

# Tool Manager Getting Started Guide

**April 2016**

***Mastercam***<sup>®</sup> 2017

## **Mastercam® 2017 Tool Manager GSG**

Date: April 2016

Copyright © 2016 CNC Software, Inc.— All rights reserved.

Software: Mastercam 2017

### **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 been changed or added since this document was published. The latest version of this 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.

# Contents

<b>Introduction .....</b>	<b>5</b>
▶ Tutorial Goals.....	5
<b>1. Mastercam's Tool Manager.....</b>	<b>7</b>
▶ Lesson Goals .....	7
▶ Exercise 1: Tool Manager Overview.....	8
▶ Exercise 2: Stand-Alone vs. Classic Tool Managers.....	10
▶ Exercise 3: Reasons to Use the Stand-Alone Tool Manager.....	14
▶ Exercise 4: Recommended Workflow .....	15
<b>2. Customizing the Tool Manager Workspace .....</b>	<b>17</b>
▶ Lesson Goals .....	17
▶ Exercise 1: Docking, Hiding, and Repositioning Tool Manager Panes .	17
▶ Exercise 2: Using the Functions on the View tab .....	20
▶ Exercise 3: Displaying, Sorting, and Customizing Documents.....	21
<b>3. Working with Tool Library Files (.TOOLDB) .....</b>	<b>27</b>
▶ Lesson Goals .....	27
▶ Exercise 1: Creating Tool Libraries .....	27
▶ Exercise 2: Opening and Editing a Tool Library.....	29
▶ Exercise 3: Searching for a Library Component.....	31
<b>4. Working with Components in a Tool Library.....</b>	<b>35</b>
▶ Lesson Goals .....	35
▶ Exercise 1: Creating New Tools and Holders .....	35
▶ Exercise 2: Editing Existing Tools.....	40
▶ Exercise 3: Creating and Editing Assemblies .....	42
▶ Exercise 4: Adding or Editing Materials and Cut Parameters .....	47

- ▶ Exercise 5: Accessing Cut Parameters from Mastercam ..... 50
- ▶ Exercise 6: Copying Components Between Tool Libraries ..... 53
- 5. Using Tools and Holders in Toolpaths ..... 55**
  - ▶ Lesson Goal ..... 55
  - ▶ Exercise 1: Creating New Tools and Holders ..... 55
  - ▶ Exercise 2: Editing Tools and Holders ..... 60
- Conclusion ..... 62**
  - ▶ Mastercam Resources ..... 62
  - ▶ Mastercam Documentation ..... 63
- A. Rendering Mastercam Tools ..... 65**
  - ▶ How Mastercam Renders Tool Profiles ..... 65
  - ▶ Exercise 1: Creating New Tools from Profiles ..... 67

# Introduction

Welcome to Mastercam's *Tool Manager Getting Started Guide*. This guide provides information about Mastercam's new stand-alone Tool Manager. Some of the information is in a step-by-step format, and some of the information is 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
- Learn how to customize the Tool Manager workspace
- Work with tool libraries (.TOOLDB) and their components
- Create and edit tools and holders in Mastercam's classic Tool Manager

*Estimated time to complete this tutorial: 3 hours*

## General Tutorial Requirements

All Mastercam tutorials have the following general requirements:

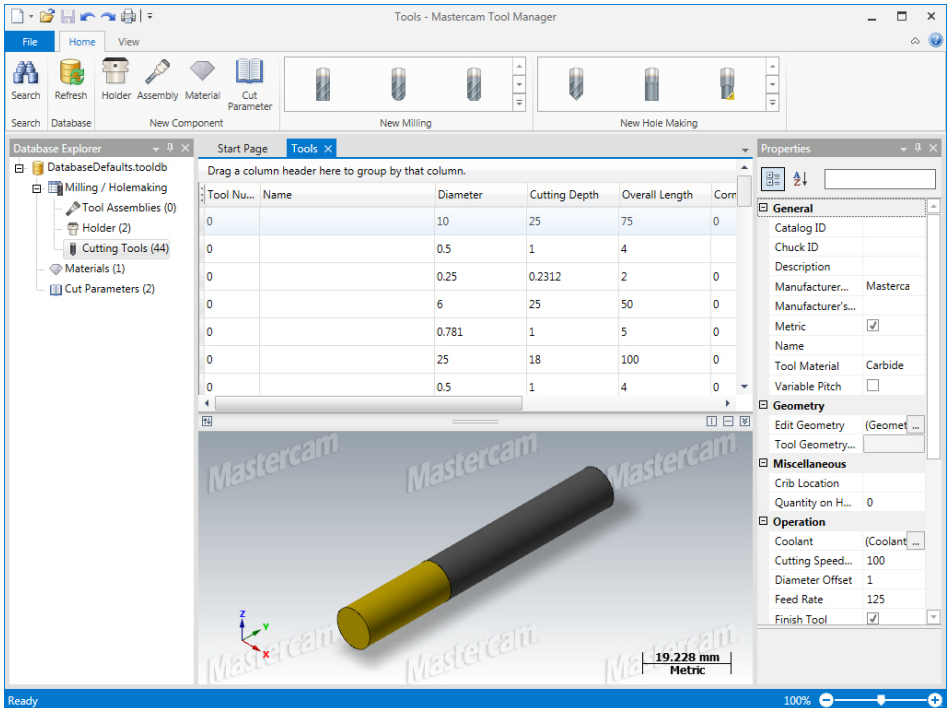
- You must be comfortable using the Windows® operating system.
- Each lesson in the tutorial builds on the mastery of preceding lesson's skills. We recommend that you complete them in order.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be accessed from the Mastercam 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](http://www.youtube.com/user/MastercamTechDocs).



# LESSON 1

## Mastercam's Tool Manager

In this lesson, you learn about the stand-alone Tool Manager interface, how it compares to the classic Mill/Router and Lathe Tool Managers in Mastercam, and the recommended general workflow.



### Lesson Goals

- Become familiar with the stand-alone Tool Manager
- Learn how the stand-alone Tool Manager differs from Mastercam's Classic Mill and Lathe Tool Managers
- Understand when and why to use it

## ***Exercise 1: Tool Manager Overview***

Mastercam's 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. You can also create your own cut parameter data and store it in the database.

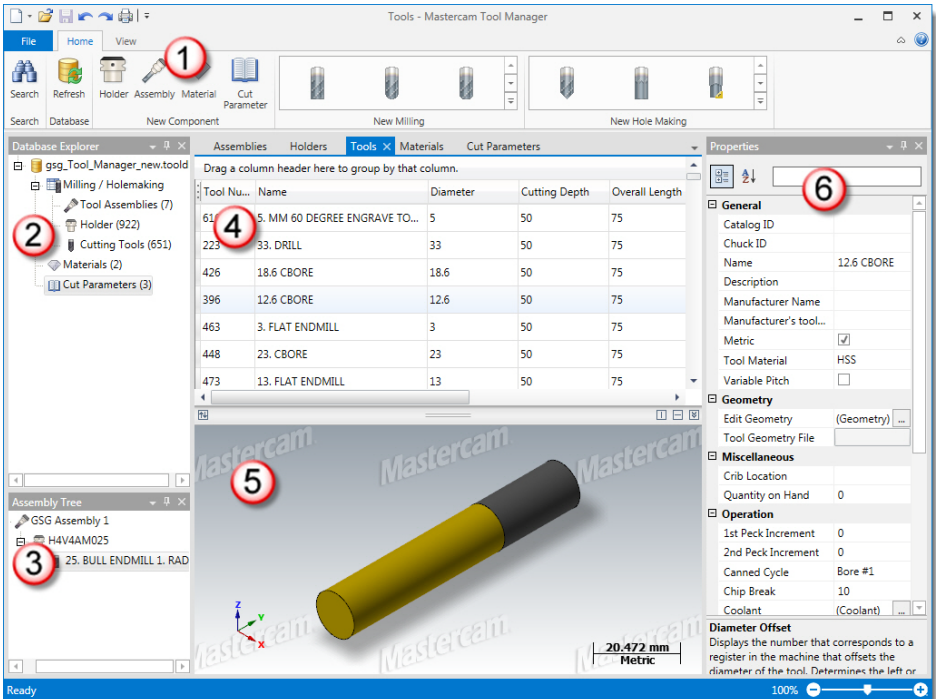
Launch the stand-alone Tool Manager separately from Mastercam by choosing **Tool Manager** from your Start menu's Mastercam folder. Certain Tool Manager functions are integrated into the Mastercam interface and are launched from within Mastercam when you create new tools, holders, and assemblies. While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

Tool Manager's database format (TOOLDB) is faster and more reliable than older database formats. Its non-proprietary format is easily accessed by third-party contributors, such as tooling manufacturers, who want to keep you up to date with the most recent tooling and tooling components. Although you can only open the TOOLDB database format in Tool Manager, in Mastercam, you can still open both the TOOLDB and TOOLS-x database formats. However, the Save As function only saves the database out in the TOOLDB format.



## Tool Manager Interface

This section contains brief descriptions of the stand-alone Tool Manager interface. These interface objects are covered in detail in the step-by-step procedure lessons of this Getting Started Guide.



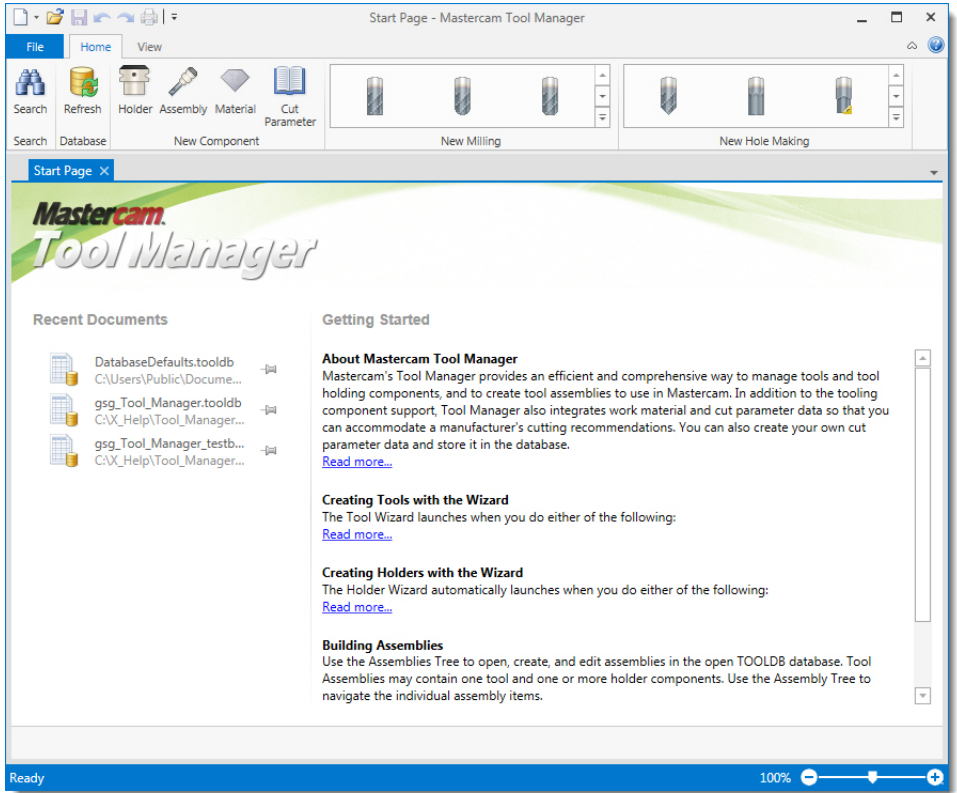
- 1. Tabs** - Tabs function somewhat like dialog boxes but look similar to toolbars. You cannot undock or dock a tab, nor are they customizable.
- 2. Database Explorer** - Use the Tool Manager Database Explorer to access content in tool database files (TOOLDB). Right-click an item to expand or collapse 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 select (double-click) Tool Assemblies in the Database Explorer or choose the **Assembly** button in 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.

## ***Exercise 2: 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.

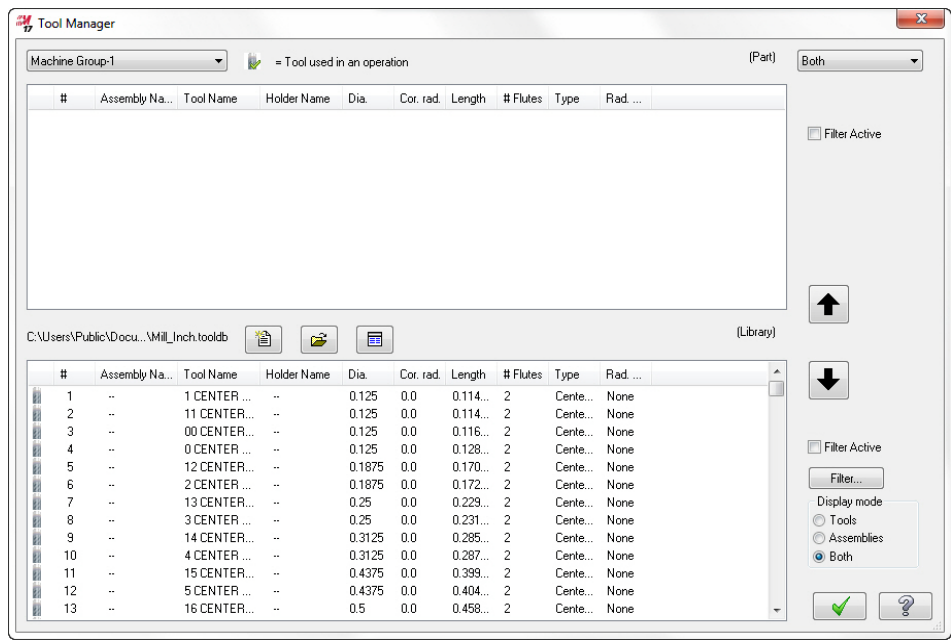
# Stand-alone Tool Manager



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.

To work in the stand-alone Tool Manager, select the Windows’ **Start** menu, and choose **Mastercam, Tool Manager**. Once you access the stand-alone Tool Manager, click the **Help** button for more details. You do not have to have Mastercam open to access and use the stand-alone Tool Manager.

# Classic Mill/Router Tool Manager

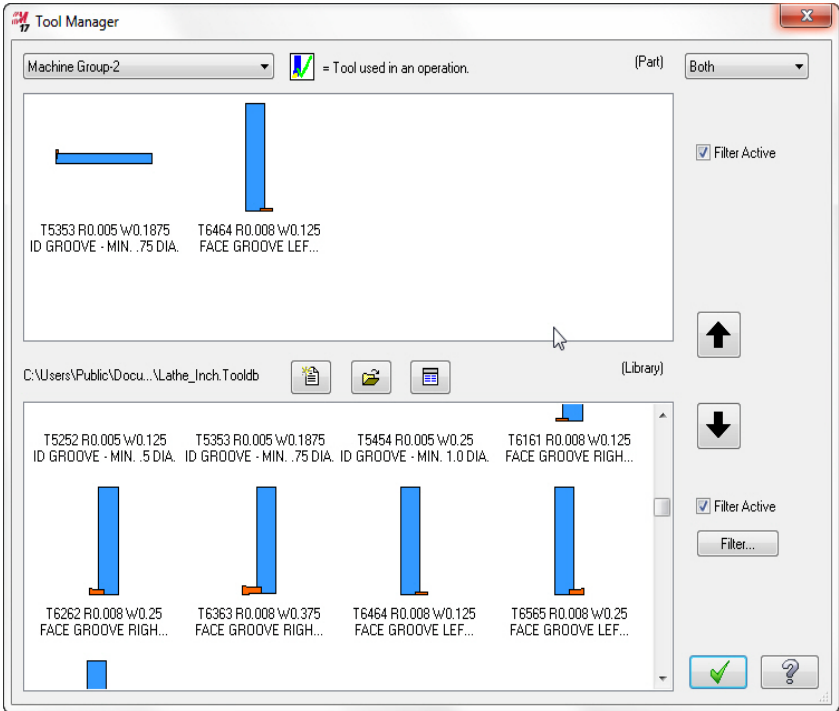


The classic Mill/Router Tool Manager allows you to view and manage tool libraries, the tools in your part file, and tool definitions. Use it to:

- 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.

Many functions of the classic Mill/Router Tool Manager interface use the integrated stand-alone Tool Manager functions such as the editing and creating wizards for tools and holders. Access the classic Tool Manager by selecting **Tool Manager** from the Mill or Router Toolpaths tab. (This option is not available in Design.)

# Classic Lathe Tool Manager



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



**NOTE:** To access the classic Lathe Tool Manager, you must have a Lathe machine definition loaded and have the insertion arrow in that machine group.

Access the Lathe Tool Manager by right-clicking in the **Toolpath parameters** tab of a lathe operation's dialog box, or selecting **Lathe Tool Manager** from the **Toolpaths, Turning** tab in the ribbon bar. (This option is not available in Design.)

## *Exercise 3: Reasons to Use the Stand-Alone Tool Manager*

There are many reasons to use the stand-alone Tool Manager instead of the classic Tool Manager. Here are some of the main reasons:

- **Work in Tool Manager 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 for library components using Search window** - Mastercam's stand-alone Tool Manager offers a Search window that shows you results that match your search criteria. You can save searches for future use as favorites, and you can view recent searches. See **Searching for a Library Component** (page 31) for details.
- **Copy tool library components between libraries** - You can have multiple instances of the stand-alone Tool Manager open. To 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. See **Copying Components Between Tool Libraries** (page 53) for details.
- **Create and edit assemblies** - Mastercam's stand-alone Tool Manager provides several ways to 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. See **Working with Components in a Tool Library** (page 35) for details.
- **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. See **Customizing the Tool Manager Workspace** (page 17) for details.
- **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.

- **Multi-Editing** - The stand-alone Tool Manager allows you to select multiple document items, such as multiple tools, and edit the properties for all selected items with the Properties Grid.

### *Exercise 4: 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.





## LESSON 2

# Customizing the Tool Manager Workspace

In this lesson, you learn how to customize, save, and reset your workspace.

### Lesson Goals

- Undock, move, and hide Tool Manager panes
- Use the window functions in the ribbon to customize your workspace
- Move, display, sort, and view document datagrids and the graphics window

### Exercise 1: Docking, Hiding, and Repositioning Tool Manager Panes

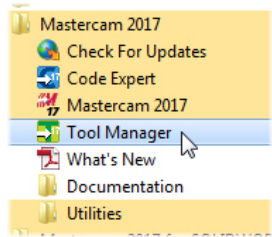
Panes in Tool Manager, such as the Database Explorer, Assembly Tree, and Properties Grid, can all be undocked, repositioned, re-docked, and hidden. For this exercise, you will use the Properties Grid for a tool as an example of how to perform these tasks.

The screenshot displays the Tool Manager workspace. On the left, the 'Tools' list shows a table of tools. The tool '23. BULL ENDMILL 3. RAD' is selected and highlighted with a red oval. On the right, the 'Properties' pane is open, showing the 'General' tab with fields for Catalog ID, Chuck ID, Description, Manufacturer Name, Manufacturer's tool code, Metric (checked), Name (23. BULL ENDMILL), Tool Material (HSS), and Variable Pitch. Below this, the 'Geometry' tab shows 'Edit Geometry' (Geometry) and 'Tool Geometry File'. The 'Miscellaneous' tab shows 'Crib Location' and 'Quantity on Hand' (0). The 'Operation' tab is also visible. At the bottom, a 3D model of a yellow and black tool is shown, with a dimension of 28.695 mm Metric displayed.

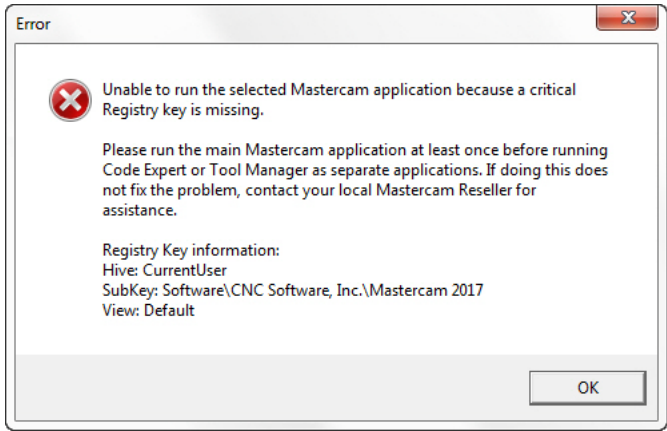
Tool Nu...	Name	Diameter	Cutting Depth
475	15. FLAT ENDMILL	15	50
249	1.10-0.25 TAP RH	1.1	50
47	1.15 DRILL	1.15	50
592	23. BULL ENDMILL 3. RAD	23	50
646	45. MM 90 DEGREE ENGRAVE TO...	45	50

28.695 mm  
Metric

- 1 Open Tool Manager from your Mastercam folder in the Windows' **Start** menu.



**IMPORTANT:** After installing Mastercam, you *must* open Mastercam before opening the stand-alone Tool Manager for the first time. Opening Mastercam builds Tool Manager support files. After opening Mastercam, you can close it and successfully open the stand-alone Tool Manager. You don't have to have Mastercam running to use 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.



- 2 Click the **Open** button and navigate to the install location of this guide.



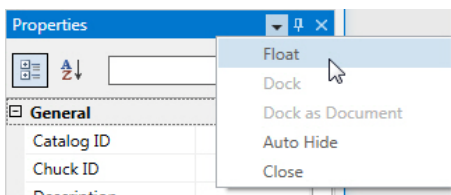
- 3 Select `gsg_Tool_Manager.tooldb` and click **Open** to load the database.

- 4 Double-click **Cutting Tools** in the **Database Explorer** to open the Tools document, and choose any tool to populate the Properties grid.
- 5 Click the **Properties** grid title bar, and drag and drop it to a position outside of the Tool Manager.

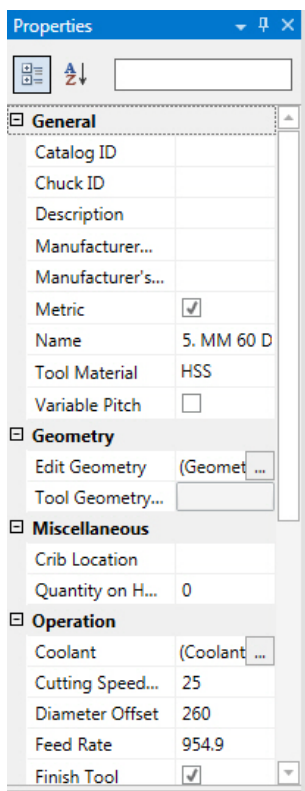


### TIPS:

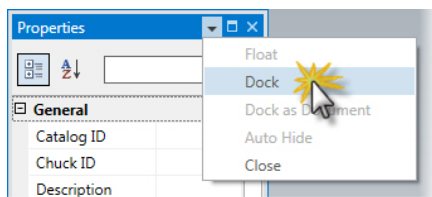
- If you use multiple monitors, you can dock the panes on another monitor, providing more work-space in Tool Manager.
- You can also use the **Float** function in the **Options** drop-down list.



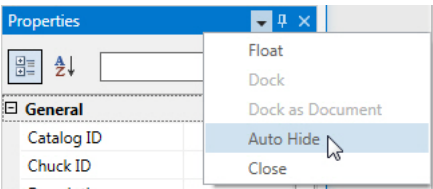
- The **Dock as Document** option is unavailable when working in the Properties grid.



- 6 To re-dock the Properties Grid to the default position in Tool Manager, click the drop-down **Options** button and choose **Dock**. You can also double-click the title bar to re-dock the grid.



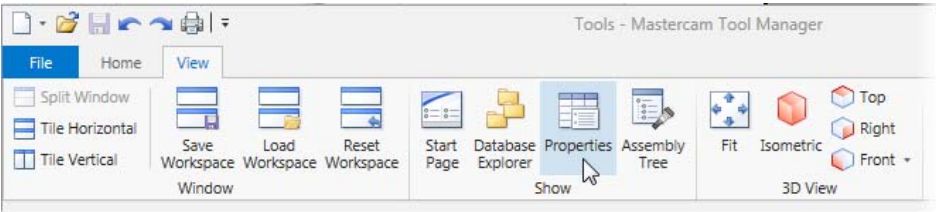
- 7 To hide the Properties Grid in Tool Manager, click the drop-down **Options** button and choose **Auto Hide**.



**NOTE:** When you hide a pane, Tool Manager docks it as a tab. Click the tab to open the Properties grid, and then click the drop-down **Options** button and choose **Dock** to re-dock it to its default state.



- 8 After closing the Properties Grid, or any other Tool Manager pane, re-open it by clicking its button in the **Show** group of the **View** tab.



## Exercise 2: Using the Functions on the View tab

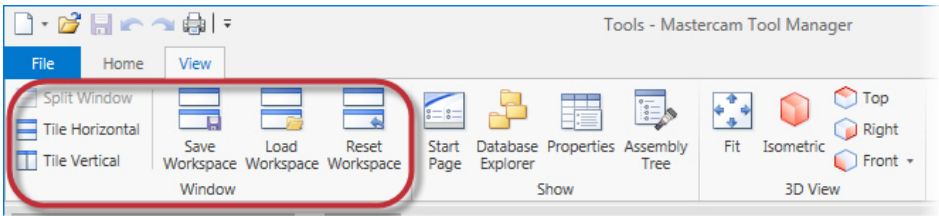
In this exercise, you learn to use the functions in the **View** tab.

- 1 If you closed Tool Manager, open it and load the `gsg_Tool_Manager.tooldb` database.
- 2 Click the **View** tab to access workspace viewing options.

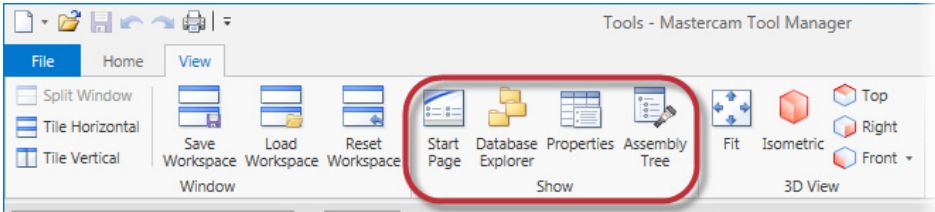


**TIP:** Hover your mouse over any of the buttons to view tooltips.

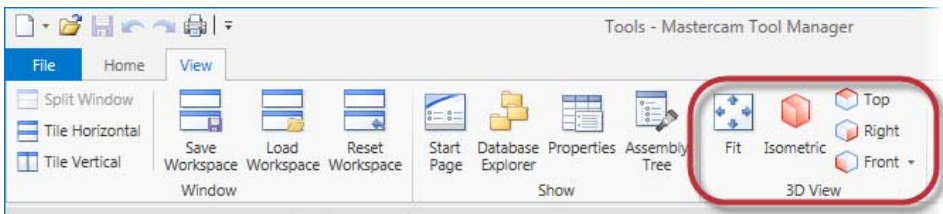
3 Use the **Window** buttons to change the arrangement of your workspace.



4 Use the **Show** buttons to display the different Tool Manager panes and grids.



5 Use the **3D View** buttons to fit and change the view of the tool, holder, or assembly in the graphics window.




---

**NOTE:** The **3D View** options are only available when you have a tool, holder, or assembly document open.

---

### *Exercise 3: Displaying, Sorting, and Customizing Documents*

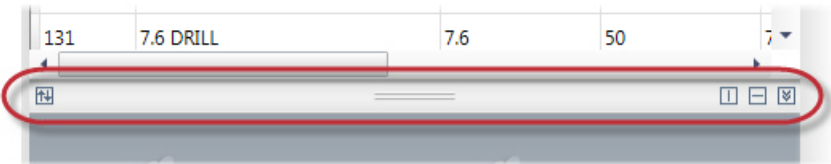
In this exercise, you learn to display, sort, and customize documents. You will also learn how to split (tile) documents, which includes customizing the graphics window's display.

## Customizing the Document Display

- 1 You should still have the Tools document open. If not, double-click **Cutting Tools** in the **Database Explorer** to open the Tools document, and choose any tool to populate the Properties grid.
- 2 Open the other document types (Assemblies, Holders, Materials, and Cut Parameters) using the same double-clicking action.
- 3 To sort a document by columns, click a column heading to sort the data by either ascending or descending order. Click the heading again to reverse the order. Click a third time in the same column header to restore the original sort order. This example shows sorting by tool number. However, you can sort by any column such as **Name**, **Diameter**, **Cutting Depth**, etc.





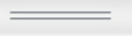
Tools X Assemblies Holders		
Drag a column header here to group by that column.		
Tool Number	Name	Diameter
1	12 Bull-Nosed Endmill	12
1	0.50 CENTER DRILL	3.15
2	FLAT ENDMILL - GSG	5
2	0.80 CENTER DRILL	3.15

- 4 To customize the documents and graphics window layout, use the buttons in the splitter control located in the display separator between the datagrid and graphics window.





## NOTES:

-  Moves the document and graphics window above or below each other.
-  Splits the document and graphics window vertically (side-by-side).
-  Splits the document and graphics window horizontally (over/under).
-  Removes the splitter and maximizes the view that is displayed on the top or left side (depending on the orientation of the split).
-  The double lines in the splitter control indicate that you can reposition the splitter by dragging it with your mouse.

## Grouping Document Data

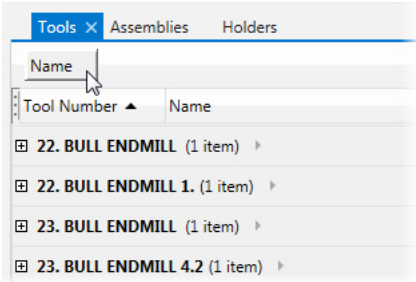
- 1 To view a datagrid by column groups, drag and drop the column header into the area just below the document tabs (as shown below).

Tools <span>×</span> Assemblies    Holders		
Drag a column header here to group by that column.		
Tool Number ▲	Name	Diameter
1	12 Bull-Nosed Endmill	12
1	0.50 CENTER DRILL	3.15
2	FLAT ENDMILL - GSG	5



**TIP:** Use the plus and minus symbols to expand and collapse grouped results.

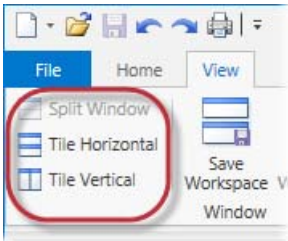
- 2 Click the button with the column name to toggle between ascending and descending the sort order. Clicking the button a third time removes sorting and restores the list to its default database order.



- 3 To ungroup your datagrid results, drag and drop the button with the column name back into the datagrid's column heading area.

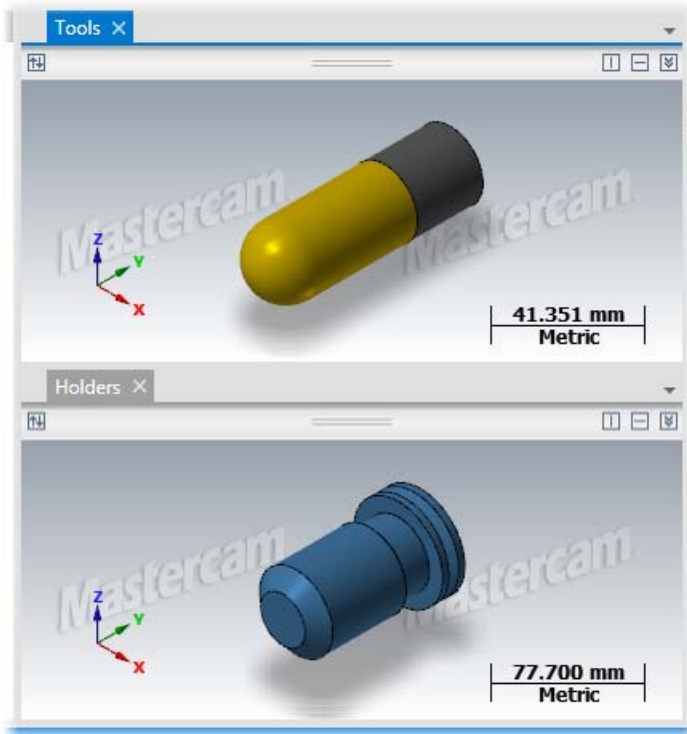
## Splitting (Tiling) Documents

Use the **View** tiling options (as shown below) to split the document workspace to view Documents concurrently.

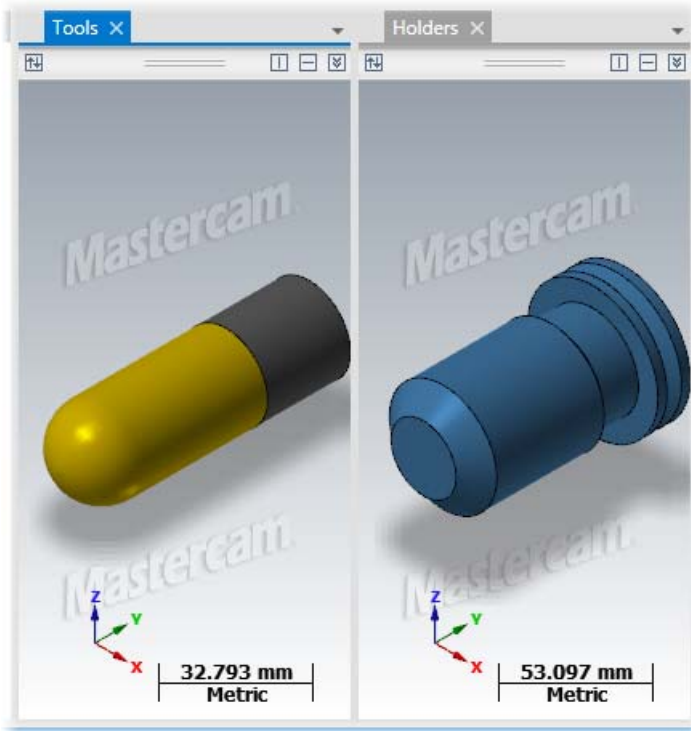




- 1 To tile documents horizontally, open the documents and click the **Tile Horizontal** button.



- 2** To tile documents vertically, open the documents and click the **Tile Vertical** button.



**NOTES:**

- To save your customized workspace, click the **Save Workspace** button in the **View** tab.
- You can have multiple saved workspaces. To load a workspace, click the **Load Workspace** button, also in the **View** tab.

- 3** Click **Reset Workspace** and close Tool Manager. Your workspace resets to its default layout when you restart Tool Manager.

## LESSON 3

# 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.



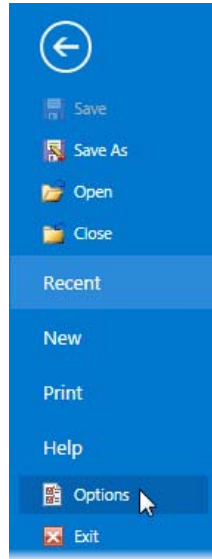
---

#### NOTES:

- The default unit setting is read from Mastercam's System Configuration. For more information see the **Start/Exit property page (System Configuration)** help topic in Mastercam.
  - 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 library (database) components are created with metric values as long as the default unit value is set to metric. For example, 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).
-

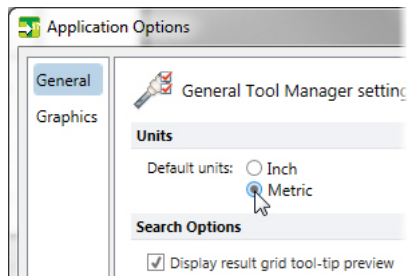
## Creating a Tool Library from an Existing Library

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



**TIP:** The Tool Manager default unit measurements are based on your Mastercam configuration. If you have Mastercam already set to metric for the unit defaults, you can skip to step 6 of this procedure.

- 4 In the **Application Options** dialog box, choose the **General** page.
- 5 If it's not already selected, click the **Metric** option under **Units**.
- 6 Click **OK**.



- 7 In the **File** tab, choose **Save As**.
- 8 Save the TOOLDB as  
gsg\_Tool\_Manager\_Tut.tooldb.



**NOTE:** You save the library with a different file name so that you don't overwrite the file provided with this guide.



## Creating a New Tool Library from Scratch

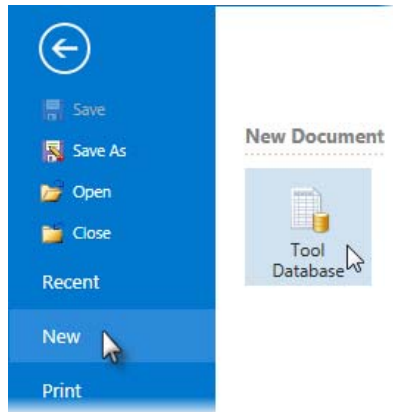
- 1 In the **File** tab (backstage view), click **New**, and then click the **Tool Database** button.

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:** You can create and add library components to the new database as detailed in **Working with Components in a Tool Library** (page 35). As you work in Tool Manager, your changes are automatically saved to the database.



- 3 Close both sessions of Tool Manager, choosing to save any changes if prompted. By closing both sessions, you ensure that all your changes are saved.

## 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.

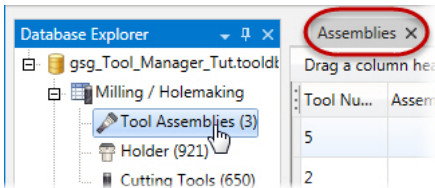
- 1 Open Tool Manager from your Windows' **Start** menu.

- 2 Click **Open** and navigate to the `gsg_Tool_Manager_Tut.tooldb` file you created in the previous exercise.



**IMPORTANT:** Do not open the `gsg_Tool_Manager.tooldb` file installed with this guide. We recommend that you leave that file unchanged in case you need to start over.

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



**TIP:** As with any document type, you can close the **Start** page by clicking the **X** in its tab. Closing the **Start** page is optional.

- 4 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. Detailed procedures are covered in the **Working with Components in a Tool Library** (page 35).



- **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.

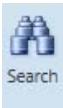


**TIP:** Single-click an item in a document to edit its parameters in the Properties Grid. Alternatively, double-click any tool or holder to open the Tool Wizard or Holder Wizard where you can edit parameters. You can also edit assemblies in the Assembly Tree. Edits you make in the Assembly Tree update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

### Exercise 3: Searching for a Library Component

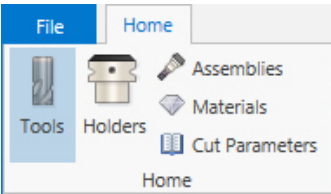
In this exercise, you search for a tool library component using the Search window.

- 1 In the Home tab, click the **Search** button. This opens the Search window.



**NOTE:** If you do not already have a database (TOOLDB) open, this button takes you to an **Open** dialog box, where you can navigate to and open a database. If you cancel the Open dialog box, the Search window still opens, and you can access a database from the **Open** button.

- 2 Select the **Tools** button in the Search window **Home** tab. You can select any button to search for a particular type of database item, but for this exercise you search for tools.



- 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 procedure, enter values as shown below.

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



NOTES:

- 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 the **[Delete]** key on your keyboard.
- To execute the search through shortcut keys, press **[Ctrl] + [Enter]** on your keyboard.

4 When you finish entering search criteria, click the **Search** button to display your search results. Your search results should be similar to those shown 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 Ra
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 clicking the **Recent** button.





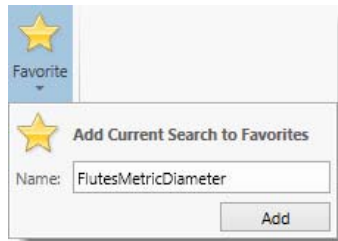
**NOTE:** If you do not enter any search criteria before executing a search, the search results display all items matching the type of search you are conducting. For example, searching for tools without entering search criteria results in a list of all tools in the current database.

**5** Save the search as a favorite for future reference as follows:

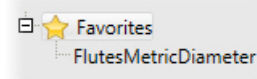
- a** Click the **Favorite** button in the toolbar.



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



- c** The name of the saved Favorite is listed in the **Favorites** folder, where you can double-click it to recall those specific search results.

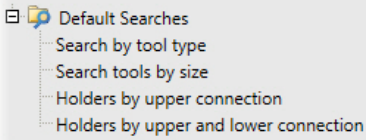


- 6** To add an item to a database already open in Tool Manager, drag the item from your Search Window and drop it into Tool Manager's Database Explorer. The item is added to the appropriate document. For example, if you drag and drop a tool into the Database Explorer, Tool Manager will always add it to the Tools document, even if you try to drop it into the Assemblies document.



---

**TIP:** Use the **Default Searches** folder in the Search Window to open common database searches installed with Mastercam. Double-click a search to open it.



- 
- 7 Click the **X** in the Search window upper right corner to close the window.

## LESSON 4

# 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 New Tools and Holders*

For this exercise, you create a bull-nosed end mill, but the procedure is similar for any milling or drilling tool type. For a different tool type, select it from the **New Milling** or **New Hole Making** gallery buttons. To expand the display of all tool type choices, use the controls shown below.



## Creating New Tools

- 1 If necessary, click the **Open** button and open the `gsg_Tool_Manager_Tut.tooldb` file.
- 2 Click the **Bull-Nosed Endmill** tool type in the **New Milling** group. This opens the Tool Wizard.



- 3** In the Tool Wizard **Define Tool Geometry** page, change the following parameters:
- Enter **85** in the **Overall length** field.
  - Enter **30** in the **Cutting length** field.
- 4** Make sure your parameters in the Define Tool Geometry page match those shown in the graphic below. The tool image updates as you make changes.

Overall dimensions

Cutting diameter: 12

Overall length: 85

Cutting length: 30

Tip / corner treatment

Radius: 3

Non-cutting geometry

Shoulder length: 30

Shoulder diameter: 12

Shank diameter: 12

- 5** Click **Next** to work in the Finalize Properties page.
- 6** In the **Finalize Properties** page, make the following changes:
- Enter **1** in the **Tool number** field.
  - Enter **21** in the **Length offset** field and **41** in the **Diameter offset** field.
- 7** Make sure your edits in the Finalize Properties page match those shown in the graphic below.



## NOTES:

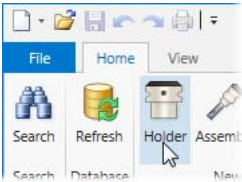
- The Speeds and Feeds Calculator is only available when the Tool Manager is accessed through Mastercam. It is unavailable in the Stand-alone version. These calculations require an active machine group with a workpiece material defined.
- To view or edit other options (Coolant button, Cutting Speed, FPT, etc.), click the **Show More Options** bar, and use the vertical scrolling bar.

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

## Creating New Holders

For this exercise, you create a new tool holder.

- 1 Click the **Holder** button in the New Component group of the Home tab. This opens the Holder Wizard.



- 2 In the Holder Wizard **Define Holder Geometry** page, change the following parameters:
  - For the **Upper, Type** drop-down field, select **CAT**.
  - For the **Upper, Size** field, type **30** and press **[Enter]**. This displays and highlights the holder that matches your entered parameters.
- 3 Make sure your parameters in the Define Holder Geometry page match those shown in the graphic below.

Connection

Upper:

TypeCAT

Size30

Lower:

TypeID

Size0

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, click the **Add Segment** button and define the segment parameters in the database grid. If you add a segment and then decide that you do not need it, click the leftmost cell for that row and press the **[Delete]** key.

- 4 Click **Next** to work in the Finalize Properties page.
- 5 In the **Finalize Properties** page, enter **GSG Main Holder** in the **Description** field and click the **Thru coolant** checkbox to set the coolant option to flow through the holder.
- 6 Make sure your edits in the Finalize Properties page match those shown in the graphic below.

General

Name: CAT 30 - ID 0

Description: GSG Main Holder

Manufacturer name: Mastercam +

Catalog ID:

☒ Metric

☒ Thru coolant

- 7 Click **Finish** to save your changes. This adds the new holder to the open library (database), and closes the Holder Wizard.

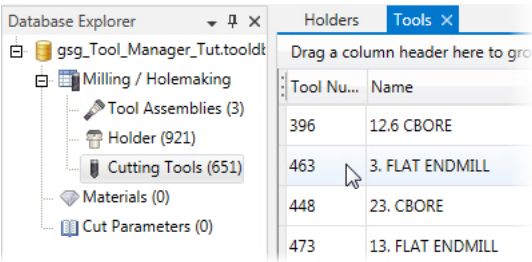
## Exercise 2: Editing Existing Tools

In this exercise, you edit a tool by accessing the Tool Wizard through the stand-alone Tool Manager. 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 through wizards, you can select the 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). This opens the tool in the Tool Wizard.

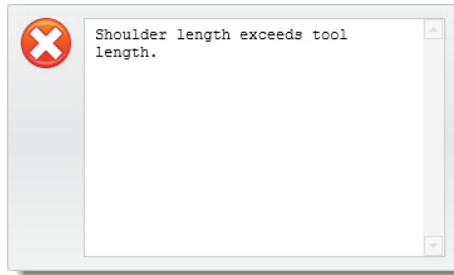


- 2 In the Tool Wizard **Define Tool Geometry** page, change the following parameters:
  - Enter **5** in the **Cutting diameter** field.
  - Enter **50** in the **Overall length** field.
  - Enter **25** in the **Cutting length** field.
  - Enter **30** in the **Shoulder length** field.





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

Overall dimensions		⬆
Cutting diameter:	<input type="text" value="5"/>	
Overall length:	<input type="text" value="50"/>	
Cutting length:	<input type="text" value="25"/>	
Tip / corner treatment		⬆
<input checked="" type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>		
Non-cutting geometry		⬆
Shoulder length:	<input type="text" value="30"/>	
Shoulder diameter:	<input type="text" value="3"/>	
Shank diameter:	<input type="text" value="3"/>	
<input checked="" type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>		

- 4 Click **Next** to work in the Finalize Properties page.

- 5 In the **Finalize Properties** page, make the following changes:
- Enter **2** in the **Tool number** field.
  - Enter **22** in the **Length offset** field and **42** in the **Diameter offset** field.
  - Enter **FLAT ENDMILL - GSG** in the **Name** field.
- 6 Make sure your edits in the Finalize Properties page match those shown in the following graphic.

Operation

Tool number: 2

Length offset: 22

Diameter offset: 42

General

Name: FLAT ENDMILL - GSG

Description:

Manufacturer name:

- 7 Click **Finish** to save your changes. This adds the edited tool to the open library (database), and closes the Tool Wizard.

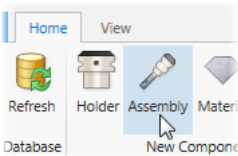
*Exercise 3: Creating and Editing Assemblies*

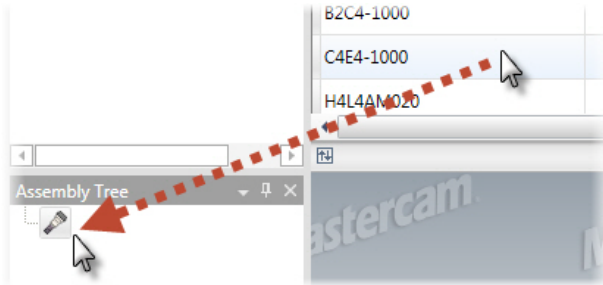
In this exercise, you create a new assembly and edit it by removing and adding assembly components.

Creating Assemblies

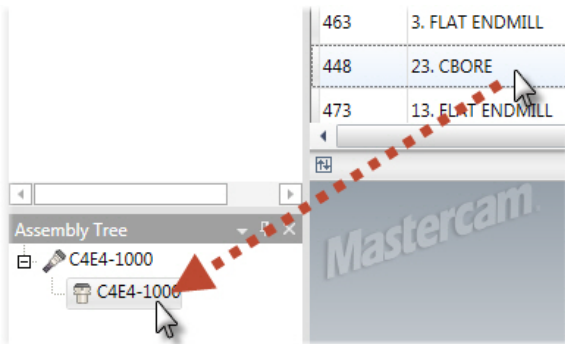
Create assemblies using a drag and drop method, as described in the following procedure. Although here you only add one holder to the assembly, you can add multiple holders as segments of a single holder using the same drag and drop method.

- 1 Click the **Assembly** button in the New Component group of the Home tab. This opens the Assembly Tree pod and the Assemblies document.
- 2 Drag and drop a **CAT-40** (name C4E4-1000) holder from the Holders document into the Assembly Tree.

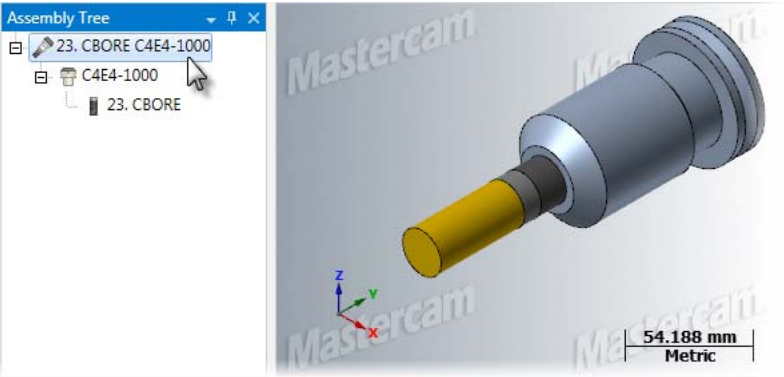




- 3 Drag and drop tool number **448** (23. CBORE) from the Tools document into the assembly.



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



## Deleting and Adding Assembly Components

Replace an assembly component by deleting it and adding a different component. The following process shows how to change a tool in an assembly. This same process applies to any component type in assemblies.

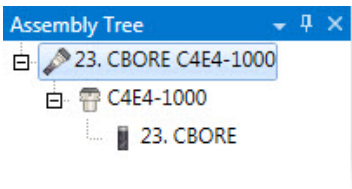
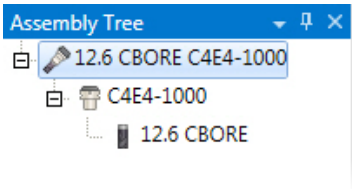
To change the tool in the assembly:

- 1 Click the **23. CBORE** tool in the assembly and press the **[Delete]** key.
- 2 Drag and drop the **12.6 CBORE** tool (tool number 396) from the Tools document onto the holder in the Assembly Tree.

Delete that tool from the assembly and add the **23. CBORE** tool (tool number 448) back into the assembly.



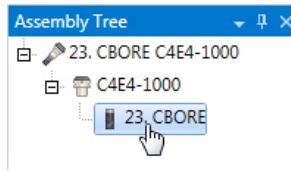
**NOTE:** As you add and remove assembly components, the name of the assembly changes.



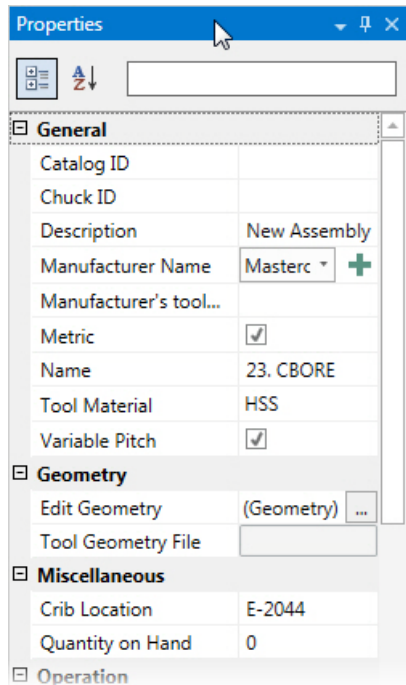
## Editing Assemblies in the Properties Grid

Edit general assembly properties or assembly components by editing the properties in the Properties Grid. Follow these steps to edit the 23. CBORE C4E4-1000 assembly you created in the previous exercise.

- 1 Click the **23. CBORE** tool in the assembly.



- 2 In the Properties Grid, make the following changes:
  - Enter **New Assembly Tool** in the **Description** field.
  - In the **Manufacturer Name** field, choose **Mastercam** from the drop-down list.
  - Select the **Variable Pitch** checkbox.
  - In the **Crib Location** field, enter **E-2044**.
- 3 Make sure your changes match the graphic shown.



### NOTES:

- Use ellipses buttons for the **Edit Geometry** and **Coolant** fields to access the Tool Wizard and Coolant dialog box to edit those properties.
- Use editable fields to change properties and parameters for the item you have selected.

Make changes to holders and assemblies in the Assembly Tree by editing properties in the Properties Grid. Changes are automatically saved to the database as you work.

For detailed information on using the Assembly Tree and the Properties Grid, click the **Help** button in Tool Manager.

## 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.

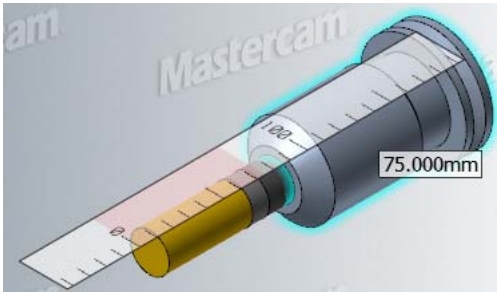
- 1 In the Assembly document datagrid, select the assembly you worked on in the previous procedure (**23. CBORE C4E4-1000**).
- 2 In the graphics window, place your cursor over the holder, hold down the **[Ctrl]** key, and click the holder. The assembly is highlighted and a dynamic ruler displays.



---

### NOTES:

- 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.



### TIPS:

- 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 4: Adding or Editing Materials and Cut Parameters*

### Adding or Editing Materials

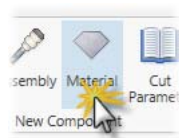
Workpiece materials are part of Mastercam Tool Manager's TOOLDB database files. They display in the Materials document in Tool Manager. Although you can edit some of the materials' properties in the Properties Grid, the Materials document is primarily used for reference information when working with 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 If necessary, open the Tool Manager from the **Start** menu and load the TOOLDB you have been working with.

- 2 Click the **Material** button. This opens the Materials document, and adds a new row to the datagrid.



- 3 Select the first row in the Materials document and set the material parameters in the Properties Grid as described below:

- In the **Name** field, enter **Aluminum**.
- In the **Hardness** field, enter **60**.
- In the **Hardness Units** drop-down, choose **Rockwell B Hardness**.
- In the **ISO Group Value** drop-down, choose **N-Aluminum, Copper, and Non-Metallic**.

General	
Description	
Name	Aluminum
Material	
Density	0
Hardness	60
Hardness Units	Rockwell B Hardness
ISO Group Value	N - Aluminum, Copper, and Non-Metallic

- 4 To edit the materials properties for that same material, change the following values in the Properties Grid as described below:

- In the **Name** field, enter **Aluminum - 6061**.
- In the **Hardness** field, enter **95**.
- In the **Hardness Units** drop-down, choose **Brinell Hardness Scale**.

General	
Description	
Name	Aluminum - 6061
Material	
Density	0
Hardness	95
Hardness Units	Brinell Hardness Scale
ISO Group Value	N - Aluminum, Copper, and Non-Metallic



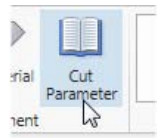
**NOTE:** Remember that as you work in Tool Manager, including the Properties Grid, your changes are automatically saved to the database.



## Adding or Editing Cut Parameters

Cut parameters are part of Mastercam Tool Manager's TOOLDB database files. They display in the Cut Parameters document in Tool Manager, 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 Click the **Cut Parameter** button.  
This opens the Cut Parameters document, and adds a new row to the datagrid.



- 2 Select the first row in the Cut Parameters document and set the material parameters in the Properties Grid as described below:

- In the **Material Name** field, select **Aluminum - 6061**.
- Make sure the **Metric** checkbox is checked.
- In the **Name** field, enter **Shouldering Aluminum with 16mm Endmill**.
- In the **Operation Type** field, select **Shouldering**.
- In the **Feed Per Tooth** field, enter **0.075**.
- In the **Axial Depth of Cut** fields (both Finish and Rough), enter **10**.
- In the **Radial Depth of Cut** fields (both Finish and Rough), enter **8**.
- In the **Surface Speed** field, enter **90**.
- In the **Maximum Diameter** field, enter **18**.
- In the **Minimum Diameter** field, enter **16**.

Make sure your values match those shown in the graphic.

A screenshot of the 'Properties' dialog box in Mastercam. The 'General' tab is selected, showing fields for Cut Quality (Rough), Cut Source (User), Description, Machine Name, Material Name (Aluminum - 6061), Metric (checked), Name (Shouldering Aluminum with), Operation Type (Shouldering), and Proven (unchecked). The 'Manufacturers Data' section shows Manufacturer Name (Mastercam). The 'Operation' section shows Estimated Tool Life (min...), Feed Per Tooth (0.075), Finish Axial Depth of Cut (10), Finish Radial Depth of Cut (8), Rough Axial Depth of Cut (10), Rough Radial Depth of Cut (8), and Surface Speed (90). The 'Tool' section shows Maximum Diameter (18), Minimum Diameter (16), and Tool Grade (Mastercam Default).

General	
Cut Quality	Rough
Cut Source	User
Description	
Machine Name	
Material Name	Aluminum - 6061
Metric	<input checked="" type="checkbox"/>
Name	Shouldering Aluminum with
Operation Type	Shouldering
Proven	<input type="checkbox"/>

Manufacturers Data	
Manufacturer Name	Mastercam

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

Tool	
Maximum Diameter	18
Minimum Diameter	16
Tool Grade	Mastercam Default



---

**NOTES:**

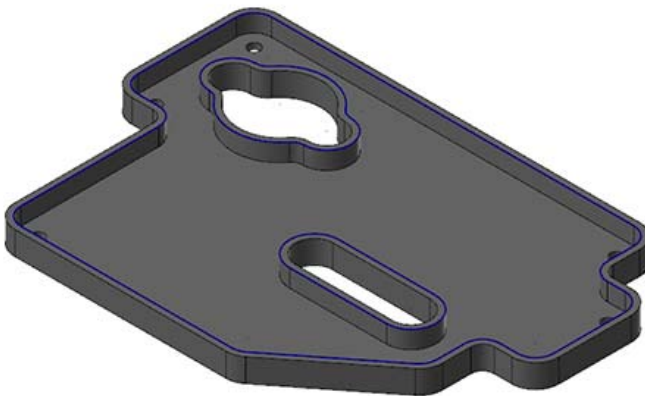
- Some of the fields are automatically populated and non-editable because that information comes from the Material you choose.
  - If you are creating a new database or if the desired workpiece material does not exist, you need to create the material before creating the cut parameters.
  - Although you do not edit these values again for this guide, you can edit the cut parameters in the Properties Grid as you did for the materials in the **Adding or Editing Materials** section.
- 

- 3 Leave the stand-alone Tool Manager open.

### *Exercise 5: Accessing Cut Parameters from Mastercam*

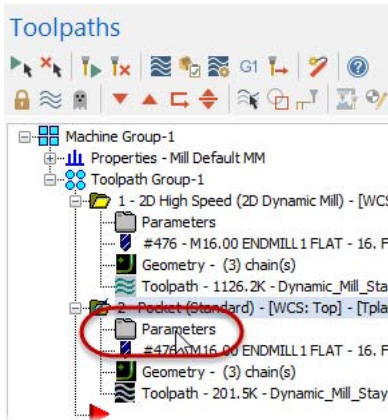
In this exercise, you apply the cut parameters you created to a Mastercam toolpath to update the surface speed and feed per tooth values.

- 1 Start Mastercam.
- 2 Open the part file `Dynamic_Mill_Rest.mcam`, provided with this guide.

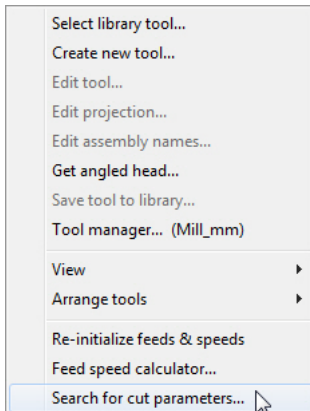


- 3 Choose **File, Save As**, and save the part under a different file name. This protects the original part file from being overwritten.

- 4 Click the **Parameters** icon for the second operation in the Toolpaths Manager.



- 5 In the Tool page, right-click and choose **Search for cut parameters**.



This opens the Search window.





- 6 Click the **File** tab in the Search window and open the `gsg_Tool_Manager_Tut.tooldb` file containing the cut parameters you created.

- 7 Make sure the search criteria is set as shown below, and click the **Search** button.

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

Your search results will display the cut parameters you created in the previous exercise.

- 8 Select the resulting cut parameters and click the **Accept** button.



Search Accept Recent Favorite

Actions Searches

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

Name	Description	Material
Shouldering Aluminum with 16mm Endmill		N - Aluminum

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

- 9 Click **OK** in the operations dialog, regenerate the toolpath, save the part file, and close Mastercam.

## Exercise 6: Copying Components Between Tool Libraries

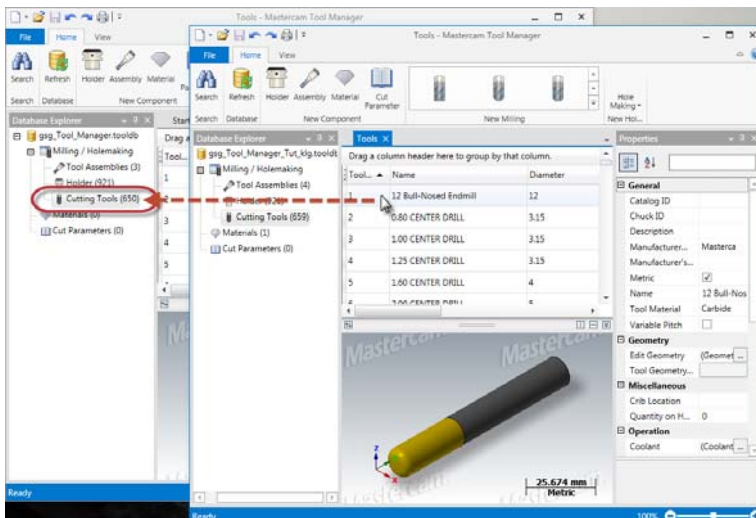
Use drag and drop to copy items from one database to another. To do this, you must have two or more sessions of Tool Manager open concurrently.

- 1 Start another session of Tool Manager through the Windows' **Start** menu.



**NOTE:** You should still have the `gsg_Tool_Manager_Tut.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 `gsg_Tool_Manager.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 `gsg_Tool_Manager_Tut.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 `gsg_Tool_Manager.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.





---

## **NOTES:**

- 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 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.

## LESSON 5

# 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*

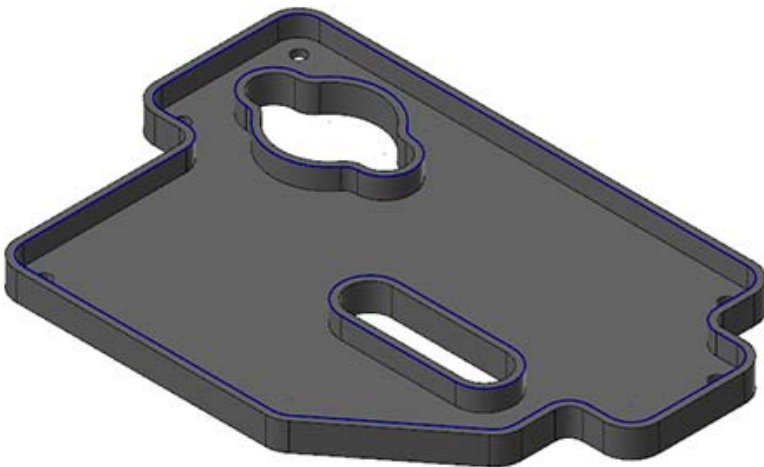
- Create and edit tools and holders using classic Tool Manager

### *Exercise 1: Creating New Tools and Holders*

In this exercise, you create a new tool and holder from within Mastercam. These tooling components are applied to a dynamic mill toolpath on the part supplied with this guide.

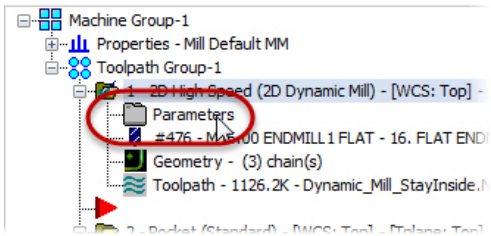
### Creating a New Tool

- 1 Start Mastercam.
- 2 Open the part file `Dynamic_Mill_Rest.mcam`, provided with this guide.

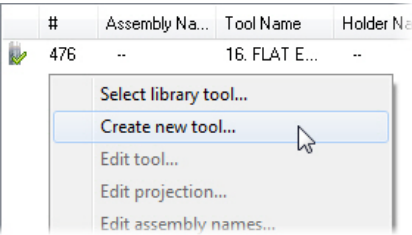


- 3 Choose **File, Save As**, and save the part under a different file name. This protects the original part file from being overwritten.

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



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



6 In the Select Tool Type page, select **End Mill**, and click **Next**.

7 In the Define Tool Geometry page set the following tool parameters:

- Enter **18** in the **Cutting diameter** field.
- Enter **90** in the **Overall length** field.
- Enter **60** in the **Cutting length** field.
- Make sure the **Shoulder length** field has a value of 60.
- Make sure the **Shoulder diameter** field has a value of 18.
- Make sure the **Shank diameter** field has a value of 18.

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



The screenshot shows the Tool Wizard dialog box with the following settings:

- Overall dimensions:**
  - Cutting diameter: 18
  - Overall length: 90
  - Cutting length: 60
- Tip / corner treatment:**
  - Four icons for corner treatment: the first is selected (blue), and the others show different corner radii.
- Non-cutting geometry:**
  - Shoulder length: 60
  - Shoulder diameter: 18
  - Shank diameter: 18
  - Four icons for shank geometry: the first is selected (blue), and the others show different shank profiles.

- 9 Click **Next** to work in the Finalize Properties page.
- 10 In the **Finalize Properties** page, make the following changes:
  - Enter **18 Flat Endmill - GSG** in the **Name** field.
  - Enter **New Tool for GSG** in the **Description** field.
- 11 Make sure your parameters in the Finalize Properties page match those shown in the graphic below.

The screenshot shows the General tab of the Finalize Properties dialog box with the following settings:

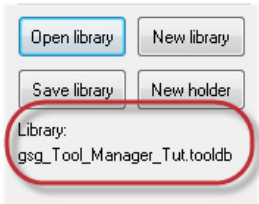
- Name:** 18 Flat Endmill - GSG
- Description:** New Tool for GSG
- Manufacturer name:** Mastercam Importer +
- Manufacturer's tool code:** (empty field)
- Tool Grade:** Mastercam Default +

- 12 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.

- 13 In the Holder page of the Dynamic Mill dialog box, choose the **Open library** button, navigate to your `gsg_Tool_Manager_Tut.tooldb`, and open that library.



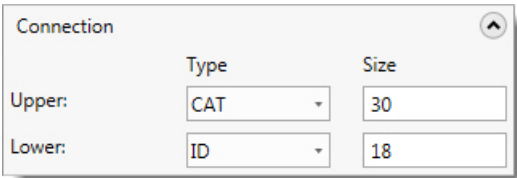
**NOTE:** Text in the window indicates the current library.



- 14 Choose the **New holder** button. This opens the Holder Wizard.



- 15 In the Define Holder Geometry page, set the following tool parameters:
- Choose **CAT** from the **Upper connection Type** drop-down field.
  - Enter **30** in the **Upper connection Size** field and press **[Enter]**.
  - Enter **18** in the **Lower connection Size** field.
- 16 Make sure your parameters in the Define Holder Geometry page match those shown in the graphic below.



- 17 Click **Next** to work in the Finalize Properties page.
- 18 In the **Finalize Properties** page, make the following changes:
- Enter **New Holder for GSG** in the **Description** field.
  - Check the **Thru coolant** checkbox to set the coolant to flow through the holder.
- 19 Make sure your parameters in the Finalize Properties page match those shown in the following graphic.

General

Name: CAT 30 - ID 18

Description: New Holder for GSG

Manufacturer name: Mastercam Importe +

Catalog ID:

☒ Metric

☒ Thru coolant

- 20** 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.



**NOTE:** 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.

CAT 30 - ID 18\*

B2C3-0018

B2C3-0020

B2C3-0020

B2C3-0032

B2C3-0032

B2C4-0011

B2C4-0016

B2C4-0020

B2C4-0032

B2C4-1000

B2E4-0125

B2E4-0187

B2E4-0250

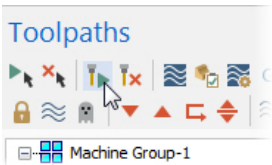
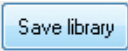
Open library New library

Save library New holder

Library:  
gsg\_Tool\_Manager\_tut.tooldb

\* = Indicates holder is not saved in current library

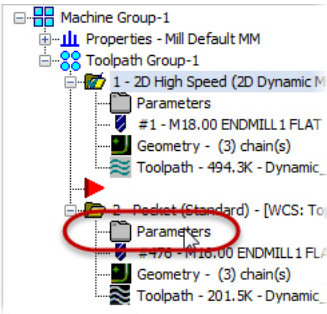
- 21 Click the **Save library** button to open a dialog box, navigate to the .TOOLDB location, and save the new holder to your library.
- 22 Click **OK** in the Dynamic Mill dialog box. This marks the toolpath dirty.
- 23 Click the **Regenerate all selected operations** button to update the toolpath.



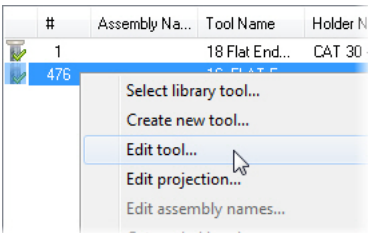
*Exercise 2: Editing Tools and Holders*

In this exercise, you edit a tool and holder.

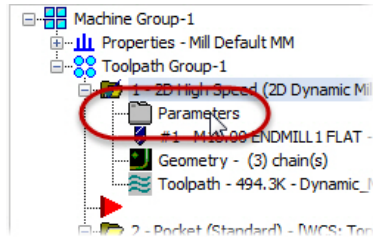
- 1 Click the **Parameters** icon for the second operation in the Toolpaths Manager, and go to the **Tool** page.



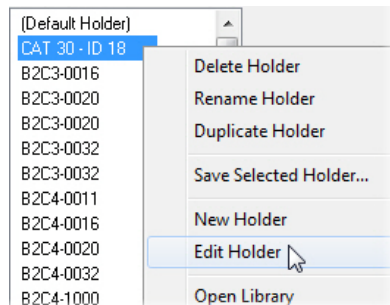
- 2 Right-click tool **#476** and choose **Edit tool**. This opens the Tool Wizard.



- 3 In the Define Tool Geometry page, change the **Shank diameter** to **22**.
- 4 Click **Finish** to save your changes. This closes the Tool Wizard, and returns you to the Tool page.
- 5 Click **OK** in the Pocket dialog box, and regenerate the operation.
- 6 Click the **Parameters** icon for the first operation in the Toolpaths Manager.



- 7 Go to the **Holder** page, right-click the **CAT 30 - ID 18** holder and choose **Edit Holder**. This opens the Holder Wizard. Your list may not look exactly like the graphic shown.



- 8 In the Define Holder Geometry page, click **Next** to leave settings as is.
- 9 In the Finalize Properties page, change the **Name** to **CAT 30 - ID 18 - GSG**.
- 10 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.
- 11 Save your changes to the library as described in the previous exercise.
- 12 Click **OK** in the **Dynamic Mill** dialog box.
- 13 Save the part file.

# Conclusion

Congratulations! You have completed the *Tool Manager Getting Started Guide* 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. The Mastercam tutorial series is in continual development, and we will add modules as we complete them.

## Mastercam Resources

Enhance your Mastercam experience by using the following resources:

- *Mastercam Help*— Also, most dialog boxes, function panels, and ribbon bars feature a Help button that opens Mastercam Help directly to related information.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—CNC Software's Technical Support department (860-875-5006 or [support@mastercam.com](mailto:support@mastercam.com)) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Mastercam Tutorials*—CNC offers a series of tutorials to help registered users become familiar with basic Mastercam features and functions. The Mastercam tutorial series is in continual development, with new modules added as we complete them.
- *Mastercam University*—CNC Software sponsors Mastercam University, an affordable online learning platform that gives you 24/7 access to Mastercam training materials. Take advantage of more than 180 videos to master your skills at your own pace and help prepare yourself for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit [www.mastercamu.com](http://www.mastercamu.com), or email [training@mastercam.com](mailto:training@mastercam.com).
- *Online communities*— You can find a wealth of information, including many videos, at [www.mastercam.com](http://www.mastercam.com). For tech tips and the latest Mastercam news, follow us on Facebook ([www.facebook.com/mastercam](http://www.facebook.com/mastercam)), Twitter ([www.twitter.com/mastercam](http://www.twitter.com/mastercam)), or Google+ ([plus.google.com/+mastercam](https://plus.google.com/+mastercam)). Visit our YouTube channel to see Mastercam in action ([www.youtube.com/user/MastercamCadCam](http://www.youtube.com/user/MastercamCadCam))!

Registered users can search for information or ask questions on the Mastercam Web forum, [forum.mastercam.com](http://forum.mastercam.com), or use the knowledge base at [kb.mastercam.com](http://kb.mastercam.com).

## Mastercam Documentation

Mastercam installs the following documents in the \Documentation folder of your Mastercam installation:

- *What's New in Mastercam 2017*
- *Mastercam 2017 Installation Guide*
- *Mastercam 2017 Administrator Guide*
- *Mastercam 2017 Transition Guide*
- *Mastercam 2017 Quick Reference Card*
- *Mastercam 2017 ReadMe*

## Contact Us

For questions about this or other Mastercam documentation, contact the Technical Documentation department by email at [techdocs@mastercam.com](mailto:techdocs@mastercam.com).







## APPENDIX A

# 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.

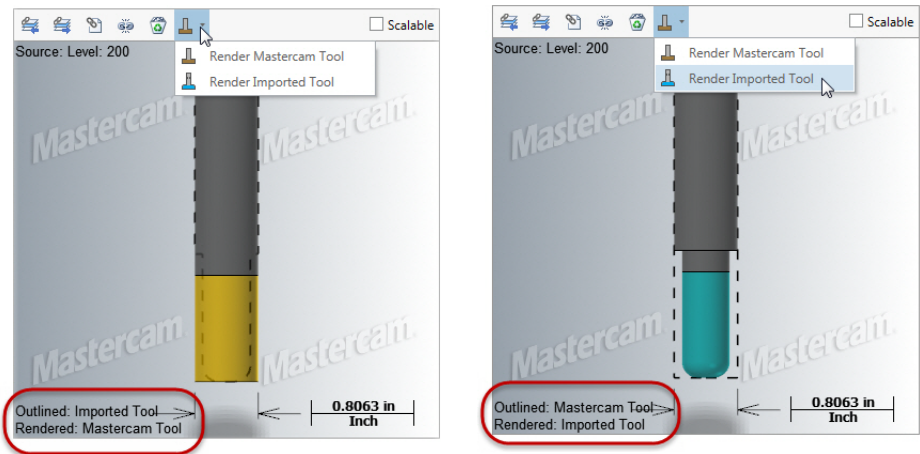
The Tool Wizard renders the custom profile as a 3D shaded tool and overlays the parametric profile using a dotted line. You can toggle this behavior to render the parametric tool and overlay the custom profile by selecting the desired option from the **Render Mastercam Tool** list.

### *How Mastercam Renders Tool Profiles*

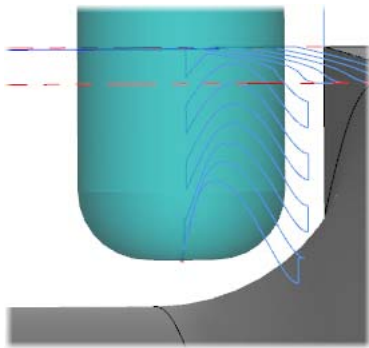
When you use a tool profile to create a new tool, whether you import the profile from an external file (DXF) or from a profile stored in your part file, if you choose to create the tool based on the Custom Tool type, Mastercam tries to approximate its parameters. Frequently, Mastercam fails to populate 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 bull endmill tool from a profile stored on level 200 in the `Custom_tool_Testing.mcam` 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_Testing.mcam` 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 tool-paths 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 guide.



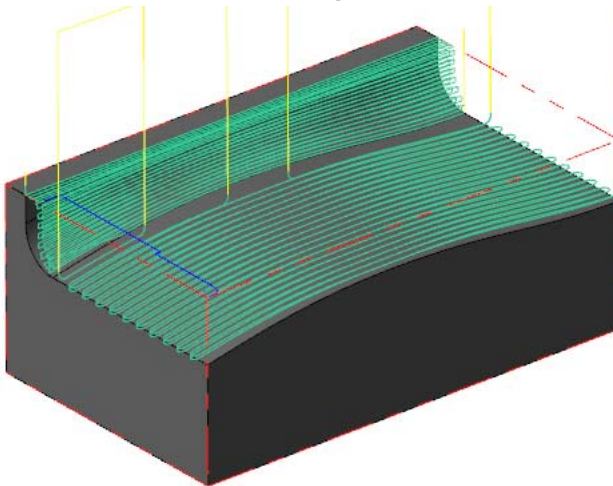

---

**NOTE:** This part is in English, based on the default inch Mill machine definition.

---

## Creating a New Tool from a Profile

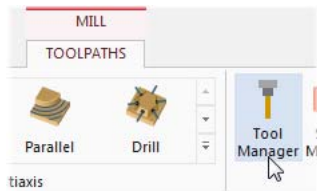
- 1 Start Mastercam.
- 2 Open the part file `Custom_tool_Testing.mcam`, provided with this guide.
- 3 Right-click in the graphics window and choose **Fit** to fit the part to the screen. Your part should look similar to the graphic below.



4 Choose **File, Save As**, and save the part as Custom\_tool\_Testing\_Tut.mcam. This protects the original part file from being overwritten.

5 Choose **Toolpaths, Tool Manager**.

This will open the Tool Manager where you will be creating a custom tool similar to the 0.375 Bull-Nosed Endmill tool used in operation 4.



6 In the Tool Manager, right-click in the Tool list window (upper window) and choose **Create new tool**. This opens the Tool Wizard.

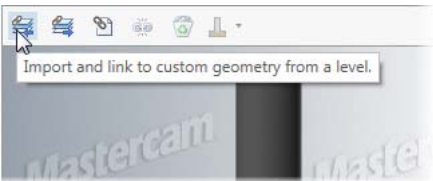
7 Choose the **Bull Mill** tool type because the custom tool profile is created as a bull-nosed endmill.



8 Click **Next** to advance to the Define Tool Geometry page.

9 Click **Import and link to custom geometry from a level**.

This opens the Select Level dialog box.

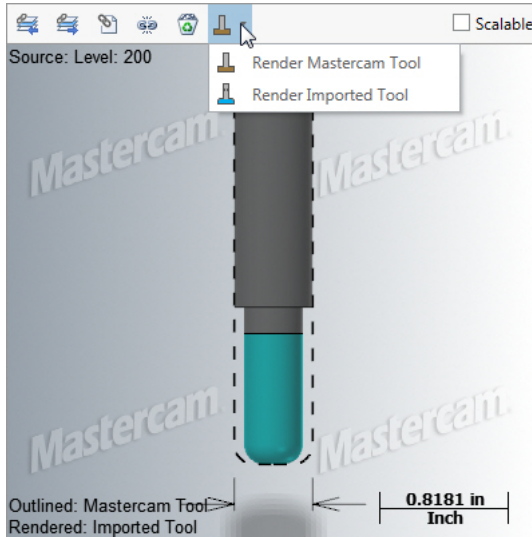


10 Select **Bull mill shoulder** on level 200 and click **OK**.

A screenshot of the 'Select Level' dialog box in Mastercam. It contains a table with three columns: 'Number', 'Level Name', and '# Entities'. The row for level 200, 'Bull mill shoulder', is highlighted with a mouse cursor.

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

- 11** Toggle between viewing the custom tool as a rendered Mastercam Tool or a rendered 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:

- Enter 0.375 in the **Cutting diameter** field.
- Enter 2.5 in the **Overall length** field.
- Make sure the **Cutting length** field has a value of 0.83333.
- Enter 0.125 in the **Radius** field.
- Enter 1 in the **Shoulder length** field.
- Enter 0.375 in the **Shoulder diameter** field.
- Enter 0.5 in the **Shank diameter** field.

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

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

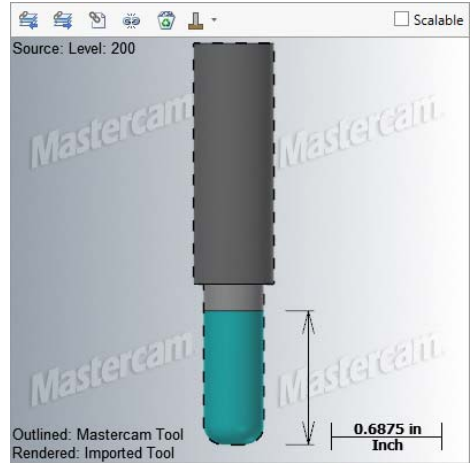
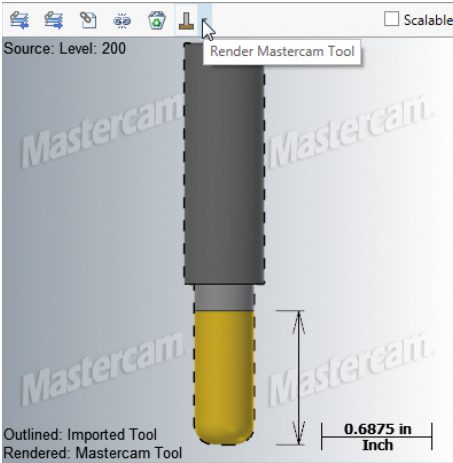
Shank diameter: 0.5



**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** Click **Next** and edit the parameters as shown below on the Finalize miscellaneous properties page, renumbering the tool as needed.

Finalize miscellaneous properties.

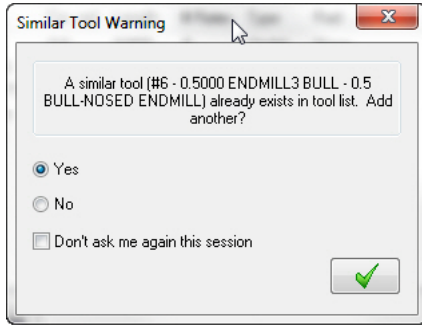
Adjust any miscellaneous properties before finalizing tool creation.

Operation	
Tool number:	3
Length offset:	3
Diameter offset:	3
Head number:	0
SFM:	130.85733
FPT:	0.00125
Number of flutes:	4
Feed rate:	6.665
Plunge rate:	25
Retract rate:	50
Spindle speed:	1333
Spindle direction:	Clockwise
Material:	Carbide

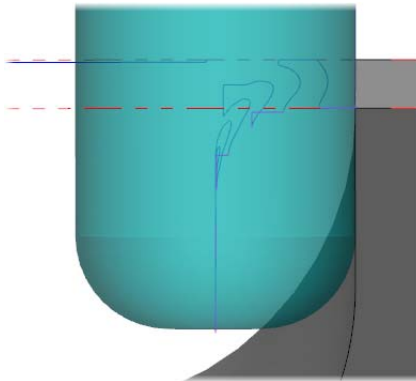
General	
Name:	Tutorial-0.375 Bull-Nosed E
Description:	Tutorial-0.375 Bull-Nosed E
Manufacturer name:	Mastercam Importe
Manufacturer's tool code:	
Tool Grade:	Mastercam Default
Milling	
<input checked="" type="checkbox"/> Rough Tool	
<input checked="" type="checkbox"/> Finish Tool	
Rough XY step (%):	0
Rough Z step (%):	0
Finish XY step (%):	0
Finish Z step (%):	0

- 16** Click **Finish** when done.

- 17** If you have numbered the new tool with a number already assigned to a tool in the current library, a similar tool warning will appear (as shown below).



- 18** Click **OK** to add the similar tool. Tool libraries allow multiple tools with the same number.
- 19** Click **OK** in the Tool Manager to accept your changes.
- 20** To view a similar, correctly-defined tool, backplot the fourth operation in the `Custom_tool_Testing_Tut.mcam` file, 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.







**Attention! Updates may be available.  
Go to [Mastercam.com/Support](http://Mastercam.com/Support) for the latest downloads.**

***cnr software, inc.***

671 Old Post Road  
Tolland, CT 06084 USA  
[www.mastercam.com](http://www.mastercam.com)