

Mastercam 2019

IMPERIAL – TRAINING TUTORIAL SERIES



Demo Software Download Instructions Included

LATHE C & Y AXIS

Mastercam 2019

Lathe C and Y Axis Toolpaths Training Tutorial

To order more books:

Call 1-800-529-5517 or

Visit www.emastercam.com or

Contact your Mastercam dealer

Mastercam 2019 Lathe - C and Y Axis Toolpaths Training Tutorial

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

Software: Mastercam 2019

Author: Mariana Lendel

ISBN: 978-1-77146-774-2

Revision date: May 14, 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, MS, and MS-DOS are registered trademarks of Microsoft Corporation; N-See is a registered trademark of Microcompatibles, Inc.; Windows 7 and Windows 8 are registered trademarks of Microsoft Corporation.

MASTERCAM SHORTCUTS

Function	Keyboard Shortcut	Function	Keyboard Shortcut
Analyze entities	F4	Mastercam version, SIM serial number	Alt+V
AutoSave	Alt+A	Motion controller rotation point	Alt+F12
C-Hook or user app	Alt+C	Pan	Arrow keys
Configure Mastercam	Alt+F8	Paste from clipboard	Ctrl+V
Copy to clipboard	Ctrl+C	Redo an event that has been undone	Ctrl+Y
Cut to clipboard	Ctrl+X	Repaint	F3
Delete entities	F5	Rotate	Alt+Arrow keys
Drafting global options	Alt+D	Select all	Ctrl+A
Exit Mastercam	Alt+F4	Selection grid parameters	Alt+G
Fit geometry to screen	Alt+F1	Shading on/off	Alt+S
Gview–Back	Alt+3	Show/hide all axes (WCS, Cplane, Tplane)	Alt+F9
Gview–Bottom	Alt+4	Show/hide coordinate axes	F9
Gview–Front	Alt+2	Show/hide displayed toolpaths	Alt+T
Gview–Isometric	Alt+7	Show/hide Operations Manager panel	Alt+O
Gview–Left	Alt+6	Undo the last creation or event	Ctrl+U, Ctrl+Z
Previous Plane	Alt+P	Unzoom to 80% of original	Alt+F2
Gview–Right	Alt+5	Unzoom to previous or 50% of original	F2
Gview–Top	Alt+1	Zoom around target point	Ctrl+F1
Help	Alt+H	Zoom with window selection	F1
Hide entities	Alt+E	Zoom/unzoom by 5%	Page Up/Page Down
Levels Manager	Alt+Z		
Main attributes, set from entity	Alt+X		

CUSTOMIZE MASTERCAM

Create Your Own Keyboard Shortcuts

- ♦ Right mouse click in the **Ribbon**.
- ♦ **Customize the Ribbon**.
- ♦ **Keyboard shortcuts**, select the **Customize** button.
- ♦ Select the **Category**.
- ♦ Select a command and under **Press new shortcut key** enter the key combinations you want to assign to it.

Customize Quick Access Toolbar

- ♦ Right mouse click on a command in the **Ribbon**.
- ♦ Select **Add to Quick Access Toolbar**.

Customize the Ribbon

- ♦ Right mouse click in the **Ribbon**.
- ♦ Select **Customize the Ribbon**.
- ♦ Select a Mastercam command.
- ♦ Select a **Tab** or create a **New Tab**.
- ♦ Press **Add** button.

Customize the right-click menu

- ♦ Right mouse click in the **Ribbon**.
- ♦ Select **Customize the Ribbon**.
- ♦ Select **Context Menu**.
- ♦ Select the **Category** and then the command that you want to add.
- ♦ Once you click on the **Add** button the function will be added to the **Right mouse button menu**.

WAYS TO GET THE MOST FROM MASTERCAM

Mastercam Training

In-House Solutions offers unsurpassed industrial training for Mastercam and Octopuz. We have training facilities in a number of cities across Canada and some of our courses can also be offered onsite, depending on trainer availability. Learn more at eMastercam.com/store.

Our library of **Mastercam Training Solutions** consists of several product lines that cater to any learning style. Learn Mastercam at your own pace with our **Training Tutorials**, teach your students with the help of our **Instructor Kits**, learn the theory behind Mastercam with our **Handbooks**, get projects à-la-carte with our **Single Projects**, let our instructors show you best practices with our **Video Training** or go digital with our **eBooks**.

Mastercam Community

eMastercam is the one-stop web resource for Mastercam users. People from all over the world visit the site whether they are teaching, learning or working with Mastercam daily. Members can post questions, comments or share projects and success stories. Visit eMastercam.com and sign up for your free account today!

For downloaded pdf please visit

www.emastercam.com/qrc

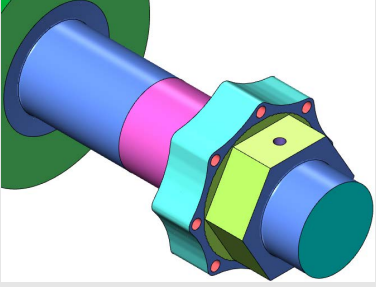
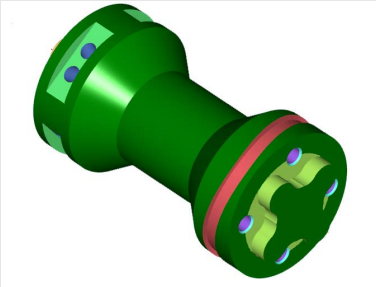
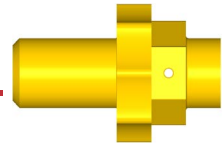
Tutorial	Geometry Functions	Surface and Toolpath Creation
<p>#1</p> 	<p>Create Rectangle. Create Line Parallel. Create Point. Edit Trim. Change Cplane. Create Polygon. Create Arc Endpoints. Transform Rotate. Create Fillet Chains. Create Bolt Circle.</p>	<p>Face . Rough. Finish. Groove. C-Axis Face Contour. C-Axis Face Drill. C-Axis Drill. Cutoff.</p>
<p>#2</p> 	<p>Levels Manager. Planes. Create Line Endpoints. Transform Mirror. Manual Spline. Create Ellipse. Create Circle Center Point. Edit Trim. Transform Rotate. Create Fillet. Create Point Position. Create Spline.</p>	<p>C-Axis Face Contour. C-Axis Cross Contour with Y-Axis. C-Axis Cross Drill. C-Axis Face Drill. Stock Transfer. C-Axis Face Contour. C-Axis Face Drill. C-Axis Contour.</p>

Table of Contents

Getting Started	1
Tutorials:	
Tutorial #1 - C-axis Face Contour, Face Drilling & C-axis Drilling Toolpath	15
Tutorial #2 - Cross Contour With Y axis Toolpath And C-axis Contour Toolpaths	139
Quiz Answers	285



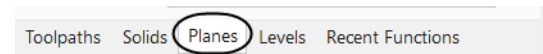
GEOMETRY CREATION

STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

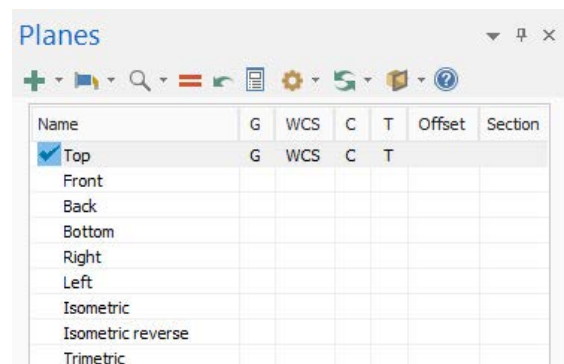
Before starting the geometry creation, we should customize the toolbars to see the toolbars required to create the geometry and machine a 3D part. See **Getting Started** for details.

NOTE: Because we will be creating **C-Axis** toolpaths, we will be using the **Top** and **Right** construction planes for this tutorial.

- ♦ To open the **Planes Manager** panel, select the **Planes** tab as shown.



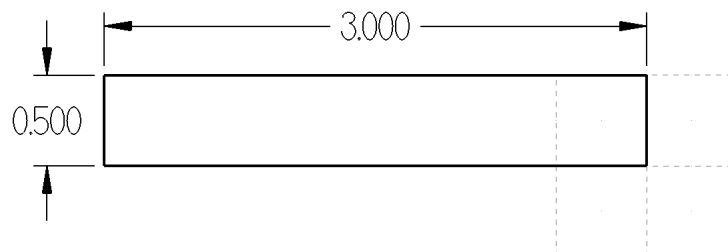
- ♦ Make sure that **WCS**, planes and the graphics view are all set to **TOP** as shown.

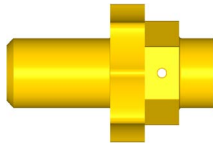


STEP 2: CREATE A RECTANGLE

In this step, you will create a rectangle knowing the width, height, and anchor position.

Step Preview:

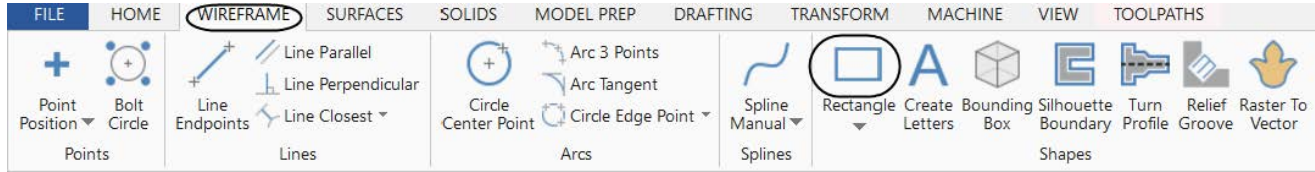




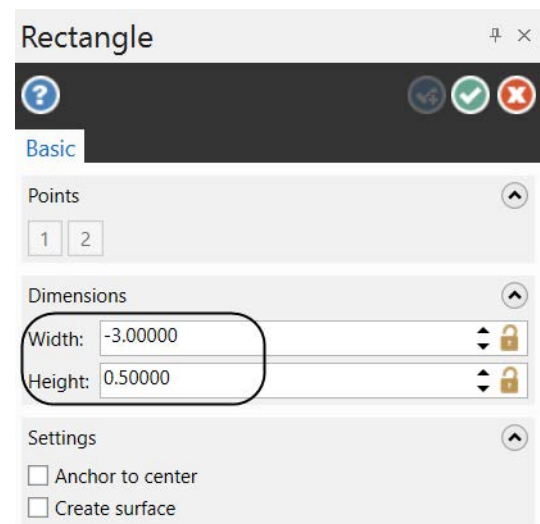
2.1 Create the -3.0" by 0.5" rectangle

WIREFRAME

- From the **Shapes** group, select the **Rectangle** icon as shown.



- Enter the **Width** and the **Height** and make sure that **Anchor to center** and **Create surface** buttons are not selected (highlighted) as shown.



- Press **Enter** after typing the values to see a preview of the rectangle.
- [Select position for first corner]: Select the **Origin** as shown in [Figure: 2.1.1](#).

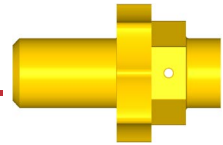
Figure: 2.1.1



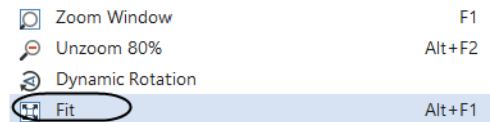
- Make sure that when selecting the origin, the visual cue of the cursor changes as shown.





- Select the **OK** button to exit the **Rectangle** panel.

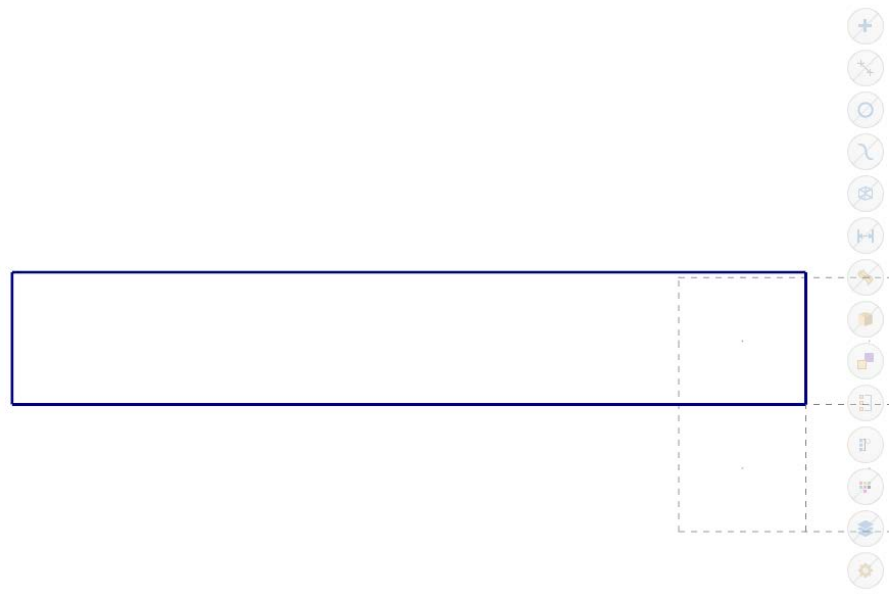


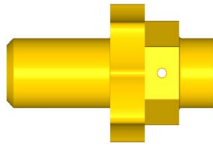
- ♦ Right mouse click in the graphics window and select **Fit** to fit the drawing to the screen. You can also press **Alt + F1**.



NOTE: During the geometry creation of this tutorial, if you make a mistake you can undo the last step using the **Undo** icon.  You can undo as many steps as needed. If you delete or undo a step by mistake, just use the **Redo** icon.  To delete unwanted geometry, select it first and then press **Delete** from the keyboard. You can zoom or unzoom the geometry by scrolling the mouse wheel up or down.

- ♦ The geometry should look as shown.

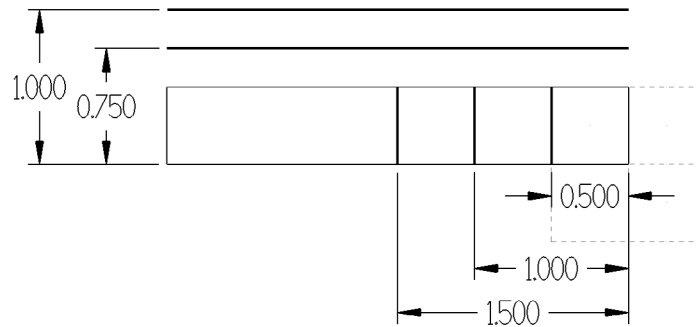




STEP 3: CREATE PARALLEL LINES

In this step, you will create parallel lines to represent the grooves on the left hand side of the part.

Step Preview:

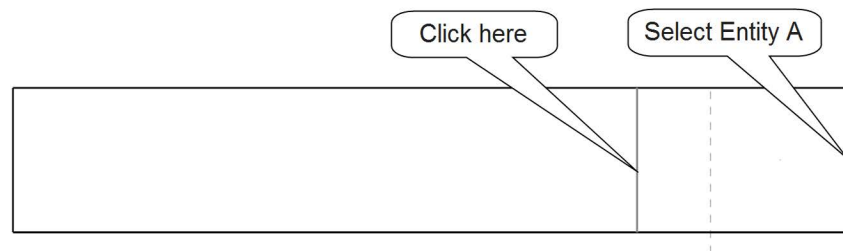


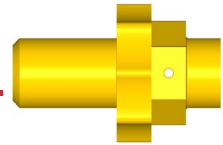
3.1 Create the vertical lines

WIREFRAME

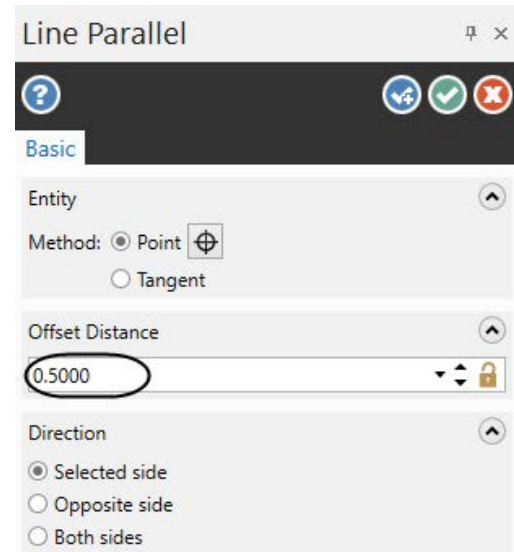
- ♦ From the **Lines** group, select the **Line Parallel** icon.
- ♦ [Select a line]: Select **Entity A** as shown in [Figure: 3.1.1](#).
- ♦ [Select the point to place a parallel line through]: Click on a point to the left of **Entity A** as shown in [Figure: 3.1.1](#).

Figure: 3.1.1





- ◆ In the **Line Parallel** panel, enter the **Offset Distance 0.5** and press **Enter**.



- ◆ Select the **OK and Create New Operation** button to stay within the command.
- ◆ [Select a line]: Select **Entity A** as shown before.
- ◆ [Select the point to place a parallel line through]: Click on a point to the left of **Entity A** as shown before.
- ◆ In the **Line Parallel** panel, enter the **Offset Distance 1.0** and press **Enter**.



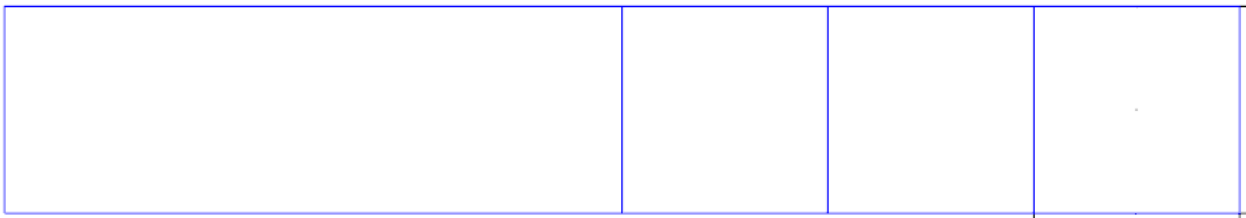
- ◆ Select the **OK and Create New Operation** button to stay within the command.
- ◆ [Select a line]: Select **Entity A** as shown before.
- ◆ [Select the point to place a parallel line through]: Click on a point to the left of **Entity A** as shown before.
- ◆ In the **Line Parallel** panel, enter the **Offset Distance 1.5** and press **Enter**.

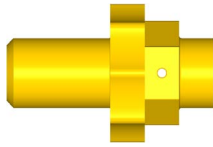


- ◆ Select the **OK and Create New Operation** button to stay within the command.
- ◆ The part should appear as shown in [Figure: 3.1.2](#).



Figure: 3.1.2

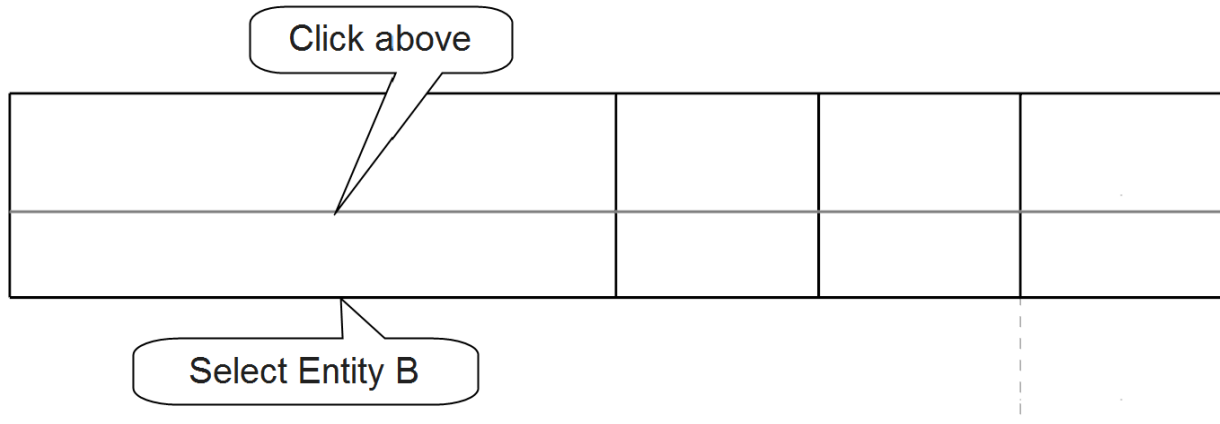




3.2 Create the horizontal lines

- ♦ [Select a line]: Select **Entity B** as shown in [Figure: 3.2.1](#).
- ♦ [Select the point to place a parallel line through]: Click on a point above **Entity B** as shown in [Figure: 3.2.1](#).

Figure: 3.2.1



- ♦ In the **Line Parallel** panel, enter the **Offset Distance 0.75** and press **Enter**.

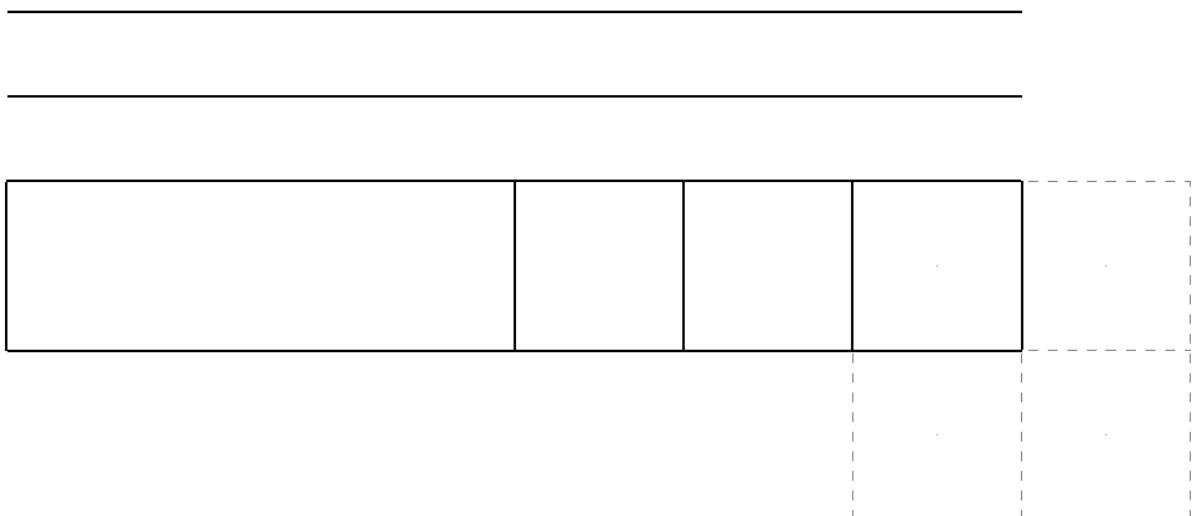


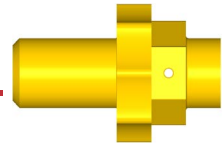
- ♦ Select the **OK and Create New Operation** button to stay within the command.
- ♦ [Select a line]: Select **Entity B** as shown in [Figure: 3.2.1](#).
- ♦ [Select the point to place a parallel line through]: Click on a point above **Entity B** as shown in [Figure: 3.2.1](#).
- ♦ In the **Line Parallel** panel, enter the **Offset Distance 1.0** and press **Enter**.



- ♦ Select the **OK** button to exit the command.
- ♦ The part should appear as shown in [Figure: 3.2.2](#).

Figure: 3.2.2

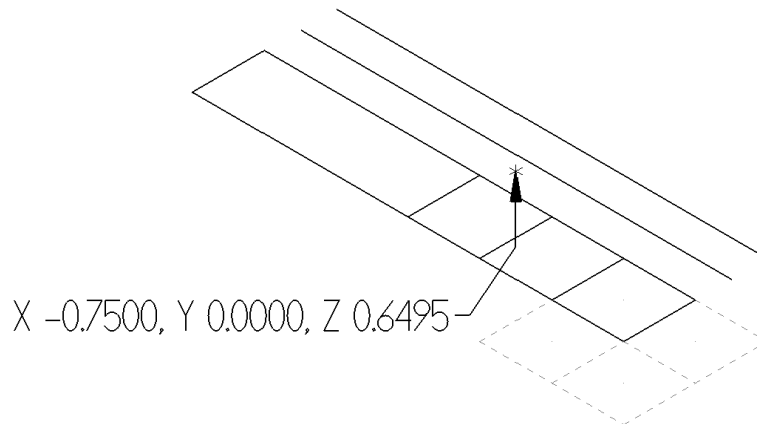




STEP 4: CREATE A POINT

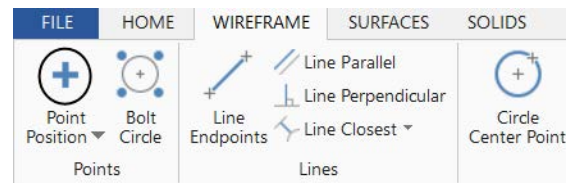
In this step, we will use the **Create Point Position** command to create a point. This point will be used by Mastercam as a drill position locator while creating the **C-Axis Drill** toolpath.

Step Preview:



WIREFRAME

- From the **Points** group, select the **Point Position** icon as shown.



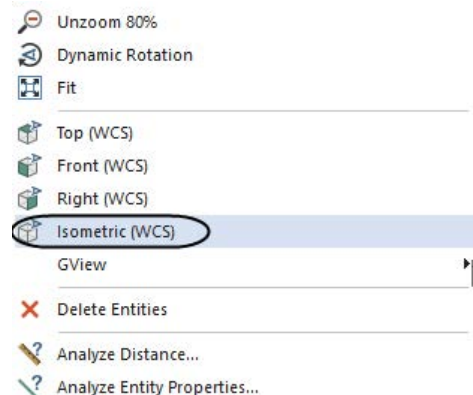
- [Create point position]: Select the **AutoCursor Fast Point** icon in the **General Selection** toolbar as shown.

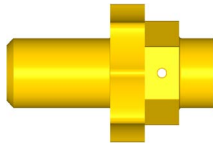


- In the coordinate field, enter the coordinates **-0.75, 0, 0.6495** as shown. Press the **Enter** key once you have finished.

-0.75,0,0.6495

- Right mouse click in the graphics window and select the **Isometric** view as shown.

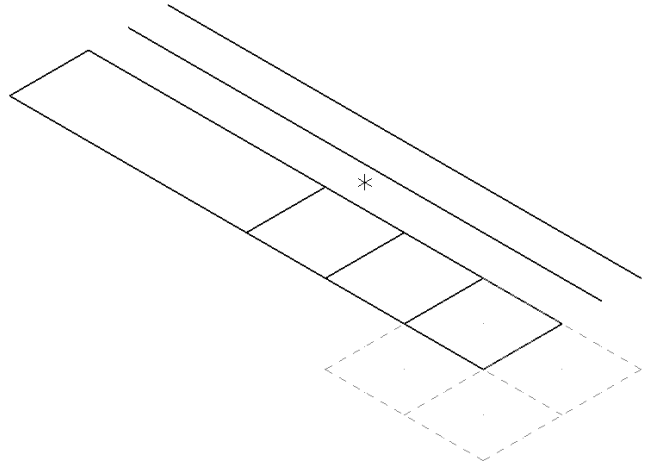




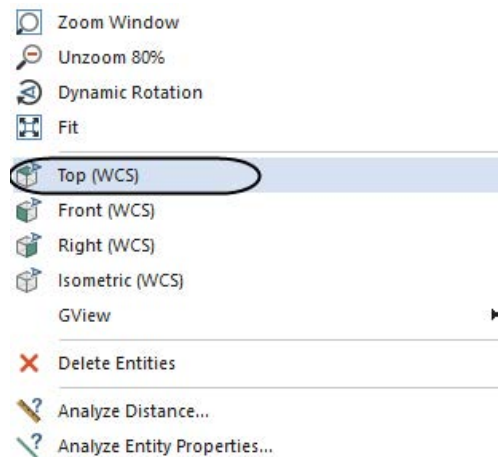
- ♦ Select the **OK** button to exit the command.
- ♦ The point should appear as shown in [Figure: 4.0.1](#).

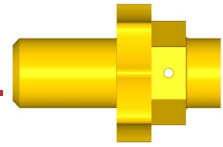


Figure: 4.0.1



- ♦ Right mouse click in the graphics window and select the **Top** view as shown.

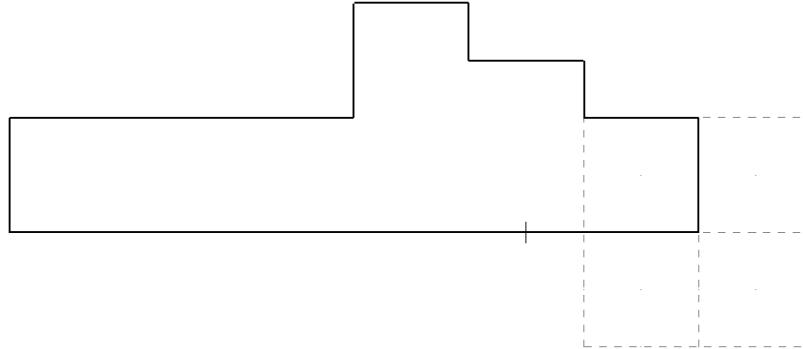




STEP 5: TRIM TO COMPLETE THE GEOMETRY IN THE TOP VIEW

In this step, we will use the **Trim Break Extend** command to trim the lines to complete the geometry in the top view.

Step Preview:

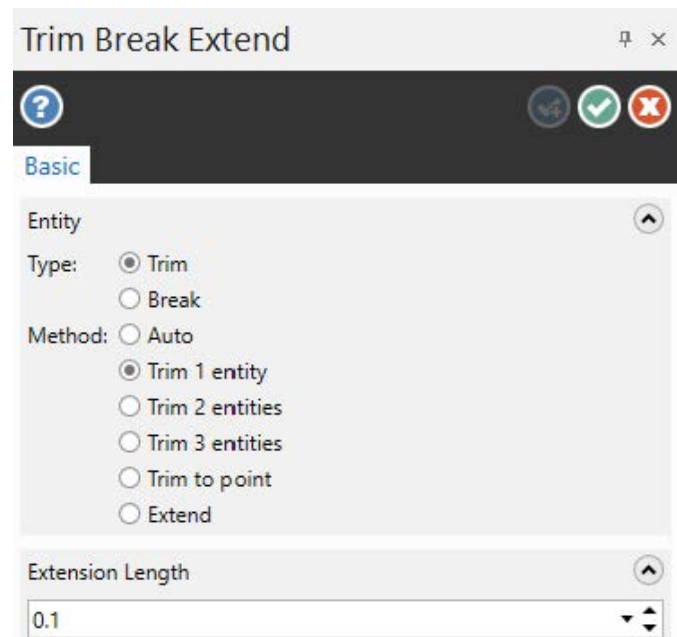


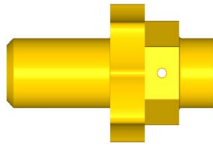
WIREFRAME

- ♦ From the **Modify** group, select the **Trim Break Extend** icon as shown.



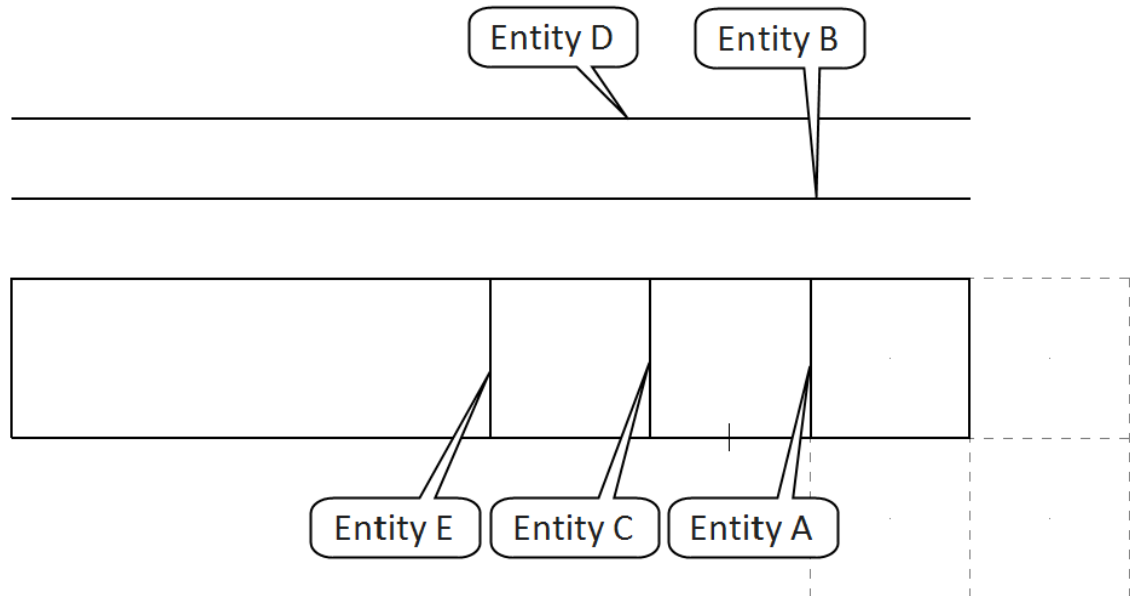
- ♦ In the **Trim Break Extend** panel, enable **Trim 1 entity** as shown.





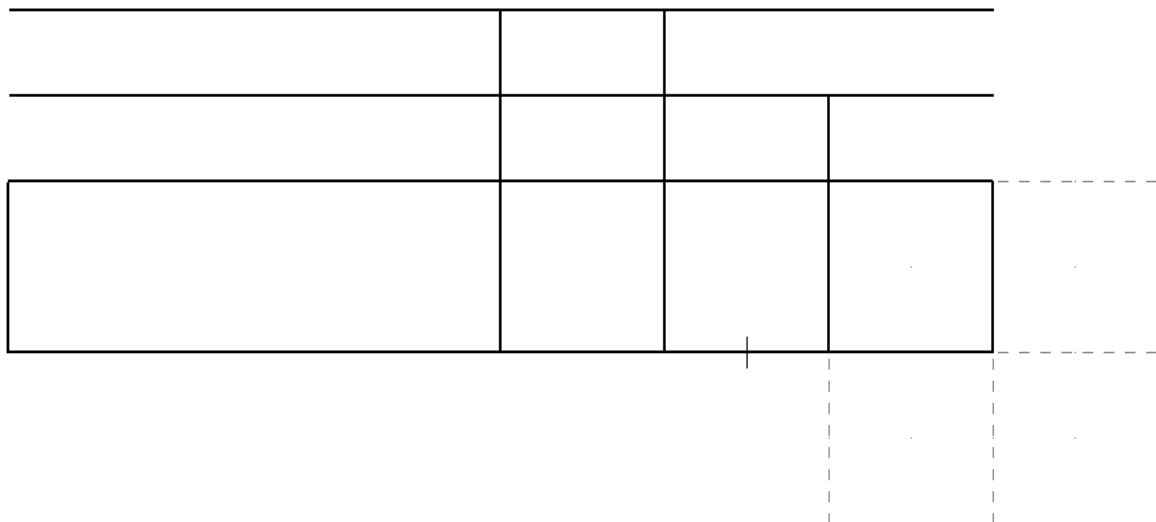
- ♦ [Select the entity to trim/extend]: Select **Entity A** as shown in [Figure: 5.0.1](#).
- ♦ [Select the entity to trim/extend to]: Select **Entity B** as shown in [Figure: 5.0.1](#).

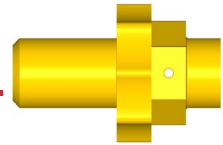
Figure: 5.0.1



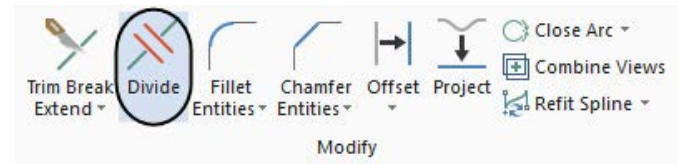
- ♦ [Select the entity to trim/extend]: Select **Entity C** as shown in [Figure: 5.0.1](#).
- ♦ [Select the entity to trim/extend to]: Select **Entity D** as shown in [Figure: 5.0.1](#).
- ♦ [Select the entity to trim/extend]: Select **Entity E** as shown in [Figure: 5.0.1](#).
- ♦ [Select the entity to trim/extend to]: Select **Entity D** as shown in [Figure: 5.0.1](#).
- ♦ The part should appear as shown in [Figure: 5.0.2](#).

Figure: 5.0.2

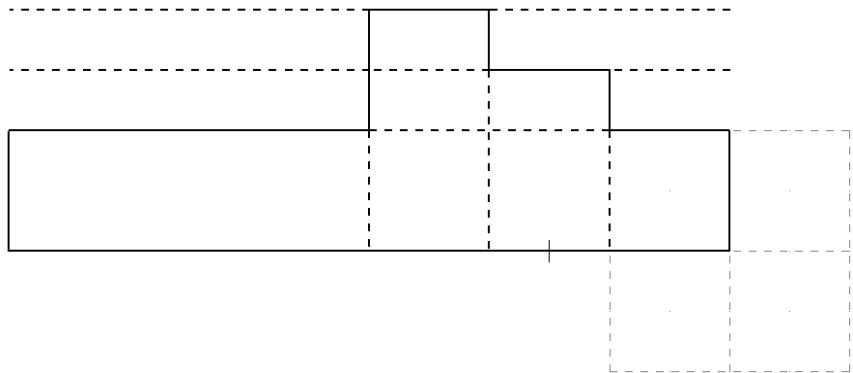




- ♦ From the **Modify** group, select the **Divide** icon as shown.

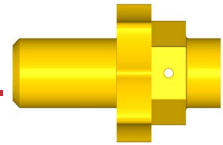
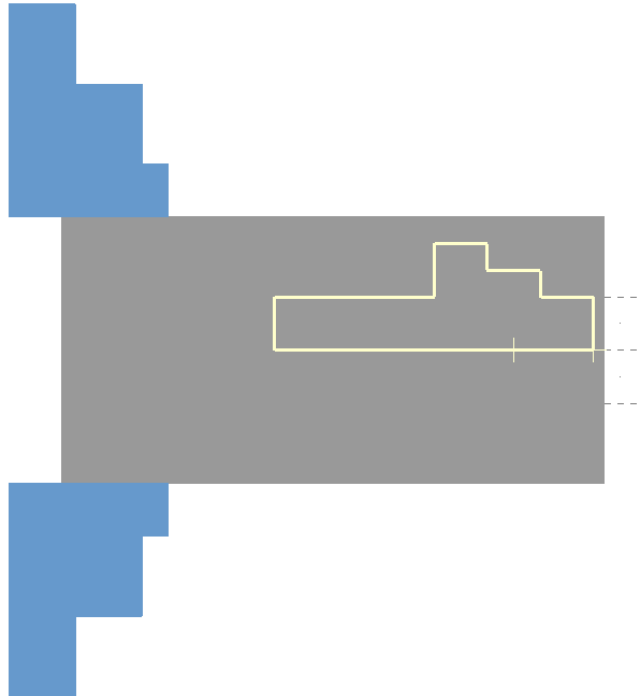


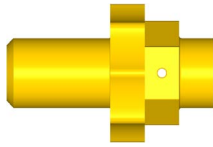
- ♦ [Select the curve to divide/delete]: Select the portions of line that appear as hidden (dotted) lines below to delete them with the divide function.

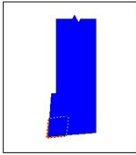
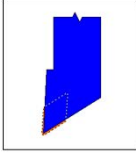
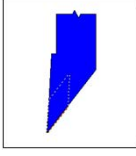
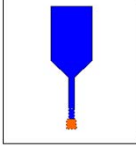
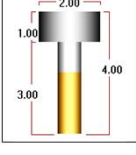
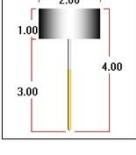
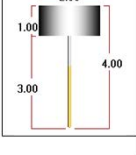
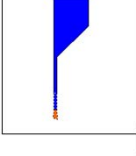


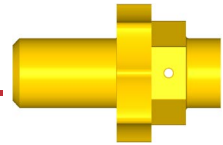
- ♦ Select the **OK** button to exit the command.



**TOOLPATH CREATION****PART SETUP:**

**SETUP SHEET:**

	TYPE: General Turning Tool DIA OFFSET: HOLDER: NUMBER: 1 LENGTH OFFSET: T0101: General Turning Tool - OD ROUGH RIGHT - 80 DEG.	FLUTE LENGTH: OVERALL LENGTH: CORNER RAD: # OF FLUTES:
	TYPE: General Turning Tool DIA OFFSET: HOLDER: NUMBER: 2 LENGTH OFFSET: T0202: General Turning Tool - OD 55 deg Right	FLUTE LENGTH: OVERALL LENGTH: CORNER RAD: # OF FLUTES:
	TYPE: General Turning Tool DIA OFFSET: HOLDER: NUMBER: 3 LENGTH OFFSET: T0303: General Turning Tool - OD FINISH RIGHT - 35 DEG.	FLUTE LENGTH: OVERALL LENGTH: CORNER RAD: # OF FLUTES:
	TYPE: Grooving Tool DIA OFFSET: HOLDER: NUMBER: 4 LENGTH OFFSET: T0404: Grooving Tool - OD GROOVE CENTER - MEDIUM	FLUTE LENGTH: OVERALL LENGTH: CORNER RAD: # OF FLUTES:
	TYPE: Endmill1 Flat DIA OFFSET: 0 HOLDER: DEFAULT HOLDER NUMBER: 1 LENGTH OFFSET: 0 #1 - 0.7500 ENDMILL1 FLAT - 3/4 FLAT ENDMILL	FLUTE LENGTH: 2.0 OVERALL LENGTH: 3.0 CORNER RAD: 0.0 # OF FLUTES: 4
	TYPE: Spot Drill DIA OFFSET: 0 HOLDER: DEFAULT HOLDER NUMBER: 2 LENGTH OFFSET: 0 #2 - 0.1250 SPOT DRILL - 1/8 SPOTDRILL	FLUTE LENGTH: 2.0 OVERALL LENGTH: 3.0 CORNER RAD: 0.0 # OF FLUTES: 2
	TYPE: Drill DIA OFFSET: 0 HOLDER: DEFAULT HOLDER NUMBER: 3 LENGTH OFFSET: 0 #3 - 0.1250 DRILL - 1/8 DRILL	FLUTE LENGTH: 2.0 OVERALL LENGTH: 3.0 CORNER RAD: 0.0 # OF FLUTES: 2
	TYPE: Grooving Tool DIA OFFSET: HOLDER: NUMBER: 5 LENGTH OFFSET: T0505: Grooving Tool - OD CUTOFF RIGHT	FLUTE LENGTH: OVERALL LENGTH: CORNER RAD: # OF FLUTES:

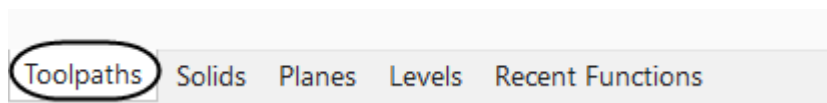


STEP 14: SELECT THE MACHINE AND SET UP THE STOCK

In Mastercam, you select a **Machine Definition** before creating any toolpath. 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 a Lathe exercise (2D toolpaths), we need just a basic machine definition.

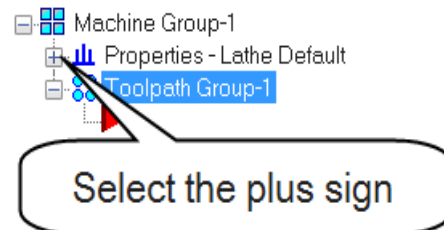
NOTE: For the purpose of this tutorial, we will be using the **Default Lathe** machine.

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

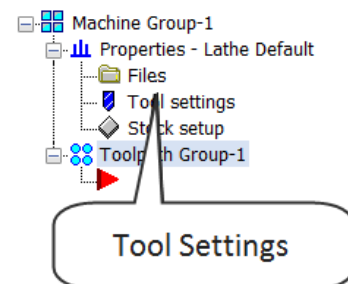


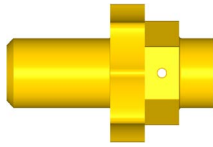
MACHINE

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



- ♦ Select **Tool Settings** to set the tool parameters.





- ♦ Select the **Tool Settings** tab and change the parameters to match the screen shot in [Figure: 14.0.1](#).

Figure: 14.0.1

The screenshot shows the 'Machine Group Properties' dialog box with the 'Tool Settings' tab selected. The 'Default program number' is set to 1. Under 'Feed Calculation', 'From tool' is selected. Under 'Toolpath Configuration', 'Assign tool numbers sequentially' and 'Warn of duplicate tool numbers' are checked. Under 'Advanced options', 'Override defaults with modal values' is unchecked. Under 'Tool Clearance', 'Rapid moves' is 0.03 and 'Entry/Exit' is 0.015. Under 'Sequence number', 'Start' is 100.0 and 'Increment' is 10.0. The 'Material' field is 'STEEL inch - 1030 - 200 BHN'. Buttons for 'Lathe Edit...', 'Select...', and 'Mill Edit...' are visible. At the bottom are 'OK', 'Cancel', and 'Help' buttons.

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

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

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

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

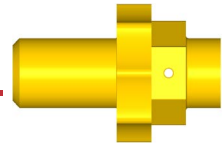
Feed Calculation set **From tool** uses feed rate, plunge rate, retract rate and spindle speed from the tool definition.

- ♦ Select the **Stock Setup** tab.

The screenshot shows the 'Machine Group Properties' dialog box with the 'Stock Setup' tab selected. The 'Left Spindle' is selected with a radio button, and the 'Right Spindle' is also selected with a radio button. Both are labeled '(Not Defined)'. There is a 'Properties...' button and a 'Delete' button.

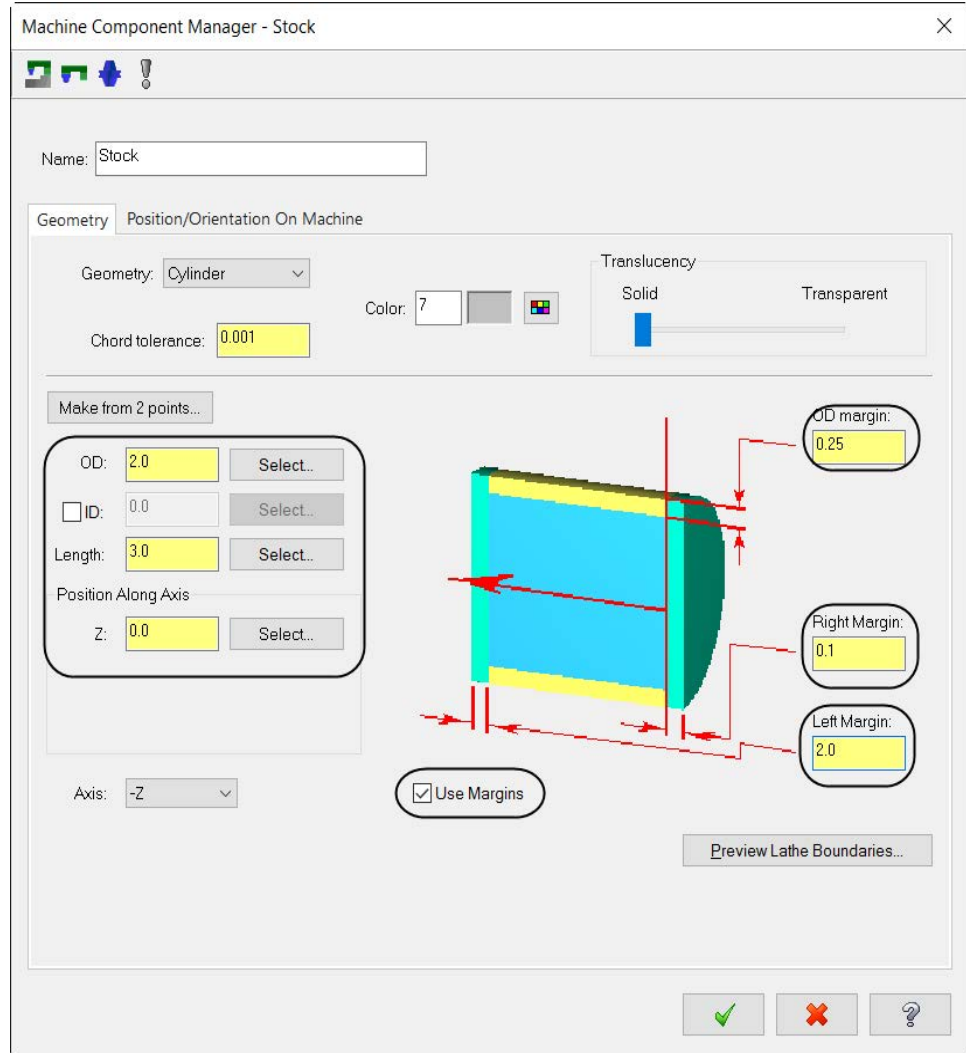
- ♦ Choose the **Properties** button to set up the stock for the **Left Spindle**.

The screenshot shows the 'Stock Setup' dialog box. It has two radio buttons: 'Left Spindle' (selected) and 'Right Spindle'. Both are labeled '(Not Defined)'. There is a 'Properties...' button and a 'Delete' button.



- ◆ Define the stock by setting the stock geometry to **Cylinder** and entering the stock dimensions. Ensure you enable **Use Margins** and enter in the values as shown in [Figure: 14.0.2](#).

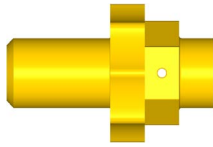
Figure: 14.0.2



NOTE: The **stock** model that you create can be displayed with the part geometry when viewing the file or the toolpaths, during backplot, or while verifying toolpaths. You can create stock on the left or right spindle.

- ◆ Select the **OK** button to exit the **Stock Setup** page.

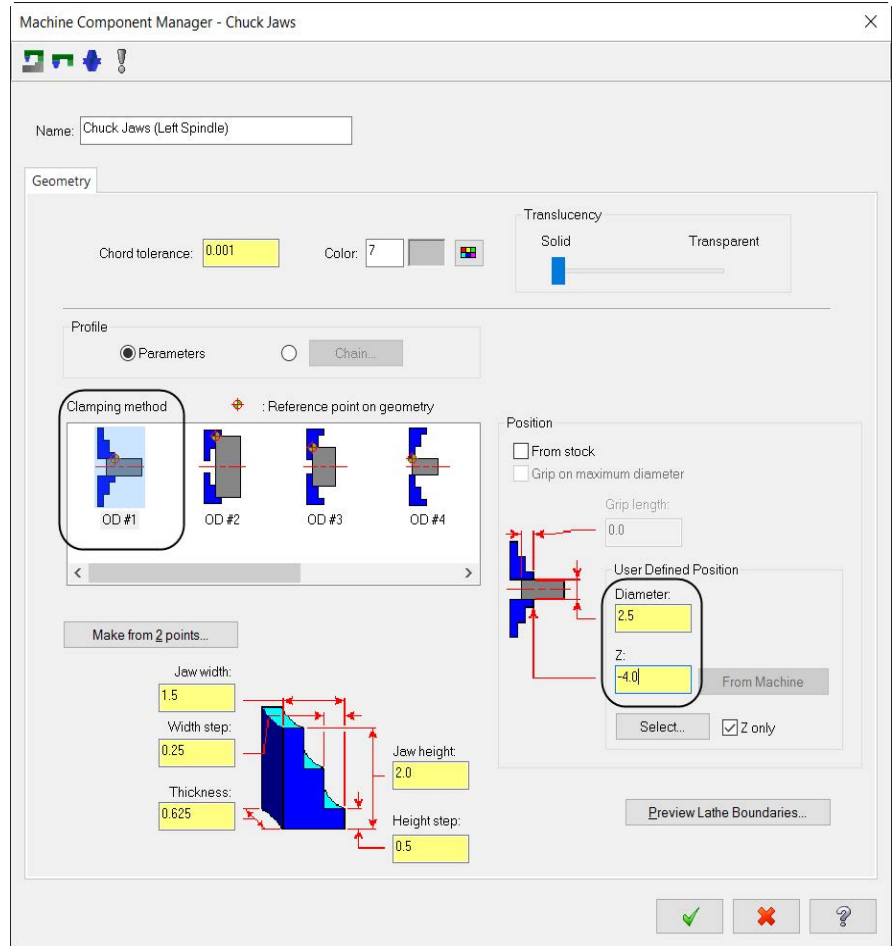




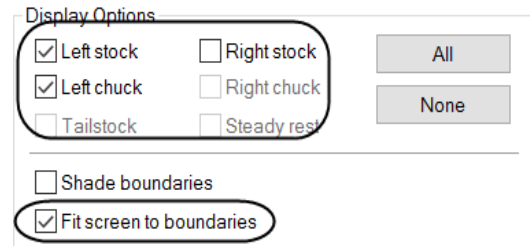
- ◆ Ensure that **Left spindle** is selected and then select the **Properties** button in the **Chuck Jaws** area as shown.

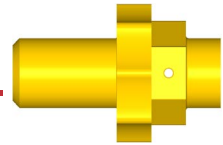



- ◆ Make the necessary changes to define the chuck size, the **Clamping method**, and the stock **Position**.
- ◆ Ensure that you choose the clamping method **OD #1** as shown in the graphic below.

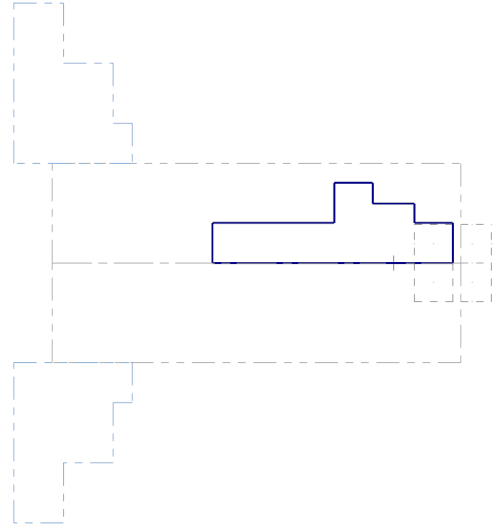


- ◆ Select the **OK** button to exit the **Chuck Jaws** dialog box.
- ◆ Enable **Fit screen to boundaries** and disable **Right stock** in the **Display Options** area.





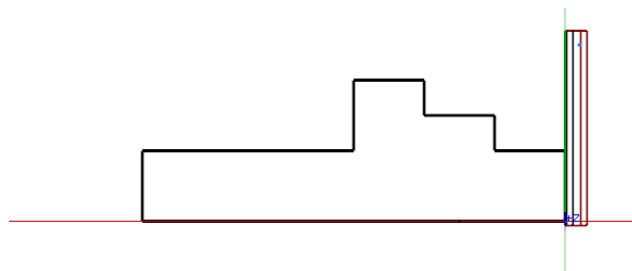
- ♦ Select the **OK** button to exit **Machine Group Properties**. 
- ♦ Press **Alt + F1** to fit the stock and the chuck to the graphics window as shown.



STEP 15: FACE THE PART

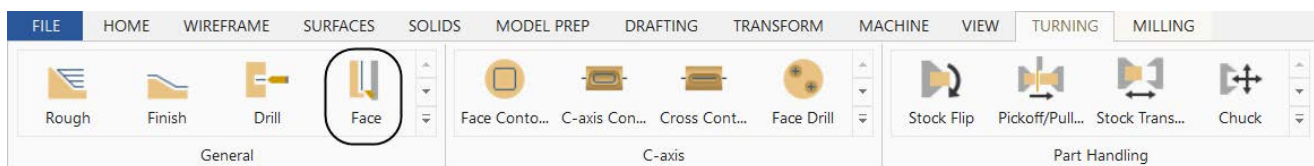
Facing toolpaths allow the user to quickly clean the stock from one end of the part and create an even surface for future operations. Note that we do not have to chain any geometry to create the toolpath because of the extra material we specified on the right face in the stock setup. You can also select points to dictate where Mastercam will create the facing operation.

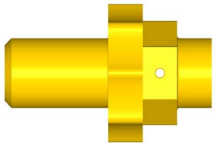
Toolpath Preview:



TURNING

- ♦ From the **General** group, select the **Face** icon as shown.





- ♦ Select the **OD Rough Right - 80 Degree Tool** and enter a comment as shown.

Lathe Face

Toolpath parameters Face parameters

Tool number: 1 Offset number: 1
Station number: 1 Tool Angle...

Feed rate: 0.01 ☒ in/rev ☐ in/min ☐ micro-in
☐ Finish feed rate: 0.005 ☐ in/rev ☐ in/min ☐ micro-in
Spindle speed: 200 ☒ CSS ☐ RPM
☐ Finish spindle speed: 1000 ☐ CSS ☒ RPM
Max. spindle speed: 10000 Coolant... (*)

Home Position
D:10. Z:10. From Machine Define

☐ Force tool change ☐ To batch

Comment:
Face the part.

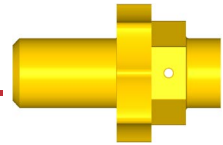
☒ Show library tools Right-click for options
Select library tool... ☒ Tool Filter...

Axis Combination / Spindle Origin
Left/Upper
Spindle origin: Lathe upper left Z0.


Misc values... ☒ Stock Update... ☐ Ref point...
☐ Tool Display... Canned Text...

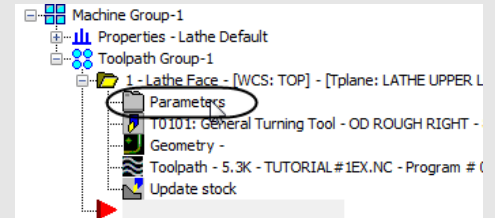
✓ ✗ ?


NOTE: The **Feed rate** and the **Spindle speeds** are based on the **Mastercam Tool Definitions**. They can be changed at any time based on the material that you are going to machine.



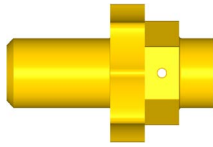
- ◆ Next you will have to set parameters in the **Face parameters** page.

NOTE: If accidentally you click on the **OK** button  before you set the parameters in all of the pages the toolpath has, from the **Toolpaths Manager**, click on the **Parameters** icon as shown.



Once you complete all of the pages and select the **OK** button , from the **Toolpaths Manager**, click on the **Regenerate all dirty operations** icon to ensure that all the changes you made are applied to the toolpath.





- ♦ Select the **Face parameters** tab and make all of the necessary changes as shown.

Entry Amount sets the height at which the tool rapids to or from the part.

Rough Stepover sets the roughing pass value.

Finish Stepover sets the finish pass value.

Overcut Amount determines how far past the center of the part the tool will cut.

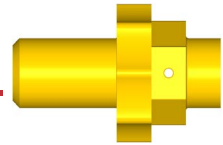
Retract Amount determines the distance the tool moves away from the face of the part before it moves to the start of the next cut.

Stock to Leave sets the remaining stock after the tool completes all passes.

Cut away from the center line sets the tool to start cutting closest to the center line and cut away from the center line at each pass.

- ♦ Once you have entered in all of the information, select the **OK** button to exit the **Lathe Face** dialog box.

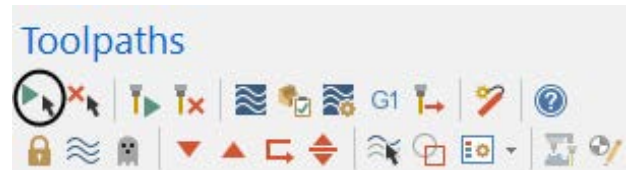




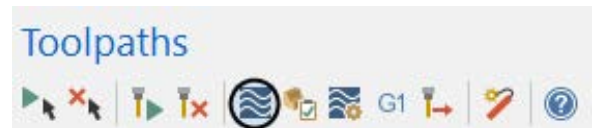
STEP 16: BACKPLOT THE TOOLPATH

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 collision between the workpiece and the tool.

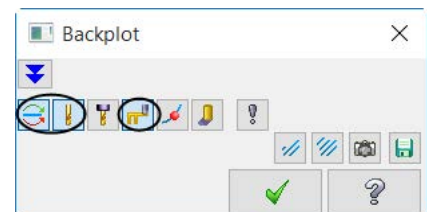
- ♦ Make sure that the toolpaths are selected (signified by the green check mark on the folder icon). If the operation is not selected, choose the **Select all operations** icon.



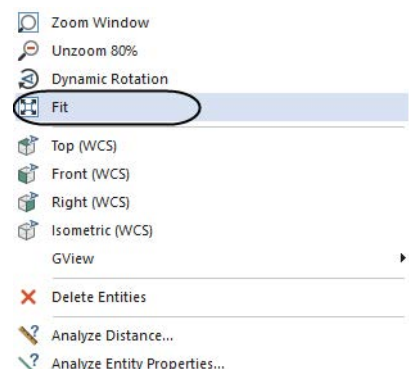
- ♦ Select the **Backplot selected operations** button.






- ♦ In the **Backplot** dialog box, enable **Display with color codes**, **Display tool**, **Display holder** and **Display rapid moves** icons as shown.



- ♦ To fit the workpiece onto the screen, if needed, right mouse click in the graphics window again and select **Fit** or press **Alt +F1**.



- ♦ You can step through the **Backplot** by using the **Step forward**  or **Step back**  buttons.
- ♦ You can adjust the speed of the backplot using the **Run speed slider** as shown. 

A

Absolute	92
Arc Endpoints	38, 43
Attributes	7
2D / 3D Construction	7
Clear Color	7
Color	7
Level	7
Line Style	7
Line Width	7
Point Style	7
Z Depth	7
Axis Combination	255
Axis Substitution	117, 275

B

Break Many Pieces	173
Breakthrough	98

C

C-Axis Contour	268
C-Axis Cross Drill	112, 119, 235, 240
C-Axis Face Contour	209, 248
C-Axis Face Drill	101, 257, 262
C-Axis Face Mill Type	83
Change The Construction Plane	32, 160
Change the Wireframe Color	7
Clearance Angles	67
Construction Depth	182
Create Point Position	24
Cross Contour toolpath with Y-Axis	223
Cut Away From The Center Line	57

D

Delete/Keep/Blank	175
Depth	92

E

Ellipse	162
Entry Amount	57
Exact Distance	175

F

Face Toolpath	54
Finish Stepover	57

G

GUI - Graphical User Interface	3
Backstage (FILE)	3
Graphic Area	3
QAT - Quick Access Toolbar	3

Quick Mask Buttons	3
Ribbon	3
Right Click Menu	3
Scale	3
Selection Bar	3
Tabs	3
Toolpaths/Solids/Planes Manager	3
WCS	3

I

Incremental	92
-------------------	----

L

Left Spindle	204
Leftover Material	76
Line Endpoints	4
Endpoint Coordinates	5
Endpoint, Length and Angle	5

M

Manager Panels	9
Auto Hide	10
Toolpaths Manager	9
Mastercam To Metric	
Current Session Only	11
Default	12
Multipasses	216

O

Overcut Amount	57
----------------------	----

P

Peck Drilling	109
Plunge Cutting	67
Plunge Parameters	67
Polygon	33

Q

Quiz Answers	285
--------------------	-----

R

Reorganizing The Toolbars	4
Retract Amount	57
Right Spindle	205
Rotary Axis Control	117
Rough Stepover	57
Rounded Distance	175

S

Set the Grid	13
Spline	195
Start Of Cut	67
Stock To Leave	57

Stock Transfer 244

T

Tip Comp 266

V

Verify

Color Loop 60

X

Xform Rotate 41, 177

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

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.



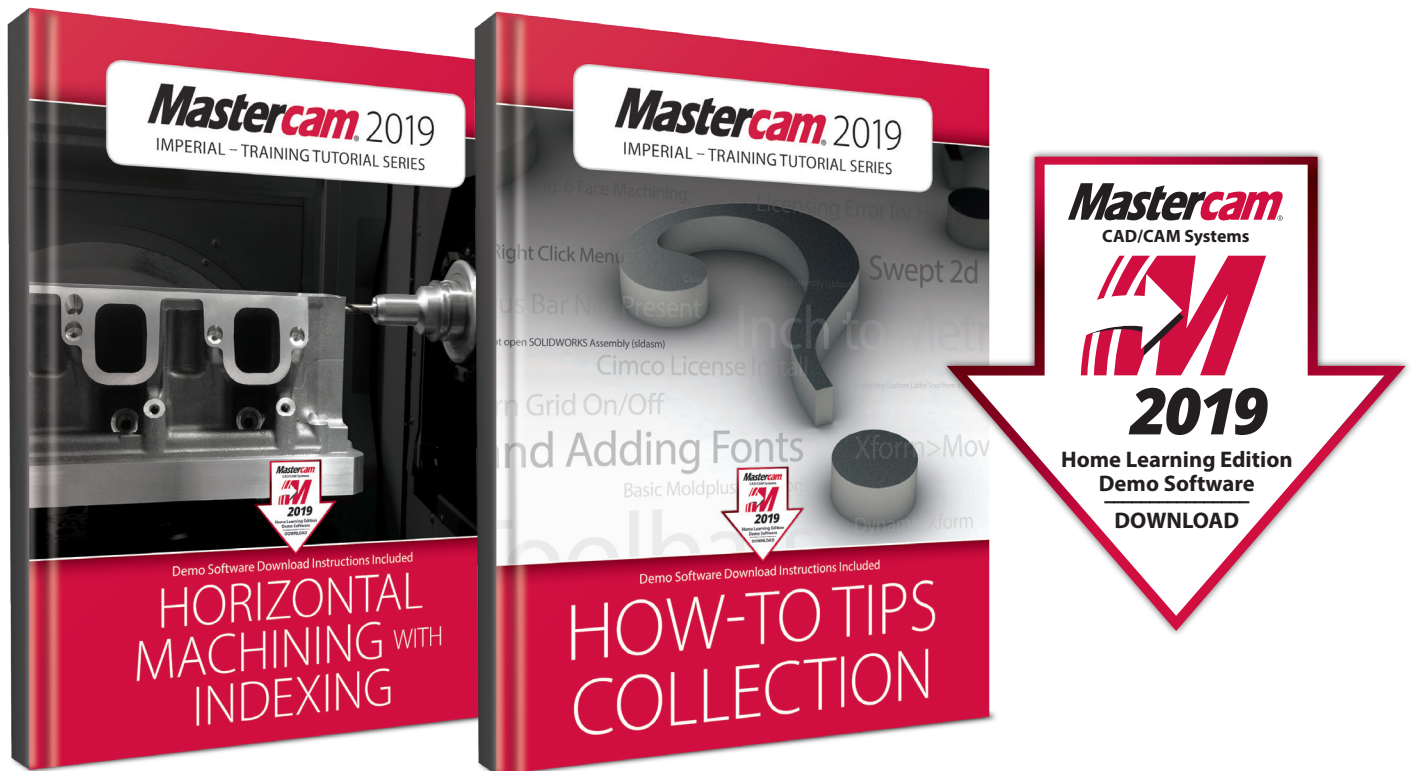
We speak your machine's language.

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



Mastercam 2019



Demo Software Download Instructions Included

FREE eBooks

Download today
www.eMastercam.com/freebooks

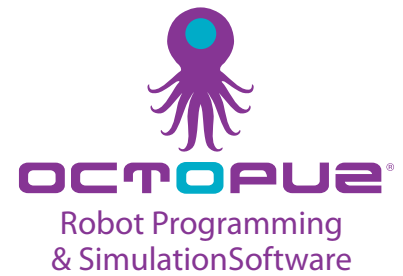
DOES YOUR SCHOOL HAVE ROBOTS? YOU SHOULD CHECK OUT OCTOPUZ.

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



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.



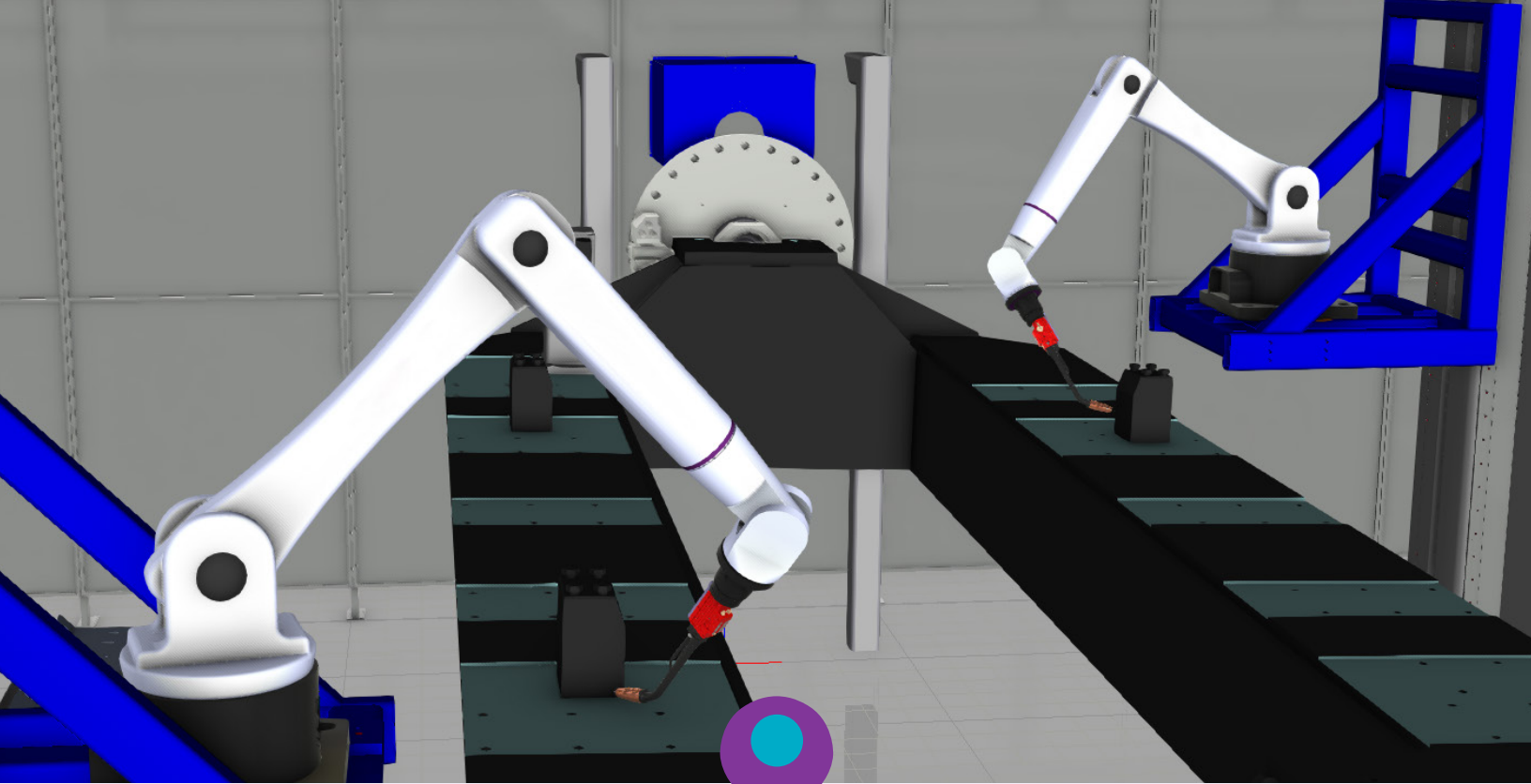
Protect your investment!
OCTOPUZ can detect
collisions offline.



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