



MASTERCAM TOOL MANAGER TUTORIAL

June 2017

POWERED BY MASTERCAM'S
DYNAMIC MOTION™ TECHNOLOGY



Mastercam®
Be Dynamic.

MASTERCAM TOOL MANAGER 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	5
General Tutorial Requirements	5
Mastercam's Tool Manager	7
Tool Manager Interface	7
Stand-Alone vs. Classic Tool Managers	9
Stand-Alone Tool Manager	9
Classic Mill/Router Tool Manager	10
Classic Lathe Tool Manager	11
Working with the Stand-Alone Tool Manager	12
Recommended Workflow	13
1 Working with Tool Library Files (.TOOLDB)	15
Exercise 1: Creating Tool Libraries	16
Creating a Tool Library from an Existing Library	16
Creating a New Tool Library from Scratch	19
Exercise 2: Opening and Editing a Tool Library	19
Exercise 3: Searching for a Library Component	21
2 Working with Components in a Tool Library	25
Exercise 1: Creating a New Tool	26
Exercise 2: Creating a New Holder	28

Exercise 3: Editing Existing Tools and Holders	30
Exercise 4: Creating and Editing Assemblies	33
Editing Assemblies in the Graphics Window	36
Exercise 5: Adding or Editing Materials	37
Exercise 6: Adding and Editing Cut Parameters	39
Exercise 7: Accessing Cut Parameters from Mastercam	41
Exercise 8: Copying Components Between Tool Libraries	44
3 Using Tools and Holders in Toolpaths	47
Exercise 1: Creating a New Tool	48
Exercise 2: Creating a New Holder	51
Exercise 3: Editing Tools and Holders	54
4 Rendering Mastercam Tools	57
How Mastercam Renders Tool Profiles	58
Exercise 1: Creating New Tools from Profiles	60
Conclusion	67
Mastercam Resources	67
Contact Us	68

Introduction

Welcome to *Mastercam Tool Manager Tutorial*. This guide provides information about Mastercam's stand-alone Tool Manager. Some of the information is in a step-by-step format, and some of the information is in a paragraph format. Both formats walk you through how and when to use the stand-alone Tool Manager.

Tutorial Goals

- Become familiar with Mastercam's stand-alone Tool Manager
- Work with tool libraries (.TOOLDB) and their components
- Create and edit tools and holders in Mastercam's classic Tool Manager
- Properly render imported tools

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: 3 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.

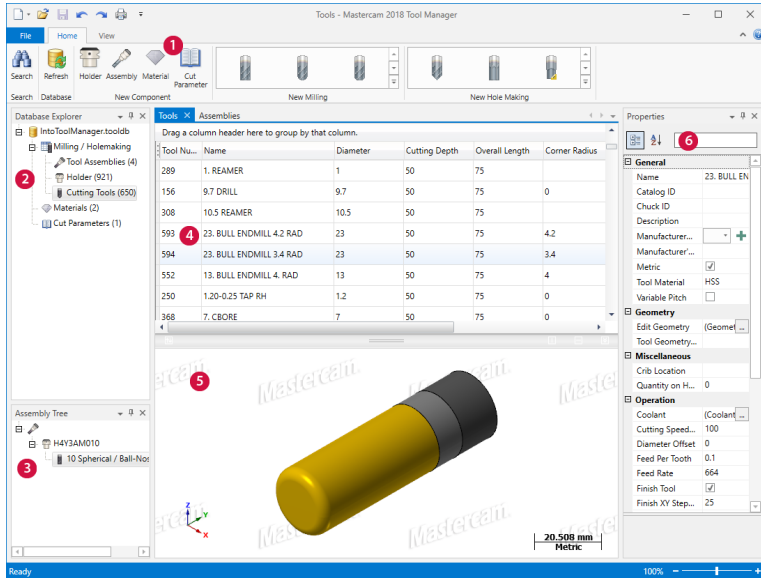
Mastercam's Tool Manager

Mastercam's stand-alone Tool Manager provides an efficient and comprehensive way to manage tools and tool holding components, and to create tool assemblies to use in Mastercam. Besides the tooling component support, Tool Manager also integrates work material and cut parameter data so that you can accommodate a manufacturer's cutting recommendations or create your own cut parameter data and store it in the database.

Launch the stand-alone Tool Manager separately from Mastercam by choosing **Tool Manager** from your Start menu's Mastercam folder. Certain Tool Manager functions are integrated into the Mastercam interface and are launched from within Mastercam when you create new tools, holders, and assemblies. While you are in Tool Manager, your data is automatically saved as you work.

Tool Manager Interface

This section contains brief descriptions of the stand-alone Tool Manager interface.



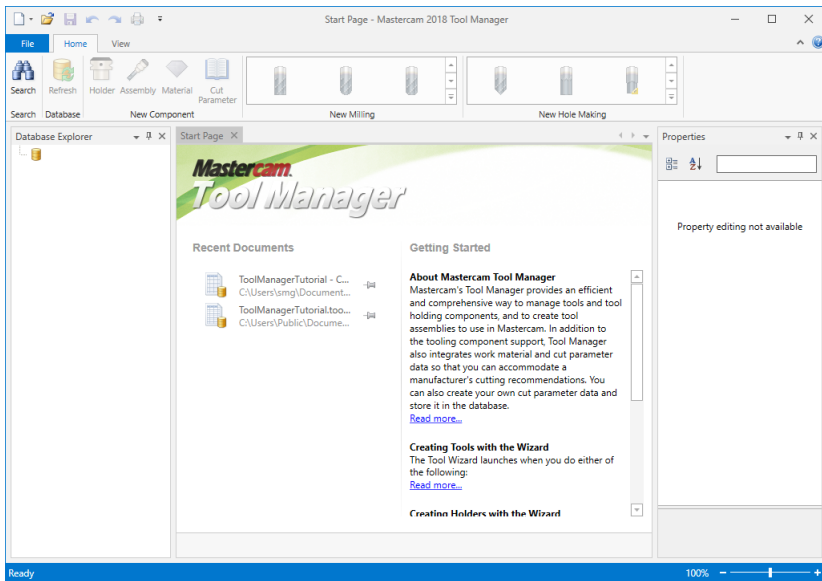
1. **Tabs:** Tabs function somewhat like dialog boxes but look similar to toolbars. You cannot undock or dock a tab, nor are they customizable.
2. **Database Explorer:** Use the Tool Manager Database Explorer to access content in tool database files (TOOLDB). Right-click an item to expand or collapse it. To work with items in the database tree view (i.e., Cutting Tools, Tool Assemblies, Holders, Materials, and Cut Parameters), double-click the item.
3. **Assembly Tree:** Use the Assembly Tree to open, create, and edit assemblies in the open TOOLDB database. Tool Assemblies may contain one tool and one or more holder components. Use the Assembly Tree to navigate the individual assembly items. If you double-click on **Tool Assemblies** in the Database Explorer or choose **Assembly** on the **Home** tab, the Assembly Tree appears below the Database Explorer and the Assemblies document appears in the documents area.
4. **Document Datagrids:** Tool Manager uses "documents" to organize data such as tools, holders, and assemblies. The main workspace uses a tab-style interface to present these documents.

5. **Graphics Window:** The graphics window is part of a document, and displays what you have selected in the document datagrid. For example, if you are working in the Assembly document, the graphics window displays the assembly you have selected in the datagrid.
6. **Properties Grid:** Tool Manager Properties Grid displays properties for cutting tools, assemblies, holders, extensions, materials, and cutting parameters. The properties displayed in the Properties Grid depend on which document type you have open and which item you have selected in the document.

Stand-Alone vs. Classic Tool Managers

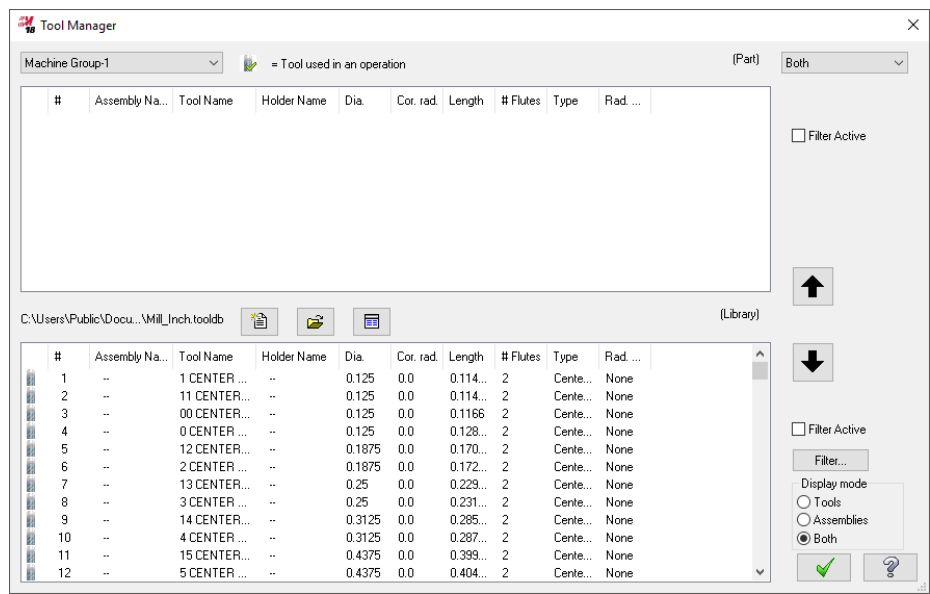
Although this document focuses on the stand-alone Tool Manager, Mastercam offers more than one Tool Manager to create, edit, and manage your tools, tooling components, and tool libraries. Each Tool Manager is accessed with different workflows and circumstances.

Stand-Alone Tool Manager



The stand-alone Tool Manager provides an interface to manage Mill and Router tools and tool holding components, and to create tool assemblies to use in Mastercam. In addition to the tooling component support, Tool Manager also integrates work material and cut parameter data so that you can accommodate a manufacturer's cutting recommendations. You can also create your own cut parameter data and store it in the database.

Classic Mill/Router Tool Manager

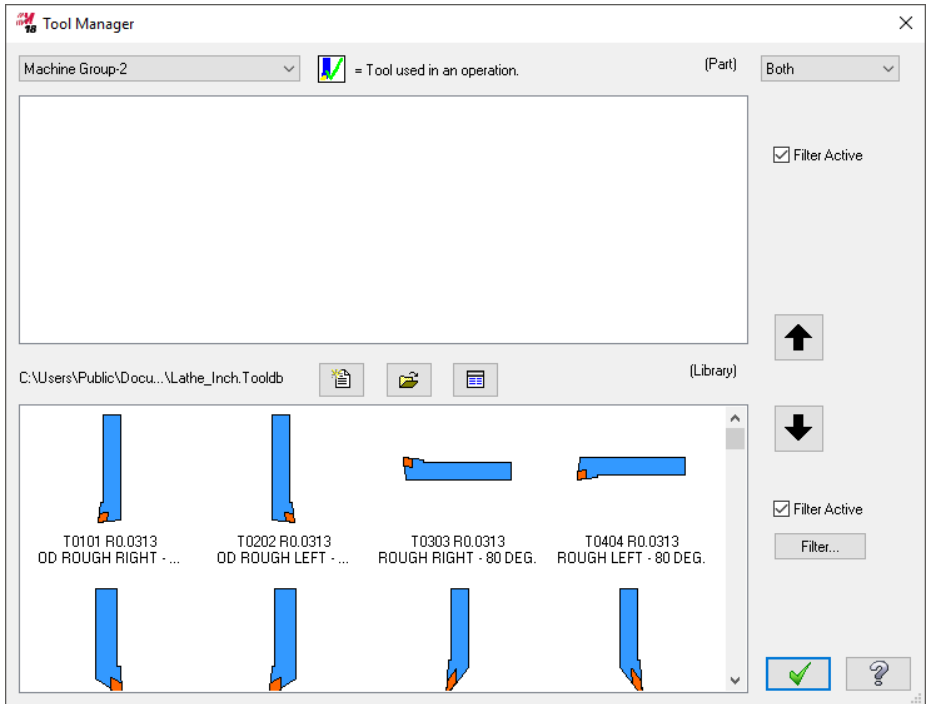


The classic Mill/Router Tool Manager allows you to view and manage tool libraries, the tools in your part file, and tool definitions. Use it to:

- View tools stored in a tool library, machine group, or both.
- Create new tools and store them in a tool library or machine group.
- Copy tools to/from a library to a machine group, or between different libraries or machine groups.

The Mill Tool Manager is accessed by selecting **Tool Manager** on the **Mill** or **Router** contextual tab.

Classic Lathe Tool Manager



The classic Lathe Tool Manager has the same features as the classic Mill/Router Tool Manager.

The Lathe Tool Manager is accessed by selecting **Tool Manager** on the **Lathe Turning** or **Milling** contextual tab.

Working with the Stand-Alone Tool Manager

By using the stand-alone Tool Manager instead of the Classic Mill or Lathe Tool Manager, you can do the following:

- **Work without Mastercam open:** Although you need to have Mastercam installed to run the stand-alone Tool Manager, you do not have to have Mastercam open. You don't even need to have a SIM plugged in. When you create, edit, and organize your tool libraries without running Mastercam, you increase your computer's performance while reserving that licensed Mastercam seat for someone else.
- **Search:** You can search for tools, assemblies, and other components that match your search criteria. You can save searches for future use as **Favorites** and you can view recent searches.
- **Copy components between libraries:** You can copy tools, holders, and assemblies from one TOOLDB to another, open the different tool libraries in separate instances of Tool Manager and drag and drop the tooling component from one Tool Manager to the other.
- **Create and edit assemblies:** You can create and edit tooling assemblies. You can drag and drop assembly components into the Assembly Tree, you can work directly in the Tool Manager's graphics window to edit assemblies, and you can change the parameters for an assembly or for each of the assembly components in the Properties Grid.
- **Customizable workspace** - The workspace of the stand-alone Tool Manager allows you to undock the major interface objects. This is useful if you work with multiple monitors. It is also useful if you just want to reposition or hide the interface windows to suit your preferences.
- **Create and edit materials and cut parameters:** Tool Manager allows you to create and edit materials and cut parameters. These settings are stored in the tool library and are easily accessed through Mastercam when you create or edit a tool with the Tool Wizard.

- **Edit multiple components:** You to select multiple document items, such as multiple tools, and edit the properties for all selected items with the Properties Grid.

In this tutorial, you will go through some of these concepts in the next few chapters.

Recommended Workflow

The stand-alone Tool Manager is designed to create tool libraries (TOOLDB) that can be specific to a job and/or machine. These tool libraries can be used as part of a job “kit” that reside on one machine, or travel with the part.

The recommended workflow is to set up your tool libraries using the stand-alone Tool Manager, and then use those libraries in Mastercam to apply to specific jobs, parts, and machines.

After installing Mastercam, you *must* open Mastercam before opening the stand-alone Tool Manager for the first time, so that it can build support files. After opening Mastercam, you can close it and successfully open the stand-alone Tool Manager. If you are working from a new Mastercam install and try to open the stand-alone Tool Manager without at least opening and closing Mastercam, you will receive an error message prompting you to open Mastercam.

Working with Tool Library Files (.TOOLDB)

In this lesson, you learn how to work with tool libraries.

Lesson Goals

- Switch to metric database output
- Create and edit tool libraries
- Use the stand-alone Tool Manager's Search feature

Exercise 1: Creating Tool Libraries

In this exercise, you learn to create tool libraries by either copying an existing library or creating a TOOLDB from scratch. Because you create metric tools and holders later in this guide, you set the default units to metric in this exercise.

The default unit setting is read from Mastercam's System Configuration.

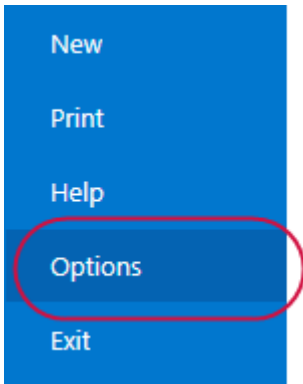
Tool Manager remains in the metric state until you manually change it back to inch, even if you use multiple sessions of Tool Manager or turn off your computer.

All new tools and holders created in the stand-alone Tool Manager or from within Mastercam, are created using metric values until this option is changed back to inch units.

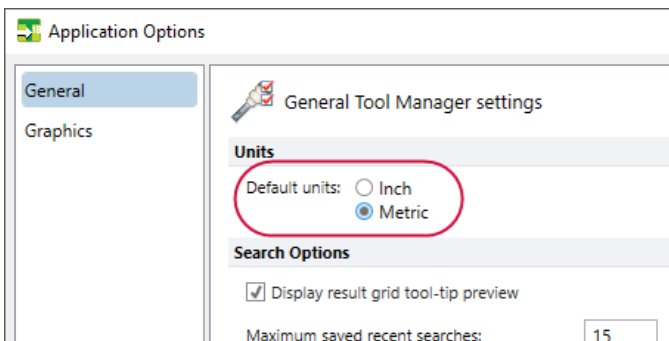
You can have both inch and metric tooling components in the same library (.TOOLDB file).

Creating a Tool Library from an Existing Library

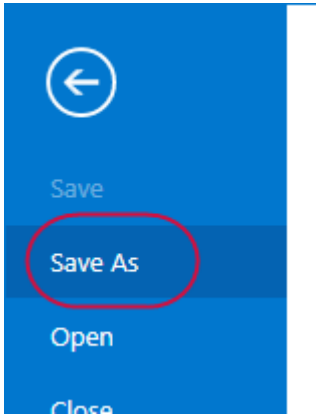
1. If the stand-alone Tool Manager is not already running, Open Tool Manager from your Mastercam folder in the Windows' Start menu.
2. Open `ToolManagerTutorial.tooldb`, provided with this tutorial.
3. After the tool library loads, click the **File** tab (backstage view) and choose **Options**.



4. In the **Application Options** dialog box, choose the **General** page.
5. If it's not already selected, set **Default units** to **Metric**.



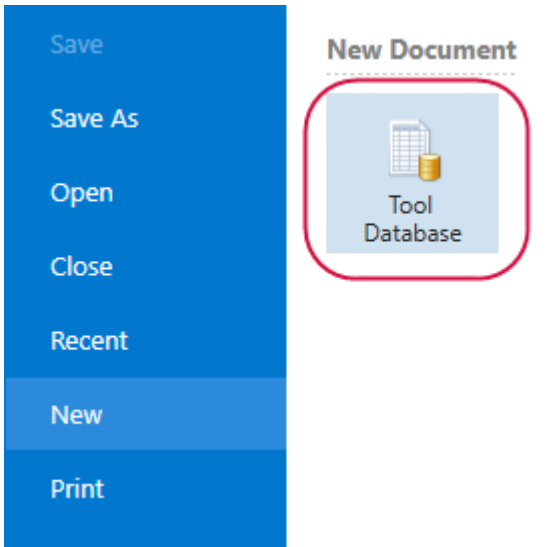
6. Click **OK**.
7. Select **File, Save As**.



8. Save the TOOLDB as `ToolManagerTutorial_XXX.tooldb`, where XXX are your initials. You save the library with a different file name so that you do not overwrite the file provided with this guide.

Creating a New Tool Library from Scratch

1. Select **File, New, Tool Database**.



This opens a second instance of Tool Manager.

2. In the Save As dialog box, enter a new file name of your choice and click **Save**.

Note: As you work in Tool Manager, your changes are automatically saved to the database.

3. Close both sessions of Tool Manager. Save any changes if prompted.

Exercise 2: Opening and Editing a Tool Library

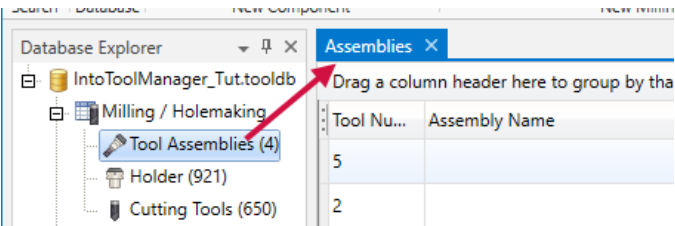
In this exercise, you open an existing library in Tool Manager and edit one or more of its components.

1. Open the stand-alone Tool Manager.
2. Select **Open** and navigate to the `ToolManagerTutorial_XXX.tooldb`

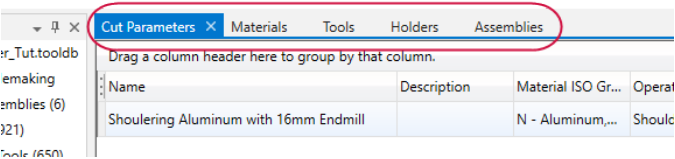
file you created in the previous exercise.



3. Open each of the document types by double-clicking its library component in the **Database Explorer**.

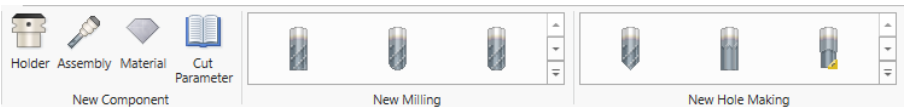


4. You should have the following documents open:



5. Review the buttons in the **New Component**, **New Milling**, and **New Hole Making** groups. Hover your mouse over each icon to view a pop-up tooltip.

Below the following graphic are brief descriptions of each item.



- **Holder:** Puts focus on the Holders document, and opens the Holder Wizard where you can create a new holder and add it to the library

database.

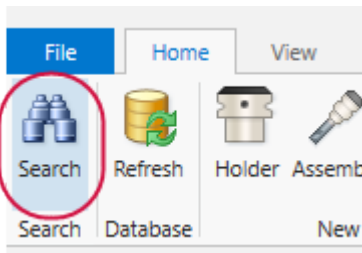
- **Assembly:** Puts focus on the Assemblies document, and opens the Assembly Tree where you can create a new assembly.
- **Material:** Puts focus on the Materials document, adds a blank row, and allows you to create a new material by defining its properties in the Properties Grid.
- **Cut Parameter:** Puts focus on the Cut Parameters document, adds a blank row, and allows you to create a new cut parameter by defining its properties in the Properties Grid.
- **New Milling:** Puts focus on the Tools document and opens the Tool Wizard where you can create a new milling tool.
- **New Hole Making:** Puts focus on the Tools document and opens the Tool Wizard where you can create a new hole-making tool.

Single-click an item in a document to edit its parameters in the Properties Grid. Alternatively, double-click any tool or holder to open the Tool Wizard or Holder Wizard where you can edit parameters. You can also edit assemblies in the Assembly Tree. Edits you make in the Assembly Tree update the parameters in the Properties Grid.

Exercise 3: Searching for a Library Component

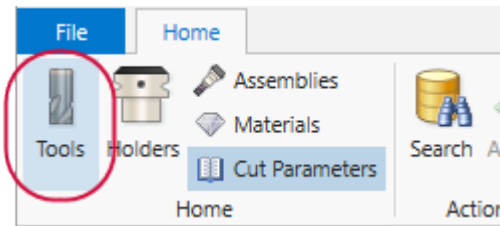
In this exercise, you search for a tool library component using the Search window.

1. Select **Search** on the **Home** tab.



The Search window displays.

- 2. Select **Tools** on the **Home** tab.



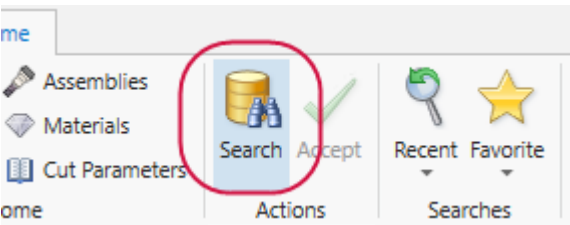
In this exercise, you are searching for a specific tool.

- 3. Select items from the **Search Item** and **Condition** drop-down lists, and enter appropriate search criteria in the **Value** field to further narrow (filter) your search. For this exercise, enter values as shown below:

Search Item	Condition	Value
Number of Flutes	=	4
Metric	=	True
Diameter	=	25

As soon as you choose a **Search Item**, a new row in the datagrid appears, allowing you to search by more than one Search Item. To remove a line of search criteria, select it and press **[Delete]**.

- 4. Select **Search** to display your search results.



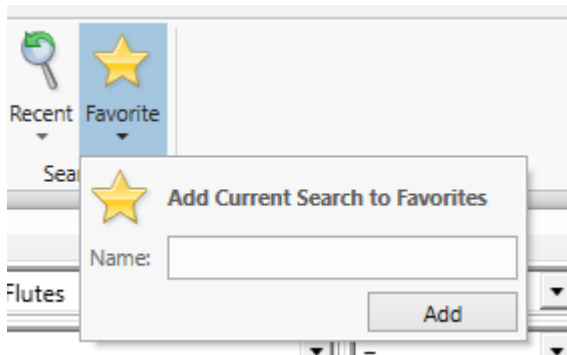
5. Your search results should be similar to the image below:

Tool Nu...	Name	Diameter	Cutting Depth	Overall Length	Corner Radius	Radius
612	25. BULL ENDMILL 4. RAD	25	50	75	4	Corner
513	25 / 45 CHAMFER MILL	25	25	50	0	None
510	25. BALL ENDMILL	25	50	75	12.5	Full Rac
632	25. MM 120 DEGREE ENGRAVE T...	25	50	75	0	None
633	25. MM 60 DEGREE ENGRAVE TO...	25	50	75	0	None
613	25. BULL ENDMILL 2. RAD	25	50	75	2	Corner
634	25. MM 30 DEGREE ENGRAVE TO...	25	50	75	0	None
635	25. MM 90 DEGREE ENGRAVE TO...	25	50	75	0	None
485	25. FLAT ENDMILL	25	50	75	0	None
614	25. BULL ENDMILL 1. RAD	25	50	75	1	Corner
615	25. BULL ENDMILL 3. RAD	25	50	75	3	Corner

These results are automatically added to your Recent Search list, accessed by selecting **Recent**. To execute the search through shortcut keys, press **[Ctrl] + [Enter]**.

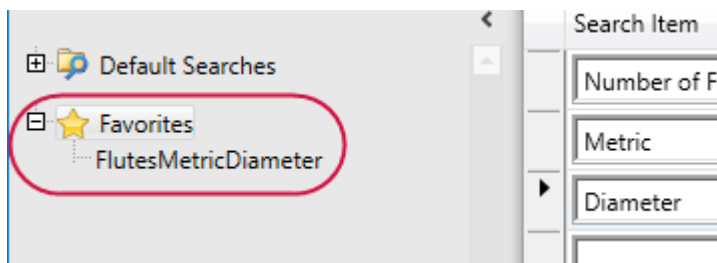
To add an item to a database already open in Tool Manager, drag the item from your Search Window and drop it into Tool Manager's Database Explorer. The item is added to the appropriate document.

6. Select **Favorite** on the **Home** tab. This saves the search as a favorite for future reference.

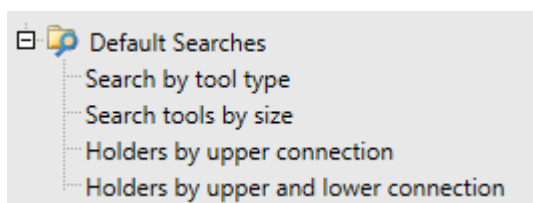


7. Enter **FlutesMetricDiameter** for the name of the search, and click **Add**.

8. The name of the saved Favorite is listed in the **Favorites** folder:



9. You can also use the **Default Searches**, provided with the Tool Manager, to open common database searches.



10. Exit the Search window.

Working with Components in a Tool Library

In this lesson, you learn how to create and edit tool library components.

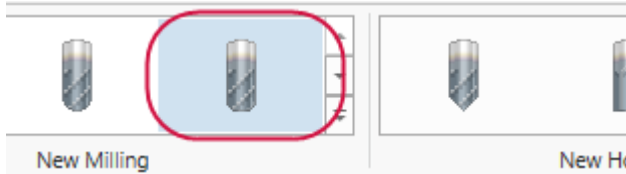
Lesson Goals

- Create and edit tools, holders, and assemblies
- Use the graphics window display to edit assemblies
- Add and edit materials and cut parameters

Exercise 1: Creating a New Tool

In this exercise, you create a new tool.

1. If necessary, open the `ToolManagerTutorial_XXX.tooldb` file.
2. Select the **Bull-Nosed Endmill** tool type in the **New Milling** group.



The Tool Wizard displays.

3. In the Define Tool Geometry page, change the following parameters:
 - **Overall length** to 85.
 - **Cutting length** to 30.

4. Ensure that your other parameters match those shown in the image below. The tool image updates as you make changes.

The image shows a screenshot of the Mastercam Tool Manager dialog box, which is divided into three sections, each with a collapse/expand arrow in the top right corner.

- Overall dimensions:** This section contains three input fields:
 - Cutting diameter: 12
 - Overall length: 85
 - Cutting length: 30
- Tip / corner treatment:** This section contains a row of four corner treatment icons (fillets). The third icon, which shows a 90-degree fillet, is selected and highlighted in blue. Below the icons is a "Radius:" input field with the value 3.
- Non-cutting geometry:** This section contains three input fields:
 - Shoulder length: 30
 - Shoulder diameter: 12
 - Shank diameter: 12Below these fields is a row of three icons representing different non-cutting geometries. The first icon, which shows a simple cylindrical shank, is selected and highlighted in blue.

5. Click **Next** to work in the Finalize Properties page.
6. In the Finalize Properties page, make the following changes:
- Set **Tool number** to 1.
 - Set **Length offset** to 21.
 - Set **Diameter offset** to 41.

7. Ensure that your other parameters match those shown in the image below.

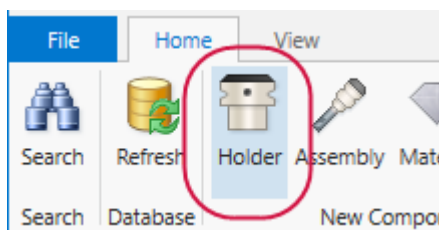
Operation	
Tool number:	1
Length offset:	21
Diameter offset:	41
Head number:	0
CS:	37.70028
FPT:	0.03
Number of flutes:	4
Feed rate:	120
Plunge rate:	600
Retract rate:	1200
Spindle speed:	1000
Spindle direction:	Clockwise
Material:	Carbide

8. Click **Finish** to save your changes. This adds the new tool to the open library, and closes the Tool Wizard.

Exercise 2: Creating a New Holder

For this exercise, you create a new tool holder.

1. Select **Holder** on the **Home** tab.



The Holder Wizard displays.

2. In the Define Holder Geometry page, change the following parameters:
 - Set **Upper Type** to **CAT**.
 - Set **Upper Size** to **30**.
3. Press **[Enter]**. This displays and highlights the holder that matches your entered parameters.
4. Ensure that your parameters match those shown in the image below.

Connection ▲

Upper:	<div style="border: 1px solid gray; padding: 2px;">Type</div> <div style="border: 1px solid gray; padding: 2px;">CAT ▼</div>	<div style="border: 1px solid gray; padding: 2px;">Size</div> <div style="border: 1px solid gray; padding: 2px;">30</div>
Lower:	<div style="border: 1px solid gray; padding: 2px;">ID</div> <div style="border: 1px solid gray; padding: 2px;">0</div>	

	Top	Height	Bottom
	31.75	3.175	31.75
	46.025	4.544	46.025
	46.025	1.914	39.394
	39.394	2.958	39.394
	39.394	1.914	46.025
	46.025	4.544	46.025
	31.75	16	31.75

Note: If you need to add and define a new segment, select **Add Segment** and define the segment parameters in the database grid. To remove a segment, select the leftmost cell for that row and press the **[Delete]** key.

5. Select **Next**.
6. Enter **TUTORIAL MAIN HOLDER** in the **Description** field and select **Thru coolant** to set the coolant option to flow through the holder.

General

Name: CAT 30 - ID 0

Description: TUTORIAL MAIN HOLDER

Manufacturer name: Mastercam +

Catalog ID:

☒ Metric

☒ Thru coolant

7. Select **Finish** to save your changes. This adds the new holder to the open library, and closes the Holder Wizard.

Exercise 3: Editing Existing Tools and Holders

In this exercise, you edit a tool through the Tool Wizard. Editing a holder is similar so it is not covered. The only difference is, when you edit holders, you work in the Holder Wizard as described in the previous exercise.

Note: In addition to editing tools and holders, you can also select a component in the document, and edit its properties in the Properties Grid.

1. Open the Tools document and double-click tool number **463** (3. FLAT ENDMILL).

The screenshot shows the Mastercam Tool Manager interface. On the left is the Database Explorer showing a tree structure with folders like 'Milling / Holmaking', 'Tool Assemblies (6)', 'Holder (922)', 'Cutting Tools (650)', 'Materials (3)', and 'Cut Parameters (1)'. On the right is the 'Tools' tab, which displays a table of tool data. The table has columns for 'Tool Nu...', 'Name', and 'Diameter'. The row for tool number 463, named '3. FLAT ENDMILL' with a diameter of 3, is highlighted with a red circle.

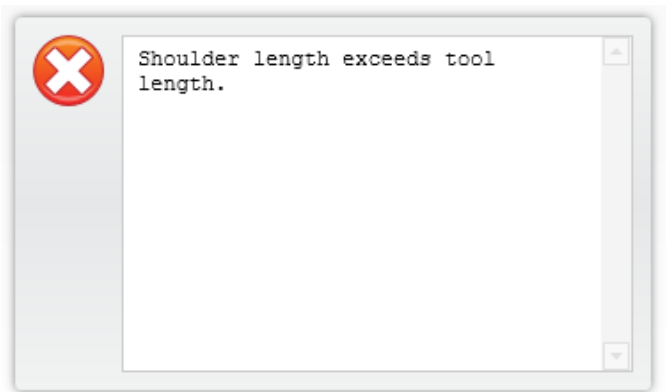
Tool Nu...	Name	Diameter
396	12.6 CBORE	12.6
463	3. FLAT ENDMILL	3
448	23. CBORE	23
473	13. FLAT ENDMILL	13

The Tool Wizard displays.

2. In the Define Tool Geometry page, change the following parameters:

- Set **Cutting diameter** to 5.
- Set **Overall length** to 50.
- Set **Cutting length** to 25.
- Set **Shoulder length** to 30.

Note: If you see a message in the graphics display similar to the one shown below, ignore it for this exercise. As you complete entering the values, the error messages are replaced with the tool graphic because you have resolved all geometric property conflicts.



3. Ensure your parameters match those shown in the image below:

The image shows three sections of a tool definition dialog:

- Overall dimensions:** Contains three input fields: "Cutting diameter:" with value 5, "Overall length:" with value 50, and "Cutting length:" with value 25.
- Tip / corner treatment:** Contains four icons representing different corner treatments. The first icon, showing a 90-degree internal corner, is selected and highlighted in blue.
- Non-cutting geometry:** Contains three input fields: "Shoulder length:" with value 30, "Shoulder diameter:" with value 3, and "Shank diameter:" with value 3. Below these are three icons representing different shank profiles. The first icon, showing a simple cylindrical shank, is selected and highlighted in blue.

4. Select **Next**.
5. Set the following parameters:
- Set **Tool number** to 2.
 - Set **Length offset** to 22.
 - Set **Diameter offset** to 42.
 - Enter **FLAT ENDMILL - TUTORIAL** in **Name**.

The image shows two sections of a tool definition dialog:

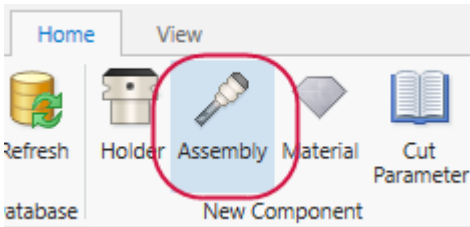
- Operation:** Contains three input fields: "Tool number:" with value 2, "Length offset:" with value 22, and "Diameter offset:" with value 42.
- General:** Contains three input fields: "Name:" with value FLAT ENDMILL - TUTORIAL, "Description:" (empty), and "Manufacturer name:" (empty with a dropdown arrow and a plus icon).

6. Select **Finish** to save your changes.

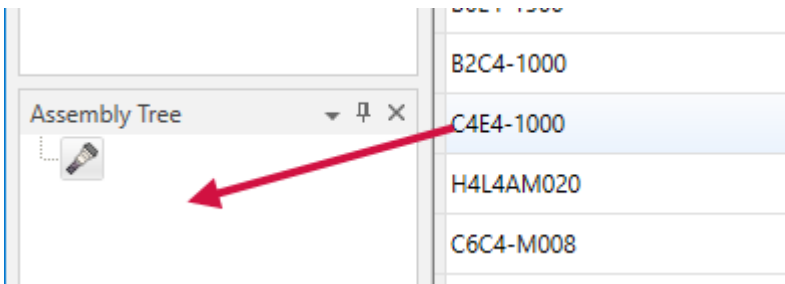
Exercise 4: Creating and Editing Assemblies

Create assemblies using a drag and drop method, as described in the following procedure. Although here you only add one holder to the assembly, you can add multiple holders as segments of a single holder using the same drag and drop method.

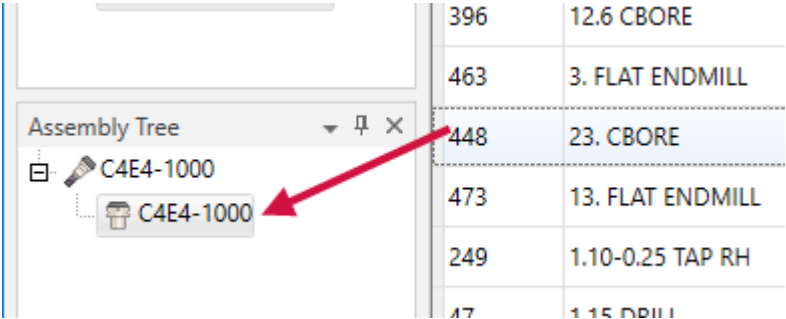
1. Select **Assembly** on the **Home** tab. This opens the Assembly Tree pod and the Assemblies document.



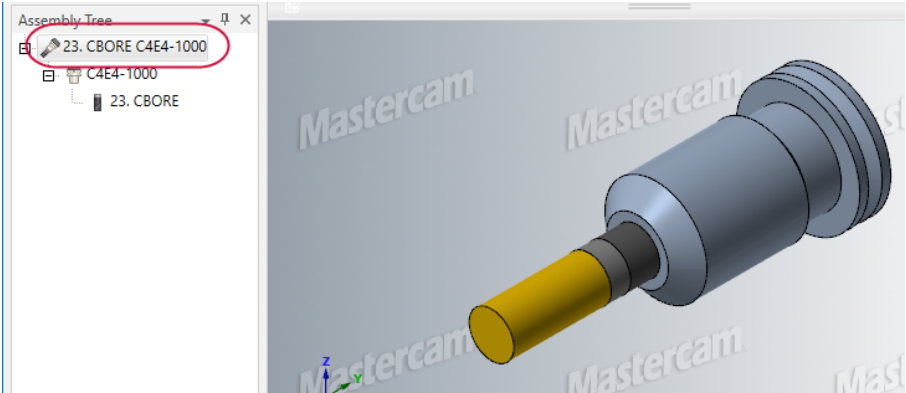
2. Select the **Holders** document.
3. Drag and drop the holder, **C4E4-1000**, from the Holders document into the Assembly Tree.



4. Drag and drop the tool, **23. CBORE**, from the Tools document into the Assembly Tree.

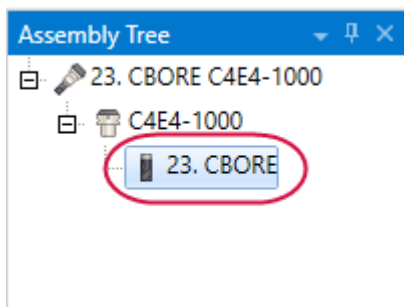


5. View a graphical representation of your assembly in the graphics window by clicking the top line of the assembly.

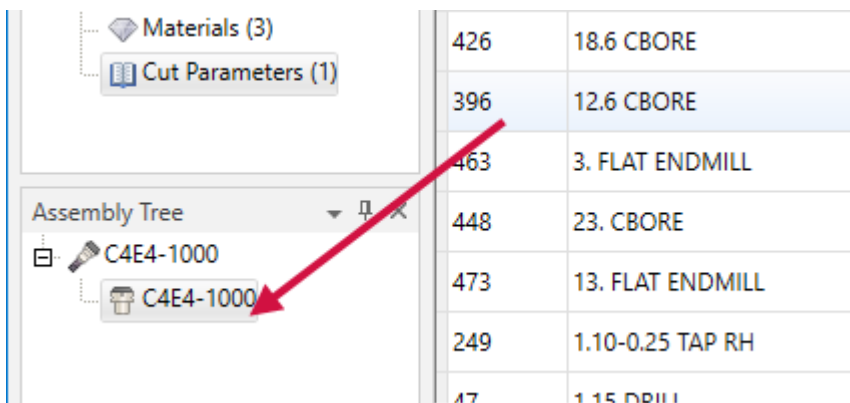


The following steps show how to change a tool in an assembly. This same process applies to any component type in assemblies.

6. Select the **23. CBORE** tool in the Assembly Tree and press the **[Delete]** key.



7. Drag and drop the tool, **12.6 CBORE**, from the Tools document into the Assembly Tree.



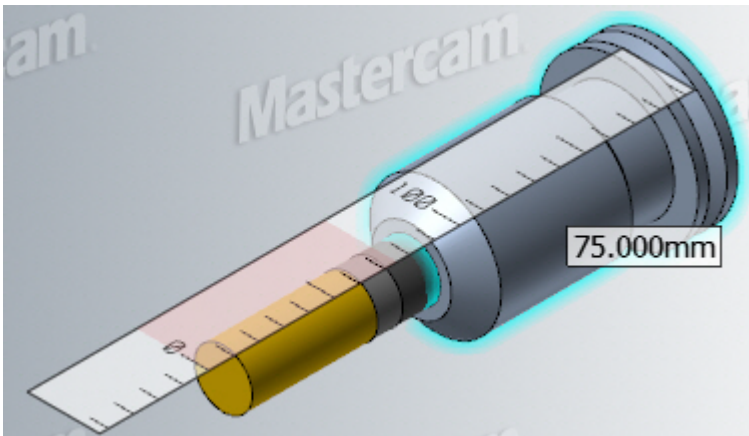
As you add and remove assembly components, the name of the assembly changes.

Editing Assemblies in the Graphics Window

Besides editing assemblies by changing settings in the Properties Grid, you can reposition the assembly (or its extensions) along the center line of the tool in the graphics window.

1. In the Assembly document datagrid, select the assembly you worked on in the previous exercise.
2. In the graphics window, place your cursor over the holder, hold down the **[Ctrl]** key, and click the holder. The assembly is highlighted and a dynamic ruler displays.

The ruler always displays absolute dimensions, which means that the tool projection value is always measured from the tool tip to the face of the holder.



When you hover over the holder, press the **[Ctrl]** key to have the cursor indicate this mode. If the cursor does not change, you may need to click the holder to put focus in the graphics window.

3. Once the ruler displays, move your mouse back and forth along the center line of the tool to change the length that the tool extends from the holder. You will see the dynamic ruler floating, and the tool projection value in the text box changes with your mouse movement.

4. Click anywhere off the tool or holder in the graphics window to set the projection distance of the tool.

When working with assemblies, you can change the Tool Projection value in the graphics window. The Tool Projection field in the Properties Grid updates as you manipulate the assembly graphic. You can also use the Tool Projection field in the Properties Grid to change those values.

When the dynamic ruler is active, change the ruler increments by zooming in or out with your mouse wheel. This is helpful if you want to be more or less precise in your Tool Projection value. The number of graduations are appropriate to the selected units specified in the Options dialog box.

With the ruler active, click and drag it to snap your Tool Projection value to the ruler's incremental distance lines. The movement is only between the shoulder length and overall length values.

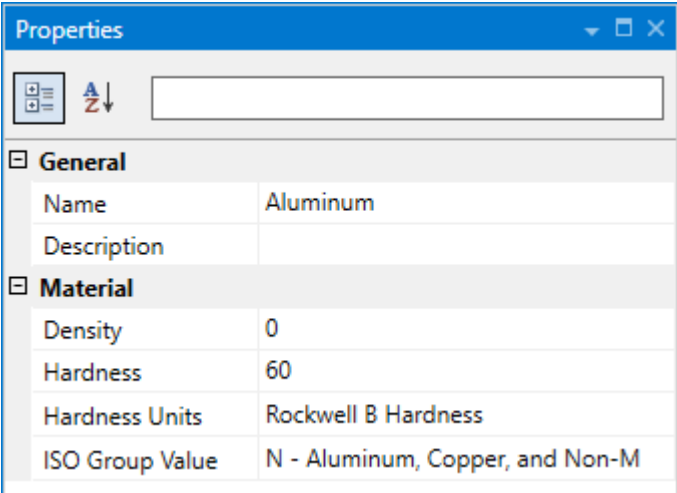
Exercise 5: Adding or Editing Materials

Workpiece materials are part of the TOOLDB database files. They display in the Materials document in Tool Manager. Although you can edit some of the materials' properties in the Properties Grid, the Materials document is primarily used for reference information when working with cut parameters.

Note: Since it isn't possible for a tool manufacturer to test every possible material for every tool, the workpiece materials specified in a tool library may not match the actual material for a given job. Use the ISO group property to find a suitable cut parameter setting as a starting point.

1. Select **Material** on the **Home** tab to display the Materials document.
2. Select the **Materials** document to view the materials grid.
3. Select the first row in the data grid and set the following parameters in the Properties grid:

- Enter **Aluminum** for Name.
- Set **Hardness** to **60**.
- Set **Hardness Units** to **Rockwell B Hardness**.
- Set **ISO Group Value** to **N - Aluminum, Copper, and Non-Metallic**.



The screenshot shows the 'Properties' dialog box in Mastercam. At the top, there is a blue title bar with the text 'Properties' and standard window controls. Below the title bar is a toolbar with icons for a list, a text field, and a dropdown menu. The main area of the dialog is divided into two sections: 'General' and 'Material'. The 'General' section contains a table with two rows: 'Name' with the value 'Aluminum' and 'Description' which is empty. The 'Material' section contains a table with four rows: 'Density' with the value '0', 'Hardness' with the value '60', 'Hardness Units' with the value 'Rockwell B Hardness', and 'ISO Group Value' with the value 'N - Aluminum, Copper, and Non-M'.

General	
Name	Aluminum
Description	

Material	
Density	0
Hardness	60
Hardness Units	Rockwell B Hardness
ISO Group Value	N - Aluminum, Copper, and Non-M

4. To edit the materials properties for that same material, change the following parameters:
- Enter **Aluminum - 6061** for Name.
 - Set **Hardness** to **95**.
 - Set **Hardness Units** to **Brinell Hardness Scale**.

The screenshot shows the 'Properties' dialog box with the 'General' tab selected. The 'Name' field is set to 'Aluminum - 6061'. The 'Material' tab is also visible, showing fields for 'Density' (0), 'Hardness' (95), 'Hardness Units' (Brinell Hardness Scale), and 'ISO Group Value' (N - Aluminum, Copper,).

General	
Description	
Name	Aluminum - 6061

Material	
Density	0
Hardness	95
Hardness Units	Brinell Hardness Scale
ISO Group Value	N - Aluminum, Copper,

Your changes are automatically saved to the database.

Exercise 6: Adding and Editing Cut Parameters

Cut parameters are also part of the TOOLDB database files. They display in the Cut Parameters document, where you can edit them as needed. These parameters are primarily used in Mastercam when you import cutting speed and feed per tooth data into an operation.

1. Select **Cut Parameters** on the **Home** tab to display the Cut Parameters document.
2. Select the **Cut Parameter** document to view the materials grid.
3. Select the first row in the Cut Parameters document and set the parameters in the Properties Grid:
 - Enter **Aluminum - 6061** for **Material Name**.
 - Ensure that **Metric** is checked.
 - Enter **Shouldering Aluminum with 16mm Endmill** for **Name**.

- Set **Operation Type** to **Shouldering**.
- Set **Feed Per Tooth** to **0.075**.
- Set **Rough Axial Depth of Cut** and **Finish Axial Depth of Cut** to **10**.
- Set **Rough Radial Depth of Cut** and **Finish Radial Depth of Cut** to **8**.
- Set **Surface Speed** to **90**.
- Set **Maximum Diameter** to **18**.
- Set **Minimum Diameter** to **16**.

The screenshot shows the 'Properties' dialog box in Mastercam. It has a blue title bar and a search bar at the top. The dialog is divided into three main sections: 'General', 'Manufacturers Data', and 'Operation'. The 'General' section includes fields for Cut Quality (Rough), Cut Source (User), Description, Machine Name, Material Name (Aluminum), Metric (checked), Name (Shouldering Aluminum with 16m), Operation Type (Shouldering), and Proven (unchecked). The 'Manufacturers Data' section has a dropdown for Manufacturer Name (Mastercam) with a plus icon. The 'Operation' section includes fields for Estimated Tool Life (m...), Feed Per Tooth (0.075), Finish Axial Depth of..., Finish Radial Depth of..., Rough Axial Depth of..., Rough Radial Depth o..., and Surface Speed (90).

General	
Cut Quality	Rough
Cut Source	User
Description	
Machine Name	
Material Name	Aluminum
Metric	<input checked="" type="checkbox"/>
Name	Shouldering Aluminum with 16m
Operation Type	Shouldering
Proven	<input type="checkbox"/>

Manufacturers Data	
Manufacturer Name	Mastercam +

Operation	
Estimated Tool Life (m...	15
Feed Per Tooth	0.075
Finish Axial Depth of...	10
Finish Radial Depth of...	8
Rough Axial Depth of...	10
Rough Radial Depth o...	8
Surface Speed	90

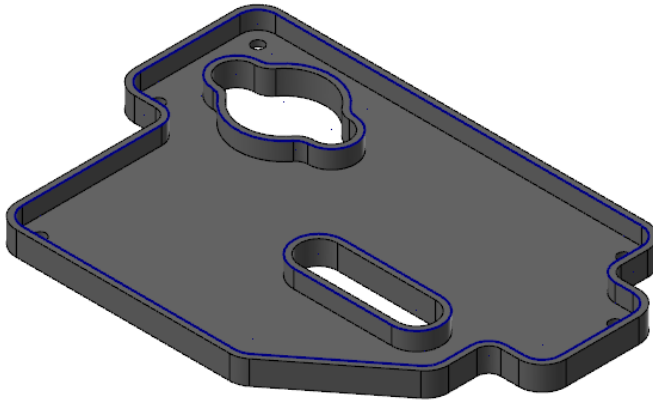
Notes:

- Some of the fields are automatically populated and non-editable because that information comes from the material you choose.
- If you are creating a new database or if the desired workpiece material does not exist, you need to create the material before creating the cut parameters.

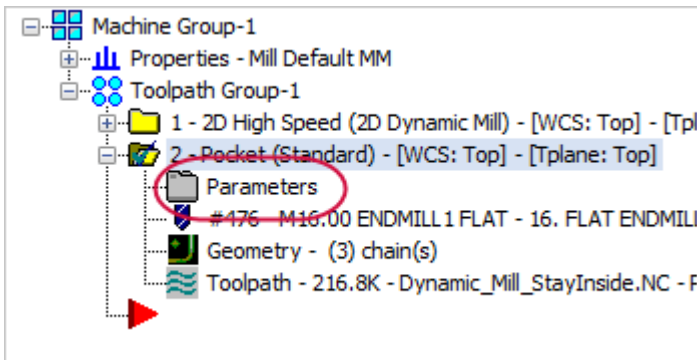
Exercise 7: Accessing Cut Parameters from Mastercam

In this exercise, you apply the cut parameters you created to a Mastercam toolpath to update the surface speed and feed per tooth values.

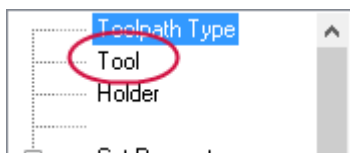
1. Start Mastercam 2018.
2. Open the part file `Dynamic_Mill_Rest`, provided with this tutorial.



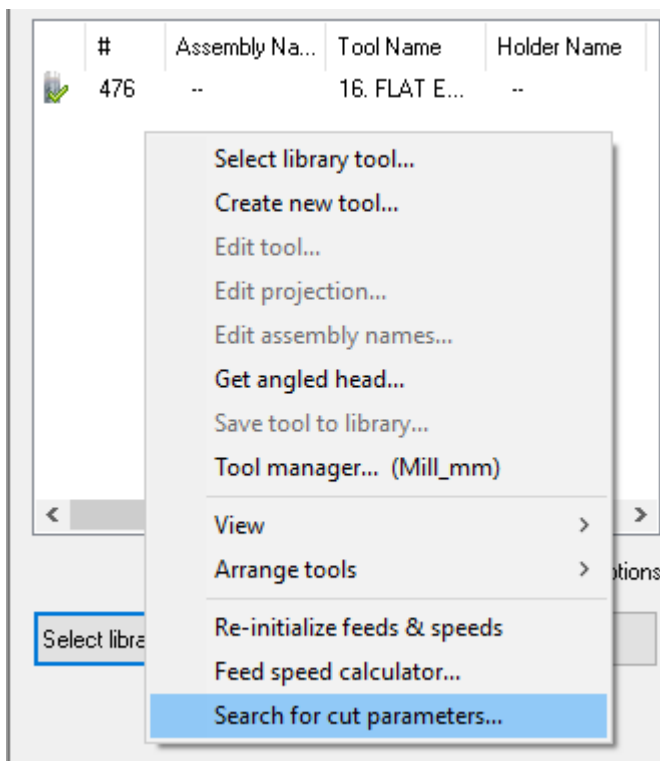
3. Select **File, Save As**, and save the part as `Dynamic_Mill_Rest_XXX`, where XXX are your initials.
4. Select **Parameters** under the second operation in the Toolpaths Manager.



5. Select the **Tool** page.



6. Right-click in the tool list and select **Search for cut parameters**.



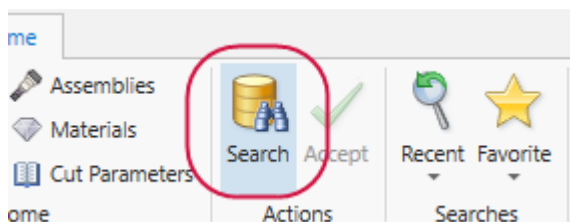
The Search window displays.

7. Select **File, Open**, and select the `ToolManagerTutorial_XXX.tooldb` file.

8. Ensure that the search criteria is as shown below:

Search Item	Condition	Value
Maximum Diameter	>=	16
Minimum Diameter	<=	16
Material ISO Group	=	N - Aluminum, Copper, and Non-Metallic
	=	

9. Select **Search**. Your search results will display the cut parameter you created in the previous exercise.



10. Select the resulting cut parameter and click **Accept**.

Name	Description	Ma
Shouldering Aluminum with 16mm Endmill		N -

This closes the Search window, and populates the Tool page feed per tooth (FPT) and cutting speed (CS) values with the parameter values you accepted.

Feed rate:	537.0	Spindle speed:	1790
FPT:	0.075	CS:	90.0
Plunge rate:	268.5	Retract rate:	268.5

11. Click **OK** to accept the changes and close the dialog box.

Note: The Assembly Has Changed dialog box may display. This dialog box alerts you that the tool or holder has been modified, and is referenced in more than one operation in the current part file. If this dialog box displays, select **Update assembly and all operations that use the assembly** and click **OK**.

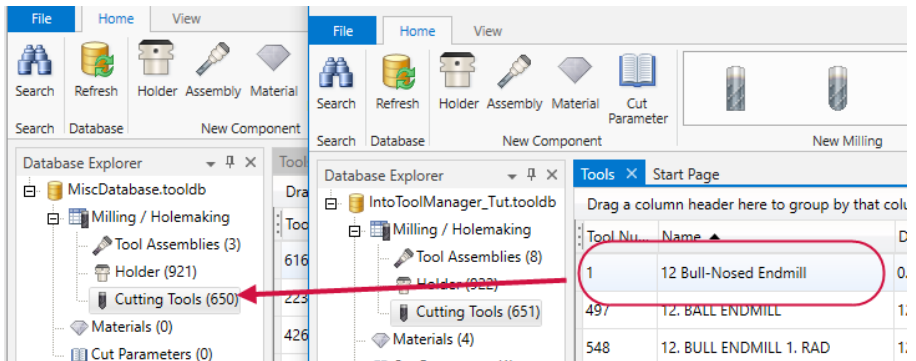
12. Regenerate the toolpath.
13. Save the part file, and close Mastercam.

Exercise 8: Copying Components Between Tool Libraries

Use drag and drop to copy items from one database to another. To do this, you must have two or more sessions of Tool Manager open concurrently.

1. Start another session of Tool Manager through the Windows' **Start** menu. You should still have the `ToolManagerTutorial_XXX.tooldb` open in the current session of Tool Manager.
2. Click the **Open** button and navigate to this guide's install location.
3. Select the `ToolManagerTutorial.tooldb` file and click **Open** to load the database.
4. Position the two sessions of Tool Manager side-by-side or one partly covering the other, resizing the windows as necessary.
5. Open the **Tools** document in each Tool Manager window.
6. In the `ToolManagerTutorial_XXX.tooldb` session of Tool Manager, select tool number **1 (12 Bull-Nosed Endmill)**.

7. Drag and drop the **12 Bull-Nosed Endmill** onto **Cutting Tools** in the ToolManagerTutorial.tooldb session of Tool Manager. Because the copied tool has the same tool number as an existing tool, it is renumbered to the next available number.



You can copy database items under the following drag and drop conditions:

- From one database Assemblies, Holders, Tools, Materials, or Cut Parameters document to another database's document of the same type.
- From any Assemblies, Holders, or Tools document in one database to the Assembly Tree of another to build or copy new assemblies.

You can change also tool numbers by double-clicking the tool in the document and changing the tool number in the Tool Wizard, or you can highlight the tool in the document and change the tool number in the Properties grid.

8. Close both sessions of the stand-alone Tool Manager.

Using Tools and Holders in Toolpaths

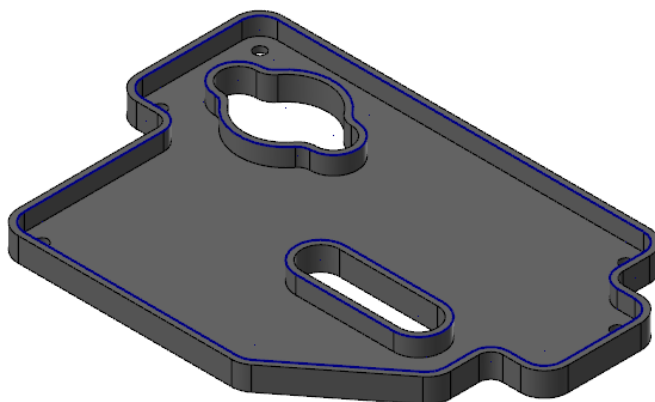
In this lesson, you learn to work in Mastercam's classic Mill Tool Manager using the stand-alone Tool Manager's integrated functions.

Lesson Goal

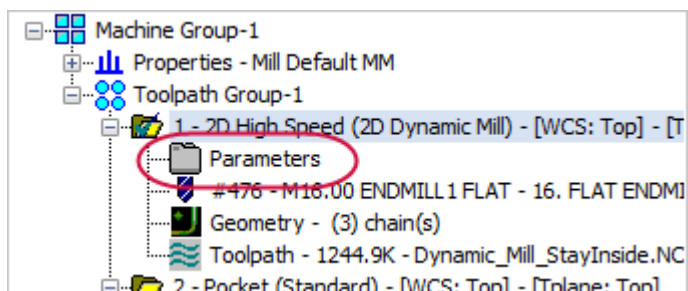
- Edit tools and holders using classic Tool Manager

Exercise 1: Creating a New Tool

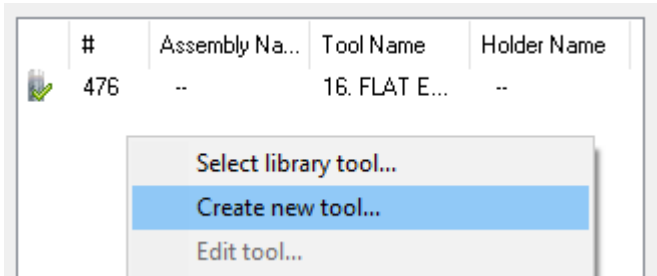
1. Start Mastercam.
2. Open the part file `Dynamic_Mill_Rest_XXX`, from the previous lesson.



3. Click the **Parameters** icon for the first operation in the Toolpaths Manager.



4. In the Tool page of the Dynamic Mill dialog box, right-click in the tool list and choose **Create new tool**.



The Tool Wizard displays.

5. In the Select Tool Type page, select **End Mill**, and click **Next**.
6. In the Define Tool Geometry page set the following tool parameters:
- Set **Cutting diameter** to **18**.
 - Set **Overall length** to **90**.
 - Set **Cutting length** to **60**.
 - Ensure that **Shoulder length** is set to **60**.
 - Ensure that **Shoulder diameter** is set to **18**.
 - Ensure that **Shank diameter** is set to **18**.

7. Make sure your parameters in the Define Tool Geometry page match those shown in the following graphic.

Overall dimensions

Cutting diameter: 18

Overall length: 90

Cutting length: 60

Tip / corner treatment

Non-cutting geometry

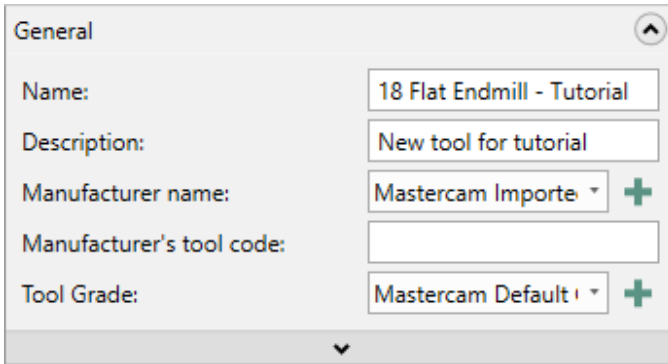
Shoulder length: 60

Shoulder diameter: 18

Shank diameter: 18

8. Click **Next** to work in the Finalize Properties page.
9. In the **Finalize Properties** page, make the following changes:
- Enter **18 Flat Endmill - Tutorial** for **Name**.
 - Enter **New tool for tutorial** for **Description**.

10. Make sure your parameters in the Finalize Properties page match those shown in the graphic below.



The screenshot shows the 'General' tab of the Mastercam Tool Manager dialog box. It contains the following fields and values:

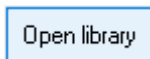
Field	Value
Name:	18 Flat Endmill - Tutorial
Description:	New tool for tutorial
Manufacturer name:	Mastercam Importe +
Manufacturer's tool code:	
Tool Grade:	Mastercam Default +

At the bottom of the dialog box, there is a downward-pointing arrow icon.

11. Click **Finish** to save your changes. This adds the new tool to the part, closes the Tool Wizard, and returns you to the Tool page.
12. Do not close the 2D High Speed Toolpath - Dynamic Mill dialog box.

Exercise 2: Creating a New Holder

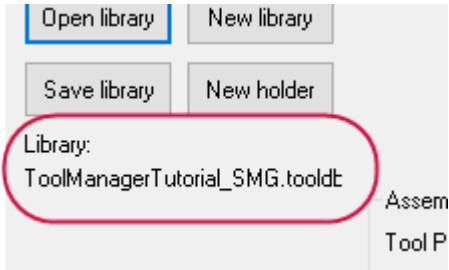
1. Select the **Holder** page.
2. Select **Open library**.



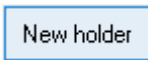
The Open dialog box displays.

3. Navigate to your ToolManagerTutorial_XXX file.

4. Select it, and click **Open**. The Holder page will display your current library.

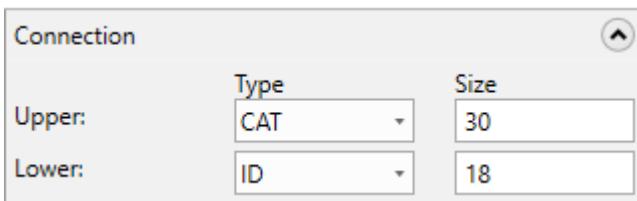


5. Select **New holder**.



The Holder Wizard displays.

6. In the Define Holder Geometry page, set the following parameters:
- Set **Upper Type** to **CAT**.
 - Set **Upper Size** to **30**.
 - Set **Lower Type** to **ID**.
 - Set **Lower Size** to **18**.
7. Make sure your parameters in the Define Holder Geometry page match those shown in the image below.



8. Click **Next**.

9. In the Finalize Properties page, set the following parameters:
 - Enter **New Holder for Tutorial** for **Description**.
 - Select **Thru coolant**, to set the coolant to flow through the holder.
10. Make sure your parameters in the Finalize Properties page match those shown in the following image.

General

Name: CAT 30 - ID 18

Description: New Holder for Tutorial

Manufacturer name: Mastercam Importer +

Catalog ID:

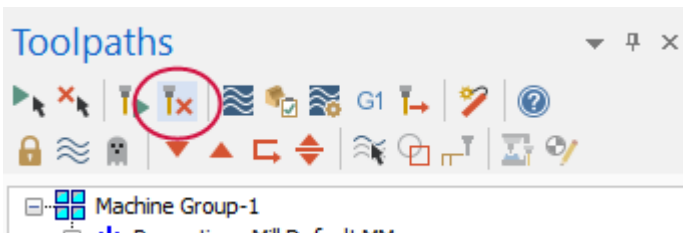
☒ Metric

☒ Thru coolant

11. Click **Finish** to save your changes. This adds the new holder to the part, closes the Holder Wizard, and returns you to the Holder page.

You can see the new holder is added to the list, but not yet saved to the library. This is indicated with a message in the window and an asterisk next to the new holder name.

12. Click **Save library** and save your tool library.
13. Click **OK** in the Dynamic Mill dialog box. This marks the toolpath dirty.
14. Click **Regenerate all dirty operations** to regenerate the toolpath.

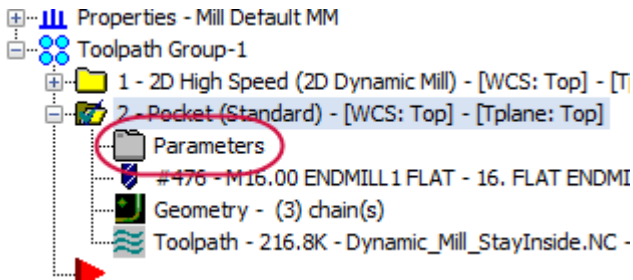


15. Save the part file.

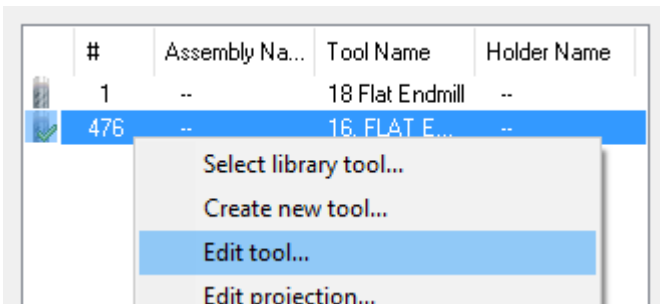
Exercise 3: Editing Tools and Holders

In this exercise, you edit a tool and holder inside of an operation in Mastercam.

1. Open the part file, `Dynamic_Mill_Rest_XXX` from the previous exercise, if it is not already open.
2. Select **Parameters** under the second operation in the Toolpaths Manager, and go to the **Tool** page.



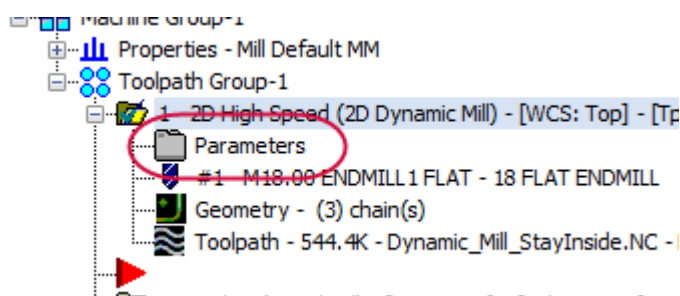
3. Right-click tool #476 and choose **Edit tool**.



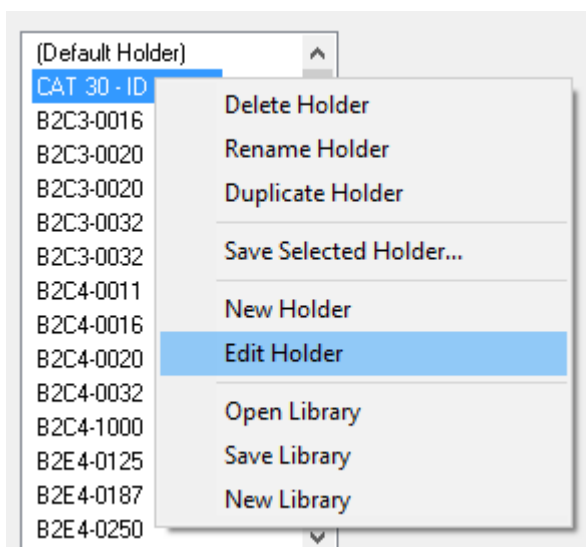
The Tool Wizard displays.

4. In the Define Tool Geometry page, change the **Shank diameter** to 22.
5. Click **Finish** to save your changes. This closes the Tool Wizard, and returns you to the Tool page.

6. Click **OK** in the Pocket dialog box.
7. Select **Parameters** under the first operation in the Toolpaths Manager.



8. Select the **Holder** page.
9. Right-click on the **CAT 30 - ID 18** holder and choose **Edit Holder**. You may need to scroll down the list to locate it.



The Holder Wizard displays.

10. In the Define Holder Geometry page, click **Next** to leave settings as is.

11. In the Finalize Properties page, change the **Name** to **CAT 30 - ID 18 - Tutorial**.
12. Click **Finish** to save your changes. This adds the edited holder to the part, closes the Holder Wizard, and returns you to the Holder page.
13. Save your changes to the library as described in the previous exercise.
14. Click **OK** in the 2D High Speed Toolpath - Dynamic Mill dialog box.
15. Save the part file.

Rendering Mastercam Tools

Mastercam has the ability to define a tool using parametric properties along with a custom profile that can be imported from a CAD file, or a part file created in Mastercam. The parametric values are used within the toolpath calculations for improved performance. The optional custom profile, when provided, is used for stock model generation and collision checking in Backplot, Verify, and Machine Simulation. Mastercam's Tool Wizard provides the option of displaying both profiles for comparison.

The Tool Wizard renders the custom profile as a 3D shaded tool and overlays the parametric profile using a dotted line. You can toggle this behavior to render the parametric tool and overlay the custom profile.

Lesson Goals

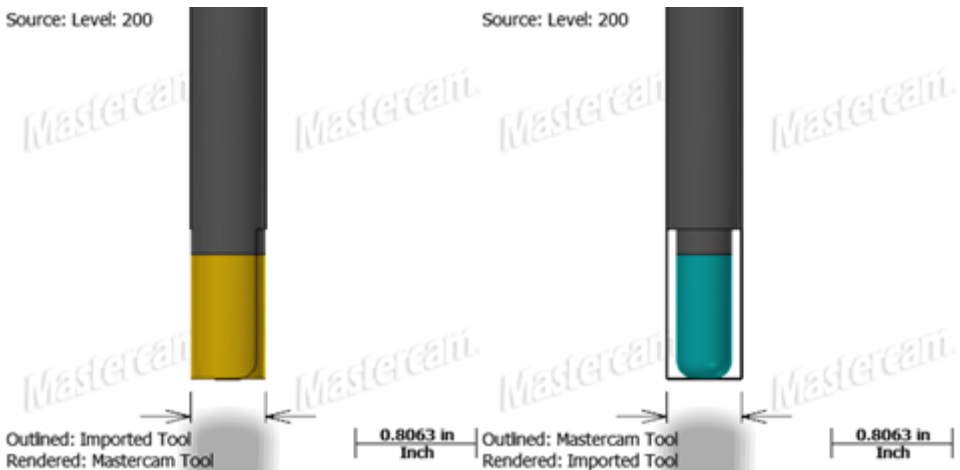
- Understand how Mastercam renders tool profiles
- Create a new tool from a profile

How Mastercam Renders Tool Profiles

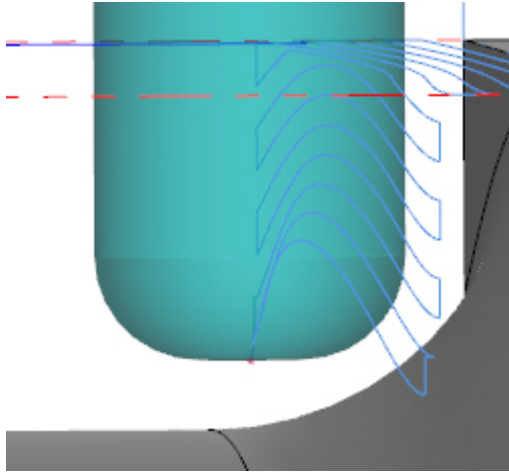
When you use a tool profile to create a new tool, whether you import the profile from an external file (DXF) or from a profile stored in your part file, if you choose to create the tool based on the Custom Tool type, Mastercam tries to approximate its parameters.

Occasionally, Mastercam fails to populate the Custom Tool parameters accurately. The tool may look correct in the Tool Wizard when rendered as an imported tool and will display as a correctly-defined tool in backplot and verify, but Mastercam uses the tool parameters from the Define Tool Geometry page in the Tool Wizard when creating toolpaths, which do not always match the true tool profile. This can result in unexpected toolpath behavior such as gouging, leaving too much stock, or random airlifts. Mastercam is using the toolpath compensation based on the parametric values rather than the imported profile shape.

The example below shows an incorrectly defined tool from a profile stored on level 200 in the `Custom_tool_Testing` part file included with this tutorial. You can see that the rendered tool that Mastercam created (shown on the left) does not match the actual definition from the imported profile (shown on the right).



Even though the rendered Imported Tool shown on the right looks correctly defined, Mastercam uses the rendered Mastercam Tool shown on the left during toolpathing. The results of the cut are shown below, viewed from the right-side view. Because Mastercam is compensating the toolpath to the Mastercam Tool, which is shaped like a flat endmill and not the Imported Tool, the tool is not cutting to the surface of the part, but rather leaving a gap. Mastercam is compensating the toolpath to the theoretical corner of the Mastercam Tool, even though it displays as the Imported Tool.



This can often be misdiagnosed as a toolpath problem. If you have a referenced profile and/or custom tool and are seeing something incorrect in the toolpath, check your tool definition and make sure that the Mastercam Tool and Imported Tool match.

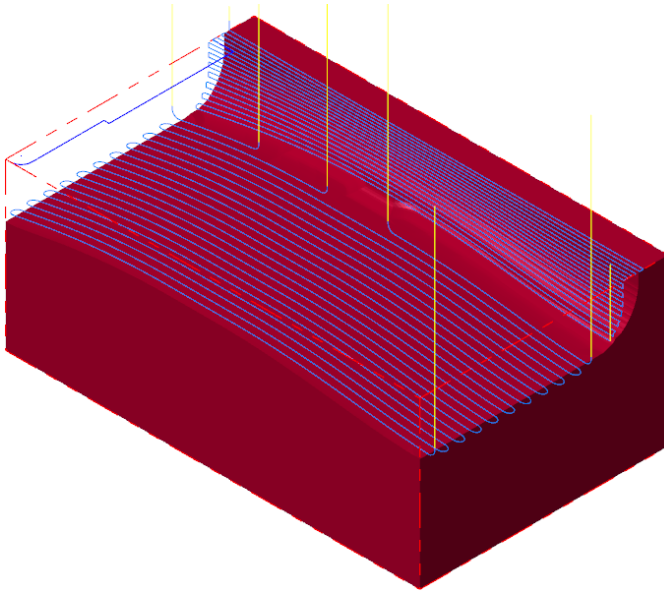
View this by backplotting operations 1 and 2 of the `Custom_Tool` part file included with this tutorial.

To create custom tools based on profile geometry, it is almost always best to select a tool type that most closely resembles the tool you are creating rather than basing the tool on the Custom Tool type. By basing your custom tool on a similar tool type, you can parametrically define the tool to match the Mastercam Tool upon which your toolpaths are calculated.

Exercise 1: Creating New Tools from Profiles

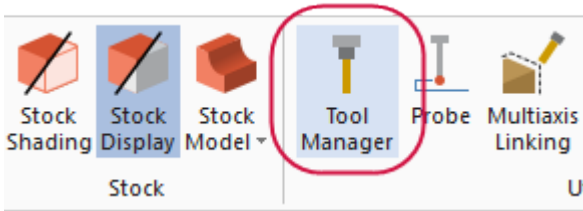
In this exercise, you create a new tool based on a tool profile from within Mastercam. These tools are used in toolpaths on the part supplied with this tutorial.

1. Start Mastercam 2018.
2. Open the part file `Custom_Tool`, provided with this tutorial.
3. Right-click in the graphics window and choose **Fit** to fit the part to the screen. Your part should look similar to the graphic below.



4. Select **File, Save As**, and save the part as `Custom_Tool_XXX`, where XXX are your initials.

5. Select **Tool Manager** on the **Mill Toolpaths** contextual tab.

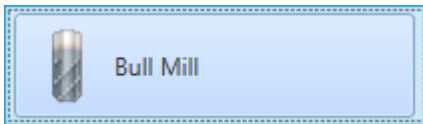


You will be creating a custom tool similar to the 0.375 Bull-Nosed Endmill tool used in operation 4.

6. Right-click in the tool list window and select **Create new tool**.

The Tool Wizard displays.

7. Select the **Bull Mill** tool type.




The custom tool profile is created as a bull-nosed endmill.

8. Select **Next**.
9. Select **Import and link to custom geometry from a level**.



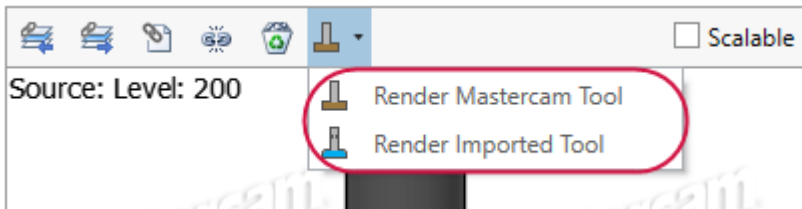
The Select Level dialog box displays.

10. Select **level 200**, and click **OK**.



Number	Level Name	# Entities
4	Solid	1
100	Feed mill Profile	8
200	Bull mill shoulder	6
1000	Junk WF	8

11. Toggle between viewing the custom tool by select **Render Mastercam Tool** and **Render Imported Tool**. They do not match. The Mastercam Tool parameters need to match the Imported Tool profile.



12. On the Define Tool Geometry page, enter the following parameters:

- Set **Cutting diameter** to **0.375**.
- Set **Overall length** to **2.5**.
- Set **Radius** to **0.125**.
- Set **Shoulder length** to **1**.
- Set **Shoulder diameter** to **0.375**.
- Set **Shank diameter** to **0.5**

13. Ensure that your parameters in the Define Tool Geometry page match those shown below:

Overall dimensions

Cutting diameter: 0.375

Overall length: 2.5

Cutting length: 0.833333

Tip / corner treatment

Radius: 0.125

Non-cutting geometry

Shoulder length: 1

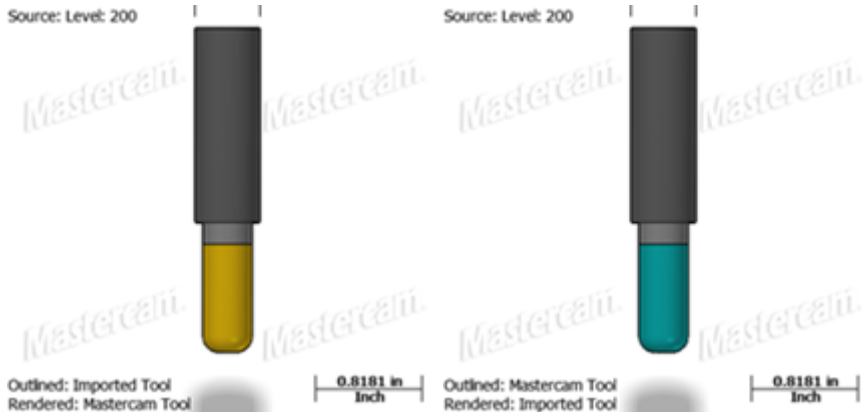
Shoulder diameter: 0.375

Shank diameter: 0.5

Notes:

- When creating custom tools from imported profiles, Mastercam populates the parametric fields with estimated values. However, you must edit these values to match the imported profile parameters. Usually these imported tool profile parameters are retrieved in one of the following methods:
- If the tool profile was created without importing it from a clearly-defined specification, you can analyze the profile geometry in Mastercam to obtain the values.
- If the tool profile was imported from a manufacturer, you can find the values from their web site or tooling catalog.

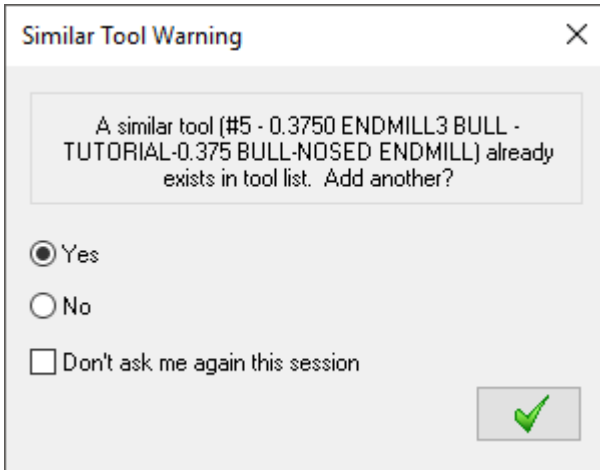
- If you have a tool but do not know the origin of that tool, you can obtain the parameters using a tool setter to measure the tool values.
14. Toggle between viewing the custom tool as a rendered Mastercam Tool or a rendered Imported Tool. You can see that the rendered shapes match, showing that the Mastercam Tool that is used to create the toolpath has the same profile values as the Imported Tool.



15. Select **Next** and edit the parameters as shown below, renumbering the tool as needed.

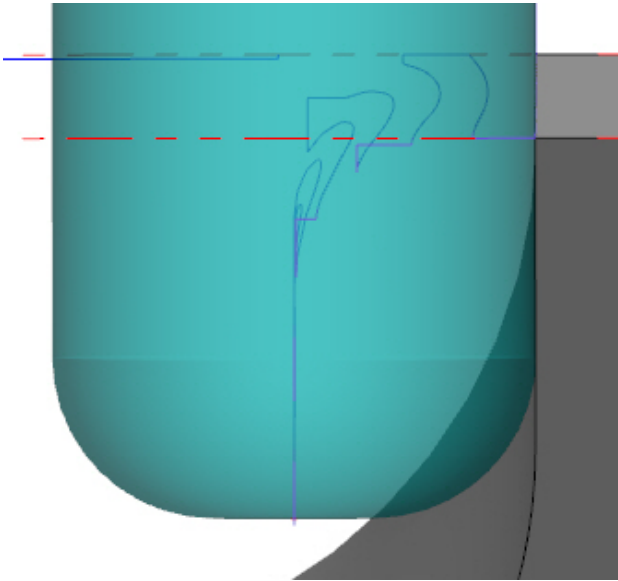
The image shows two screenshots of the Mastercam Tool Manager dialog boxes. The left screenshot shows the 'Operation' tab with the following parameters: Tool number: 3, Length offset: 3, Diameter offset: 3, Head number: 0, SFM: 130.85733, FPT: 0.00125, and Number of flutes: 4. The right screenshot shows the 'General' tab with the following parameters: Name: Tutorial-0.375 Bull-Nosed E, Description: Tutorial-0.375 Bull-Nosed E, Manufacturer name: Mastercam Importe, Manufacturer's tool code: (empty), Tool Grade: Mastercam Default, and Milling: (empty).

16. Select **Finish**. Since you have changed the new tool with a number already assigned to a tool in the current library, a warning will appear, as shown below:



17. Click **OK** to add the similar tool. Tool libraries allow multiple tools with the same number.
18. Click **OK** in the Tool Manager to accept your changes.

19. To view a similar, correctly-defined tool, backplot the fourth operation, rotating the part to view the corrected toolpaths. The tools now cut accurately and you can see that the new parametric tool definition matches the tool profile used in Backplot, tracing the surface as expected.



Conclusion

Congratulations! You have completed the *Mastercam Tool Manager 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](http://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.*