

Mastercam 2022



MULTIAXIS ADVANCED

TRAINING TUTORIAL SERIES | IMPERIAL



2022

HLE DOWNLOAD
INSTRUCTIONS INSIDE

Mastercam 2022

Multiaxis Advanced Training Tutorial

To order more books:

Call 1-800-529-5517 or

Visit www.emastercam.com or

Contact your Mastercam dealer



The graphic features a large white downward-pointing arrow with a red outline. Inside the arrow, the Mastercam logo is at the top, followed by the text 'Home Learning Edition Demo Software' and 'DOWNLOAD' at the bottom.

Mastercam
CAD/CAM Systems

Home Learning Edition
Demo Software
DOWNLOAD

HOW TO DOWNLOAD:

Mastercam Home Learning Edition (HLE)
Demo Software

Visit <https://signup.mastercam.com/try-mastercam>

Proceed to download!

Mastercam 2022 Training Tutorial - Multiaxis Advanced

Copyright: 1998 - 2022 In-House Solutions Inc. All rights reserved

Software: Mastercam 2022

Authors: Mariana Lendel

ISBN: 978-1-77146-947-0

Date: September 27, 2021

Notice

In-House Solutions Inc. reserves the right to make improvements to this manual at any time and without notice.

Disclaimer Of All Warranties And Liability

In-House Solutions Inc. makes no warranties, either express or implied, with respect to this manual or with respect to the software described in this manual, its quality, performance, merchantability, or fitness for any particular purpose. In-House Solutions Inc. manual is sold or licensed "as is." The entire risk as to its quality and performance is with the buyer. Should the manual prove defective following its purchase, the buyer (and not In-House Solutions Inc., its distributor, or its retailer) assumes the entire cost of all necessary servicing, repair, of correction and any incidental or consequential damages. In no event will In-House Solutions Inc. be liable for direct, indirect, or consequential damages resulting from any defect in the manual, even if In-House Solutions Inc. has been advised of the possibility of such damages. Some jurisdictions do not allow the exclusion or limitation of implied warranties or liability for incidental or consequential damages, so the above limitation or exclusion may not apply to you.

Copyrights

This manual is protected under International copyright laws. All rights are reserved. This document may not, in whole or part, be copied, photographed, reproduced, translated or reduced to any electronic medium or machine readable form without prior consent, in writing, from In-House Solutions Inc.


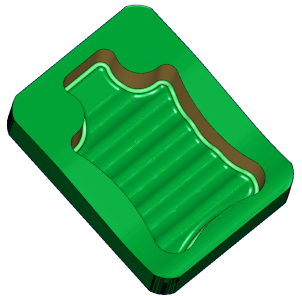
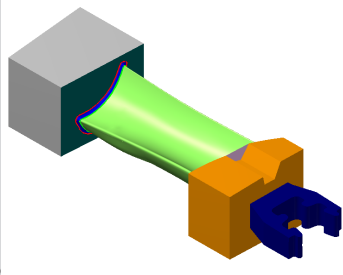
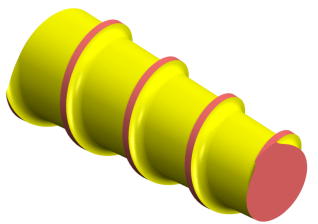
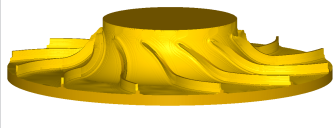
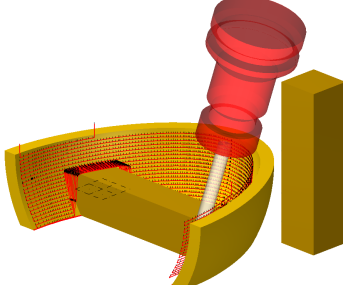
Trademarks

Mastercam is a registered trademark of CNC Software, Inc.

Microsoft, the Microsoft logo, are registered trademarks of Microsoft Corporation;

Windows 10 is a registered trademarks of Microsoft Corporation.

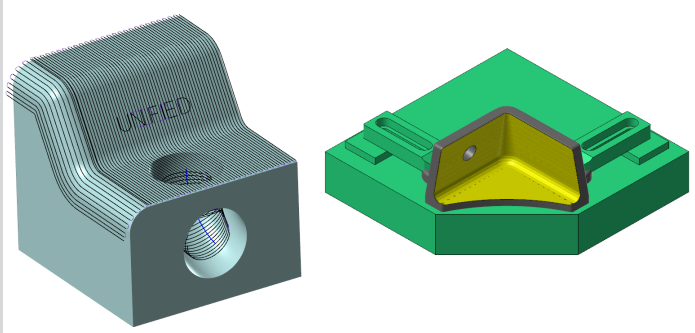
Multiaxis Advanced Projects

Tutorial	Toolpath Creation
<p>#1</p> 	<p>Project Curve Multiaxis toolpath.</p>
<p>#2</p> 	<p>Swarf Milling toolpath. Parallel toolpath with To Curve option. Machine Simulation.</p>
<p>#3</p> 	<p>Parallel toolpath with Angle option. Morph toolpath with From Curves option. Parallel toolpath with To Curve option.</p>
<p>#4</p> 	<p>Rotary Advanced toolpath. Morph (Curves) toolpath. Parallel (To Surface) toolpath. Parallel (To Curves) toolpath.</p>
<p>#5</p> 	<p>Morph with Between Two Surfaces option. Parallel (To Surface) toolpath. Parallel (To Curves) toolpath. Transform Rotate toolpaths to machine the entire impeller.</p>
<p>#6</p> 	<p>Collision Control strategies: Tilt tool away using a side tilt angle. Retract tool along tool axis. Move tool away with retract on +Z.</p>

Tutorial

Toolpath Creation

#7



Unified toolpath.

Table Of Content

Multiaxis Advanced Training Tutorial	1
Multiaxis Advanced Projects	3
Getting Started	13
Objectives	14
Step 1: Starting Mastercam	14
Step 2: GUI - Graphical User Interface	15
Step 3: Navigate Through Mastercam	16
Step 4: Set the attributes	18
Step 5: Manager Panels	20
Step 6: Setting mastercam to imperial	22
Step 7: Set the Grid	24
Conventions used in this book:	25
Mastercam® Work Flow	26
Tutorial 1	29
Step 1: Select The Geometry File	31
Step 2: Select The Machine And Set Up The Stock	32
Step 3: Project Curve Multiaxis Toolpath	35
Step 4: Backplot The Toolpath	48
Step 5: Verify The Toolpath	50
Step 6: Save The Updated MCAM File	52
Tutorial #1 Quiz	54
Tutorial 2	55
Step 1: Select The Geometry File	57
Step 2: Set The Stock Model From The RoughING Operations	58
Step 3: Finish The Walls - Swarf Milling	60
Step 4: Backplot and Verify The Toolpath	69
Step 5: Parallel Toolpath With Tool Axis Tilted to Chain	77
Step 6: Machine Simulation	87
Step 7: Save The Updated MCAM File	92

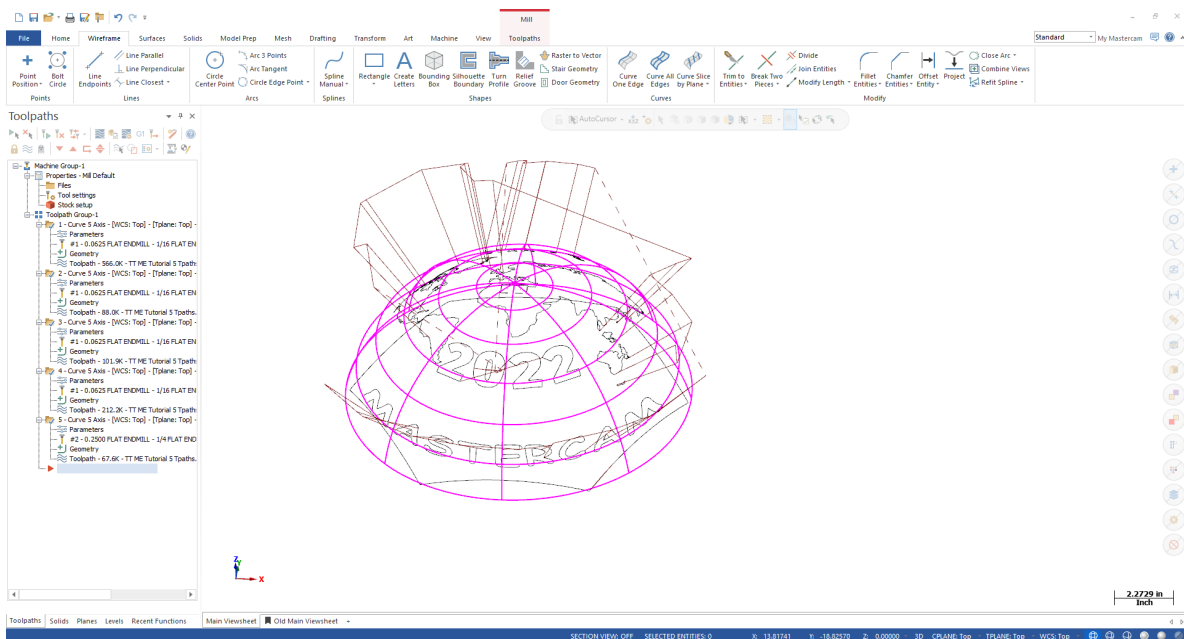
Tutorial #2 Quiz	94
Tutorial 3	95
Step 1: Select The Geometry File	97
Step 2: Select The Machine And Set Up The Stock	98
Step 3: Finish The Blade - Parallel (To Angle) Toolpath	102
Step 4: Backplot The Toolpath	109
Step 5: Finish The Blade - Morph (Between 2 Curves) Toolpath	111
Step 6: Finish The Fillets - Parallel To Curves With Limits	127
Step 7: Machine Simulation	139
Step 8: Save The Updated MCAM File	143
Tutorial #3 Quiz	145
Tutorial 4	147
Overview Of Steps Taken To Create The Final Part:	148
Step 1: Select The Geometry File	149
Step 2: Select The Machine And Set Up The Stock	150
Step 3: Rough Out The Screw Using Rotary Advanced	153
Step 4: Backplot The Toolpath	162
Step 5: Verify The Toolpath	163
Step 6: Finish The Floor Using the Morph From Curves	164
Step 7: Stock Model	175
Step 8: Finish The Fillet Surfaces - Parallel (To Curves)	178
Step 9: 3D High Speed Scallop	195
Step 10: Save The Updated MCAM File	201
Tutorial #4 Quiz	203
Tutorial 5	205
Overview Of Steps Taken To Create The Final Part:	206
Step 1: Select The Geometry File	207
Step 2: Select The Machine And Set Up The Stock	208
Step 3: Rough Out The Impeller - Morph (Between 2 Surfaces)	211
Step 4: Finish The Impeller Floor	230

Step 5: Finish The Blade Surfaces - Parallel To Curves	234
Step 6: Finish the floor close to the fillet - Parallel surface	244
Step 7: Transform - Rotate	252
Step 8: Transform rotate the second & third toolpaths	254
Step 9: Transform rotate last two toolpaths	255
Step 10: Machine Simulation	258
Step 11: Save The Updated MCAM File	262
Tutorial #5 Quiz	264
Overview Of Steps Taken To Create The Final Part:	266
Step 1: Select The Geometry File	267
Step 2: Collision Control Off	268
Step 3: Backplot The Toolpath	276
Step 4: Tilt Tool - Use Side Tilt Angle	277
Step 5: Collision Control - Retract Tool Along Tool Axis	286
Step 6: Retract Tool Along +Z	290
Step 7: Modify The Toolpath To Machine The Entire Surface	293
Step 8: Save The Updated MCAM File	295
Tutorial #6 Quiz	297
Tutorial 7	299
Overview Of Steps Taken To Create The Final Part:	300
Part 1	301
Step 1: Select The Geometry File	301
Step 2: Unified Toolpath From Curve - Guide style)	302
Step 3: Unified Toolpath From Curves - Morph style	309
Step 4: Unified Toolpath for Port Machining	316
Step 5: Unified Toolpath For Engraving - Project style	324
Part 2	331
Step 6: Select The File	331
Step 7: Unified Toolpath From Curve - Guide style	331
Step 8: Unified Toolpath From Curve - Parallel style	340

Step 9: Unified Toolpath From Curve - Parallel	349
Tutorial #7 Quiz	359
General Notes	361
Default Key Assignments	362
Customizing The Interface	363
Customizing Quick Access Toolbar	363
Create A New Tab	365
Customize The Right Mouse Button Menu	368
Example: Add The Zoom Target To The Right Mouse Button Menu	369
Key Mapping	372
Example: Assign Line Endpoints to Ctrl + I	373
Data Entry Shortcuts	376
Example On How To Use A Shortcut	376
Creating 3D wireframe	377
About Planes	377
About The Tool Plane	380
Define A New Construction Plane	381
About Construction Depth (Z Depth)	382
Chaining	384
How To Chain	386
Operation List Area	391
Right Mouse Click In The Toolpaths Manager List Area	392
Machine Group Properties	395
Tool Settings	396
Stock Setup	397
Multiaxis Toolpath Families	398
Project Curve Toolpath	398
Morph Toolpath	400
Parallel Toolpath	402
Port	405

Triangular Mesh	407
Swarf Milling Toolpath	409
Pocketing	411
Port Expert	412
Blade Expert	413
Convert to 5 axes	414
Unified Toolpath	415
Quiz Answers	419

Getting Started



OBJECTIVES

- Starting Mastercam
- The student will learn about the Graphical User Interface.
- The student will learn how to navigate through Mastercam.

STEP 1: STARTING MASTERCAM

1.1 For Windows 7

- Select the **Start** button.
- Select **All Programs** and click on Mastercam 2022.

1.2 For Windows 8

- Select the Start button.
- Click on the drop down arrow to open Apps.
- Find and click on Mastercam 2022.

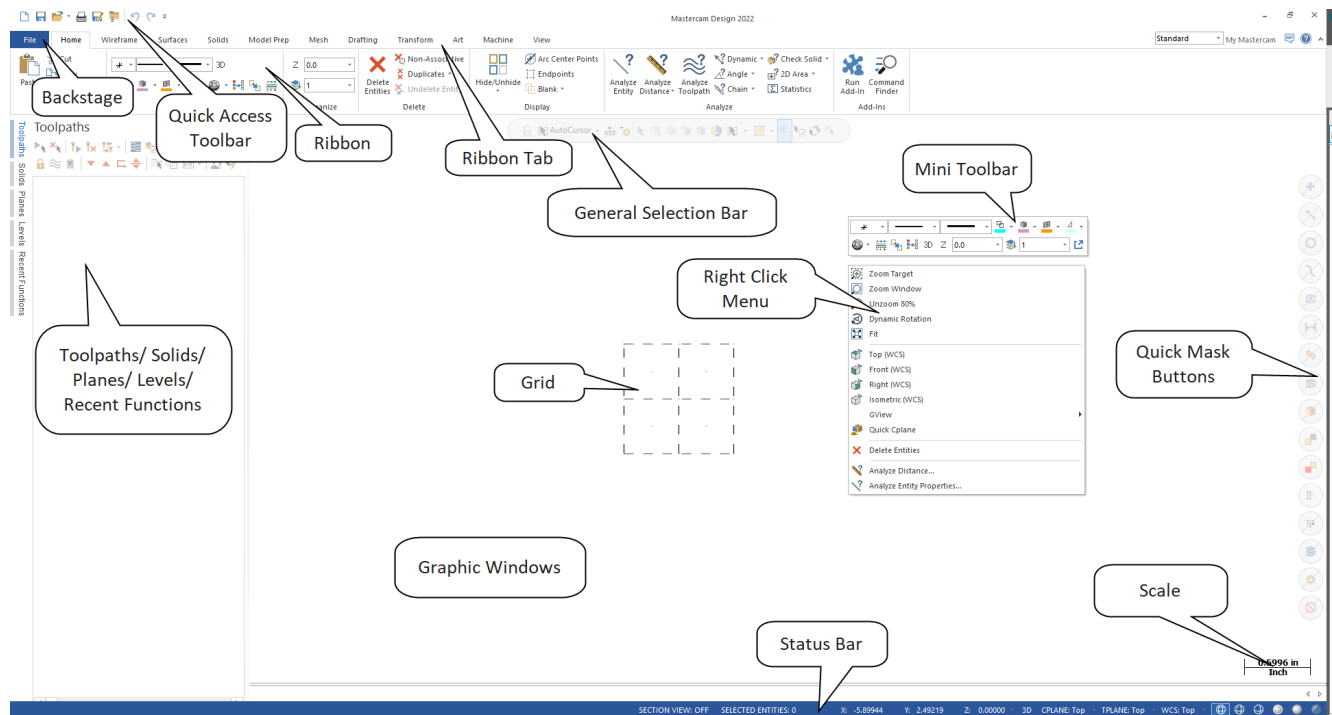
1.3 For Windows 10

- Select the Start button.
- Click on the drop down arrow to open Apps.
- Find and click on Mastercam 2022.

- To start the software, from Desktop, click on the shortcut icon as shown.



STEP 2: GUI - GRAPHICAL USER INTERFACE



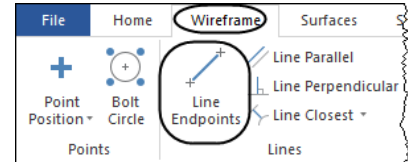
Quick Access Toolbar	QAT contains a fully customizable set of functions that can be quickly accessed by the user.
Backstage (FILE)	Allows you to manage files. You can insert information about files, start a new file, open an existing one or merge files together. You can also save, convert or print files as well as access the help resources.
Tabs	Contain all the functionality within Mastercam.
Ribbon	Displays the commands available for a selected Tab.
Selection Bar	Allows you to set the AutoCursor modes and to switch between wireframe or solid selections.
Quick Mask Buttons	Lets you select all entities of a specific type. Clicking on the left side of the button or right side of the button toggles between select all or only.
Right Click Menu	Right click menu allows quick access to functions such as zoom, graphic views or recent functions used. A mini toolbar will also appear that allows you to quickly change the attributes.
Toolpaths/Solids/Planes Manager	Lists the history of the toolpath operations and solids.
Graphics Window	Workspace area in Mastercam where the geometry is displayed.
Scale	Shows you a scale of the object on the screen.
WCS: TOP T/Cplane:	Displays the current WCS and T/Cplane information.

STEP 3: NAVIGATE THROUGH MASTERCAM

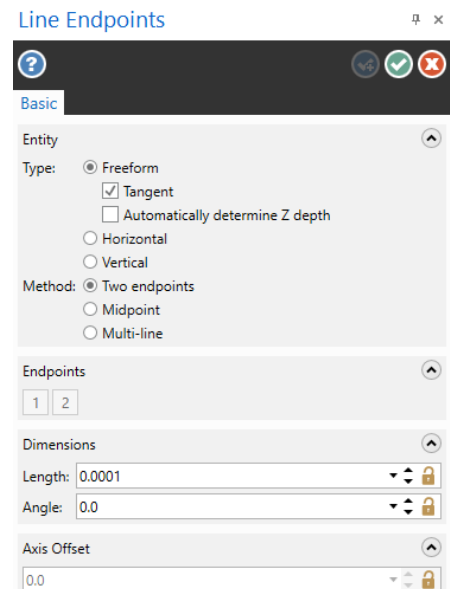
In this step, you will learn how to use the menu functions in Mastercam to create geometry.

3.1 Using the Wireframe tab to select the command to create Line Endpoints

- Left click on **Wireframe**.
- Left click on the **Line Endpoints** icon as shown.



- Once you select **Line Endpoints**, the **Line Endpoints** panel appears on the screen as shown.



Sketching A Line

- To sketch a line, left click on two locations on the screen.

Creating A Line Knowing The Endpoint Coordinates



- To make a line knowing the two endpoint coordinates, select the **AutoCursor Fast Point** icon from the **General Selection** toolbar.





- In the coordinates field that opens in the upper left corner enter the coordinates of the first endpoint as shown.
- Press **Enter** to continue.
- Select the **AutoCursor Fast Point** icon again and enter in the coordinates of the second endpoint and then press **Enter**.

0,1

Creating A Line Knowing An Endpoint, The Length, And The Angle

- You can also enter the coordinates of the first endpoint, then enter the **Length** and **Angle** if necessary.
- To continue making lines, choose the **OK** and **Create New Operation** button from the dialog box or press **Enter**. 
- To exit the current command, select the **OK** button or press the **Esc** button. 

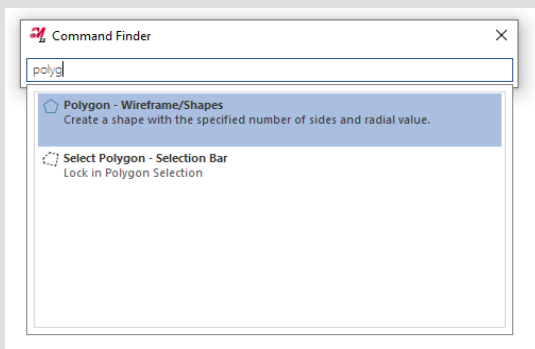
Note: To undo the last command, from the **QAT** (**Q**uick **A**ccess **T**oolbar) select the **Undo** button. 
The Undo button can be used to go back to the beginning of geometry creation or to the last point of the saved file. Mastercam also has a **Redo** button for your convenience. 

Function Prompt

Prompts the user to execute a command.

Example: this prompt is used in the **Line Endpoints** command. Specify the first endpoint

Note: To find a command, from the **Home** ribbon, select the **Command Finder** icon and type the function name in the field that opens up.
For example, to find the **Polygon** command type "polygon" in the text field. From the list, select the desired command.



STEP 4: SET THE ATTRIBUTES

Mastercam attributes are point style, line style, line thickness, color and levels. Before starting to create geometry, you should set the attributes.

4.1 Attributes Group

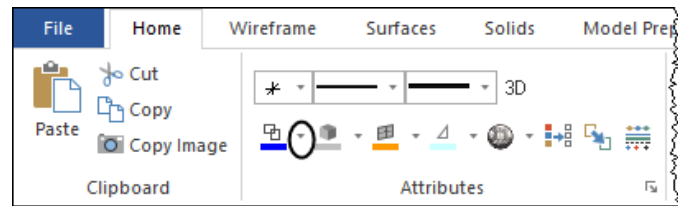
Point Style	Displays and sets the system's point style.
Line Style	Displays and sets the system's line style.
Line Width	Displays and sets the current system's line width.
Color	Assigns the current color to wireframe, solid and surface entities. To change the current color, click in the specific color field and select a color from the color pallet. To change an existing geometry color, select the geometry first and then click in the color field and select a color from the color pallet.
Clear Color	When performing a transform function (Xform), Mastercam creates a temporary group from the originals (red) and a result (purple) from the transformed entities. These system groups appear in the Groups dialog box. However, they stay in effect only until you use the Clear Colors function or perform another transform function.
2D / 3D Construction Mode	Toggles between 2D and 3D construction modes. In 2D mode, all geometry is created parallel to the current Cplane at the current system Z depth. In 3D mode, you can work freely in various Z depths, unconstrained by the current system Z depth and Cplane setting.

4.2 Organize Group

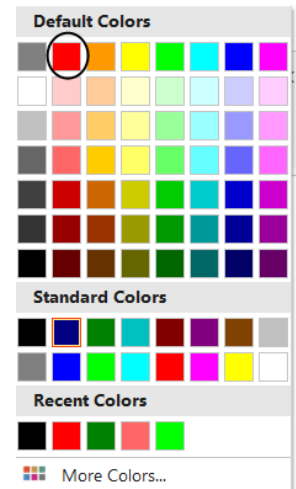
Z Depth	Sets the current construction depth. To set this, click the drop down arrow and pick one from the most recently used list or click the Z: label and pick a point in the graphics window to use the Z depth values based on the selected entity.
Level	Sets the main level you want to work with in the graphics window. To change the current working level. Type the level number in the box.

Change The Wireframe Color

- Click on the drop down arrow next to the **Wireframe Color** field as shown.



- Select the desired color from the dialog box as shown.



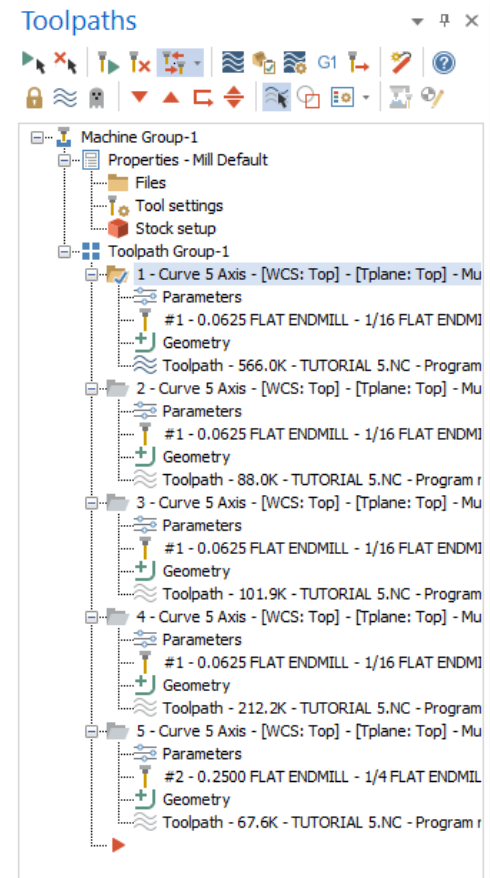
Note: Any geometry on your screen will remain in the previous system color. This change will only affect the geometry you create going forward.

To change the color of existing geometry, select the entities first and then click on the drop down arrow next to the Wireframe Color and select the desired color. The same method can be applied for any other attribute that you want to set or change.

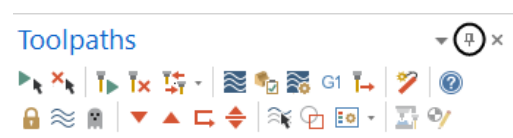
STEP 5: MANAGER PANELS

5.1 The Toolpaths Manager

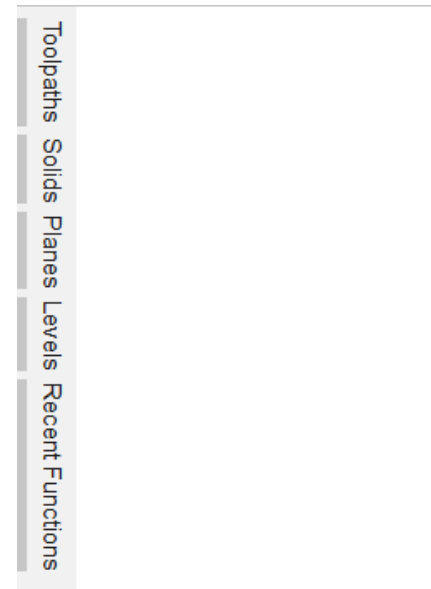
The **Toolpaths Manager** displays all the operations for the current part. You can sort, edit, regenerate, verify and post any operation. For more information on the **Toolpaths Manager**, please click on the **Help** icon.



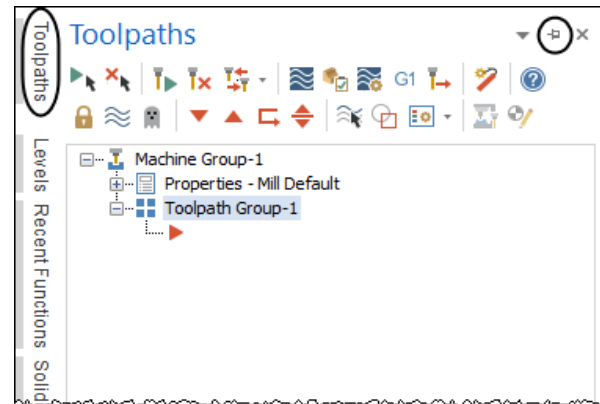
- The **Toolpaths Manager**, **Solids Manager**, or **Planes Manager** can be hidden to gain more space in the graphics area for creating geometry. Use **Auto Hide** icon to close all **Toolpaths**, **Solids**, **Planes** and **Levels Manager** panels.



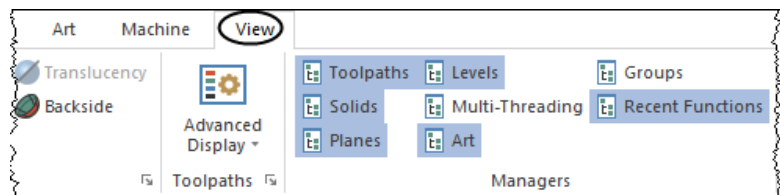
- The panels will be hidden to the left of the graphics window as shown or at the bottom of the manager as shown previously.



- To un-hide them, click on one of the managers to open it and then click again on the **Auto Hide** icon as shown.



- Selecting the **X (Close icon)** instead of the **Auto Hide**, you will close the manager panel. To re-open them, from the **View** tab, select **Toolpaths**, **Solids**, **Planes** or **Levels** as shown.



STEP 6: SETTING MASTERCAM TO IMPERIAL

In this step you will learn how to set the imperial system as your default. You will have to select the **Backstage** options and select the system configuration.

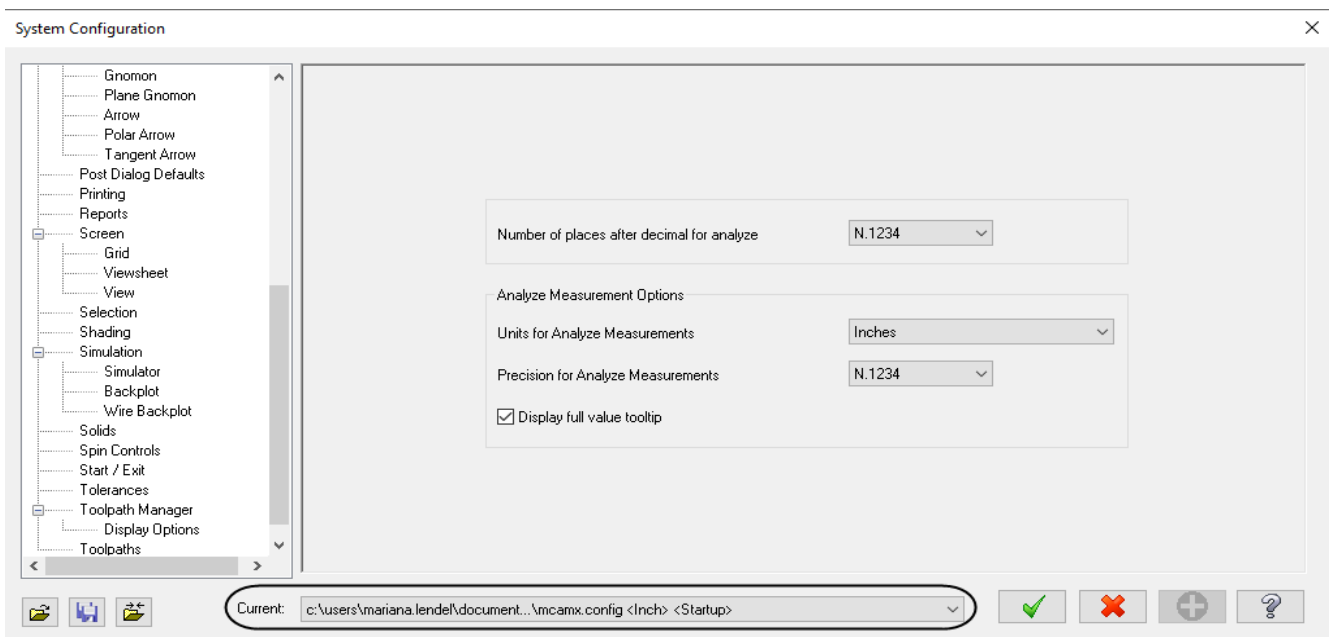
6.1 Setting Mastercam to inch for the current session only

Note: You may need to switch Mastercam to run in MetricInch mode.

File

■ Configuration.

- Select the drop down arrow beside **Current** as shown.
- Select **mcamxm.config <Inch>**.



- Select the **OK** button to exit the **System Configuration** dialog box. 

Note: If you have a drawing on the screen it may ask you to scale the current part to imperial. Choose Yes if you wish to do this.

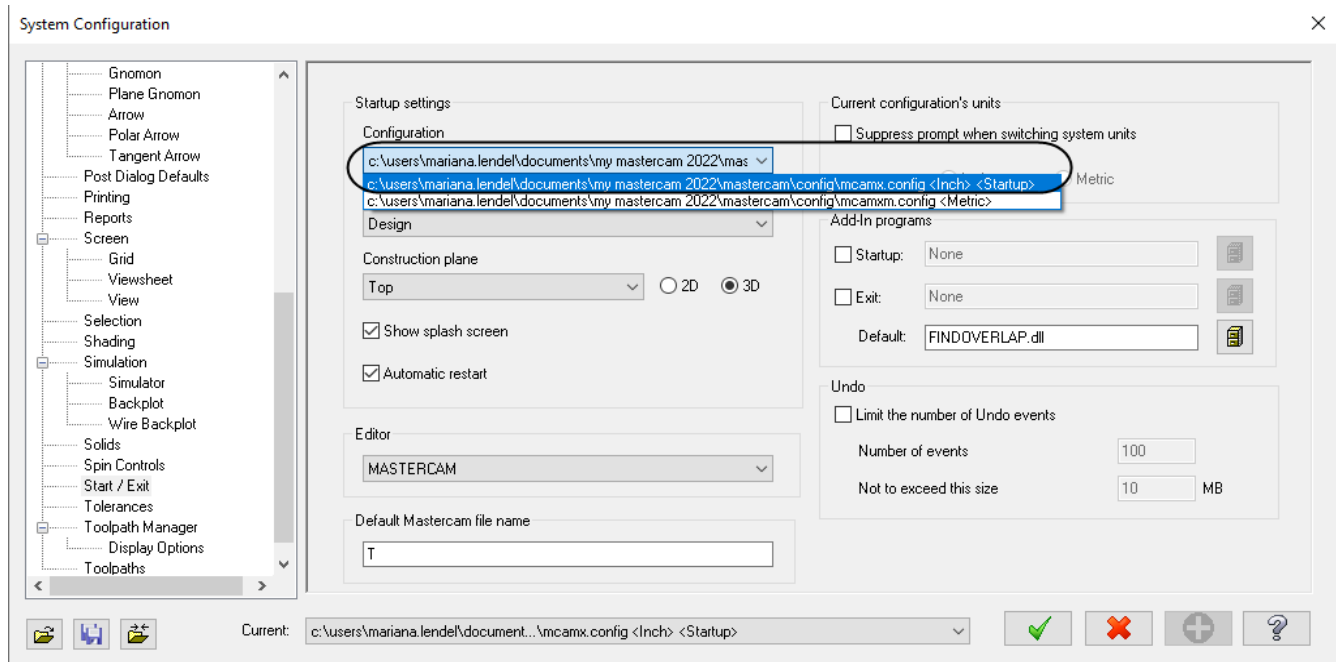
6.2 Setting Mastercam to imperial as a default

Note: If you wish to always work in Imperial mode, follow these steps to save metricmetric as your current configuration file.

File

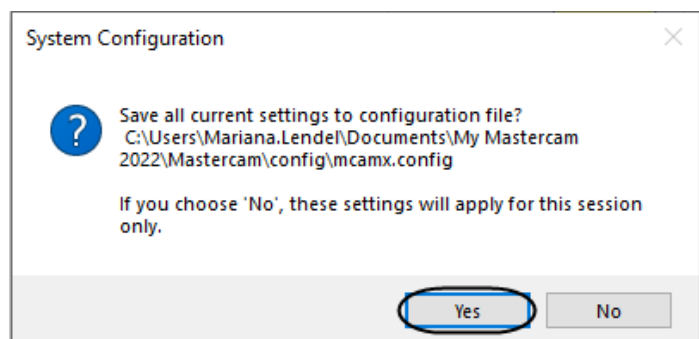
■ Configuration.

- Select **Start/Exit** from the configuration topics.
- Select the drop down arrow below **Configuration** in the **Startup** settings area as shown
- Select **mcamxm.config <Inch>** as shown.



- Select the **OK** button to exit the **System Configuration** dialog box.

- Mastercam will then prompt you to save these settings to your current configuration file, select **Yes**.



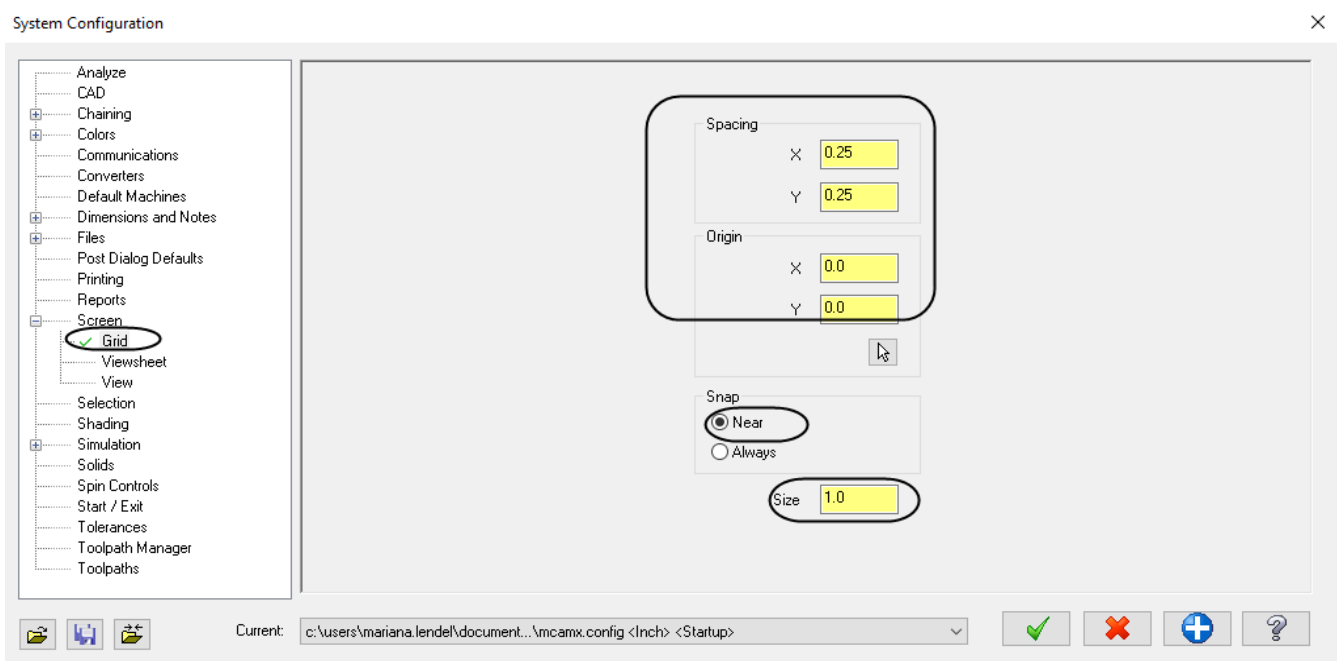
STEP 7: SET THE GRID

Before beginning to create geometry, it is highly recommended to enable the **Grid**. The grid will show you where the origin is and the orientation of the grid gives you a quick preview of the plane you are working in.

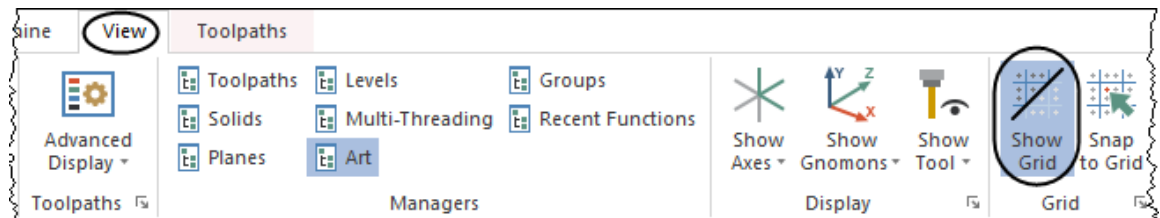
File

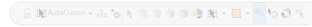
■ Configuration.

- Select **Screen** from the configuration **Topics**.
- Select the plus sign (+) beside **Screen** as shown.



- In **Grid**, change the **Spacing** to **X = 0.25** and **Y = 0.25**.
- Set the **Size** to **1.0**.
- Choose the **OK** button to exit.
- Select the **Yes** button to save the settings in the **System Configuration**.
- To see the **Grid** in the graphics window, from the **View** tab, enable **Show Grid** as shown.





- The grid should look as shown.



CONVENTIONS USED IN THIS BOOK:

We have attempted to make this manual as uncluttered as possible and provide you with reference information when it is appropriate. It is not intended to be a Reference Guide or all-encompassing user manual.

The Text Styles Used Are The Followings:

Standard Text - Represents normal wording needed to provide you the instructions.

STEP 8: STEP TITLES

8.1 Sub step titles

Information about the current step, terms or parameter definitions describing the parameters and description.

Bold Text - Represents menu commands, dialog box settings or other similar items from the screen.

Note: Represents information about the process/step that is important or may require an explanation.

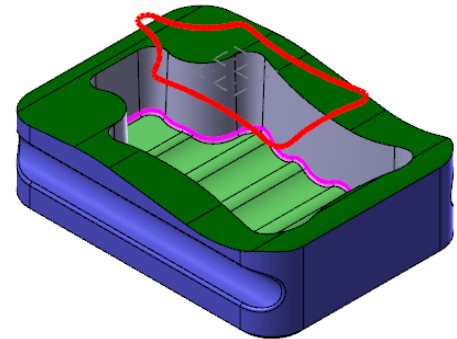
- Bulleted text are step by step instructions that are to be followed.

The files used in this book are available for download at <http://www.emastercam.com/files/>.

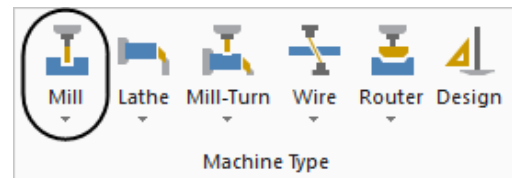
MASTERCAM® WORK FLOW

The process to create or import the geometry and to generate a toolpath will be repeated over and over through the tutorials in this book. You will find the process simple and straightforward once you have programmed a few parts. The following is an outline of the process we will follow to create programs:

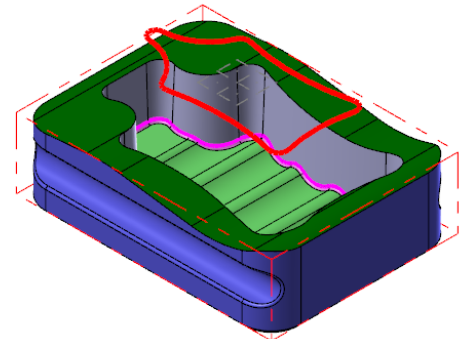
1. Create or import the part geometry.



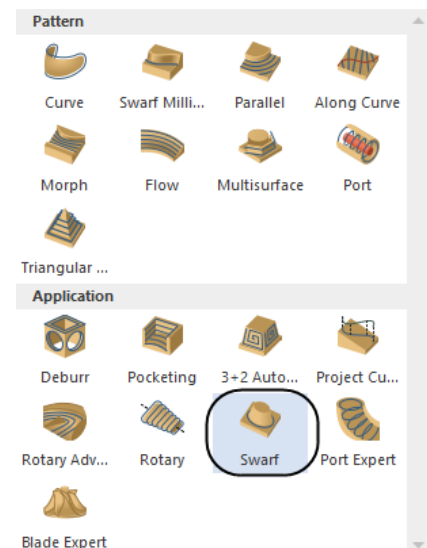
2. Select the Machine type.



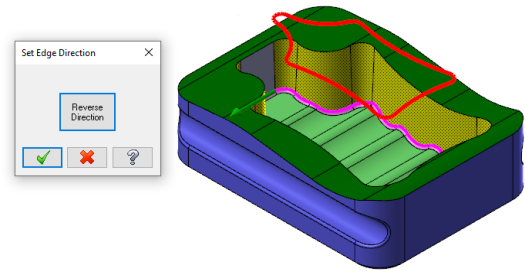
3. Define the stock size that your part will be cut from and set tool information.



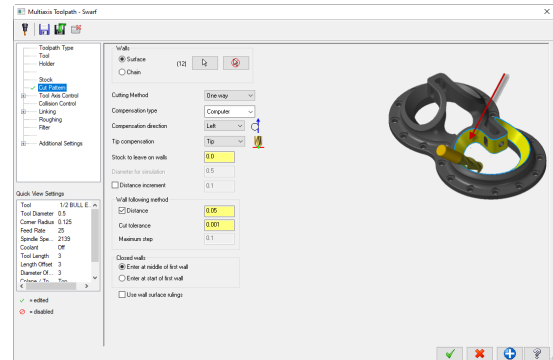
4. Select a toolpath type such as Swarf.



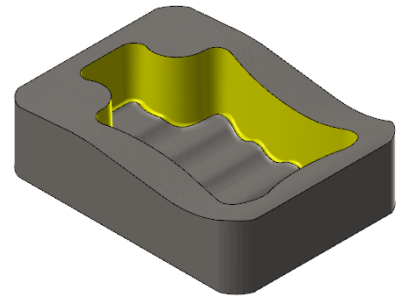
5. Select the geometry of the part you will cut with the different selection options.



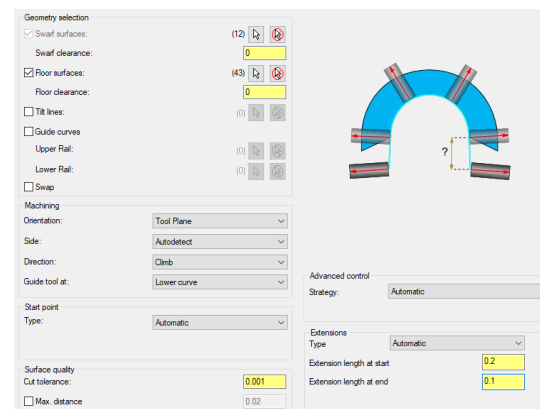
6. Fill in the necessary information on the parameter pages that appear for the toolpath type you selected.



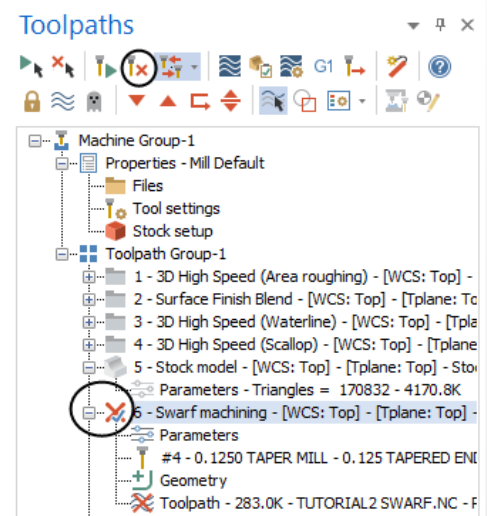
7. Verify the toolpath on your computer screen to confirm the results are as you expected, using Backplot and/or Solid Verify.



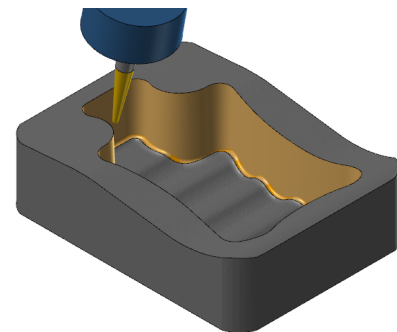
8. Make any changes as required by changing parameters.



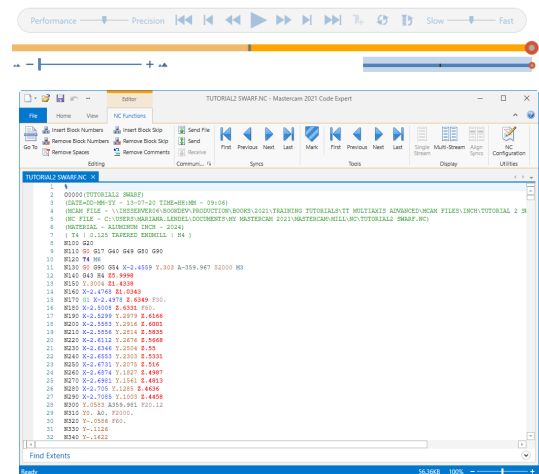
9. Make sure that the Generate toolpath is enabled otherwise, Regenerate the “Dirty” operation to update the parameter changes.



10. Verify again to make sure the toolpath is correct.



11. Convert the graphical toolpath information into machine code by Post Processing and sending it to the CNC machine.



Tutorial 1



OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

Open the CAD Model:

- The student will download the file that contains the geometry from www.emastercam.com.

Create the necessary Toolpaths to machine the part:

- The student will set up the stock size to be used and the appropriate tool settings.
- A Project Curve Multiaxis toolpath will be created to engrave the letters.

Backplot and Verify the file:

- The Backplot will be used to simulate a step-by-step process of the tool's movements.
- The Verify will be used to watch a tool machine the part out of a solid model.



This tutorial takes approximately 20 minutes to complete.

This tutorial covers Mastercam's 5 Axis toolpath functionality. To apply these toolpaths to a 5 Axis machine tool, a customized post processor for your machine is required.

There are no default 5 Axis post processors included with Mastercam due to the variation in multi-axis machine configurations. Contact your Mastercam reseller to request post processor services such as multi-axis post development.

Your post processor may require certain additional programming information not covered in this tutorial. For instance:

You may be required to position your stock in machine space rather than at Mastercam's origin.

You may be required to use the Misc Values.

Your post may also prompt you for tool gage lengths.

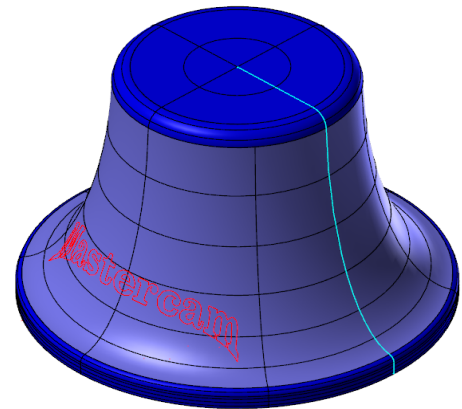
The nature of the additional information required largely depends on your machine's configuration. Contact the developer of your post processor for details.

STEP 1: SELECT THE GEOMETRY FILE

Resources - Download the file from www.emastercam.com/trainingfiles/.

File

- **Open.**
- Select "**TUTORIAL 1 PROJECT5 AXIS.MCAM**" from the directory you saved the file in.
- Press **Alt + F1** to fit the geometry to the graphics window.



STEP 2: SELECT THE MACHINE AND SET UP THE STOCK

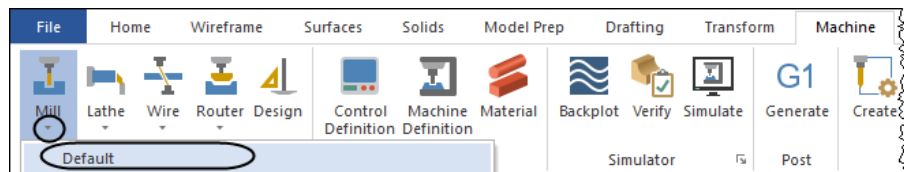
In Mastercam, you select a **Machine Definition** before creating any toolpaths. The **Machine Definition** is a model of your machine tool's capabilities and features and acts like a template for setting up machining jobs. The machine definition ties together three main components: the schematic model of your machine tool's components, the control definition that models your control unit's capabilities and the post processor that will generate the required machine code (G-code). For the purpose of this tutorial, we will be using the MILL Default machine.

Note: If you already have the default machine in the Toolpaths Manager, do not select another machine.

2.1 Select the Machine type

Machine

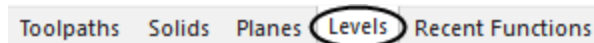
- From the **Machine Type** group, select the drop down arrow below **Mill**.
- Select the **Default**.



Note: Once you select the **Mill Default**, the **Ribbon** bar changes to reflect the toolpaths that could be used with **Mill Default**.

2.2 Make Level 2 visible to be able to select the stock

- Select the **Levels** tab to open **Levels Manager** as shown.

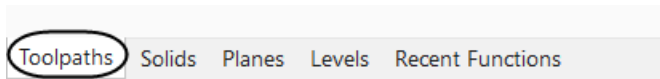


- Click in the **Visible** column next to **Number 2**.

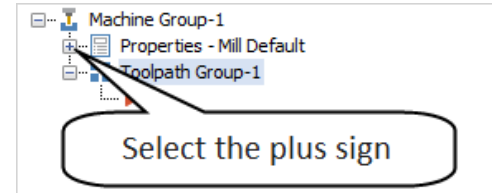
Num...	Visible	Name	Entities	Level Set
1	X	Geometry	149	
2	X	Stock-Solid	1	
10			259	

2.3 Set the tool parameters in the Tool settings

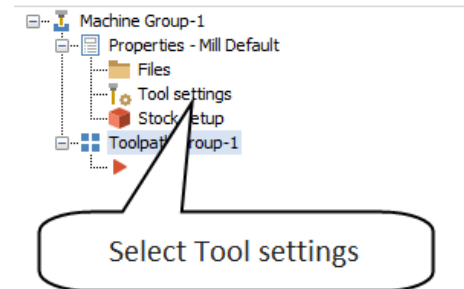
- To display the **Toolpaths Manager** panel select the **Toolpaths** tab as shown.



- Select the plus sign (+) in front of **Properties** in the **Toolpaths Manager** to expand the **Toolpaths Group Properties**.



- Select the **Tool settings** to set the tool parameters.



- Change the parameters to match the image shown below.

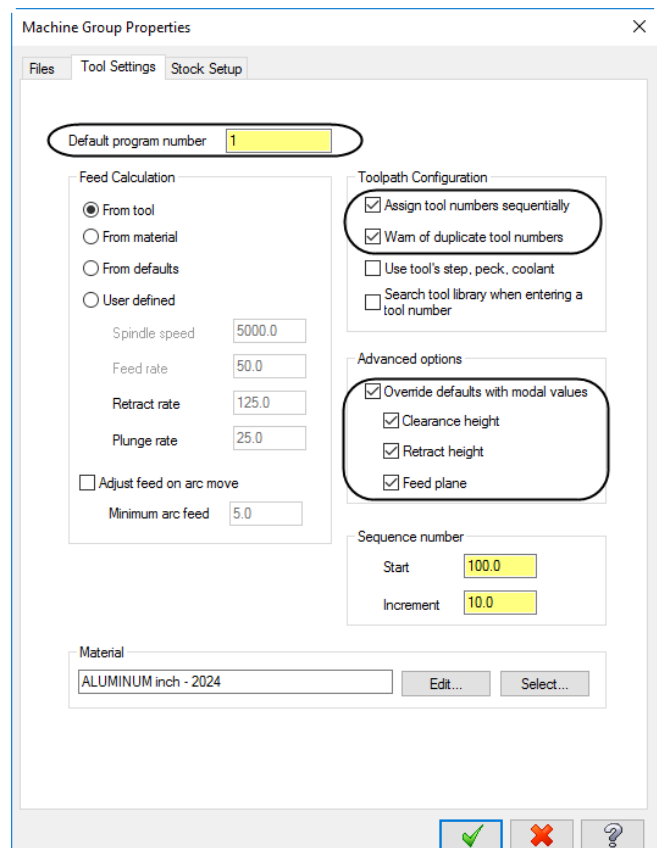
Default program number is used to enter a number if your machine tool requires a number for a program name.

Assign tool numbers sequentially allows you to overwrite the tool number from the library with the next available tool number (First operation tool number 1; Second operation tool number 2, etc.).

Warn of duplicate tool numbers allows you to get a warning if you enter two tools with the same number.

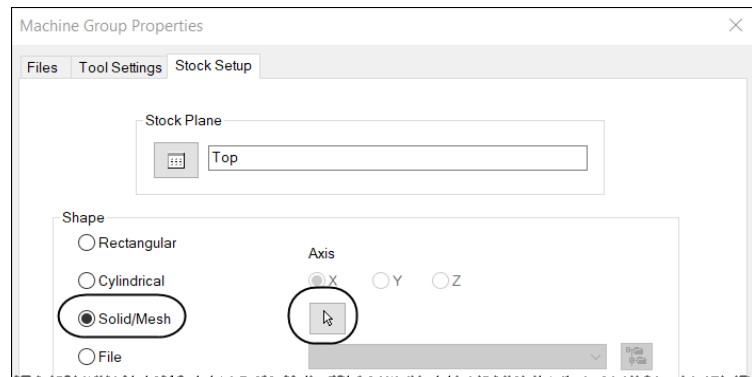
Override defaults with modal values enables the system to keep the values that you enter.

Feed Calculation set **From tool** uses feed rate, plunge rate, retract rate, and spindle speed from the tool definition.

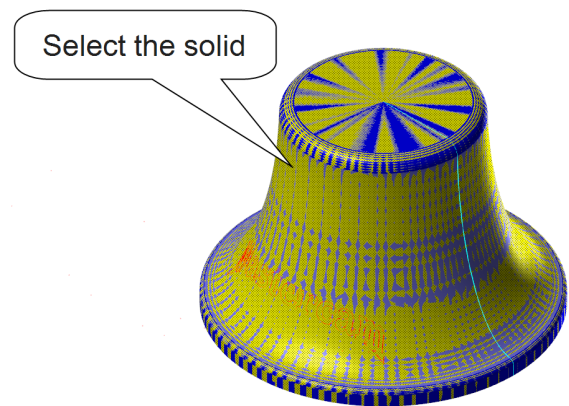


2.4 Set the stock shape and the size

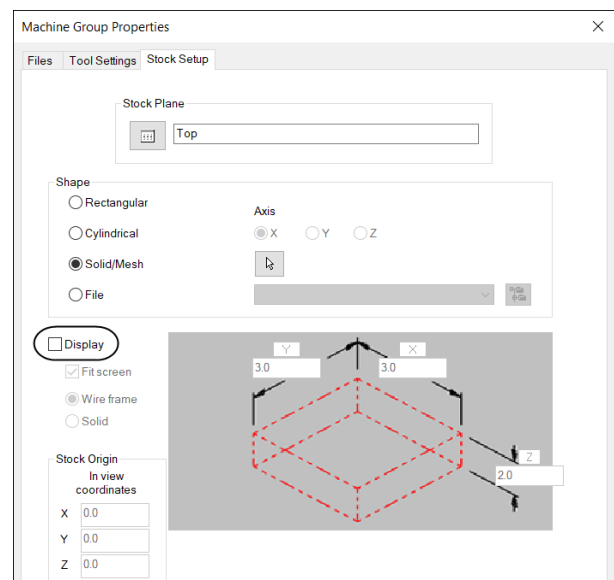
- Select the **Stock Setup** tab.
- In the **Shape** area, enable **Solid** and click on the **Select** button.



- [Select an entity]: Select the solid from the graphics window.



- The **Stock Setup** page should look as shown.



- Select the **OK** button to exit the **Machine Group Properties** dialog box.



2.5 Make Level 2 invisible

- Select the **Levels** tab to open the **Levels Manager** as shown.



- Click in the **Visible** column next to **Number 2** to remove the **X** and make **Level 2** invisible.

Num...	Visible	Name	Entities	Level Set
✓ 1	X	Geometry	149	
2		Stock-Solid	1	
10			259	

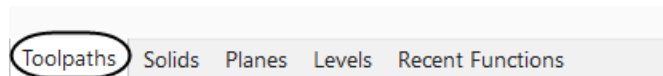
Note: Remember that the stock is not geometry and cannot be selected. You will use the defined stock while verifying the part.

STEP 3: PROJECT CURVE MULTIAXIS TOOLPATH

Advanced Multiaxis provides enhanced 5 Axis multisurface machining strategies. You can work with the full interface that gives you access to all the available parameters and options. You can also choose from a number of simplified interfaces that have been customized for specific applications and machining strategies.

These toolpaths work on surfaces. Solid selection is available for most advanced multiaxis toolpath strategies, with the following exceptions: toolpaths that require the selection of a defined edge (solid edge) and toolpaths that require the selection of only a single surface (solid face).

- To display the **Toolpaths Manager** panel select the **Toolpaths** tab as shown.



3.1 Create the Projection 5 Axis toolpath with the tool axis oriented normal to the surfaces

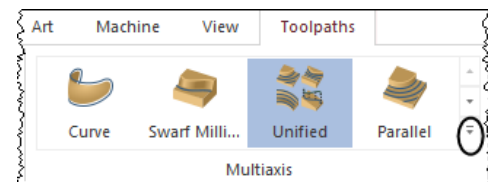
Project Curve is a toolpath that follows a curve that has been projected onto a set of surfaces. This is useful for engraving and similar applications. Projection curves should lie on or above the surface, within the maximum projection distance.

Toolpath Preview:

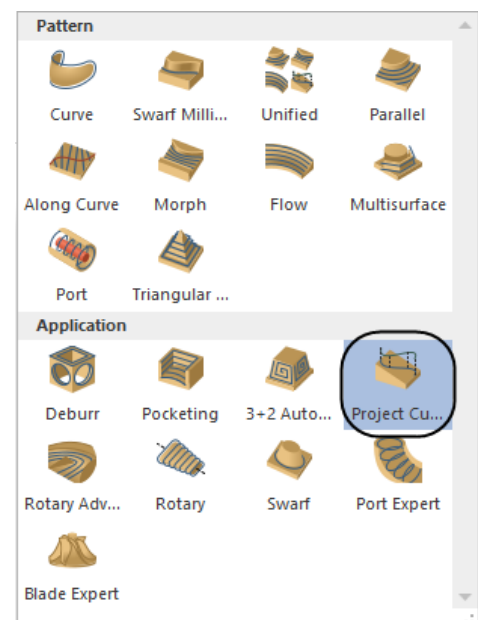


Toolpaths

- From the **Multiaxis** group, select the **Expand gallery** arrow as shown.



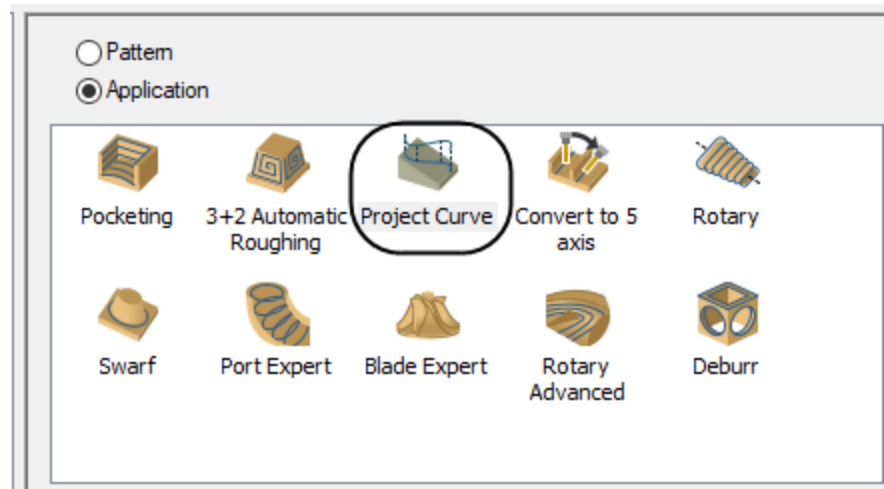
- From the **Application** gallery, select the **Project Curve** icon as shown.



3.2 Toolpath Type

The **Toolpath Type** page allows you to establish the type of multiaxis toolpath to create. The toolpath type controls the options available as you proceed down the tree structure. The **Toolpath Type** page allows you to select between the two toolpath groups, **Pattern** or **Applications**. You can then choose one of the toolpaths inside of that group.

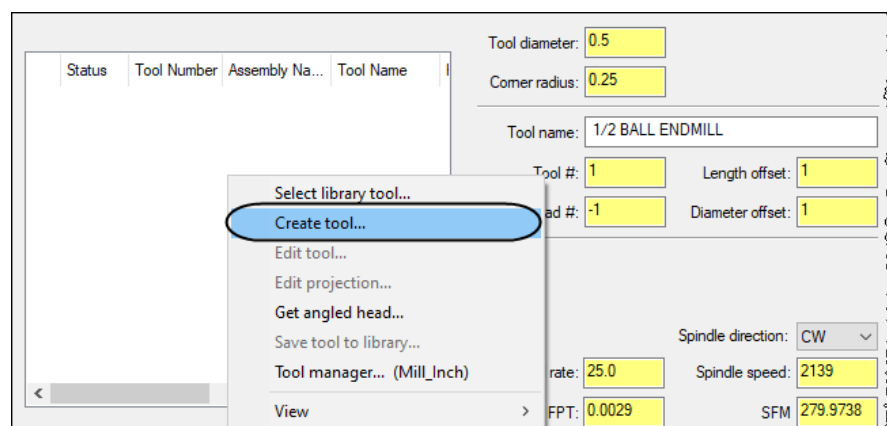
- Select **Toolpath Type** from the **Tree View List**.
- In the **Toolpath Type** area, **Project Curve** should be already selected as shown.



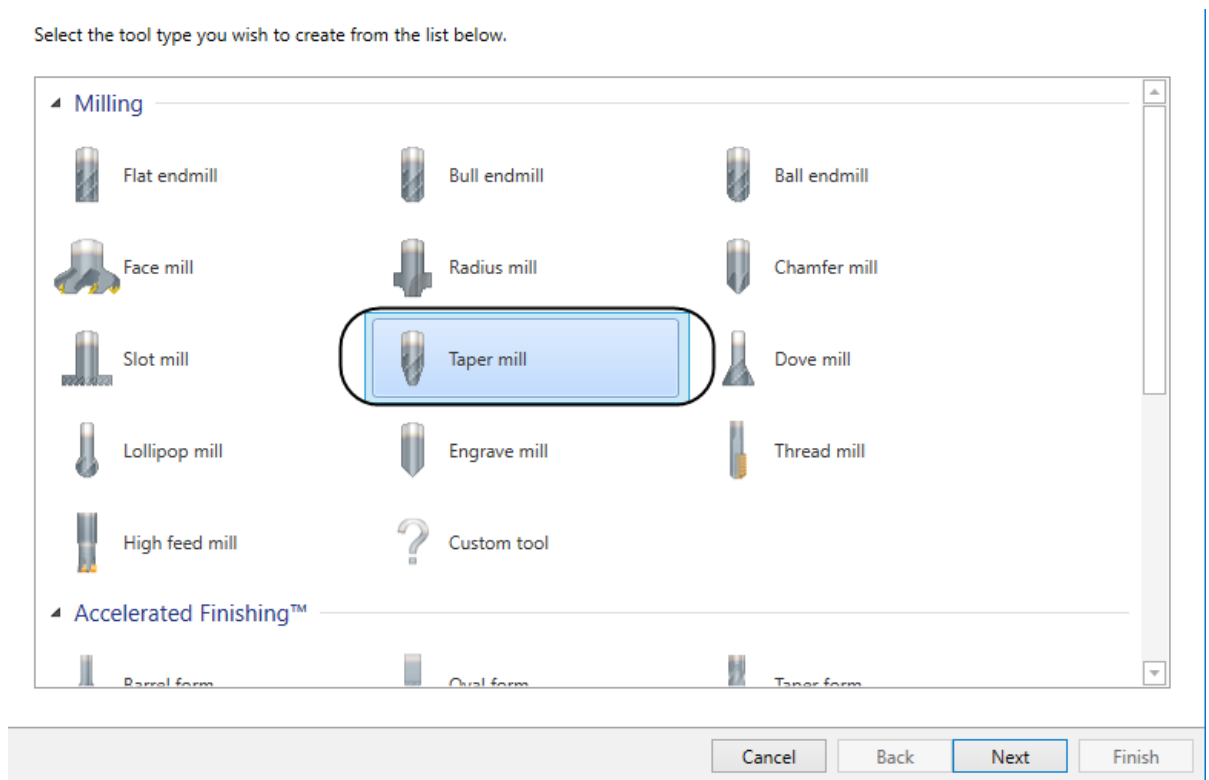
3.3 Tool

The **Tool** page allows you to select a tool, set the feeds and speeds, enter a comment about the operation, and set other general toolpath parameters.

- From the **Tree View list**, select **Tool**.
- Right-mouse click in the tool list window and select **Create tool**.



- As the tool type, select the **Taper Mill** as shown.



- Select the **Next** button and change the parameters as shown.

Note: Make sure that you enter the **Tip diameter** first. Once you set the **Corner type** to **Full Radius**, the **Tip diameter** will be grayed out.

×

Define Taper Mill

Adjust geometric properties used to define the tool shape.

Overall dimensions

Tip diameter: 0.05

Angle: 5

Cutting length: 0.25

Shoulder length: 1.2

Overall length: 2

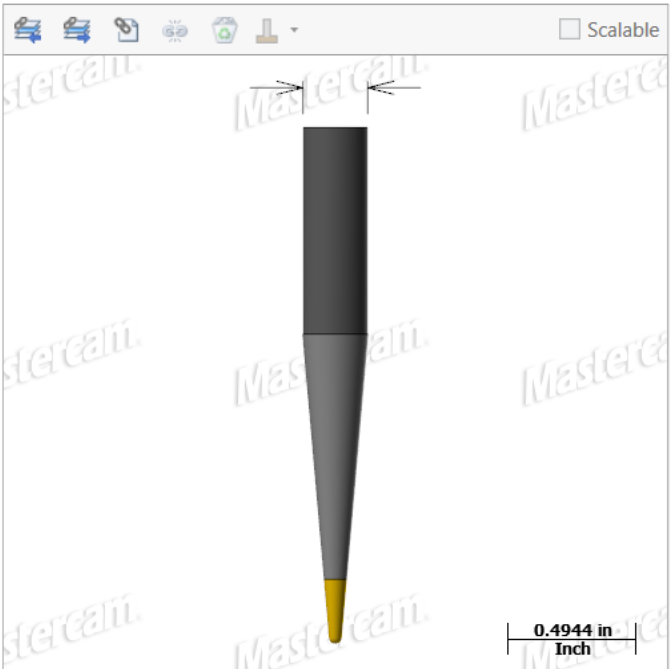
Tip / corner treatment

Corner type: Full Radius

Radius: 0.025

Non-cutting geometry

Shank diameter: 0.25



Scalable

Cancel

Back

Next

Finish

- Select the **Next** button and change the parameters as shown.

Finalize miscellaneous properties.

Adjust any miscellaneous properties before finalizing tool creation.

Operation	
Tool number:	1
Length offset:	1
Diameter offset:	1
Head number:	0
SFM:	57.06806
FPT:	0.00125
Number of flutes:	4
Feed rate:	50
Plunge rate:	25
Retract rate:	50
Spindle speed:	4360
Spindle direction:	Clockwise
Material:	Carbide

General	
Name:	0.25 Tapered Endmill
Description:	
Manufacturer name:	Mastercam Importer +
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

Cancel Back Next **Finish**

- Select the **Finish** button to continue.

3.4 Tool

- Add a comment in the **Comment** area to identify the toolpath in the **Toolpaths Manager** and also in the NC file.

- The **Tool** page should look as shown.
- Make sure that the **Tool diameter** value is **0.05** and the **Corner radius** value is **0.025**.

Note: The Feed rate, Plunge rate and Spindle speed values are the ones you set on the Parameters page of the Tool definition.

3.5 Holder

- In the **Holder** page, select the **B2C4-0016** holder as shown.

3.6 Cut Pattern

Cut Pattern settings determine the geometry that the tool follows and how it travels along that geometry.

The **Projection** button allows you to return to the graphics window to select the curves. Projection curves should lie on or above the surface, within the Max. projection distance.

The **Drive surface** button allows you to return to the graphics window to select the surfaces.

Drive surfaces offset allows you to set the depth the project toolpath will cut to.

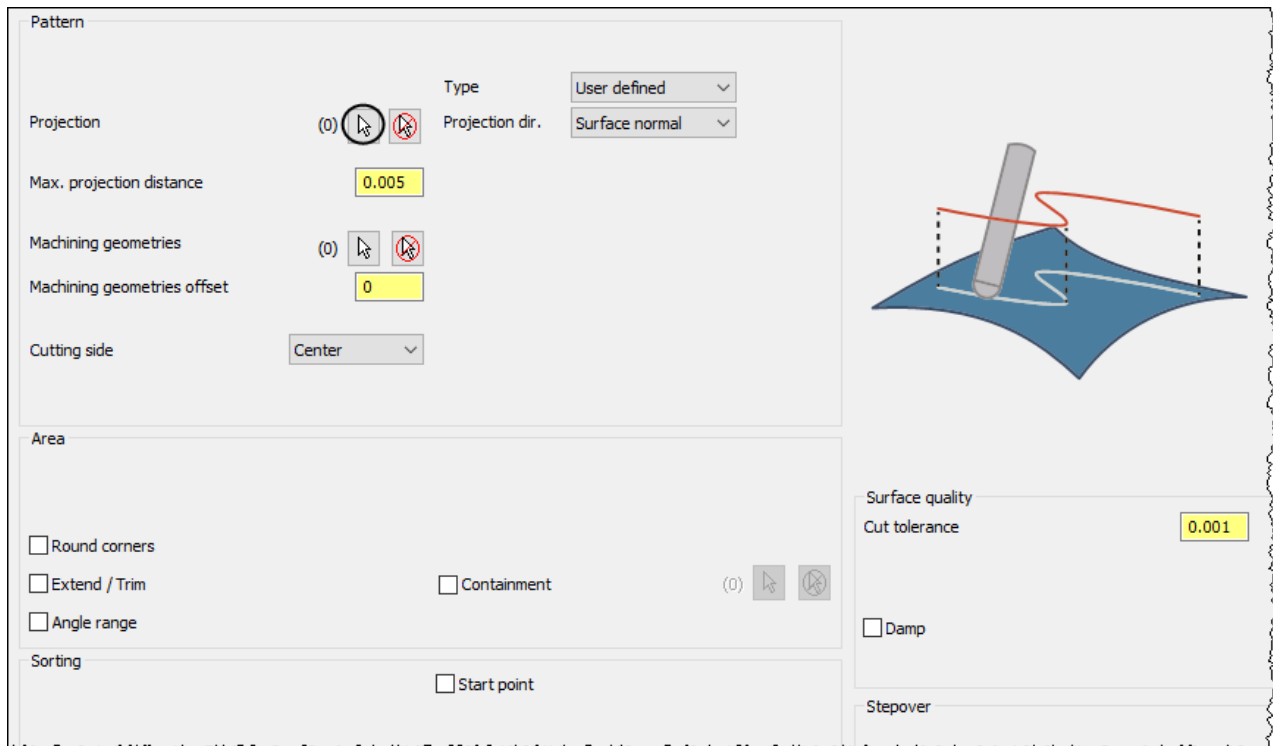
Projection dir allows you to select the projection method between along one of the axes, along a line or using the surface normal.

Type allows you to select a pattern from a drop down list.

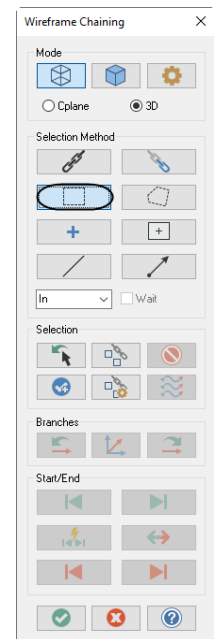
Cut tolerance sets the accuracy of the Multiaxis toolpath.

Max. projection distance sets the maximum distance between toolpath positions in the cutting direction.

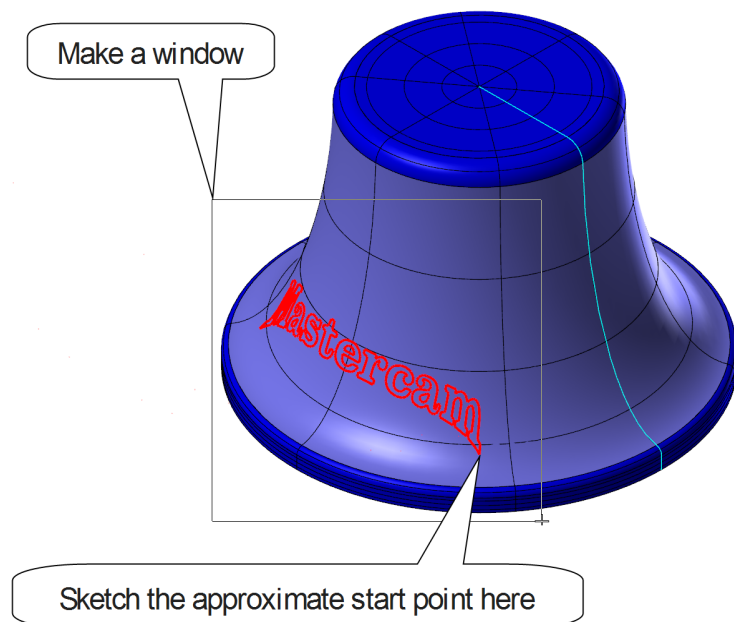
- From the **Tree View list**, select **Cut Pattern**.
- Click on **arrow** in the **Pattern** area to select **Projection**.



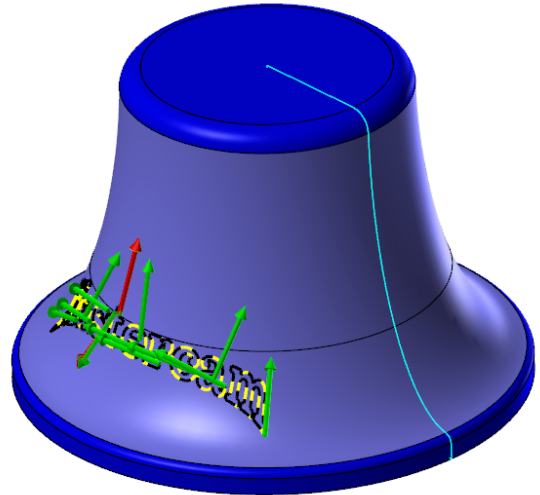
- From the **Chaining** dialog box, select the **Window** button as shown.




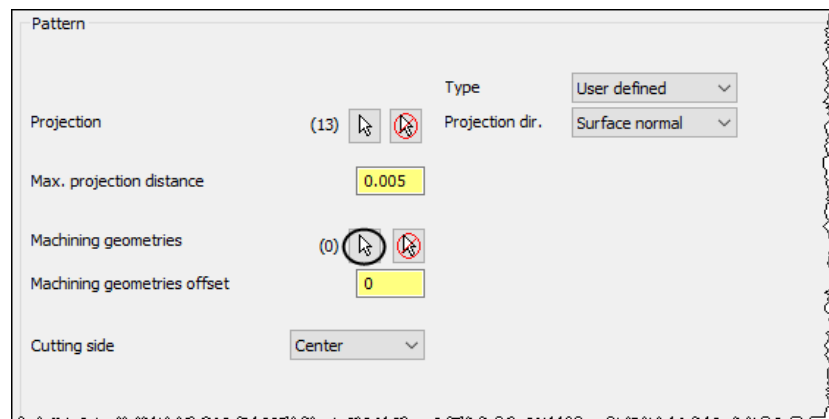
- Make a window around the letters as shown.
- [Sketch approximate start point]:
Select the point as shown.



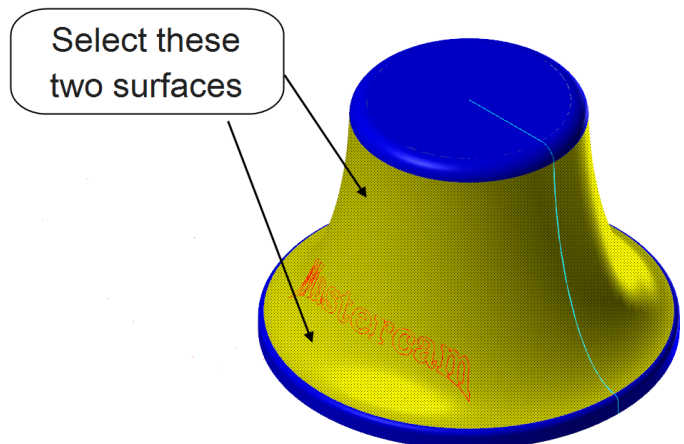
- The geometry should be selected as shown.



- Select the **OK** button to exit the **Chaining** dialog box. 
- Click on the **arrow** button as shown to select **Machining geometries**.



- [Select Solid Face, Surface or Mesh]:
Select the surfaces shown.
- Press **Enter** or the **End selection** button to finish the selection.



- Change the **Machining geometries offset** to **-0.005** and make sure that the rest of the parameters are set as shown.

Note: The **Machining geometries offset** sets the depth of the engraving.

Pattern

Projection (13) Type: User defined Projection dir.: Surface normal

Max. projection distance: 0.005

Machining geometries (2)

Machining geometries offset: -0.005

Cutting side: Center

Area

☐ Round corners

☐ Extend / Trim ☐ Containment (0)

☐ Angle range

Sorting ☐ Start point

Surface quality

Cut tolerance: 0.001

☐ Damp

Stepover

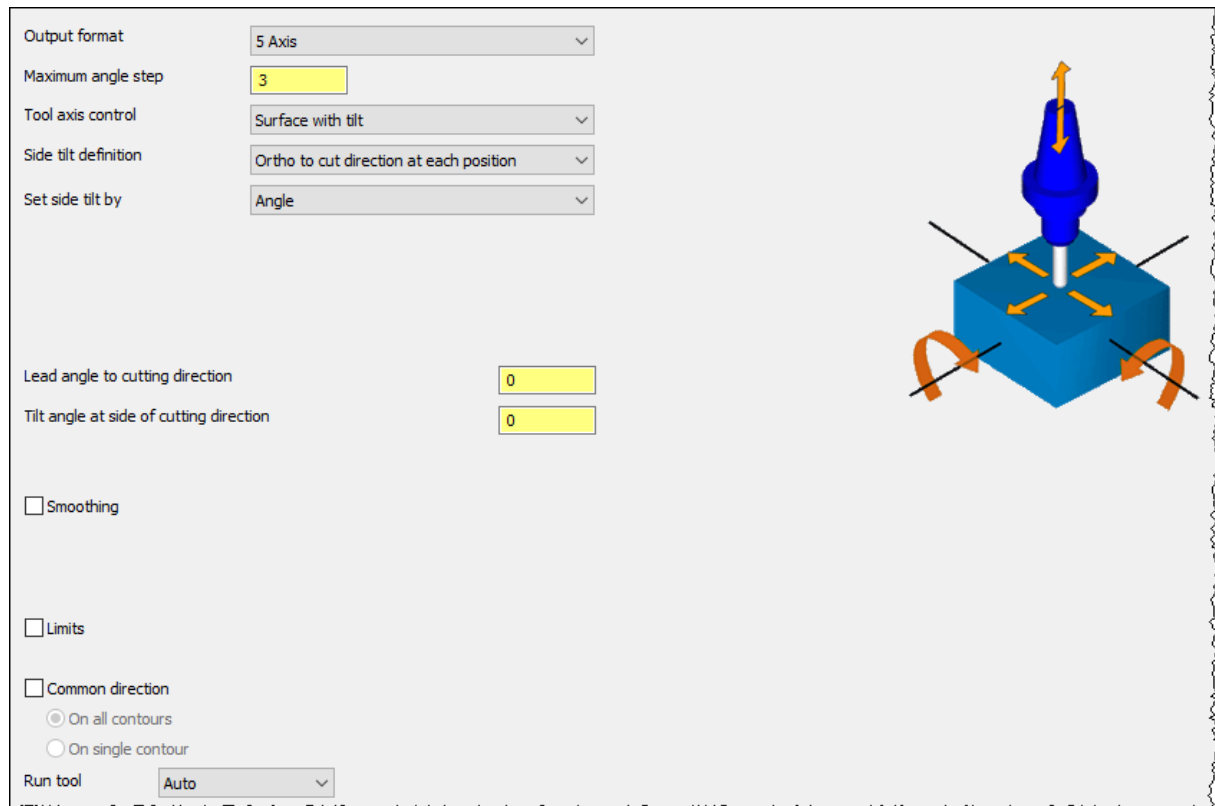
Note: The graphics in all of the toolpath parameter pages will change as you modify a parameter. This helps you visualize how that parameter influences the toolpath. Your image might be different from the one shown based on the parameter you selected to change.

3.7 Tool Axis Control

The **Tool Axis Control** page allows you to set the parameters that control the tool axis orientation in relation to the geometry being cut.

The **Maximum angle step** sets the maximum angle between the adjacent tool vectors generated along the curves.

- From the **Tree View list**, select **Tool Axis Control** and make sure the parameters are set as shown.



Output format: 5 Axis

Maximum angle step: 3

Tool axis control: Surface with tilt

Side tilt definition: Ortho to cut direction at each position

Set side tilt by: Angle

Lead angle to cutting direction: 0

Tilt angle at side of cutting direction: 0

☐ Smoothing

☐ Limits

☐ Common direction

☒ On all contours

☐ On single contour

Run tool: Auto

3.8 Linking

Linking determines how the tool moves when not cutting material.

Distances set the distances above the part, as incremental values, where the tool is moving at rapid feed rates or at the cutting feed rate. It also sets the minimum distance above the part for the tool to travel in the clearance area.

- From the **Tree View list**, select **Linking** and set the **Small gaps** and **Large gaps** to **Retract to feed distance** and leave the rest of the parameters as shown.

Entry/Exit

First entry: Approach from clearance area (dropdown), Don't use Lead-In (dropdown)

Last exit: Retract to clearance area (dropdown), Don't use Lead-Out (dropdown)

☐ Start from home position

☐ Return to home position

Home position (button)

Default links

Small gaps: Blend spline (dropdown), Don't use Lead-In/Out (dropdown)

Large gaps: Retract to rapid distance (dropdown), Don't use Lead-In/Out (dropdown)

Small gap size: 100.0 (text), ☒ in % of tool diameter, 0 (text), ☐ as value

Clearance area

Type: Plane (dropdown)

Direction: Z-axis (dropdown)

Height: User defined (dropdown), 6 (text), (button)

Incremental clearance plane

Direction: Z-axis (dropdown)

Type: Step (dropdown)

Incremental height: 6 (text), (button)

Advanced options for clearance area

☐ Interpolation tilt angles

☐ Keep initial orientation until distance: 0 (text)

Angle step for rapid moves: 5 (text)

Angle step for feed moves: 5 (text)

Distances

Rapid distance: 0.8 (text)

Entry feed distance: 0.4 (text)

Exit feed distance: 0.4 (text)

Air move safety distance: 0.4 (text)

☐ Rapid distance in tool plane

Arc fit

☐ Clearance area

☐ Rapid distance

☐ Feed distance

Arc radius: 0 (text)

Diagram: A 3D model of a part with a tool path. A red arrow points to the 'Z=?' label, indicating the clearance height.

Note: You have to retract the tool after each letter is machined to the feed distance to ensure no gouges are made into the part.

- Select the **OK** button to exit the **Multiaxis Toolpath - Project Curve** dialog box.

