

# **Mastercam** 2019

METRIC – TRAINING TUTORIAL SERIES



Demo Software Download Instructions Included

# MULTIAXIS ESSENTIALS

# ***Mastercam***® 2019

---

## MULTIAXIS ESSENTIALS TRAINING TUTORIAL

---

## **Mastercam 2019 Multiaxis Essentials Metric Training Tutorial**

Copyright: 1998 - 2019 In-House Solutions Inc. All rights reserved

Software: Mastercam 2019

Authors: Mariana Lendel

ISBN: 978-1-77146-779-7

Date: August 7, 2018

### **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**

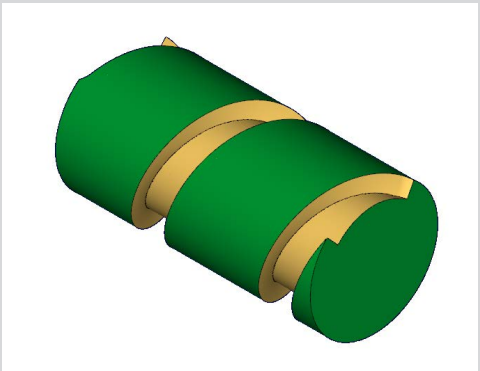
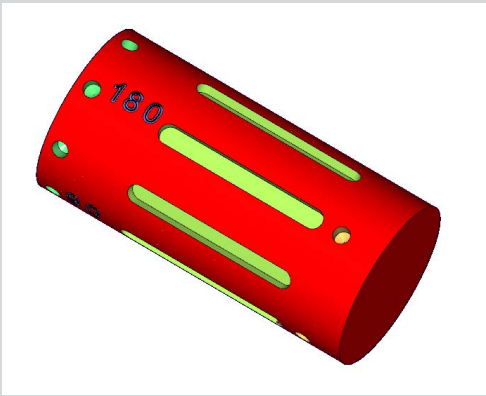
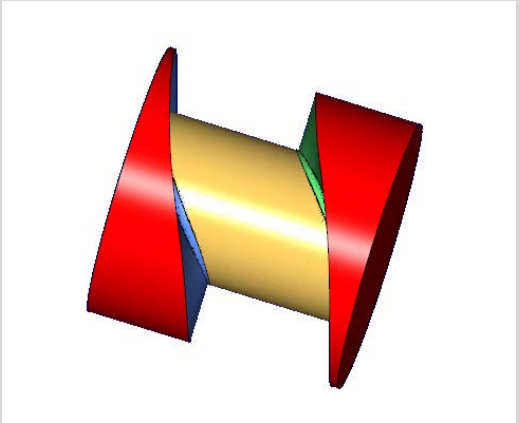
Mastercam is a registered trademark of CNC Software, Inc.

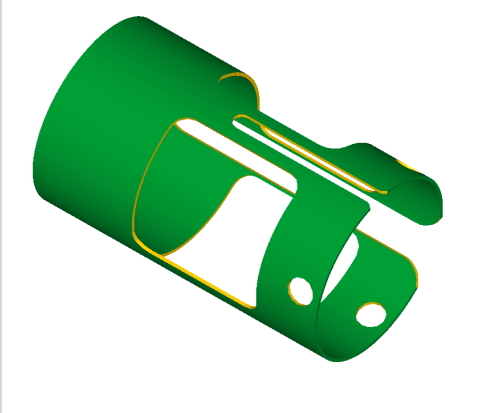
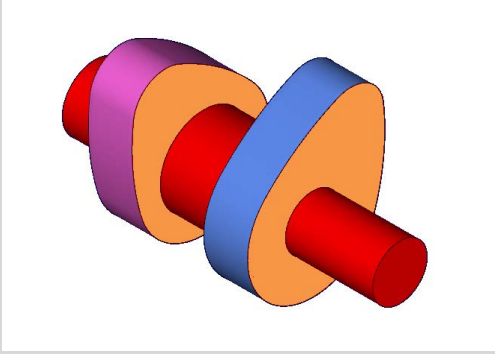

Microsoft, the Microsoft logo, are registered trademarks of Microsoft Corporation;

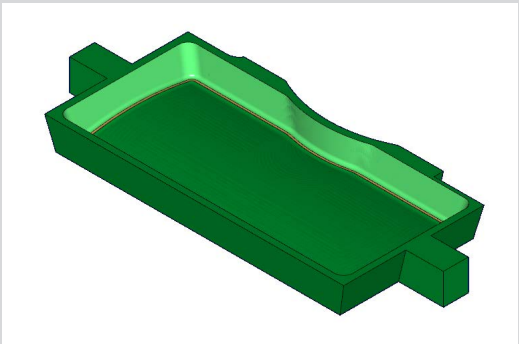
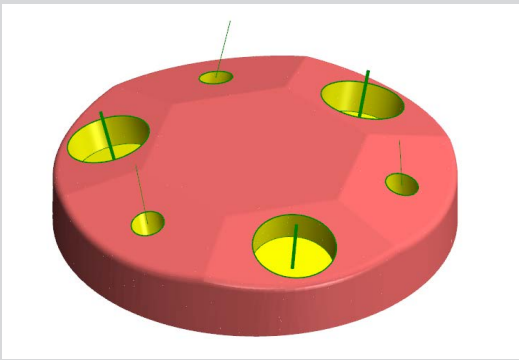
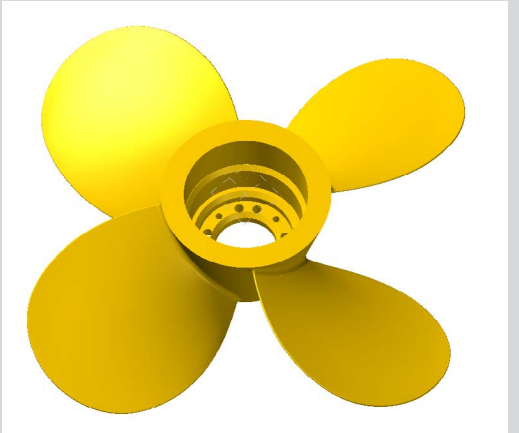
Windows 10 is a registered trademarks of Microsoft Corporation.

## 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 Rotatry Axis Positioning. Contour With Axis Substitution. Transform Rotate.</p>
<p>#3</p> 	<p>Contour with Axis Substitution. Roldie C-Hook.</p>

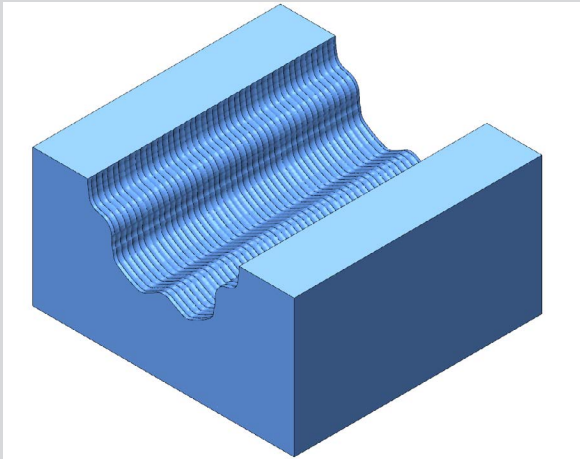
Tutorial	Toolpath Creation
<p>#4</p> 	<p>Contour With Axis Substitution. Contour Indexing Using Planes.</p>
<p>#5</p> 	<p>Rotary 4-Axis.</p>
<p>#6</p> 	<p>Curve 5 -Axis.</p>

Tutorial	Toolpath Creation
<p>#7</p> 	<p>Swarf 5-Axis.</p>
<p>#8</p> 	<p>Drill 5-Axis. Circle Mill 5-Axis.</p>
<p>#9</p> 	<p>Flow 5-Axis.</p>

**Tutorial**

**Toolpath Creation**

#10



Multisurface 5-Axis

## Table Of Content

<b>Multiaxis Essentials Projects</b>	<b>3</b>
<b>Getting Started</b>	<b>13</b>
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	19
Step 5: Manager Panels	21
Step 6: Setting Mastercam To Metric	23
Step 7: Set The Grid	25
<b>Tutorial 1</b>	<b>27</b>
Step 1: Create The Geometry Of The Cylindrical Helix	29
Step 2: Select The Machine And Set Up The Stock	37
Step 3: Machine The Cylindrical Helix - Axis Substitution About Y Axis	43
Step 4: Backplot The Toolpaths	53
Step 5: Verify The Toolpath	55
Step 6: Post The File To Generate The NC Code (Use A Different Machine)	57
Step 7: Save The Updated MCAM File	58
Step 8: Create The Geometry Of The Conical Helix	59
Step 9: Machine The Conical Helix - Axis Substitution About Y Axis	62
Step 10: Save The Updated MCAM File	64
Tutorial #1 Quiz	66
<b>Tutorial 2</b>	<b>67</b>
Step 1: Select The Geometry File	69
Step 2: Select The Machine And Set Up The Stock	70
Step 3: Drilling Toolpath- Axis Substitution About Y Axis	74
Step 4: Backplot The Toolpath	82

---

Step 5: Verify The Toolpath .....	84
Step 6: Drill The 8 Holes Using Axis Substitution Toolpath .....	85
Step 7: Axis Substitution About Y Axis - Contour Toolpath .....	91
Step 8: Machine The Top Slot - Contour Toolpath .....	102
Step 9: Transform-Rotate The Contour Toolpath About The Origin .....	110
Step 10: Post The File To Generate The NC Code .....	115
Tutorial #2 Quiz .....	119
<b>Tutorial 3 .....</b>	<b>121</b>
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 .....	142
Step 5: Rolldie C-Hook Toolpath .....	144
Step 6: Post The File To Generate The NC Code .....	164
Step 7: Save The Updated MCAM File .....	165
Tutorial #3 Quiz .....	171
<b>Tutorial 4 .....</b>	<b>173</b>
Step 1: Select The Geometry File .....	175
Step 2: Select The Machine And Set Up The Stock .....	176
Step 3: Axis Substitution About Y Axis - Contour Toolpath .....	181
Step 4: Backplot The Toolpath .....	190
Step 5: Verify The Toolpath .....	193
Step 6: Define A New Tool Plane To Drill The Hole .....	195
Step 7: Drill The Hole In The New Plane .....	200
Step 8: Transform - Rotateto Machine The Rest Of The Holes .....	205
Step 9: Machine The Top Slot In The Top Tool Plane .....	211
Step 10: Machine The Bigger Slot In A New Tool Plane .....	220

---

Step 11: Post The File To Generate The NC Code .....	230
Step 12: Save The Updated MCAM File .....	231
Tutorial #4 Quiz .....	233
<b>Tutorial 5 .....</b>	<b>235</b>
Step 1: Select The Geometry File .....	237
Step 2: Select The Machine And Set Up The Stock .....	238
Step 3: Rotary 4-Axis With Rotary Cut .....	243
Step 4: Rotary 4-Axis With Axial Cut .....	258
Step 5: Post The File To Generate The NC Code .....	272
Step 6: Save The Updated MCAM File .....	273
Tutorial #5 Quiz .....	275
<b>Tutorial 6 .....</b>	<b>277</b>
Step 1: Select The Geometry File .....	279
Step 2: Select The Machine And Set Up The Stock .....	280
Step 3: Curve With Tool Axis Control Oriented To A Point .....	286
Step 4: Curve With Tool Axis Control Oriented From A Point .....	301
Step 5: Curve With Tool Axis Control Oriented Normal To A Plane .....	313
Step 6: Curve With Tool Axis Control Oriented Towards A Chain .....	324
Step 7: Curve With Tool Axis Control Oriented To Lines .....	333
Step 8: Post The File To Generate The NC Code .....	342
Step 9: Save The Updated MCAM File .....	343
Tutorial #6 Quiz .....	345
<b>Tutorial 7 .....</b>	<b>347</b>
Step 1: Select The Geometry File .....	349
Step 2: Select The Machine And Set Up The Stock .....	350
Step 3: Verify The Surface High Speed Toolpaths .....	355
Step 4: Create A Stock Model .....	356
Step 5: Swarf Toolpath With Walls Defined From The Chains .....	358



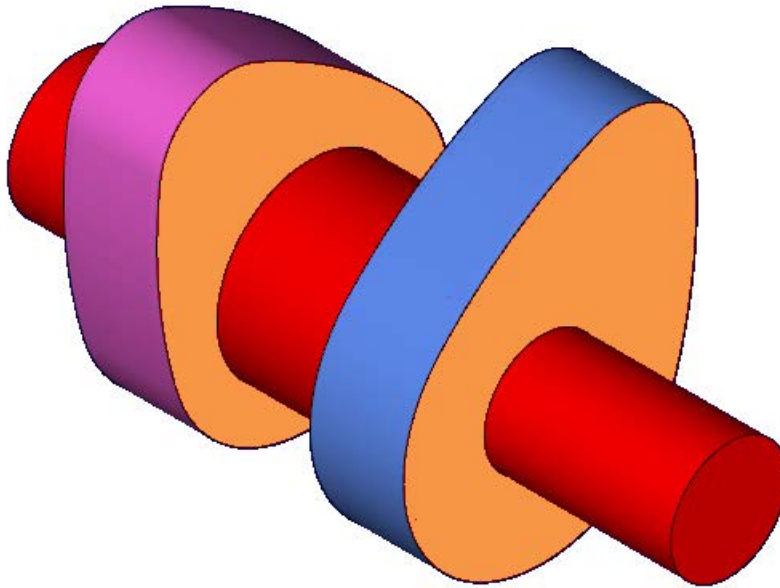
---

Step 6: Swarf Toolpath With Walls Defined From Surfaces .....	374
Step 7: Post The File To Generate The NC Code .....	382
Step 8: Save The Updated MCAM File .....	383
Tutorial #7 Quiz .....	385
<b>Tutorial 8 .....</b>	<b>387</b>
Step 1: Select The Geometry File .....	389
Step 2: Select The Machine And Set Up The Stock .....	390
Step 3: Multiaxis - Drill Toolpath With Entity Type Set To Points/lines .....	395
Step 4: Create The Hole Axis .....	409
Step 5: Swarf Toolpath With Walls Defined From The Chains .....	412
Step 6: Post The File To Generate The NC Code .....	422
Step 7: Save The Updated MCAM File .....	423
Tutorial #8 Quiz .....	425
<b>Tutorial 9 .....</b>	<b>427</b>
Step 1: Select The Geometry File .....	429
Step 2: Select The Machine .....	430
Step 3: Flow 5 Axis Toolpath With Tool Axis Control Parallel To Lines .....	432
Step 4: Machine Simulation To Check The Flow 5 Axis .....	448
Step 5: Flow 5 Axis Toolpath With Tool Axis Oriented To A Point .....	455
Step 6: Transform - Rotate The Toolpath To Finish All Blades .....	469
Step 7: Post The File To Generate The NC Code .....	474
Step 8: Save The Updated MCAM File .....	475
Tutorial #9 Quiz .....	477
<b>Tutorial 10 .....</b>	<b>479</b>
Step 1: Select The Geometry File .....	481
Step 2: Select The Machine .....	482
Step 3: Multisurface 5 Axis Toolpath To Rough The Cavity .....	485

---

Step 4: Machine Simulation To Check The Multisurface 5 Axis .....	500
Step 5: Multisurface 5 Axis Toolpath To Finish The Part .....	505
Step 6: Post The File To Generate The NC Code .....	517
Step 7: Save The Updated MCAM File .....	518
Tutorial #10 Quiz .....	520
<b>Quiz Answers .....</b>	<b>521</b>

## Tutorial 5



## 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

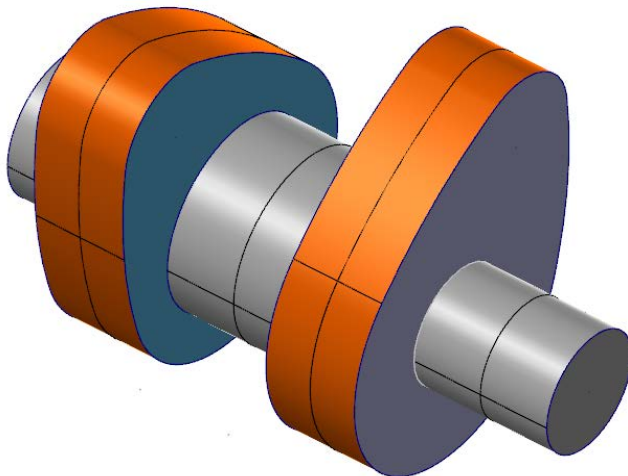
---

## STEP 1: SELECT THE GEOMETRY FILE

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

### FILE

- ◆ **Open.**
- ◆ Select "TUTORIAL 5.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

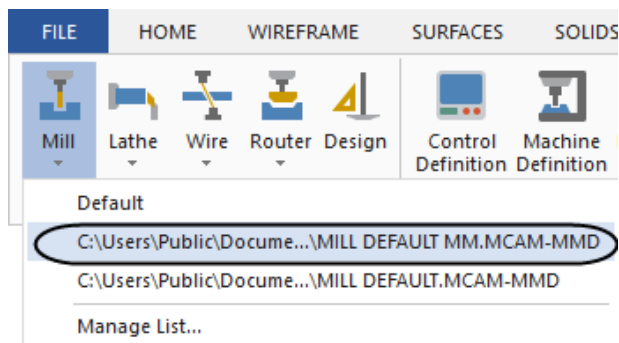
In Mastercam, you select a **Machine Definition** before creating any toolpaths. The **Machine Definition** is a model of your machine tool's capabilities and features and acts like a template for setting up machining jobs. The machine definition ties together three main components: the schematic model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

### 2.1 Open the Toolpaths Manager

*Note: If the **Toolpaths Manager** is not open, follow the instructions provided in **Tutorial 1** on **page 25**.*

### 2.2 Select the Machine Type

- ◆ From the **Machine Type Group**, select the drop down arrow below **Mill**.
- ◆ Select **MILL DEFAULT MM.MCAM-MMD**.



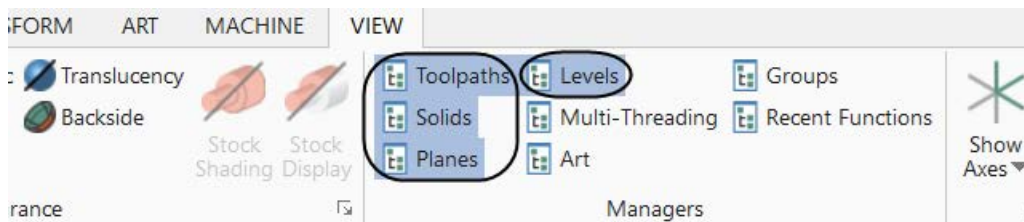
*If MILL DEFAULT MM.MCAM-MMD does not appear, follow the instructions on **Page 38**.*

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

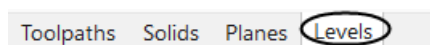
### 2.3 Make Level 10 Visible

#### VIEW

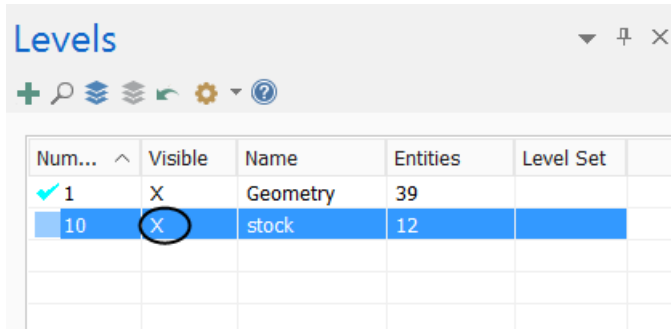
- ◆ In the **Managers** group, make sure that **Toolpaths**, **Solids**, **Planes**, and **Levels** are selected as shown.



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



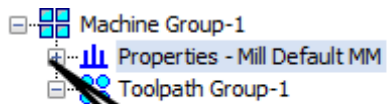
- ◆ Click in the **Visible** column next to **Number 10** as shown..



Num...	Visible	Name	Entities	Level Set
1	X	Geometry	39	
10	X	stock	12	

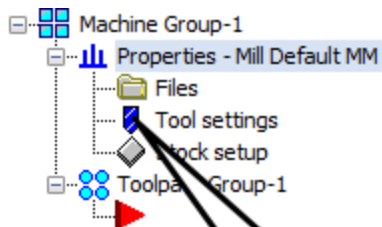
#### 2.4 Set the Tool Parameters in the Tool Settings page

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



Select the plus sign

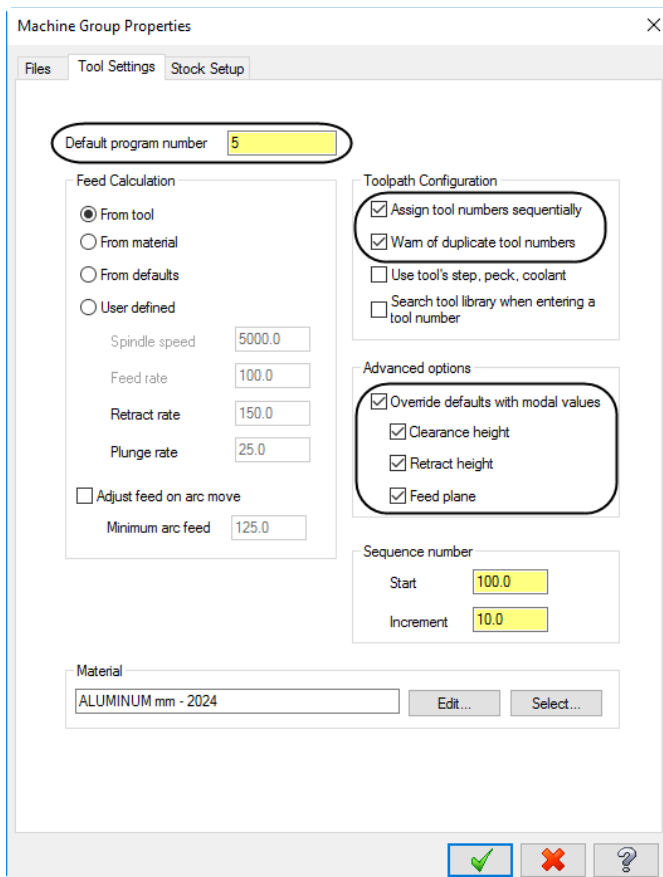
- ◆ Select **Tool settings** to set the tool parameters as shown.



Select Tool settings



- ◆ 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.

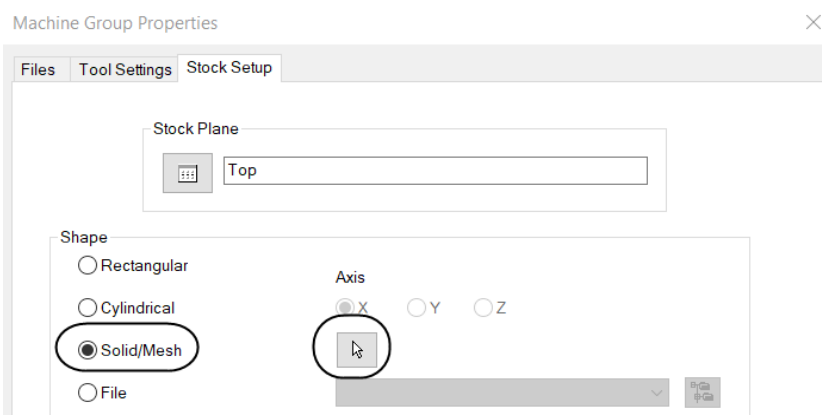
**Assign tool numbers sequentially** allows you to overwrite the tool number from the library with the next available tool number (First operation tool number 1; Second operation tool number 2, etc.).

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

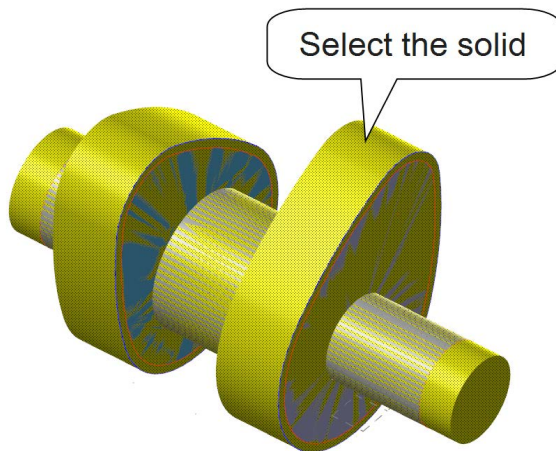
**Override defaults with modal values** enables the system to keep the values that you enter.

## 2.5 Set the Stock Shape and Size

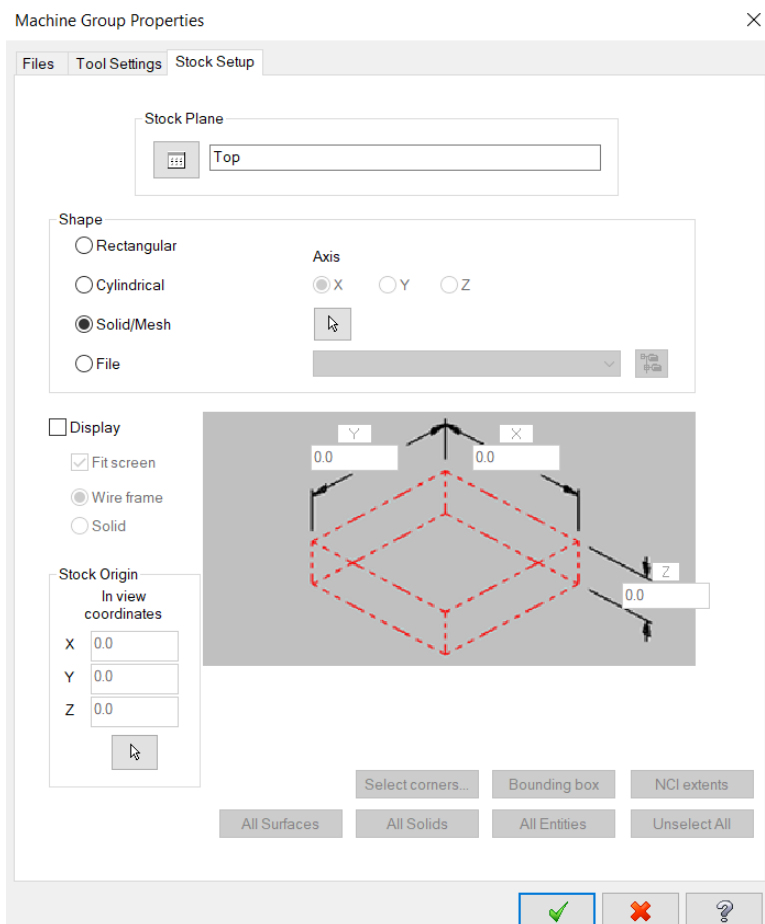
- ◆ Select the **Stock Setup** tab to define the stock.
- ◆ In the **Shape** area, enable **Solid/Mesh** and click on the **Select** button as shown.



- ◆ Select the solid from the graphics window as shown.



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

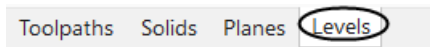


- ◆ Select the **OK** button to exit the **Machine Group Properties** dialog box.



### 2.5 Make Level 10 Invisible

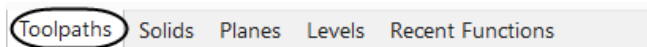
- ◆ Select the **Levels** tab.



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

Num...	Visible	Name	Entities	Level Set
✓ 1	X	Geometry	39	
10	X	stock	12	

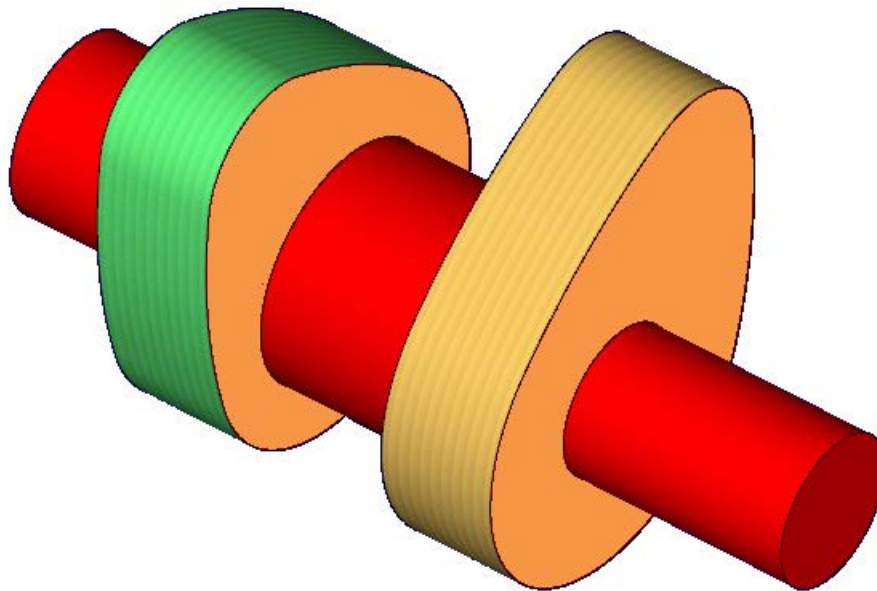
- ◆ Select the **Toolpaths** tab.



## 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.

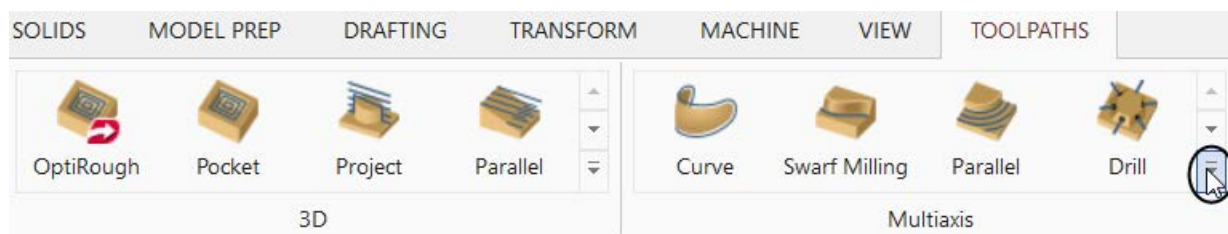
*Toolpath Preview:*



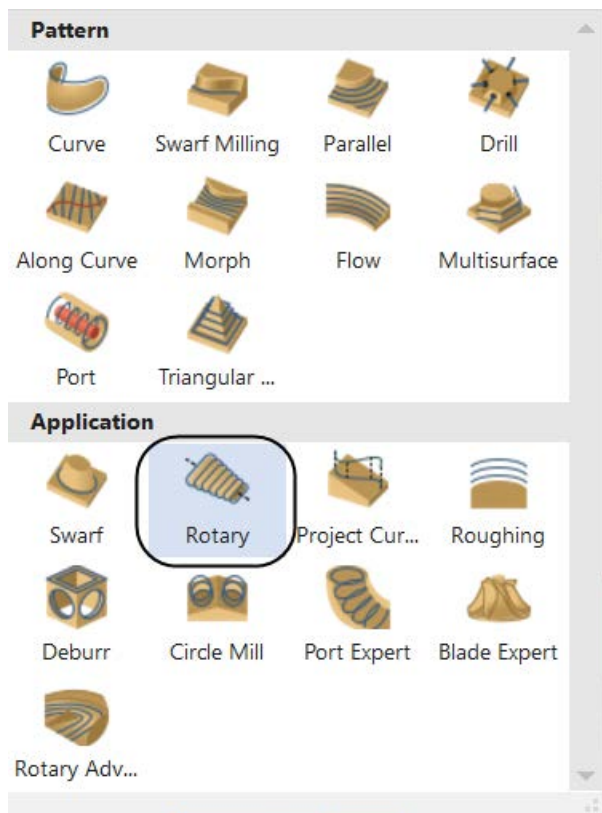
### 3.1 Machine the Right Side Cam using the Rotary Cut Method

#### TOOLPATHS

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



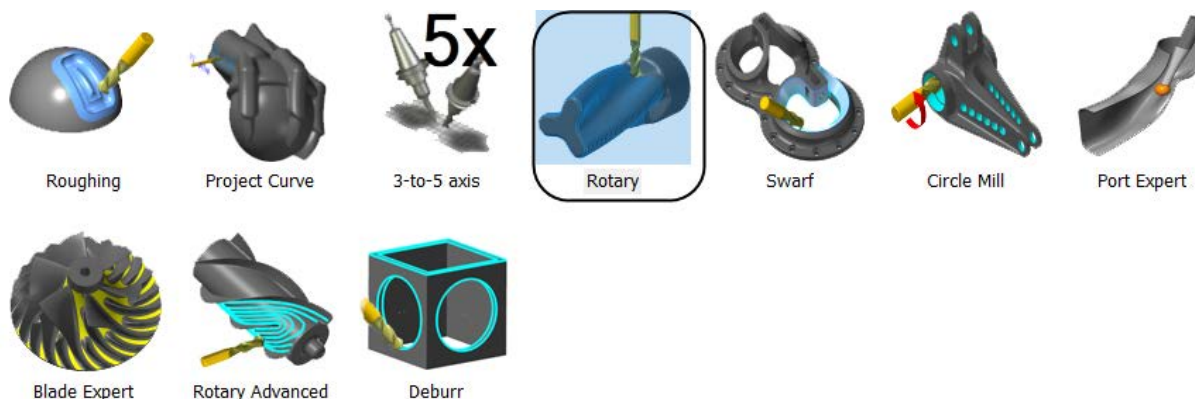
- ◆ From the **Application** group, select the **Rotary** icon.



### 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.



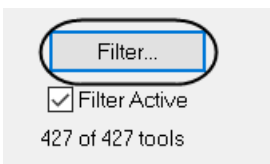
### 3.3 Set the Tool page parameters

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.

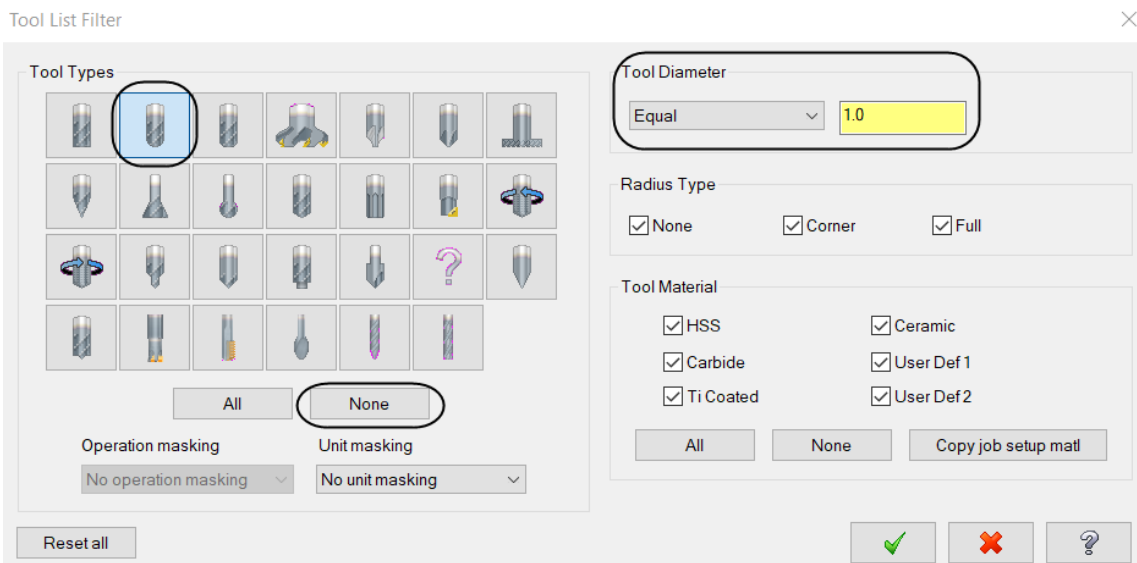
- ◆ From the **Tree View list**, select the **Tool** page.
- ◆ To select the tool, click on the **Select library tool** button as shown.



- ◆ 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.

#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	L
1	--	1 INCH B...	--	1.0	0.5	2

Right-click for options

Select library tool... ☐ Filter Active Filter...

☐ To batch

Tool diameter: 1.0  
Corner radius: 0.5

Tool name: 1 INCH BALL ENDMILL

Tool #: 1 Length offset: 1  
Head #: 0 Diameter offset: 1

Spindle direction: CW

Feed rate: 25.0 Spindle speed: 2139  
FPT: 0.0029 SFM: 559.9476  
Plunge rate: 15.0 Retract rate: 25.0


☐ Force tool change ☒ Rapid Retract

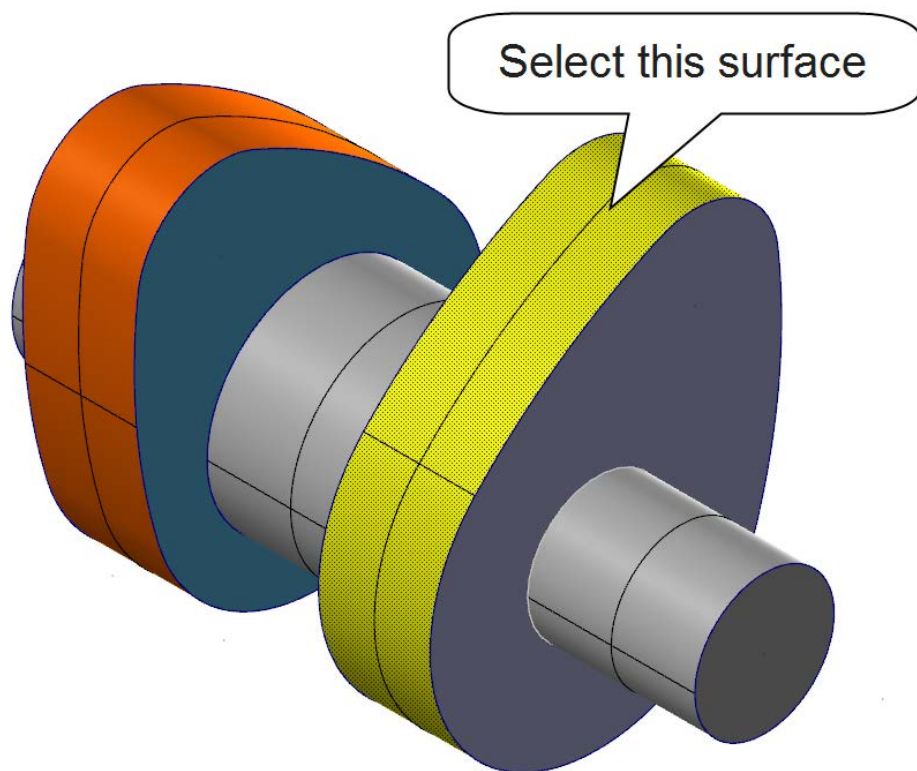
Comment  
Finish the right side cam using Rotary with the Rotary Cut method.



### 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 = 0.1**.
- ◆ **Cut tolerance = 0.025**.
- ◆ 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.

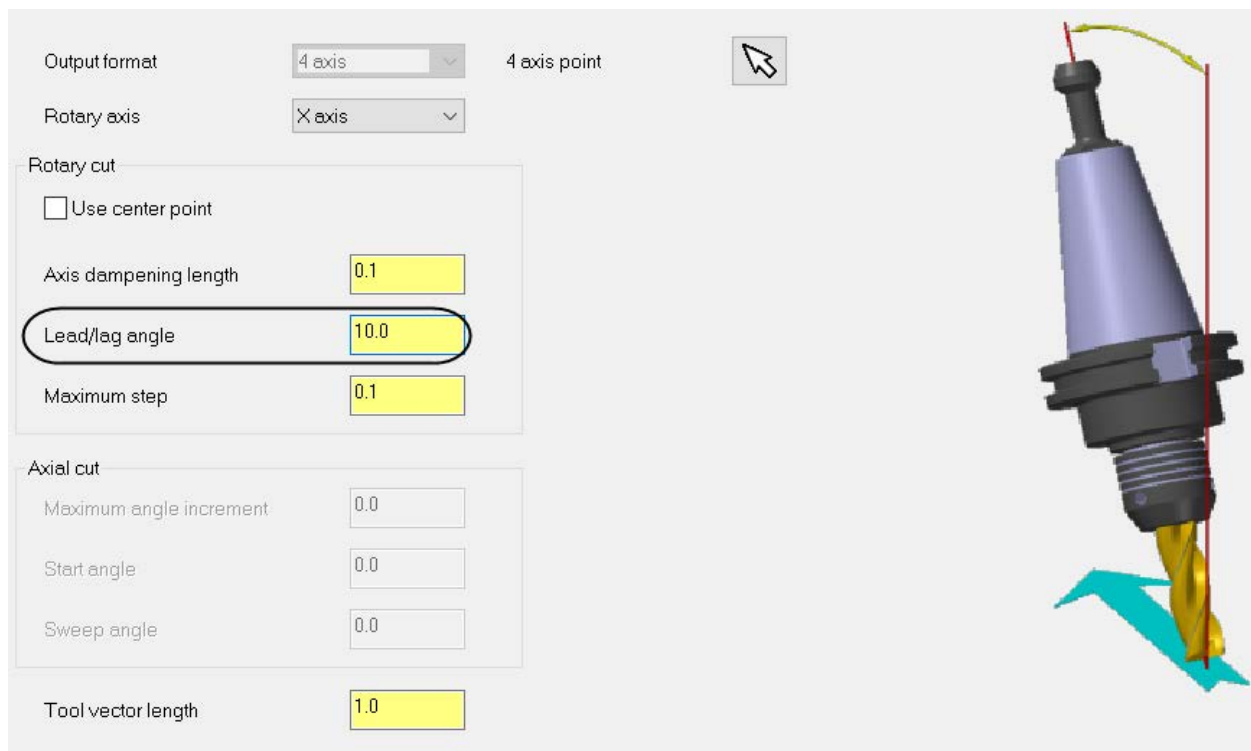
**Axis dampening** determines the angular change between the vectors. A longer axis dampening length creates less angular change between the vectors. A shorter length provides more tool positions and a toolpath that adheres closely to the surface.

**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 **25mm**.

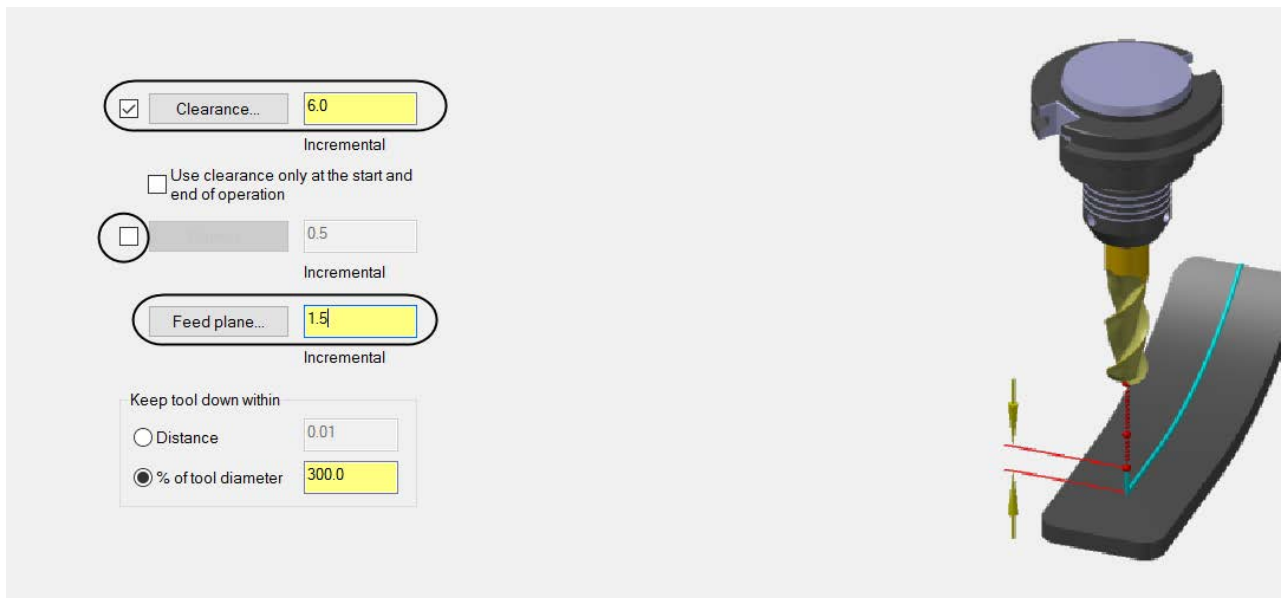
- ◆ Select the **Tool Axis Control** page and make the changes as shown.



### 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.



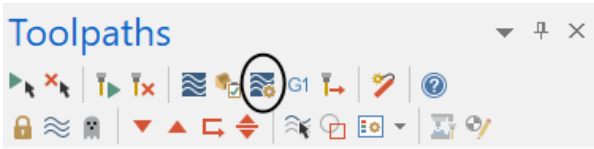
- ◆ Select the **OK** button to exit the **Multiaxis Toolpath - Rotary** dialog box.



### 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.

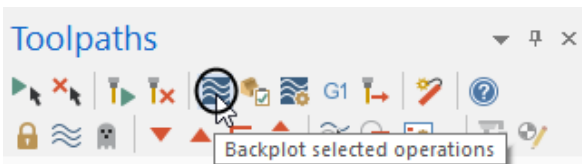
- ◆ From the **Toolpaths Manager**, select the **Simulator Options** icon.



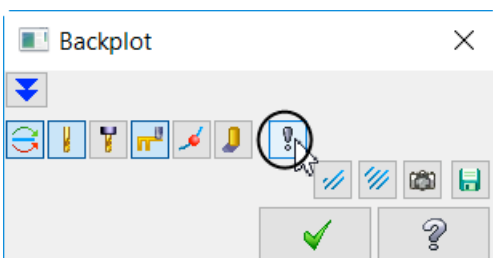
- ◆ Make sure that the **Stock Setup** is enabled as shown.



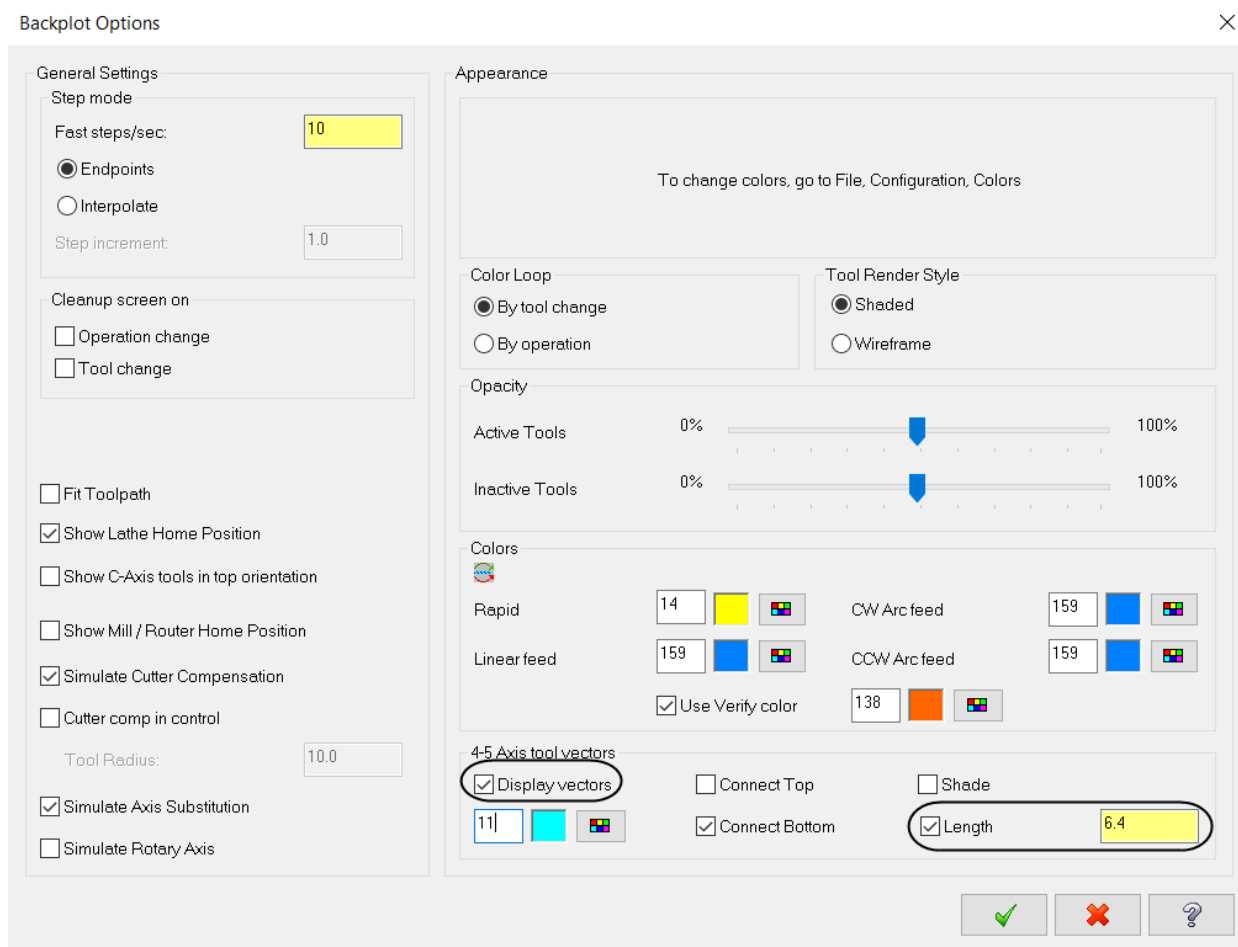
- ◆ Select the **OK** button to exit the **Simulator Options** dialog box. 
- ◆ Make sure that the toolpath is selected (signified by the green check mark on the folder icon).
- ◆ Select the **Backplot selected operations** icon.



- ◆ 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 **6.4**.



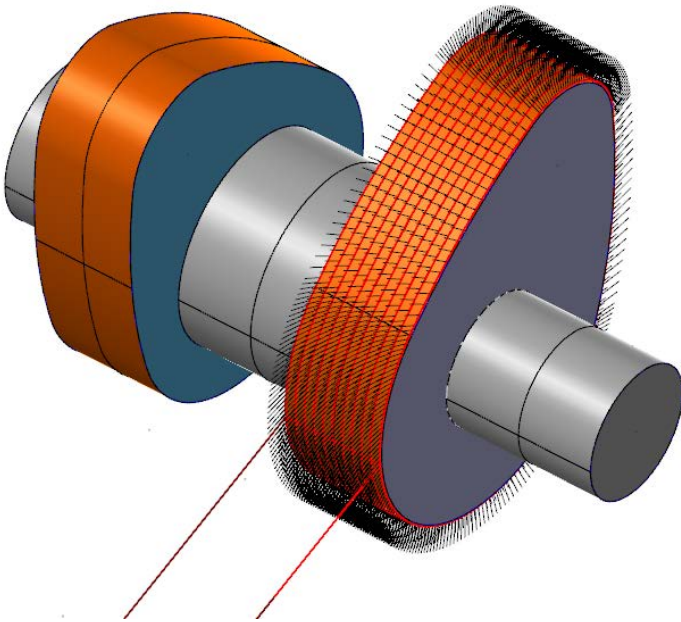
- ◆ Select the **OK** button to exit the **Backplot Options** dialog box.



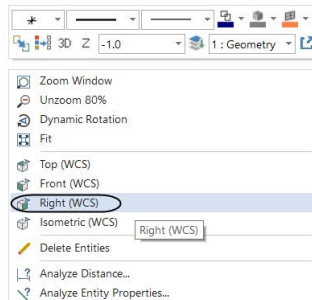
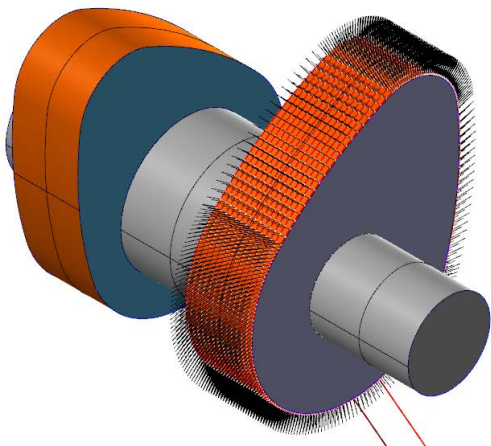
- ◆ Select the **Play** button to run **Backplot**.



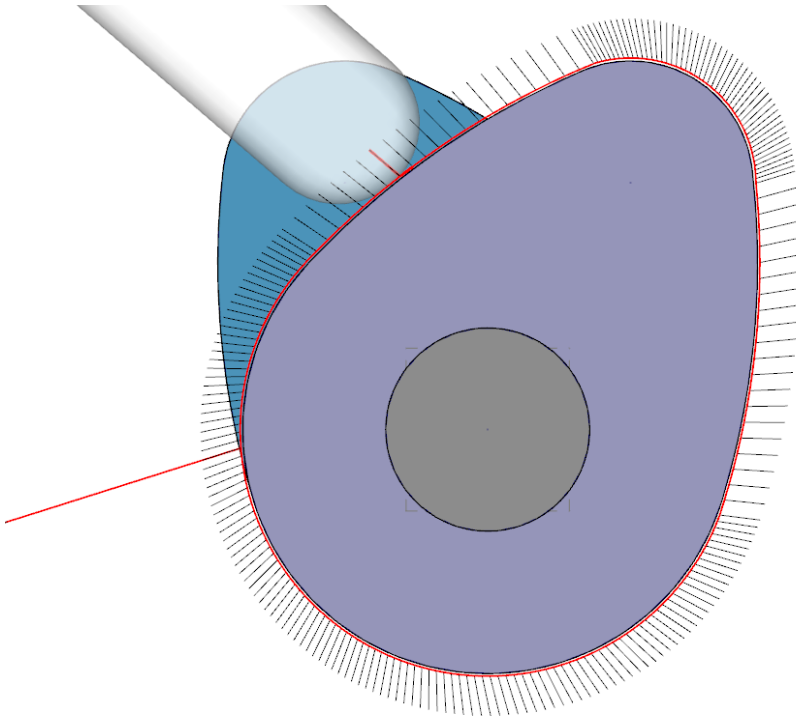
- ◆ The toolpath should look as shown.




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

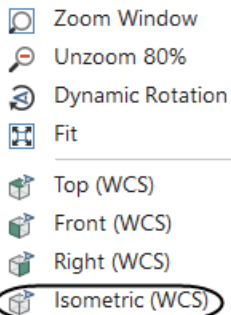


- ◆ Backplot the toolpath again as shown.



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

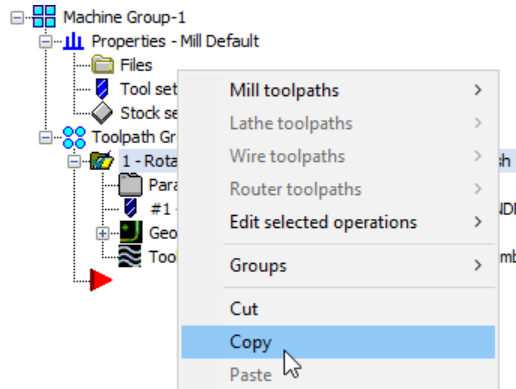
- ◆ Select the **OK** button to exit the **Backplot**. 
- ◆ Press **Alt + T** to remove the toolpath display from the graphics window.
- ◆ Right mouse click in the graphics window and select the **Isometric** view as shown.



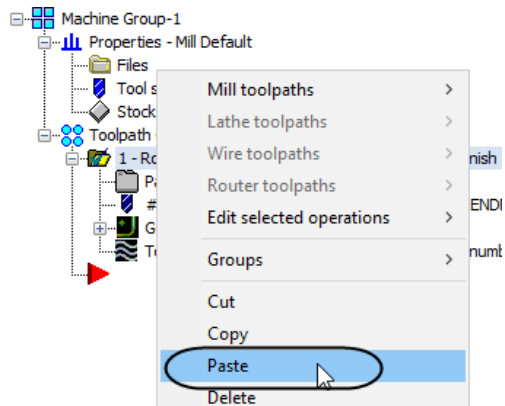


### 3.8 Machine the Left Side Cam using the Rotary Cut Method

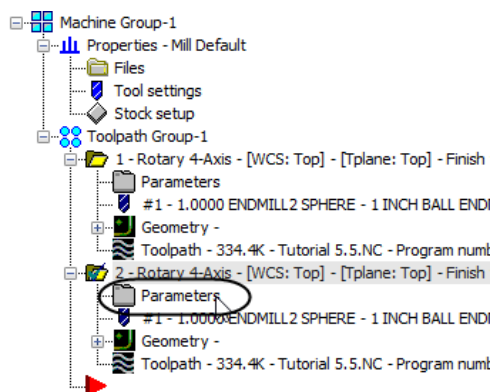
- ◆ Right mouse click in the **Toolpaths Manager** and select **Copy** as shown.



- ◆ 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.



# Did you like this book?

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

eMastercam has the largest Mastercam community on the web with over 56,000 members and over a million forum posts. While there, be sure to join the conversation!



**Training  
Solutions  
Offered**

Training Tutorials • Instructor Materials • Professional Courseware  
Handbooks • Video Training • Online Video eCourses  
Sitewide Subscription • Site Licenses • Online Community

# Mastercam® @Courses

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



## GETTING STARTED



## MILL ESSENTIALS



## MILL ADVANCED



## LATHE



## MULTIAxis ESSENTIALS



## MASTERCAM FOR SOLIDWORKS



## LOG IN & LEARN TODAY!

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

- NEW – FREE Getting Started eCourse
- Online previews with table of contents including the time it takes to complete each session.
- Mastercam 2019 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**

Only \$149 each!



[www.eMastercam.com/ecourses](http://www.eMastercam.com/ecourses)



[www.eMastercam.com](http://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.



**\$39.99**  
/1 month

**\$129.99**  
/4 months  
(save \$29.97)

**BEST VALUE:**

**\$359.99**  
/12 months  
(save \$119.89)

[www.eMastercam.com/  
sitewide-subscription](http://www.eMastercam.com/sitewide-subscription)

# Post processors that are requested worldwide.



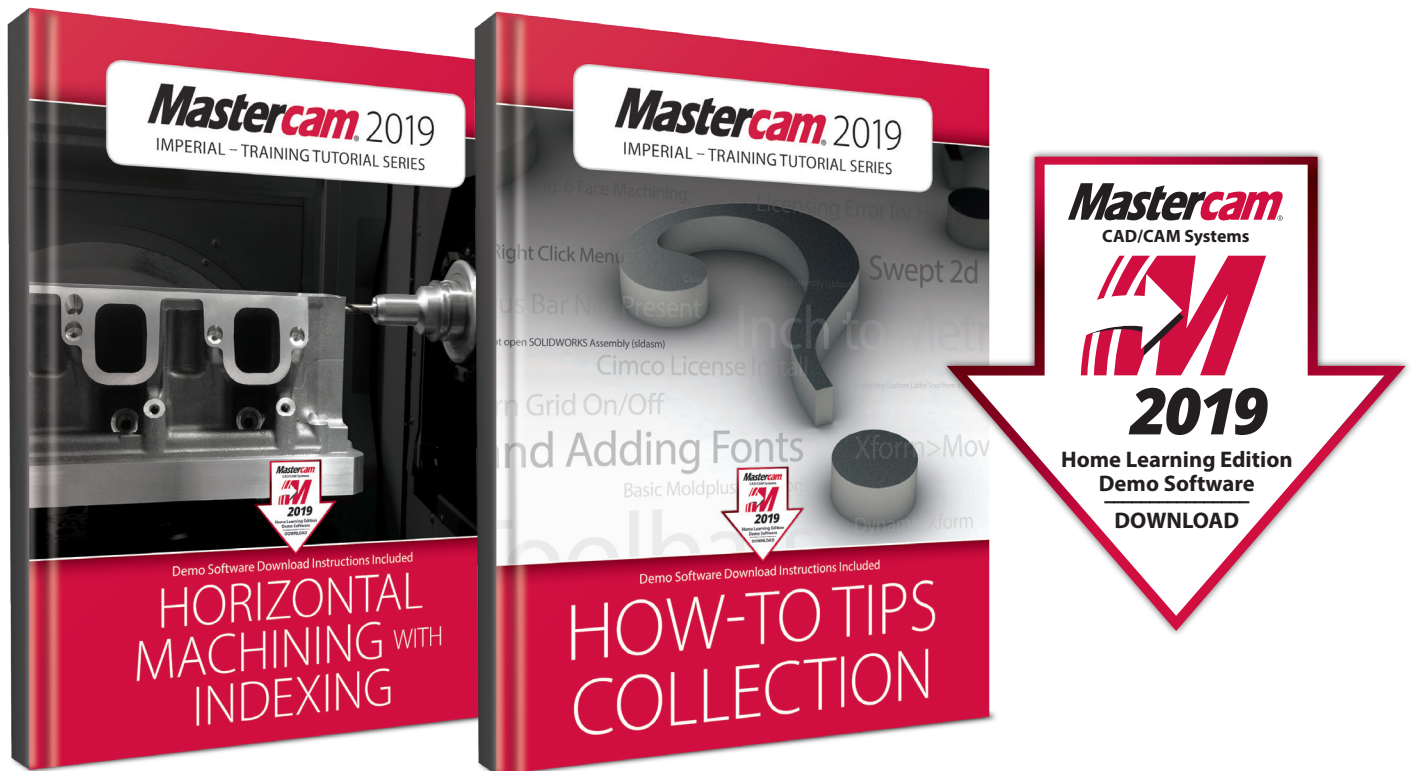
## We speak your machine's language.

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 that is continually developed and expanded to include current machines and controllers. 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.

[www.inhousesolutions.com/posts](http://www.inhousesolutions.com/posts)



# **Mastercam** 2019



Demo Software Download Instructions Included

# FREE eBooks

Download today  
[www.eMastercam.com/freebooks](http://www.eMastercam.com/freebooks)



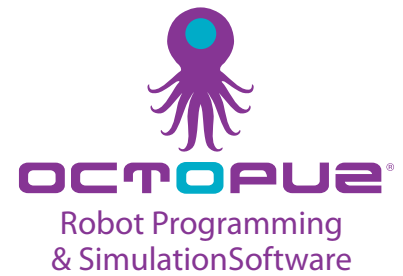
# DOES YOUR SCHOOL HAVE ROBOTS? YOU SHOULD CHECK OUT OCTOPUZ.

As easy to teach as it is to learn, OCTOPUZ is an offline robot programming and simulation software used world-wide in industry. Prepare your students for the real world of robotics, with OCTOPUZ they will gain the necessary skills to not only be hired, but to excel.



## DESIGN. PROGRAM. LEARN.

[www.OCTOPUZ.com/for-education](http://www.OCTOPUZ.com/for-education)



### With OCTOPUZ you can:

1

Create your  
custom cell.

2

Program your  
application.

3

Verify using  
simulation.

4

Generate and  
post code to ALL your  
robot brands.

#### Why use OCTOPUZ?



Get the most out of lab  
time, program offline and  
run online on the robot.



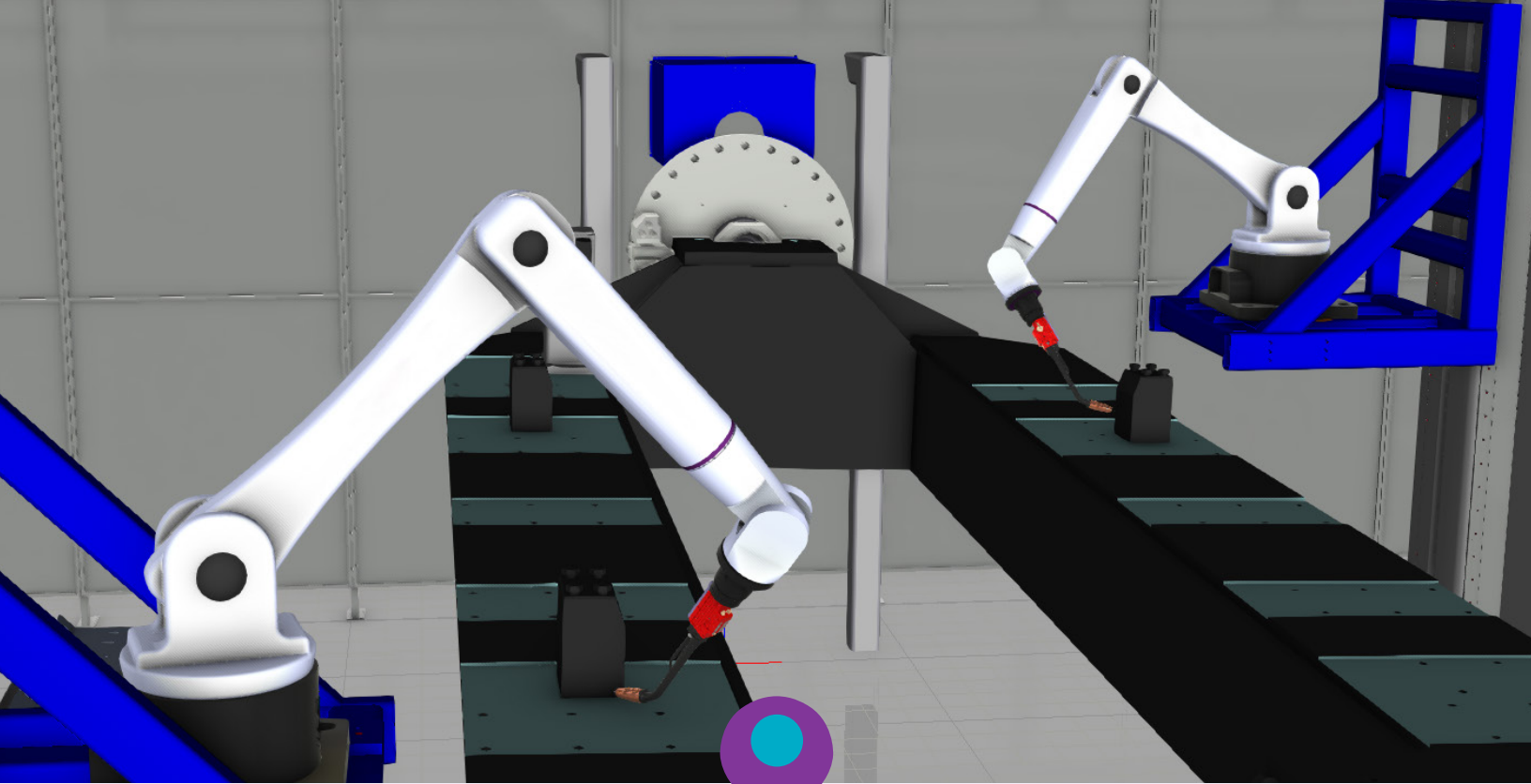
Protect your investment!  
OCTOPUZ can detect  
collisions offline.



Expose students to  
components and robots  
not available in your lab.



Bring 3D printing to life.



**OCTOPUZ®**

## Robot Programming & Simulation Software

The robot brands YOU use.  
As many robots as YOU need.  
Together, in YOUR cell... at the same time

### Why choose OCTOPUZ

Multiple robots? No problem!  
Most robot brands supported  
Simple simulation building  
Extensive component library

Full customization  
Application versatility  
CAM interface  
Complex kinematic systems

**CONTACT US**

SET UP YOUR FREE DEMO

[www.OCTOPUZ.com](http://www.OCTOPUZ.com)