Mastercam® 2020

IMPERIAL | TRAINING TUTORIAL SERIES

HLE DOWNLOAD
INSTRUCTIONS INSIDE
2020

MULTIAXIS ESSENTIALS

Mastercam 2020

MULTIAXIS ESSENTIALS TRAINING TUTORIAL

To order more books:

Call 1-800-529-5517 or

Visit www.emastercam.com or

Contact your Mastercam dealer

Mastercam 2020 Multiaxis Essentials Training Tutorial

Copyright: 1998 - 2020 In-House Solutions Inc. All rights reserved

Software: Mastercam 2020

Authors: Mariana Lendel

ISBN: 978-1-77146-841-1

Date: August 21, 2019

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 distributer, 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.

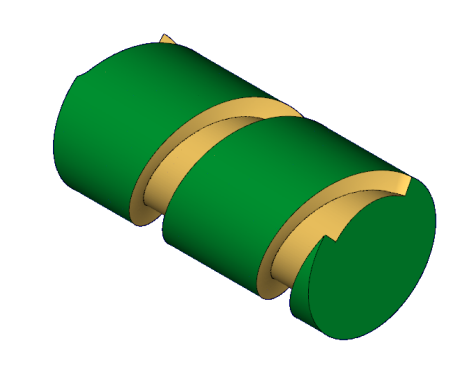
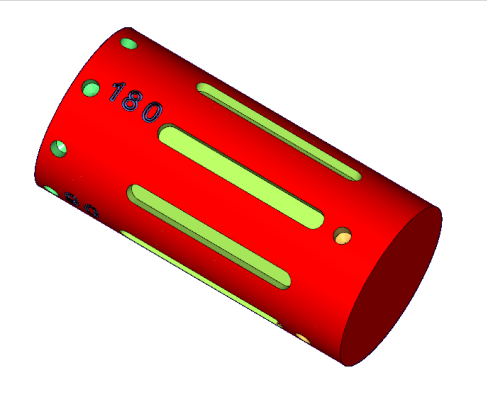
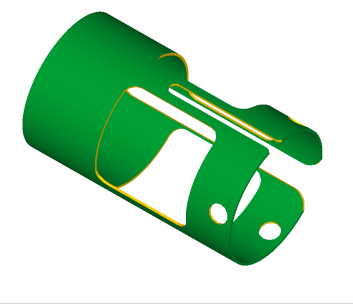
Trademarks

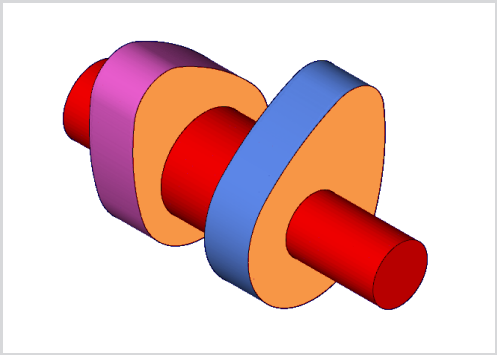
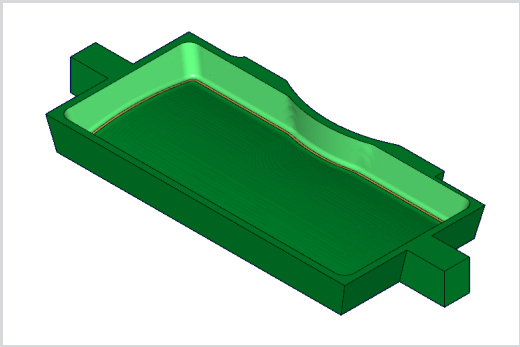
Mastercam is a registered trademark of CNC Software, Inc.

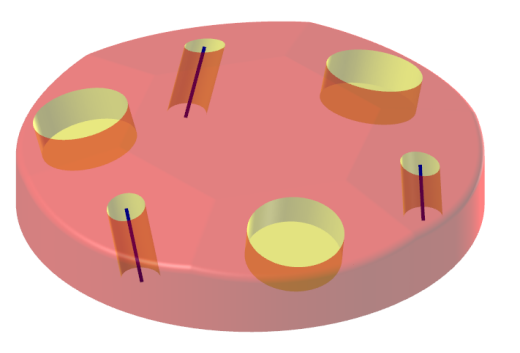
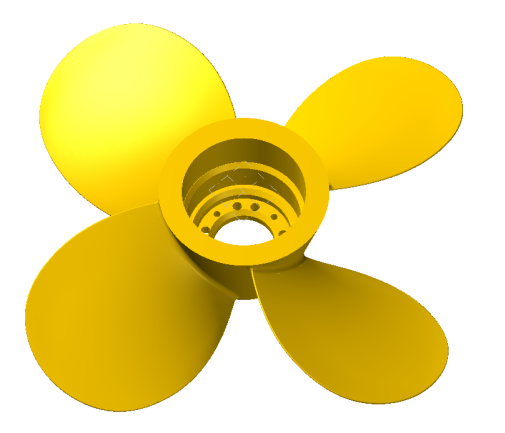
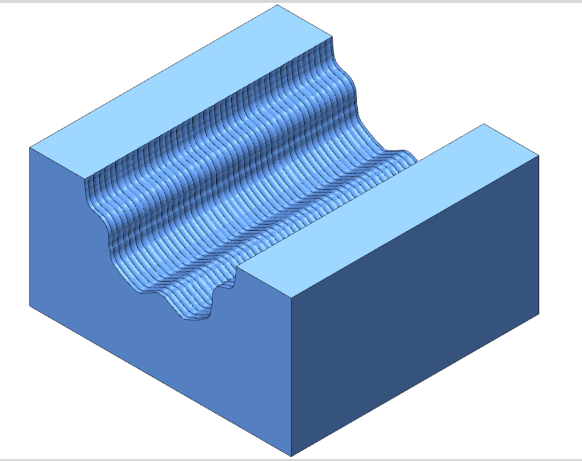
Microsoft, the Microsoft logo, are registered trademarks of Microsoft Corporation;

Windows 10 is a registered trademarks of Microsoft Corporation.

Multiaxis Essentials Projects

Tutorial	Toolpath Creation
#1 	Contour With Axis Substitution.
#2 	Drilling With Axis Substitution. Drilling with Rotatry Axis Positioning. Contour With Axis Substitution. Transform Rotate.
#3 	Contour With Axis Substitution. Contour Indexing Using Planes.

Tutorial	Toolpath Creation
#4 	Rotary 4-Axis.
#5 	Curve 5 -Axis.
#6 	Swarf 5-Axis.

Tutorial	Toolpath Creation
#7 	Drill 5-Axis. Circle Mill 5-Axis.
#8 	Flow 5-Axis.
#9 	Multisurface 5-Axis

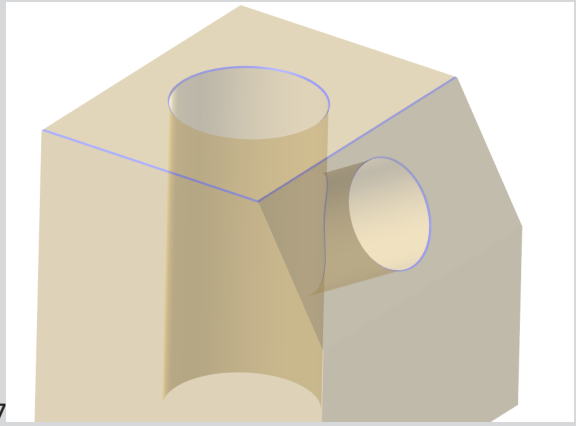
Tutorial	Toolpath Creation
 #7	Debur

Table Of Content

Multiaxis Essentials Projects **3**

Getting Started **15**

Objectives 16

Step 1: Starting Mastercam 16

Step 2: GUI - Graphical User Interface 17

Step 3: Navigate Through Mastercam 18

Step 4: Set The Attributes 21

Step 5: Manager Panels 23

Step 6: Setting Mastercam To Imperial 25

Step 7: Set The Grid 27

Tutorial 1 ..... **29**

Step 1: Create The Geometry Of The Cylindrical Helix 31

Step 2: Select The Machine And Set Up The Stock 39

Step 3: Machine The Cylindrical Helix - Axis Substitution About Y Axis 44

Step 4: Backplot The Toolpaths 54

Step 5: Verify The Toolpath 56

Step 6: Post The File To Generate The NC Code (Use A Different Machine) 58

Step 7: Save The Updated MCAM File 59

Step 8: Create The Geometry Of The Conical Helix 60

Step 9: Machine The Conical Helix - Axis Substitution About Y Axis 63

Step 10: Save The Updated MCAM File 65

Tutorial #1 Quiz 67

Tutorial 2 ..... **69**

Step 1: Select The Geometry File 71

Step 2: Select The Machine And Set Up The Stock 72

Step 3: Drilling Toolpath- Axis Substitution About Y Axis 76

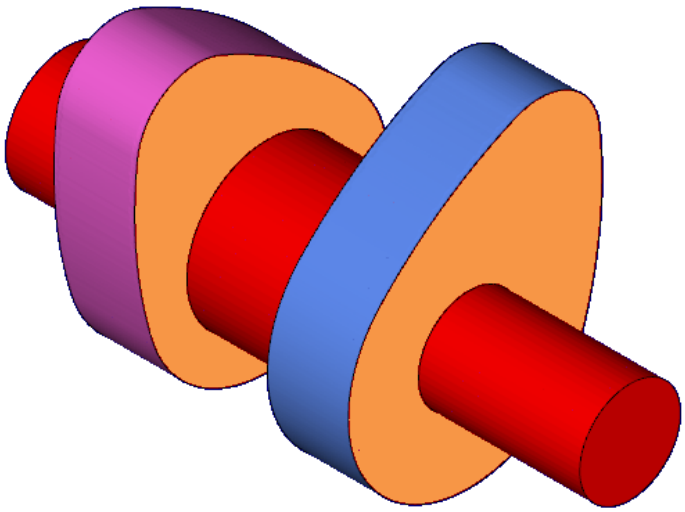
Step 4: Backplot The Toolpath 83
Step 5: Verify The Toolpath 85
Step 6: Drill The 8 Holes Using Axis Substitution Toolpath 86
Step 7: Axis Substitution About Y Axis - Contour Toolpath 93
Step 8: Machine The Top Slot - Contour Toolpath 103
Step 9: Transform-Rotate The Contour Toolpath About The Origin 111
Step 10: Post The File To Generate The NC Code 116
Tutorial #2 Quiz 120
Tutorial 3 ..... **121**
Step 1: Select The Geometry File 123
Step 2: Select The Machine And Set Up The Stock 124
Step 3: Axis Substitution About Y Axis - Contour Toolpath 129
Step 4: Backplot The Toolpath 138
Step 5: Verify The Toolpath 141
Step 6: Define A New Tool Plane To Drill The Hole 143
Step 7: Drill The Hole In The New Plane 148
Step 8: Transform - Rotate To Machine The Rest Of The Holes 153
Step 9: Machine The Top Slot In The Top Tool Plane 159
Step 10: Machine The Bigger Slot In A New Tool Plane 168
Step 11: Post The File To Generate The NC Code 178
Step 12: Save The Updated MCAM File 179
Tutorial #4 Quiz 181
Tutorial 4 ..... **183**
Step 1: Select The Geometry File 185
Step 2: Select The Machine And Set Up The Stock 186
Step 3: Rotary 4-Axis With Rotary Cut 191
Step 4: Rotary 4-Axis With Axial Cut 206
Step 5: Post The File To Generate The NC Code 219

Step 6: Save The Updated MCAM File 220
Tutorial #5 Quiz 222
Tutorial 5 ..... **223**
Step 1: Select The Geometry File 225
Step 2: Select The Machine And Set Up The Stock 226
Step 3: Curve With Tool Axis Control Oriented To A Point 232
Step 4: Curve With Tool Axis Control Oriented From A Point 245
Step 5: Curve With Tool Axis Control Oriented Normal To A Plane 257
Step 6: Curve With Tool Axis Control Oriented Towards A Chain 268
Step 7: Curve With Tool Axis Control Oriented To Lines 277
Step 8: Post The File To Generate The NC Code 286
Step 9: Save The Updated MCAM File 287
Tutorial #5 Quiz 289
Tutorial 6 ..... **291**
Step 1: Select The Geometry File 293
Step 2: Set Up The Stock 294
Step 3: Verify The Surface High Speed Toolpaths 299
Step 4: Create A Stock Model 300
Step 5: Swarf Toolpath With Walls Defined From The Chains 303
Step 6: Swarf Toolpath With Walls Defined From Surfaces 319
Step 7: Post The File To Generate The NC Code 327
Step 8: Save The Updated MCAM File 328
Tutorial #6 Quiz 330
Tutorial 7 ..... **331**
Step 1: Select The Geometry File 333
Step 2: Select The Machine And Set Up The Stock 334
Step 3: Create The Smaller Hole Axes 339

Step 4: Spot Drill Toolpath 342
Step 5: Drill The Three 1/2" Diameter Holes 351
Step 6: Circle Mill Toolpath With Tool Axis Control 356
Step 7: Post The File To Generate The NC Code 366
Step 8: Save The Updated MCAM File 367
Tutorial #7 Quiz 369
Tutorial 8 371
Step 1: Select The Geometry File 373
Step 2: Select The Machine 374
Step 3: Flow 5 Axis Toolpath With Tool Axis Control Parallel To Lines 376
Step 4: Machine Simulation To Check The Flow 5 Axis 392
Step 5: Flow 5 Axis Toolpath With Tool Axis Oriented To A Point 399
Step 6: Transform - Rotate The Toolpath To Finish All Blades 413
Step 7: Post The File To Generate The NC Code 418
Step 8: Save The Updated MCAM File 419
Tutorial #9 Quiz 421
Tutorial 9 423
Step 1: Select The Geometry File 425
Step 2: Select The Machine 426
Step 3: Multisurface 5 Axis Toolpath To Rough The Cavity 429
Step 4: Machine Simulation To Check The Multisurface 5 Axis 444
Step 5: Multisurface 5 Axis Toolpath To Finish The Part 449
Step 6: Post The File To Generate The NC Code 461
Step 7: Save The Updated MCAM File 462
Tutorial #9 Quiz 464
Tutorial 10 465
Step 1: Select The Geometry File 467
Step 2: Select The Machine And Set Up The Stock 468

Step 3: 3 Axis Deburr Toolpath 472
Step 4: 3 + 2 Deburr Toolpath 484
Step 5: 5-Axis Deburr Toolpath 492
Step 6: Post The File To Generate The NC Code 499
Step 7: Save The Updated MCAM File 501
Tutorial #10 Quiz 503
Quiz Answers **505**

Tutorial 4



OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

Open the CAD Model:

- The student should examine the part file to understand what part is being created in the tutorial.

Create the necessary Toolpaths to machine the part:

- The student will set up the stock size to be used and the appropriate tool settings.
- The left and right side cams will be roughed using a Rotary 4-Axis toolpath with the axial cut method in a one way direction.
- The left and right side cams will then be finished using a Rotary 4-Axis toolpath with the rotary cutting method in a zig zag direction.

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 one hour to complete.

This tutorial covers Mastercam's 4 and 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 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

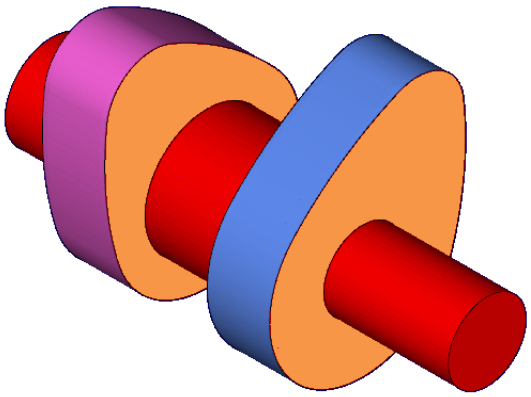
.

STEP 1: SELECT THE GEOMETRY FILE

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

File

- **Open**.
- Select "TUTORIAL 4.MCAM" from the directory you saved the file in.
- Press **Alt + F1** to fit the drawing to the screen.



STEP 2: SELECT THE MACHINE AND SET UP THE STOCK

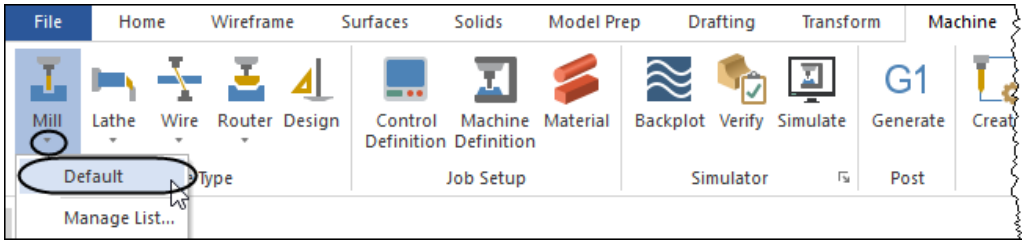
In Mastercam, you select a **Machine Definition** before creating any toolpaths. The **Machine Definition** is a model of your machine tool's capabilities and features and acts like a template for setting up machining jobs. The machine definition ties together three main components: the schematic model 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.

2.1 Open the Toolpaths Manager

*Note: If the **Toolpaths Manager** is not open, follow the instructions provided in **Tutorial 1** on **page 37**.*

2.2 Select the Machine Type

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

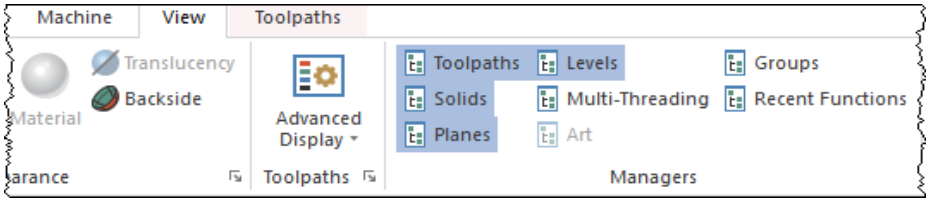


***Note:** Once you select the **Default**, the **Ribbon bar** changes to reflect the toolpaths that could be used with **Mill Default**.*

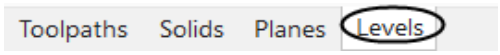
2.3 Make Level 10 Visible

View

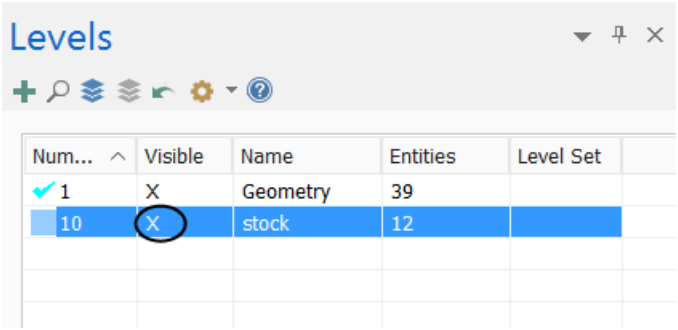
- In the **Managers** group, make sure that **Toolpaths**, **Solids**, **Planes**, and **Levels** are selected as shown.



- From the lower left corner below the **Toolpaths Manager** select the **Levels** tab as shown.

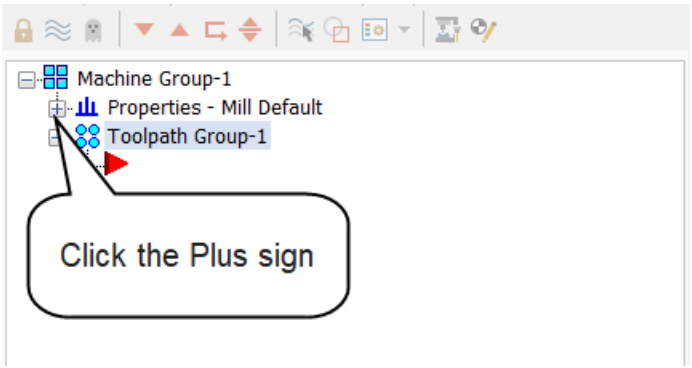


- Click in the **Visible** column next to **Number 10** as shown.

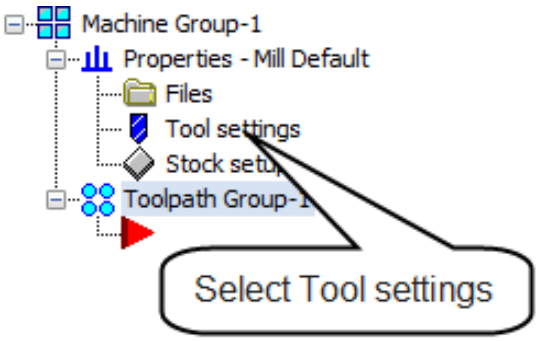


2.4 Set the Tool Parameters in the Tool Settings page

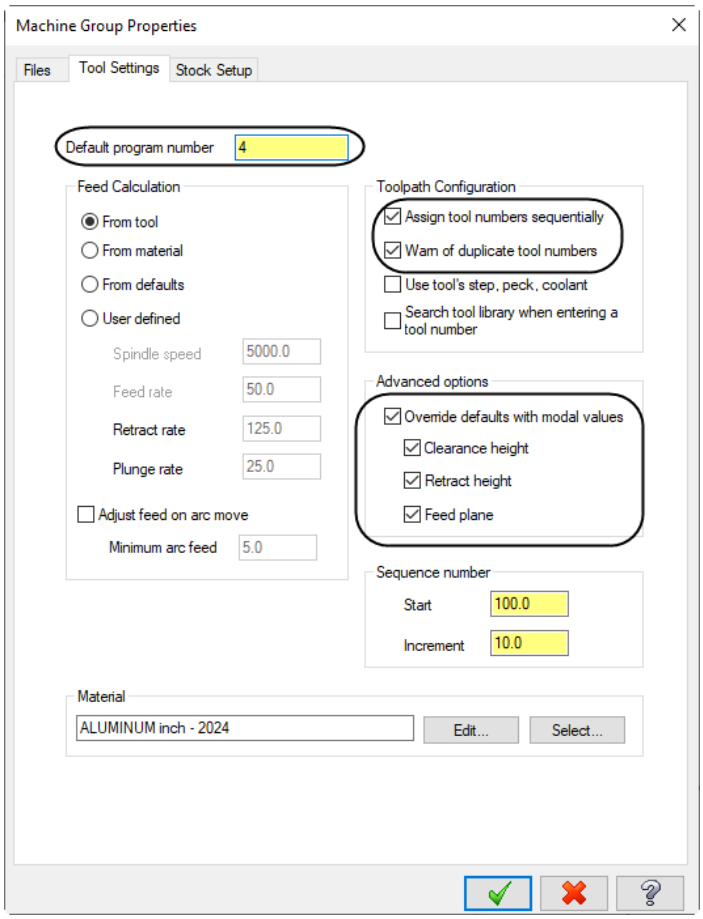
- Select the plus sign in front of **Properties** in the **Toolpaths Manager** to expand the **Toolpaths Group Properties** as shown.



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



♦ Change the parameters to match the image below.



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

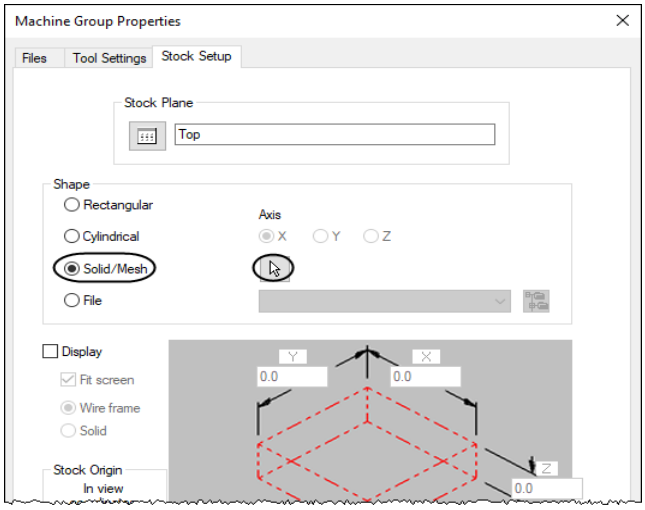
Assign tool numbers sequentially allows you to overwrite the tool number from the library with the next available tool number (First operation tool number 1; Second operation tool number 2, etc.).

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

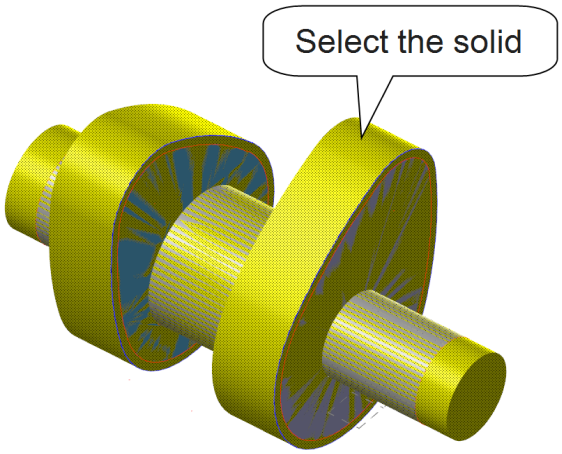
Override defaults with modal values enables the system to keep the values that you enter.

2.5 Set the Stock Shape and Size

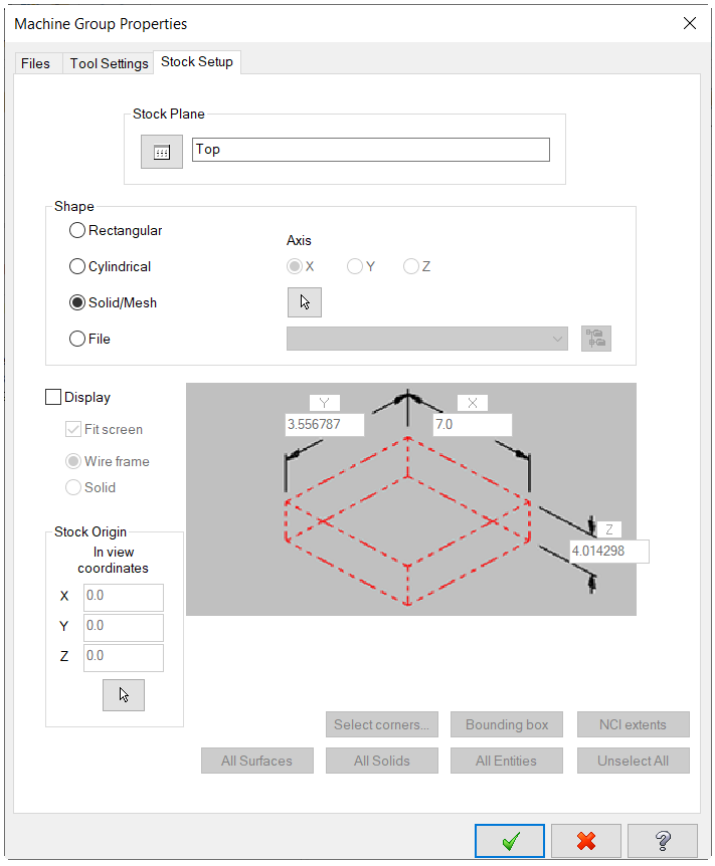
♦ Select the **Stock Setup** tab to define the stock.
♦ In the **Shape** area, enable **Solid/Mesh** and click on the **Select** button as shown**.**



♦ Select the solid from the graphics window as shown.



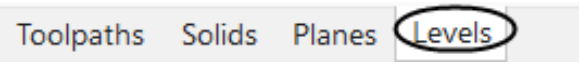
♦ The **Stock Setup** page should look as shown below.



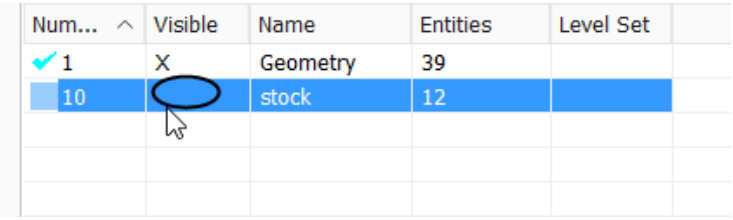
♦ Select the **OK** button to exit the **Machine Group Properties** dialog box.

2.6 Make Level 10 Invisible

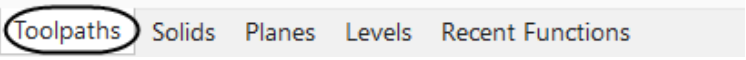
♦ Select the **Levels** tab.



♦ Click in the **Visible** column next to **Number 10** to remove the **X** as shown.



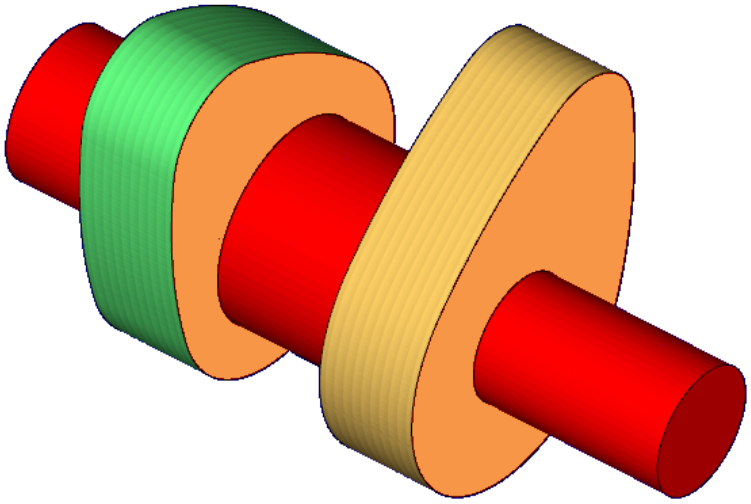
♦ Select the **Toolpaths** tab.



STEP 3: ROTARY 4-AXIS WITH ROTARY CUT

Rotary toolpaths are used on machines that allow simultaneous 4-Axis movements to create a 4-Axis surface machining toolpath which cuts around or along the axis of rotation of the machine (the **X axis** in our case). The **Rotary** toolpath is a finish toolpath. To demonstrate both rotary cut and axial cut, we are going to finish the part first using the rotary cutting method and then the axial cutting method.

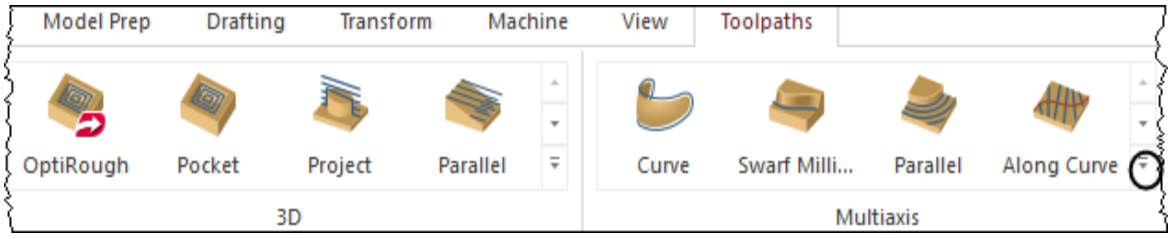
Toolpath Preview:



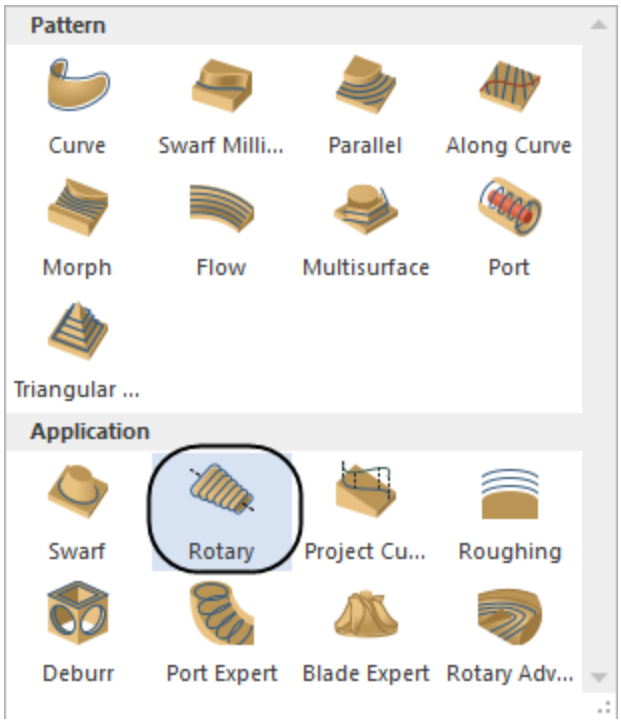
3.1 Machine the Right Side Cam using the Rotary Cut Method

Toolpaths

- From the **Multiaxis** group, select the **Expand Gallery** icon as shown.



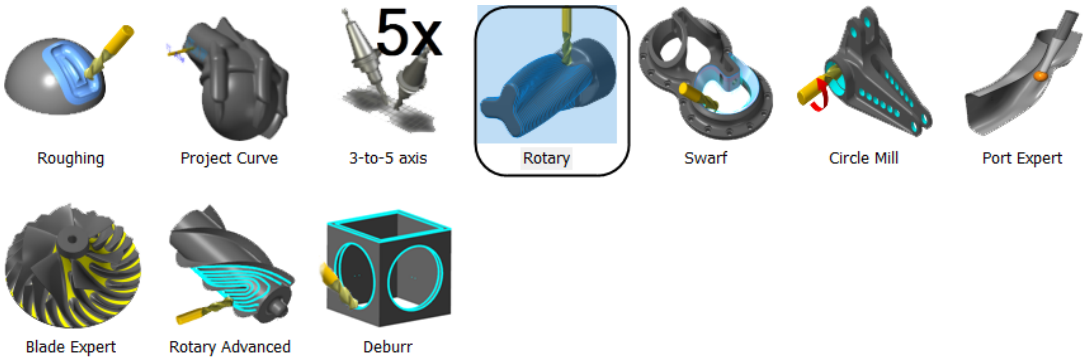
♦ From the **Application** group, select the **Rotary** icon.



3.2 Toolpath Type

The **Toolpath Type** page allows you to establish the type of Multiaxis toolpath to create. Toolpath type controls the options available as you proceed down the tree structure. The options are based on the toolpath family selected using the calculation based on buttons, as well as by selecting the picture of the toolpath you wish to create.

♦ In the **Toolpath Type** page, the **Rotary** icon should be already selected as shown.

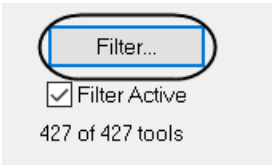


3.3 Set the Tool page parameters

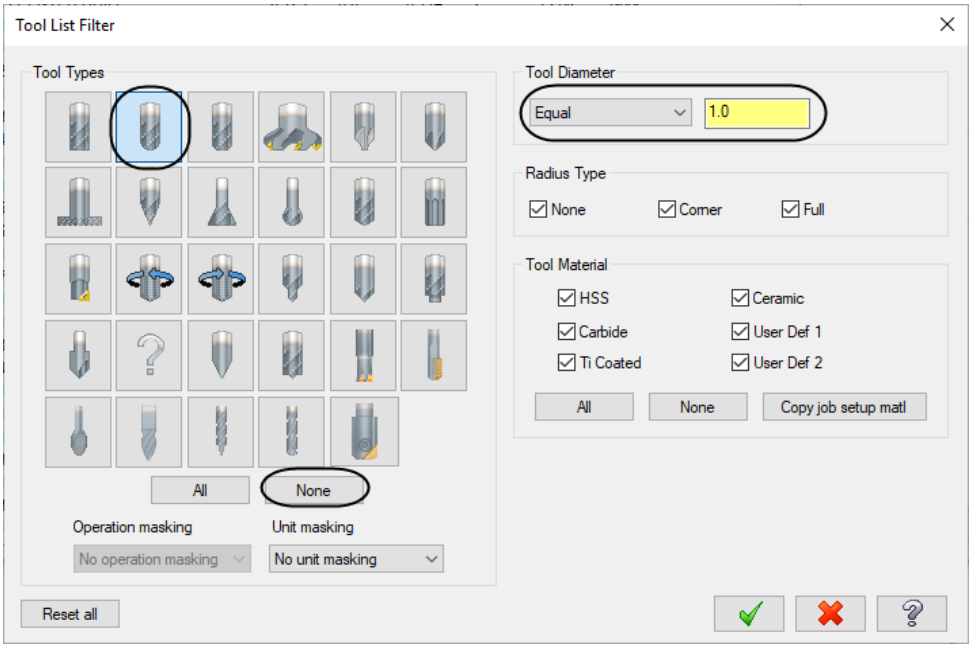
The **Tool** page allows you to select a tool, set the feeds and speeds, enter a comment about the operation, and set other general toolpath parameters.

- From the **Tree View list**, select the **Tool** page.
- To select the tool, click on the **Select library tool** button as shown.

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



- Select the **None** button to unselect any previous tool selection as shown.
- Select the **Ball 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.

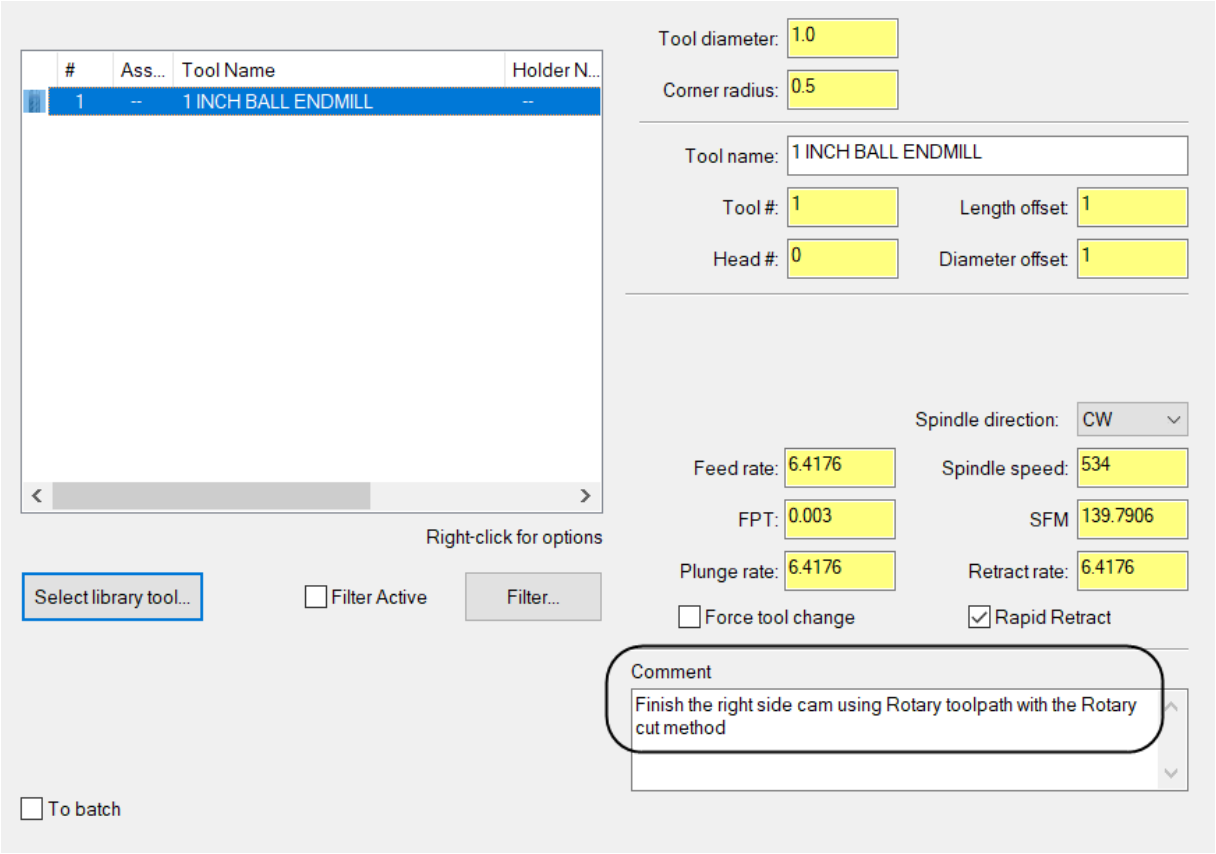


- Select the **OK** button to exit the **Tool List Filter** dialog box.

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

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.

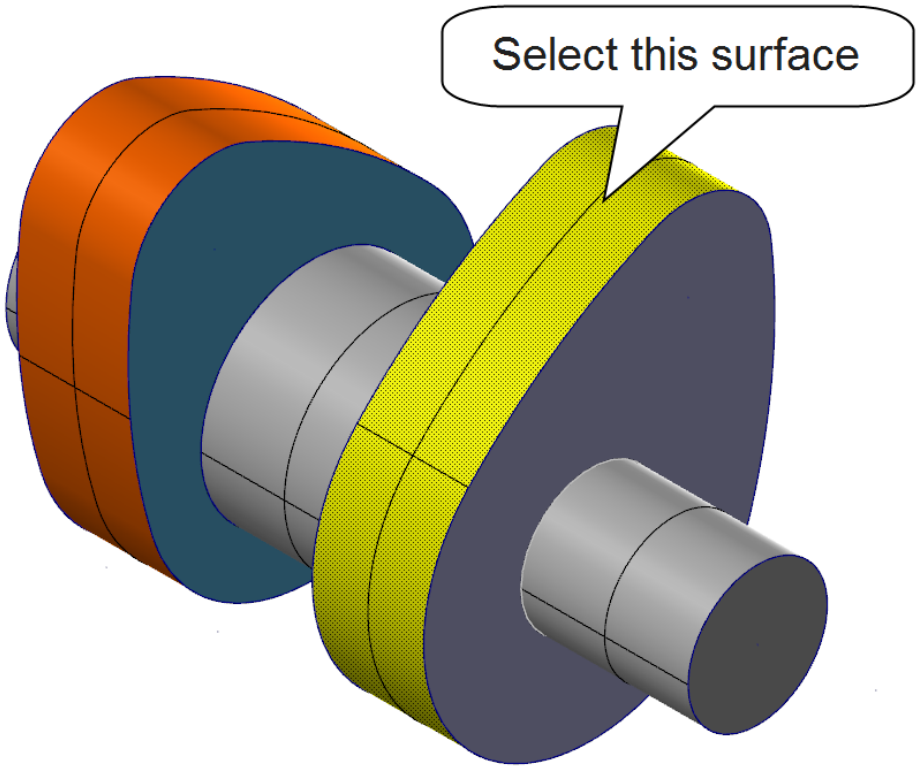
- Add a comment in the **Comment** area to identify the toolpath in the **Toolpaths Manager** and also in the **NC** file.
- The **Tool** page should look as shown.



3.4 Set the Cut Pattern Parameters

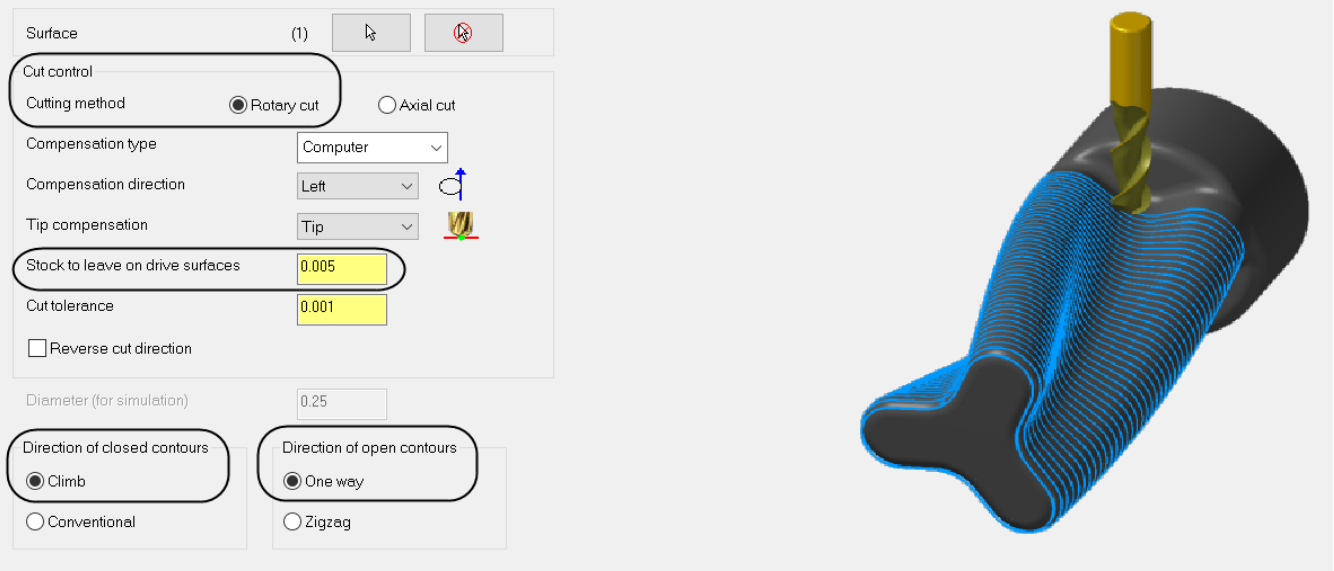
The **Cut Pattern** settings determine the geometry that the tool follows and how it travels along that geometry. The **Rotary cut** method is designed to cut around the diameter using 4-Axis output.

- From the **Cut Pattern** page, click on the **Select** button to return to the graphics view and select the surface.
- [Select Solid Face or Surface]: Select the right side cam surface as shown.



- Press **Enter** to finish the selection

- Enable **Rotary cut.**
- **Stock to leave on drive surfaces = 0.005.**
- **Cut tolerance = 0.001.**
- Enable **Climb** and **One way** as shown.



3.5 Set the Tool Axis Control parameters

Tool Axis Control settings determine the tool's orientation in relation to the geometry being cut.

Use center point allows you to generate tool vectors that terminate at the selected point. Deselect it to generate tool vectors that are normal to the surface.

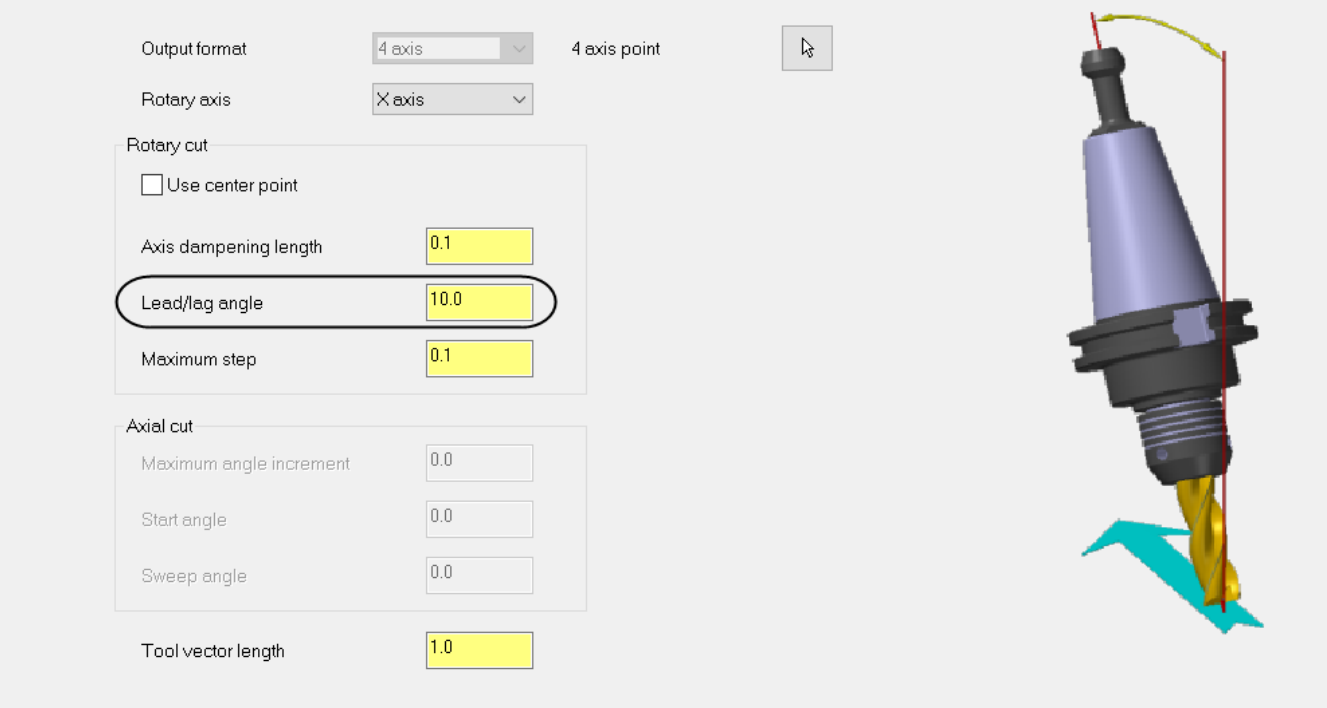
Axis dampening determines the angular change between the vectors. A longer axis dampening length creates less angular change between the vectors. A shorter length provides more tool positions and a toolpath that adheres closely to the surface.

Lead/lag angle (positive/negative value) tilts the tool forward/backward in the direction of the toolpath.

Maximum step controls the distance between passes. The distance is measured parallel to the axis of rotation.

Tool vector length controls the toolpath display by determining the length of the tool axis at each tool position. The recommended value is 1.0".

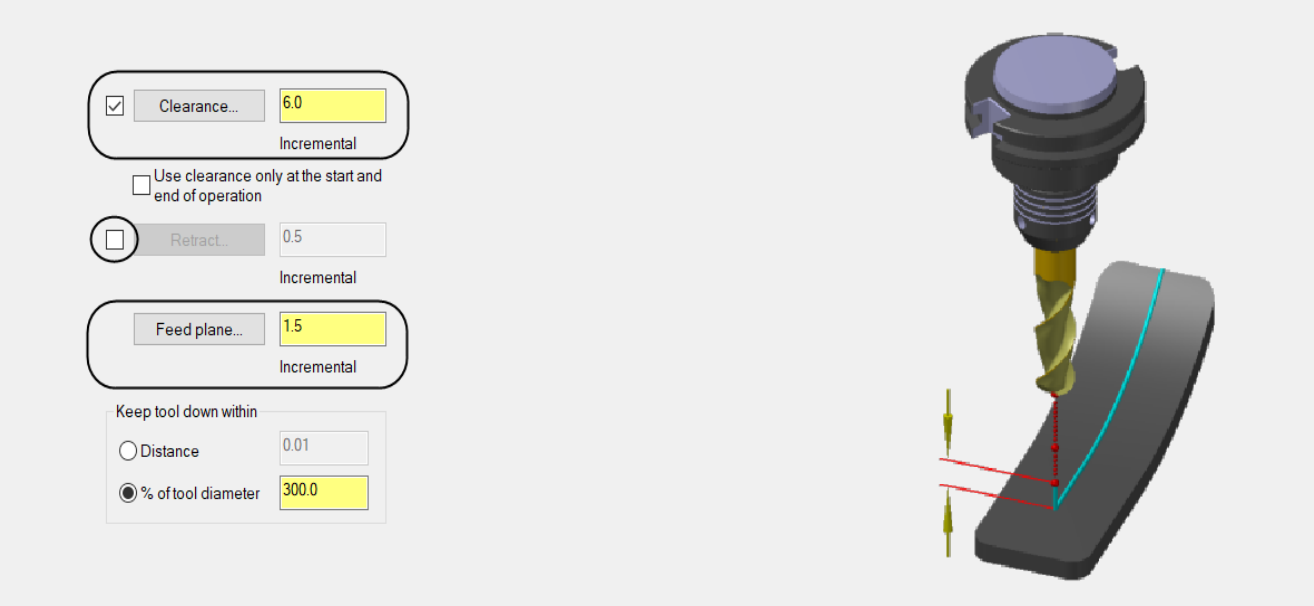
♦ Select the **Tool Axis Control** page and make the changes as shown.



3.6 Set the Linking Parameters

Linking parameters determine how the tool moves when it is not cutting material. In this page you can set the **Clearance**, **Retract**, and **Feed plane** heights and force the tool to move directly between the passes at the feed rate.

♦ Select the **Linking Parameters** page and make the following changes as shown.

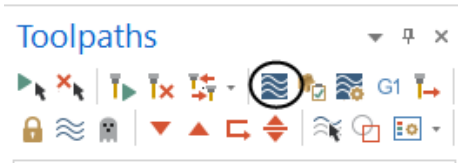


♦ Select the **OK** button to exit the **Multiaxis Toolpath - Rotary** dialog box.

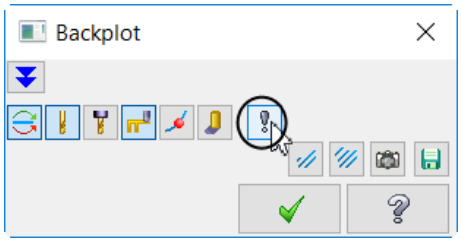
3.7 Backplot the Toolpath

Backplotting shows the path the tools take to cut the part. This display lets you spot errors in the program before you machine the part. As you backplot toolpaths, Mastercam displays additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates, and the cycle time. It also shows any collisions between the workpiece and the tool.

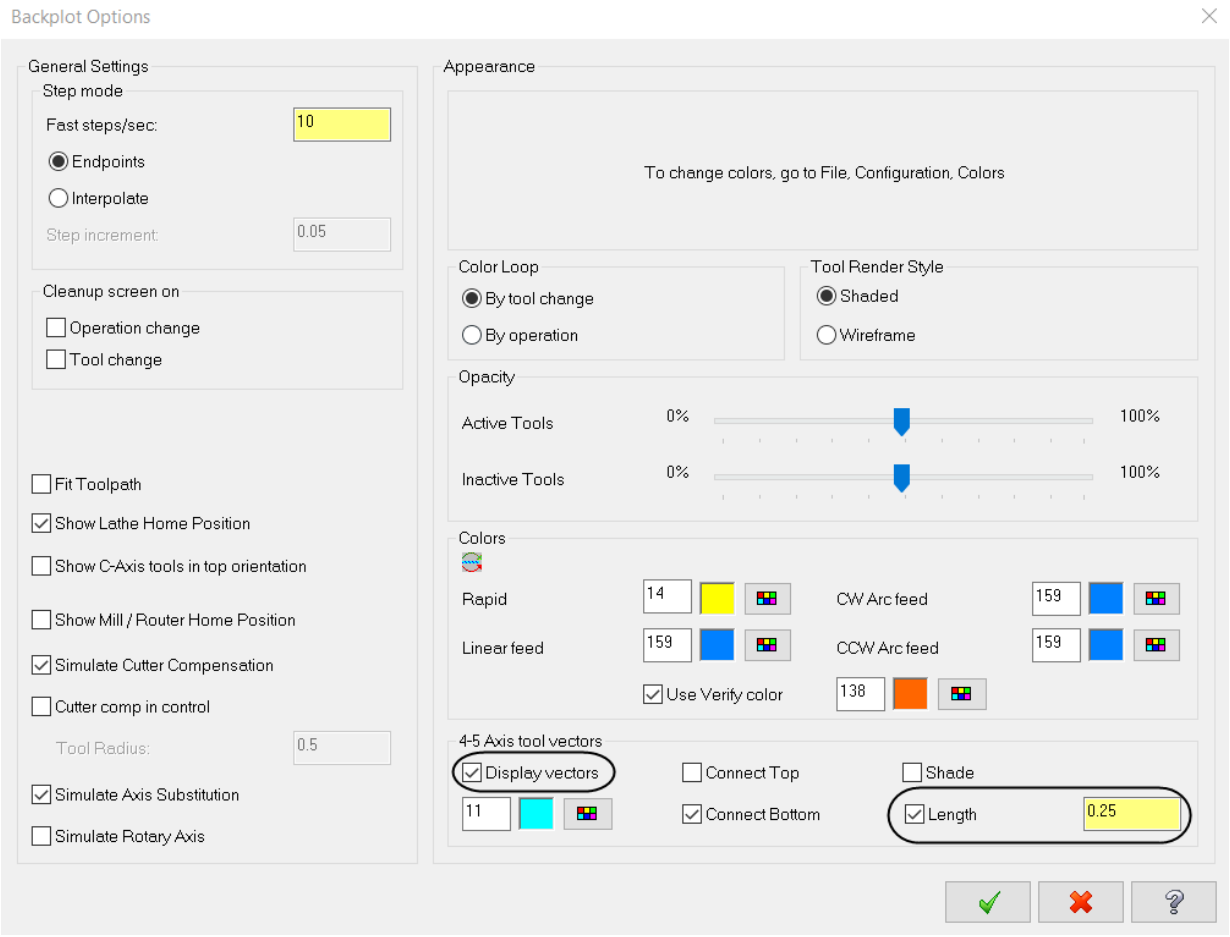
- In the **Toolpaths Manager**, make sure that the toolpath is selected (signified by the green check mark on the folder icon).
- Select the **Backplot selected operations** icon.



- Select the **Play** button to run **Backplot**.
- Click on the **Options** button.

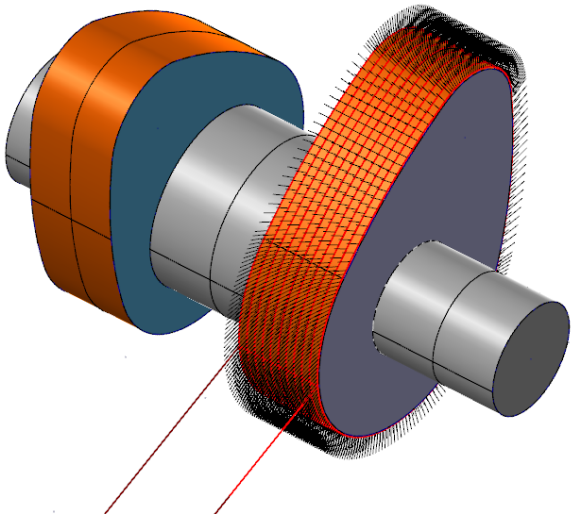


♦ In the **Backplot Options** dialog box, enable **Display vectors** and set the **Length** to **0.25**.

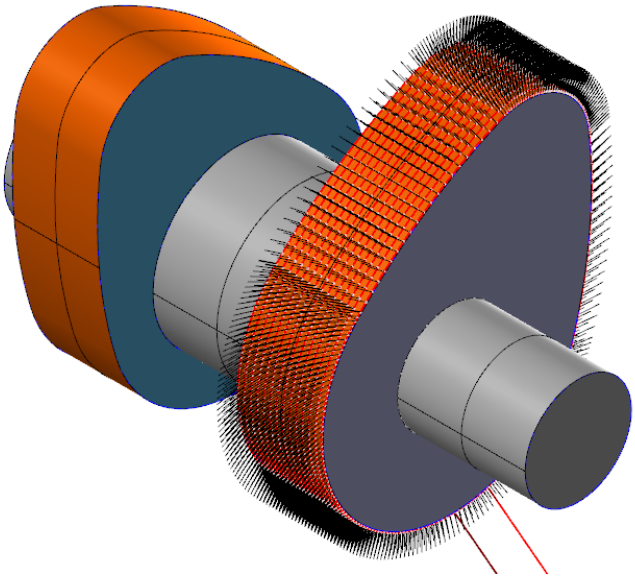


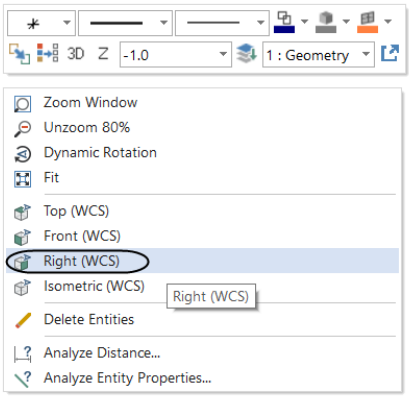
♦ Select the **OK** button to exit the **Backplot Options** dialog box.

- Select the **Play** button to run **Backplot**.
- The toolpath should look as shown.

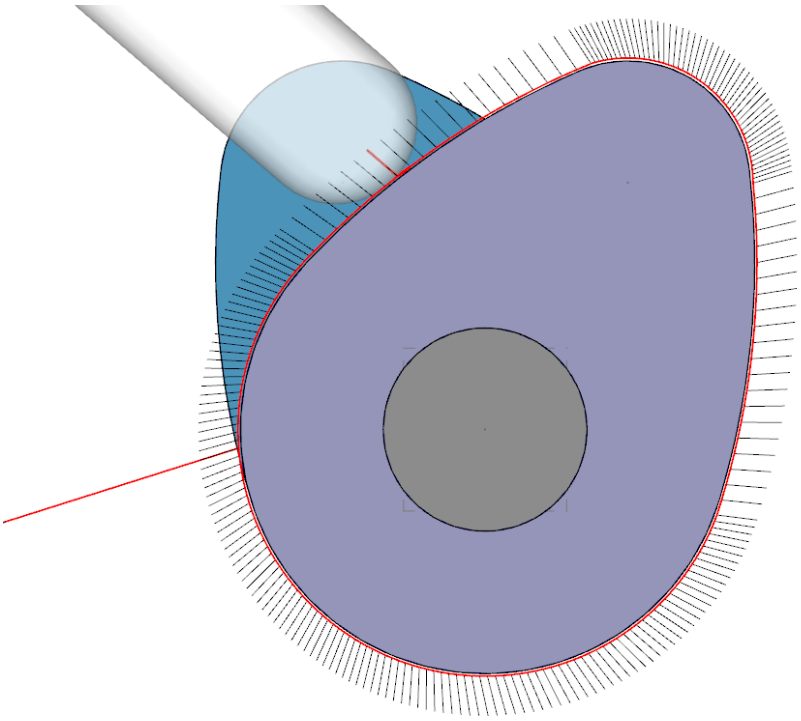


- Right mouse click and change the **Graphics view** to **Right** as shown.



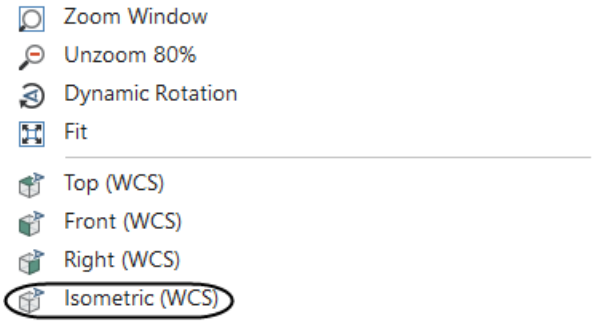


- Backplot the toolpath agan as shown.



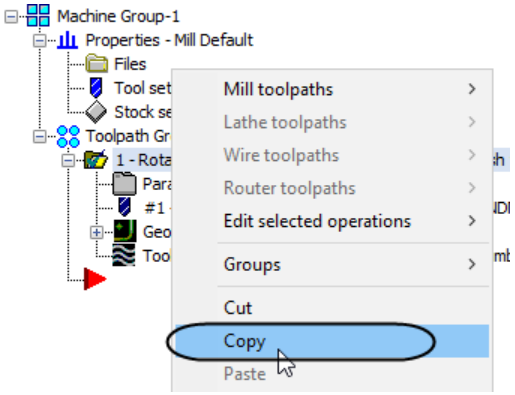
Note: The tool axis orientation due to the 10 degrees lead angle.

- Select the **OK** button to exit the **Backplot**.
- Press **Alt + T** to remove the toolpath display from the graphics window.
- Right mouse click in the graphics window and select the **Isometric** view as shown.

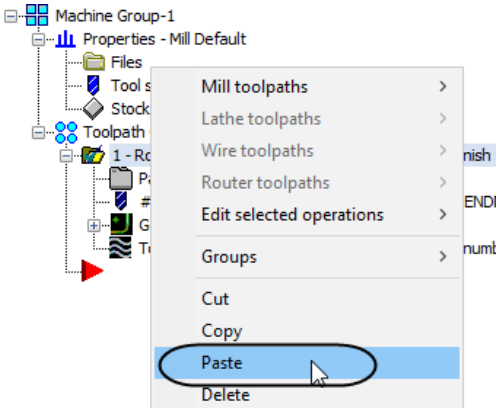


3.8 Machine the Left Side Cam using the Rotary Cut Method

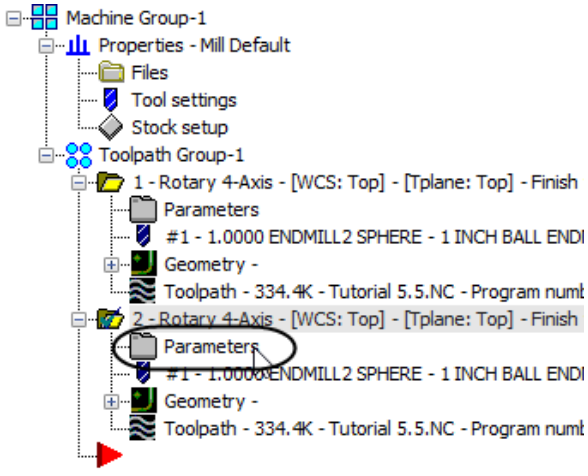
- Right mouse click in the **Toolpaths Manager** and select **Copy** as shown.



- Right mouse click again in the **Toolpaths Manager** and select **Paste** to duplicate the existing toolpath as shown.



- From the **Toolpaths Manager**, select the **Parameters** in the second toolpath as shown.



www.**eMastercam**.com

Sitewide Subscription

An eMastercam sitewide subscription will grant you instant access to all of our online resources, including our full library of eBooks and eCourses. Once subscribed, you will be able to watch all of our streaming eCourses in your browser or read any of our Mastercam eBooks using our Webviewer.

www.**eMastercam**.com/
sitewide-subscription

Decades of experience. Thousands of post processors.

Choose In-House Solutions for all your post needs.

Implementing successful post processors for decades, the post department at In-House Solutions has earned a reputation for quality, resulting in one of the largest post departments in the world. We have an extensive library built using our new post engine, "IKE". We are continually developing and expanding our product to include new machines and controllers everyday, and if a post is not already available, we will develop a custom one for nearly any machine. We have great relationships with OEMs who provide technical information for both machines and controls, which in turn allows us to produce post processors that will generate edit-free code and run your machine efficiently.

Be sure to ask your Mastercam reseller for an In-House Solutions IKE post!

www.**inhousesolutions**.com/**posts**

In-House Solutions

Mastercam @Courses

Included with each eCourse: HLE demo software, corresponding PDF eBook, and personalized final certificate.

LOGIN & LEARN!

Mastercam CAD/CAM Systems 2020 Home Learning Edition Demo Software DOWNLOAD

Expand your Mastercam knowledge, anytime and at your own pace, with our selection of step-by-step online video eCourses.

Mastercam eCourses provide both novice and seasoned Mastercam users with the tools and information they need to excel.

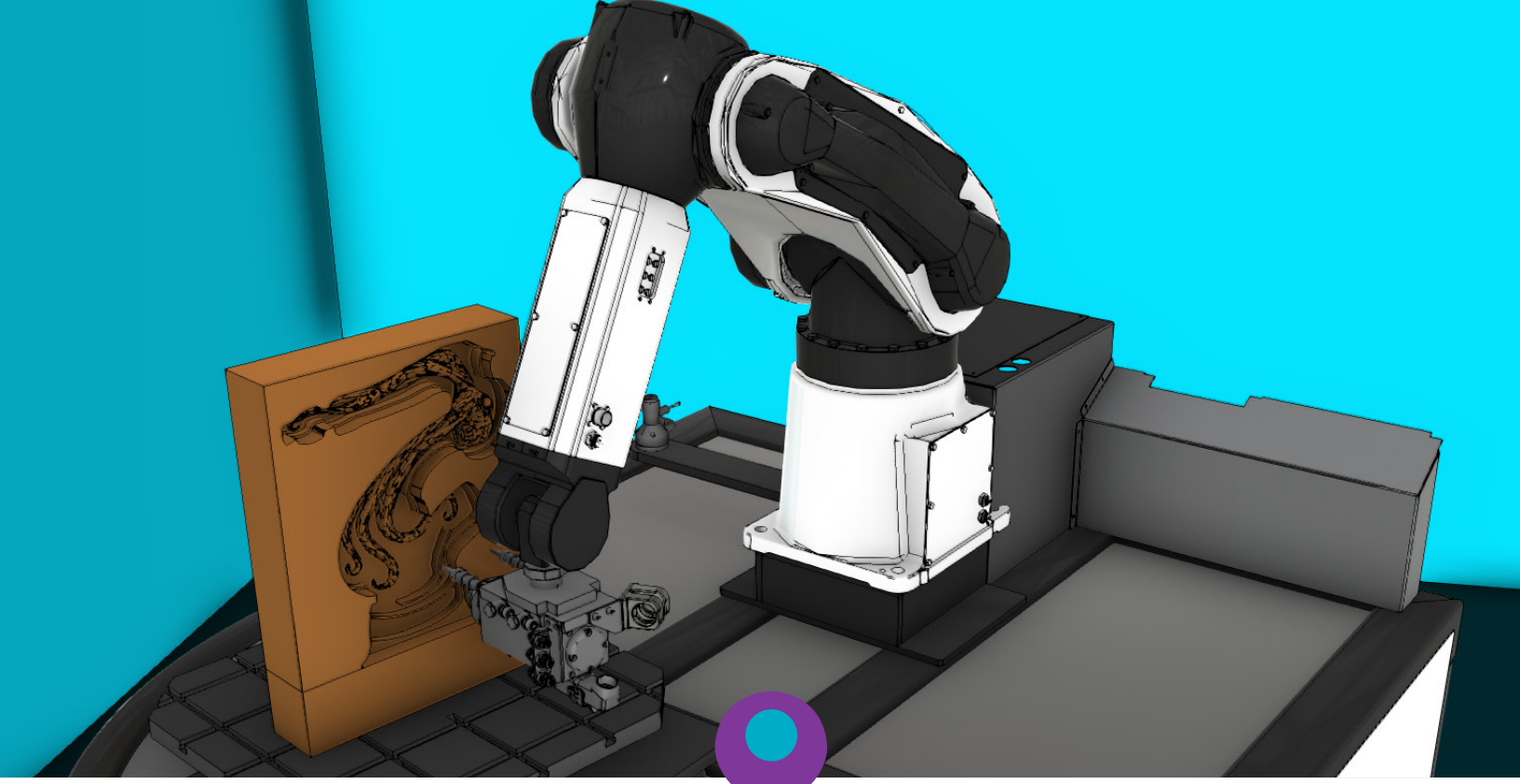
The *"log in & learn"* format of each eCourse allows users to set up their own online classroom, where each user's experience is customized and tracked. Because the eCourses are not subscription-based and have no expiration date, users can log on and off at any time, and finish the program at their convenience without additional expense.

Approximate completion time ranges from 7.5 to 15 hours depending on the eCourse, making it possible for employees or students to gain new skills outside of work or studies.

Highlights:

- Online previews with table of contents including the time it takes to complete each session.
- Mastercam 2020 Home Learning Edition Demo Software download is included so you can follow along with our instructors.
- Corresponding eBook is provided.
- Quizzes follow each tutorial.
- Personalized certificate of completion for each successfully completed eCourse.
- **Instant, lifetime access**

www.**eMastercam**.com/**eCourses**



OCTOPUZ®

Robot Programming & Simulation Software

Program and generate robot code in a simple and easy way, regardless of application and brand. Spend less time programming and more time manufacturing!

Why choose OCTOPUZ?

- Multiple robots? No problem!
- Most robot brands supported
- Simple simulation building
- Extensive component library
- Automatic toolpath solving
- Application versatility
- CAM interface
- Complex kinematic systems

www.**OCTOPUZ**.com

eMastercam

TELL US WHAT YOU THINK

Our goal is to provide you the best Mastercam Training Solutions,
and we do that with your help.

Tell us what you like about our training solutions.
Let us know what you think can be improved.
Give us suggestions for future products.

www.**eMastercam**.com/**feedback**

We appreciate your feedback!

eMastercam.com

YOUR ONLINE MASTERCAM SOURCE

DID YOU LIKE THIS BOOK?

- Training Tutorials
- Instructor Materials
- Handbooks
- Professional Courseware
- Pro. Certification Curriculum
- Online Video eCourses
- Site Licenses
- Sitewide Subscription
- Online Community

Discover other Mastercam Training Solutions at
www.**eMastercam**.com