

Mastercam 2019

METRIC – TRAINING TUTORIAL SERIES



Demo Software Download Instructions Included

LATHE

Mastercam® 2019

LATHE TRAINING METRIC TUTORIAL

Mastercam 2019 Lathe Metric Training Tutorial

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

Software: Mastercam 2019

Authors: Mariana Lendel

ISBN: 978-1-77146-758-2978-1-77146-777-3

Date: September 19, 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.

Table Of Content

Getting Started	13
Objectives	14
Step 1: Starting Mastercam	14
Step 2: GUI - Graphical User Interface	15
Step 3: Navigate Through Mastercam	16
Step 4: Set The Attributes	18
Step 5: Manager Panels	19
Step 6: Setting Mastercam To Metric	21
Step 7: Set The Grid	23
Conventions Used In This Book:	24
Step 8: Step Titles	24
Mastercam® Work Flow	25
Tutorial 1: Geometry Creation	27
Overview Of Steps Taken To Create The Part Geometry	28
Tutorial #1 Drawing	29
Step 1: Setting Up The Graphical User Interface	30
Step 2: Create A Rectangle	32
Step 3: Create The Parallel Lines	35
Step 4: Create Line Endpoint	43
Step 5: Create The FilletS	44
Step 6: Trim The Geometry	47
Step 7: Save The File	50
Tutorial #1 Review Exercise	51
Create The Geometry For Tutorial #1 Exercise	52
Tutorial #1 Geometry Creation Quiz	54
Tutorial 1: Toolpath Creation	55

Overview Of Steps Taken To Create The Final Part:	56
Part Setup:	57
Setup Sheet:	57
Step 1: Select The Machine And Set Up The Stock	58
Step 2: Face The Part	64
Step 3: Backplot The Toolpaths	66
Step 4: Simulate The Toolpath In Verify	68
Step 5: Rough Out The Part	70
Step 6: Finish The Part	77
Step 7: Post The File	81
Step 8: Save The Updated MCAM File	82
Create The Toolpaths For Tutorial #1 Exercise	83
Tutorial #1 Toolpath Creation Quiz	86
Tutorial 2: Geometry Creation	87
Overview Of Steps Taken To Create The Part Geometry	88
Tutorial #2 Drawing	89
Step 1: Setting Up The Graphical User Interface	90
Step 2: Select The Lathe Default	90
Step 3: Set Up The Lathe Plane +D +Z	90
Step 4: Create A Rectangle	92
Step 5: Create Parallel Lines	94
Step 6: Create The Fillets	99
Step 7: Trim The Geometry	102
Step 8: Create Additional Lines	104
Step 9: Trim The Geometry	109
Step 10: Save The File	112
Tutorial #2 Review Exercise	113
Tutorial #2 Geometry Creation Quiz	116

Tutorial 2: Toolpath Creation	117
Overview Of Steps Taken To Create The Final Part:	118
Part Setup:	119
Setup Sheet:	119
Step 1: Select The Tool Settings And Set Up The Stock	120
Step 2: Face The Part	125
Step 3: Backplot The Toolpath	127
Step 4: Simulate The Toolpath In Verify	129
Step 5: Rough The Part	131
Step 6: Finish The Part	136
Step 7: Groove The Part Using The Multiple ChainS Method	140
Step 8: Center Drill The Part	148
Step 9: Drill The Part	151
Step 10: Run The Post Processor To Obtain The G-code File	154
Step 11: Save The Updated MCAM File	155
Create The Toolpaths For Tutorial #2 Exercise	157
Tutorial #2 Toolpath Creation Quiz	160
Tutorial 3: Geometry Creation	161
Overview Of Steps Taken To Create The Part Geometry	162
Step 1: Setting Up The Graphical User Interface	163
Step 2: Download The File Lathe Training Tutorial Metric	163
Step 3: Open The Parasolid File In Mastercam	163
Step 4: Change The Main Level To 2	165
Step 5: Create A Turn Profile Of The Part	166
Step 6: Save The File	169
Tutorial #3 Review Exercise	170
Tutorial #3 Geometry Creation Quiz	172

Tutorial 3: Toolpath Creation	173
Overview Of Steps Taken To Create The Final Part:	174
Part Setup:	175
Setup Sheet:	175
Step 1: Select The Lathe Default	176
Step 2: Set Up The Lathe Plane +D +Z	176
Step 3: Set Up The Tool Settings And The Stock	177
Step 4: Face The Part	182
Step 5: Backplot The Toolpaths	184
Step 6: Simulate The Toolpath In Verify	186
Step 7: Center Drill The Part	187
Step 8: Drill The Part	190
Step 9: Rough The ID Using A Canned Rough Toolpath	193
Step 10: Finish The ID Using A Canned Finish Toolpath	200
Step 11: Rough The OD Using A Rough Toolpath	203
Step 12: Finish The OD	211
Step 13: Machine The Groove	216
Step 14: Machine The Angled Groove	227
Step 15: Cutoff The Part	236
Step 16: Run The Post Processor To Obtain The G-code File	239
Step 17: Save The Updated MCAM File	241
Create The Toolpaths For Tutorial #3 Exercise	242
Tutorial #3 Toolpath Creation Quiz	246
Tutorial 4: Geometry Creation	247
Overview Of Steps Taken To Create The Part Geometry	248
Step 1: Setting Up The Graphical User Interface	250
Step 2: Create A Rectangle To Represent The Outside Profile	250
Step 3: Create Parallel Lines	252

Step 4: Create A Line Given The Endpoints	258
Step 5: Delete The Construction Line	259
Step 6: Trim The Geometry	260
Step 7: Save The File	264
Tutorial #4 Review Exercise	265
Tutorial #4 Geometry Creation Quiz	268
Tutorial 4: Toolpath Creation	269
Overview Of Steps Taken To Create The Final Part:	270
Part Setup:	271
Setup Sheet:	271
Step 1: Select The Tool Settings And Set Up The Stock	272
Step 2: Face The Part	277
Step 3: Rough The OD	277
Step 4: Backplot The Toolpaths	280
Step 5: Simulate The Toolpath In Verify	282
Step 6: Finish The OD Using A Finish Toolpath	283
Step 7: Center Drill The Part	287
Step 8: Drill The Part	290
Step 9: Rough The Inside Diameter	293
Step 10: Finish The ID Before Grooving	298
Step 11: Finish The Grooves On The ID	302
Step 12: Cutoff The Part	309
Step 13: Run The Post Processor To Obtain The G-code File	313
Step 14: Save The Updated MCAM File	314
Create The Toolpaths For Tutorial #4 Exercise	315
Tutorial #4 Toolpath Creation Quiz	320
Tutorial 5: Geometry Creation	321

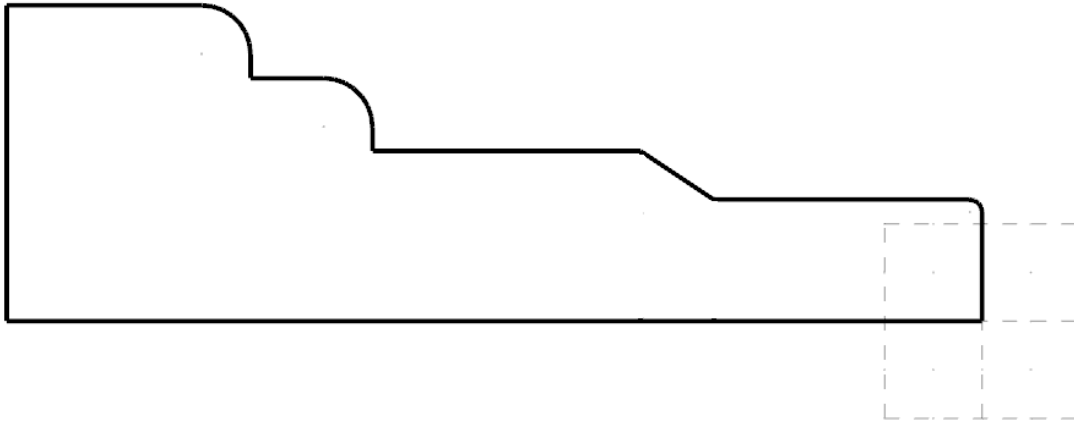
Overview Of Steps Taken To Create The Part Geometry	322
Step 1: Setting Up The Graphical User Interface	324
Step 2: Select The Lathe Default	324
Step 3: Set Up The Lathe Plane +D +Z	324
Step 4: Create Lines To Represent The Outside Profile	325
Step 5: Create An Arc Tangent With Dynamic Tangency	337
Step 6: Delete The Extra Construction Line	340
Step 7: Create Relief Grooves	341
Step 8: Chamfer The Part	345
Step 9: Save The File	347
Tutorial #5 Review Exercise	348
Tutorial #5 Geometry Creation Quiz	351
Tutorial 5: Toolpath Creation	353
Overview Of Steps Taken To Create The Final Part:	354
Part Setup:	355
Setup Sheet:	355
Step 1: Select The Tool Settings And The Stock	356
Step 2: Face The Part	361
Step 3: Rough The OD	361
Step 4: Finish The OD	366
Step 5: Machine The M12 X 1 Relief Groove	370
Step 6: Machine The M12 X 1 Thread	376
Step 7: Center Drill The Part	381
Step 8: Drill The Part	384
Step 9: Flip The Part Using Stock Flip	389
Step 10: Face The Flipped Part	393
Step 11: Rough The OD Of The Flipped Part	393
Step 12: Finish The OD Of The Flipped Part	397

Step 13: Machine The Grooves Of The Flipped Part	397
Step 14: Machine The M16 X 1.5 Thread	403
Step 15: Center Drill The Flipped Part	407
Step 16: Drill The Flipped Part	410
Step 17: Run The Post Processor To Obtain The G-code File	413
Step 18: Save The Updated MCAM File	414
Create The Toolpaths For Tutorial #5 Exercise	415
Tutorial #5 Toolpath Creation Quiz	420
Tutorial 6: Geometry Creation	421
Overview Of Steps Taken To Create The Part Geometry	422
Step 1: Setting Up The Graphical User Interface	424
Step 2: Create A Rectangle To Represent The Outside Profile	424
Step 3: Create Parallel Lines	426
Step 4: Create Angled Lines Given Their Endpoints	432
Step 5: Delete The Extra Construction Lines	434
Step 6: Fillet The Part	435
Step 7: Trim The Groove	438
Step 8: Chamfer The Grooves	439
Step 9: Create Parallel Lines	441
Step 10: Chamfer The Corners And Trim Simultaneously	449
Step 11: Trim The Geometry	451
Step 12: Create Relief Grooves	454
Step 13: Save The File	459
Tutorial #6 Review Exercise	460
Tutorial #6 Geometry Creation Quiz	463
Tutorial 6: Toolpath Creation	465
Overview Of Steps Taken To Create The Final Part:	466

Toolpath Creation	467
Part Setup 1:	467
Step 1: Select The Tool Settings And The Stock	469
Step 2: Face THE PART	474
Step 3: Rough The OD	474
Step 4: Backplot And Verify The Toolpaths	478
Step 5: Finish The OD With Down Cutting	480
Step 6: Machine The M12 X 1.25 And The M12 X 1 Relief Grooves	484
Step 7: Machine The M12 X 1 Thread	490
Step 8: Machine The M24 X 2 Thread	494
Step 9: Center Drill The Part	498
Step 10: Advance The Stock	501
Step 11: Advance The Tailstock	503
Step 12: Machine The Grooves	504
Step 13: Cutoff The Part	514
Step 14: Run The Post Processor To Obtain The G-code File	514
Step 15: Save The Updated MCAM File	515
Create The Toolpaths For Tutorial #6 Exercise	516
Tutorial #6 Toolpath Creation Quiz	522
Tutorial 7: Geometry Creation	523
Overview Of Steps Taken To Create The Part Geometry	524
Step 1: Setting Up The Graphical User Interface	525
Step 2: Download The File Lathe Training Tutorial Metric	525
Step 3: Removing Any Machine From The Toolpaths Manager	525
Step 4: Open The Solidworks File In Mastercam	527
Step 5: Change The Main Level To 2	529
Step 6: Create A Turn Profile Of The Part	530
Step 7: Save The File	533

Tutorial #7 Review Exercise	534
Tutorial #7 Geometry Creation Quiz	536
Tutorial 7: Toolpath Creation	537
Overview Of Steps Taken To Create The Final Part:	538
Toolpath Creation	539
Part Setup:	539
Setup Sheet 1:	540
Step 1: Select The Tool Settings And The Stock	541
Step 2: Face The Part	550
Step 3: Backplot The Toolpaths	551
Step 4: Rough The OD	554
Step 5: Finish The OD	558
Step 6: Spot Drill The Center Of The Part	562
Step 7: Drill The Part With A Large Drill	566
Step 8: Rough And Finish The Inside Diameter Of The Part	571
Step 9: Machine The Grooves On The ID Of The Part	577
Step 10: Thread The OD Of The Part	586
Step 11: Run The Post Processor To Obtain The G-Code File	591
Step 12: Save The Updated MCAM File	592
Create The Toolpaths For Tutorial #7 Exercise	593
Tutorial #7 Toolpath Creation Quiz	598
Creating A Lathe Tool Library	599
Editing A Lathe Tool Library	613
Quiz Answers	621

Tutorial 1: Geometry Creation



OVERVIEW OF STEPS TAKEN TO CREATE THE PART GEOMETRY

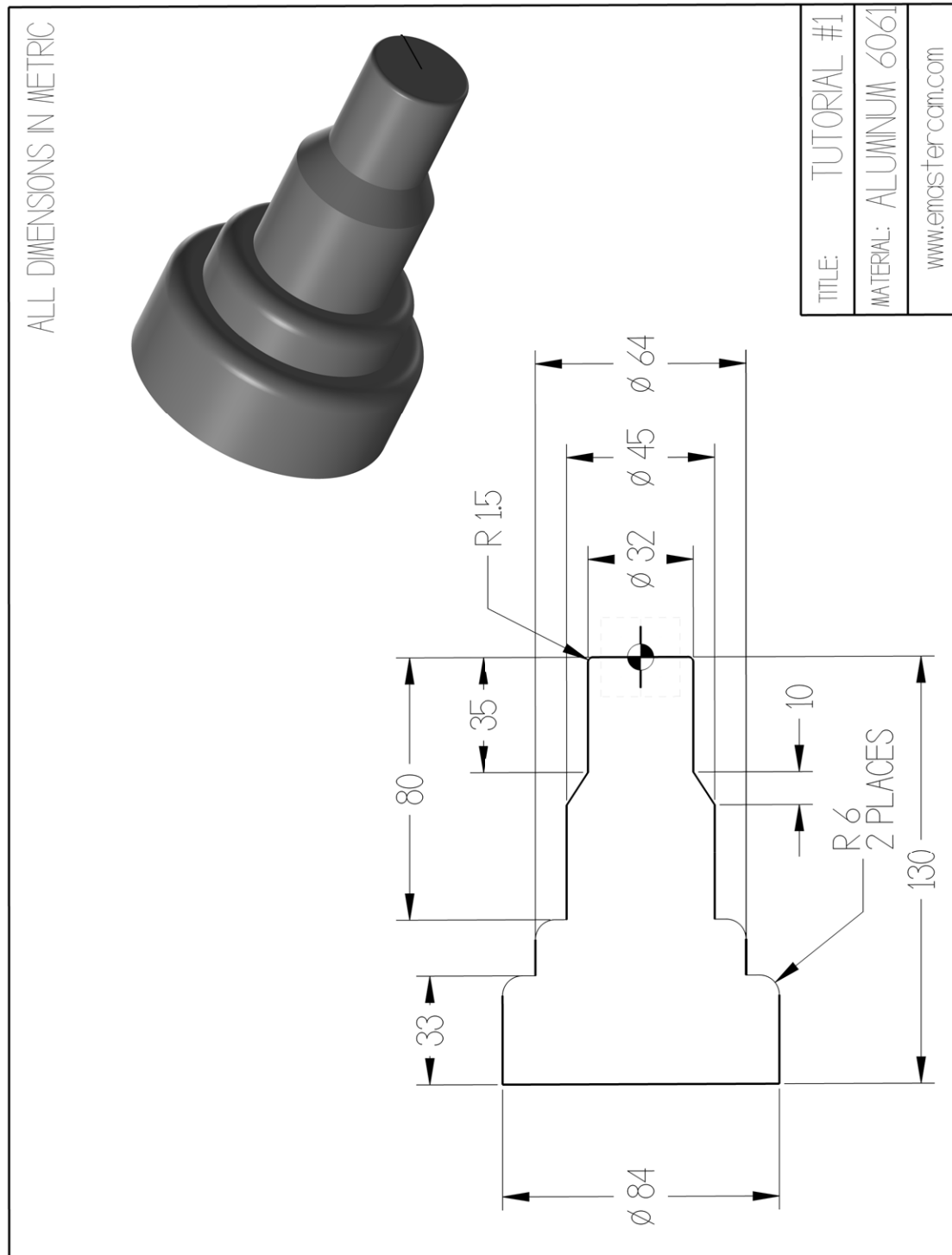
From Drawing to CAD Model:

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

Create the 2D CAD Model:

- ◆ The student will create the upper profile of the part. Only half of the geometry is needed to create the necessary toolpaths to machine the part.
- ◆ Geometry creation commands such as Line Endpoints, Line Parallel, Rectangle, Fillet Entities, and Trim will be used.

TUTORIAL #1 DRAWING



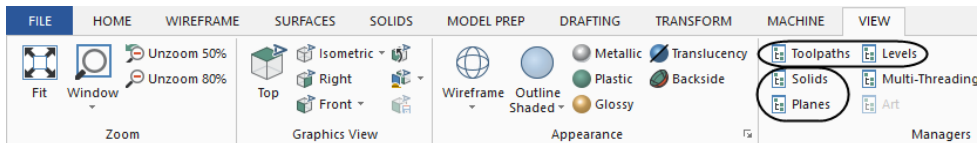
STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

Please refer to the **Getting Started** section for more info on how to set up the graphical user interface. In this step, you will learn how to hide the manager panels to gain more space in the graphics window.

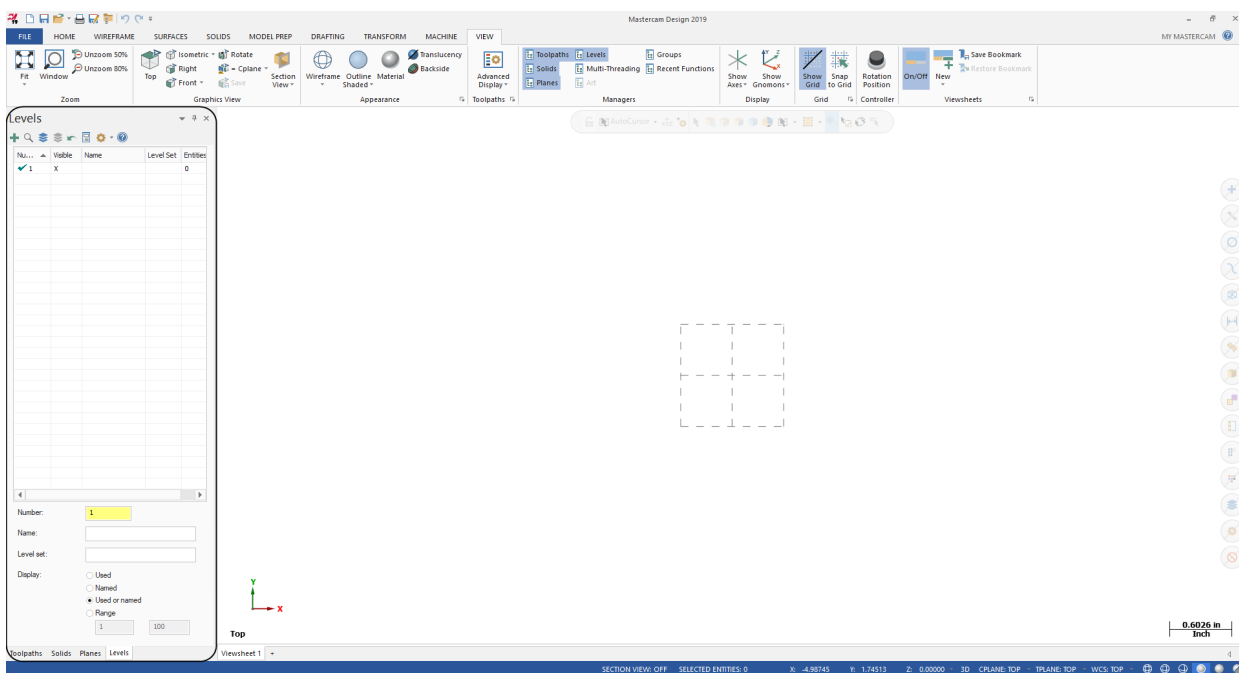
1.1 Hide the manager panels

VIEW

- ◆ From the **Managers** group, enable all four managers as shown.

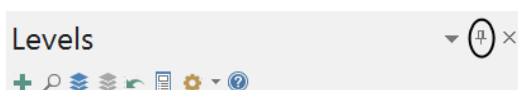


- ◆ The panels should be on the left side of the graphics window as shown.

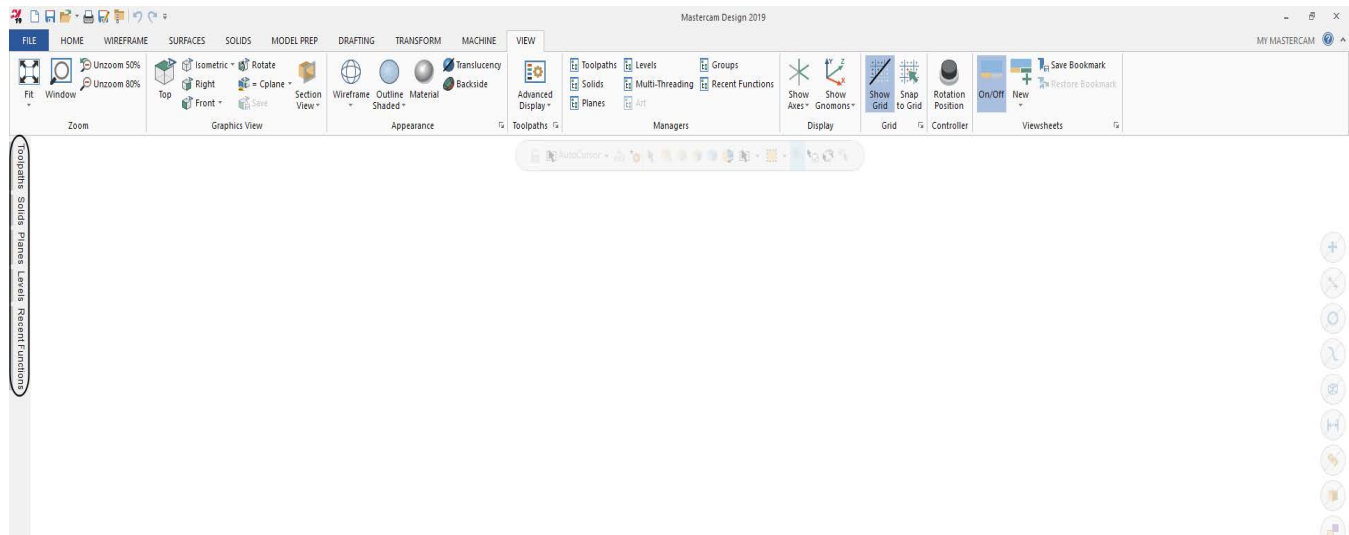


*Note: It does not matter which panel is currently opened. It could be the **Toolpaths**, the **Solids**, the **Planes** or the **Levels** panel as shown.*

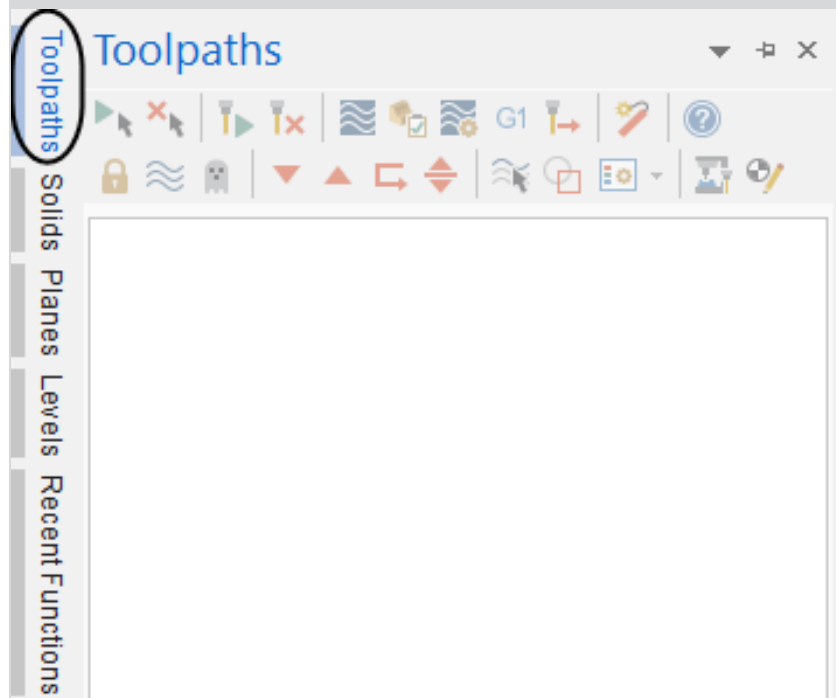
- ◆ To hide all panels, click on the **Auto Hide** icon as shown.



- ◆ The panels will be hidden to the left of the graphics window as shown.



*Note: To un-hide them temporarily, you can click on one of the **Managers** to open it as shown.*

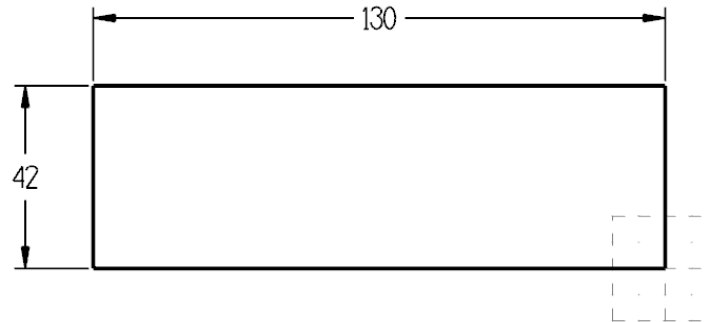


*While creating the geometry, keep the **Manager** panels hidden. This ensures more space in the graphics window for the geometry.*

STEP 2: CREATE A RECTANGLE

In this step you will learn how to create a rectangle given the width, the height, and the anchor position.

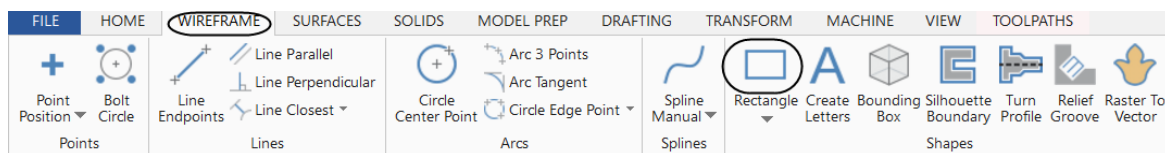
Step Preview:



2.1 Create the 130mm by 42mm rectangle

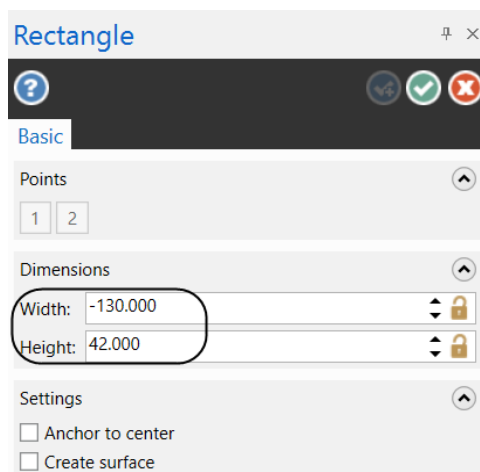
WIREFRAME

- ◆ From the **Shapes** group, select Rectangle.



Note: Select the rectangle icon as shown. If you click too close to the drop down arrow, a fly-out list of commands appears and you can select the top Rectangle command.

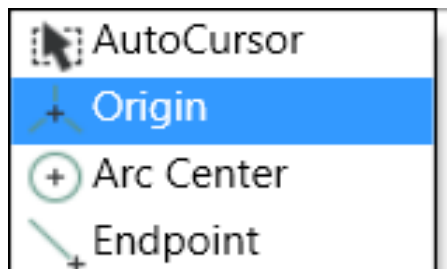
- ◆ Enter the **Width** and the **Height** and press Enter.



- ◆ To select the position of the base point, from the **General Selection** toolbar, click on the drop down arrow next to **AutoCursor** as shown.



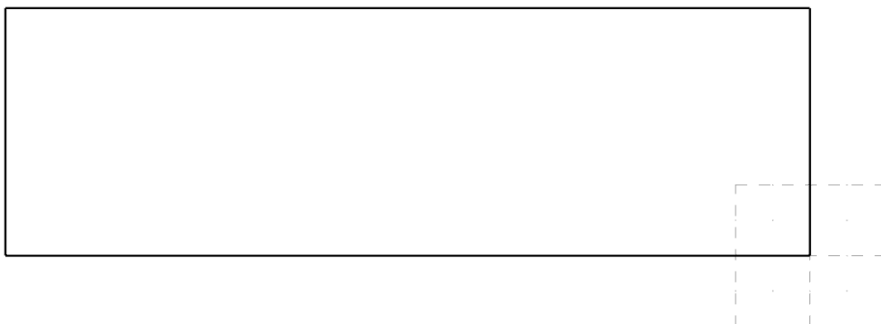
- ◆ From the fly-out menu select Origin.



- ◆ To see the entire rectangle, right mouse click in the graphics window and select **Fit** as shown.

*Note: To fit the geometry to the screen you can also press **Alt + F1**.*

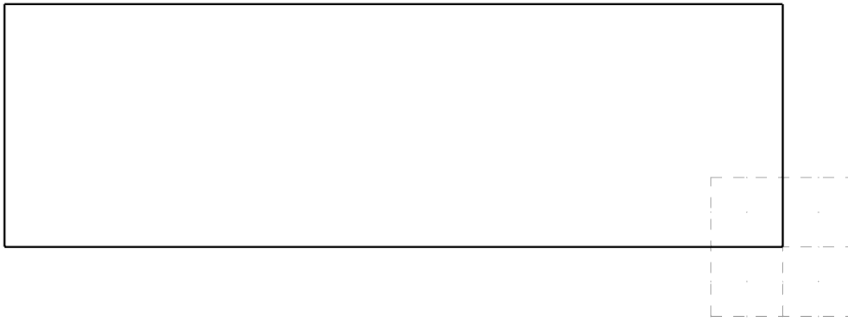
- ◆ A preview of the geometry should look as shown.





Note: The geometry should appear in a cyan blue color which is the color for live entities.

While the rectangle is live you can adjust the dimensions or select a new base point.

- ◆ Select the **OK** button to exit the **Rectangle** command.
- ◆ The geometry should look as shown.



*Note: While creating geometry for this tutorial, if you make a mistake, you can undo the last step using the **Undo** icon  or by pressing **Ctrl + Z**. You can undo as many steps as needed. If you delete or undo a step by mistake, just use the **Redo** icon  or press **Ctrl + Y**.*

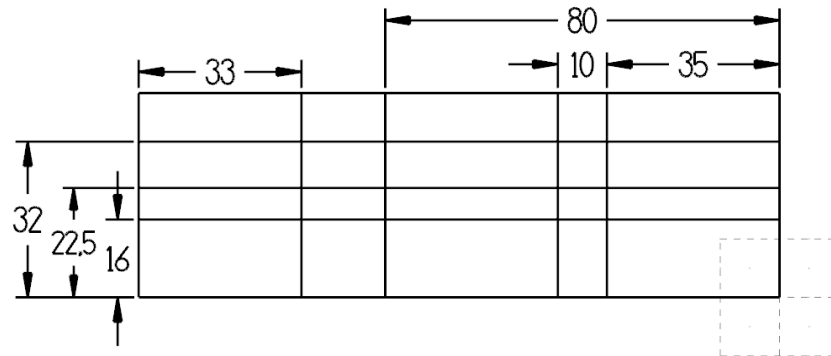
*To delete unwanted geometry, select the geometry first and then press **Delete** from the keyboard.*

To zoom tor unzoom, move the cursor to the center of the geometry and scroll up or down on the mouse wheel.

STEP 3: CREATE THE PARALLEL LINES

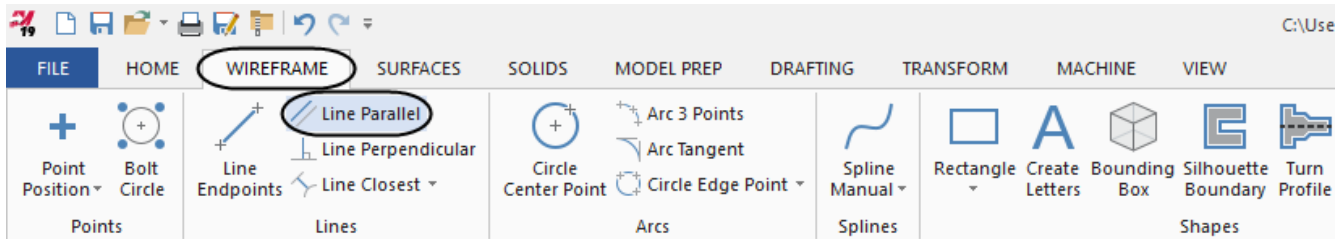
In this step you will learn how to create parallel lines to existing lines given the distance between the lines. We are creating the lines to use as part of the geometry as well as the construction lines.

Step Preview:

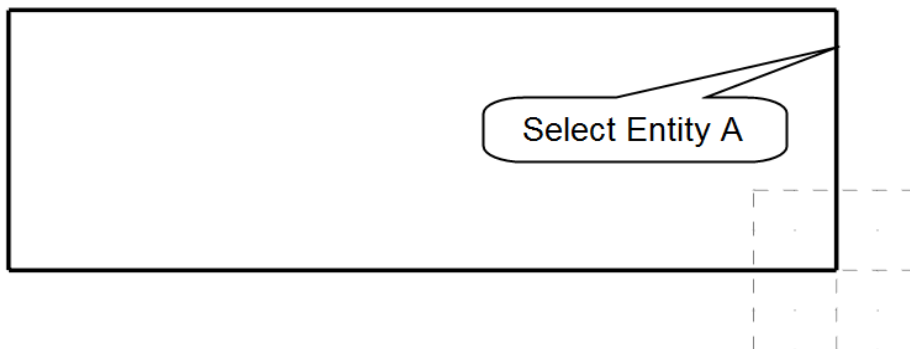


WIREFRAME

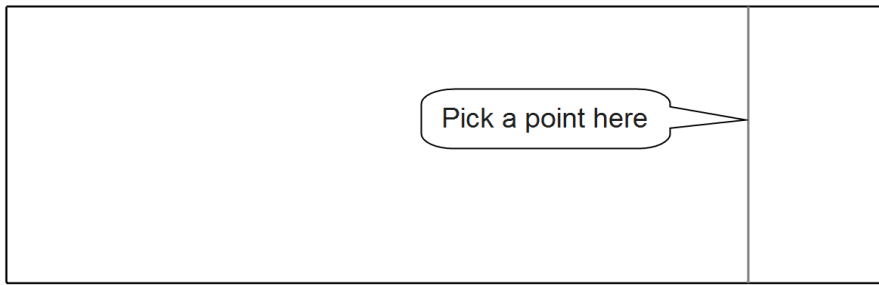
- ◆ From the **Lines** group, select **Line Parallel**.



- ◆ [Select a line]: Select **Entity A** as shown.

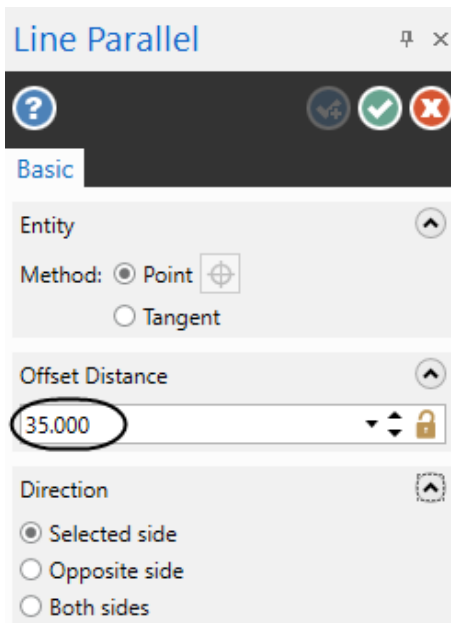


- ◆ [Select the point to place a parallel line through]: Pick a point to the left of the selected line.

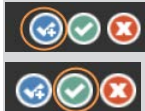


Note: The color of the geometry is cyan which means that the entity is "live" and you can still change the line parameters if needed.

- ◆ In the **Line Parallel** panel, enter the **Distance 35.0**.
- ◆ Press **Enter** to move the line to the proper distance.



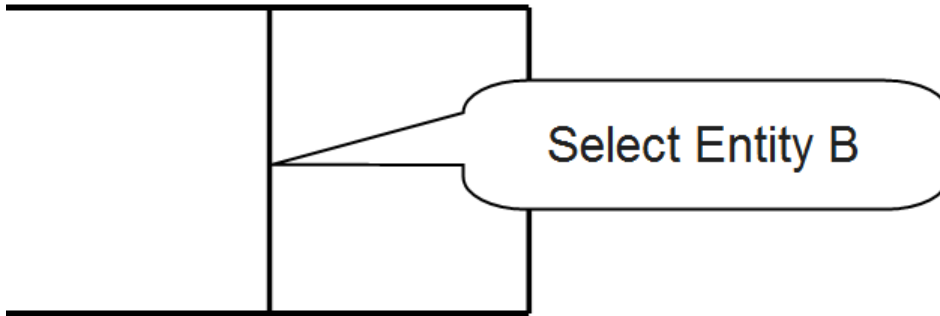
*Note: To continue using the same command you can either select the **OK and Create New Operation** button*



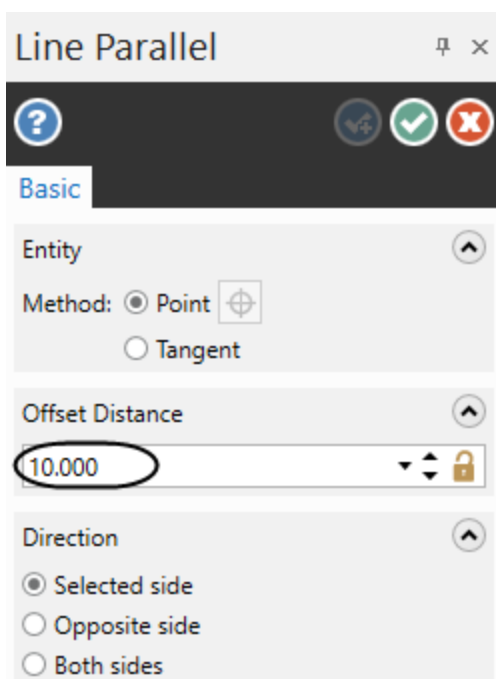
*or press **Enter**. To exit the command you can either start a new command or select the **OK** button.*

- ◆ Press **Enter** to continue.

- ◆ [Select a line]: Select **Entity B** as shown.



- ◆ [Select the point to place a parallel line through]: Pick a point to the left of the selected line.
- ◆ Enter the **Distance 10.0**.



- ◆ Press **Enter** to move the line to the proper distance.
- ◆ Press **Enter** to continue.

Tutorial 1: Toolpath Creation



OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

Create the necessary Toolpaths to machine the part:

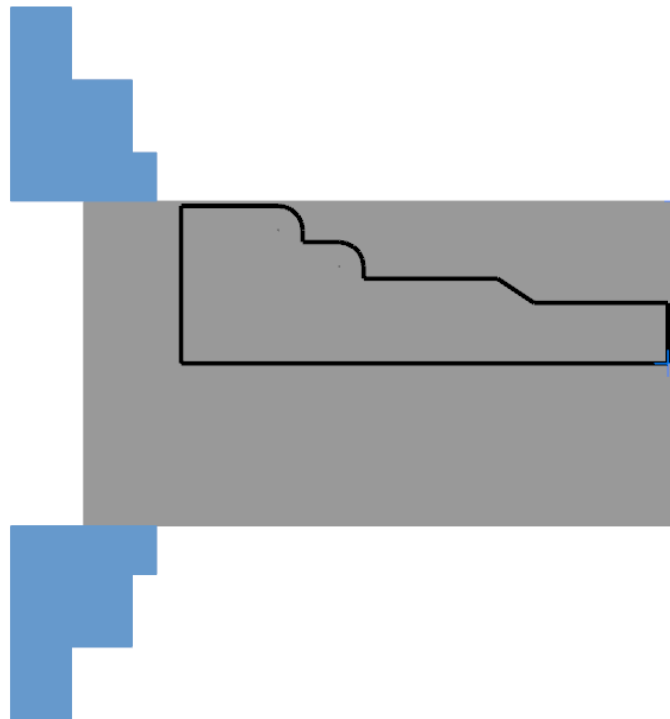
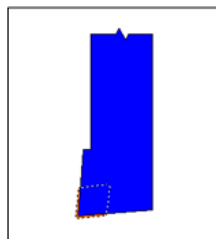
- ◆ The student will set up the stock size to be used and the clamping method used.
- ◆ A Facing toolpath will be created to remove material from the face of the part.
- ◆ A Roughing toolpath will be created to remove the bulk of material in preparation for a finish toolpath.
- ◆ A Finish toolpath will be created to machine the leftover material from the roughing operation.

Backplot and Verify the file:

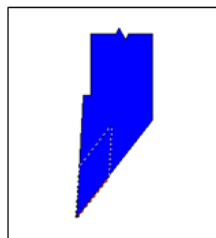
- ◆ Backplot will be used to simulate a step-by-step process of the tool's movements.
- ◆ Verify will be used to watch a tool machine the part out of a solid model.

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.

PART SETUP:**SETUP SHEET:**

TYPE:	General Turning Tool	FLUTE LENGTH:
DIA OFFSET:		OVERALL LENGTH:
HOLDER:		CORNER RAD:
NUMBER:	1	# OF FLUTES:
LENGTH OFFSET:		
T0101: General Turning Tool - OD ROUGH RIGHT - 80 DEG.		



TYPE:	General Turning Tool	FLUTE LENGTH:
DIA OFFSET:		OVERALL LENGTH:
HOLDER:		CORNER RAD:
NUMBER:	21	# OF FLUTES:
LENGTH OFFSET:		
T2121: General Turning Tool - OD FINISH RIGHT - 35 DEG.		

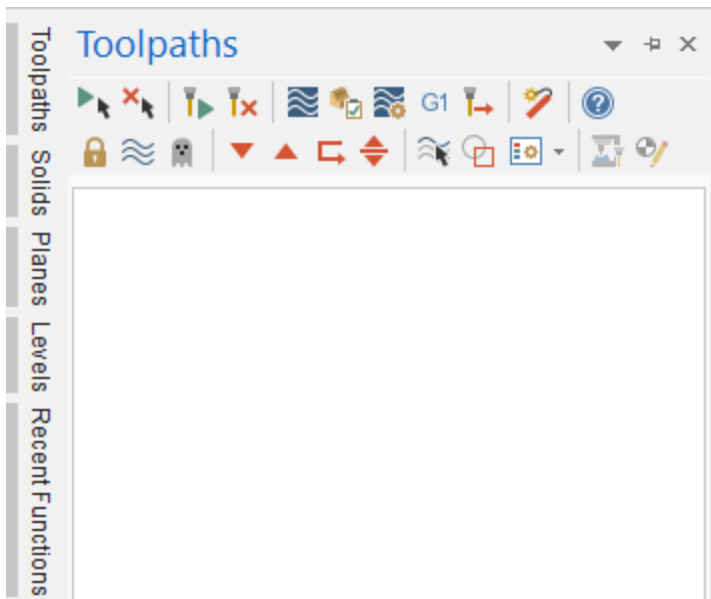
STEP 1: SELECT THE MACHINE AND SET UP THE STOCK

In Mastercam, you select a **Machine Definition** before creating any toolpath. The **Machine Definition** is a model of your machine's capabilities and features. It acts like a template for setting up your machine. The machine definition ties together three main components: the schematic model of your machine's components, the control definition that models your control capabilities, and the post processor that will generate the required machine code (G-code). For a Mill Essentials exercise (2D toolpaths), we need just a basic machine definition.

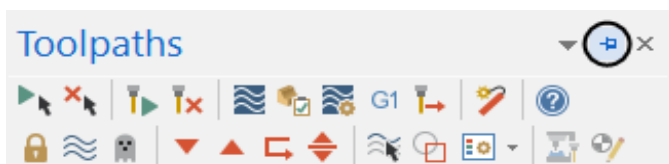
*Note: For the purpose of this tutorial, we will be using the **Default Mill MM** machine.*

1.1 Unhide the Toolpaths Manager panel

- ◆ From the left side of the graphics window, click on the **Toolpaths** tab as shown.



- ◆ Pin the **Toolpaths Manager** by clicking on the **Auto Hide** icon as shown.



- ◆ The **manager panels** will be translated to the **lower left corner** of the graphics window.

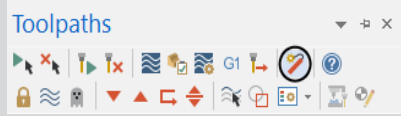


*Note: If a **Machine Group** already exists in the Toolpaths Manager, skip the next step.*

1.2 Select the machine

*Note: If another machine is already selected, from the **Machine** ribbon in the Machine Type group select **Design**. Then in the **Toolpaths Manager**, click on the **Delete all operations, groups and tools** icon to remove*

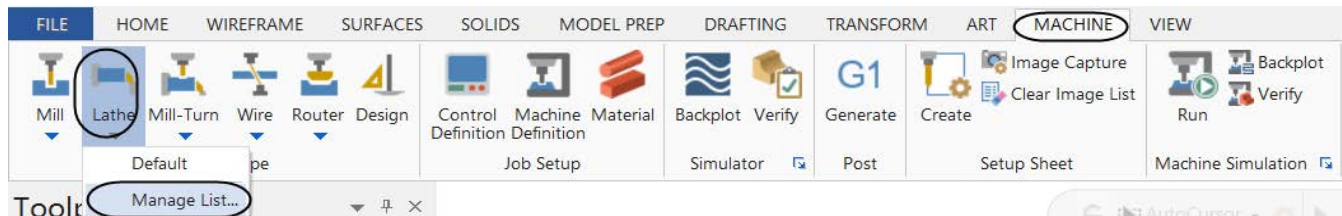
*the existing **Machine Group**.*



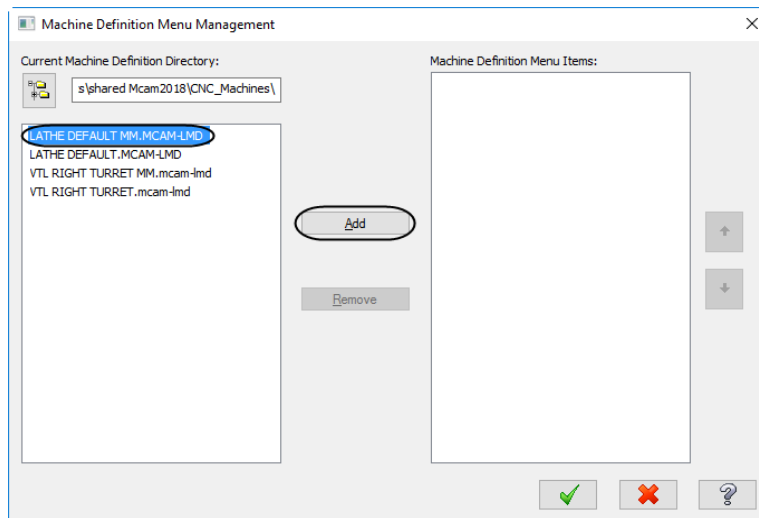
- ◆ Press **Alt + F1** to fit the geometry to the screen.

MACHINE

- ◆ From the **Machine** area, click on the drop down arrow below **Lathe** and select **Manage List**.

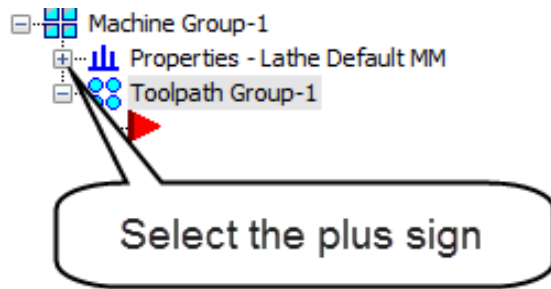


- ◆ Select **LATHE DEFAULT MM.MCAM-LMD** from the list and press **Add**.

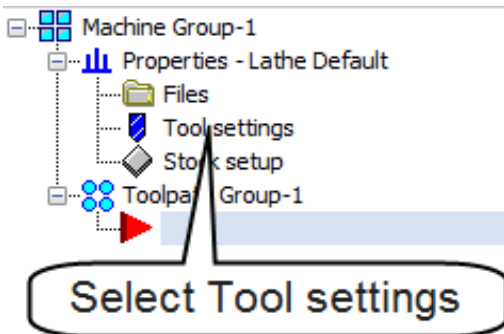


*Note: Once you select the **Lathe Default** the ribbon bar changes to reflect the toolpaths that could be used with **Lathe** machine.*

- ◆ 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.



- ◆ Change the parameters to match the screen shot in [Figure: 1.2.1](#).

Figure: 1.2.1

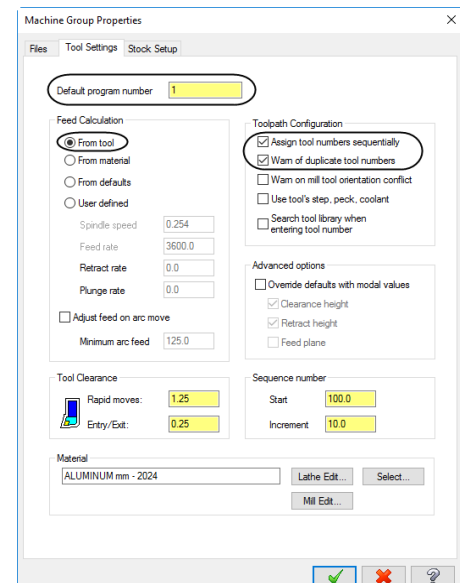
Default program number is used to enter a number if your machine requires a number for a program name.

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.

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



- ◆ Select the **Stock Setup** tab to define the stock.



- ◆ Choose the **Properties** button to set up the stock for the **Left Spindle**.
- ◆ Define the stock by setting the stock geometry (Cylinder) and entering the stock dimensions.
- ◆ Ensure you enable **Use Margins** and enter the values as shown in [Figure: 1.2.2](#).

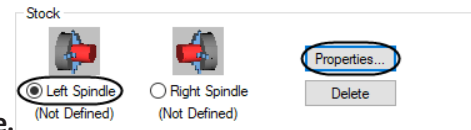
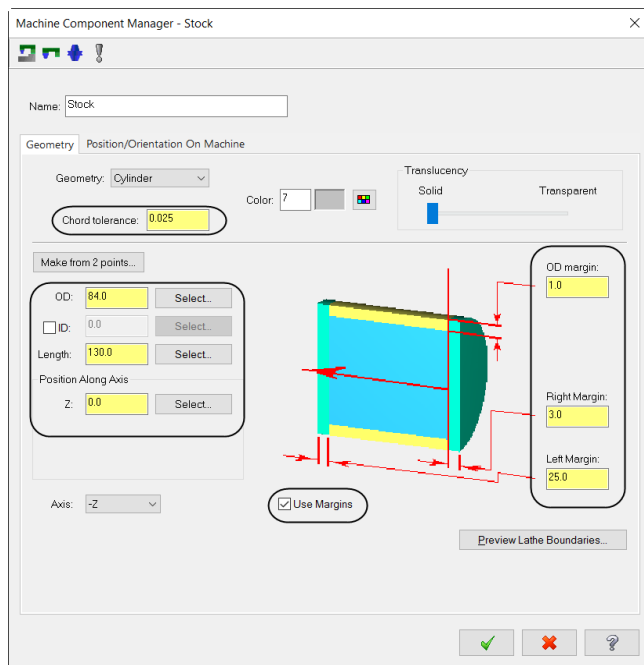



Figure: 1.2.2

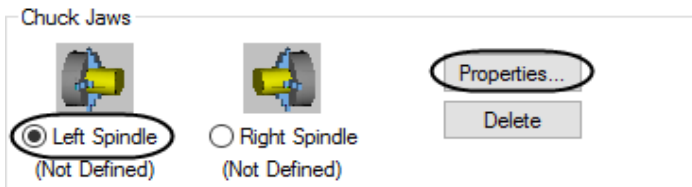


OD is used to enter the outer diameter of the final part. You can also click on the Select button and pick a point from the geometry. **Length** is used to enter the length of the finished part. Again you can click on the Select button and pick a point from the geometry.

Use Margins allows you to add extra stock to the final part size. **OD margin** allows you to enter a value as a radius value that will be added to the final part to define the stock's outer diameter. **Left margin** allows you to add a value to the stock's left side. **Right margin** allows you to add a value to the stock's right side.

*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 **Machine Component Manager - Stock** dialog box. 
- ◆ 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.



Position parameters determine where the reference point is in relation to the stock.

From stock positions the chuck jaw on the stock using the selected clamping method's reference point. You can choose how much of the stock the jaw grips.

- ◆ Select the **OK** button to exit the **Machine Component Manager - Chuck Jaws** dialog box. 


- ◆ Set the **Display Options** as shown.

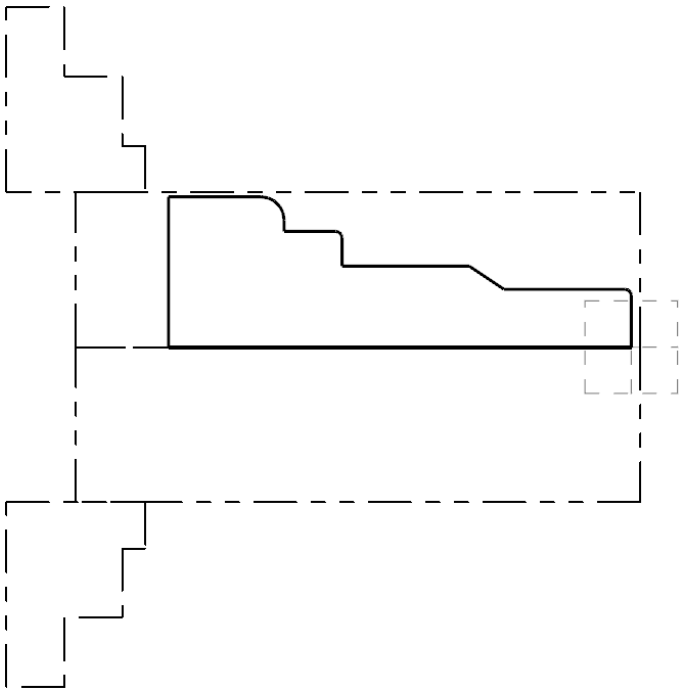
Display Options

<input checked="" type="checkbox"/> Left stock	<input type="checkbox"/> Right stock	All
<input checked="" type="checkbox"/> Left chuck	<input type="checkbox"/> Right chuck	
<input type="checkbox"/> Tailstock	<input type="checkbox"/> Steady rest	None

☐ Shade boundaries

☒ Fit screen to boundaries

- ◆ Select the **OK** button to exit the **Machine Group Properties** dialog box. 
- ◆ Press **Alt + F1** to fit the drawing to the screen.
- ◆ The stock should look as show.

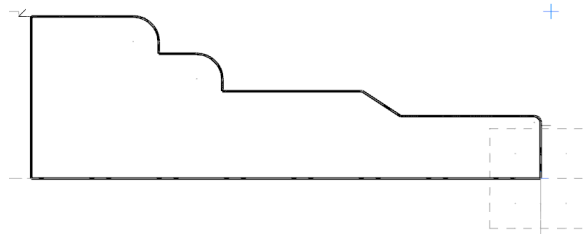


Note: The stock is not geometry and cannot be selected.

STEP 2: FACE THE PART

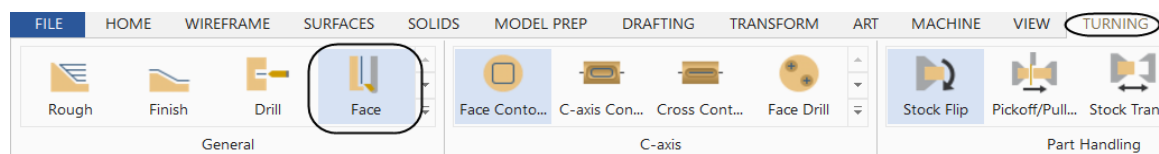
Face toolpaths allow the user to quickly clean the stock from one end of the part and create an even surface for future operations. Note that we do not have to chain any geometry to create the toolpath because of the extra material we specified on the right face in the stock setup. You can also select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

Toolpath Preview:

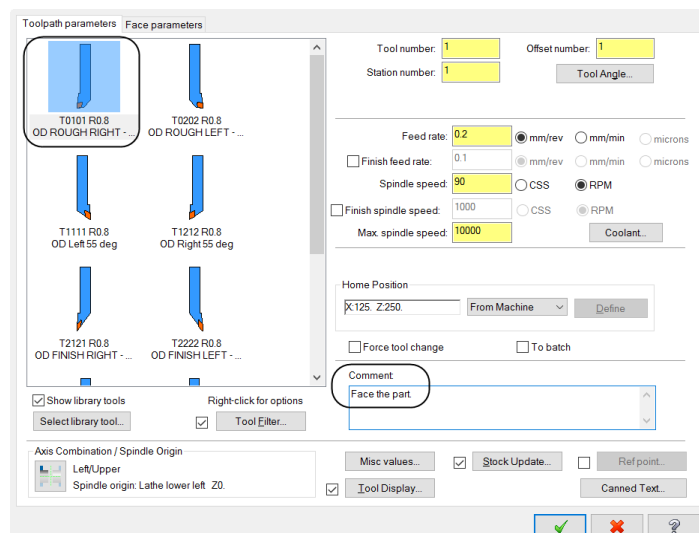


TURNING

- ◆ From the **General** group, select the **Face** icon.




- ◆ Select the **OD Rough Right - 80 Degree** tool and enter the comment as shown.

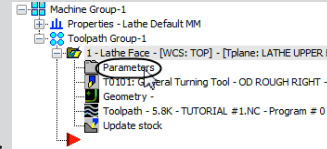



*Note: The **Feed rate** and the **Spindle speeds** are based on the **Mastercam Tool Definitions**. They can be changed at any time based on the material that you are going to machine.*

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

*Note: If you accidentally click on the **OK** button  before you set the parameters in all of the pages the*

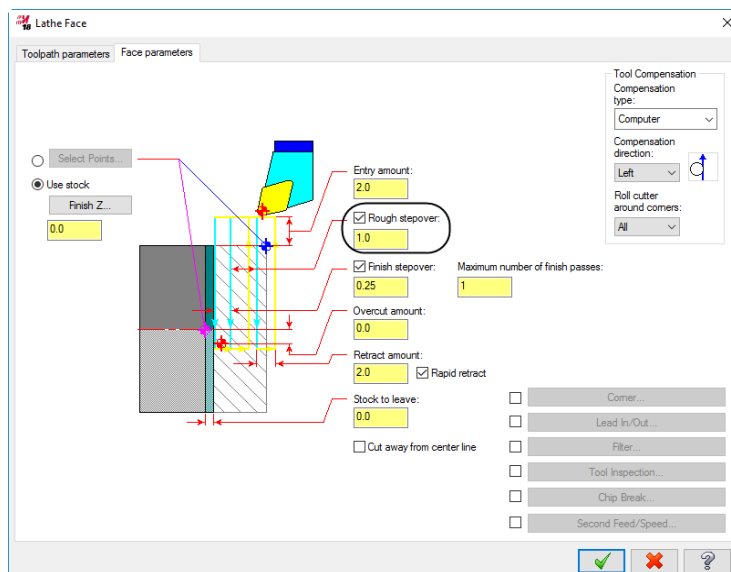
*toolpath has, from the **Toolpaths Manager**, click on **Parameters** as shown.*



*Once you complete all 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 were 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.

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

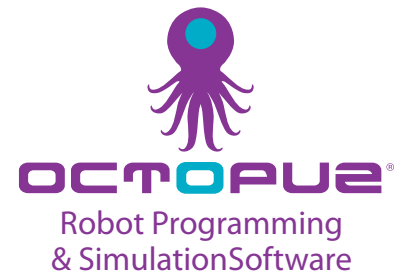
DOES YOUR SCHOOL HAVE ROBOTS? YOU SHOULD CHECK OUT OCTOPUZ.

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.



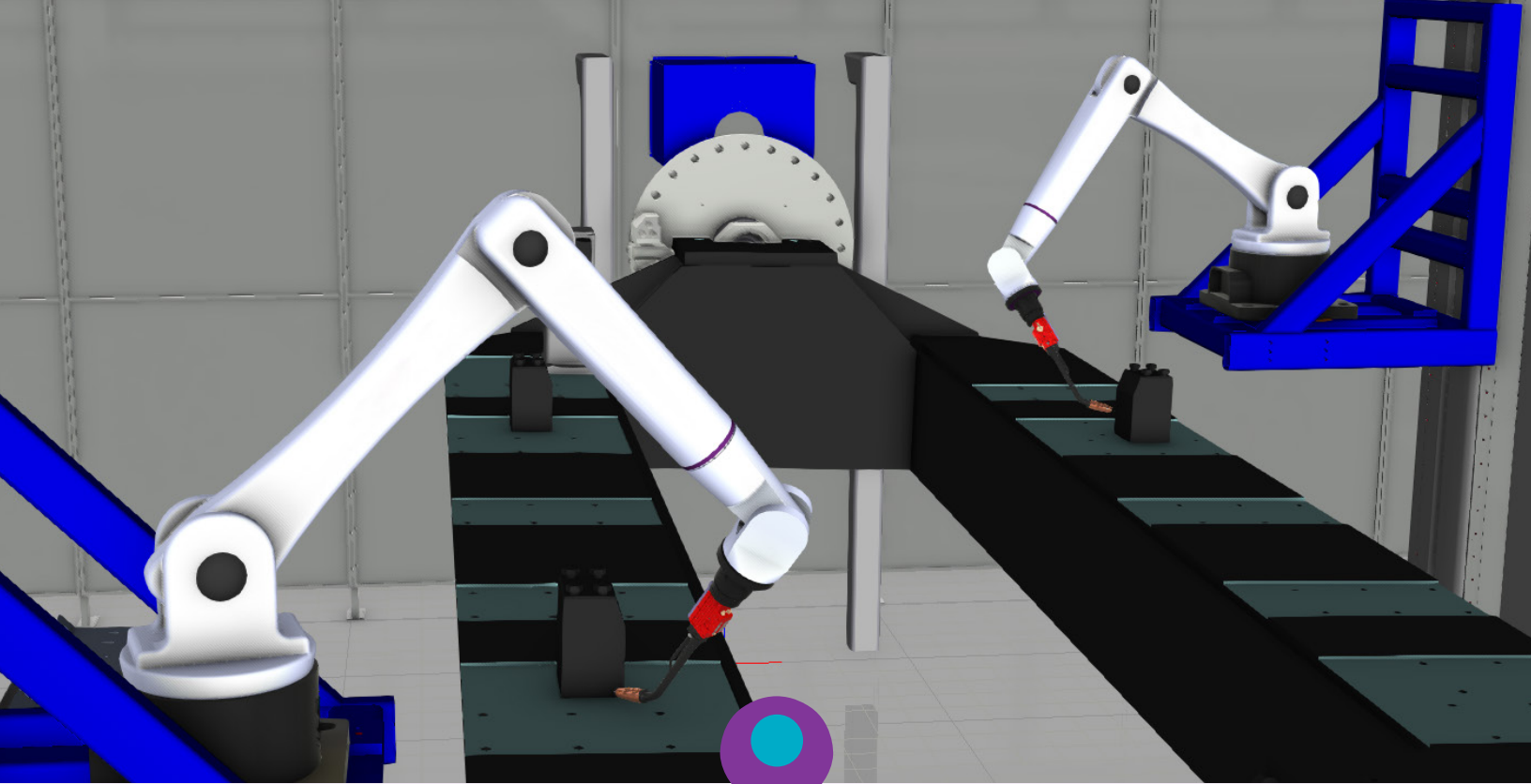
Protect your investment!
OCTOPUZ can detect
collisions offline.



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