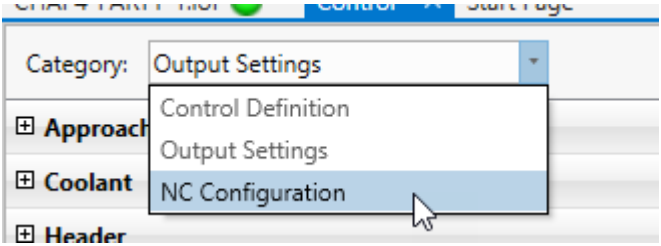
3. Open the **NC Output** group.
4. Select the **Open post output in Multi-Stream view** option.



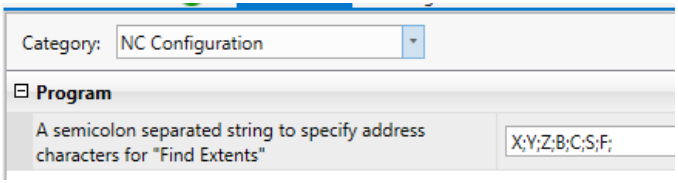
Configuring the Find Extents Feature

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

1. Go to the **NC Configuration** category.



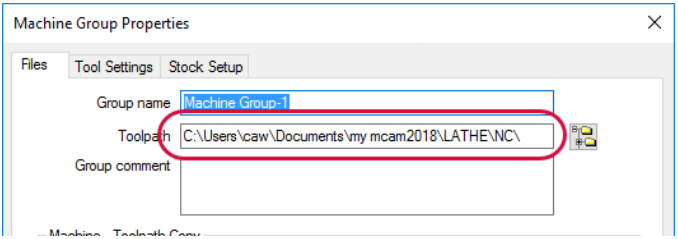
2. Open the **Program** group.
3. Enter the desired letter addresses in the list, separated by semi-colons (;).



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

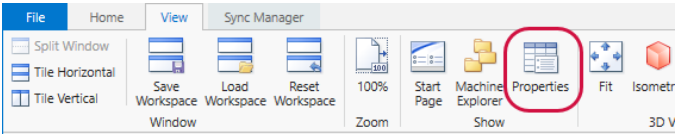
Setting the Toolpath Directory

In non-Mill-Turn Mastercam, the toolpath directory setting is in the Machine Group Properties dialog box, as shown below:

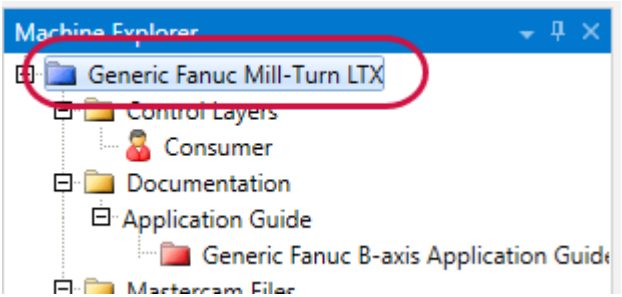


In Mill-Turn, the setting in Machine Group Properties is only used to override the default toolpath directory. The default toolpath directory (the one used for newly created files) is a property of the machine file. To set it, follow these steps:

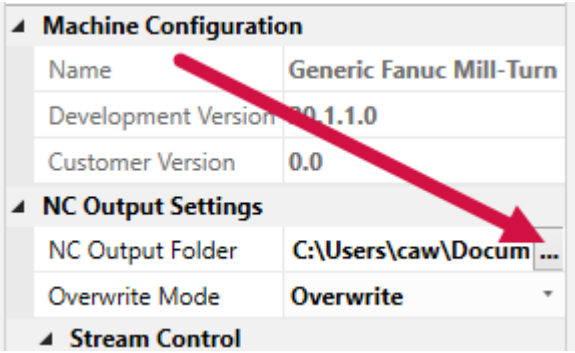
1. Open the machine file in Code Expert.
2. If necessary, open the **Properties** window.



3. In **Machine Explorer**, click the machine name.



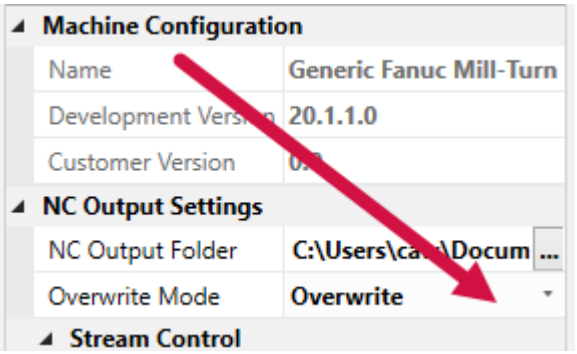
4. In **Properties**, select the desired NC output Folder.



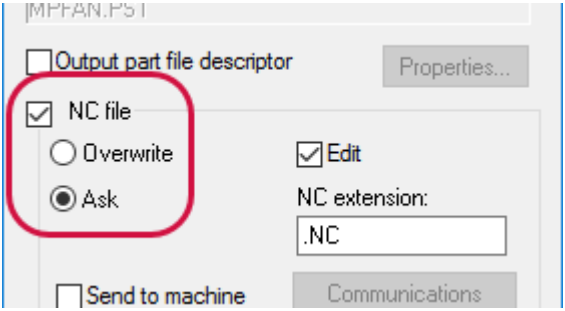
Mastercam will write your NC files for this machine to this folder.

5. Select the desired **Overwrite Mode**.

   You can choose whether Mastercam automatically overwrites NC files with the same file name or prompts you to enter a different name.



   This **Overwrite Mode** setting replaces the setting from the Posting dialog box in non-Mill-Turn Mastercam.



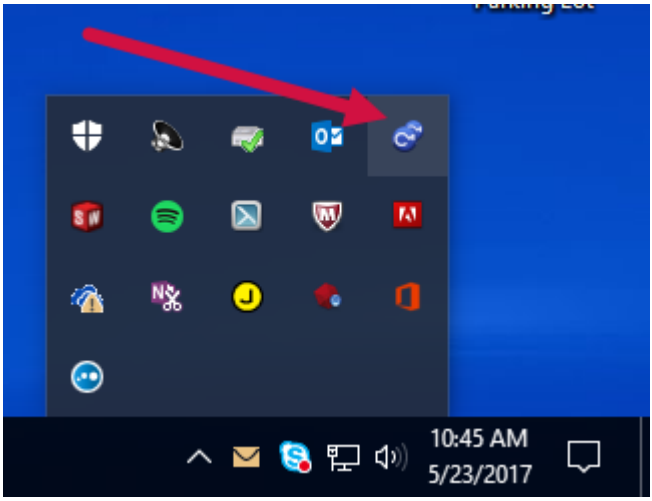
6. Save the machine file.

   Note: For more information on your machine's settings and capabilities with Mill-Turn, please refer to the application guide that comes packaged with each machine. You can also access this document in Machine Explorer's Documentation folder.

Managing Machine Licenses

Before you can use a Mill-Turn machine, you must have a license for it. If you use other Mastercam products, you are familiar with HASPs, the small device that plugs into your USB port and verifies your Mastercam license.

Mill-Turn machines, however, use software licenses administered by a product named CodeMeter. You can access CodeMeter from your System Tray, as shown in the picture below. Use CodeMeter to inspect or manage your Mill-Turn licenses.



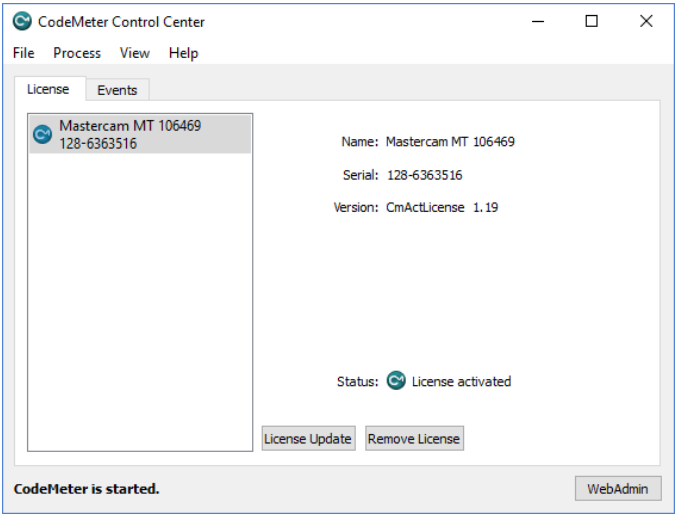
Verifying that You Have the Proper License

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

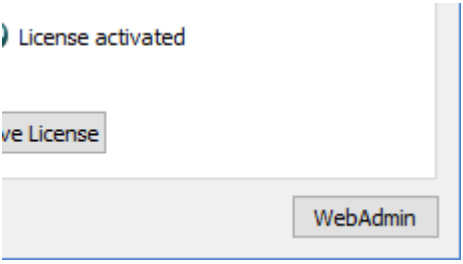
1. Click the **CodeMeter** icon in the System Tray.

   The **CodeMeter Control Center** opens. You should see a single item in the License tab. This is the container that stores all of the Mill-Turn licenses

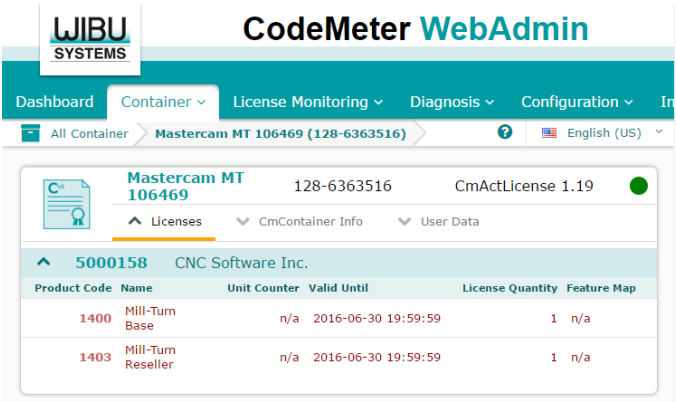
installed on your system. The **Status** should tell you that the license is activated.



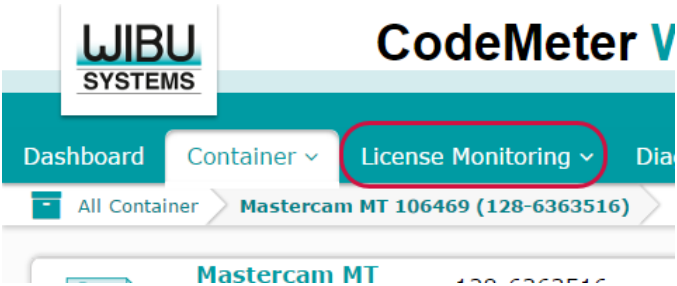
2. Click the **WebAdmin** button to inspect the individual licenses.



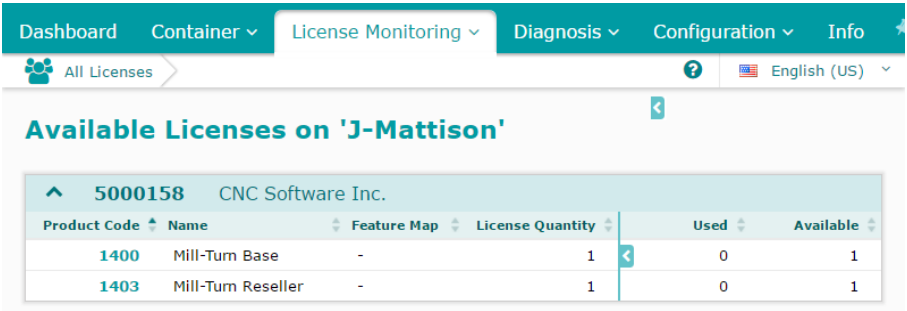
Your Web browser opens to the CodeMeter WebAdmin page, as shown in the following image.



3. Click the **License Monitoring** tab to see the licenses that are installed.



The example below shows this user's installed Mill-Turn licenses. If this user had licenses for machines, they would be listed here, as well.



Note: In Mastercam, if you try to open a machine for which you do not have a license—or open a part file that uses such a machine—Mastercam displays an error message.

Reactivating Disabled Licenses

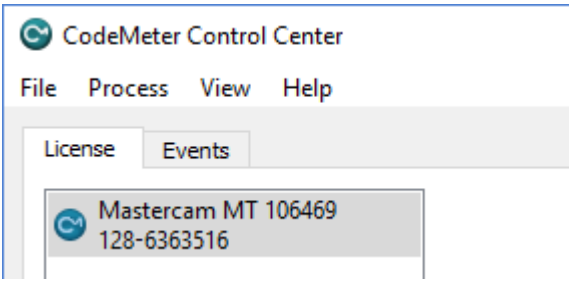
If you have a license installed on your system, and you still receive a license message, the license may have become disabled. To run properly, the CodeMeter application needs to maintain a constant connection with your HASP. If this connection breaks, your software licenses cannot be activated, and you will be unable to run Mill-Turn.

Common reasons this might happen include:

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

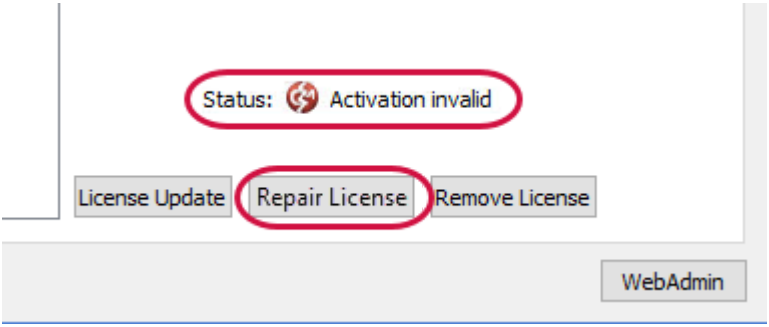
If your CodeMeter licenses are invalid for any reason, the icon in the System Tray changes from blue to red. To repair the license, follow these steps:

1. Make sure that your workstation is properly connected to the HASP.
2. Start CodeMeter by clicking its icon in the System Tray.
3. When CodeMeter starts, make sure your Mastercam MT license container is visible.



CodeMeter will tell you that the license is invalid, as shown in the picture for the following step.

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



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

5

Working with Tools and Spindles

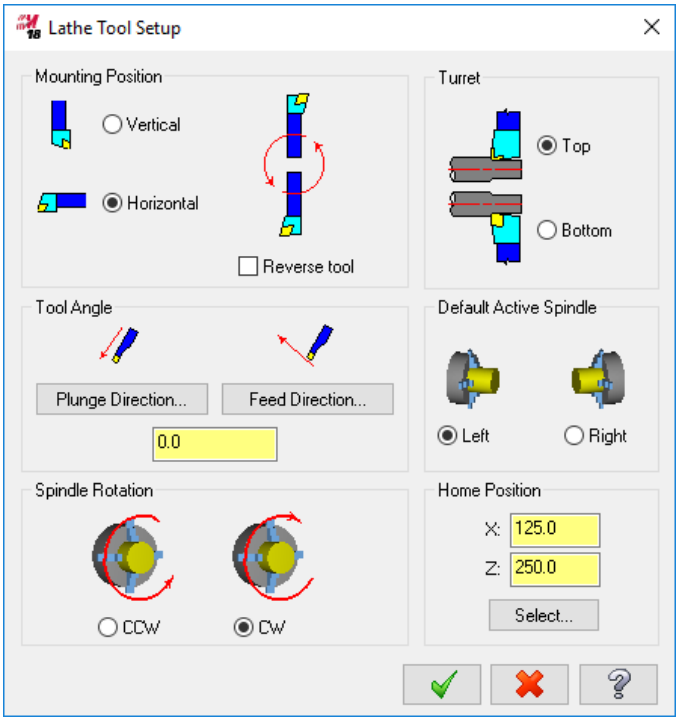
This lesson explains how tool and toolpath settings work together to produce the proper NC output for your Mill-Turn machine's lathe operations and tools.

Lesson Goals

- Initial tool setup.
- Edit tool settings.
- Change insert orientation.
- Switch between LH and RH holders.
- Specify a tool's compensation point.
- Set up for left or right spindle.
- Setup for pinch turning.

Initial Tool Setup

Tools for both upper and lower turrets should be defined so that they are oriented correctly for the intended turret at the tool change position, with the correct vertical or horizontal orientation. These settings are stored in the Mastercam tool definition, shown below.



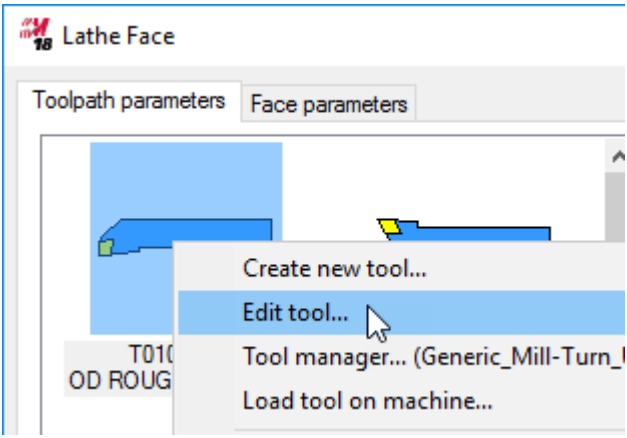
When you first select the tool, it should look as it is oriented at the tool change regardless of which spindle you are working on. The position shown in the Toolpath parameters page will be different depending on whether the tool is in the B-axis head or a lower turret.

Editing Tool Settings

To review or edit the tool settings, follow these steps. (This general procedure is informational only and not part of the tutorial.)

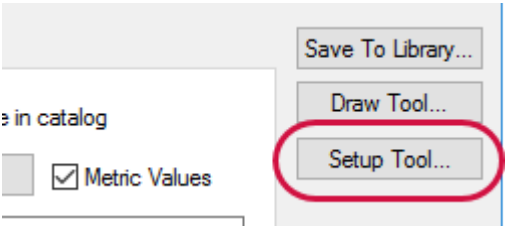
1. Right-click the tool in the tool selection window.
2. Choose **Edit tool** from the pop-up menu.

   The **Define Tool** dialog box opens.



3. Click **Setup Tool**.

   The **Lathe Tool Setup** dialog box opens.



4. Make changes to the tool settings, as needed.

5. Click **OK** in the **Lathe Tool Setup** and **Define Tool** dialog boxes to finalize your changes.

Note: When altering the tool definition settings, use the **Draw Tool** button to quickly view the tool's orientation.

Insert Up/Down and LH/RH Holders

When the tool is mounted at the tool change position, it can be in any of four 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 orientations is acceptable with Mill-Turn; the correct choice depends on how you need the tool oriented when you rotate it into its cutting position.

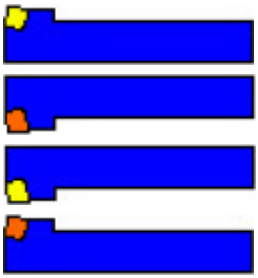
The tool preview window in Mastercam shows you how the tool is oriented. Notice in the following picture that the insert color changes to show insert up (yellow) or insert down (orange).

Left-hand holder, insert up

Left-hand holder, insert down

Right-hand holder, insert up

Right-hand holder, insert down

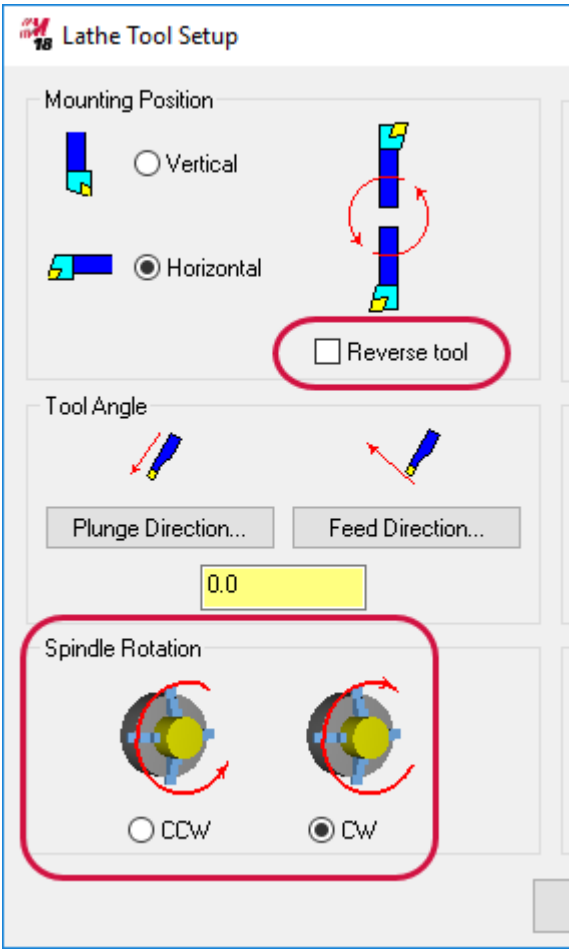


Changing Insert Orientation

To change the tool from insert up to insert down, do the following. (This general procedure is informational only and not part of the tutorial.)

1. Go to the **Lathe Tool Setup** dialog for the tool. (Click the tool in the operation, and then click **Setup Tool** in the Define Tool dialog box.)
2. Toggle the **Reverse tool** setting.

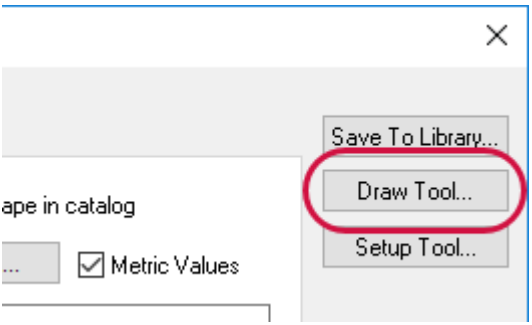
3. Change the **Spindle Rotation** direction.



Switching Between LH and RH Holders

To switch between left- and right-hand holders, do the following. (This general procedure is informational only and not part of the tutorial.)

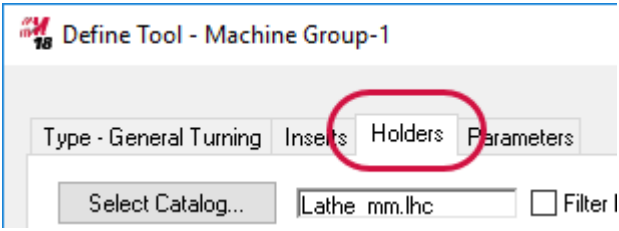
1. Go to the **Define Tool** dialog box. (Click the tool in the operation.)
2. Click the **Draw Tool** button.



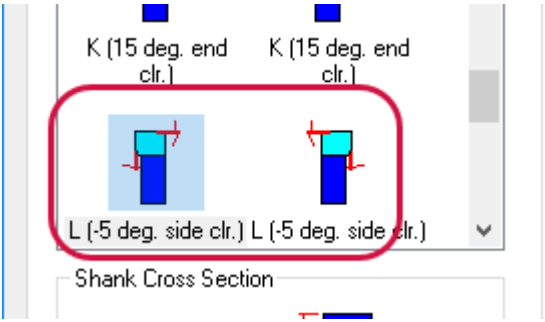
For your verification, Mastercam displays the tool in its current configuration.

3. Press [**Enter**] to return to the **Define Tool** dialog box.
4. Select the **Holders** tab.

In the **Style** box, the holders are arranged in left/right pairs.



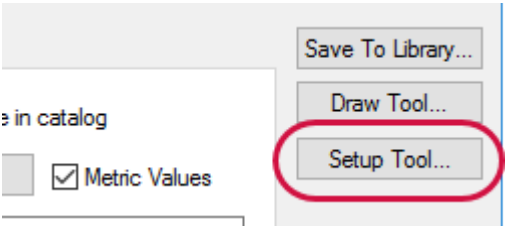
5. Select the desired left- or right-hand holder.



6. Click the **Draw Tool** button to see the change, and then press [**Enter**] to return to the **Define Tool** dialog box.

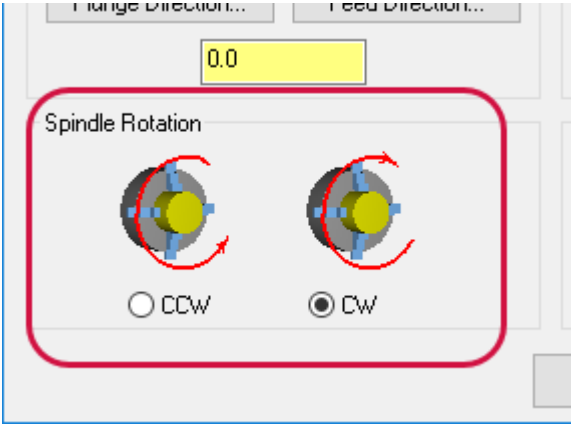
7. Click **Setup Tool**.

   The **Lathe Tool Setup** dialog box opens.



8. Change the **Spindle Rotation** setting, and click **OK**.

   The tool positions correctly for the operation.

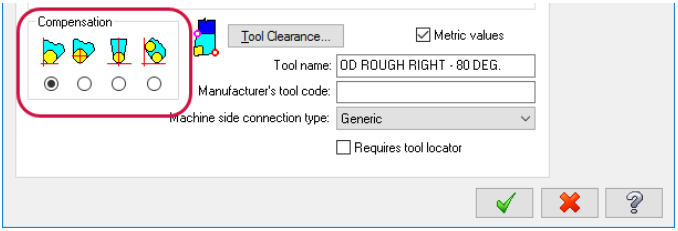


9. Click the **Draw Tool** button to see and verify your change, and then press [**Enter**] to return to the **Define Tool** dialog box.

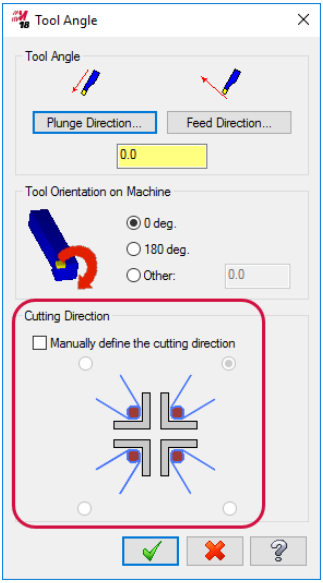
10. Click **OK** to dismiss the **Define Tool** dialog box.

Specifying the Tool's Compensation Point

Depending on how you set up your tools on the machine, you may need to change the compensation point Mastercam uses to calculate the toolpath. Do this in the Parameters tab of the Define Tool dialog box, as shown here. If your machine has a tool eye (touch setter for turning tools) the standard setting is the first option.



If you are using a tool that is being rotated in the B-axis head, and your toolpath is not compensated as you expect, use the Cutting Direction options of the Tool Angle dialog box (shown below) to specify how your tool was touched off on the machine. Click the **Tool Angle** button on the Toolpath parameters tab to open this dialog box.



If you use this compensation method, you must touch off your tools differently at the machine.

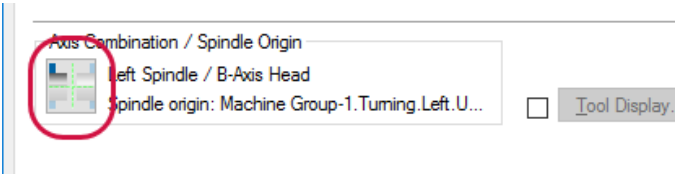
Selecting the spindle and turret

In Mastercam, to specify which spindle and turret to use, you select an axis combination. There is a specific axis combination for each possible combination of B-axis head, left/right spindle, and turret. Even if your machine has only a B-axis head and left spindle, Mastercam defines at least one axis combination.

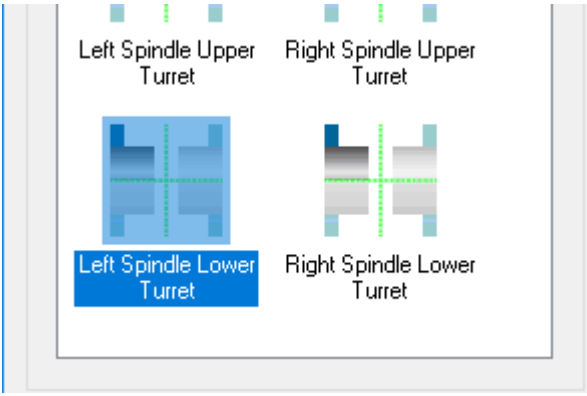
Selecting an Axis Combination

The following general procedure demonstrates multiple axis combos. Follow these steps. (This general procedure is informational only and not part of the tutorial.)

1. In the **Toolpath parameters** tab, click the **Axis Combination** button.



2. Select the desired turret/spindle combination. This picture shows the axis combination set to use the lower turret on the left spindle.



Setting Up Tools for Pinch Turning

When creating a pinch turn operation, the settings for the two tools must be correctly coordinated. Mastercam cannot create the operation if the settings conflict. Follow these guidelines.

- *Insert direction*—The insert directions for the two tools must 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 and 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 created correctly, you might need to select a different tool holder.

6

Reference Positions

Use reference positions (not to be confused with reference points) to specify where to start and end operations. For example, you can move to the tool change position between operations by selecting a reference position for the first operation's retract and selecting the same reference position for the second operation's starting point.

The following exercises demonstrate how to use and define reference positions in your Mill-Turn operations.

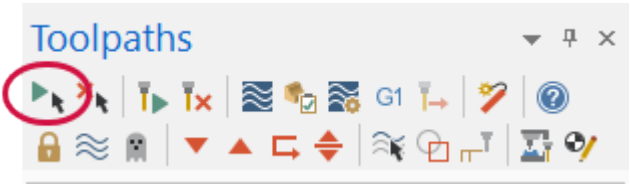
Lesson Goals

- Specify reference positions.
- Define reference positions.

Exercise 1: Using Reference Positions

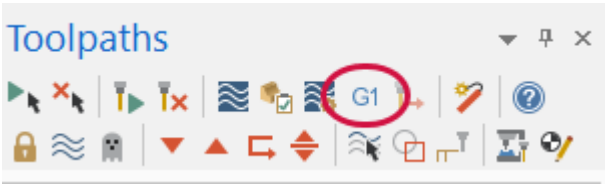
Mill-Turn machines come with standard reference positions already defined. You can use these predefined reference positions without changing the machine or your part files. Just select them as needed in Sync Manager, as the following exercise shows.

1. Load the file `RefPositions` into Mastercam, and save it as RefPositions–*XX*, where *XX* is your initials. This ensures that you do not accidentally overwrite the original file.
2. Select all operations in **Toolpaths Manager**.



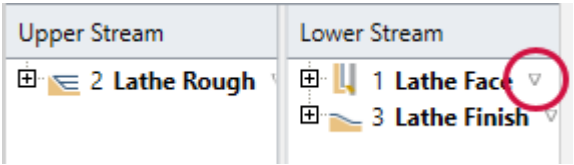
3. Click the **G1** button.

   Sync Manager opens and displays the part's data.



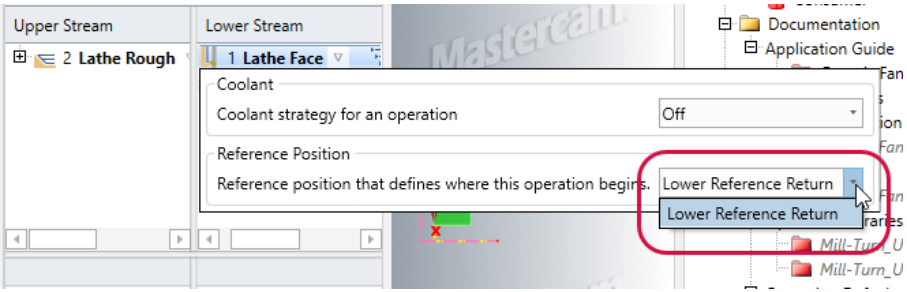
4. Click the small inverted triangle next to operation 1, **Lathe Face**.

   A set of options appears in a drop-down box. One of these options is **Reference Position**.



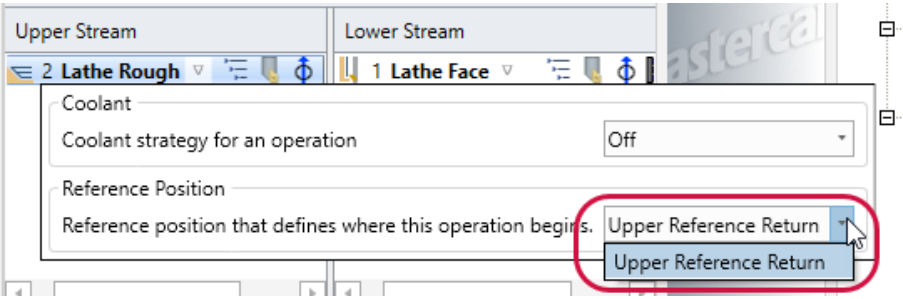
5. Open the drop-down menu for the **Reference position** option, and note that it contains only one item, named **Lower Reference Return**.

   This is the only reference position currently available for this operation on the machine's left spindle, lower turret.



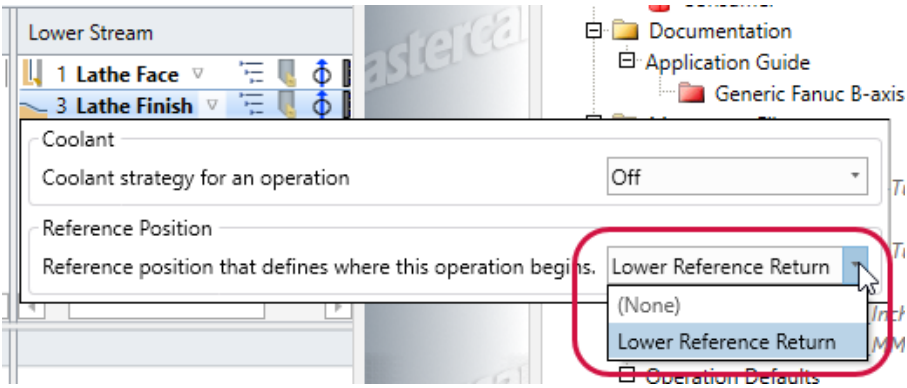
6. Look at the reference position for operation 2, **Lathe Rough**.

   Upper Reference Return is the only reference position currently available for this operation on the machine's left spindle, B-axis head.



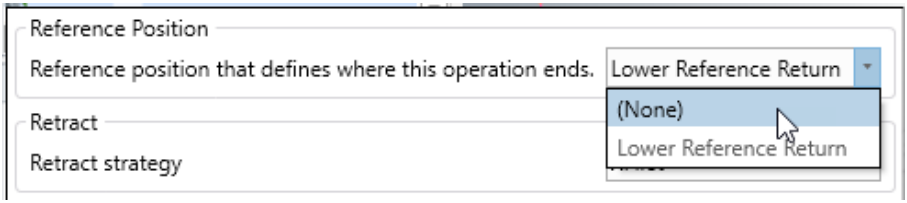
7. Look at the reference position for operation 3, **Lathe Finish**.

   You see two reference positions, **(None)** and **Lower Reference Return**.



8. Expand the **Lathe Finish** operation, so that you can see the sub-operations. (If you cannot see the "plus" icon for expanding the operation, make sure the pane is scrolled all the way to the left.)

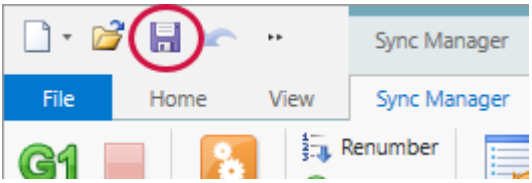
9. On operation 3's **Retract**, change the Reference position setting from **Lower Reference Return** to **(None)**.



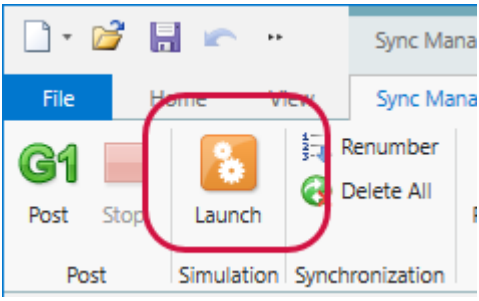
Note: Reference positions are available only on the operation and retract nodes in the Operations pane.

10. Save the file.

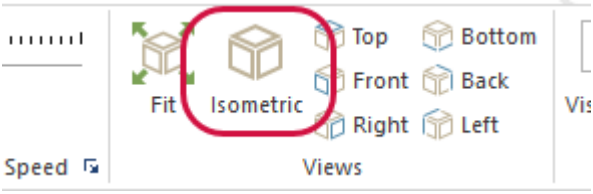
    Saving the file in the Sync Manager updates the part in Mastercam.



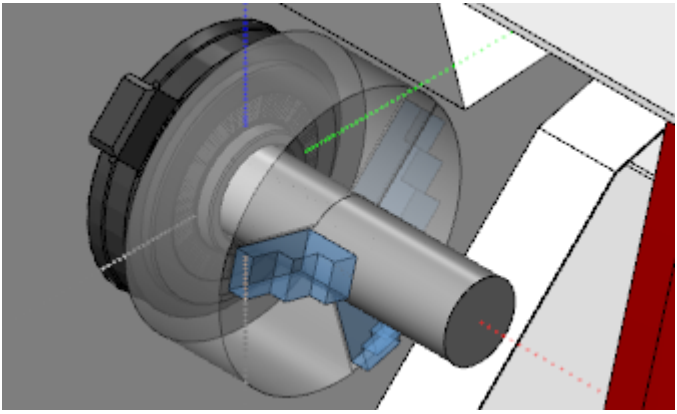
11. Run Machine Simulation.



12. Set the view to **Isometric**.

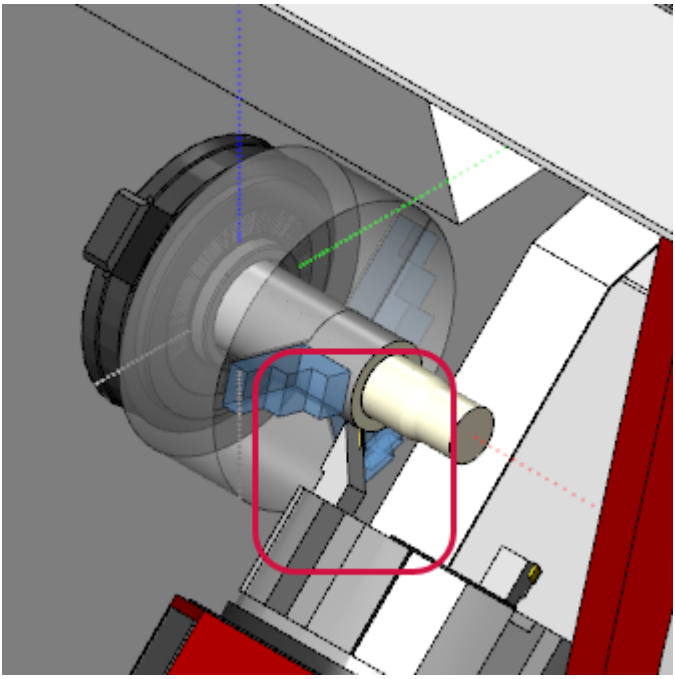


13. Use your mouse wheel to zoom in on the chuck.

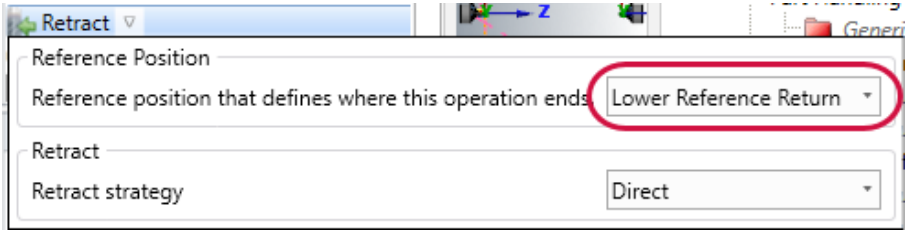


14. Click **Run** on the Simulation tab to run the simulation, and notice that the lower turret does not retract to a reference position after completing the finish cut.

This is because you set this retract's reference position to **(None)**.

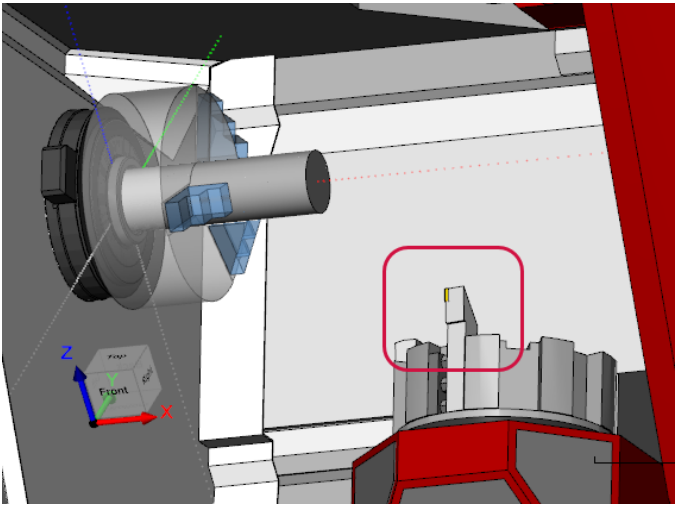


15. In Sync Manager, change the reference position for the third operation's **Retract** back to **Lower Reference Return**.



16. Save the file, relaunch Machine Simulation, and click the **Run** button.

    Now, after completing all cuts, the spindle retracts to the position specified by the Lower Reference Return reference position.



17. Close Machine Simulation.

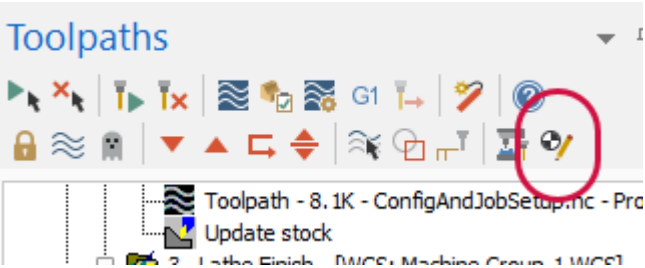
Exercise 2: Creating Reference Positions

Although each machine file includes a set of predefined reference positions, you can use the main Mastercam application to create your own. You might want a custom reference position, for example, to accommodate special part fixtures or tooling. Unlike the predefined reference positions, custom reference positions are stored in the part file, not the machine file.

The following exercise demonstrates how to define and use a custom reference position. Continue with the file from the previous exercise.

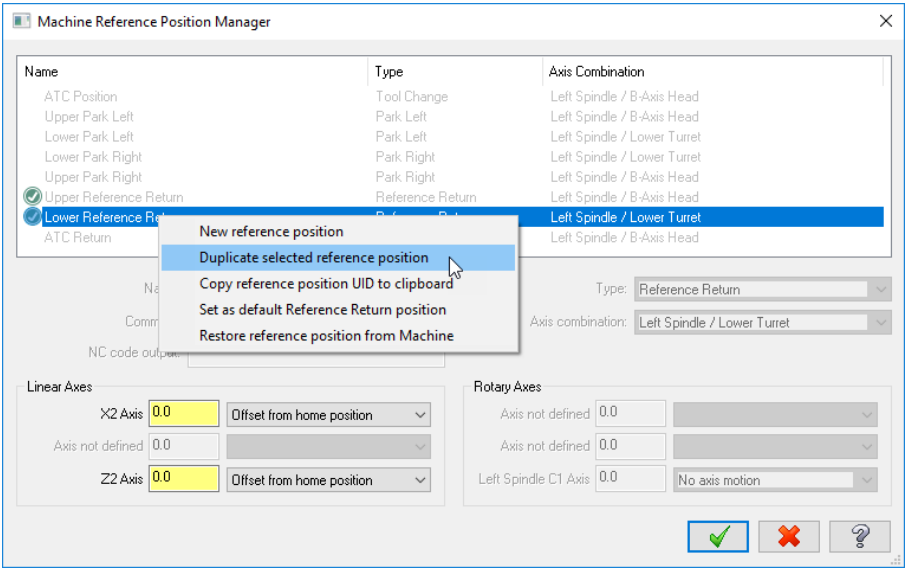
1. In Mastercam's **Toolpaths Manager**, click the **Edit reference positions** button.

   The **Machine Reference Position Manager** opens.



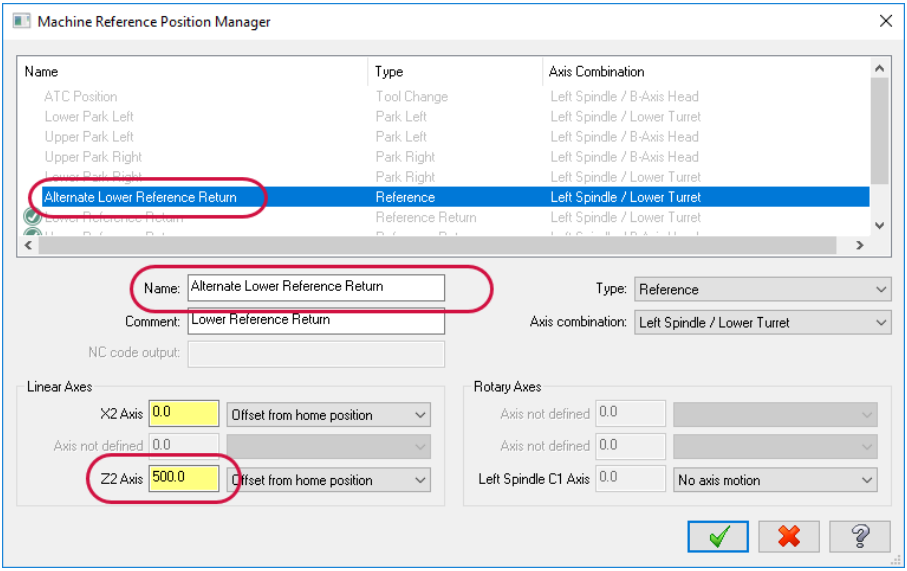
2. Right-click **Lower Reference Return**, and choose **Duplicate selected reference position** from the pop-up menu.

   Mastercam creates a copy of the reference position.



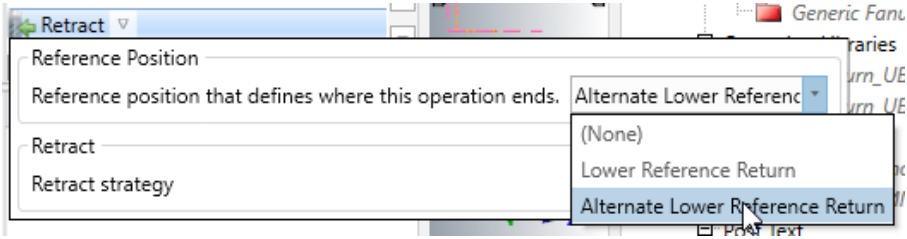
3. Change the new reference position's name to **Alternate Lower Reference Return**, and set the **Z2 Axis** value to **500**. Click **OK** to save your changes.

   Now, **Alternate Lower Reference Return**, the reference point you just created, is identical to **Lower Reference Return**, except for its name and Z2 Axis value.



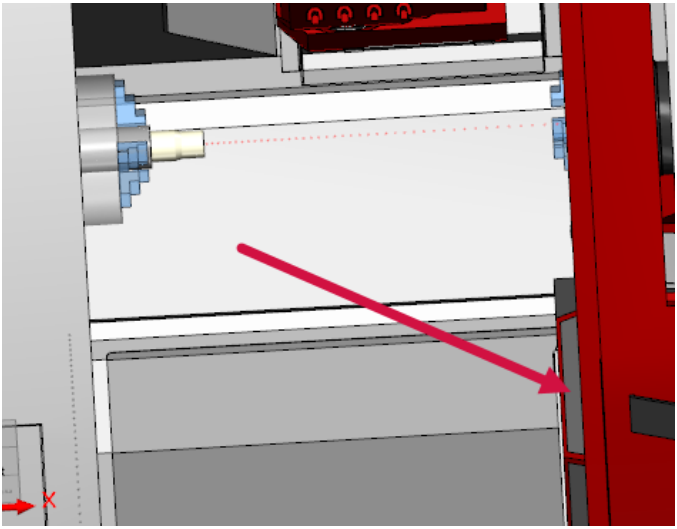
4. Click the **G1** button to update Sync Manager with the new reference position.

5. Change the reference position for operation 3's **Retract** to the new **Alternate Lower Reference Return**.



6. Save the file, and run the simulation.

Now the turret retracts to a position 500 mm farther to the right, compared to where it started at the original Lower Reference Return reference position.



7. Close Machine Simulation, Code Expert, and Mastercam.

7

Configuration and Job Setup

Before you create toolpaths with Mastercam Mill-Turn, you must complete configuration and setup tasks. These include choosing a machine and setting the appropriate options for your job.

Lesson Goals

- Add a machine file.
- Select a machine file.
- Complete Job Setup.

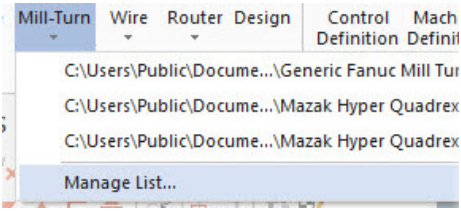
Adding a Machine File

Mill-Turn supports many types of machines, each having its own machine solution (`MACHINE` file). To see how to add a machine, start Mastercam and perform the following steps.

Exercise 1: Adding Your Machine

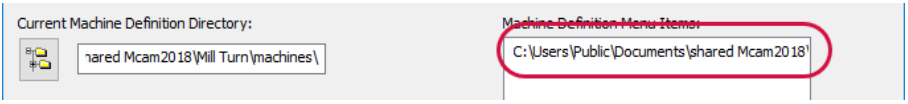
1. If you have not already done so, place a copy of `Generic Fanuc Mill-Turn LTX.machine` (also included with this tutorial) in your `C:\Users\Public\Documents\shared mcam2018\Mill Turn\machines` directory.
2. On Mastercam's Machine tab, choose **Mill-Turn, Manage List**.

   The **Machine Definition Menu Management** dialog box displays.



3. For the purposes of the exercises in this tutorial, choose `Generic Fanuc Mill-Turn LTX.machine`, and click **Add**.

   The machine moves to the **Machine Definition Menu Items** box, which means that it will now appear in Mastercam's **Machine Type, Mill-Turn** submenu.



4. Click **OK** to dismiss the **Machine Definition Menu Management** dialog box.

Setting Up Your Job

You can now select the machine from the **Machine, Mill-Turn** drop-down menu. When you do, Job Setup, in which you must define your job's parameters, displays. These parameters include the following:

- The Work Coordinate System (WCS) setting
- The spindle on which work will begin
- The type of work to be done (single piece or bar)
- The part geometry
- The stock definition
- The chuck jaw definitions
- The part stickout and pickoff positions

Exercise 2: Selecting a Machine

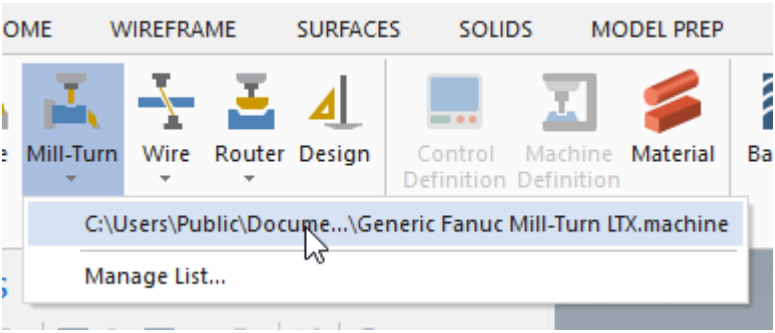
1. Load the part file `TurningToolpaths`, which is included with this tutorial.

   Note: This part file is in Metric units. If your Mastercam system is set for English units, you will be asked about switching from English to Metric. Just click **OK** to proceed.

   The part's geometry displays in Mastercam's graphics window. However, because no machine is assigned to the part, the Toolpaths Manager is empty.

2. Save the part file as `TurningToolpaths_`*XX*, where *XX* is your initials. This ensures that you do not accidentally overwrite the original file.

3. From Mastercam's **Machine, Mill-Turn** drop-down menu, choose the `Generic Fanuc Mill-Turn LTX.machine` file.



Mastercam creates a new machine group, and then displays the Job Setup tab of the Machine Group Properties dialog box.

Note: If Code Experts opens on top of Mastercam, minimize Code Expert. Do not close it.

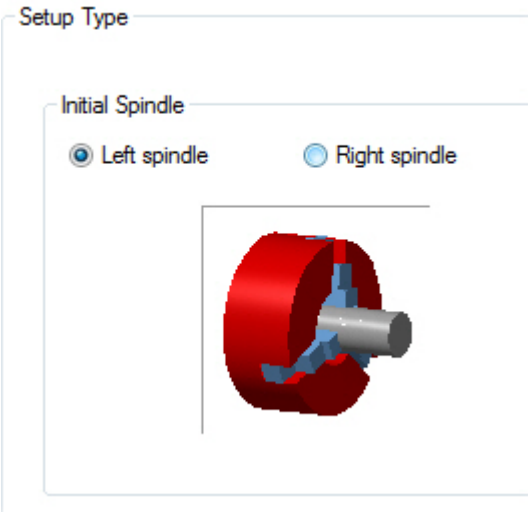
Completing Job Setup

The Job Setup tab contains the general parameters for your job. These parameters are arranged in top-down fashion, providing a helpful work flow for successfully configuring your job. In this section of the tutorial, you work through the parameter list, setting up a continuous bar stock job on the left spindle. This job includes pickoff and cutoff operations.

Exercise 3: Setting Setup Type and WCS

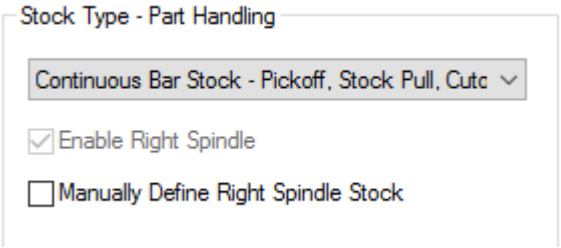
1. Select **Setup Type** in the list, and ensure that **Initial Spindle** is set to **Left spindle**.

   The **Left spindle** setting specifies that the job starts on the left spindle. If a part transfer is required, the part moves from the left spindle to the right spindle.



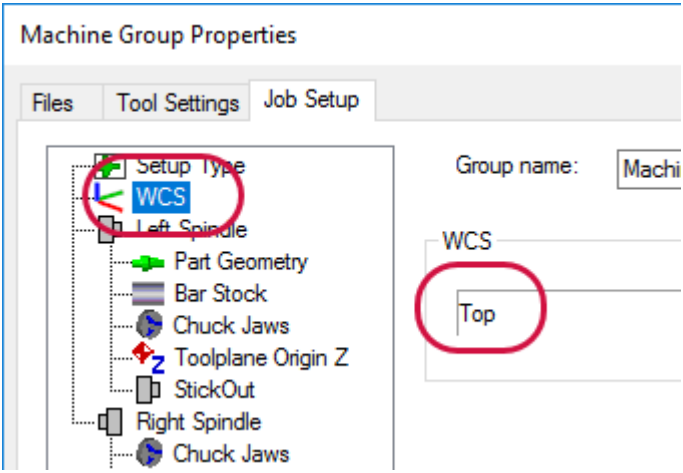
2. Ensure that **Continuous bar stock - Pickoff, Stock Pull, Cutoff** is selected in the drop-down list.

   Mill-Turn includes options for continuous bar stock and single stock pieces. Your selection determines the part handling operations you can create and the options that are available in the Pickoff page.



3. Select the **WCS** page, and ensure that **WCS** is set to **TOP**.

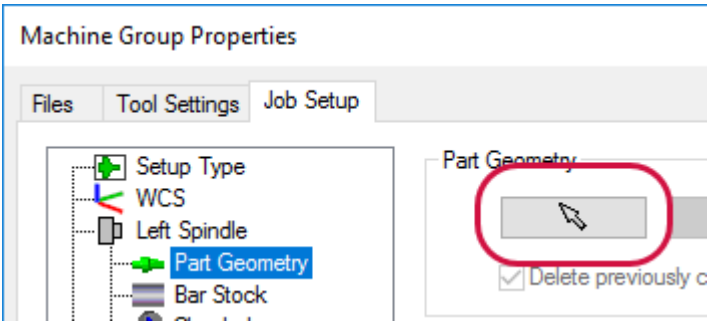
Note: In your production work, you can use the WCS setting to reorient parts that are not aligned correctly for machining.



Exercise 4: Specifying Part Geometry

1. Select **Part geometry** from the list, and click the geometry selection arrow.

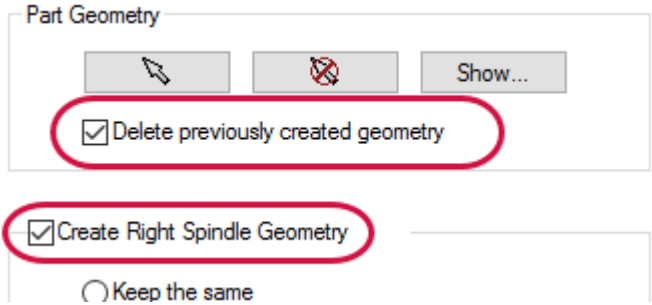
   The dialog box minimizes so that you can access Mastercam's graphics window.



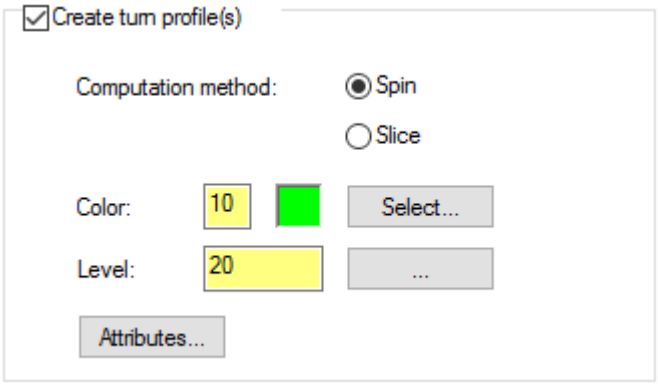
2. Click anywhere on the part, and press [**Enter**].

Mastercam returns to the **Job Setup** tab, with the part's geometry selected.

Mastercam also enables the **Delete previously created geometry** and **Create Right Spindle Geometry** options. The first ensures that geometry created in previous operations (if you've previously run Setup) is removed from the part. The second creates geometry for the opposite spindle, in this instance, the right spindle.



3. Select the **Create turn profile(s)** option, and ensure that **Spin** is active.

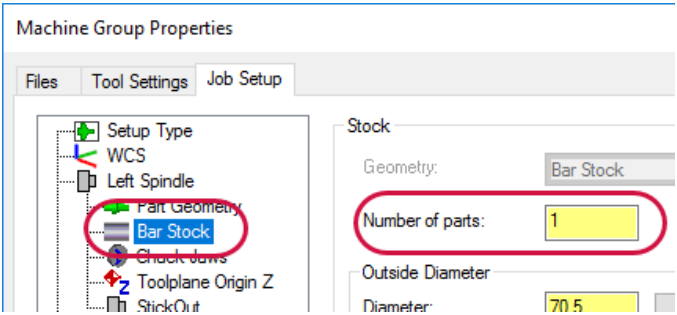


The **Create turn profile(s)** option tells Mastercam to create turn-profile geometry using either of the following methods: **Spin** rotates the part geometry around an axis, generating the profile from the farthest extents of the spun geometry. **Slice** slices the part geometry in the Lathe ZX (WCS XY) plane. The

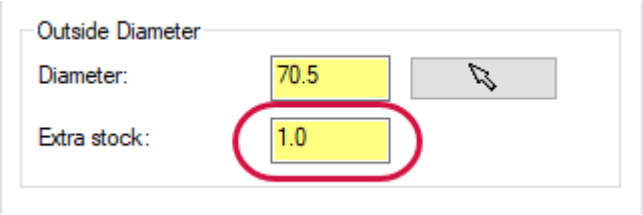
profile is limited to what the slice encounters and may not include the geometry's full extents. For example, bosses not in the slice plane will be unrepresented in the turn profile.

Exercise 5: Specifying Bar Stock Settings

1. Select **Bar Stock** in the job setup list, and ensure that **Number of parts** is set to **1**.

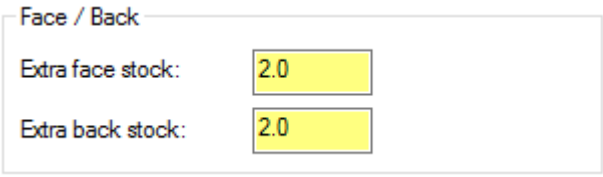


2. Set **Extra stock** for the outside diameter to **1.0** mm.



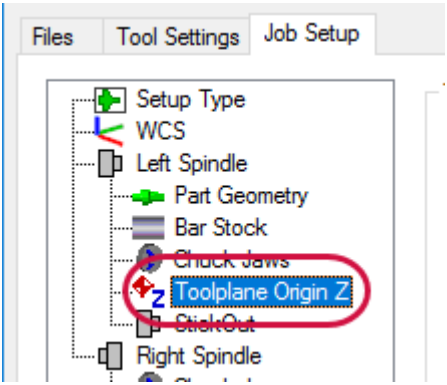
Note: The initial stock settings, such as diameter values, appear when a solid has been selected as the part or stock model. This is not the case with wireframe geometry.

3. Set extra stock for face and back to **2.0** mm.

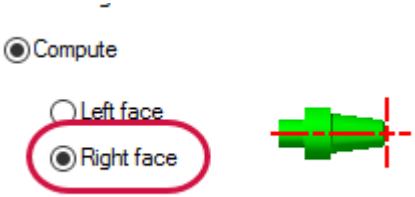


Exercise 6: Setting Spindle and Chuck Parameters

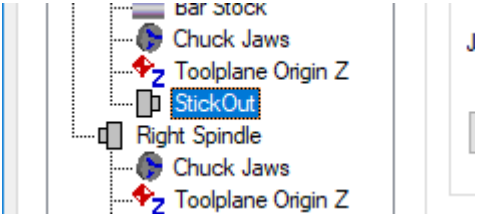
1. In the job setup list, click **Left Spindle, Toolplane origin Z**.



2. Notice that the toolplane origin is on the part's right face.

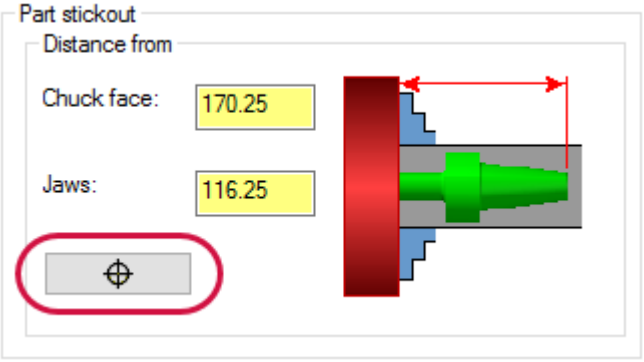


3. Choose **StickOut** in the Job Setup list.

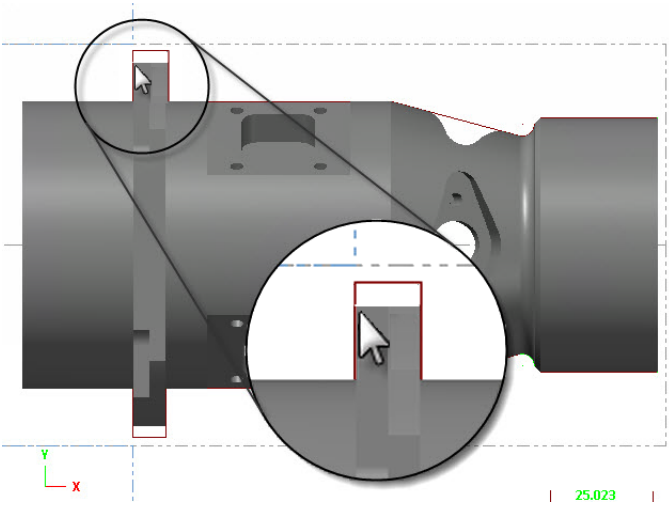


4. In the **Part Stickout** box, click the "target" button.

   Mastercam minimizes the dialog box so that you can access the graphics window.

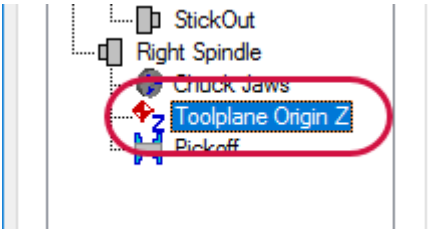


5. Click the part where shown in the following picture.

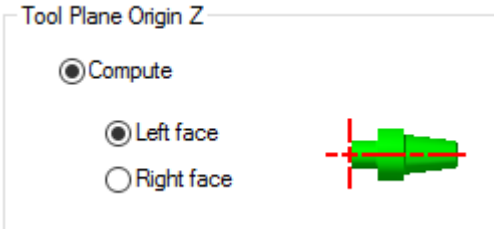


The **Machine Group Properties** dialog box redisplays, and **Chuck face** is set to **150.0**.

6. In the job setup list, click **Right Spindle, Toolplane origin Z**.



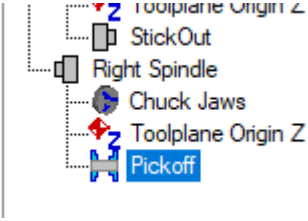
7. Notice that the toolplane origin is on the part's left face.



Note: The jaw options that you see are based on the Stock type and other stock settings. For example, you will not see ID jaw options unless an inner diameter has been specified on the Bar stock page.

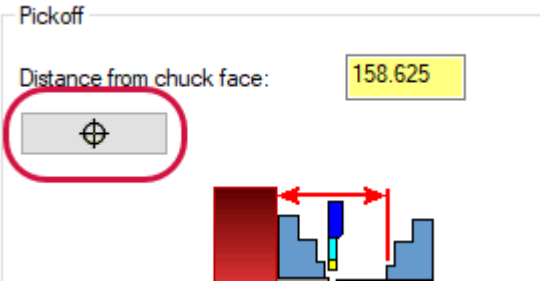
Exercise 7: Defining the Pickoff Operation

1. In the job setup list, select **Pickoff**.

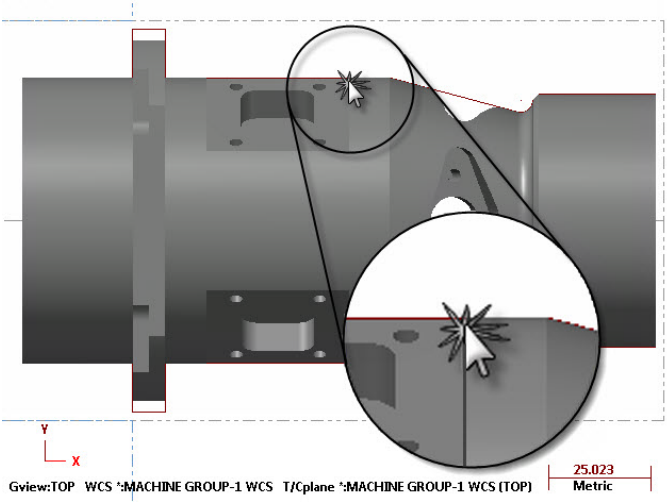


2. In the **Pickoff** box, click the "target" button.

   Mastercam minimizes the dialog box so that you can access the graphics window.

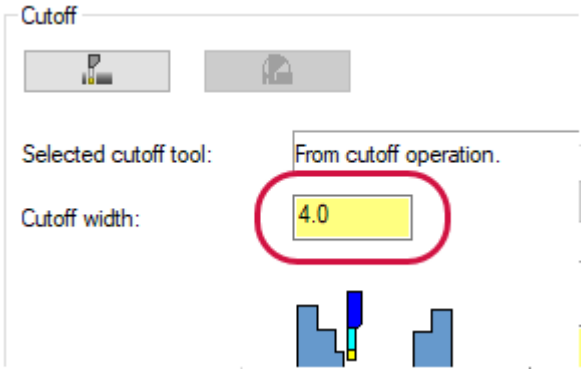


3. Click the part where shown in the following picture.



The **Machine Group Properties** dialog box redisplays, and **Distance from chuck face** is set to **93.75**. This is where the right spindle will grip the part.

4. In the Cutoff box, set **Cutoff width** to **4.0** mm.



5. Click **OK** to close the **Machine Group Properties** dialog box.
6. Save the file as `TurningToolpaths2_XX`, where *XX* is your initials.

Your job setup is now complete. In the next lesson, you create the toolpath operations needed to face the part in the left spindle with the lower turret.

8

Turning Toolpaths

In this lesson, you create toolpaths to cut the face and OD of a part with both roughing and finishing operations. You program the initial operations on the left spindle. You then transfer the part to the right spindle and cut the areas that were inaccessible in the left spindle.

Lesson Goals

- Perform facing, using the left spindle and lower turret
- Perform roughing, using the left spindle and upper turret
- Perform finishing, using the left spindle and lower turret
- Perform part transfer from the left spindle to the right spindle, which includes pickoff and cutoff operations
- Perform facing, using the right spindle and lower turret
- Perform pinch turn, using the right spindle, upper turret, and lower turret

Programming the Left Spindle

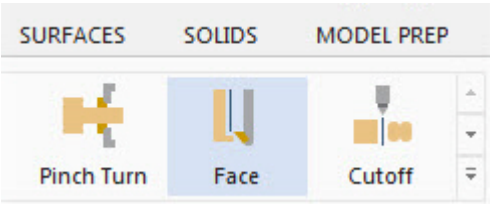
In this section, you create a facing toolpath. You also create roughing and finishing operations for the part's OD. You program these toolpaths on the left spindle, alternating between the lower and upper turrets.

Continue with your part from the previous lesson. Alternatively, you can load the file `TurningToolpaths2`, included with this tutorial. If you choose to load `TurningToolpaths2`, remember to save it as `TurningToolpaths2_XX`, where *XX* is your initials.

Exercise 1: The Facing Toolpath

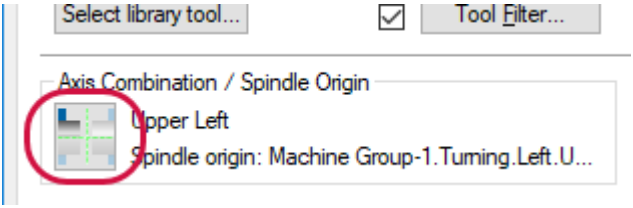
1. Click Mastercam's **Turning** tab, and choose **Face** from the General toolpaths.

   The **Lathe Face** dialog box appears. The tools displayed, however, are not appropriate for the operation you will create.



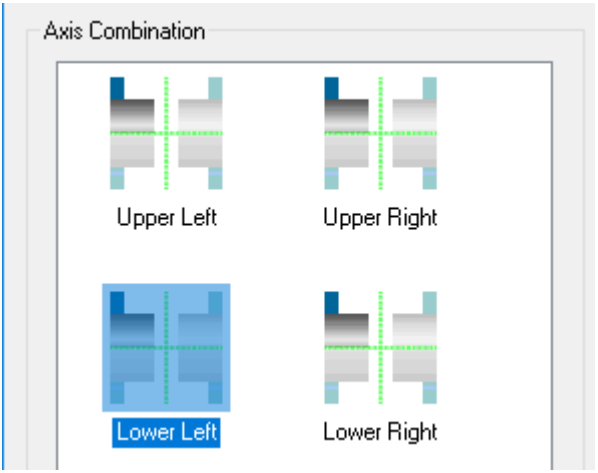
2. Click the **Axis Combination** button.

   The **Axis Combination / Spindle Origin** dialog box displays.

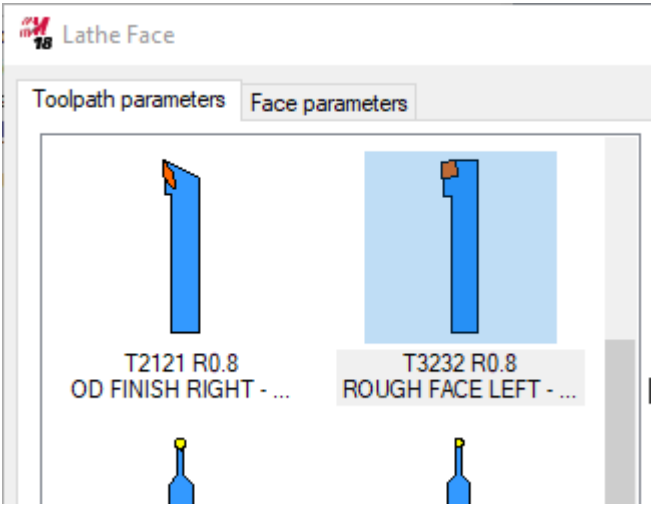


3. Select the **Lower Left** axis combination, and click **OK**.

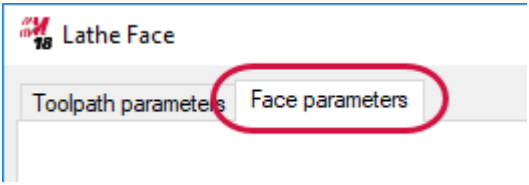
The **Lathe Face** dialog box now shows tools that are appropriate for the lower-left axis combination.



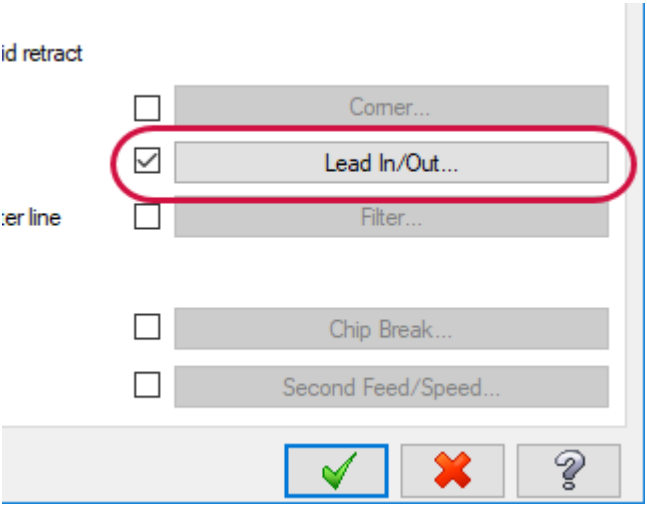
4. Select the **ROUGH FACE LEFT - 80 DEG** tool (your tool may be in a different position than the one shown).



5. Click the **Face parameters** tab.



6. Activate the **Lead In/Out** checkbox.



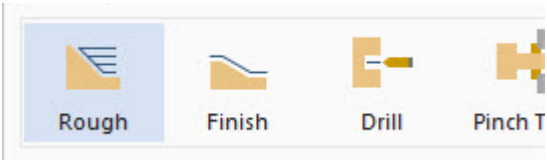
7. Click **OK** to exit the dialog box.

   Mastercam creates the facing toolpath based on your selected parameters, and the operation appears in Toolpaths Manager.

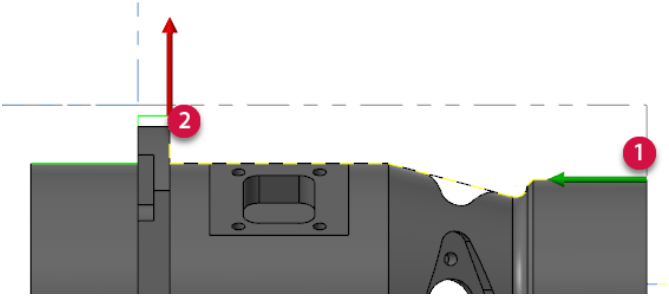
Exercise 2: The Roughing Toolpath

1. Choose **Rough** from the General toolpaths.

   The **Chaining** dialog box displays.



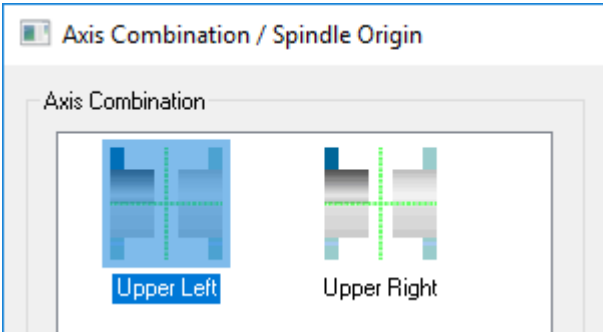
2. Select the chain shown in the picture below, by clicking points 1 and 2.



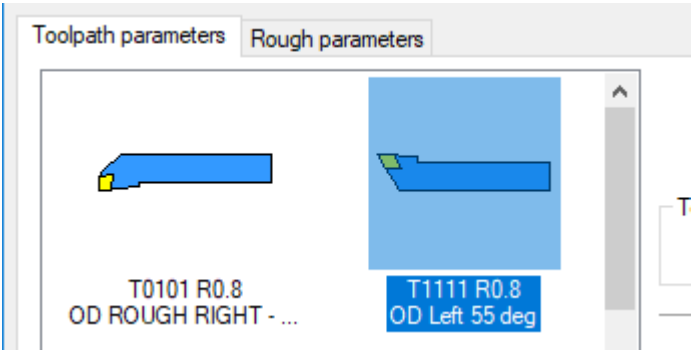
3. Click **OK** to close the **Chaining** dialog box.

   The **Lathe Rough** dialog box displays.

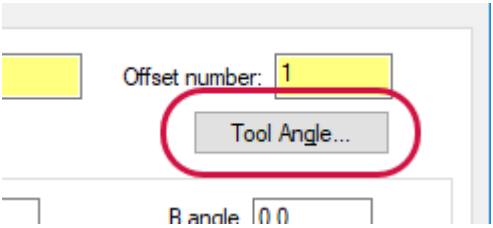
4. Click the **Axis Combination** button, ensure that the **Upper Left** axis combination is selected, and click **OK**.



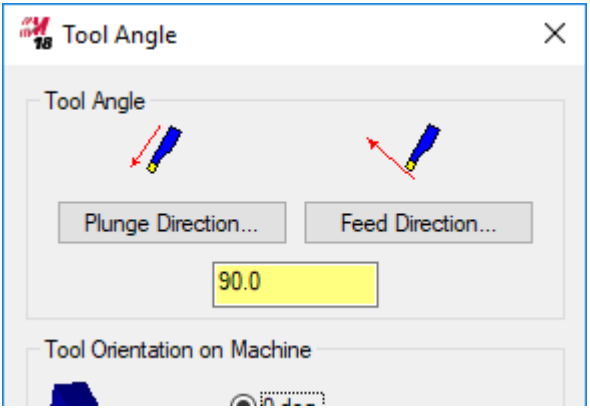
5. Choose the **OD Left 55 deg** tool.



6. Click the **Tool Angle** button.

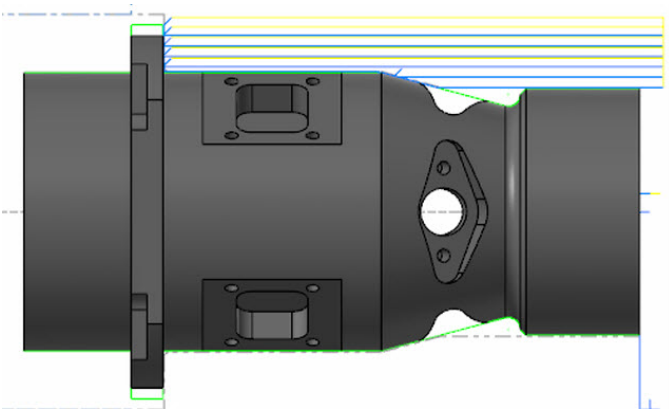


7. In the **Tool Angle** dialog box, set the angle to **90**, and click **OK**.



8. Click **OK** to close the **Lathe Rough** dialog box.

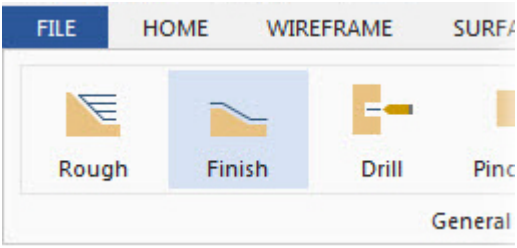
Mastercam creates the roughing toolpath, as shown below.



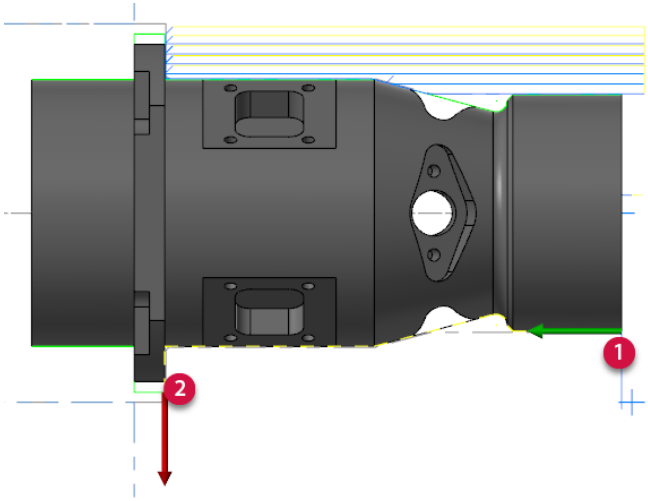
Exercise 3: The Finishing Toolpath

1. Choose **Finish** from the General toolpaths.

   The **Chaining** dialog box displays.



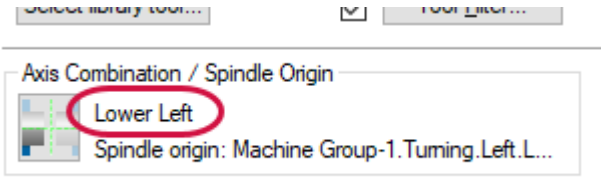
2. Select the chain shown in the picture below, clicking points 1 and 2.



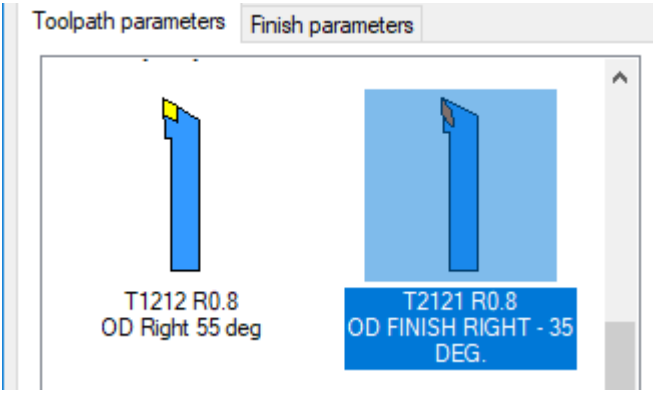
3. Click **OK** to close the **Chaining** dialog box.

   The **Lathe Finish** dialog box displays.

4. Ensure that the **Axis Combination** is set to **Lower Left**.



5. Select the **OD FINISH RIGHT - 35 DEG** tool.



6. Click **OK** to close the **Lathe Finish** dialog box.

   Mastercam creates the finishing toolpath.

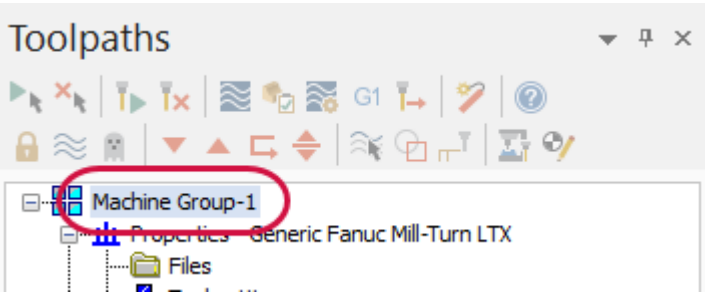
7. Save the file as `TurningToolpaths3-`*XX*, where *XX* is your initials.

Exercise 4: Simulating the Toolpaths

To test your operations, you will now simulate the job in the Machine Simulation application.

1. In Mastercam's Toolpaths Manager, click **Machine Group-1**.

   Mastercam selects all of the operations.

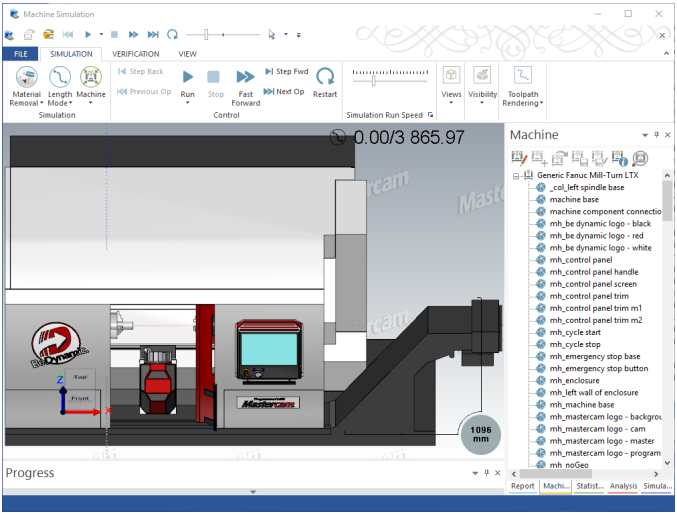


2. Click the **G1** (Post) button.

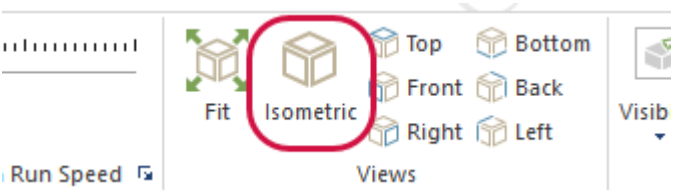
After a few moments, Code Expert displays Sync Manager. If you see the Properties pane, close it to free up space in the window.

3. In the Sync Manager tab, click the **Launch** button.

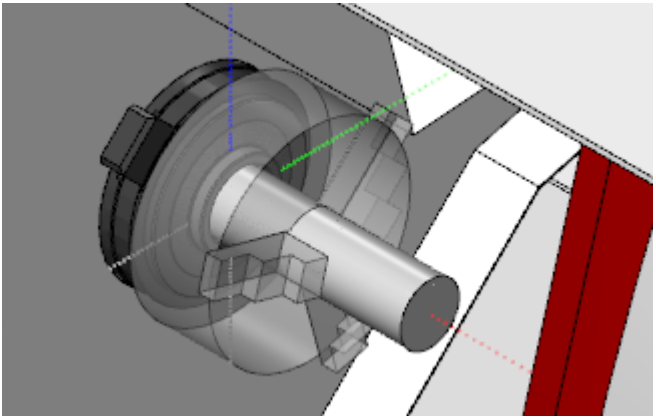
   After a few moments, the Machine Simulation window displays, as shown in the following picture.



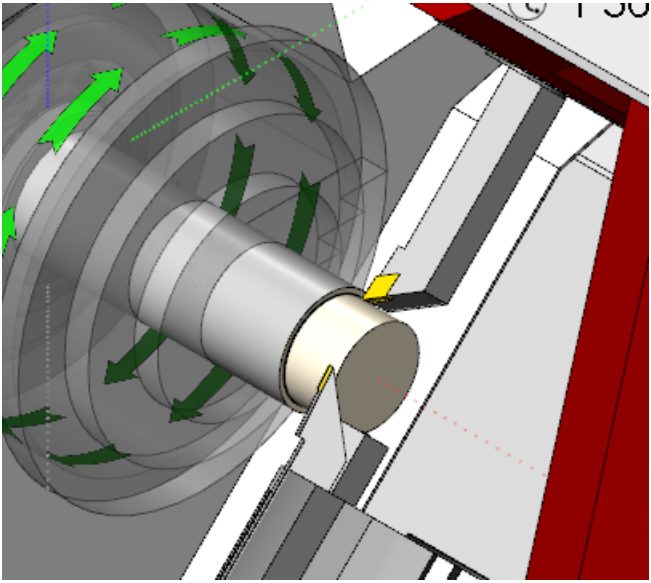
4. In the Simulation ribbon, click **Isometric**.



5. In the graphics window, place your mouse pointer over the left spindle, and use your mouse wheel to zoom in closer to the part's stock, as shown in the following picture.



6. In the Simulation tab, click the **Run** button.
7. Watch as Machine Simulation simulates the three operations you created. Notice how the machine tries to make two cuts simultaneously, as seen in the following picture.



Note: Use the **Simulation Run Speed** slider to speed up or slow down the simulation as needed.

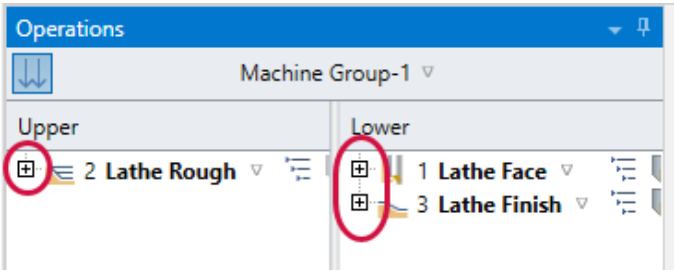
Exercise 5: Synchronizing the Toolpaths

1. Close Machine Simulation.

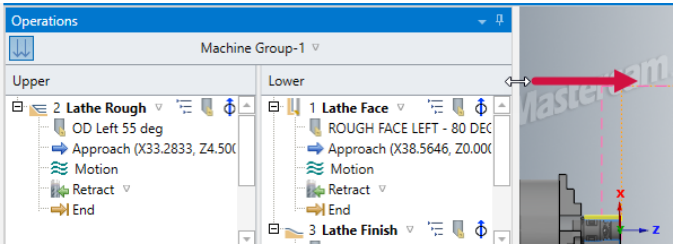
   You are now ready to do more work in Sync Manager.

2. In Sync Manager, click the plus signs (+) next to the operations in both the upper and lower streams. Alternatively, to expand all operations, right-click in either operation stream, and choose **Expand All Operations**.

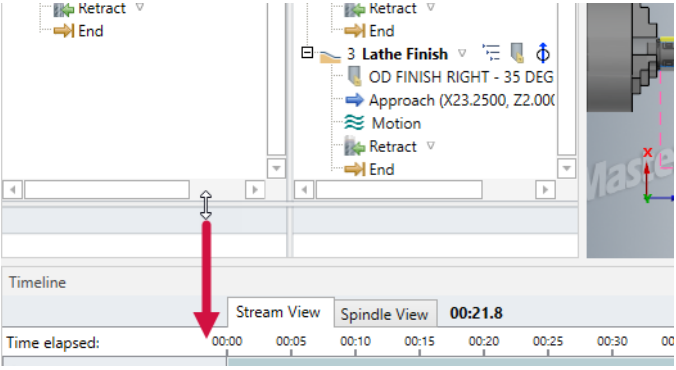
   Sync Manager expands the operations to display the sub-operations.



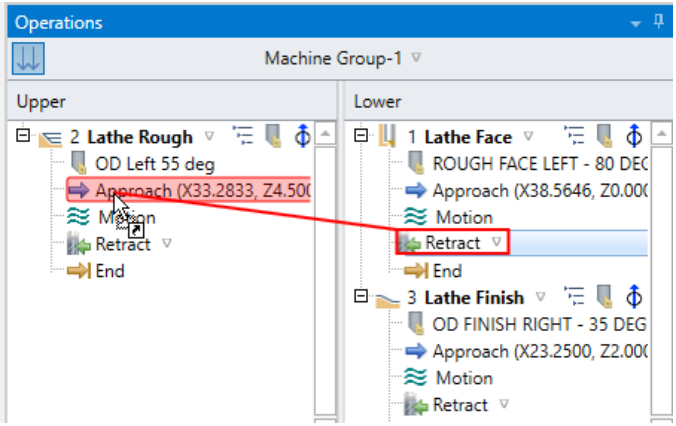
3. Drag the right-hand edge of the **Operations** pane to the right, expanding the pane until you can see more of the operations' descriptions, as shown below.



4. Drag down the bottom edge of the pane, expanding the pane vertically.

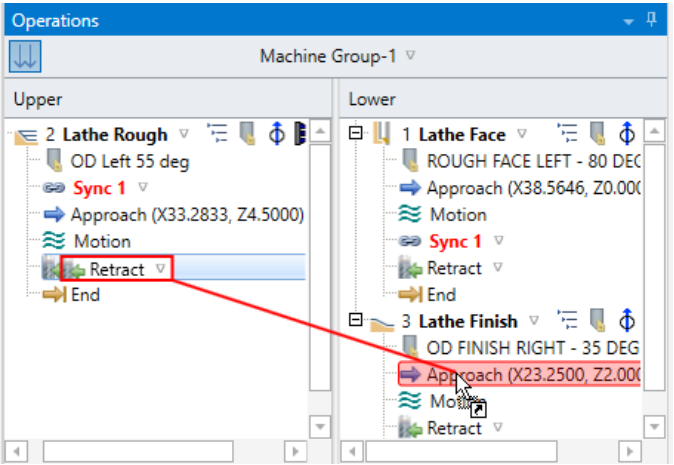


5. Use your mouse to drag-and-drop the **Lathe Face** operation's **Retract** to the **Lathe Rough** operation's **Approach**.



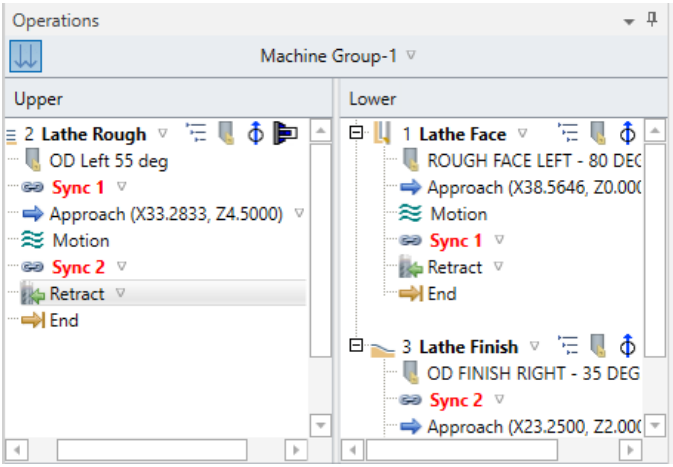
You have now synchronized the end of the facing toolpath with the start of the roughing toolpath. Sync Manager labels the sync point as **Sync 1**.

6. Drag the **Lathe Rough** operation's **Retract** to the **Lathe Finish** operation's **Approach**.

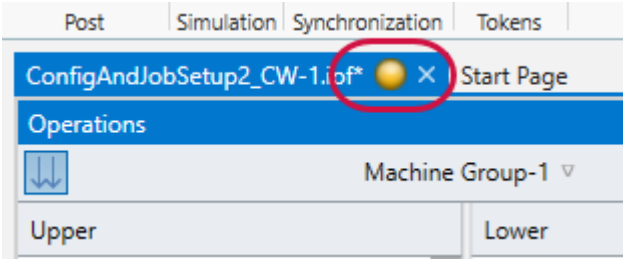


Sync Manager labels the second sync point as **Sync 2**.

You have now synchronized the end of the roughing toolpath with the start of the finishing toolpath. Your upper and lower streams should now look like the following picture.



At this point, the file's status icon is yellow, which means that Mastercam must be updated from Sync Manager. For more information on how Sync Manager and Mastercam share data, please see Sync Manager and Mastercam Interactions.



7. With the focus on the IOF file's tab, choose **File, Save**.

   Sync Manager writes the synchronized data back to Mastercam, and the status icon returns to green.

8. Click the **Launch** button, and run the simulation again.

Now the operations are synchronized. That is, they are guaranteed to run in the correct order, without one operation overlapping another.

9. Close Machine Simulation.
10. Back in Mastercam (not Sync Manager), save the part as `TurningToolpaths4-XX`, where *XX* is your initials.

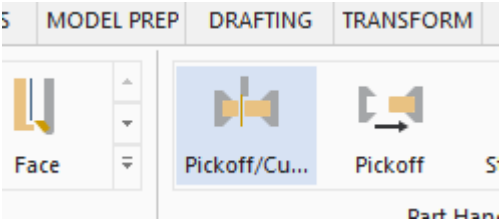
The Part Handling Operations

You now must transfer the part from the left spindle to the right, so that you can create the toolpaths to complete the job. To do this, you program pickoff, cutoff, or transfer operations that leave the part clamped in the right chuck, ready for cutting.

Exercise 6: Creating Part Handling Operations

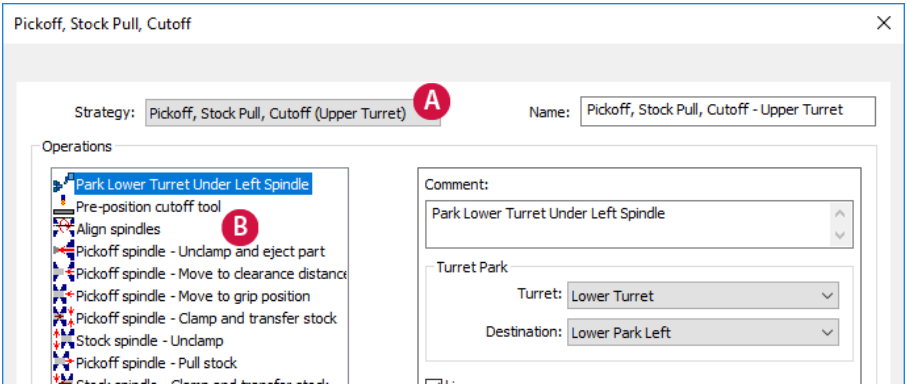
1. From Mastercam's **Turning** tab, choose **Part Handling, Pickoff/Cutoff**.

   The **Pickoff, Stock Pull, Cutoff** dialog box displays.



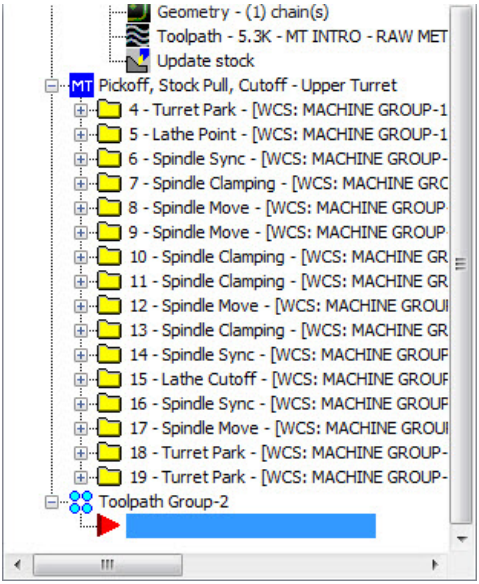
2. Verify that the Strategy is set to **Pickoff, Stock Pull, Cutoff (Upper Turret)**, as shown in the picture below.

   The Strategy selection (**A**) determines which operations appear in the Operations list (**B**).



3. Click **OK** to accept the default settings for your part handling strategy.

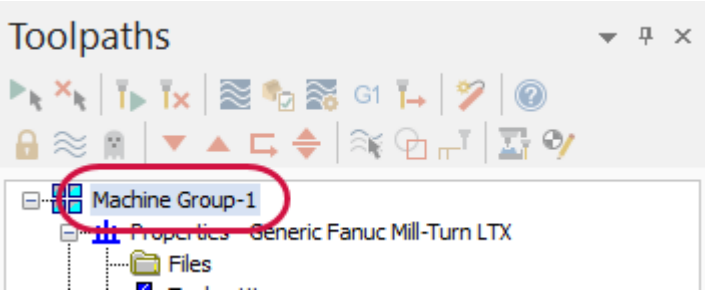
   Mastercam Mill-Turn creates the requested set of operations, as seen in the picture below.



Exercise 7: Simulating the New Operations

1. In Toolpaths Manager, click **Machine Group-1**.

   Mastercam selects all of the operations.

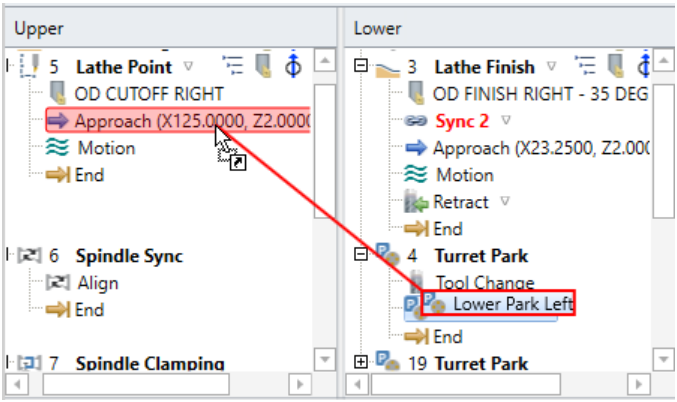


2. Click the **G1** button.

   Mastercam sends the new data to Sync Manager, and Sync Manager displays in Code Expert's window.

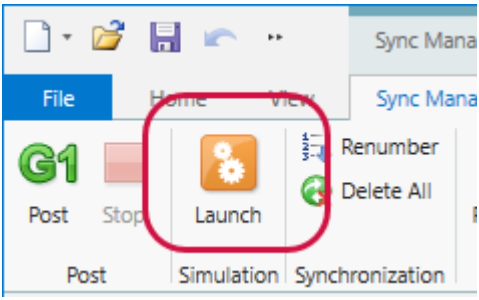
3. Sync operation **4 Turret Park, Lower Park Left** to operation **5 Lathe Point, Approach**, as shown in the picture below.

   Sync Manager creates a sync point labeled **Sync 3**. You have now synchronized the end of the finishing operation with the right spindle approach.

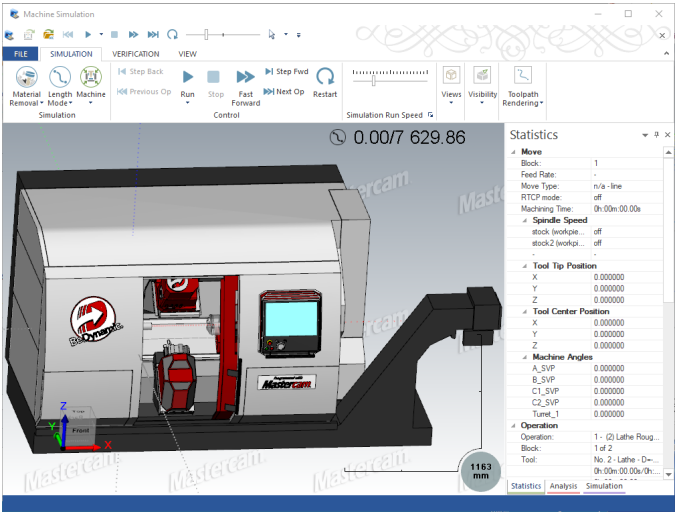


4. Save the file back to Mastercam, and then click the **Launch** button.

   In a few moments, Machine Simulation displays.



5. Rotate the graphics window so that you can see both spindles. (See the picture below.) To do this, place your mouse pointer over the graphics window, hold down your left mouse button, and drag the mouse.



6. Use your mouse to zoom in on the two spindles.

   Now you can see the simulation more clearly.

7. Click the **Run** button.

   Machine Simulation simulates the job's operations. When the simulation completes, the right chuck will be holding the part.

8. Close Machine Simulation.

9. In Mastercam, save the part as `TurningToolpaths5–XX`, where *XX* is your initials.

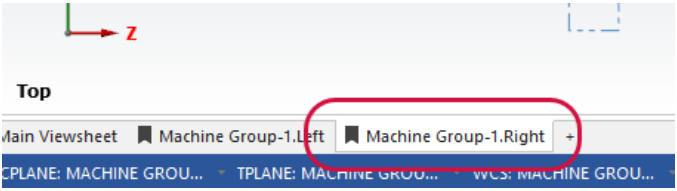
Programming the Right Spindle

Your part is now in the right spindle, and you can create the toolpaths to complete the part. For the following exercise, continue with your part from the previous section.

Exercise 8: The Facing Toolpath, Right Spindle

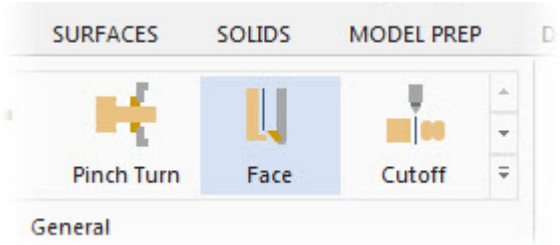
1. In Mastercam, select the **Machine Group-1.Right** viewsheet tab.

   The graphics window now displays the right spindle.



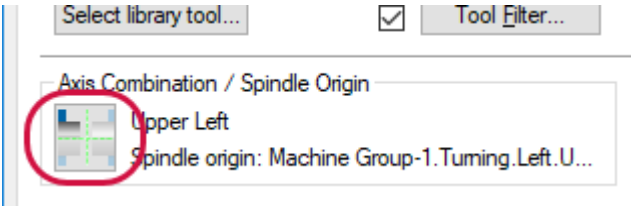
2. From Mastercam's Turning tab, choose **General, Face**.

   The **Lathe Face** dialog box displays.



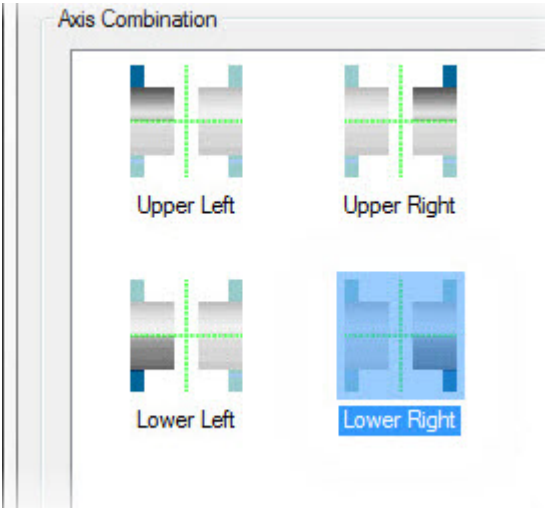
3. Click the **Axis Combination** button.

   The **Axis Combination / Spindle Origin** dialog box displays.

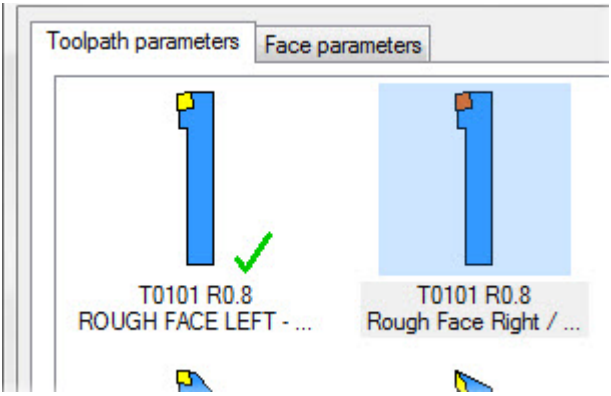


4. Select the **Lower Right** axis combination, and click **OK**.

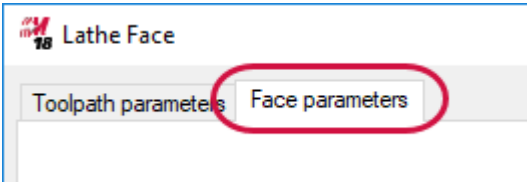
   The **Lathe Face** dialog box now shows tools that are appropriate for the lower-right axis combination.



5. Select the **Rough Face Right / 80 Deg** tool.



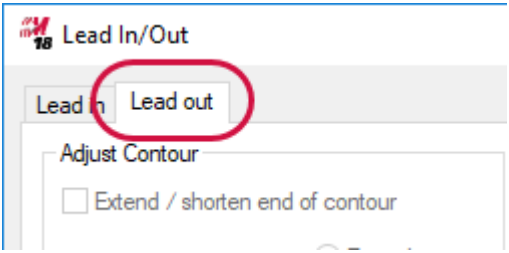
6. Click the **Face parameters** tab.



7. Ensure that the **Lead In/Out** checkbox is selected, and then click the **Lead In/Out** button.

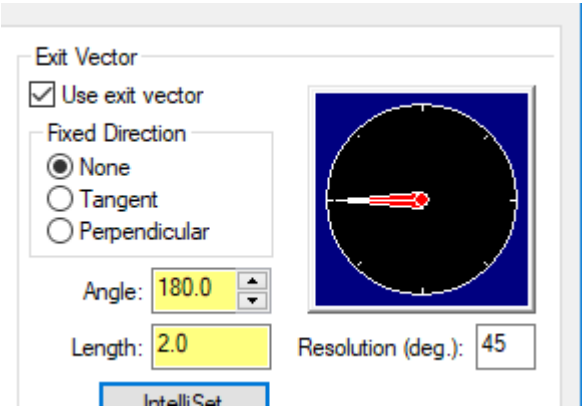
   The **Lead In/Out** dialog box displays.

8. Click the **Lead out** tab.

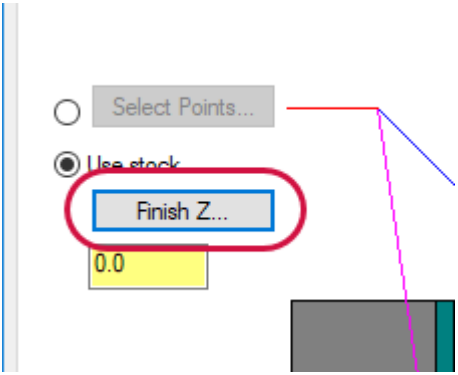


9. Change **Angle** to **180**.

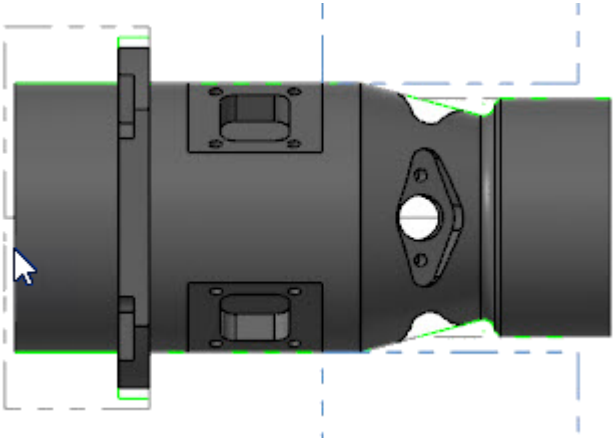
   The tool will now exit away from the part, rather than toward it.



10. Click **OK** to exit the **Lead In/Out** dialog box.
11. Click the **Finish Z** button.



12. Click the left face of the part, as shown in the picture below.



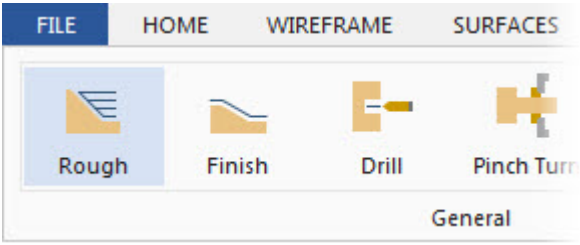
13. Click **OK** to exit the dialog box.

    Mastercam creates the facing toolpath based on your selected parameters, and the operation appears in Toolpaths Manager. (Your tool numbers may vary.)

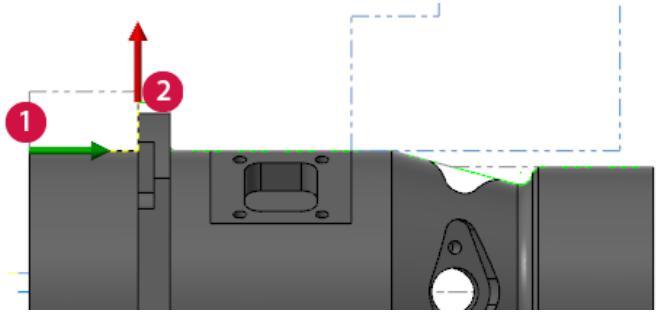
Exercise 9: The Roughing Toolpath, Right Spindle

1. From Mastercam's Turning tab, choose **General, Rough**.

    The **Chaining** dialog box displays.



2. Select the chain shown in the following picture, by clicking points 1 and 2.

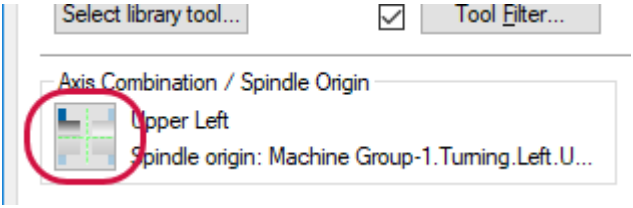


3. Choose **OK** to close the **Chaining** dialog box.

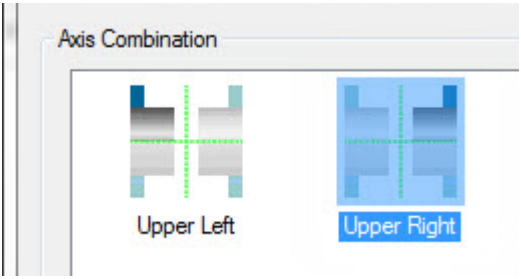
   The Lathe Rough dialog box displays.

4. Click the **Axis Combination** button.

   The **Axis Combination / Spindle Origin** dialog box displays.



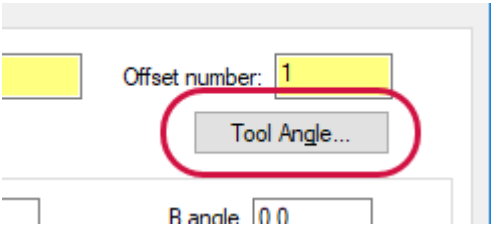
5. Ensure that the **Upper Right** axis combination is selected, and click **OK**.



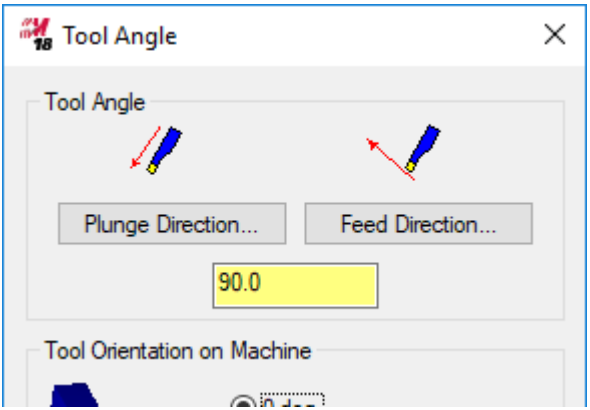
6. Choose the **OD ROUGH RIGHT - 80 DEG** tool.

7. Click the **Tool Angle** button.

   The **Tool Angle** dialog box appears.

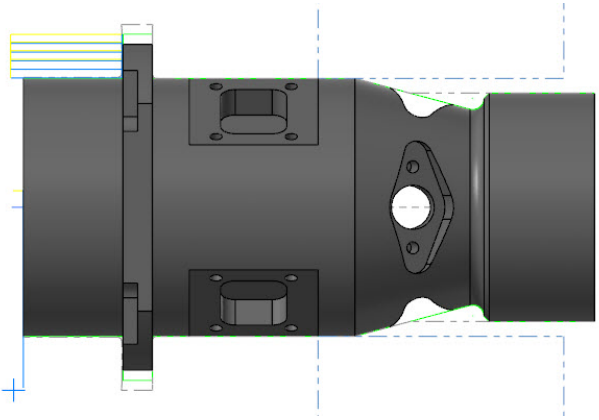


8. Set **Tool Angle** to **90**, and click **OK**.



9. Click **OK** to close the **Lathe Rough** dialog box.

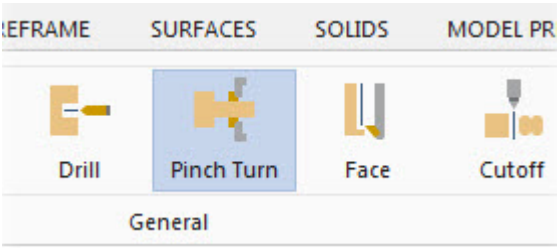
   Mastercam creates the roughing toolpath, as shown in the following picture.



Exercise 10: The Pinch-Turn Toolpath

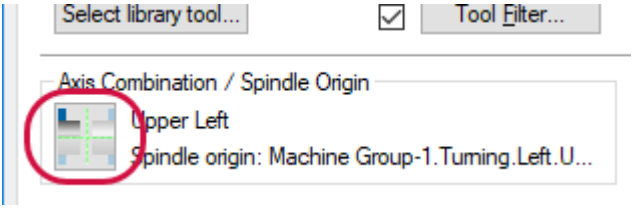
1. From Mastercam's Turning tab, choose **General, Pinch Turn**.

   The **Pinch Turn Associated** dialog box opens.

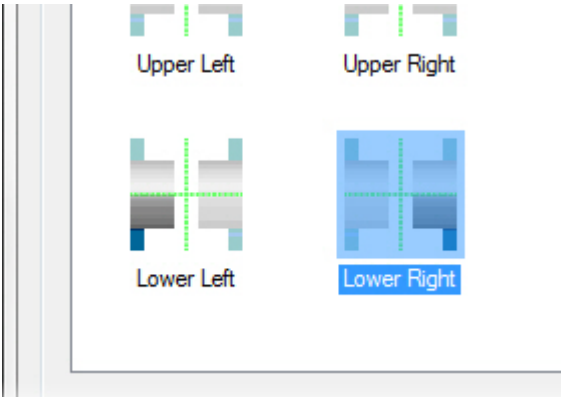


2. Click the **Axis Combination** button.

   The **Axis Combination / Spindle Origin** dialog box opens.

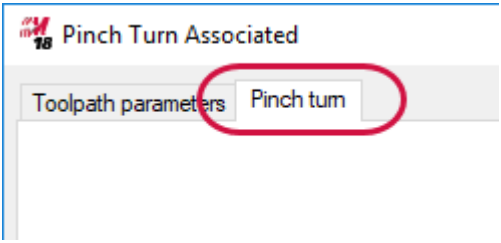


3. Set the **Axis Combination** to **Lower Right**, and click **OK**.



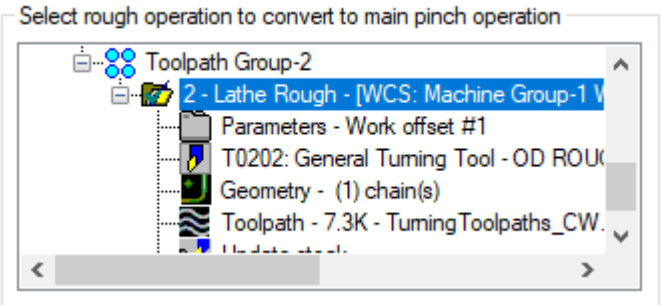
4. Choose the **OD RIGHT 55 DEG** tool.
5. Click the **Pinch turn** tab.

   The Pinch Turn settings appear in the dialog box.



6. In the operation list, select the **Lathe Rough** operation in **Toolpath Group-2**.

   This is the operation Mastercam will use to generate the pair of operations that make up the Pinch Turn toolpath.



7. Click **OK** to exit the dialog box.

   Mastercam creates the Pinch Turn toolpath.

Exercise 11: Synchronizing the Toolpaths

1. In Toolpaths Manager, click **Select all operations**.

   Mastercam selects all of the operations.

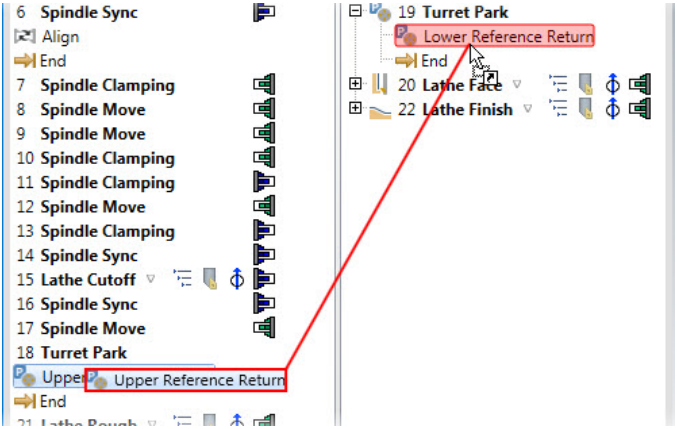
2. Click the **G1** button.

   After a few moments, Code Expert displays Sync Manager with the updated part data.

3. In the **Upper Stream** list, scroll down to operation **18 Turret Park**, and click the plus (+) to show the sub-operations.

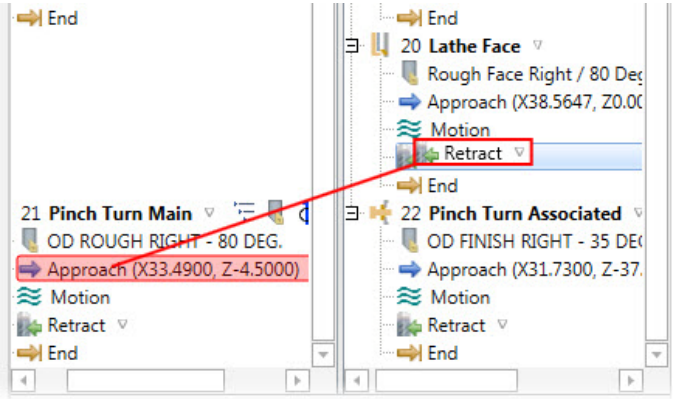
4. In the **Lower Stream** list, open operation **19 Turret Park**.

5. Drag-and-drop from operation **18 Turret Park, Upper Reference Return** to operation **19 Turret Park, Lower Reference Return**, as shown in the picture that follows.



You have now synchronized the end of the part handling operations with the turret park operation.

6. Drag operation 20's **Retract** to operation 21's **Approach**.



Sync Manager creates the sync point.

7. Choose **File, Save**.

   Sync Manager writes the synchronized data back to Mastercam.

8. Click the **Launch** button, and run the simulation.

All operations are synchronized.

9. Close Machine Simulation.
10. Back in Mastercam (not Sync Manager), save the part as `TurningToolpaths6-`*XX*, where *XX* is your initials.
11. Close Mastercam and Code Expert.

9

Milling Toolpaths

These exercises step through basic C- and Y-axis toolpathing using a partially programmed sample file. For simplicity, you import toolpaths from an existing file.

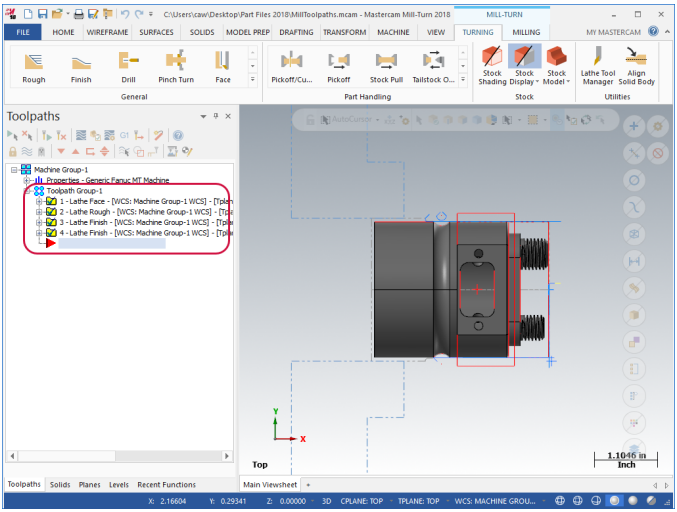
Lesson Goals

- Set up the part.
- Import milling operations.
- Complete the pocket operation.
- Complete the facing operation.
- Complete the rough contour operation.
- Complete the finish contour operation.

Exercise 1: Setting Up the Part

1. Launch Mastercam.
2. Open the file `MillToolpaths`, which is included with this tutorial, and if the **System Configuration** dialog box displays, click **OK** to switch system units.

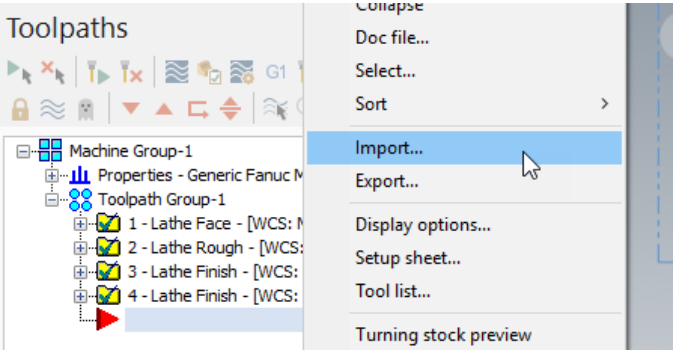
In Toolpaths Manager, you can see that four lathe operations are already programmed (see the picture below). To start the C- and Y-axis milling examples, you will import operations from a sample part.



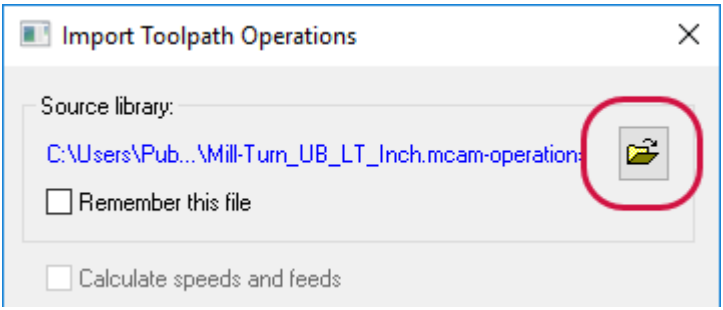
Exercise 2: Importing the Milling Operations

1. Right-click in **Toolpaths Manager**, and select **Import**.

   The **Import Toolpath Operations** dialog box appears.



2. Click the **Select Operations File** button.

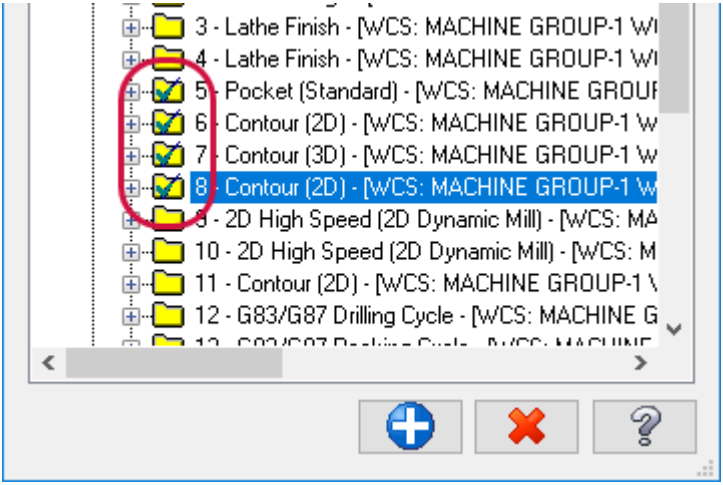


Note: To see the file when browsing, you must change the file extension filter from **Mastercam Operation Files** to **Mastercam Files**.

3. Browse to and open the file `FixSyncs`, which is included with this tutorial.

   The operations contained in the selected file appear in the **Import Toolpath Operations** dialog box.

4. Select operations **5** through **8** (hold down [**Shift**] to select multiple items), and click the **Import Operations** button (the one that looks like a plus sign).



5. Click **Done** to close the **Import Toolpath Operations** dialog box.

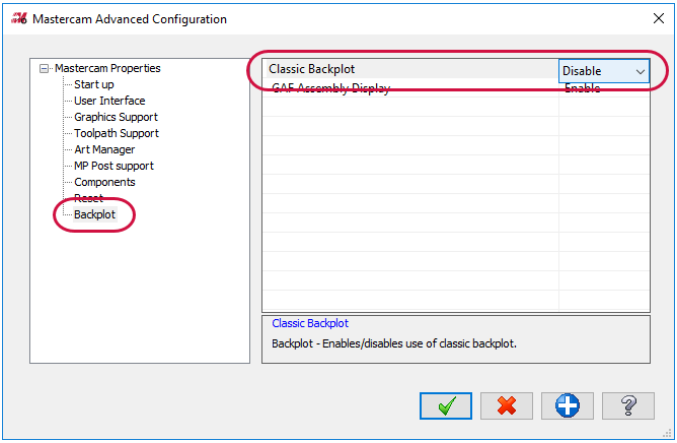
   After the import, **Toolpaths Manager** contains the operations.

6. In your Windows Start menu, go to **All Programs, Mastercam 2018, Advanced Configuration**.

   The **Mastercam Advanced Configuration** dialog box opens.

7. Select the **Backplot** properties page, and disable the **Classic Backplot** entry, as shown in the following picture. Then click **OK**.

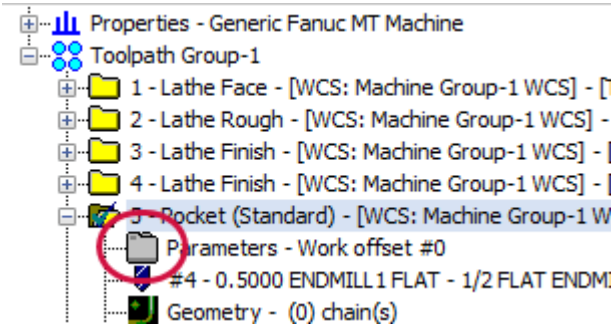
   Mastercam will now backplot and verify with Mastercam Simulator.



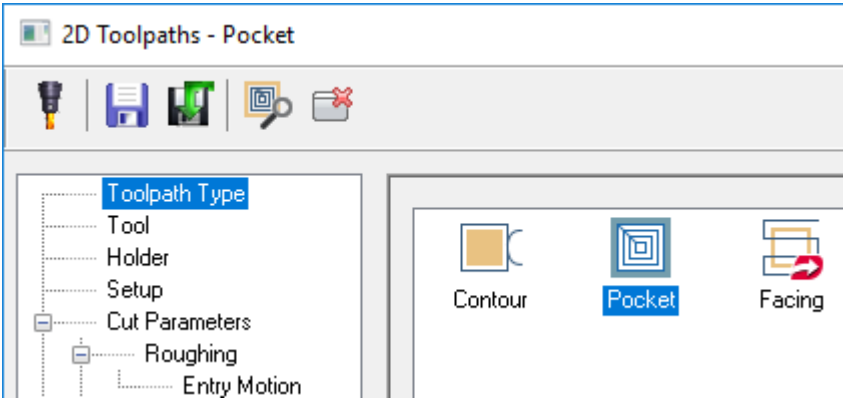
Exercise 3: Completing the Pocket Operation

1. Click the **Parameters** icon in operation 5.

   The **2D Toolpaths - Pocket** dialog box displays.

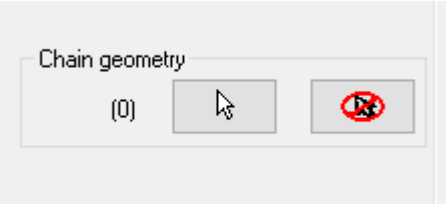


2. Verify that the **Toolpath Type** is set to **Pocket**.



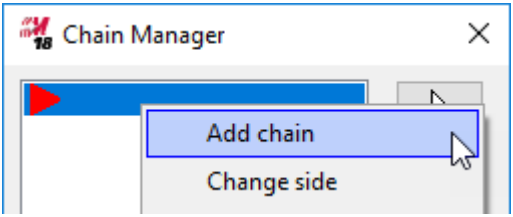
3. Click the **Select chains** button.

   The **Chain Manager** dialog box appears.

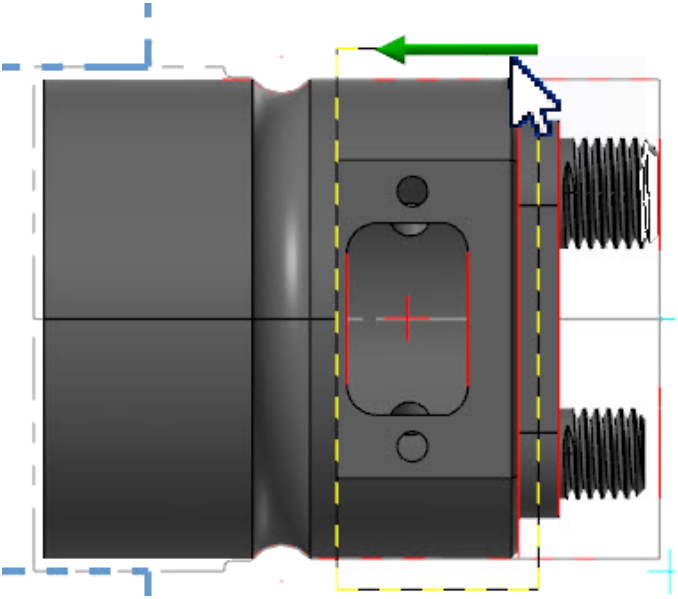


4. Right-click in the dialog box, and choose **Add chain** from the pop-up menu.

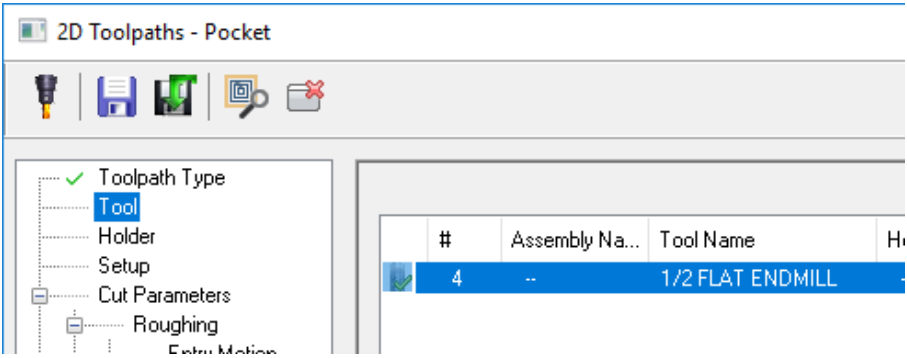
   The **Chaining** dialog box appears.



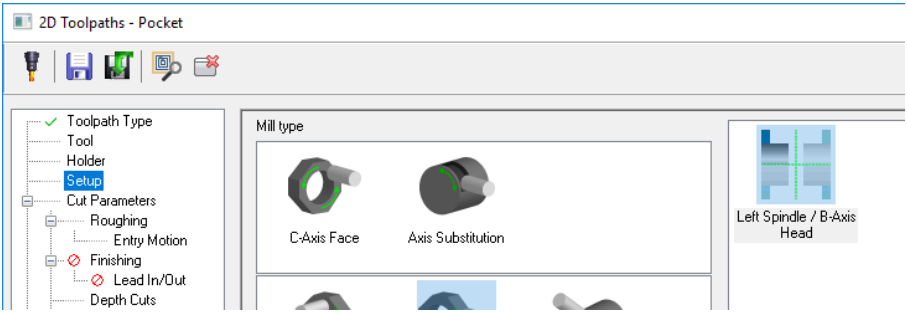
5. Chain the rectangle shown below, and click **OK** in the **Chaining** and **Chain Manager** dialog boxes.



6. Select **Tool**, and then verify that the tool is a ½-inch flat endmill.



7. On the **Setup** page, ensure that **Y-Axis Cross** and the **Left Spindle / B-Axis Head** axis combination are selected.



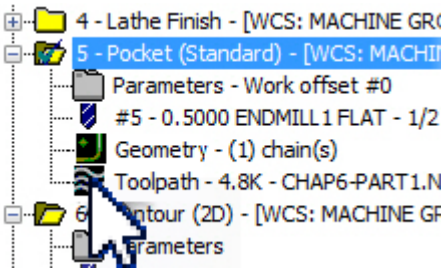
Note: Because there is no existing toolplane for the selected geometry, Mastercam automatically creates the proper plane based on the selected geometry, the operation orientation, and axis combo. The display at the bottom of the dialog box tells you the name of the new plane.

8. Click **OK** to close the **2D Toolpaths - Pocket** dialog box.
9. Click **Regenerate all selected operations**.

   Mastercam regenerates the pocket operation.

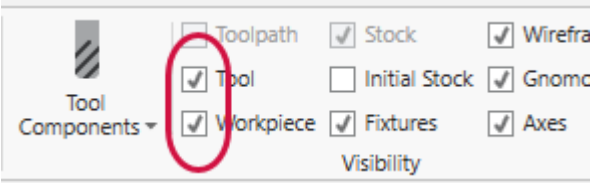
10. Select operation 5's **Toolpath** icon in the Toolpaths Manager.

    The Mastercam Simulator window opens.



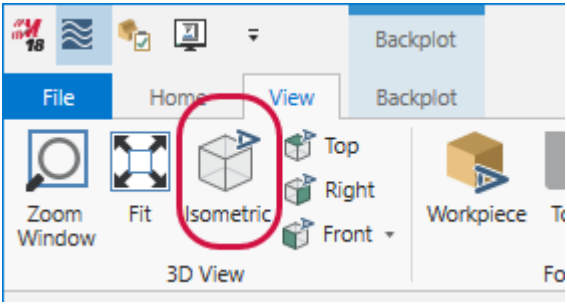
11. In the **Home** tab's **Visibility** group, ensure that **Tool** and **Workpiece** are selected.

    Mastercam Simulator will now show the tool and the workpiece in its graphics area.



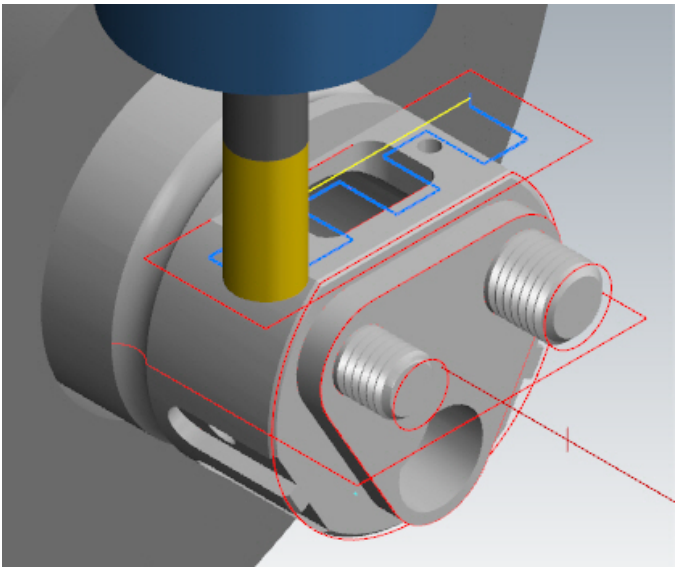
12. On the **View** tab, click **Isometric**.

    The view changes to the isometric orientation. Note that the view in Mastercam also changes to isometric.



13. Click the **Play** button to run backplot.

Mastercam backplots the operation, as shown in the second picture that follows.



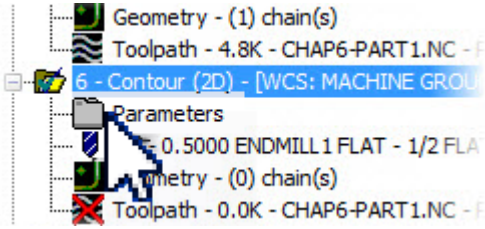
14. Close Machine Simulation.

Exercise 4: Completing the Facing Operation

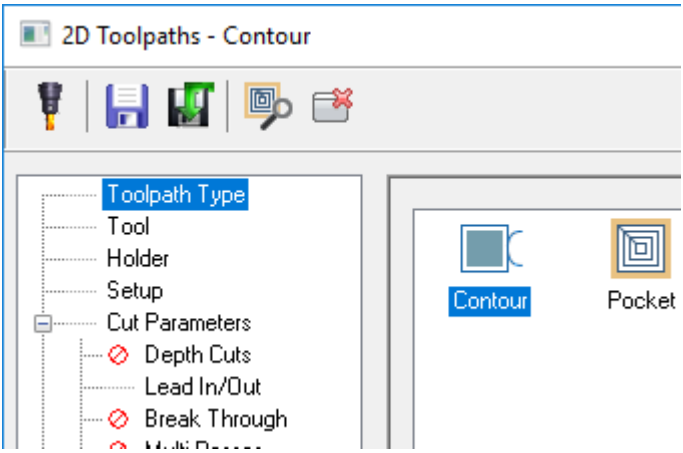
Next, you finish the short wall in back of the pocket.

1. Click operation 6's **Parameters** icon.

   The **2D Toolpaths - Contour** dialog box displays.

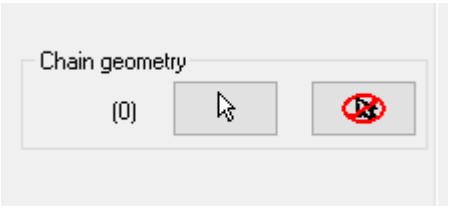


2. Ensure that **Toolpath Type** is set to **Contour**.



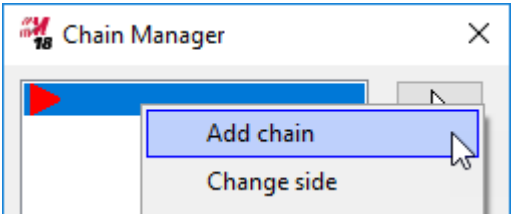
3. Click the **Select chains** button.

   Chain Manager appears.

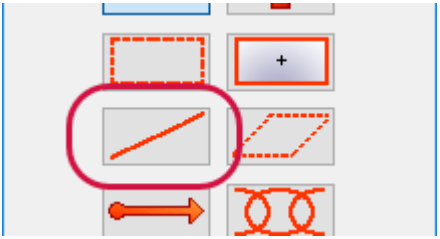


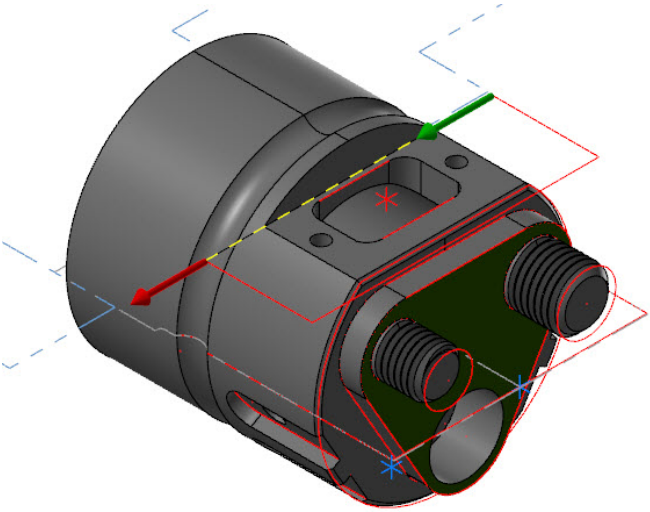
4. Right-click, and choose **Add chain**.

   The **Chaining** dialog box appears.



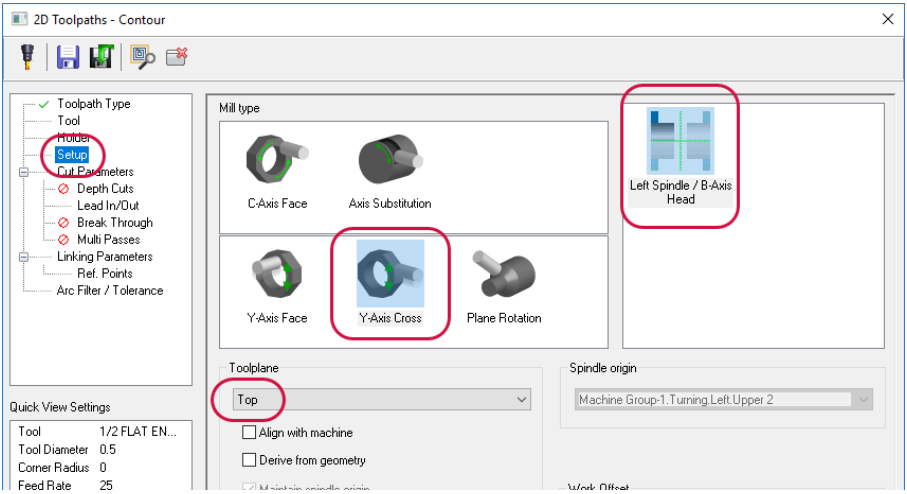
5. In the **Chaining** dialog box, choose the **Single** button, and then select the line at the base of the wall, as shown in the second following picture.





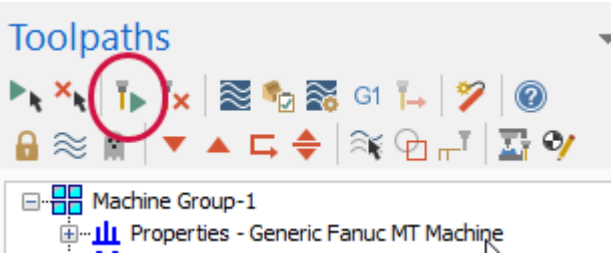
6. Click **OK** in both the **Chaining** and **Chain Manager** dialog boxes.

7. On the **Setup** page, select **Y-Axis Cross**, **Left Spindle / B-Axis Head**, and **Top**, as shown below.

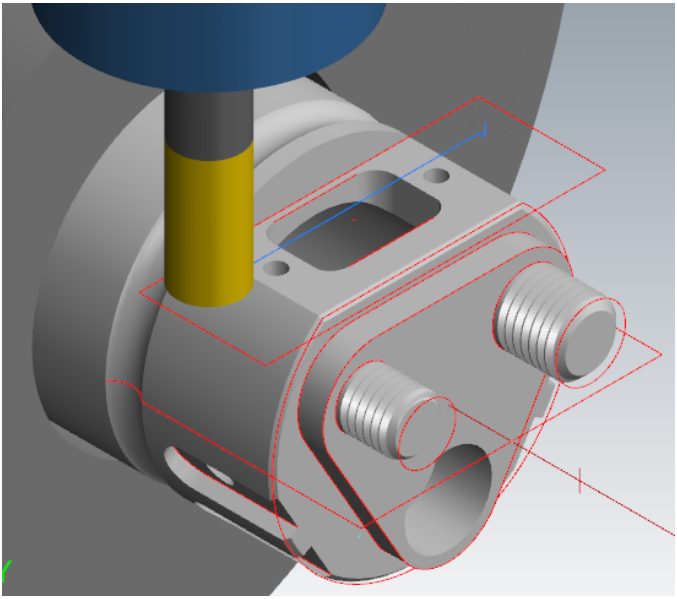


8. Click **OK** to close the **2D Toolpaths - Contour** dialog box.
9. Click **Regenerate all selected operations**.

   Mastercam regenerates the operation.



10. Select operation 6's **Toolpath** icon, and backplot the operation in isometric view, as shown below.

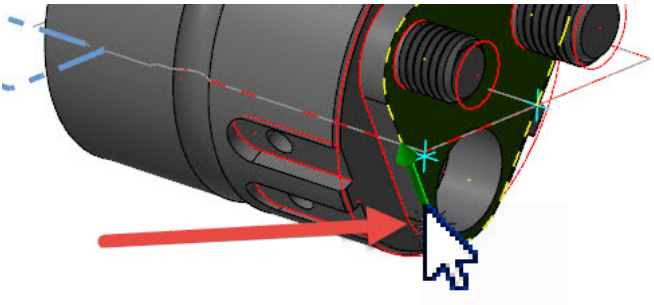


11. Close Mastercam Simulator.

Exercise 5: Completing the Rough Contour Operation

Next, you rough out the triangular section at the front of the part.

1. Re-chain geometry for operation **7 – Contour (3D)** as shown here.



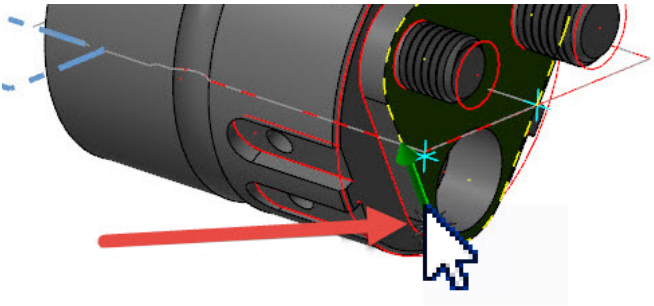
2. Set **Toolpath Type** to **Contour**, and for **Tool**, select the ½-inch flat endmill.

3. On the **Setup** page, select **C-Axis Face** and the **Left Spindle / B-Axis Head** axis combo.
4. Regen the operation, and backplot it.
5. Close Machine Simulation.

Exercise 6: Completing the Finish Contour Operation

You next finish the triangular section at the front of the part.

1. Re-chain geometry for operation **8 – Contour (2D)** as shown here. (Select the same contour as in the previous rough operation. Tip: Use the **Last** button in the chaining dialog to select the last chain selected in the previous toolpath.")



2. Set **ToolpathType** to **Contour**.
3. On the **Setup** page, select **C-Axis Face** and the **Left Spindle / B-Axis Head** axis combo.
4. Click **OK** to close the **2D Toolpaths - Contour** dialog box.
5. Regen the operation, and backplot it.
6. In Mastercam, save the part as `MillToolpaths2_`*XX*, where *XX* is your initials.
7. Close the Simulation, Code Expert, and Mastercam windows.

10

More about Sync Manager

Sync Manager lets you create sync points for multi-stream machines and create custom job setups that ensure valid output. You have already used Sync Manager in previous lessons. This lesson demonstrates additional features you might have not yet discovered.

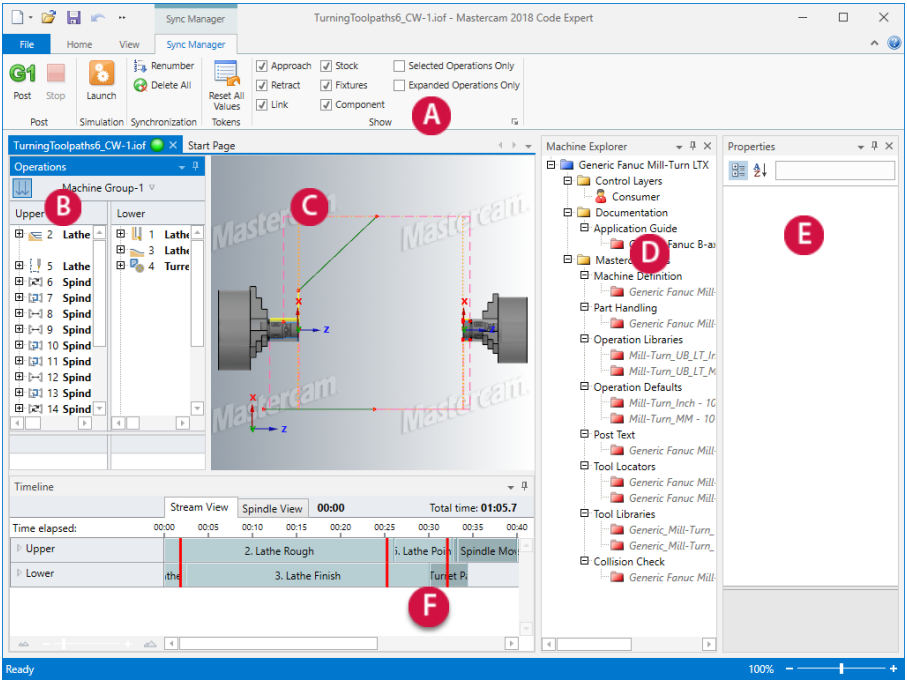
Lesson Goals

- Explore Sync Manager's window.
- Customize the display.
- Customize the operation settings.
- Use the Timeline.
- Understand Sync Manager and Mastercam interactions.

Sync Manager's Window

Sync Manager's window has six main areas. Refer to the following picture as you work through the procedures in this lesson.

- **A** Ribbon
- **B** Operations pane
- **C** Graphics window
- **D** Machine Explorer
- **E** Properties grid
- **F** Timeline



Exercise 1: Customizing the Display

You can easily modify Sync Manager's display to suit your needs. Try the following procedure to discover some common features.

1. In Mastercam, load the part file named `SyncManager`, which is included with this tutorial. Click **OK** to change units if asked.

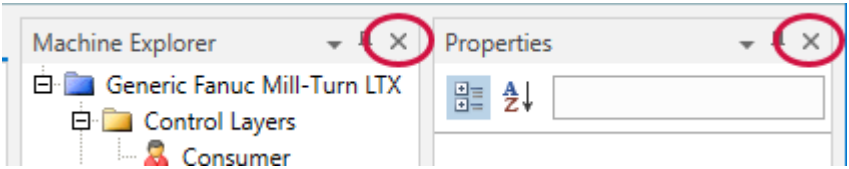
   Notice that Toolpaths Manager contains a machine group for a Generic Fanuc MT Machine. Notice also that the toolpath group contains facing, roughing, and finishing toolpaths for the part displayed in the graphics window.

2. Save the file as `SyncManager_`*XX*, where *XX* is your initials.

3. Click the **G1** button.

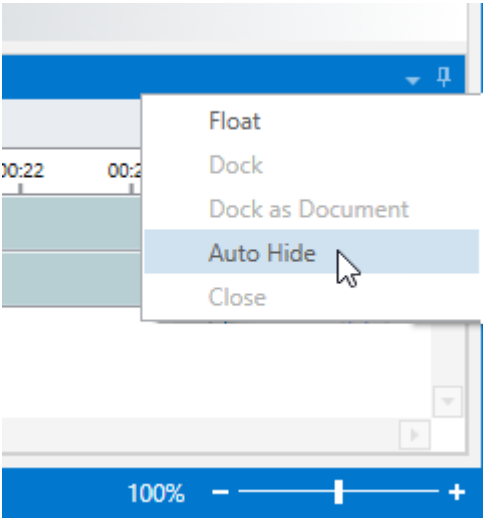
   In a few moments, Sync Manager appears in the Code Expert window.

4. If you have Machine Explorer and the Properties grid open, close them so that you can see more of the part.

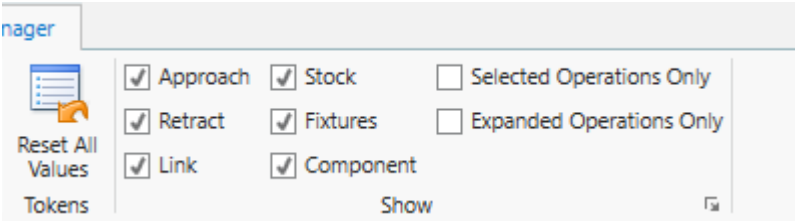


5. Click the small arrow in the Timeline's upper-right corner, and choose **Auto Hide**.

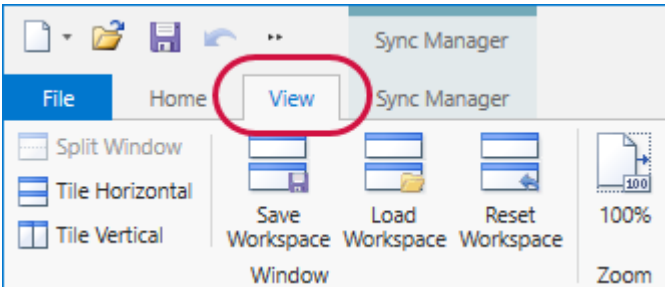
   The Timeline minimizes to a tab.



6. In the Sync Manager tab, experiment with the checkboxes in the **Show** group to toggle display elements on and off.

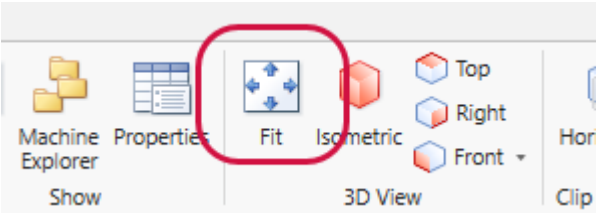


7. Click the **View** tab.

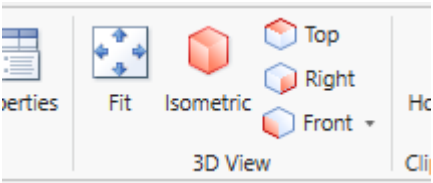


8. Click the **Fit** button.

   The geometry zooms to fit the graphics window.



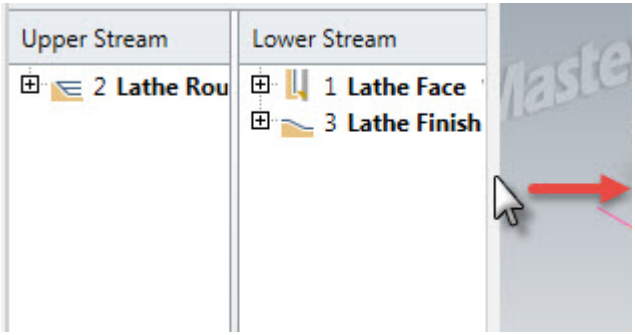
9. Click the view buttons to see the part from different angles.



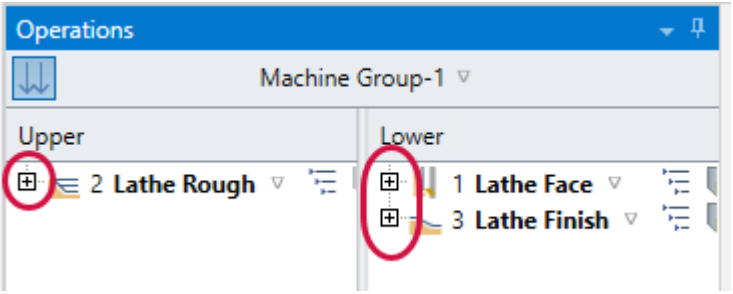
Exercise 2: Customizing Operation Settings

You can set options for individual parameters by clicking small inverted triangles in the Operations pane. Try the following procedure to see how this works.

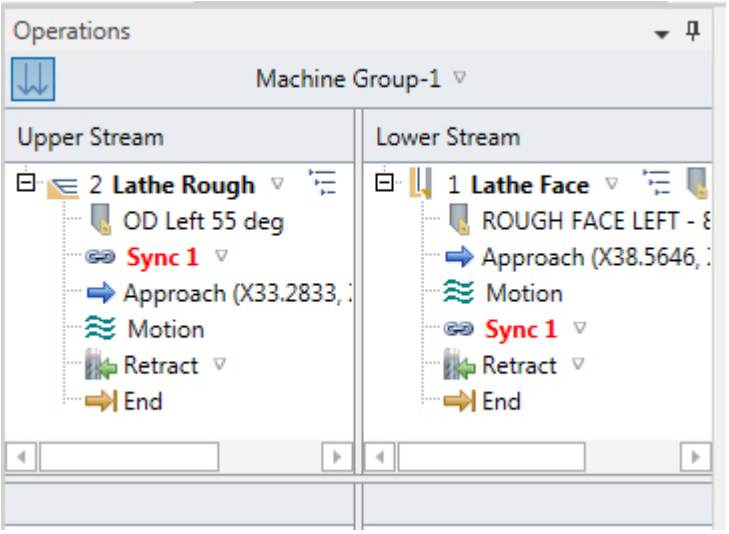
1. With your mouse, drag the right-hand edge of the **Operations** pane to increase its width.



2. Click the plus signs (+) next to the operations in the **Operations** pane.

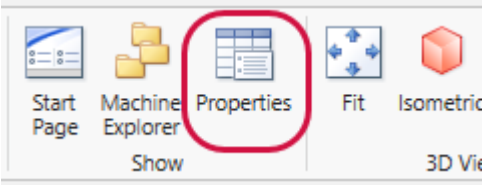


Sync Manager expands the operations, as shown below.

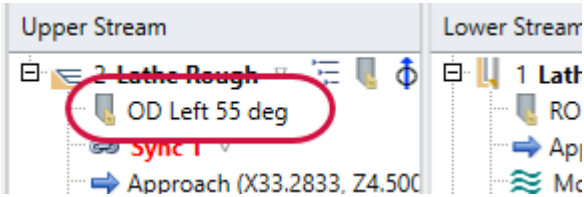


3. Go to the **View** tab, and click **Properties**.

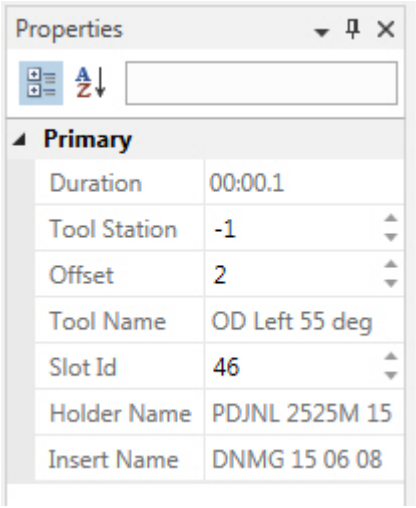
   The **Properties** grid appears.



4. Click **OD Left 55 deg** in the **Operations** pane.

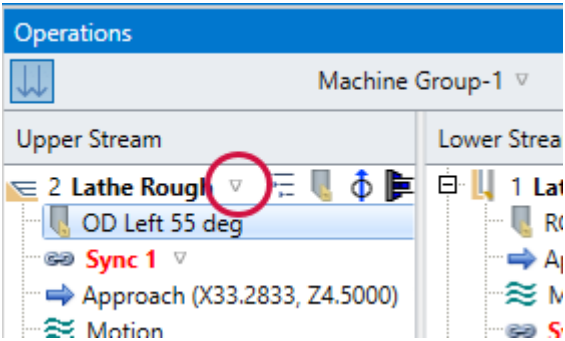


Information about the selected item appears in the Properties grid, as shown in the following picture.



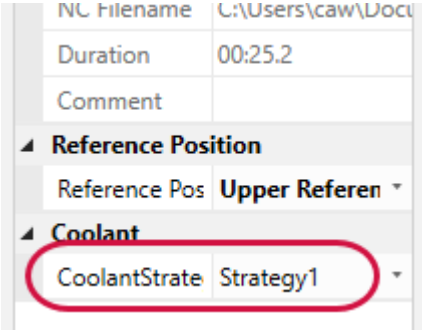
5. Back in the **Operations** pane, click the small inverted triangle to the right of **Lathe Rough**.

   A small box appears into which you can enter option values.



6. Change the **Coolant** value to **Strategy1**, and then click outside of the box to accept your change.

   The option's value in the **Properties** grid also changes. You can edit your operations in either the **Operations** pane or the **Properties** grid.

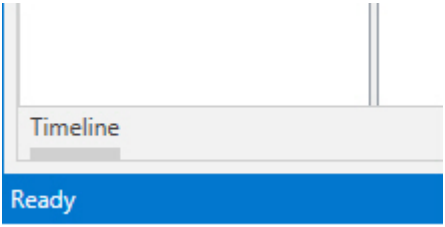


7. Save the file in Sync Manager and then in Mastercam.

Exercise 3: Using the Timeline

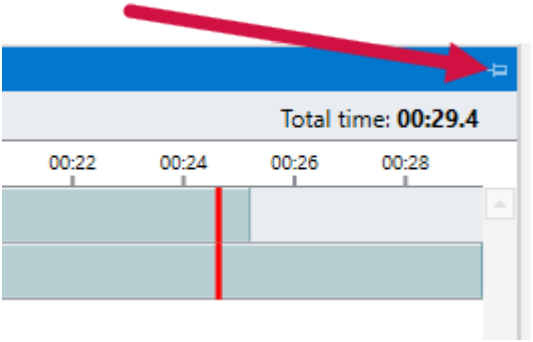
1. Click the **Timeline** tab in Code Expert's lower-left corner.

   The Timeline displays.



2. In the Timeline's upper-right corner, click the pin icon.

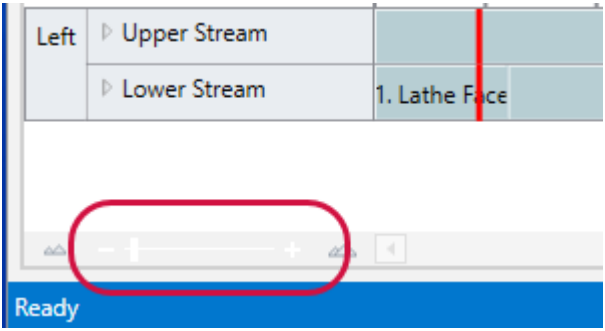
   The Timeline's **Auto Hide** option is now off, and the Timeline no longer minimizes to a tab.



   The Timeline displays the operations horizontally instead of vertically, as they are displayed in the **Operations** pane.

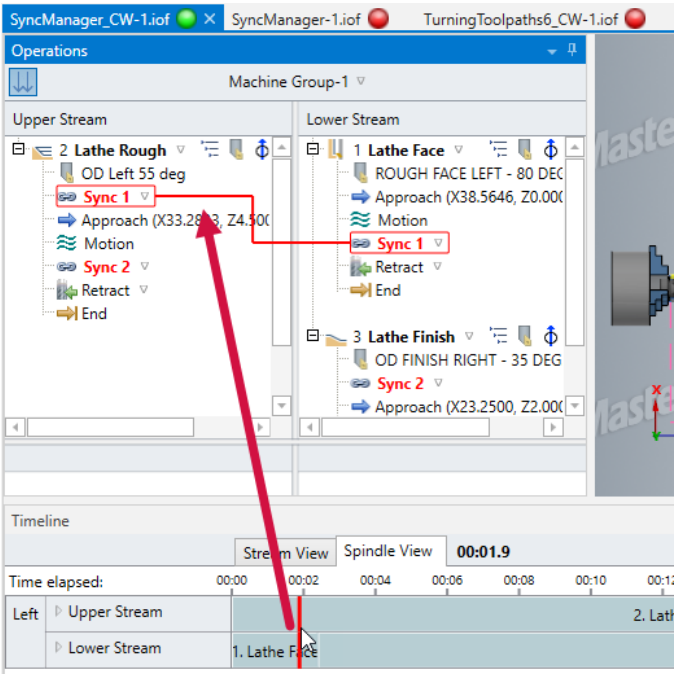
3. If necessary, in the Timeline's lower-left corner, drag the zoom slider all the way to the left.

   You should see the three operations and two red lines. (You may need to maximize the Sync Manager window.) The red lines are the sync points created in the Operations pane.



4. Hover your mouse over a sync point in the Timeline.

   Sync Manager displays a tooltip for the sync and outlines the equivalent sync point in the **Operations** pane, as shown in the following image.



5. Close Code Expert, without saving changes you may have made to the file.

Exercise 4: Sync Manager and Mastercam Interactions

Because Sync Manager and Mastercam share data, their interactions are tightly controlled. When a part is open in both, you can access either application freely only when the data in both is up to date.

Mastercam is the master application with respect to Sync Manager; the part data in Mastercam represents the current state of the part. Sync Manager uses a working copy of that data. When you save data from Sync Manager, you are updating the part file in Mastercam. To see how these interactions work, try the following exercise.

Exercise 5: Changing Data in Sync Manager

1. Load the file `SyncManager2` (provided with this tutorial) into Mastercam, and save it as `SyncManager2_`*`XX`*, where *XX* is your initials.

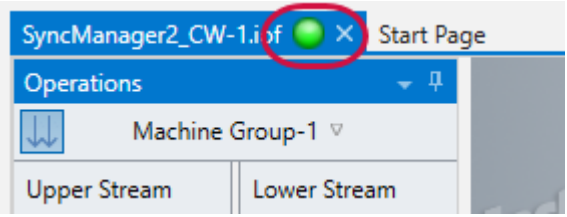
   Mastercam not only loads the part, but also starts the Code Expert application. Note that Code Expert is not yet displaying data for the part you loaded.

2. In Mastercam's **Toolpaths Manager**, select all operations, and then click the **G1** button.

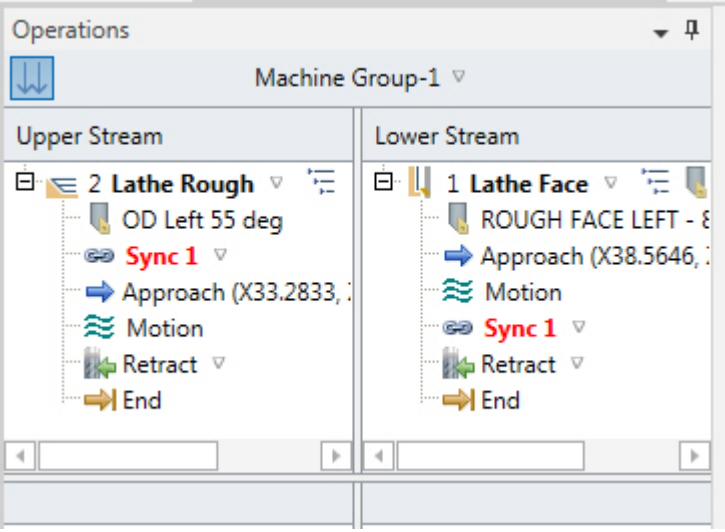
   Sync Manager opens the part. At this point, you can access both Mastercam and Sync Manager because both applications have a clean operation set.

3. In Sync Manager, notice the green circle on the file's tab.

   Green indicates that Mastercam's and Sync Manager's data sets match.



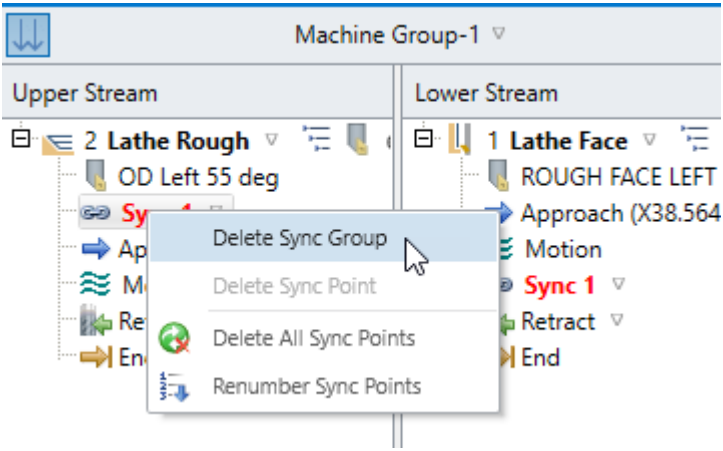
4. In Sync Manager, expand the **Lathe Rough** and **Lathe Face** operations, as shown below.



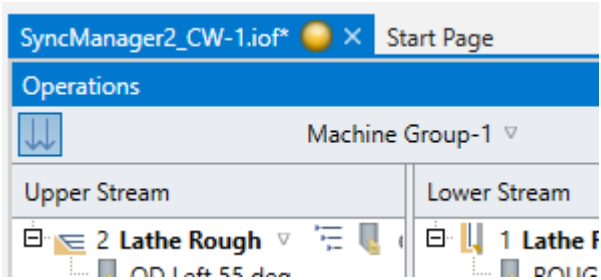
Note: You can use the **Connect Stream List Display** button (shown below) to lock together the lists, operations, and sync points so that they remain horizontally aligned.

5. Right-click on a **Sync 1** sync point, and choose **Delete Sync Group** from the pop-up menu.

   Sync Manager removes the pair of sync points.



6. Notice that the status indicator is now yellow, which means Mastercam's and Sync Manager's data sets no longer match.



7. Try to select something in the Mastercam window.

   Sync Manager retains control, disabling access to Mastercam. You cannot use Mastercam because the loaded part does not yet contain the changes you made in Sync Manager.

8. In Sync Manager, click the **Save** button.

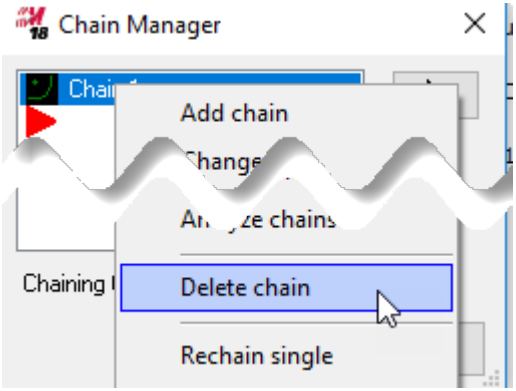
   Sync Manager's status indicator turns green.

9. Try to select something in the Mastercam window.

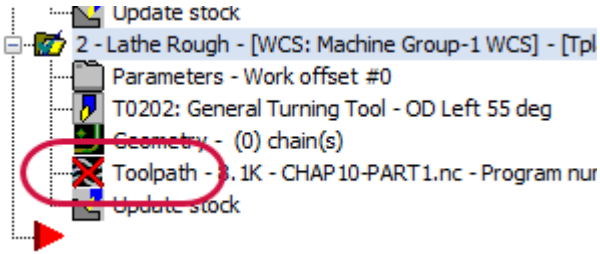
   Sync Manager lets you access Mastercam because its data is now up to date.

Exercise 6: Changing Data in Mastercam

1. In Mastercam, click the **Geometry** icon in the **Lathe Rough** operation, and delete the chain.

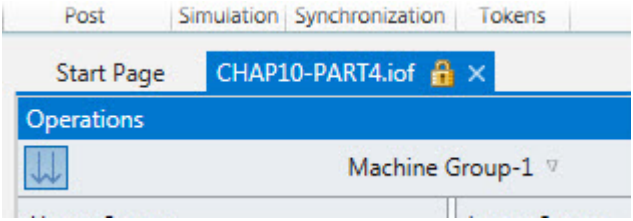


   Mastercam marks the operation as dirty.

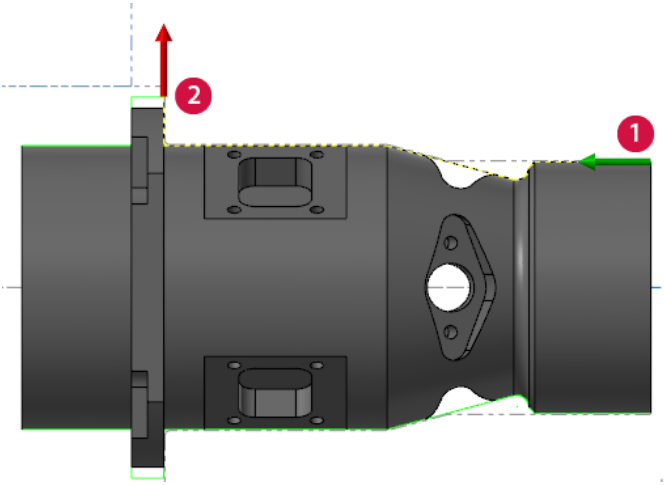


2. Notice that Sync Manager's status icon has changed to a lock.

   The lock indicates that the changed part in Mastercam does not match the data in Sync Manager.



3. Rechain the geometry (select **Geometry**, right-click in the **Chain Manager** dialog, and choose **Add chain**), clicking at the points shown below.

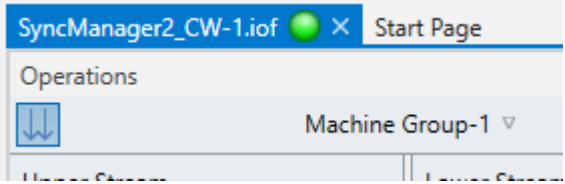


4. Click the **Regenerate all selected operations** button.

   Mastercam regenerates the operation, resolving its dirty state.

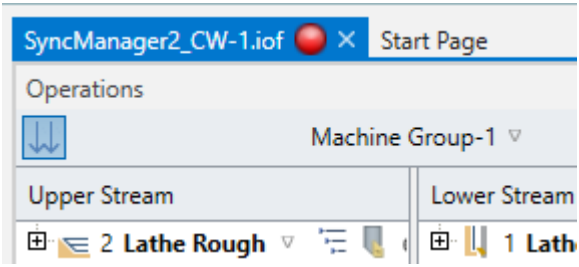
5. Select all operations, and then click the **G1** button.

   Mastercam sends the updated part data to Sync Manager, whose status indicator again shows green.



6. In Mastercam, select **File, New**, and, when prompted, choose **Don't Save**.

   Sync Manager's status indicator shows red, which indicates that Mastercam and Sync Manager are no longer linked.



7. Close all windows.

The table below describes Sync Manager's status icons.

Sync Manager's data matches Mastercam's data.	Sync Manager needs to be rerun from Mastercam.
Changes in Sync Manager need to be saved.	Sync Manager and Mastercam are disconnected.

11

More about Machine Simulation

Use Machine Simulation to verify your toolpaths, watching as a graphical representation of the machine cuts the part. Using Machine Simulation helps you find problems before the actual machining. (For more information about Machine Simulation, please read the *Machine Simulation* tutorial, available from your Reseller or the Mastercam website.)

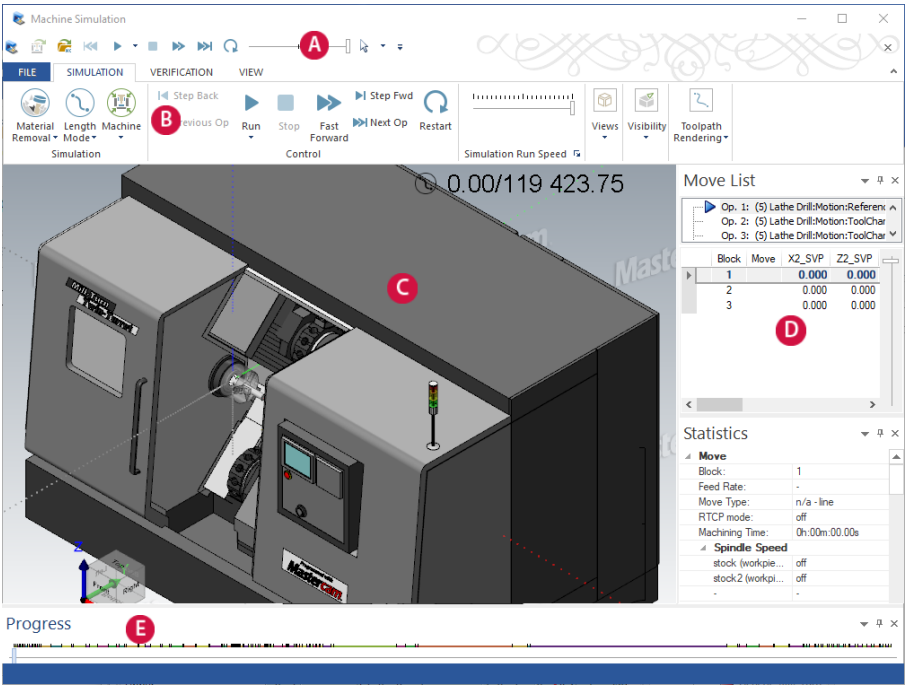
Lesson Goals

- Start machine simulation.
- Run a simulation.
- Manipulate a simulation.

Machine Simulator's Window

The Machine Simulation window has the main areas shown below. Refer to this illustration as you work through the procedures in this lesson.

- **A** Quick Access Toolbar
- **B** Ribbon
- **C** Graphics window
- **D** Data panes
- **E** Progress slider

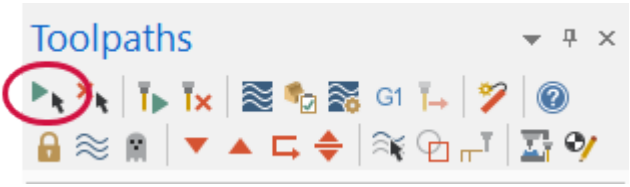


Exercise 1: Starting Machine Simulation

1. In Mastercam, open the file `MachineSimulation`, which is included with this tutorial. Then save the file as `MachineSimulation`–*XX*, where *XX* is your initials.

   Mastercam's window should look similar to the following picture.

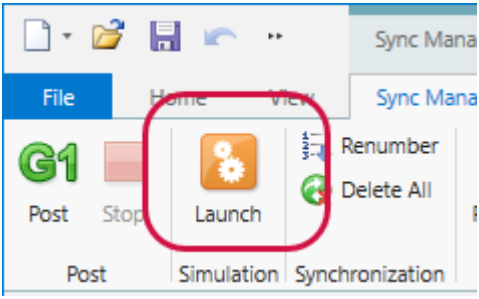
2. In Toolpaths Manager, click the **Select all operations** button.



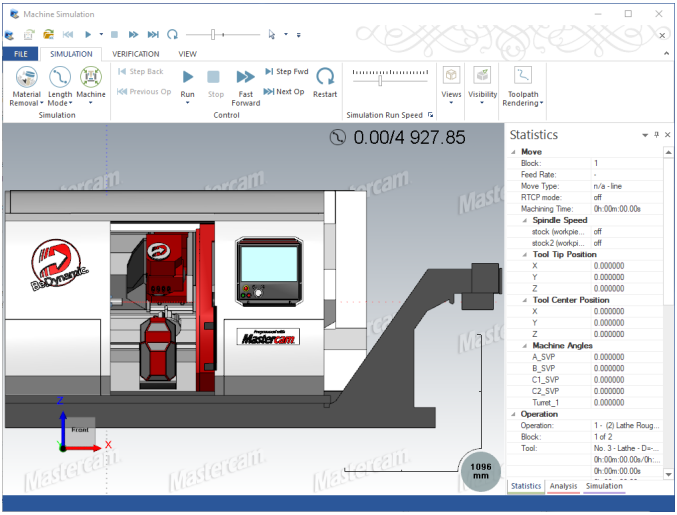
3. Click the **G1** button.

   Sync Manager opens with the part's data loaded. (Note that you may or may not see the **Machine Explorer** and **Properties** panes, depending on the state of the application when you last closed it.)

4. In Sync Manager, click the **Launch** button.



The Machine Simulation window appears, as shown in the picture below.



Note: If you do not see the tabbed panes on the right side of the window, you can display them by choosing **Docking Panes** options from the **View** tab. The following exercises require you to display the **Move List**.

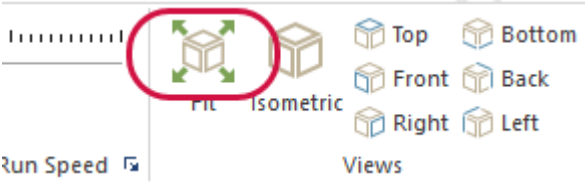
Exercise 2: Running a Simulation

1. Place your mouse over the graphics area, and move your mouse wheel.

   The view zooms in or out, depending on the direction you spin the wheel.

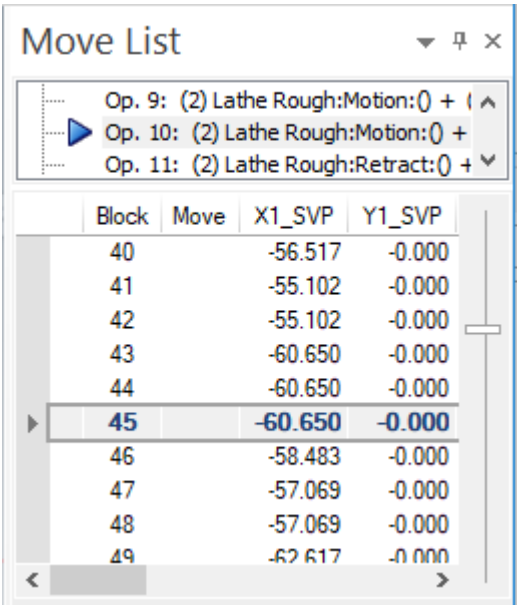
2. On the **Simulation** tab, in the **Views** group, click the **Fit** button.

   Machine Simulation zooms the graphics window to exactly display its contents.

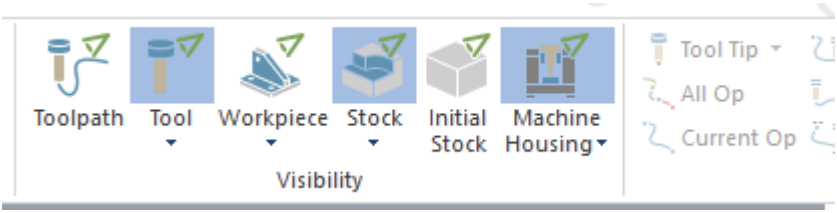


3. Click the **Run** button.

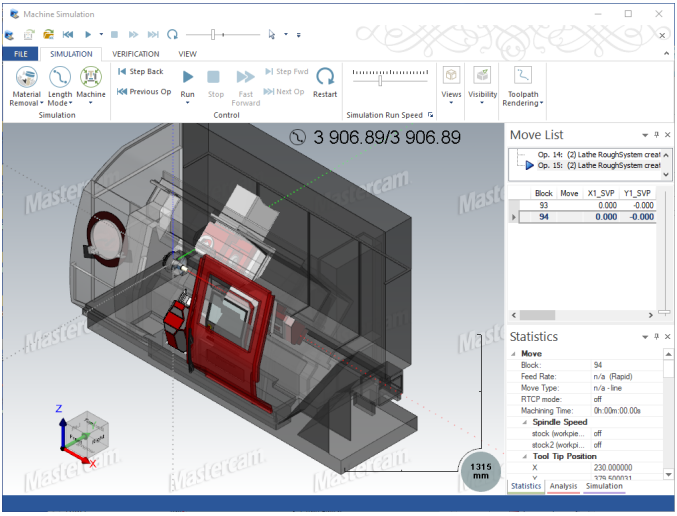
   Machine Simulation simulates the operations in your part. As it does, the **Move List** displays the running code, as shown below. (If you do not see the **Move List**, choose **View, Move List**.)



4. In the **Visibility** group, click the **Machine Housing** button.

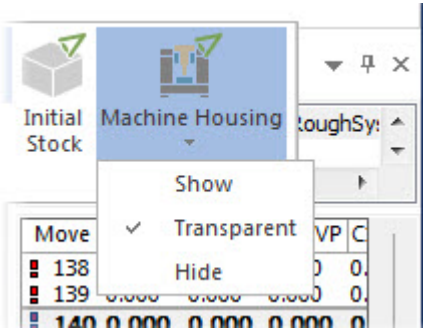


The machine housing becomes transparent, as shown in the following picture.

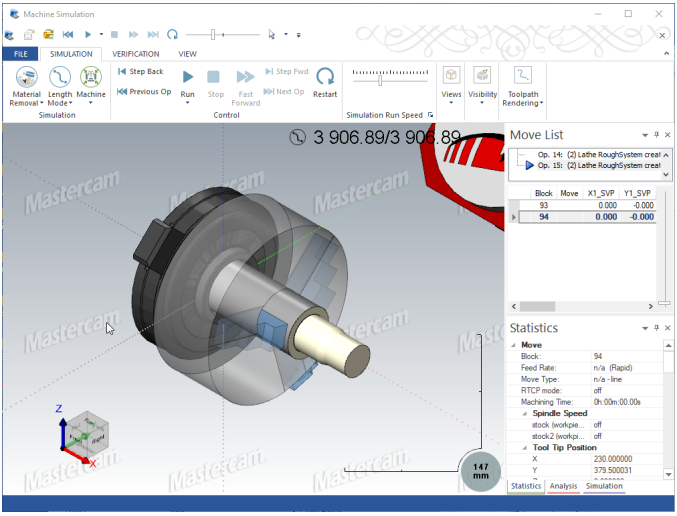


5. Click the **Machine Housing** button again.

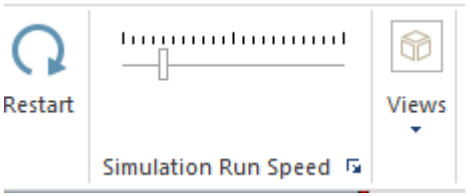
   Machine Simulation removes the machine housing from the display. You can choose a specific display state (Show, Transparent, or Hide) by clicking the small arrow below a component's button and selecting the state from the drop-down menu.



6. In the graphics window, place your mouse pointer over the left spindle, and use your mouse wheel to zoom in closer, as shown in the following picture.



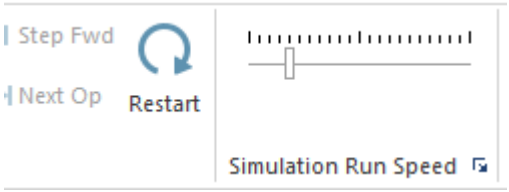
7. Drag the simulation speed slider to the left, approximately to the location shown.



Now when you next start the simulation, it will run slower than it did the first time.

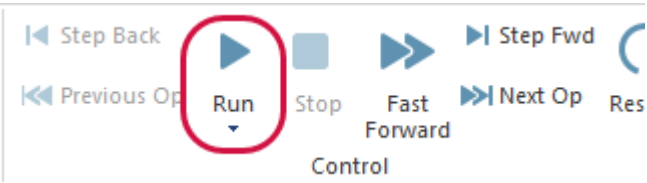
8. Click the **Restart** button.

   Machine Simulation resets the operations to the beginning.

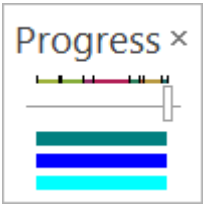


9. Click the **Run** button.

   Machine Simulation simulates the operations with your new settings.



10. Drag the **Progress** slider in the Progress window to move backward and forward through the simulation.



As you drag the slider, notice how the **Move List** updates to show the current operation and move. (If you do not see the slider, increase the size of the Progress area.)

11. Close the Machine Simulation window. You do not need to save the part.

12

Posting

After you have created your operations and verified them in Machine Simulation, you are ready to post the part. This process creates the program (G-code) that you send to your machine control to cut the part.

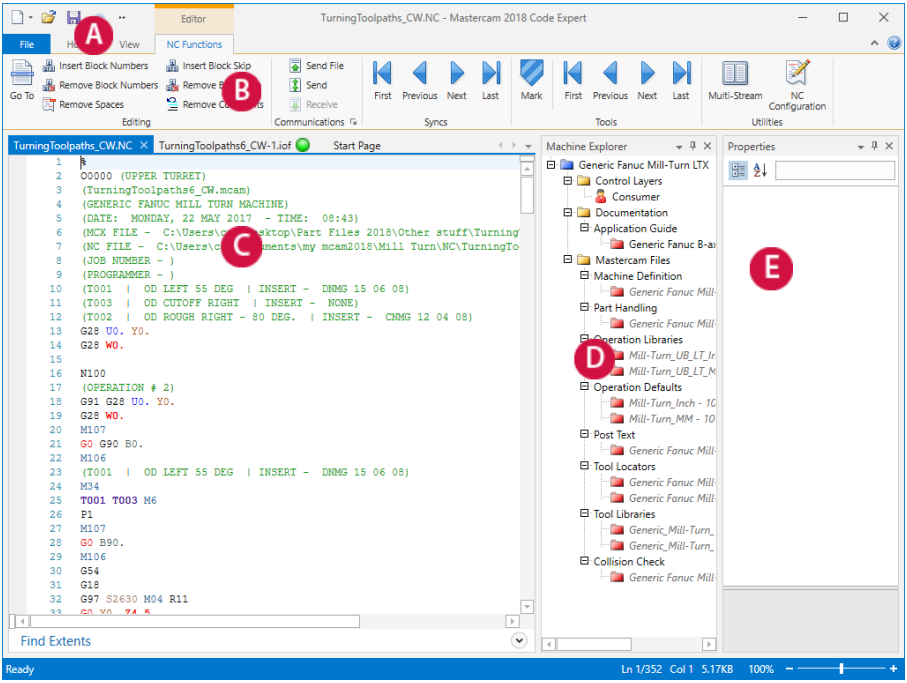
Lesson Goals

- Post a part.
- Use editing functions.
- Use syncs and tools functions.
- Use the right-click menu.
- Send NC code to a control.

The Editor's Window

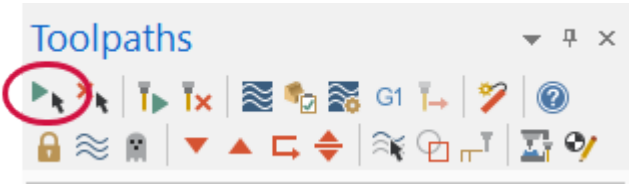
When you post a part, Code Expert opens the resulting G-code in its editor. The editor window has the main areas shown below. Refer to this illustration as you work through the procedures in this lesson.

- **A** Quick Access Toolbar
- **B** Ribbon
- **C** Code pane
- **D** Machine Explorer
- **E** Properties grid



Exercise 1: Posting a Part

1. Load the file `PostingExample` into Mastercam, and select all operations in **Toolpaths Manager**.

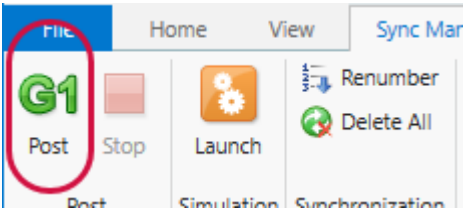


2. Click the **G1** button.

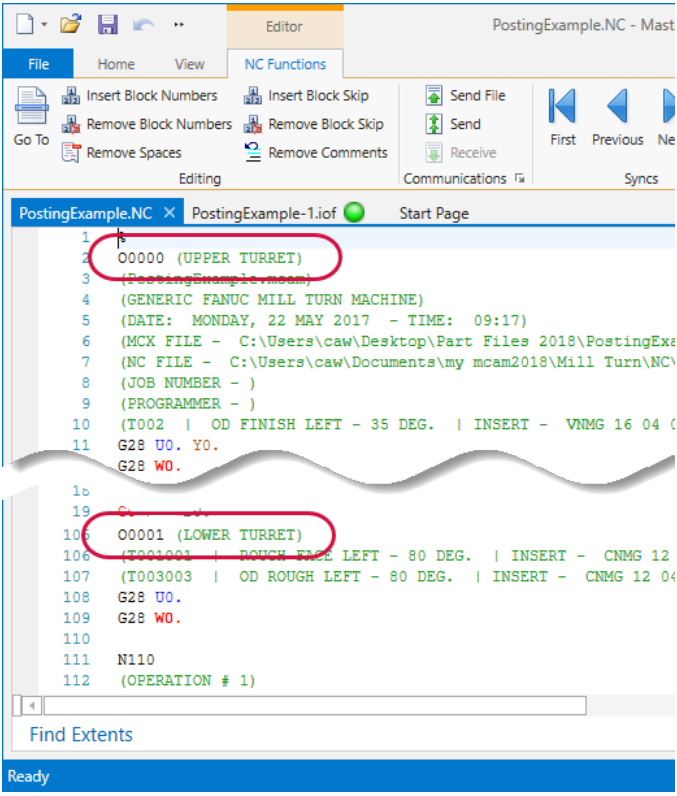
   Sync Manager opens and displays the part's data.

3. In Sync Manager, click the **Post** button.

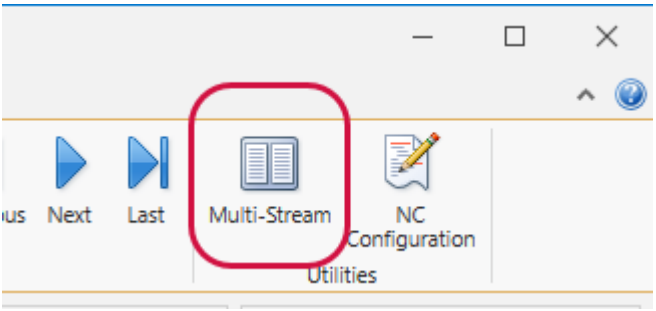
   Sync Manager processes the part, and opens the resulting G-code file into Code Expert's Editor.



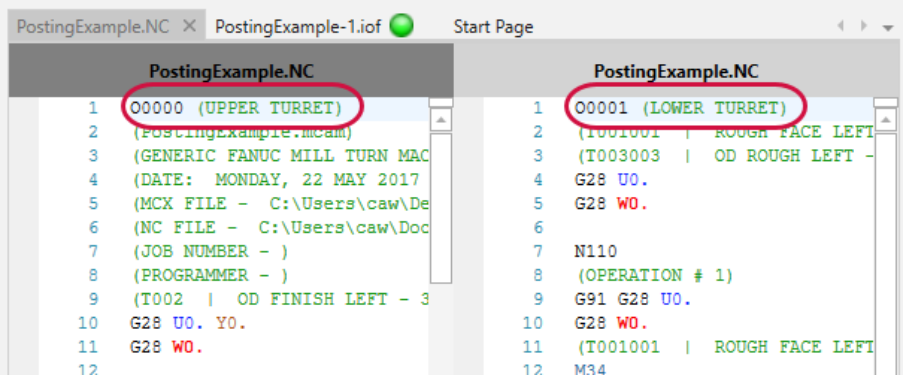
   Note that the code is merged into a single stream, with the code for both turrets shown in the same pane.



4. In the **Utilities** group, click **Multi-Stream**, and then click **OK** in the dialog box that displays.



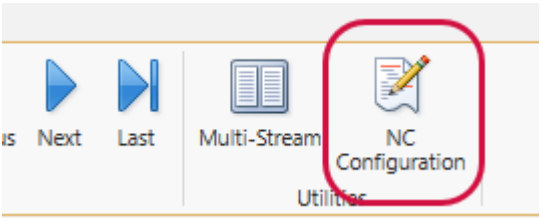
Code Expert divides the code into two streams, with the upper turret on the left and the lower turret on the right.



Exercise 2: Using the Editing Functions

1. Click **NC Configuration**.

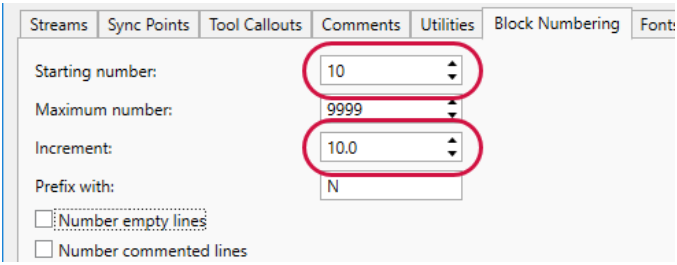
   The **NC Configuration** dialog box opens.



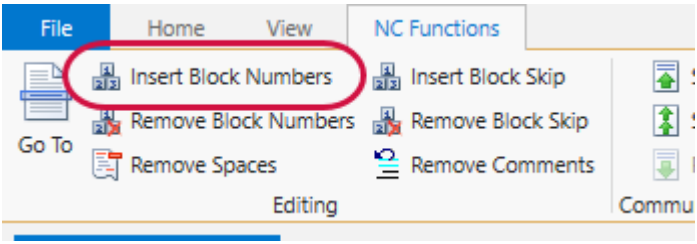
2. Click the **Block Numbering** tab.

   The **Block Numbering** page opens.

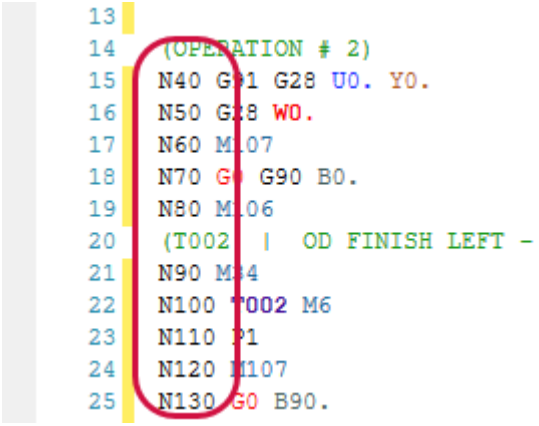
3. Change **Starting number** and **Increment** to **10**, and click **OK**.



4. In the Editing group, click **Insert Block Numbers**.

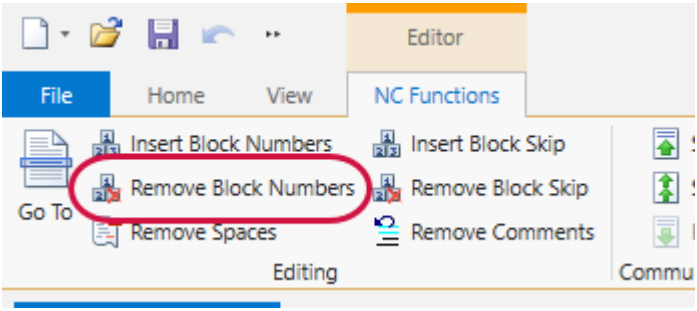


The Editor adds block numbers to the code (see the image below) based on the settings you specified in the Block Numbering page.



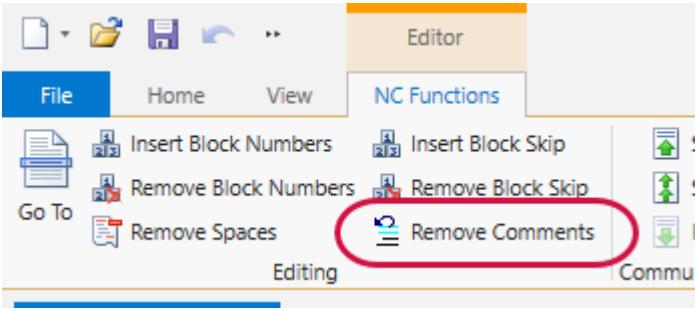
5. Click **Remove Block Numbers**.

The Editor removes the block numbers from the code.



6. Click **Remove Comments**.

   The Editor removes all comments from the active stream.



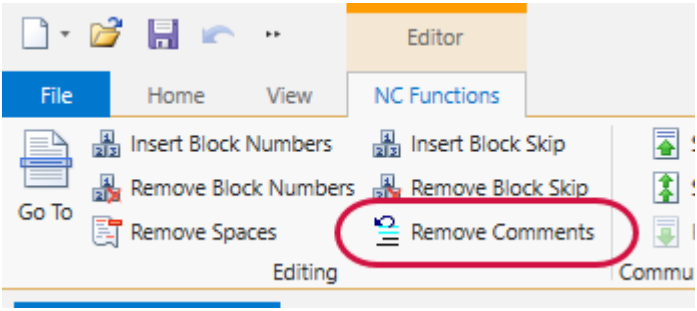
7. On your keyboard, press [**Ctrl+Z**].

   The editor undoes the **Remove Comments** command.

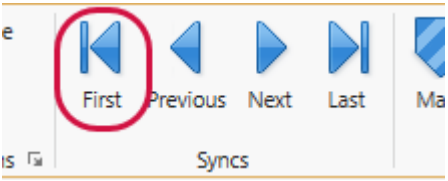
Exercise 3: Using the Syncs and Tools Functions

1. Click in the upper turret stream (the left-hand stream), and then click **Remove Comments**.

   Code Expert removes comments from the stream so that the following step in this tutorial does not find sync codes in non-code lines.



2. In the **Syncs** group, click **First**.

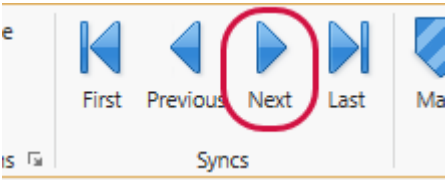


The editor jumps to the first synchronization code in the file.

```
16  N50 G28 W0.
17  N60 M107
18  N70 G0 G90 B0.
19  N80 M106
20
21  N90 M34
22  N100 T808 M6
23  N110 P1
24  N120 M107
25  N130 G0 B90.
26  N140 M106
27  N150 G54
```

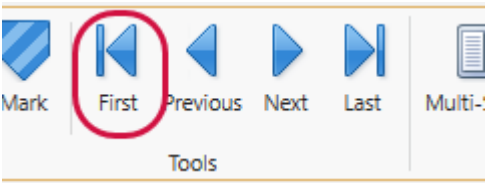
3. Click **Next** three times.

The editor shows the next synchronization code. (In this example, you click three times to skip past two occurrences of P*n* that are not actually sync codes.)



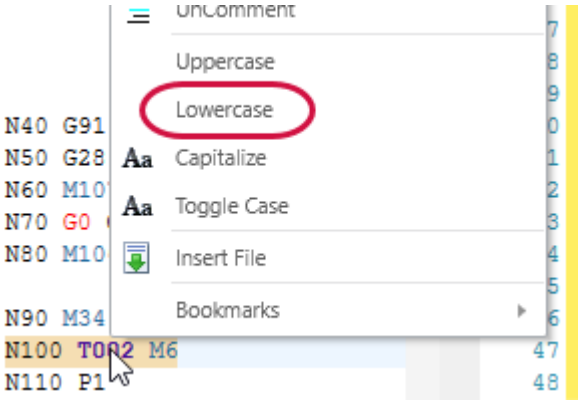
4. In the **Tools** group, click **First**.

   The editor jumps to the first tool change in the stream.



Exercise 4: Using the Right-Click Menu

1. Highlight a line of code, right-click the line, and choose **Lowercase** from the pop-up menu.

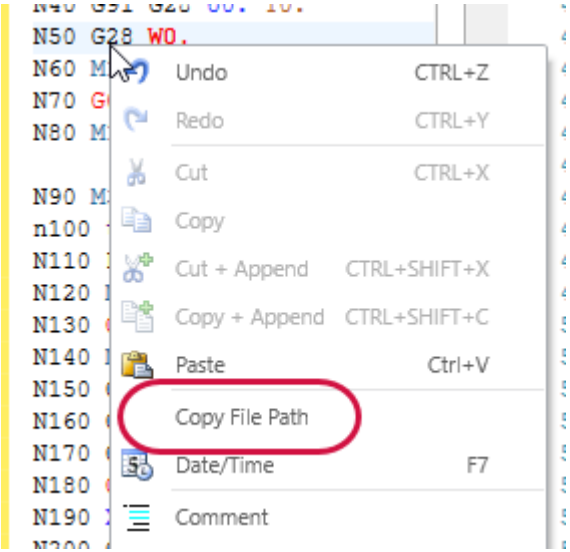


   The editor changes the selected text to lowercase, as shown below.

```
21  N90 M34
22  n100 t002 m6
23  N110 P1
```

2. Right-click anywhere in the code, and select **Copy File Path**.

   The editor places the active file's full path into the Clipboard.



3. Place the text cursor where you want to insert the file's path, and press [**Ctrl+V**] to paste the file path into your code.

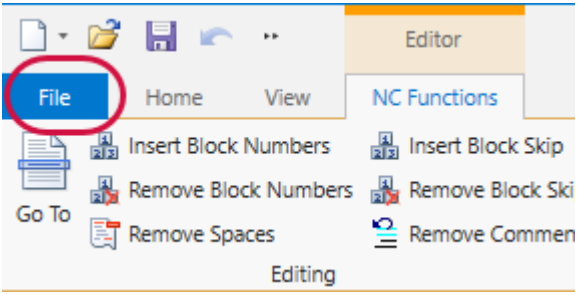
Exercise 5: Sending the NC Code to Your Control

1. Click the **PostingExample-1.iof** tab, and then click the **Post** button to post the part. (If you are asked to save changes, click **No**.)

   Sync Manager regenerates the NC file.

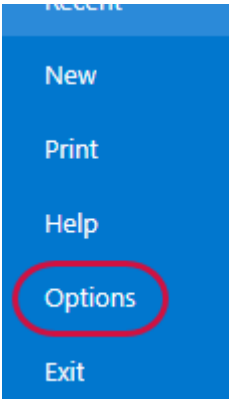
2. Click the **File** tab.

   The file commands display.



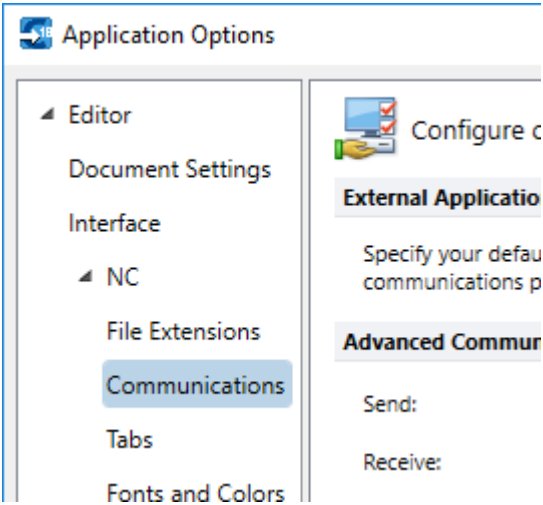
3. Click **Options**.

   The **Application Options** dialog box appears.

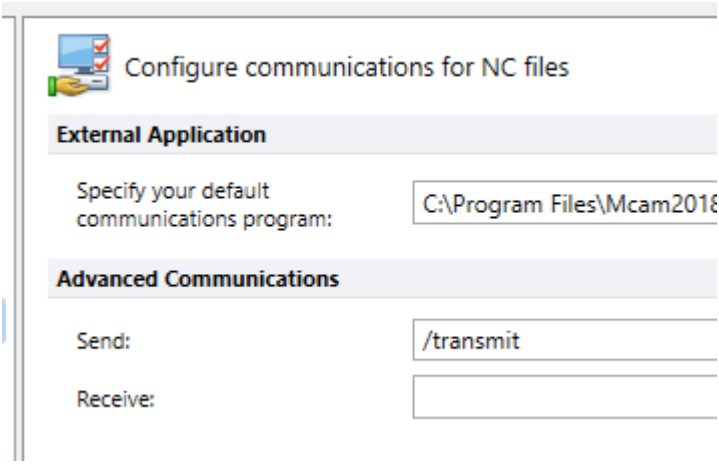


4. Open the option group **Editor, NC, Communications**.

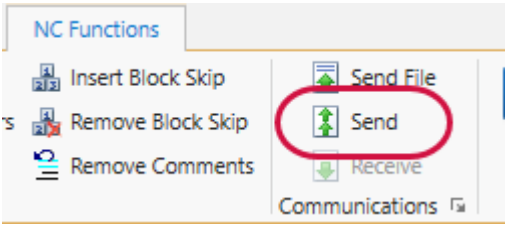
   The communications options appear in the dialog box's right pane.



5. Set the communications options appropriately for your machine control. (If you are unsure, leave them set to the defaults.)



6. Click **OK** to close the **Application Options** dialog box.
7. On the **NC Functions** tab, click the **Send** (not **Send File**) button.



Code Expert sends the G-code file to your machine control.

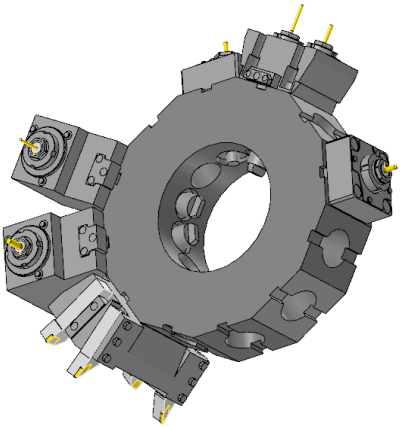
Note: For more information on your machine's settings and capabilities with Mill-Turn, please refer to the application guide that comes packaged with each machine.

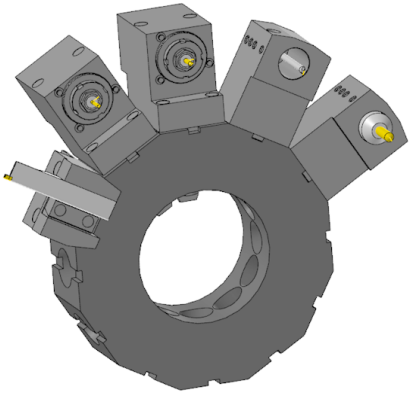
8. Close all windows.

13

Loading Multi-Station Tool Locators

Multi-station tool locators (see images below) offer extensive flexibility for your machine setup. The examples in this appendix provide an overview of loading tool locators onto a machine's turrets, as well as loading tools into the locators. The complete sample program demonstrates many tool locator and tool combinations.





Exercise 1: Loading the Part and Processing the Toolpaths

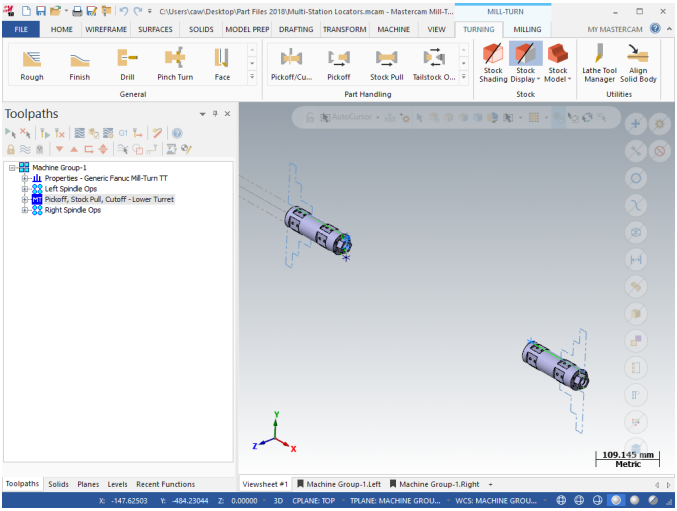
1. Copy the machine file `Generic Fanuc Mill-Turn TT.machine` into your `C:\Users\Public\Documents\shared`

`Mcam2017\Mill Turn\machines` folder.

You use this twin turret machine for the exercises that follow.

2. Open the file `Multi-Station Locators`.

   The file is included with this tutorial and contains many operations that require tools be loaded into turrets with multi-station tool locators. After loading the file, Mastercam should look like the following image:

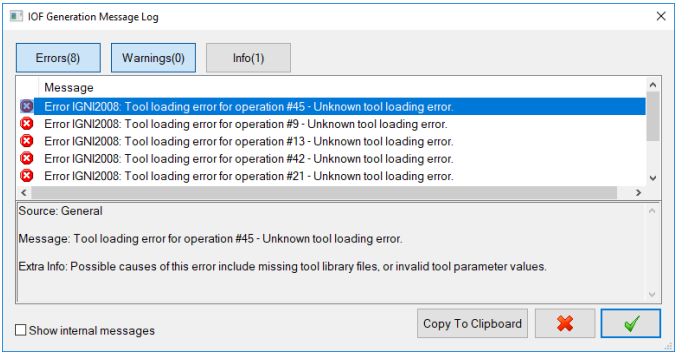


3. In Toolpaths Manager, click **Machine Group-1** to select all of the operations.

   If you want to see the operations, click the plus signs in the list to expand each group.

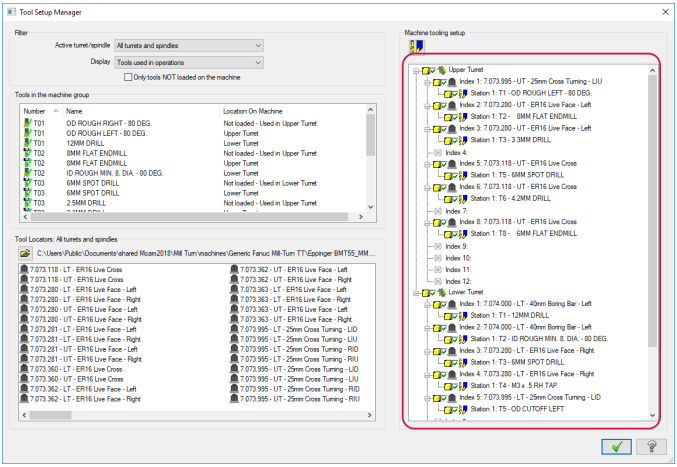
4. Click the **Post selected operations** button.

   Mastercam attempts to process the operations, but generates tool-loading errors. These errors must be addressed before you can continue.



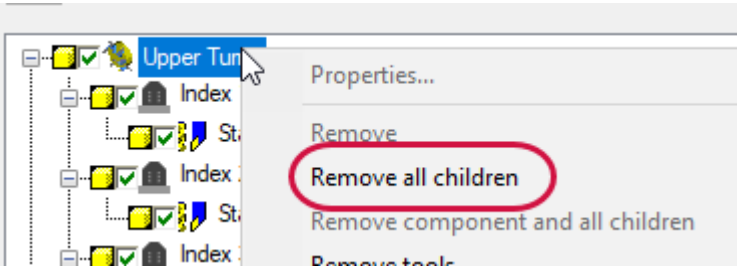
5. Click the **Cancel** button to dismiss the IOF Generation Message Log dialog box.

   The Tool Setup Manager appears, showing the tools that Mastercam loaded into the upper and lower turrets.



6. Right-click the upper turret in the **Machine tooling setup** list, and choose **Remove all children** from the menu that appears.

   Mastercam removes all tool locators and tools from the upper turret.



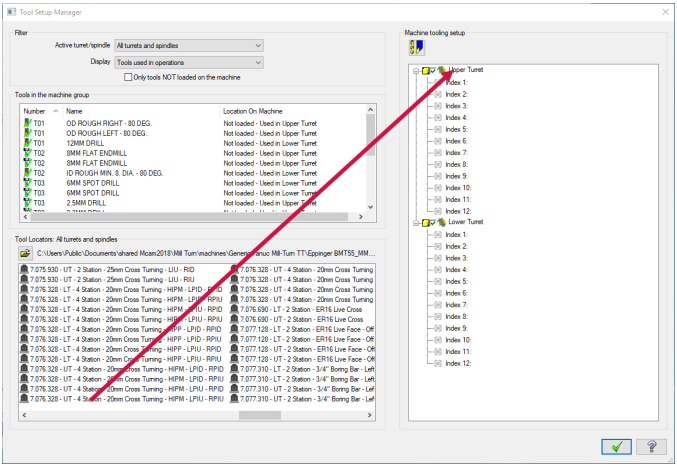
7. Right-click the lower turret in the **Machine tooling setup** list, and choose **Remove all children** from the menu that appears.

   Mastercam removes all tool locators and tools from the lower turret. Both turrets are now empty.

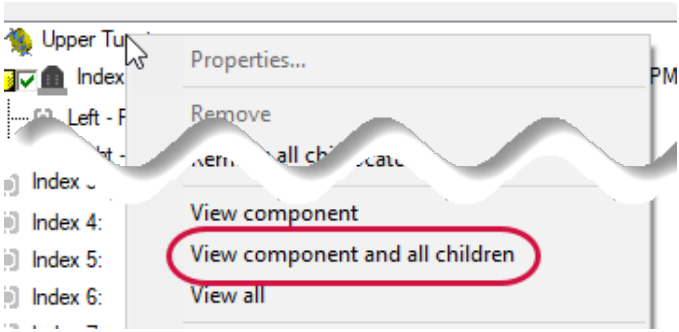
Exercise 2: Loading a Tool Locator and Tool

1. From the Tool Locators pane, drag the **7.076.328 - UT - 4 Station - 20mm Cross Turning- HIPM - LPIU - RPIU** tool locator to Index 1 on the upper turret.

   Mastercam adds the tool locator to the turret.

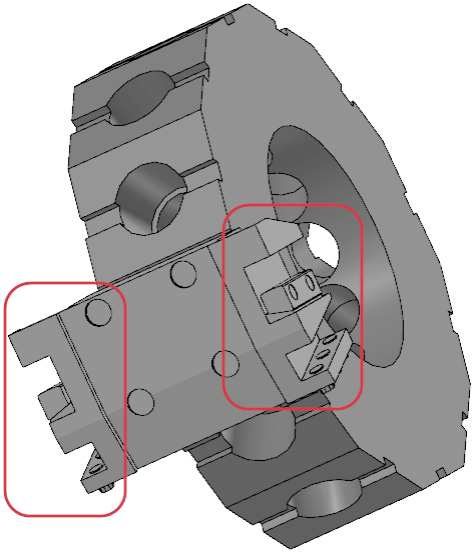


2. In the **Machine tooling setup** list, right-click the upper turret, and select **View component and all children**.



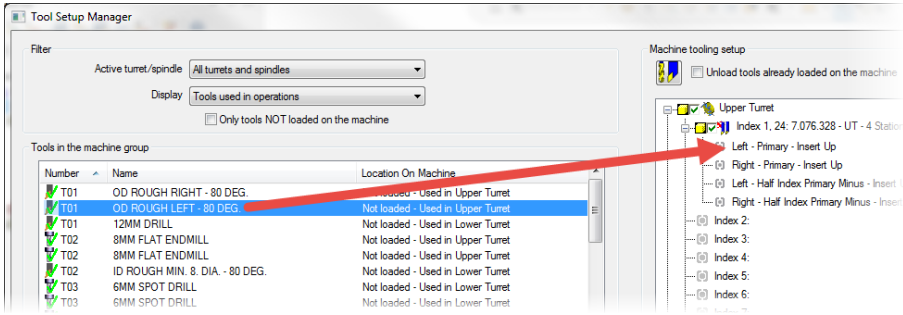
Mastercam displays the components in the graphics window.

3. Rotate the components to the view shown below. Note the four locations for tools. You load those tools in the next steps.



4. Press [**Enter**] to return to the Tool Setup Manager.

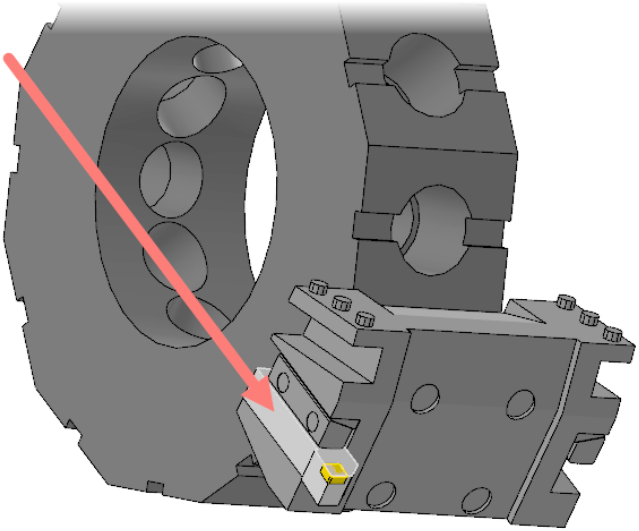
5. From the **Tools in the machine group** list, drag the tool **T01 - OD ROUGH LEFT - 80 DEG** to the location **Left - Primary - Insert Up**.



6. Right-click the upper turret, and choose **View component and all children**, as you did in Step 2.

   Mastercam displays the components in the graphics window.

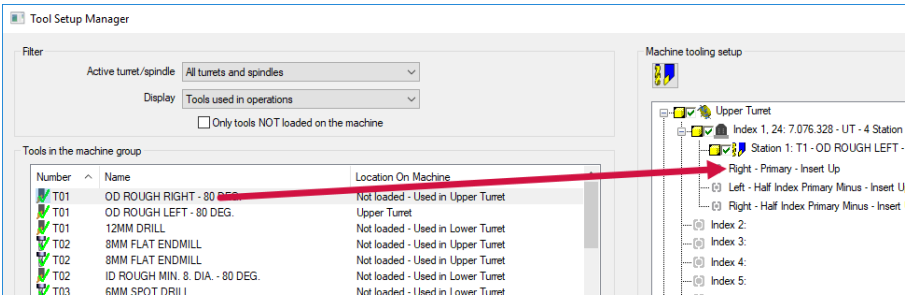
7. Rotate the components so that you can see the tool loaded into the tool locator.



8. Press [**Enter**] to return to the Tool Setup Manager.

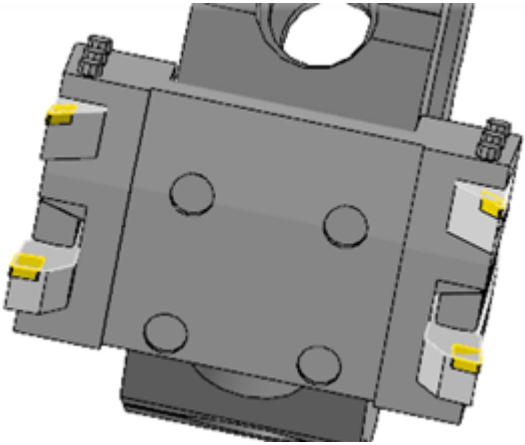
Exercise 3: Loading the Remaining Tools in the Tool Locator

1. From the **Tools in the machine group** list, drag the tool **T01 - OD ROUGH RIGHT - 80 DEG** to the location **Right - Primary - Insert Up**.



2. Drag the tool **T24 - OD FINISH LEFT - 35 DEG** to the location **Left - Half Index Primary Minus - Insert Up**.
3. Drag the tool **T24 - OD FINISH RIGHT - 35 DEG** to the location **Right - Half Index Primary Minus - Insert Up**.

   The tool locator is now fully loaded.
4. Right-click the upper turret, and choose **View component and all children**.
5. Rotate the components so that you can see the four tools loaded into the tool locator.



6. Press [**Enter**] to return to the Tool Setup Manager.

Exercise 4: Loading the Second Tool Locator and Live Mill Tools

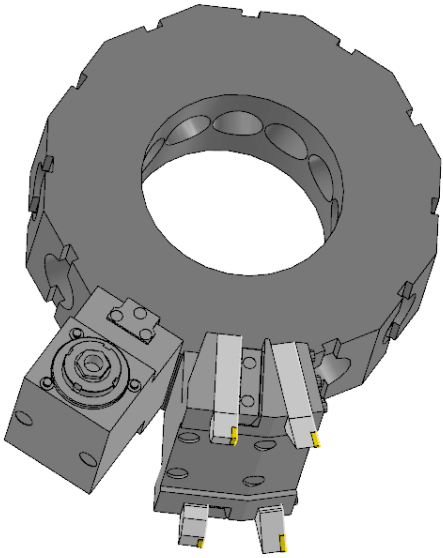
1. From the Tool Locators pane, drag the **7.077.128 - UT - 2 Station - ER16 Live Face - Offset Left** tool locator to Index 2 on the upper turret.

   Mastercam adds the tool locator to the turret.

2. In the **Machine tooling setup** list, right-click the upper turret, and select **View component and all children**.

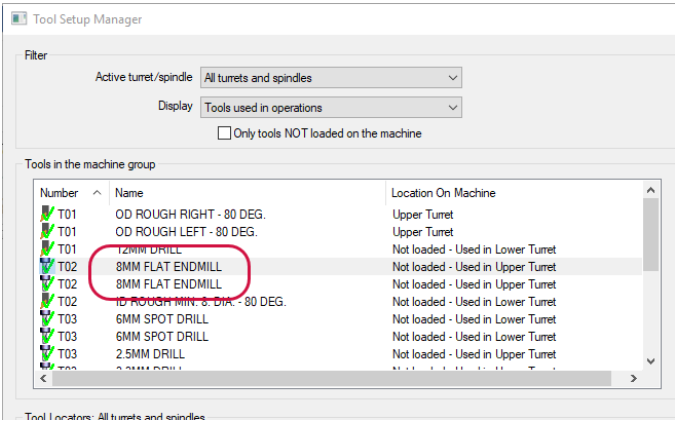
   Mastercam displays the components in the graphics window.

3. Rotate the components to the view shown below. Note the new tool locator for live tooling.



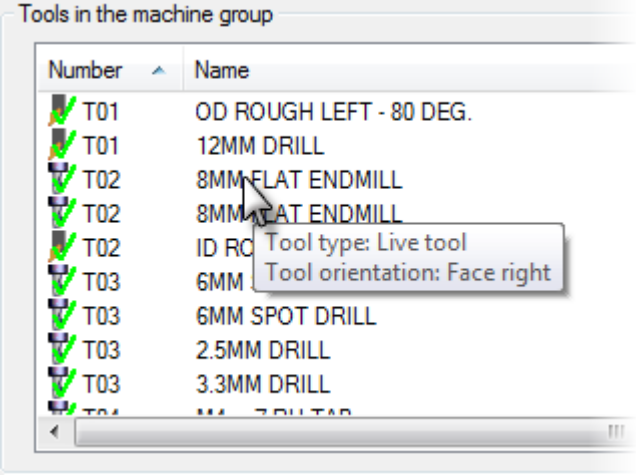
4. Press [**Enter**] to return to the Tool Setup Manager.
5. In the **Tools in the machine group** list, locate the 8MM FLAT ENDMILL tools.

   In the list, these tools' descriptions look identical.



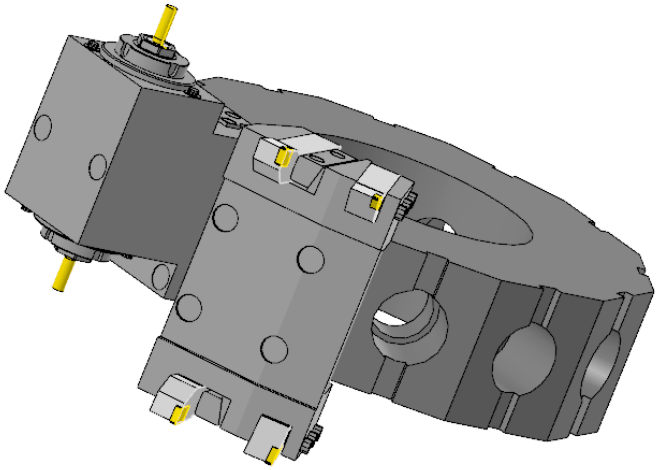
6. Hover your mouse pointer over the first 8MM FLAT ENDMILL.

   A tooltip describes this tool's orientation as **Face right**.



7. Drag the right-facing 8MM FLAT ENDMILL to the location **Right - Live Tool**.
8. Drag the other 8MM FLAT ENDMILL (left facing) to the location **Left - Live Tool**.
9. In the **Machine tooling setup** list, right-click the upper turret, and select **View component and all children**.

   Mastercam displays the components in the graphics window.
10. Rotate the components to the view shown below, and note the two live tools for milling.



11. Press [**Enter**] to return to the Tool Setup Manager.

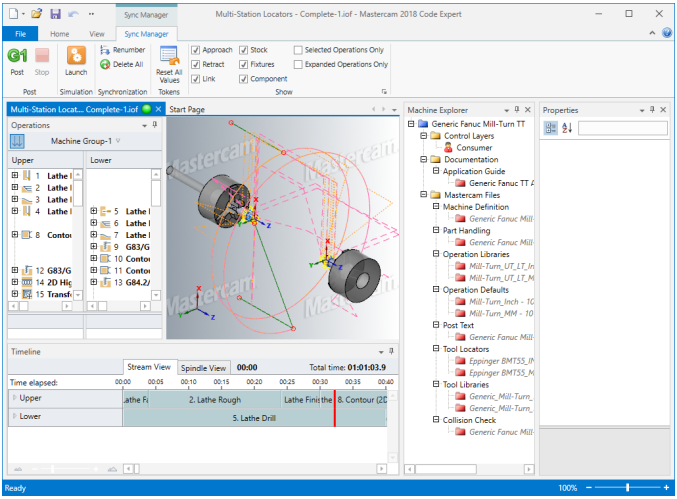
This tutorial includes the completed file, `Multi-Station Locators - Complete`. You can load this file if you wish to see complete machine tooling setup.

Exercise 5: Simulating the Operations

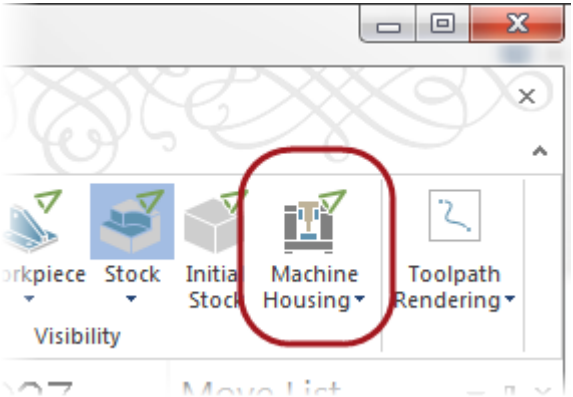
Before continuing with this part of the tutorial, load the complete file `Multi-Station Locators - Complete`. Do not save changes to the previous file.

1. In Toolpaths Manager, click **Machine Group-1** to select all of the operations.
2. Click the **Post selected operations** button.

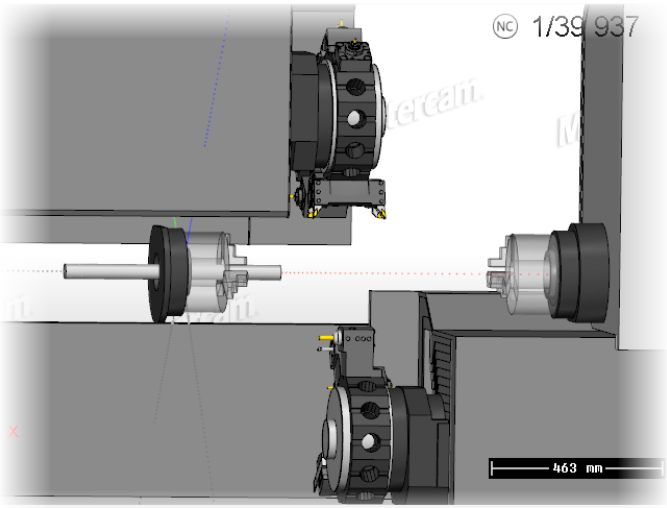
   Mastercam processes the operations and opens the part in Sync Manager, as shown here:



3. Click the **Launch** button to start Machine Simulation.
4. If necessary, click **Machine Housing** twice to hide the machine's housing.

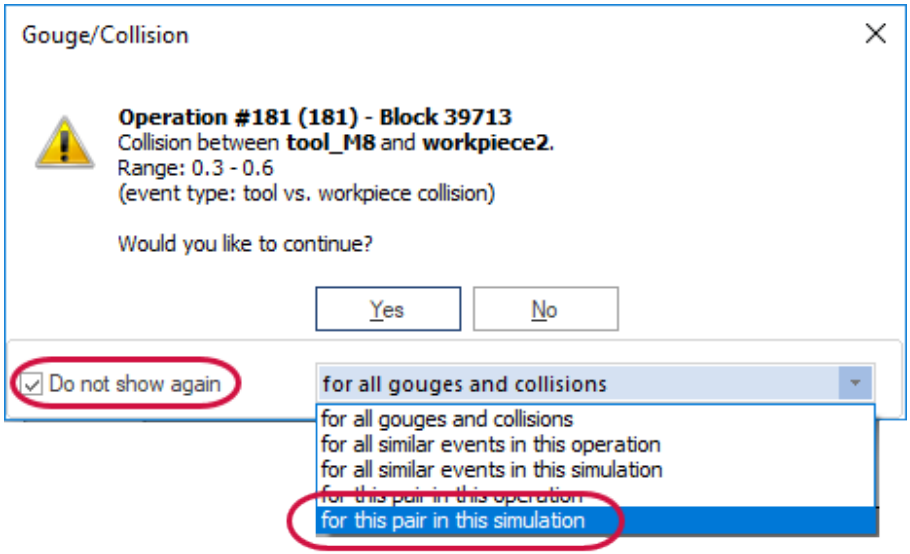


5. Rotate the view so that you can easily see the part stock and the turrets.



6. Click **Run** to watch the machine cut the part, paying special attention to the turrets and their tool locators. As the simulation runs, adjust the view by panning, rotating, and zooming so that you can see each operation clearly.

Note: When the machining gets to operation 54, the simulation will crash, displaying the dialog box shown below. This crash is because the tap tool is larger than the hole, as it should be. To prevent the simulation from constantly stopping, click **Do not show again** in the dialog box, and select **for this pair in this simulation** from the drop-down. Now the simulation will ignore further crashes between this tool and the workpiece.



Conclusion

Congratulations! You have completed the *Mastercam Mill-Turn Tutorial* tutorial! Now that you have mastered the skills in this tutorial, explore Mastercam's other features and functions.

You may be interested in other tutorials that we offer. Mastercam tutorials are being constantly developed, and we will add more as we complete them. Visit our website, or select **Help**, **Tutorials** from the **File** tab.

Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Documentation*—Mastercam installs a number of helpful documents for your version of software in the Documentation folder of your Mastercam 2018 installation.
- *Mastercam Help*—Access Mastercam Help by selecting **Help**, **Contents** from Mastercam's File tab or by pressing [**Alt+H**] on your keyboard.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—Our Technical Support department (860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Mastercam Tutorials*—We offer a series of tutorials to help registered users become familiar with basic Mastercam features and functions. Visit our website, or select **Help**, **Tutorials** from Mastercam's File tab to see the latest publications.
- *Mastercam University*—Mastercam University, an affordable online learning platform, gives you 24/7 access to Mastercam training materials. Take

advantage of more than 180 videos to master skills at your own pace and help prepare for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit www.mastercamu.com, or email training@mastercam.com.

- *Online Communities*—You can find a wealth of information at www.-mastercam.com. For tech tips and the latest Mastercam news, follow us on Facebook (www.facebook.com/mastercam), Twitter (www.twitter.com/mastercam), or Google+ (plus.google.com/+mastercam). Visit our YouTube channel to see Mastercam in action (www.youtube.com/user/MastercamCadCam)! Registered users can search for information or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the knowledgebase at kb.mastercam.com.

Contact Us

For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at techdocs@mastercam.com.

ATTENTION! UPDATES MAY BE AVAILABLE.
PLEASE REFER TO MASTERCAM.COM/SUPPORT
FOR THE LATEST DOWNLOADS.

CNC Software, Inc.
671 Old Post Road
Tolland, CT 06084 USA
www.mastercam.com

Mastercam
Be Dynamic.

Mastercam® is a registered trademark of CNC Software, Inc. © 1983-2017. All rights reserved.
All other trademarks are property of their respective owners.