

Mastercam 2022

▶ VIDEO ECOURSE PROMO CODE INCLUDED



MULTIAXIS ESSENTIALS

TRAINING TUTORIAL SERIES | IMPERIAL



2022

HLE DOWNLOAD
INSTRUCTIONS INSIDE

Mastercam 2022

Multiaxis Essentials 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, at the top, is the Mastercam logo (the word "Mastercam" in red and black, with "CAD/CAM Systems" in smaller black text below it) and a stylized red and white "M" logo. Below the logo, the text "Home Learning Edition Demo Software" is written in black, followed by "DOWNLOAD" in a larger, bold, black font.

HOW TO DOWNLOAD:

Mastercam Home Learning Edition (HLE)
Demo Software

Visit <https://signup.mastercam.com/demo-hle>

Proceed to download!

Mastercam 2022 Multiaxis Essentials Training Tutorial

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

Software: Mastercam 2022

Authors: Mariana Lendel

ISBN: 978-1-77146-946-3

Date: September 1, 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.

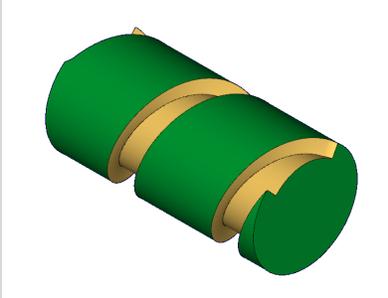
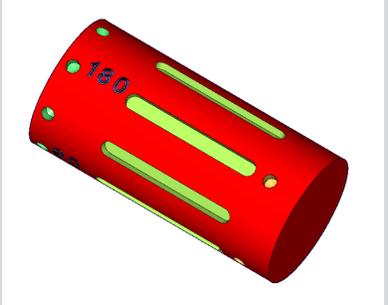
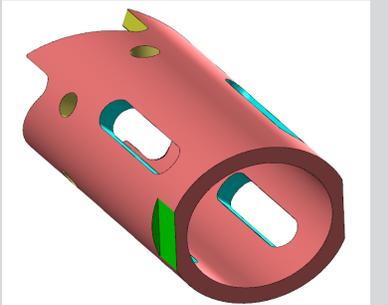
Trademarks

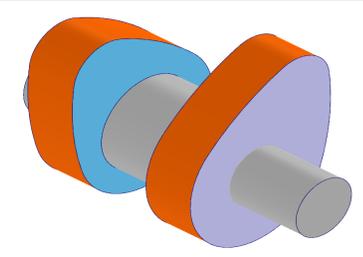
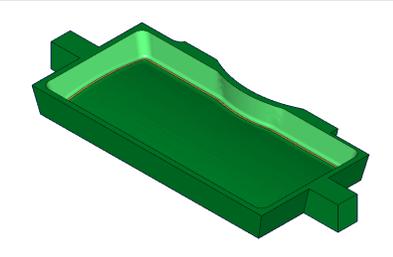
Mastercam is a registered trademark of CNC Software, Inc.

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

Windows 10 is a registered trademarks of Microsoft Corporation.

Multiaxis Essentials Projects

Tutorial	Toolpath Creation
<p>#1</p> 	<p>Contour with Axis Substitution.</p>
<p>#2</p> 	<p>Drilling with Axis Substitution. Drilling with Rotary Axis Positioning. Contour with Axis Substitution. Contour Toolpath. Transform Rotate.</p>
<p>#3</p> 	<p>2D HS Dynamic Mill with Axis Substitution. Contour with Axis Substitution. Swarf Milling. Drill the Holes -4 Axis output. Setup a new plane. 2D HS Dynamic Mill using Index about X-axis. Transform - Rotate. Setup a new plane. Contour Indexing using Planes. Transform - Rotate.</p>

Tutorial	Toolpath Creation
<p>#4</p> 	<p>Rotary 4-Axis with Rotary Cut Rotary 4-Axis with Axial Cut</p>
<p>#5</p> 	<p>Curve 5 -Axis with Tool Axis Control To Point Curve 5 -Axis with Tool Axis Control From a Point Curve 5 -Axis with Tool Axis Control Normal to a Plane Curve 5 -Axis with Tool Axis Control towards a Chain Curve 5 -Axis with Tool Axis Control to Lines</p>
<p>#6</p> 	<p>Stock Model Swarf 5-Axis with Walls defined from Chains Swarf 5-Axis with Walls defined from Surfaces</p>
<p>#7</p> 	<p>Stock Model Circle Mill 5-Axis Multiaxis Linking</p>

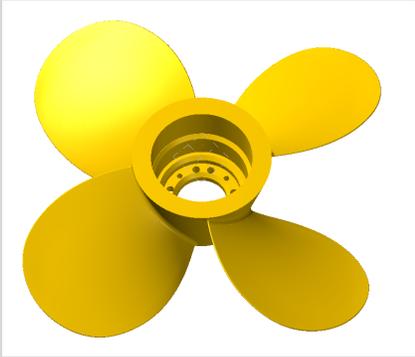
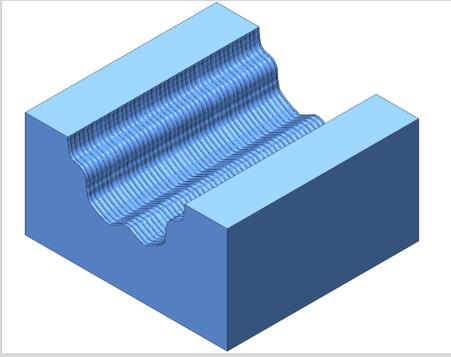
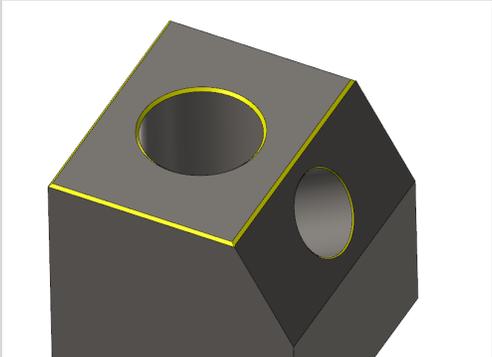
Tutorial	Toolpath Creation
<p>#8</p> 	<p>Flow 5-Axis with Tool Axis Control to Lines Machine Simulation Flow 5-Axis with Tool Axis Control to Point Transform Rotate</p>
<p>#9</p> 	<p>Multisurface 5-Axis Roughing Multisurface 5-Axis Finish Machine Simulation</p>
<p>#10</p> 	<p>Deburr 3 Axis Deburr 3+2 Axis Deburr 5 Axis</p>

Table Of Content

Multiaxis Essentials Training Tutorial	1
Multiaxis Essentials 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
Tutorial 1	27
Overview Of Steps Taken To Create The Final Part:	28
Step 1: Create the Geometry of the Cylindrical helix	29
Step 2: Select the Machine and Set Up the Stock	37
Step 3: Cylindrical Helix - Axis Substitution About Y	42
Step 4: Backplot The Toolpaths	51
Step 5: Verify the Toolpath	52
Step 6: Post the File to Generate the NC Code	54
Step 7: save the Updated MCAM File	55
Step 8: Create the Geometry of the Conical helix	56
Step 9: conical helix - axis substitution about y axis	59
Step 10: Save the Updated MCAM File	60
Tutorial #1 Quiz	62
Tutorial 2	63
Overview Of Steps Taken To Create The Final Part:	64
Step 1: Select the Geometry File	65
Step 2: Select the Machine and Set Up the Stock	66
Step 3: Drilling Toolpath - Axis Substitution About Y Axis	70

Step 4: Backplot The Toolpath	77
Step 5: Verify the Toolpath	79
Step 6: Drill the 8 Holes Using Axis Substitution Toolpath	80
Step 7: Axis Substitution About Y Axis - Contour Toolpath	84
Step 8: Machine the top Slot - Contour Toolpath	92
Step 9: Transform-Rotate the Contour Toolpath	98
Step 10: Post the File to Generate the NC Code	102
Tutorial #2 Quiz	104
Tutorial 3	105
Overview Of Steps Taken To Create The Final Part:	106
Step 1: Select the Geometry File	107
Step 2: Select the Machine and Set Up the Stock	108
Step 3: Axis Substitution about Y Axis - Dynamic Mill Toolpath	112
Step 4: Backplot The Toolpath	121
Step 5: Verify the Toolpath	122
Step 6: Axis Substitution about Y Axis - Contour Toolpath	123
Step 7: change the walls color using Change Face command	130
Step 8: Finish The Walls - Swarf Milling	133
Step 9: Drill the Holes	141
Step 10: Define a New Tool Plane to Machine one slot	150
Step 11: Machine the Slot in the New Plane	155
Step 12: Transform - Rotate to Machine the Other Slots	162
Step 13: Define a New Tool Plane using the Gnomon	167
Step 14: Machine the Right Side Flat area	171
Step 15: Transform - Rotate to Machine The other flat	177
Step 16: Post the File to Generate the NC Code	180
Step 17: Save the Updated MCAM File	181
Tutorial #3 Quiz	183
Tutorial 4	185
Overview Of Steps Taken To Create The Final Part:	186

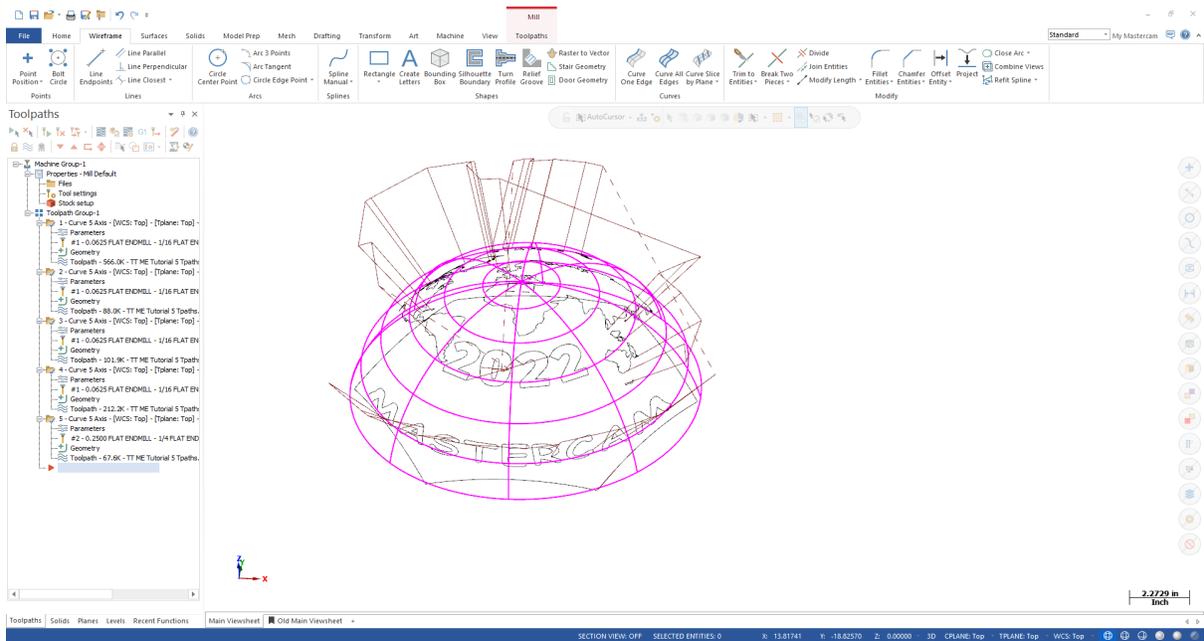
Step 1: Select the Geometry File	187
Step 2: Select the Machine and Set Up the Stock	187
Step 3: Rotary 4-Axis with Rotary Cut	190
Step 4: Rotary 4-Axis With Axial Cut	202
Step 5: Post the File to Generate the NC Code	212
Step 6: Save the Updated MCAM File	212
Tutorial #5 Quiz	214
Tutorial 5	215
Overview Of Steps Taken To Create The Final Part:	216
Step 1: Select the Geometry File	217
Step 2: Select the Machine and Set Up the Stock	218
Step 3: Curve With Tool Axis Control Oriented To A Point	222
Step 4: Curve - Tool Axis Control From A Point	232
Step 5: Curve - Tool Axis Control Normal to A Plane	242
Step 6: Curve - Tool Axis Control Towards a Chain	251
Step 7: Curve - Tool Axis Control to Lines	260
Step 8: Post the File to Generate the NC Code	269
Step 9: Save the Updated MCAM File	269
Tutorial #5 Quiz	271
Tutorial 6	273
Overview Of Steps Taken To Create The Final Part:	274
Step 1: Select the Geometry File	275
Step 2: Set Up the Stock	275
Step 3: Verify the 3D High Speed Toolpaths	278
Step 4: Create a Stock Model	279
Step 5: Swarf Toolpath with Walls Defined From the Chains	281
Step 6: Swarf Toolpath With Walls Defined From Surfaces	295
Step 7: Post The File To Generate The NC Code	301
Step 8: Save the Updated MCAM File	301
Tutorial #6 Quiz	303

Tutorial 7	305
Overview Of Steps Taken To Create The Final Part:	306
Step 1: Select the Geometry File	307
Step 2: Set the Tool Settings	308
Step 3: Create a Stock Model	309
Step 4: Helix bore the 0.531" diameter Holes	314
Step 5: Helix Bore the 0.75" diameter Holes	324
Step 6: Helix Bore the 0.7647" diameter holes	329
Step 7: Post the File to Generate the NC Code	331
Step 8: Save the Updated MCAM File	331
Tutorial #7 Quiz	333
Tutorial 8	335
Overview Of Steps Taken To Create The Final Part:	336
Step 1: Select the Geometry File	337
Step 2: Set The tool Settings	338
Step 3: Flow 5 axis - Tool Axis Control Parallel To Lines	339
Step 4: Machine Simulation Flow 5 axis	350
Step 5: Flow 5 axis - Tool Axis Oriented To A Point	356
Step 6: Transform - Rotate to Finish All Blades	369
Step 7: Post the File to Generate the NC Code	371
Step 8: Save the Updated MCAM File	371
Tutorial #8 Quiz	373
Tutorial 9	375
Overview Of Steps Taken To Create The Final Part:	376
Step 1: Select the Geometry File	377
Step 2: Set the Tool Parameters in the Tool Settings page	378
Step 3: Multisurface 5 axis - Rough the Cavity	379
Step 4: Machine Simulation - Multisurface 5 axis	391
Step 5: Multisurface 5 axis - Finish the Part	395
Step 6: Post the File to Generate the NC Code	403

Step 7: Save the Updated MCAM File	403
Tutorial #9 Quiz	405
Tutorial 10	407
Overview Of Steps Taken To Create The Final Part:	408
Step 1: Select the Geometry File	409
Step 2: Select the Machine and Set Up the Stock	410
Step 3: 3 Axis Deburr Toolpath	411
Step 4: 5-Axis Deburr Toolpath	420
Step 5: Post the File to Generate the NC Code	426
Step 6: Save the Updated MCAM File	426
Tutorial #10 Quiz	428
Quiz answers	429



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.

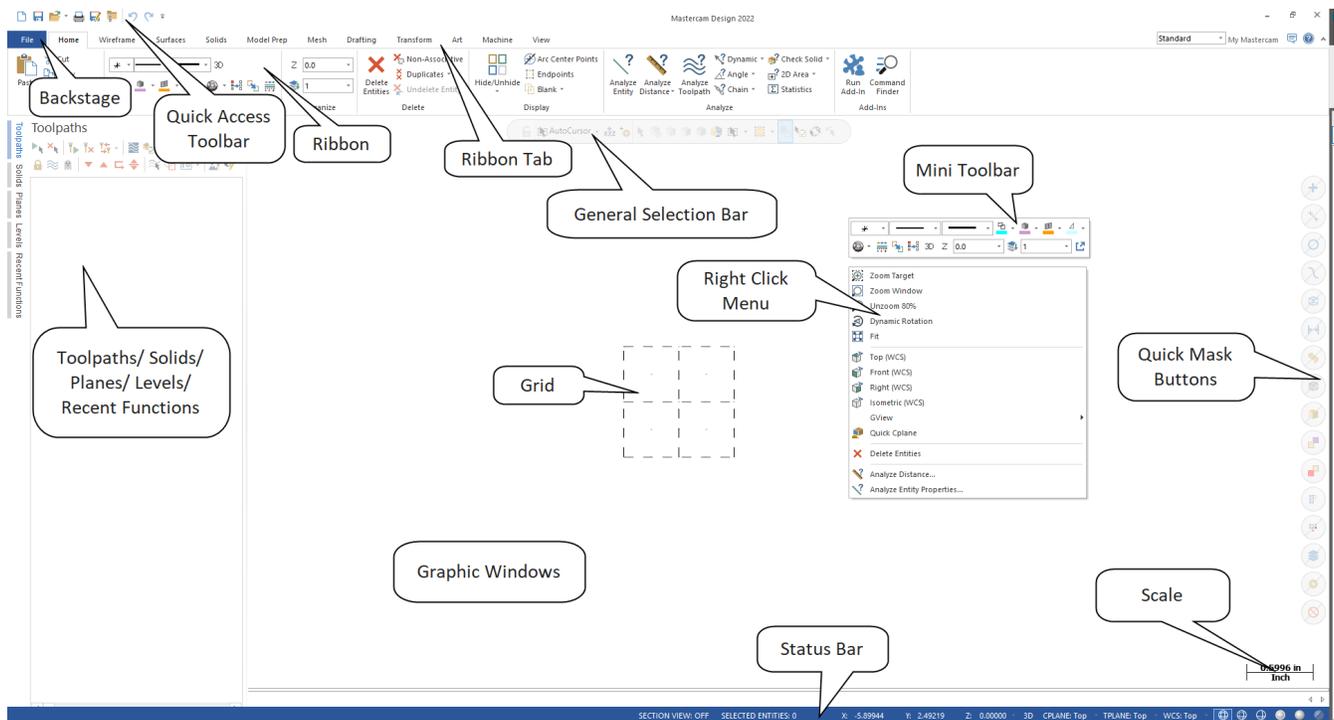
1.3 For Windows 10

- 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/C plane information.

- Press **Enter** to continue.
- Select the **AutoCursor Fast Point** icon again and enter in the coordinates of the second endpoint and then press **Enter**.

3.4 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. 
- To undo the last command, from the **QAT (Quick Access Toolbar)** 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. 

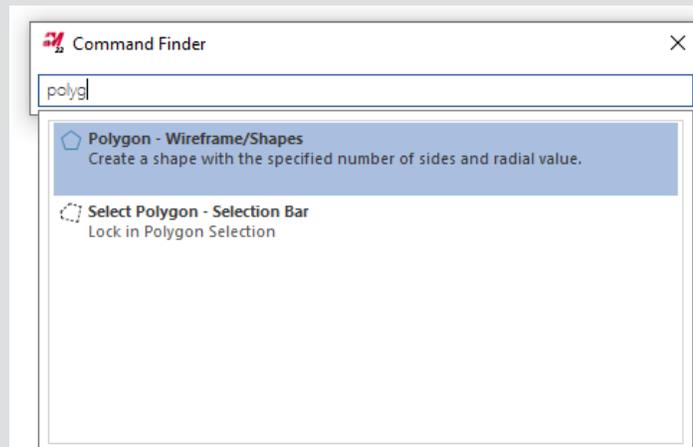
3.5 Function Prompt

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

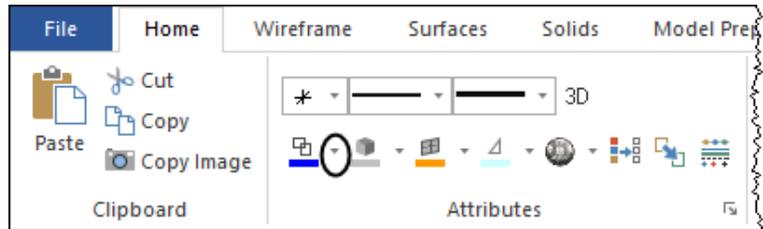
Point Style	Displays and sets the system's point style.
Line Style	Displays and sets the system's line style.
Line Width	Displays and sets the current system's line width.
Color	Assigns the current color to wireframe, solid and surface entities. To change the current color, click in the specific color 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

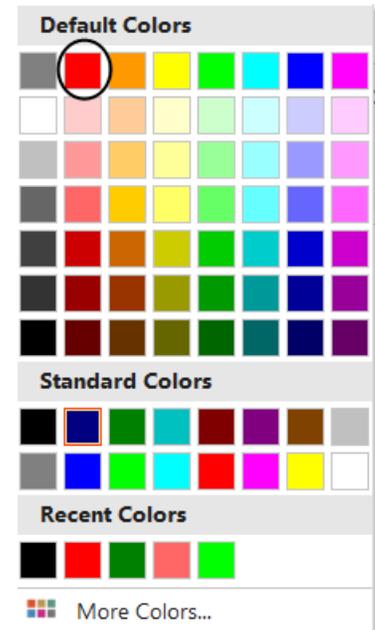
Z Depth	Sets the current construction depth. To set this, click the drop down arrow and pick one from the most recently used list or click the Z: label and pick a point in the graphics window to use the Z depth values based on the selected entity.
Level	Sets the main level you want to work with in the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

4.3 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.

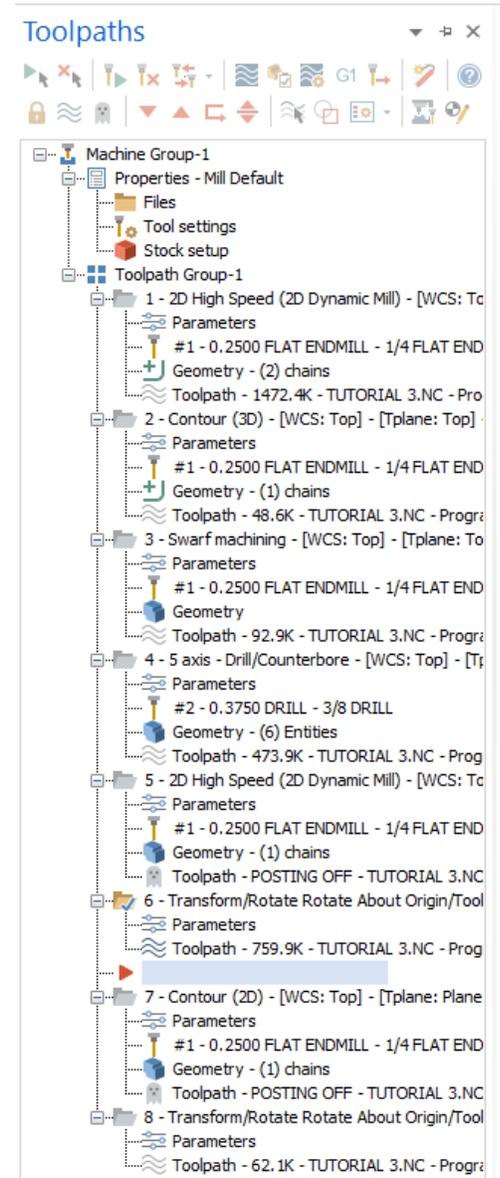


Note: Any geometry on your screen will remain in the previous system color. This change will only affect the geometry you create going forward.
To change the color of existing geometry, select the entities first and then click on the drop down arrow next to the **Wireframe Color** and select the desired color. The same method can be applied for any other attribute that you want to set or change.

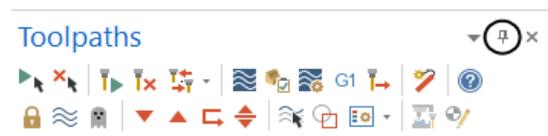
STEP 5: MANAGER PANELS

5.1 The Toolpaths Manager

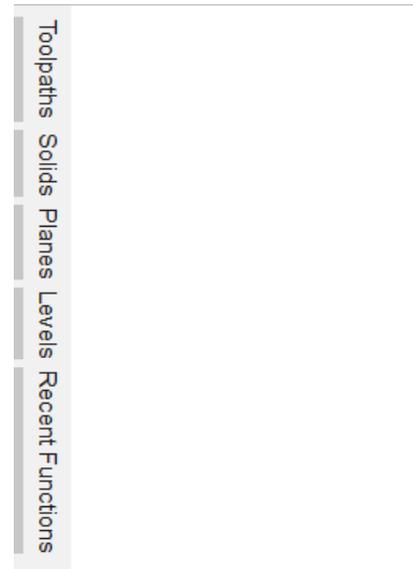
The **Toolpaths Manager** displays all the operations for the current part. You can sort, edit, regenerate, verify and post any operation as shown. For more information on the **Toolpaths Manager**, please click on the **Help** icon.



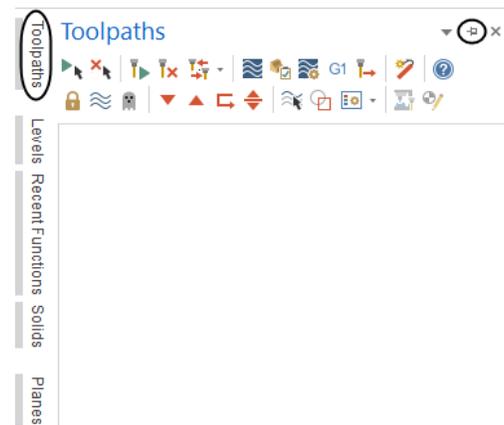
- The **Toolpaths Manager**, **Solids Manager**, or **Planes Manager** can be hidden to gain more space in the graphics area for creating geometry. Use **Auto Hide** icon to close all **Toolpaths**, **Solids**, **Planes** and **Levels Manager** panels.



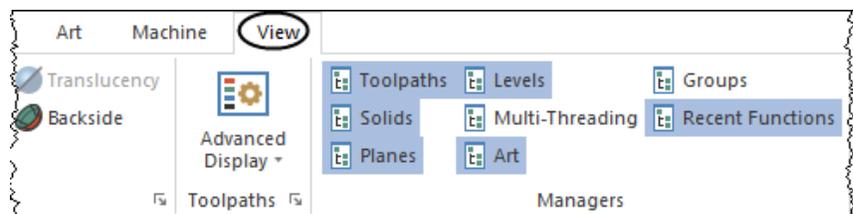
- The panels will be hidden to the left of the graphics window as shown or at the bottom of the manager as shown previously.



- 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.

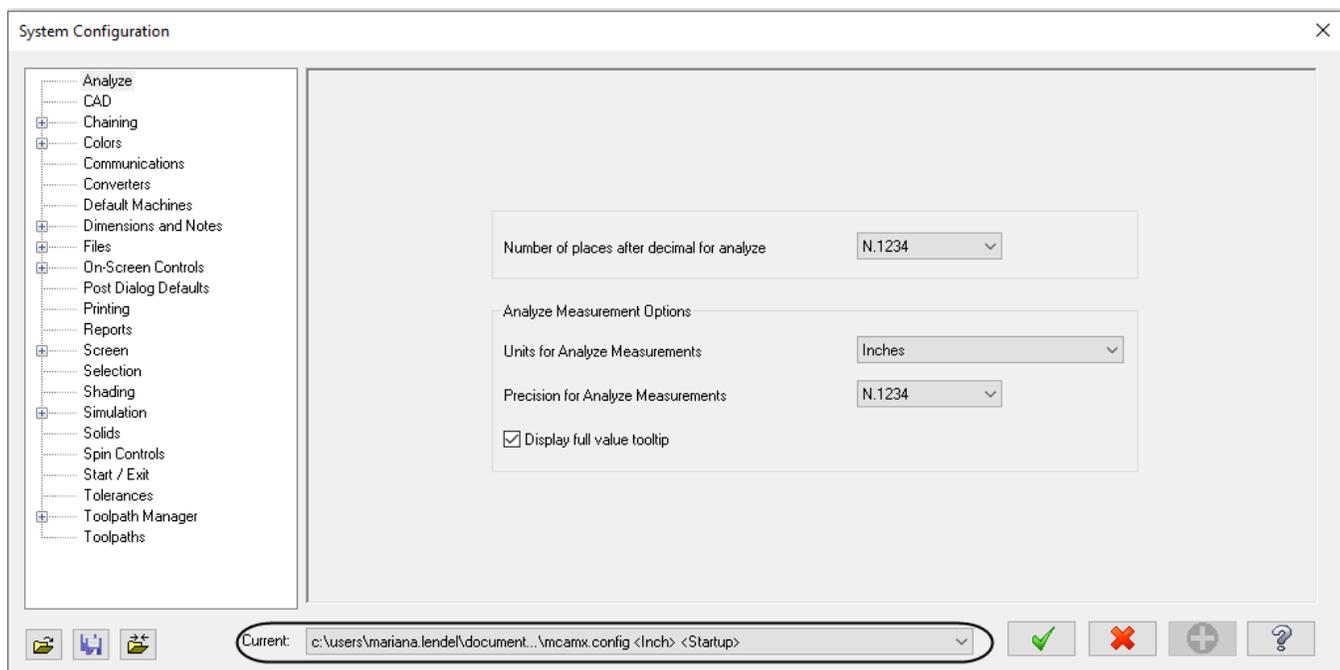
6.1 Setting Mastercam to inch for the current session only

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

File

■ Configuration.

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



- 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.

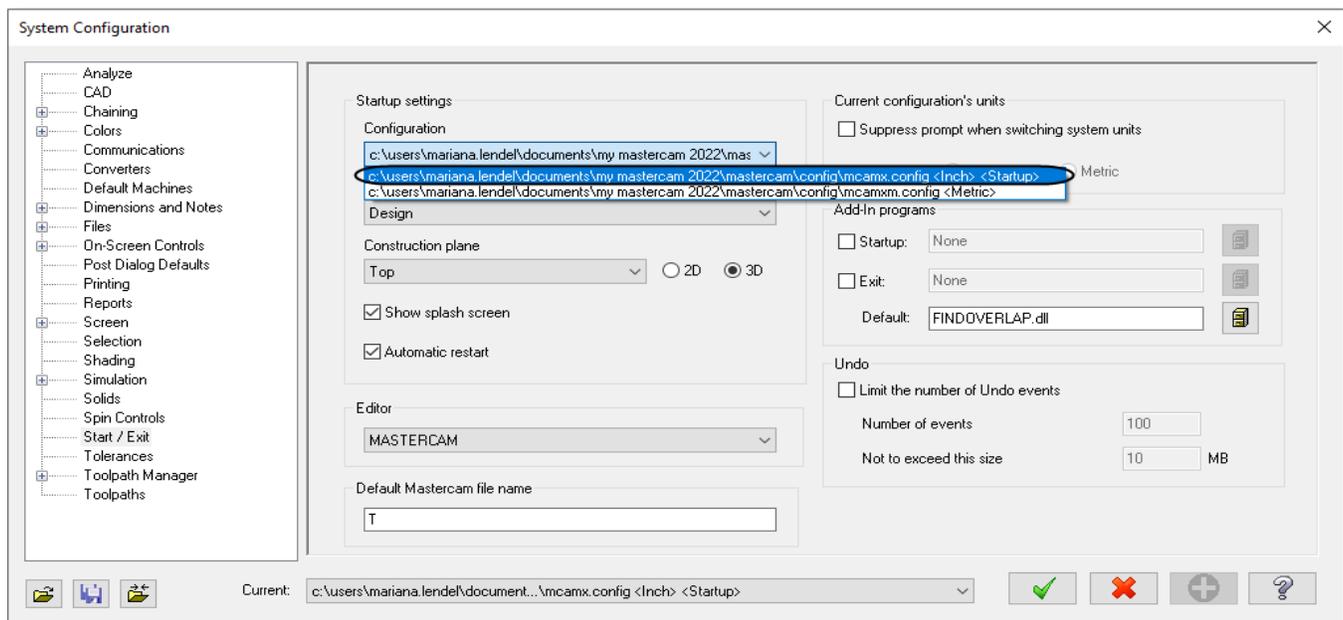
6.2 Setting Mastercam to imperial as a default

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

File

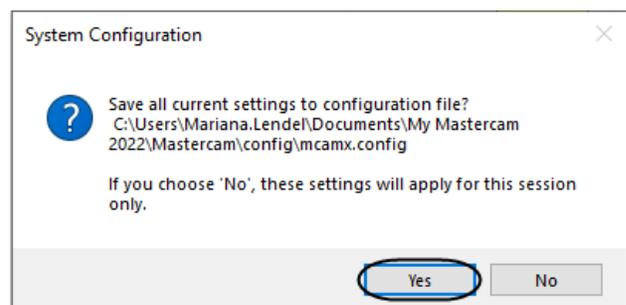
■ Configuration.

- 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**.



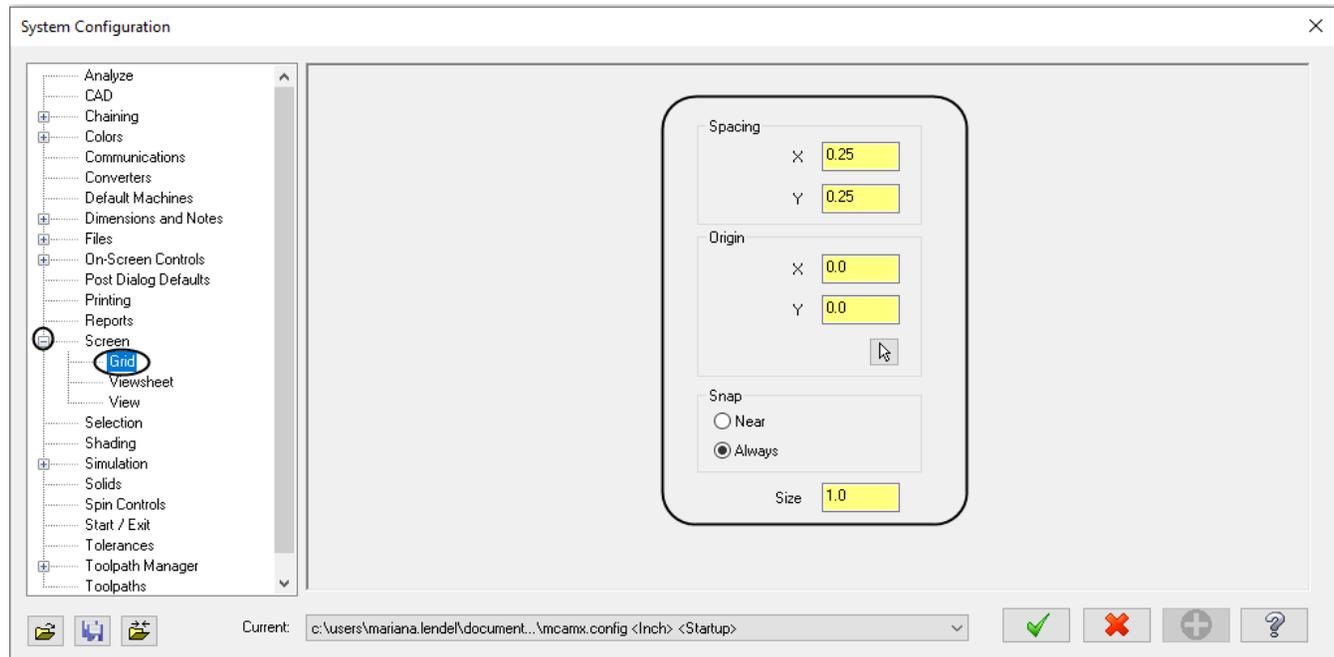
STEP 7: SET THE GRID

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

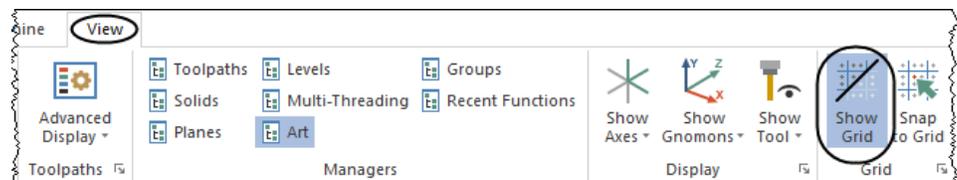
■ Configuration.

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



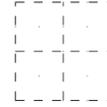
- In **Grid Settings**, change the **Spacing** to **X = 0.25** and **Y = 0.25**.
- Set the **Size** to **1.0**.
- Choose the **OK** button to exit. 
- Select the **Yes** button to save the settings in the **System Configuration**.

- To see the **Grid** in the graphics window, from the **View** tab, enable **Show Grid** as shown.

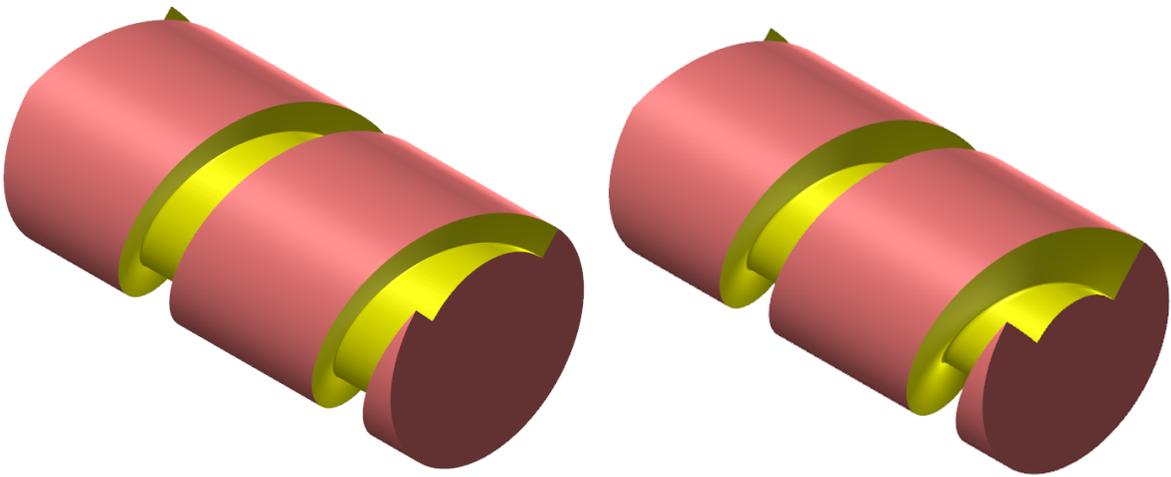




- The grid should look as shown.



Tutorial 1



OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

From Drawing to CAD Model:

- The student should examine the drawing on the following page to understand what part is being created in the tutorial.
- From the drawing we can decide how to go about creating the geometry in Mastercam.

Create the 2D CAD Model used to generate Toolpaths:

- The student will create a line in the Top Plane needed to create the toolpaths.
- Two files will be created: one for a cylindrical helix, another for a conical helix.
- Geometry creation commands such as Create Line Endpoints (horizontal and vertical), and the Delete entities command will be used.

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 contour toolpath will be created to machine a cylindrical helix using Axis Substitution.
- Another contour toolpath will be created to machine a conical helix using Axis Substitution.

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.

Post Process the file to generate the G-code:

- The student will then post process the file to obtain an NC file containing the necessary code for the machine.



This tutorial takes approximately 45 minutes to complete.

This tutorial covers Mastercam's 4 - 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 4-Axis post processors included with Mastercam due to the variation in Multiaxis machine configurations. Contact your Mastercam reseller to request post processor services such as Multiaxis 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 box.

Your post may also prompt for tool gauge lengths.

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

STEP 1: CREATE THE GEOMETRY OF THE CYLINDRICAL HELIX

Cylindrical helix parameters:

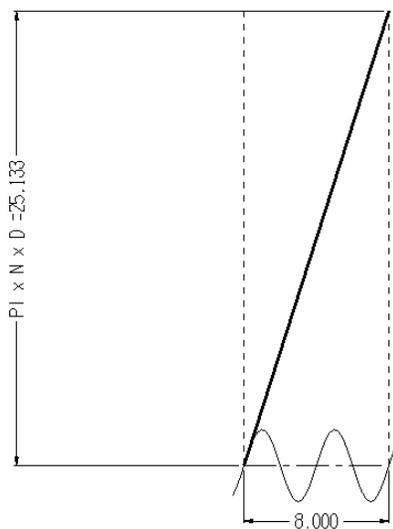
Diameter, D = 4".

Number of spirals, n = 2.

Helix length, L = 8".

ALL DIMENSIONS IN INCHES

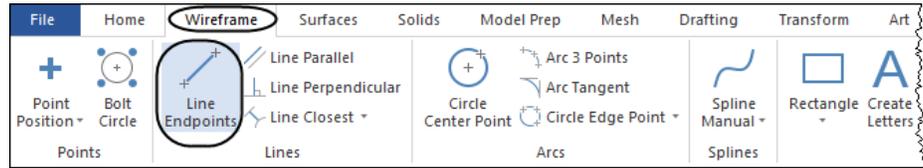
PI =3.141592653
N = 2 (Number of spirals)
D = 4 (Cylinder diameter)



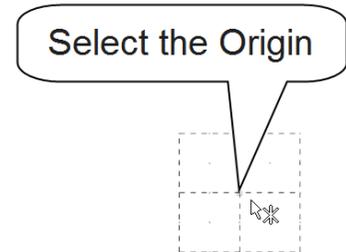
1.1 Create the Horizontal Line

Wireframe

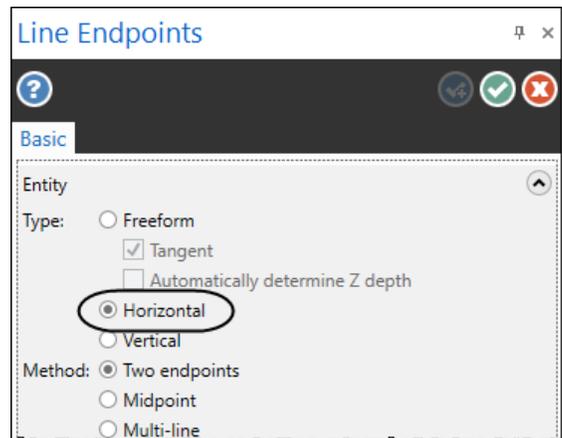
- From the **Lines** group, select **Line Endpoints** as shown.



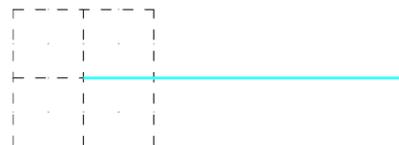
- [Specify the first endpoint]: As by moving the cursor to the center of the grid as shown.



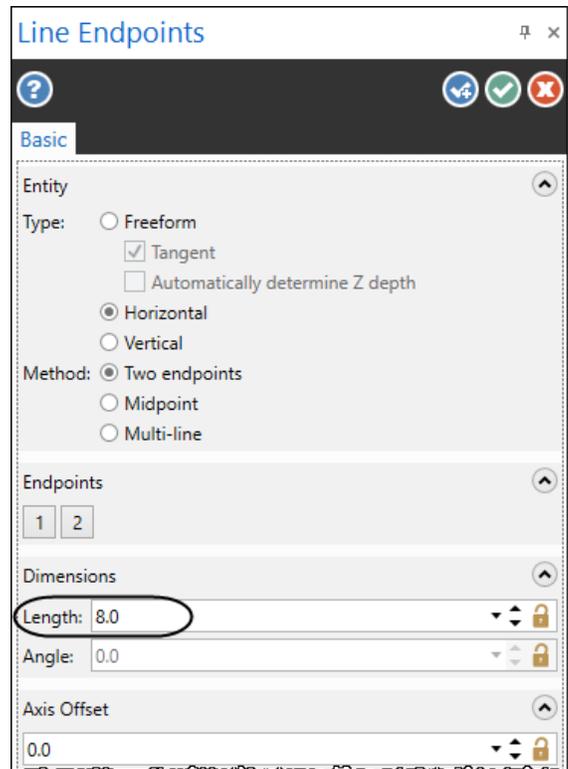
- From the **Line Endpoints** panel, enable **Horizontal**.



- [Specify the second endpoint]: Sketch the line to the right of the **Origin**.



- In the panel, enter the **Length = 8.0** as shown.



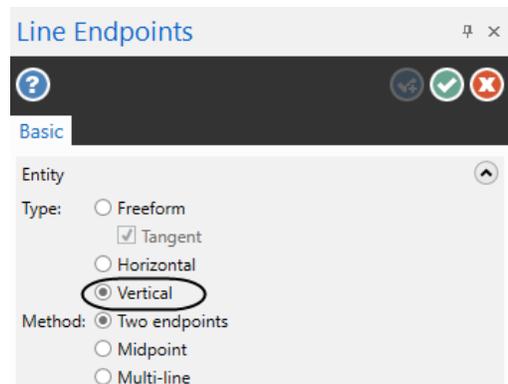
- Press the **Enter** key.
- Select **OK and Create New Operation** to continue in the same command.
- Press **Alt + F1** to fit the geometry to the screen.

1.2 Create the Vertical Line

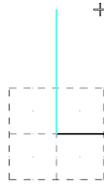
- [Specify the first endpoint]: Select the **Origin**.



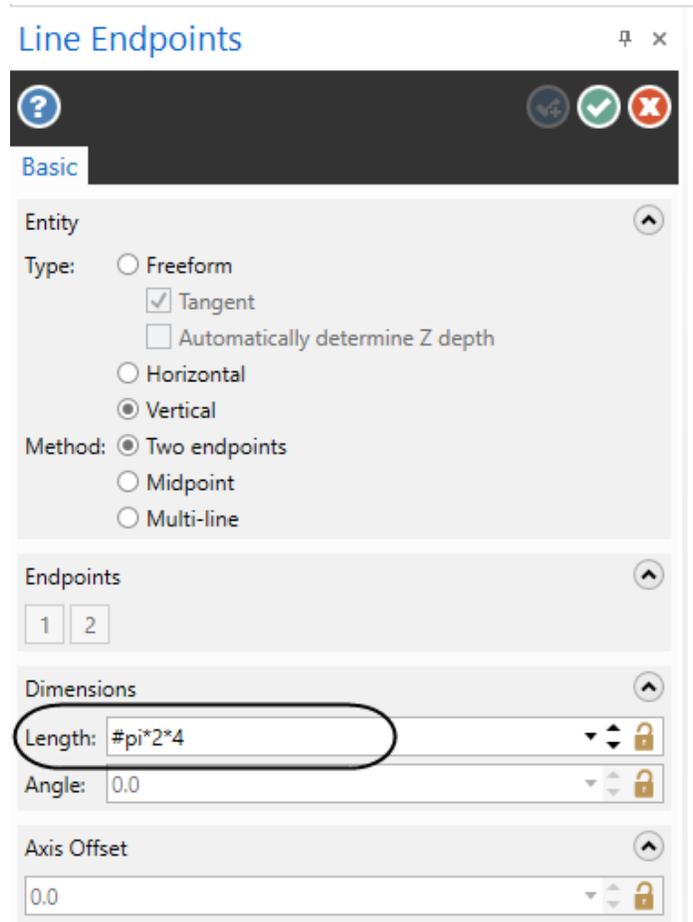
- Enable **Vertical**.



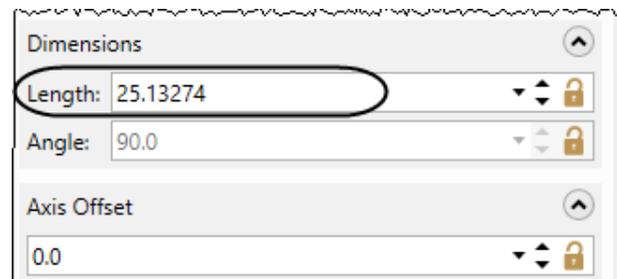
- [Specify the second endpoint]:
Sketch the line above the **Origin**.



- Click in the **Length** value field and enter **#pi*2*4** as shown.



- Press **Enter**.
- The value should look as shown.



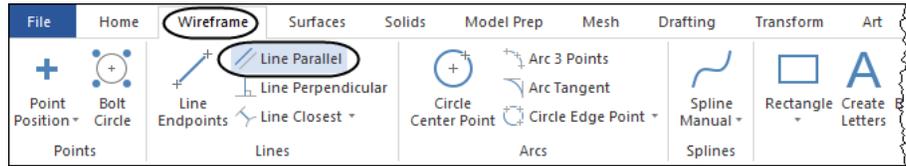
- Select the **OK** button to exit the command.
- Press **Alt + F1** to fit the drawing to the screen.



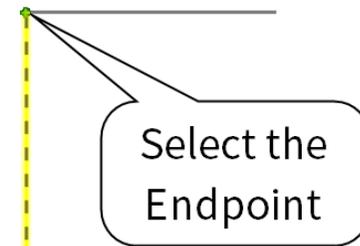
1.3 Create Line Parallel

Wireframe

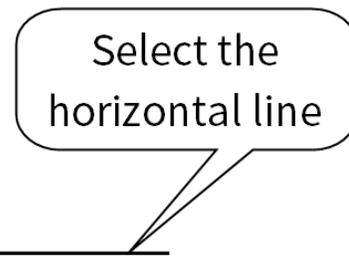
- From the **Lines** group, select **Line parallel** as shown.



- [Select a line]: Select the horizontal line.



- [Select the point to place a parallel line through]: Select the upper **Endpoint** of the vertical line as shown.

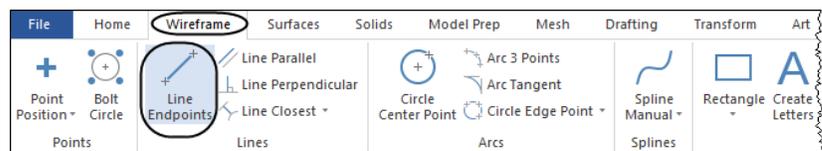


- Select the **OK** button to exit the command.

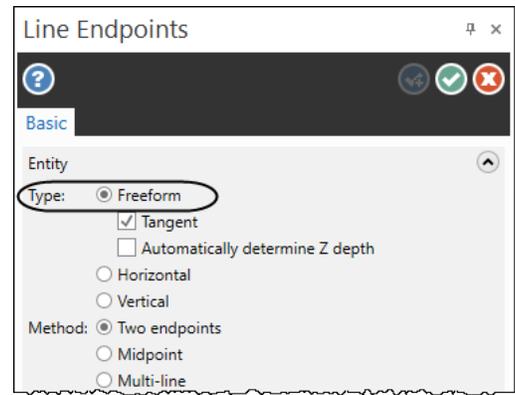
1.4 Create the Hypotenuse of the Right Angle Triangle

Wireframe

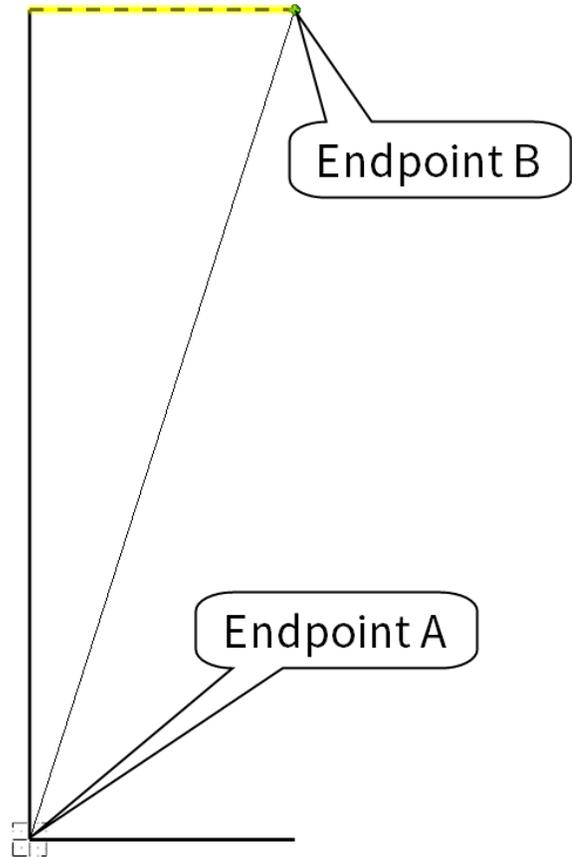
- From the **Lines** group, select **Line Endpoints** as shown.



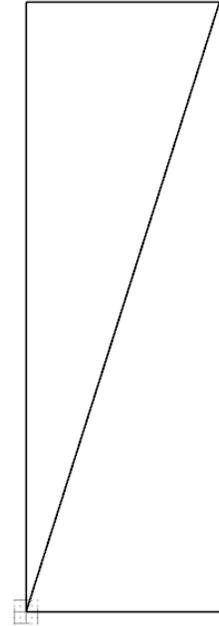
- Enable **Freeform** and select the endpoints of the lines.



- [Specify the first endpoint]: Select **Endpoint A** as shown.
- [Specify the second endpoint]: Select **Endpoint B** as shown.



- The geometry should look as shown.

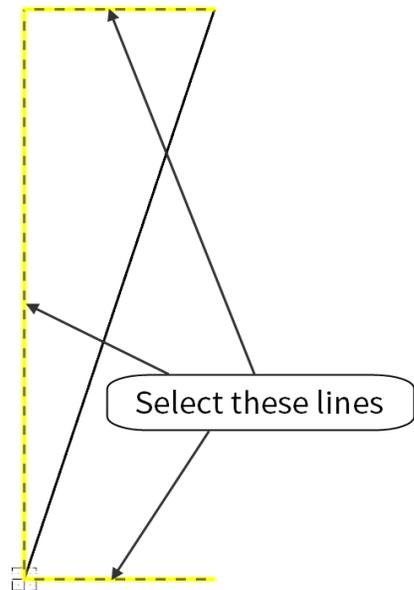


- Select the **OK** button to exit the command.



1.5 Delete the two Horizontal Lines and the Vertical Line

- Click on the horizontal lines and on the vertical line as shown.



- Press **Delete** on the keyboard.

- The geometry should look as shown.



1.6 Save the File

File

- **Save As.**

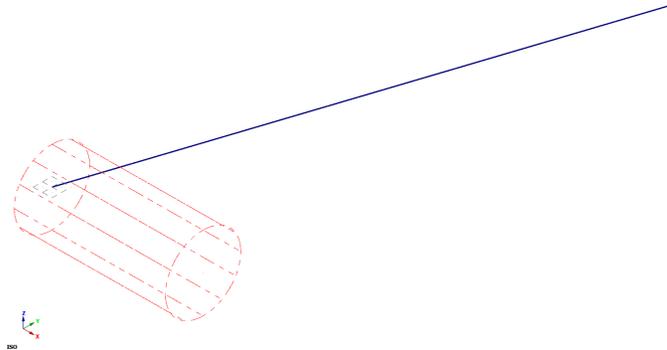


- Click on the **Browse** icon as shown.
- Find a location on the computer to save your file. File name: "TUTORIAL1_CYLINDRICALHELIX.MCAM".

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.

Step Preview:

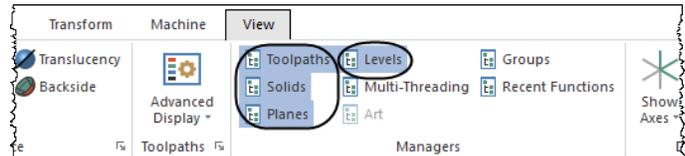


Note: If you already have the default machine in the **Toolpaths Manager**, do not select another machine.

2.1 Open the Toolpaths Manager

View

- In the Managers group, make sure that the following managers are selected.



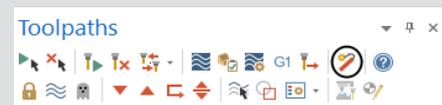
- If needed, from the lower left corner of the interface click on the **Toolpaths** tab as shown.
- Make sure that the **Toolpaths Manager** is pinned as shown.



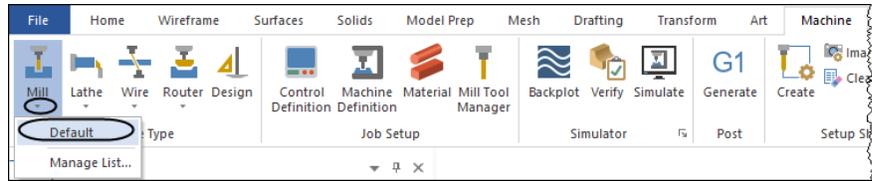
2.2 Select the Machine

Machine

Note: If a machine is already selected in the **Toolpaths manager**, from **Machine Type** select **Design** and then click on the **Delete all operations, groups and tools** icon from the **Toolpaths Manager**.



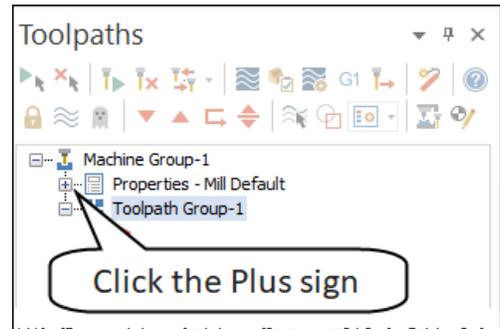
- From the **Machine Type** group, select the drop down arrow below **Mill**.
- Select the **Default**.



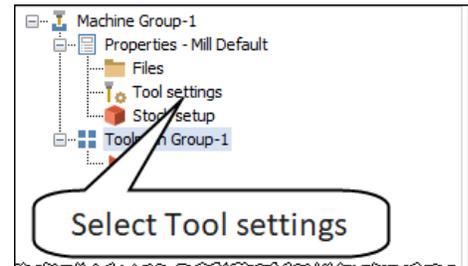
Note: Once you select the **Mill Default**, the **Ribbon bar** changes to reflect the toolpaths that could be used with **Mill Default**.

2.3 Set the Tool Parameters in the Tool Settings

- Select the plus sign in front of **Properties** in the **Toolpaths Manager** to expand the **Machine Group Properties** as shown.



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



- Change the parameters to match the image.

Machine Group Properties

Files Tool Settings Stock Setup

Default program number 1

Feed Calculation

From tool

From material

From defaults

User defined

Spindle speed 5000.0

Feed rate 50.0

Retract rate 125.0

Plunge rate 25.0

Adjust feed on arc move

Minimum arc feed 5.0

Toolpath Configuration

Assign tool numbers sequentially

Warn of duplicate tool numbers

Use tool's step, peck, coolant

Search tool library when entering a tool number

Advanced options

Override defaults with modal values

Clearance height

Retract height

Feed plane

Sequence number

Start 100.0

Increment 10.0

Material

ALUMINUM inch - 2024 Edit... Select...

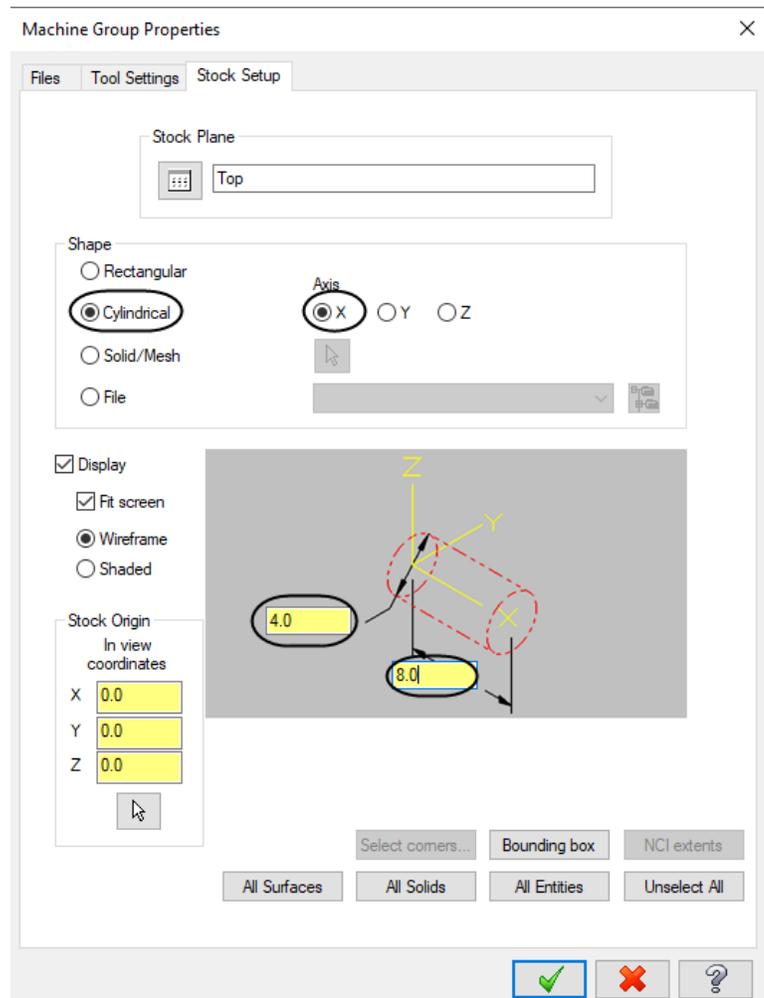
✓ ✗ ?

Default program number is used to enter a number if your machine 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 receive 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 to **From tool** uses the feed rate, plunge rate, retract rate and spindle speed from the tool definition.

2.4 Set the Stock Shape and Size

- Select the **Stock Setup** tab to define the stock.
- Set the **Stock Shape** to **Cylindrical** and the **Axis** to **X**.
- Set the **Diameter** to **4.0** and the **Length** to **8.0** as shown.



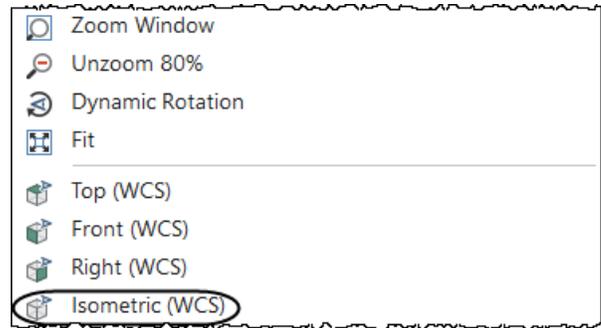
The **Stock Origin** values adjust the positioning of the stock, ensuring that you have an equal amount of extra stock around the finished part.

Display options allow you to set the stock as **Wireframe** and to fit the stock to the screen. (Fit screen)

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

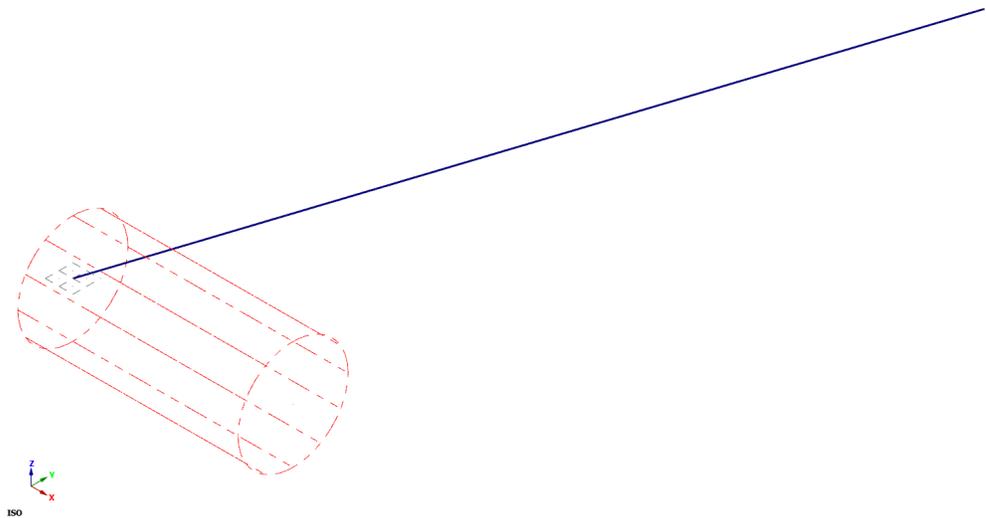


- Right mouse click in the graphics window and select the **Isometric (WCS)** icon as shown.



- Press **Alt + F1** to fit the drawing to the screen.
- Scroll down the mouse wheel to see the stock.

- The stock should look as shown.



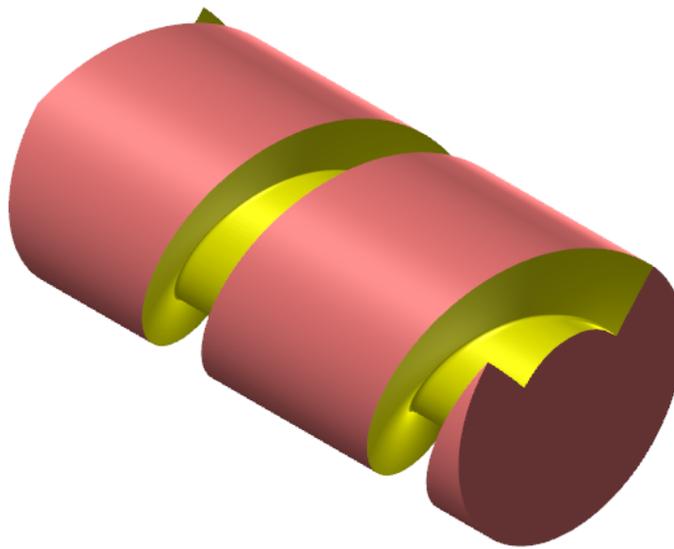
Note: Remember that the stock is not geometry and cannot be selected.

STEP 3: CYLINDRICAL HELIX - AXIS SUBSTITUTION ABOUT Y

Axis substitution is used to wrap a toolpath around a cylinder. The geometry can be either flat or already properly oriented in 3D space (select the **Unroll** option if this is so). The toolpath replaces the X or Y axis with a rotary axis, known as A-Axis or B-Axis. Using this option, the part will be rotated about the rotating axis while the tool moves perpendicular to the axis of rotation.

Rotary diameter sets the size of the cylinder that the toolpath is rotating about. It is used mainly for toolpaths on cylinders. To machine the helix you will use a contour toolpath with no cutter compensation.

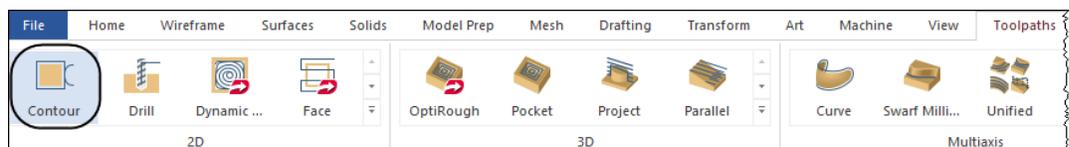
Toolpath Preview:



Note: Before you can create rotary axis motion, you need to properly configure the rotary axis components in your machine definition. You will only be able to select rotary axis options which are supported by your machine definition

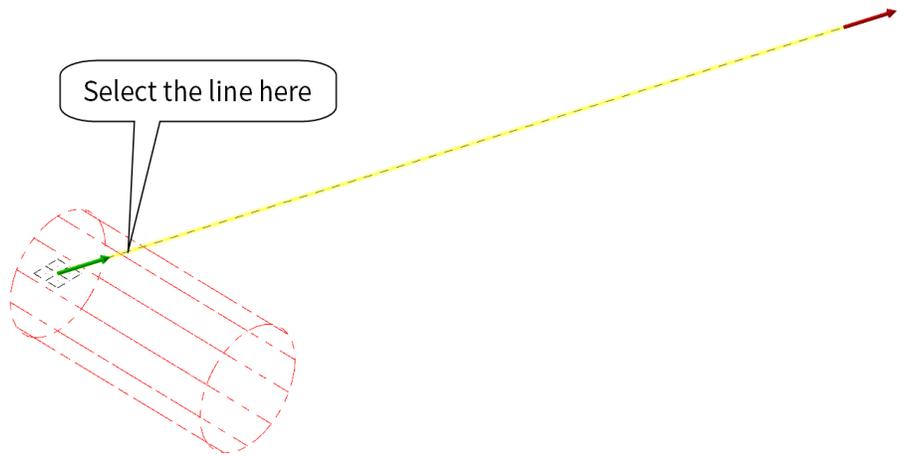
Toolpaths

- From the **2D** group, select **Contour** as shown.



- Leave the default settings in the **Chaining** dialog box.

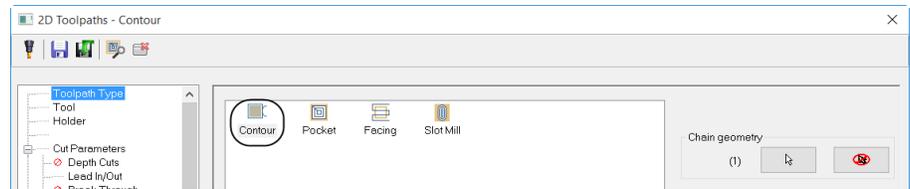
- [Select Contour chain 1]: Select the line as shown. Make sure the direction is correct. If the direction is incorrect, select the reverse icon. 



- Select the **OK** button to exit the **Chaining** dialog box. 

3.1 Toolpath Type

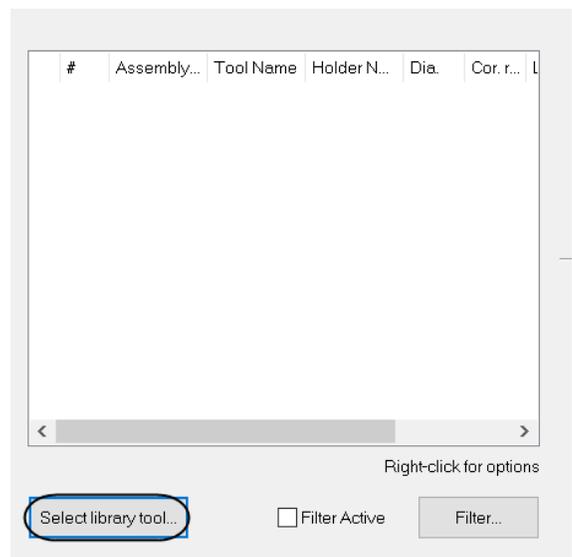
- In the **Toolpath Type** page, the **Contour** toolpath should be already selected as shown.



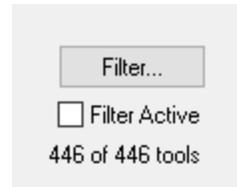
3.2 Set the Tool page parameters

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.

- Select **Tool** from the **Tree View list**.
- Click on **Select library tool** button.

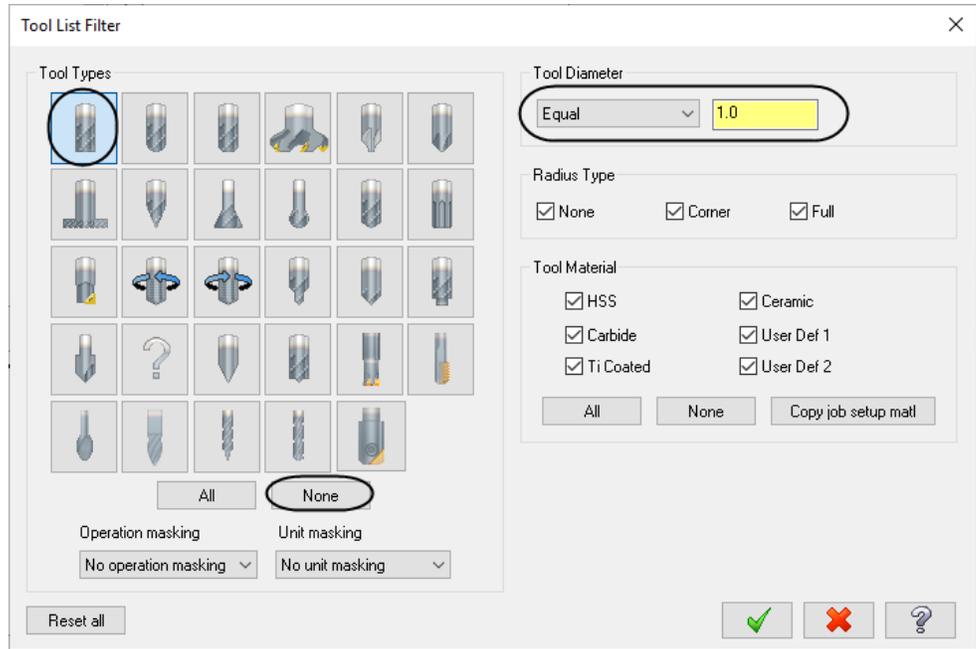


- Select the **Filter** button as shown.



- Select the **None** button to deselect any previous tool selection as shown in the image below.
- Select the **Flat Endmill** in the **Tool Types** list.
- Select the drop down arrow in the **Tool Diameter** field and select **Equal**.

- Enter **1.0** in the **Tool Diameter** value box as shown in the image.



- Select the **OK** button to exit the **Tool List Filter** dialog box.

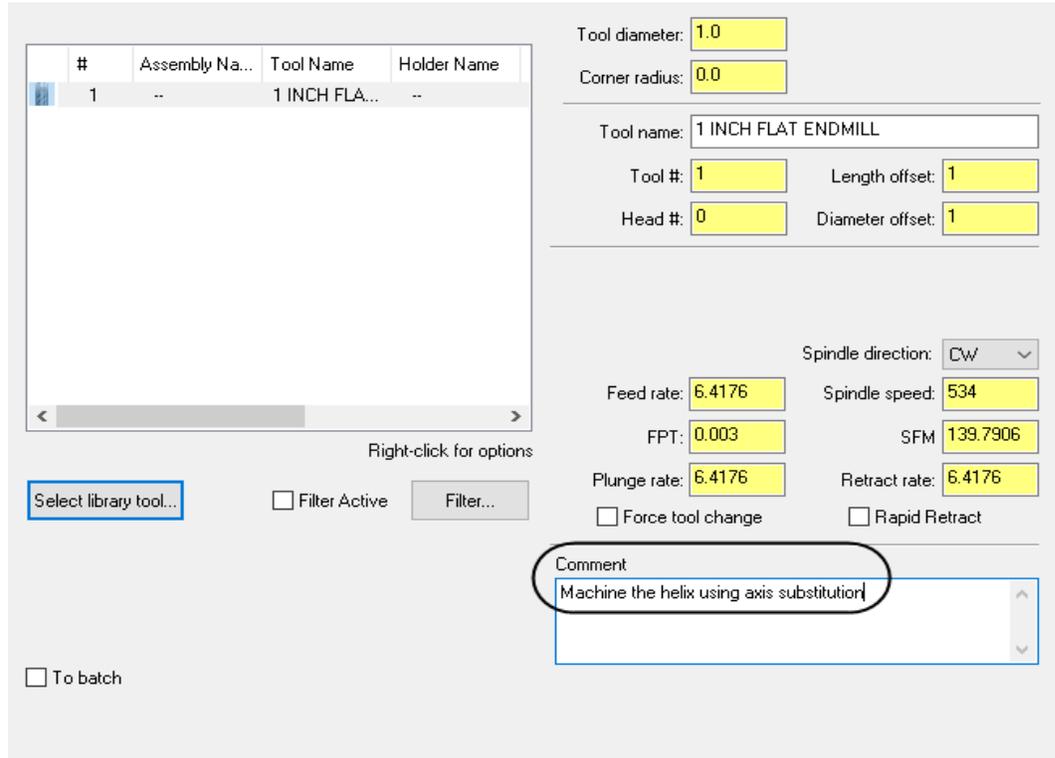
- Make sure that the tool in the **Tool Selection** window is highlighted. Otherwise select it.

#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	Length	# Flut...	Type	Rad...
297	-	1 INCH F...	-	1.0	0.0	2.0	4	End...	None

- Select the **OK** button to exit from the **Tool Selection** dialog box.

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

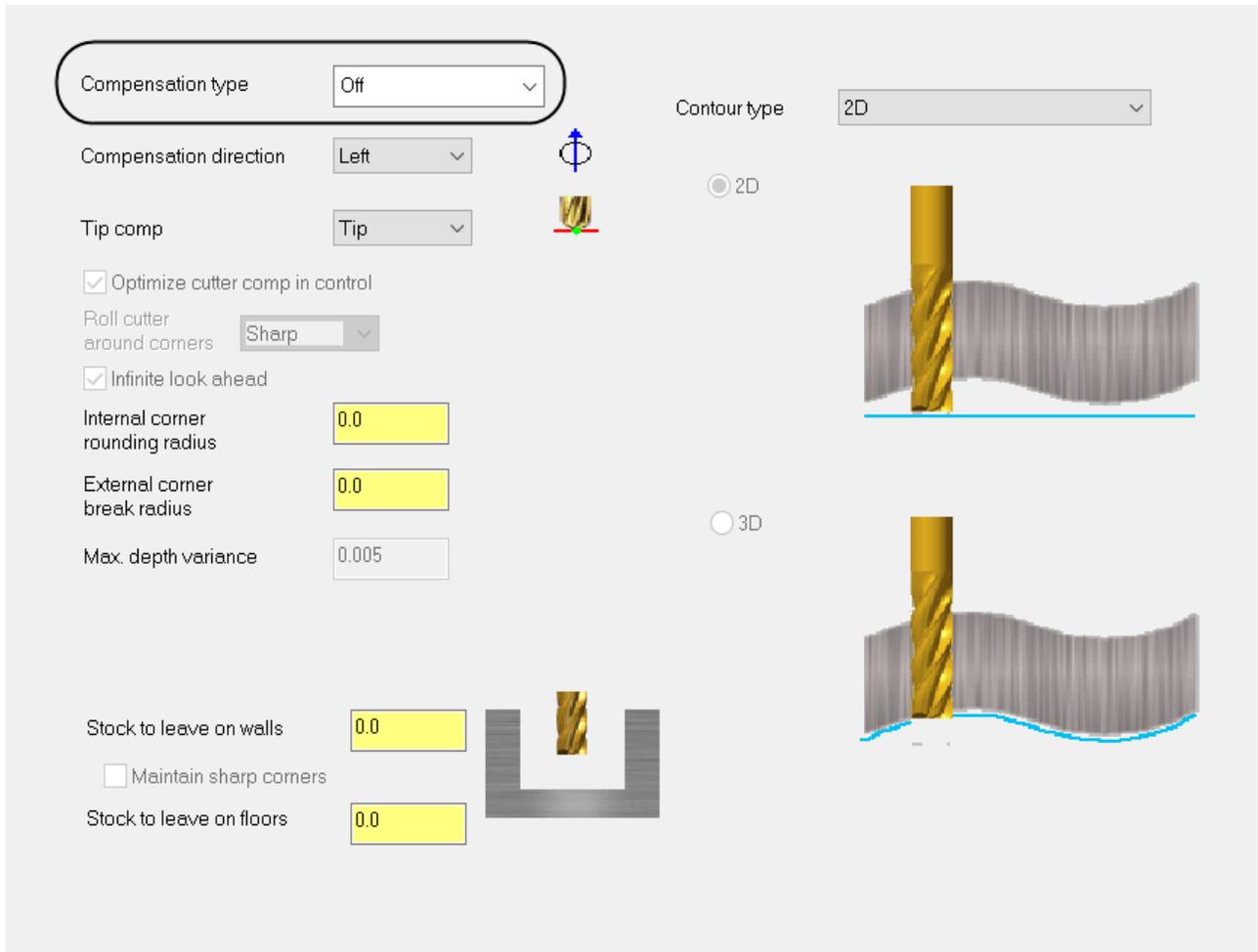
- The **Tool** page should look as shown.



Note: The feeds and spindle speed are based on the tool definition. You can overwrite them with the feeds and speed that you want to use. You can also let Mastercam calculate the feeds and speed using the Feed speed calculator. During this training tutorial, we will be using the default values for the tools.

3.3 Cut Parameters

- From the **Tree View list**, select **Cut Parameters**.
- Change the **Compensation type** to **Off** as shown.



Note: The cutter compensation is turned off as you want the tool to follow the chain with the center of the tool.

3.4 Depth Cuts

- From the **Tree View list**, select the **Depth Cuts** page.

- Enable **Depth Cuts**, set the **Max rough step** to **0.25** and set the **Finish step** to **0.0** as shown.

