

# **Mastercam** 2019

METRIC – TRAINING TUTORIAL SERIES



Demo Software Download Instructions Included

# MILL ESSENTIALS

# ***Mastercam***® 2019

---

**MILL ESSENTIALS METRIC TRAINING TUTORIAL**

---

## **Mastercam 2019 Mill Essentials Tutorial**

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

Software: Mastercam 2019

Authors: Mariana Lendel

ISBN: 978-1-77146-814-5

Date: September 17, 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

---

<b>Mill Essentials Projects</b>	<b>13</b>
<b>Getting Started</b>	<b>17</b>
Objectives	18
Step 1: Starting Mastercam	18
Step 2: GUI - Graphical User Interface	19
Step 3: Navigate Through Mastercam	20
Step 4: Set The Attributes	22
Step 5: Manager Panels	23
Step 6: Setting Mastercam To Metric	25
Step 7: Set The Grid	27
Conventions Used In This Book:	28
Mastercam® Work Flow	29
<b>Tutorial 1: Geometry Creation</b>	<b>33</b>
Tutorial #1 Drawing	35
Step 1: Setting Up The Graphical User Interface	36
Step 2: Create One Rectangle	37
Step 3: Create The 6mm Diameter Circles	40
Step 4: Create The 48mm And 18mm Diameter Circles	43
Step 5: Create The Chamfers	46
Step 6: Save The File	48
Tutorial #1 Review Exercise	49
Create The Geometry For Tutorial #1 Exercise	50
<b>Tutorial 1: Toolpath Creation</b>	<b>53</b>
Suggested Fixture	55
Setup Sheet	56
Step 1: Select The Machine And Set Up The Stock	57



---

Step 2: Face The Part .....	62
Step 3: Circle Mill The Large Hole .....	69
Step 4: Backplot The Toolpaths .....	77
Step 5: Simulate The Toolpath In Verify .....	79
Step 6: Circle Mill The Inside Hole .....	80
Step 7: Spot Drill The 6mm Holes .....	89
Step 8: Drill The 6mm Holes .....	99
Step 9: Chamfer The Large Hole .....	105
Step 10: Machine The Chamfers At The Corners Using Contour Toolpath .....	113
Step 11: Post The File .....	120
Step 12: Save The Updated MCAM File .....	121
Create The Toolpaths For Tutorial #1 Exercise .....	122
<b>Tutorial 2: Geometry Creation .....</b>	<b>125</b>
Tutorial #2 Drawing .....	127
Step 1: Setting Up The Graphical User Interface .....	128
Step 2: Create A Rectangles .....	129
Step 3: Create Two Obround Shapes .....	132
Step 4: Create A Circle .....	136
Step 5: Use Trim Divide To Clean The Circle .....	138
Step 6: Create Parallel Lines .....	141
Step 7: Use Trim Divide Delete To Clean Up The Geometry .....	144
Step 8: Create Angular Lines .....	146
Step 9: Create Polygon .....	149
Step 10: Create Fillets .....	151
Step 11: Save The File .....	157
Tutorial #2 Review Exercise .....	158
Create The Geometry For Tutorial #2 Exercise .....	159
<b>Tutorial 2: Toolpath Creation .....</b>	<b>161</b>

---

Toolpath Creation - Setup #1 .....	163
Suggested Fixture .....	163
Setup Sheet .....	163
Step 1: Select The Machine And Set Up The Stock .....	164
Step 2: Slot Milling .....	169
Step 3: Backplot The Toolpaths .....	179
Step 4: Simulate The Toolpath In Verify .....	181
Step 5: Machine The Cutout Pockets .....	182
Step 6: Finish The Inside Shapes Using 2D HS Dynamic Contour .....	192
Step 7: Rough The Outside Using High Speed Dynamic Mill .....	198
Step 8: Finish The Outside Profile Using Contour Toolpath .....	210
Suggested Fixture: .....	215
Setup Sheet: .....	215
Step 9: Creating And Renaming Toolpath Groups .....	216
Step 10: Set WCS To Bottom .....	218
Step 11: Face The Part .....	224
Step 12: Rename The NC File .....	230
Step 13: Post The File .....	232
Step 14: Save The Updated MCAM File .....	234
Create The Toolpaths For Tutorial #2 Exercise .....	235
<b>Tutorial 3: Geometry Creation .....</b>	<b>239</b>
Tutorial #3 Drawing .....	241
Step 1: Setting Up The Graphical User Interface .....	242
Step 2: Create Two Polar Arcs .....	242
Step 3: Create A Circle .....	246
Step 4: Create A Line Tangent To The Circle .....	248
Step 5: Create A Fillet .....	250

---

Step 6: Mirror The Geometry .....	252
Step 7: Create An Arc Tangent To Two Entities .....	254
Step 8: Trim The Geometry Using Trim 3 Entities Command .....	257
Step 9: Mirror The Geometry .....	260
Step 10: Create The Ellipses .....	262
Step 11: Offset The Ellipse .....	265
Step 12: Add The Text .....	268
Step 13: Save The File .....	275
Tutorial #3 Review Exercise .....	276
Create The Geometry For Tutorial #3 Exercise .....	277
<b>Tutorial 3: Toolpath Creation .....</b>	<b>279</b>
Suggested Fixture .....	281
Setup Sheet .....	281
Step 1: Select The Machine And Set Up The Stock .....	282
Step 2: 2D High Speed Area Mill .....	286
Step 3: Backplot The Toolpaths .....	301
Step 4: Simulate The Toolpath In Verify .....	303
Step 5: 2D High Speed Dynamic Mill .....	304
Step 6: Pocket .....	314
Step 7: Pocket Remachining .....	323
Step 8: Post The File .....	330
Step 9: Save The Updated MCAM File .....	331
Create The Toolpaths For Tutorial #3 Exercise .....	332
<b>Tutorial 4: Geometry Creation .....</b>	<b>335</b>
Tutorial #4 Drawing .....	337
Step 1: Setting Up The Graphical User Interface .....	338
Step 2: Create Two Arcs .....	338
Step 3: Create Vertical Lines .....	347

---

Step 4: Create Arc Using Arc Endpoints .....	349
Step 5: Rotate Geometry .....	351
Step 6: Mirror Geometry .....	353
Step 7: Create An Arc Tangent .....	355
Step 8: Trim Geometry .....	358
Step 9: Mirror Geometry To Complete Arms .....	361
Step 10: Create A Construction Line .....	364
Step 11: Create A 12mm Diameter Circle .....	366
Step 12: Delete Construction Geometry .....	368
Step 13: Create Tangent Lines .....	369
Step 14: Create Arc Polar .....	374
Step 15: Create Fillets .....	377
Step 16: Trim The Arc .....	380
Step 17: Rotate .....	382
Step 18: Translate .....	384
Step 19: Change The Main Level To 2 .....	389
Step 20: Create The Solid Body By Extruding A Closed Chain .....	390
Step 21: Extrude Cut The Pockets And The Holes .....	398
Step 22: Chamfer The Part .....	406
Step 23: Save The File .....	412
Tutorial #4 Review Exercise .....	413
Create The Geometry For Tutorial #4 Exercise .....	414
Create The Solid Geometry For Tutorial #4 Exercise .....	414
<b>Tutorial4: Toolpath Creation .....</b>	<b>417</b>
Suggested Fixture .....	419
Setup Sheet .....	420
Step 1: Select The Machine And Set Up The Stock .....	421

---

Step 2: 2D High Speed Dynamic Mill .....	425
Step 3: Backplot The Toolpaths .....	438
Step 4: Simulate The Toolpath In Verify .....	440
Step 5: Finish The Walls Using Contour Toolpath .....	441
Step 6: Area Mill Toolpath .....	449
Step 7: Transform-Rotate Toolpath .....	461
Step 8: Finish The Pocket Walls Using Contour Toolpath .....	465
Step 9: Spot Drill The Holes .....	471
Step 10: Drill All Holes .....	477
Step 11: Chamfer The Outside Diameter .....	481
Step 12: Chamfer The Pockets .....	488
Suggested Fixture 2: .....	497
Setup Sheet 2: .....	497
Step 13: Creating And Renaming Toolpath Groups .....	498
Step 14: Set The WCS To Bottom .....	500
Step 15: 2D HS Dynamic Mill .....	503
Step 16: Finish The Pocket Wall Using Contour Toolpath .....	512
Step 17: Rename The NC File .....	518
Step 18: Post The File .....	519
Step 19: Save The Updated MCAM File .....	521
Create The Toolpaths For Tutorial #4 Exercise .....	522
<b>Tutorial 5: Geometry Creation .....</b>	<b>527</b>
Tutorial #5 Drawing .....	529
Step 1: Setting Up The Graphical User Interface .....	530
Step 2: Importing The Solidworks File Geometry .....	530
Step 3: Save The File .....	532
Tutorial #5 Review Exercise .....	533
Create The Geometry For Tutorial #5 Exercise .....	534

---

<b>Tutorial 5: Toolpath Creation</b>	<b>537</b>
Suggested Fixture	539
Setup Sheet	540
Step 1: Select The Machine And Set Up The Stock	541
Step 2: 2D High Speed Area Mill	546
Step 3: Backplot The Toolpaths	559
Step 4: Simulate The Toolpath In Verify	560
Step 5: 2D High Speed Area Mill	561
Step 6: Area Mill To Remachine The Remaining Material	573
Step 7: Spot Drill The Holes	583
Step 8: Drill All Holes	591
Step 9: Tap The Holes	597
Suggested Fixture 2:	602
Setup Sheet 2:	603
Step 10: Creating And Renaming Toolpath Groups	604
Step 11: Set WCS To Front	606
Step 12: Spot Drill All 3 Holes	609
Step 13: Drill The Two 10mm Tapped Holes	617
Step 14: Tap The Two Holes	624
Step 15: Drill The 10mm Hole	630
Step 16: Counterbore The 12mm Hole	635
Step 17: Rename The NC File	641
Setup Sheet 3:	643
Step 18: Creating And Renaming Toolpath Groups	644
Step 19: Set The WCS To Left Side	645
Step 20: Machine The Slot	647
Step 21: Rename The NC File	655

---

Step 22: Post The File .....	656
Step 23: Save The Updated MCAM File .....	657
Create The Toolpaths For Tutorial #5 Exercise .....	658
<b>Tutorial 6: Geometry Creation .....</b>	<b>669</b>
Tutorial #6 Drawing .....	671
Step 1: Setting Up The Graphical User Interface .....	672
Step 2: Create Rectangles .....	672
Step 3: Create Circles And Arc Tangent .....	675
Step 4: Trim The Arcs .....	680
Step 5: Create Line Parallel .....	685
Step 6: Create Chamfers .....	687
Step 7: Create Polar Lines .....	689
Step 8: Create Line Parallel .....	692
Step 9: Delete Entities .....	694
Step 10: Trim Entities .....	695
Step 11: Save The File .....	699
Tutorial #6 Review Exercise .....	700
Create The Geometry For Tutorial #6 Exercise .....	701
<b>Tutorial 6: Toolpath Creation .....</b>	<b>703</b>
Suggested Fixture .....	705
Setup Sheet .....	705
Step 1: Select The Machine And Set Up The Stock .....	706
Step 2: 2D High Speed Dynamic Mill .....	711
Step 3: Backplot The Toolpaths .....	724
Step 4: Simulate The Toolpath In Verify .....	725
Step 5: 2D High Speed Area Mill .....	726
Step 6: 2D High Speed Blend Mill .....	737
Step 7: 2D High Speed Peel Mill .....	748



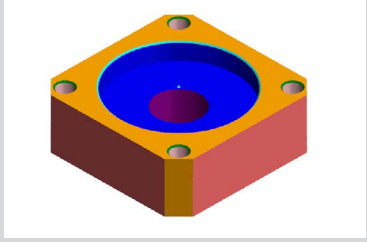
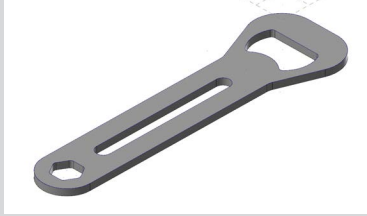
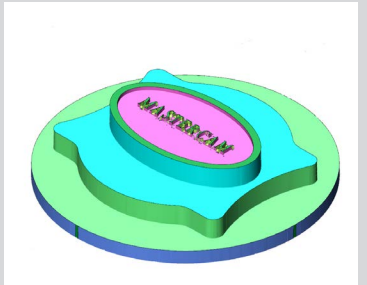
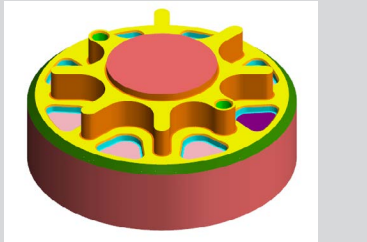
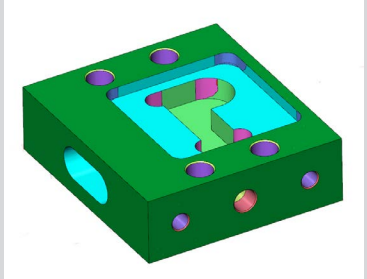
---

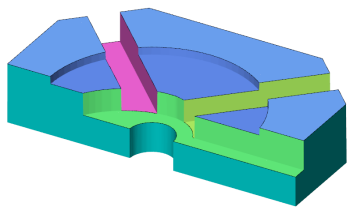

Step 8: 2D High Speed Peel Mill .....	754
Step 9: Post The File .....	761
Step 10: Save The Updated MCAM File .....	762
Create The Toolpaths For Tutorial #6 Exercise .....	763
<b>Tutorial 7: Geometry Creation .....</b>	<b>765</b>
Tutorial #7 Drawing .....	767
Step 1: Setting Up The Graphical User Interface .....	768
Step 2: Create Circle Center Point .....	768
Step 3: Create Line Tangent .....	770
Step 4: Create A Line Parallel .....	772
Step 5: Trim The Entities .....	774
Step 6: Create Rectangular Shape .....	777
Step 7: Trim Divide .....	781
Step 8: Fillet Chains .....	783
Step 9: Create The Obround Shapes .....	786
Step 10: Create Circle Center Point .....	789
Step 11: Set The Level .....	793
Step 12: Extrude The Base Of The Solid .....	795
Step 13: Chamfer The Holes .....	805
Step 14: Fillet The Edges .....	808
Step 15: Save The File .....	811
Tutorial #7 Review Exercise .....	812
Create The Geometry For Tutorial #7 Exercise .....	813
Create The Solid Geometry For Tutorial #7 Exercise .....	813
<b>Tutorial 7: Toolpath Creation .....</b>	<b>815</b>
Suggested Fixture .....	817
Setup Sheet .....	817

---

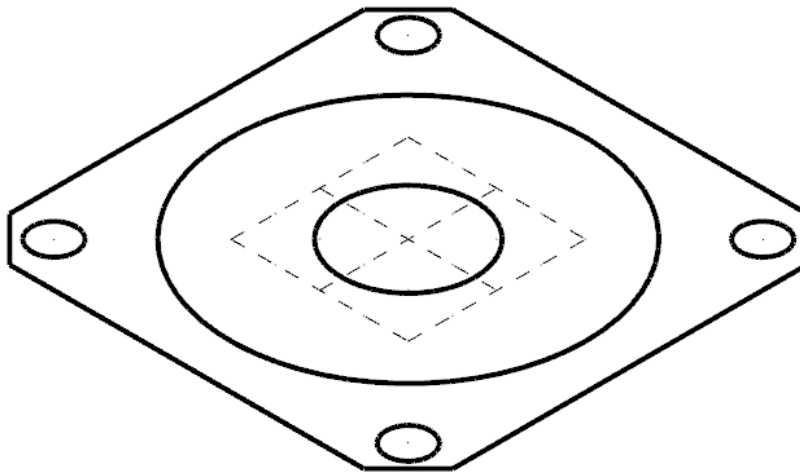
Step 1: Select The Machine And Set Up The Stock .....	819
Step 2: 2D High Speed Dynamic Mill .....	824
Step 3: Backplot The Toolpaths .....	836
Step 4: Simulate The Toolpath In Verify .....	837
Step 5: Drill The Holes Using FBM Drill .....	838
Step 6: Machine The Outside Of The Part Using Dynamic Milling .....	847
Step 7: Finish The Pockets Using A Pocket Toolpath .....	860
Step 8: Contour Toolpath .....	870
Step 9: Post The File .....	879
Step 10: Save The Updated MCAM File .....	880
Create The Toolpaths For Tutorial #7 Exercise .....	881
<b>Creating/Editing A Mill Tool Library .....</b>	<b>887</b>
<b>Quiz Answers .....</b>	<b>899</b>

## Mill Essentials Projects

Tutorial	Geometry Functions	Toolpath Creation
<p>#1</p> 	<p>Rectangle. Circle Center Point. Chamfer Entities.</p>	<p>Facing Toolpath. Circle Mill Toolpath. Contour Toolpath. Spot Drill Toolpath. Drill Toolpath. 2D Contour (Chamfer Toolpath).</p>
<p>#2</p> 	<p>Rectangle. Rectangular Shapes. Polygon. Fillet Entities. Fillet Chains. Line Endpoints. Trim Divide.</p>	<p>Setup 1 Slot Mill Toolpath. 2D HS Dynamic Mill Toolpath. Contour Toolpath. 2D HS Dynamic Contour Toolpath. Setup 2 Facing Toolpaths.</p>
<p>#3</p> 	<p>Polar Arcs. Circle Center Point. Line Tangent. Fillet Entities. Mirror. Arc Tangent to 2 Entities. Trim 3 Entities. Ellipse. Offset. Letters. Bounding Box. Translate.</p>	<p>2D High Speed Area Mill Toolpath. 2D HS Dynamic Mill Toolpath. Pocket with Island Toolpath. Pocket Remachine Toolpath.</p>
<p>#4</p> 	<p>Circle Center Point. Line Tangent. Mirror. Arc Tangent. Arc Polar. Trim. Filleets. Rotate. Translate. Solids Extrude. Chamfer.</p>	<p>Setup 1 2D High Speed Area Mill Toolpath. 2D HS Dynamic Mill Toolpath. Transform Toolpath. Drill Toolpath. Contour (Chamfer Toolpath). Setup 2 2D HS Dynamic Mill Toolpath.</p>
<p>#5</p> 	<p>Import a SolidWorks file. Translate 3D.</p>	<p>Setup 1 - Top Tool Planes. 2D HS Area Mill Toolpath. 2D HS Area Mill Rest Toolpath. Setup 2 - Front Tool Plane. Drill Toolpath. Setup 3 - Left Tool Plane. Slot Mill Toolpath.</p>

Tutorial	Geometry Functions	Toolpath Creation
<p>#6</p> 	<p>Rectangle. Circle Center Point. Arc Tangent to 1 Entity. Line Parallel. Chamfer. Line Polar. Trim.</p>	<p>2D HS Dynamic Mill Toolpath. 2D HS Core Mill Toolpath. 2D HS Blend Mill Toolpath. 2D HS Peel Mill Toolpath.</p>
<p>#7</p> 	<p>Circle Center Point. Line Tangent. Line Parallel. Rectangular Shapes. Trim. Fillet Chains. Solids Extrude. Solids Chamfer. Solids Fillet.</p>	<p>2D HS Area Mill Toolpath. Feature Based Drilling Toolpath. 2D HS Area Mill Toolpath. Pocket Toolpath. 2D Contour Toolpath.</p>

## Tutorial 1: Geometry Creation



## OVERVIEW OF STEPS TAKEN TO CREATE THE PART GEOMETRY:

### From Drawing to CAD Model:

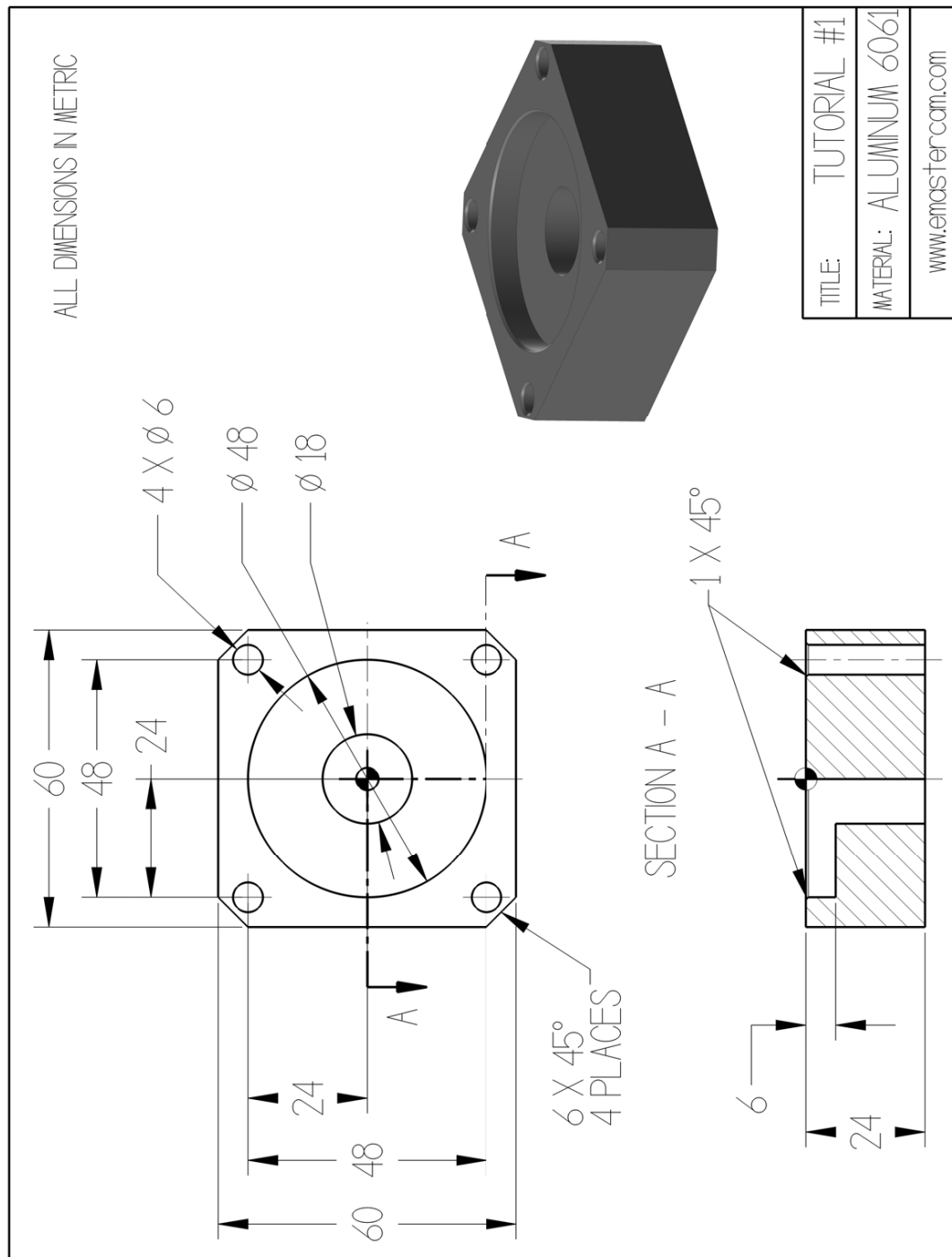
- ◆ The student should examine the drawing on the following page to understand what part is being created in the tutorial.
- ◆ From the drawing we can decide how to create the geometry in Mastercam.

### Create the 2D CAD Model:

- ◆ The student will create the Top 2D geometry needed to create the toolpaths.
- ◆ Geometry creation commands such as Rectangle, Circle Center Point, and Chamfer Entities will be used.



## TUTORIAL #1 DRAWING



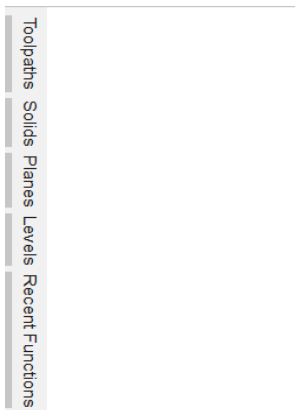
## STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

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.

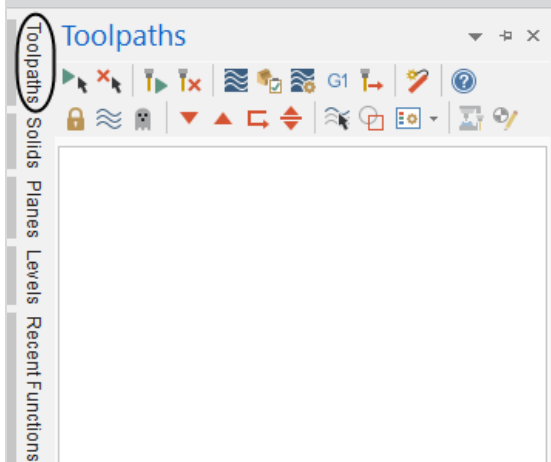
- ◆ Use **Auto Hide** icon to hide all **Manager** panels.



- ◆ 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.*

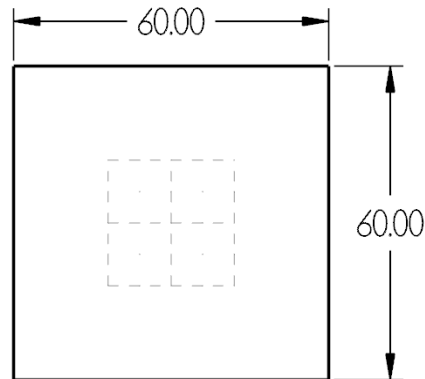


*While creating the geometry, keep the Manager panels hidden. This ensures more space in the graphics window for the geometry.*

## STEP 2: CREATE ONE RECTANGLE

In this step, you will learn how to create a rectangle given the width, the height, and the anchor position. You will create the 60 mm by 60 mm rectangle with the center anchor in the Origin.

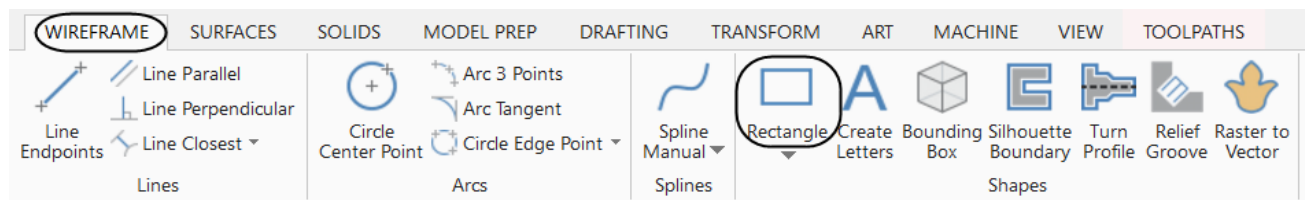
*Step Preview:*



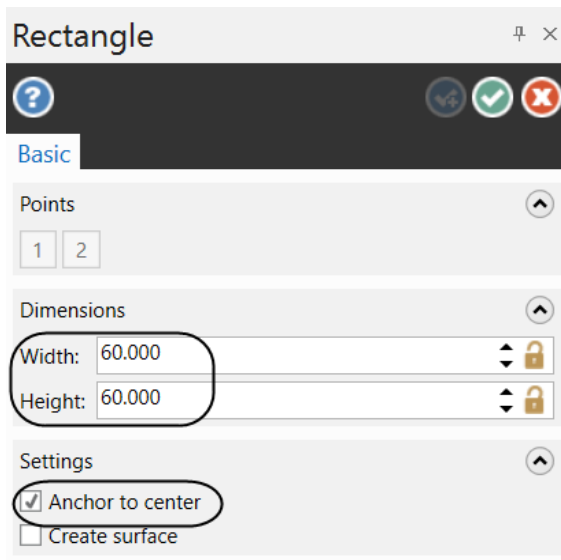
### 2.1 Create a 60 mm by 60 mm Rectangle

#### WIREFRAME

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

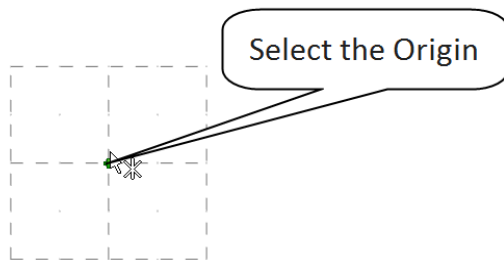


- ◆ In the **Rectangle** panel, enter the **Width** and **Height** and enable **Anchor to center** as shown.

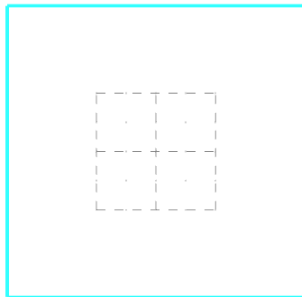


*Note: Make sure that **Create surface** is not selected. **Anchor to center** sets the base point of the rectangle to its center and draws the rectangle outward from the center. **Create surface** creates a surface inside of the rectangle. Surface creation and Surface toolpath are covered in Mill Advanced.*

- ◆ Select the position of the base point as shown.



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

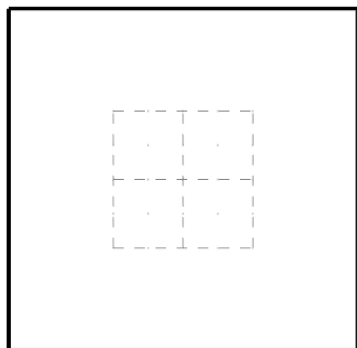




*Note: The geometry should appear in cyan blue color which is the color for the live entities. While the rectangle is live, you can adjust the dimensions or select a new base point.*

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



- ◆ The geometry should look as shown.

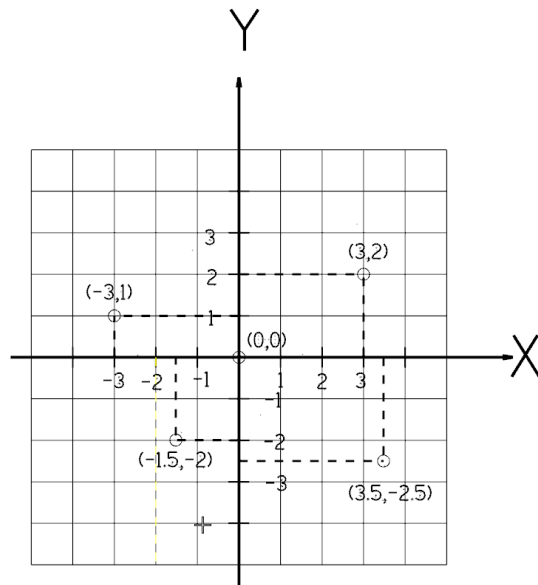


*Note: While creating geometry for this tutorial, if you make a mistake, you can undo the last step using the **Undo** icon.  You can undo as many steps as needed. If you delete or undo a step by mistake, just use the **Redo** icon.  To delete unwanted geometry, select the geometry first and then press **Delete** from the keyboard. To zoom or un-zoom, move the cursor in the center of the geometry and scroll up or down the mouse wheel.*

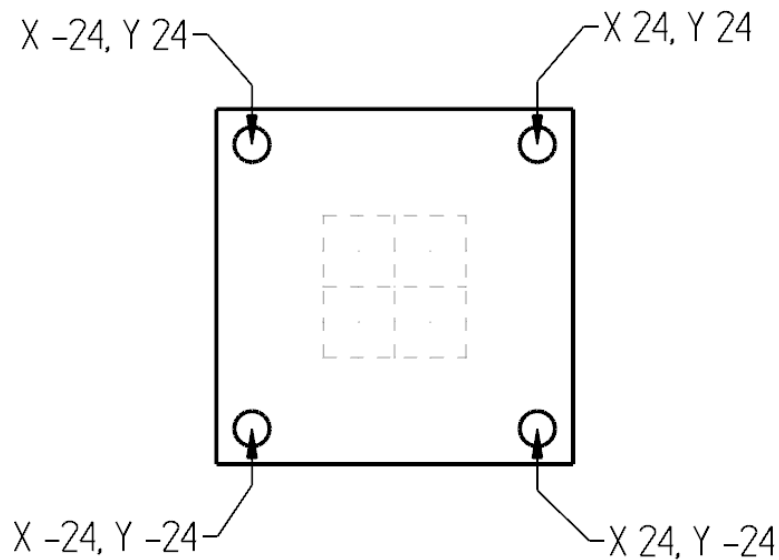
### STEP 3: CREATE THE 6MM DIAMETER CIRCLES

In this step, you will create circles for which you know the diameter and the locations. To use **Circle Center Point**, you need to know the center point and the radius or the diameter of the circle. To complete this step, you will need to know the **Cartesian Coordinate System**. A **Cartesian Coordinate System** is a coordinate system that specifies each point uniquely in a plane by a pair of numerical coordinates, which are the signed distances from the point to two fixed perpendicular directed lines, measured in the same unit of length as shown in [Figure: 3.0.1](#).

Figure: 3.0.1

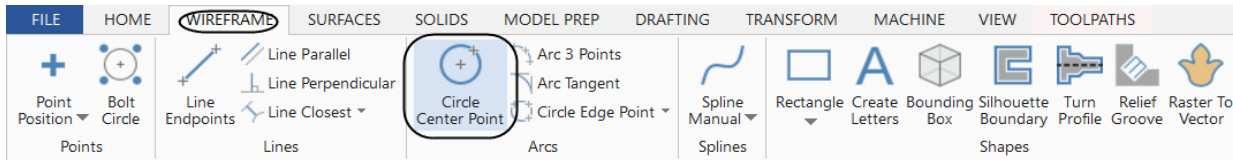


*Step Preview:*

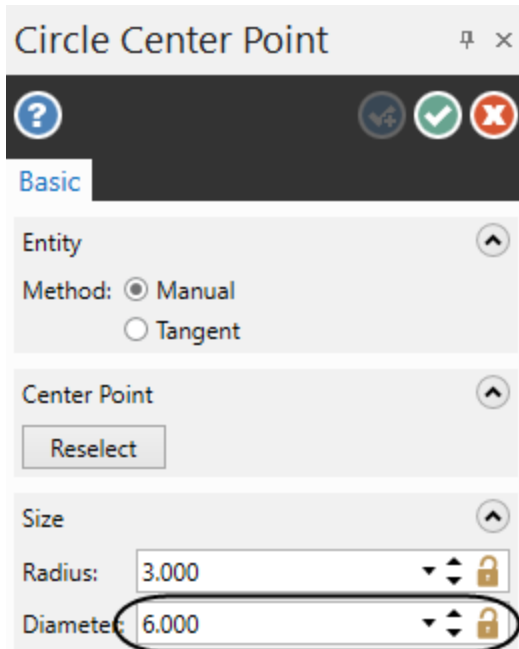


## WIREFRAME

- ◆ From the **Arcs** group, select **Circle Center Point**.



- ◆ Enter a **Diameter** of **6.0** in the panel as shown.
- ◆ To create all four circles, click on the locker icon to lock the value.



- ◆ [Enter the center point]: Select the **AutoCursor Fast Point** icon from the **General Selection** toolbar and the field where you can type the coordinates will open at the upper left side of the graphics window as shown.



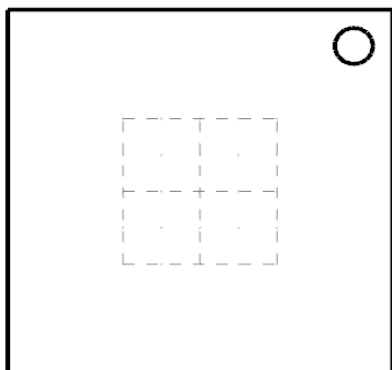
- ◆ Type **24, 24** as shown.

24,24

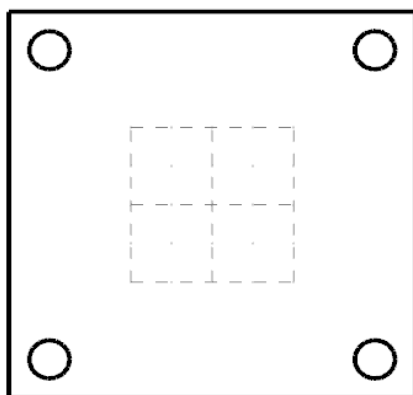
*Note: When entering the coordinates for the center point, the first value is the **X** coordinate value, then the **Y** value followed by the **Z** value only if it is different from zero. The coordinate values are separated with commas. You do not need to use the coordinate labels if you enter the values in this order.*



- ◆ Press **Enter** and the circle will be placed as shown.



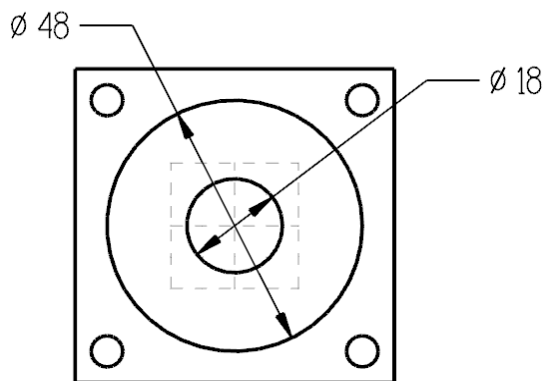
- ◆ [Enter the center point]: Select the **AutoCursor Fast Point** icon again and enter **24,-24**.
- ◆ Press **Enter** to place the circle.
- ◆ [Enter the center point]: Select the **AutoCursor Fast Point** icon again and enter **-24,24**.
- ◆ Press **Enter** to place the circle.
- ◆ [Enter the center point]: Select the **AutoCursor Fast Point** icon again and enter **-24,-24**.
- ◆ Press **Enter** to place the circle.
- ◆ Once complete choose the **OK** button to exit the command.
- ◆ The geometry should look as shown.



## STEP 4: CREATE THE 48MM AND 18MM DIAMETER CIRCLES

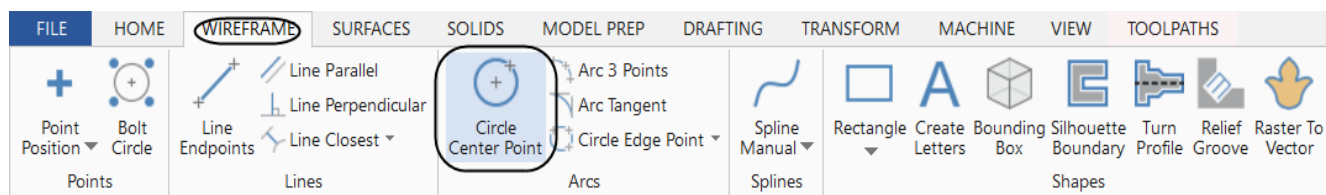
In this step, you will use the same Circle Center Point to create circles that you know the diameters and the locations.

*Step Preview:*

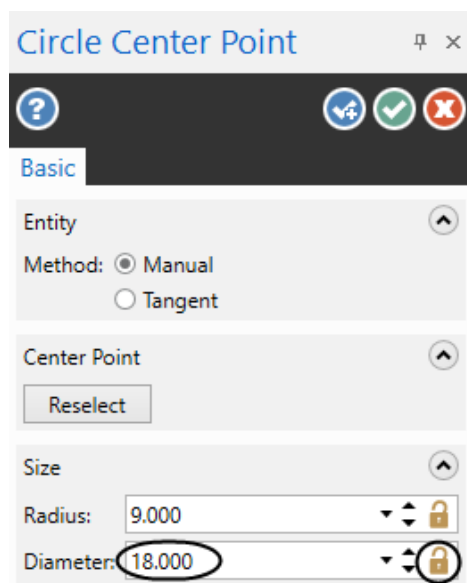


### WIREFRAME

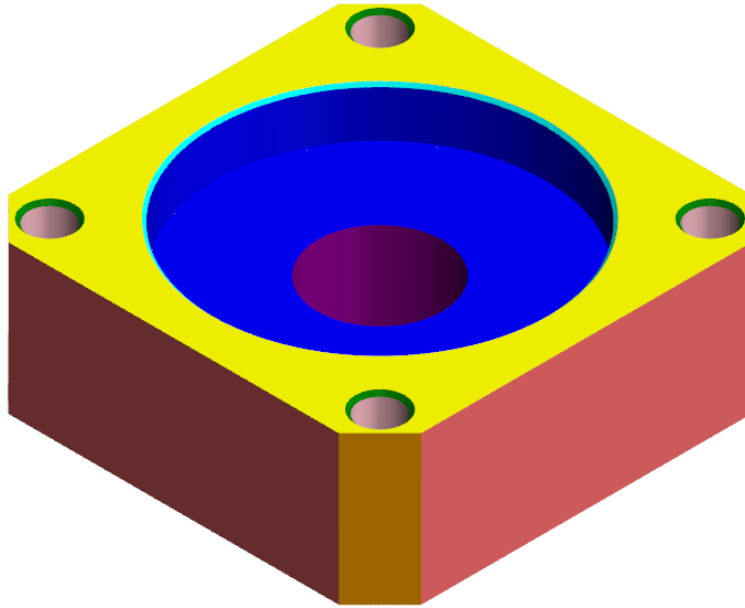
- ◆ From **Arcs** group, select **Circle Center Point**.



- ◆ Enter the **Diameter 18.0** in the panel and disable the locker icon as shown.



## 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 machine the top of the part.
- ◆ A Circle Mill toolpath will remove the material inside of the large hole.
- ◆ A Drilling toolpath will be created to spot drill the four holes.
- ◆ A Drilling toolpath will be created to machine the through holes.
- ◆ A Contour toolpath with 2D Chamfer option will be used to chamfer the top of the large hole.
- ◆ A Contour toolpath will be used to machine the corners.

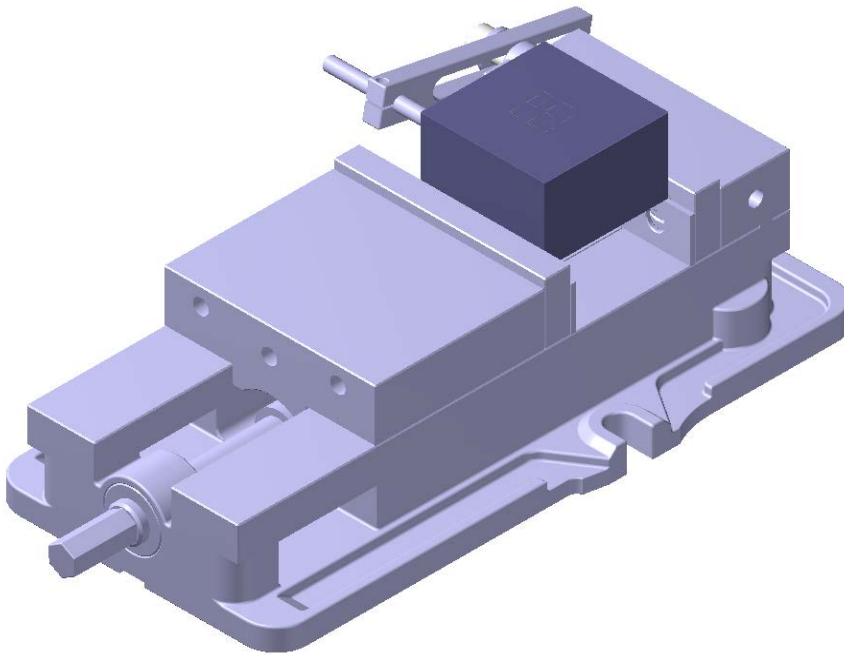
### Backplot and Verify the file:

- ◆ 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.

## SUGGESTED FIXTURE



## SETUP SHEET

### TOOL LIST

	<p>TYPE: Face mill</p> <p>DIA OFFSET: 1</p> <p>HOLDER: Default Holder</p> <p>NUMBER: 1</p> <p>LENGTH OFFSET: 1</p> <p>FLUTE LENGTH: 40.0</p> <p>OVERALL LENGTH: 65.0</p> <p>CORNER RAD: 0.0</p> <p># OF FLUTES: 4</p> <p>#1 - M50.00 FACE MILL - FACE MILL - 50/58</p>
	<p>TYPE: Endmill1 Flat</p> <p>DIA OFFSET: 2</p> <p>HOLDER: Default Holder</p> <p>NUMBER: 2</p> <p>LENGTH OFFSET: 2</p> <p>FLUTE LENGTH: 19.0</p> <p>OVERALL LENGTH: 83.0</p> <p>CORNER RAD: 0.0</p> <p># OF FLUTES: 4</p> <p>#2 - M12.00 ENDMILL1 FLAT - FLAT END MILL - 12</p>
	<p>TYPE: Endmill1 Flat</p> <p>DIA OFFSET: 3</p> <p>HOLDER: Default Holder</p> <p>NUMBER: 3</p> <p>LENGTH OFFSET: 3</p> <p>FLUTE LENGTH: 10.0</p> <p>OVERALL LENGTH: 57.0</p> <p>CORNER RAD: 0.0</p> <p># OF FLUTES: 4</p> <p>#3 - M6.00 ENDMILL1 FLAT - FLAT END MILL - 6</p>
	<p>TYPE: Spot Drill</p> <p>DIA OFFSET: 4</p> <p>HOLDER: Default Holder</p> <p>NUMBER: 4</p> <p>LENGTH OFFSET: 4</p> <p>FLUTE LENGTH: 40.0</p> <p>OVERALL LENGTH: 131.0</p> <p>CORNER RAD: 0.0</p> <p># OF FLUTES: 1</p> <p>#4 - M20.00 SPOT DRILL - NC SPOT DRILL - 20</p>
	<p>TYPE: Drill</p> <p>DIA OFFSET: 5</p> <p>HOLDER: Default Holder</p> <p>NUMBER: 5</p> <p>LENGTH OFFSET: 5</p> <p>FLUTE LENGTH: 48.0</p> <p>OVERALL LENGTH: 93.0</p> <p>CORNER RAD: 0.0</p> <p># OF FLUTES: 1</p> <p>#5 - M6.00 DRILL - HSS/TIN DRILL 8XDC- 6.0</p>
	<p>TYPE: Chamfer mill</p> <p>DIA OFFSET: 6</p> <p>HOLDER: Default Holder</p> <p>NUMBER: 6</p> <p>LENGTH OFFSET: 6</p> <p>FLUTE LENGTH: 10.0</p> <p>OVERALL LENGTH: 75.0</p> <p>CORNER RAD: 0.0</p> <p># OF FLUTES: 4</p> <p>#6 - M10.00 CHAMFER MILL - CHAMFER MILL 10/90DEG</p>

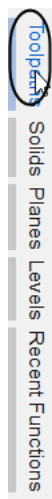
## STEP 1: SELECT THE MACHINE AND SET UP THE STOCK

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.

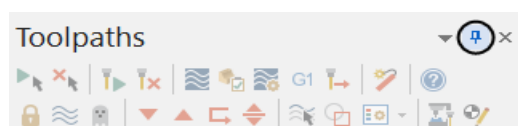
*Note: For the purpose of this tutorial, we will be using the **Default Mill MM** machine.*

### 1.1 Unhide the Toolpaths Manager panel

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



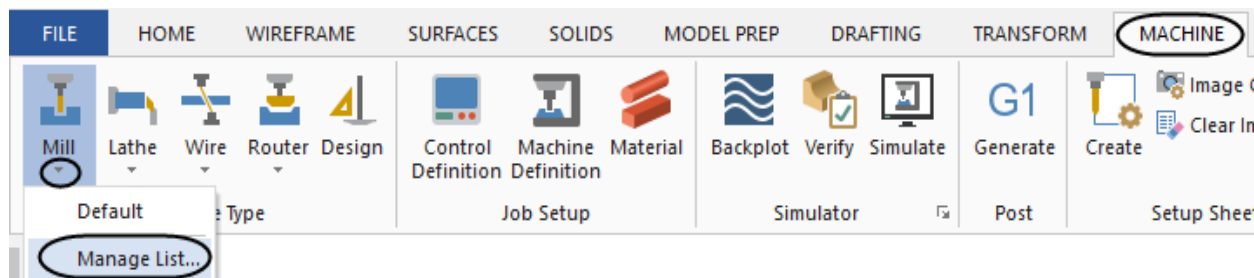
- ◆ Pin the **Toolpaths Manager** by clicking on the **Auto Hide** icon as shown.



### 1.2 Select the machine

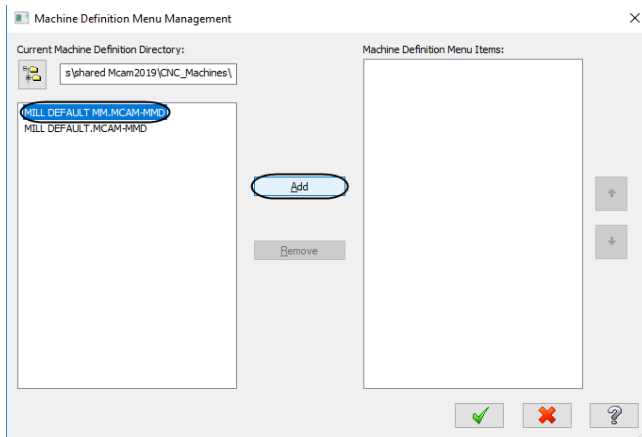
#### MACHINE


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

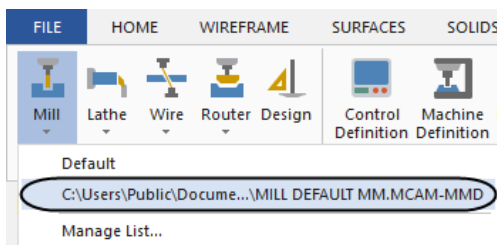




- ◆ Select **MILL DEFAULT MM.MCAM-MMD** from the list and press **Add**.

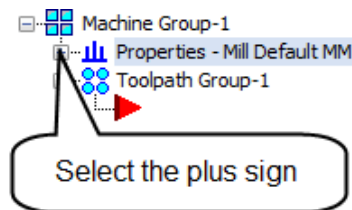


- ◆ Select the **OK** button to exit **Machine Definition Menu Management**. 
- ◆ From the Machine Type area, click on the drop down arrow and select **MILL DEFAULT MM.MCAM-MMD** as shown.

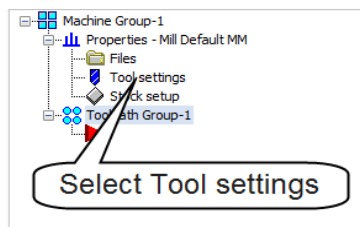


*Note: Once you select the **MILL DEFAULT MM.MCAM -MMD** the ribbon bar changes to reflect the toolpaths that can be used with this machine.*

- ◆ 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 as shown.

**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.

Machine Group Properties

Files Tool Settings Stock Setup

Default program number 1

Feed Calculation

- ☒ From tool
- ☐ From material
- ☐ From defaults
- ☐ User defined

Spindle speed 5000.0

Feed rate 100.0

Retract rate 150.0

Plunge rate 25.0

☐ Adjust feed on arc move

Minimum arc feed 125.0

Toolpath Configuration

- ☒ Assign tool numbers sequentially
- ☒ Warn of duplicate tool numbers
- ☐ Use tool's step, peck, coolant
- ☐ Search tool library when entering a tool number

Advanced options

- ☒ Override defaults with modal values
- ☒ Clearance height
- ☒ Retract height
- ☒ Feed plane

Sequence number

Start 100.0

Increment 10.0

Material

ALUMINUM mm - 2024

Edit... Select...

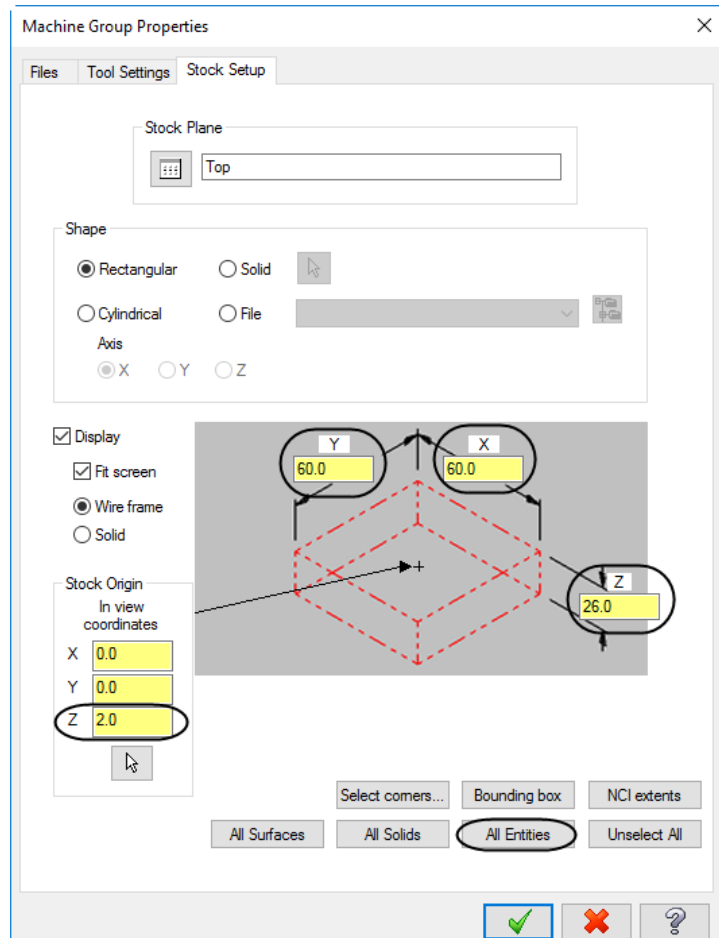
OK Cancel Help

- ◆ Select the **Stock Setup** tab to define the stock.
- ◆ Select the **All Entities** button near the bottom of the **Stock Setup** page as shown.
- ◆ In the **Stock Setup**, enter in the **Z** field **26** and the **Z Stock Origin** **2**. Make sure that the rest of the parameters are as shown.

The **X, Y, Z** values in the graphics area are the dimensions of the stock model. They are always positive values.

The **Stock Origin** values adjust the positioning of the stock, ensuring that you have an equal amount of extra stock around the finished part. In the graphics screen, the plus sign (+) shows you where the stock origin is. The default position is the middle of the stock.

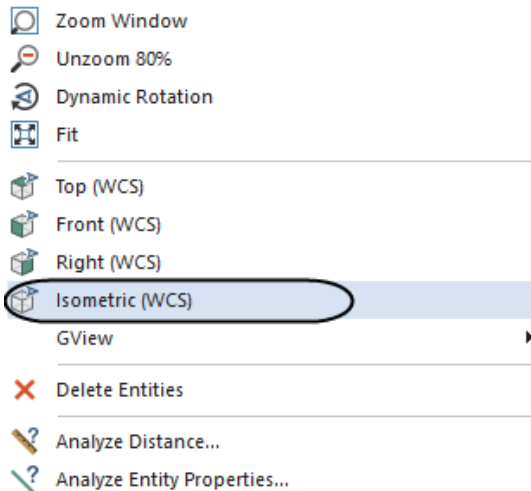
**Display options** allow you to set the stock as **Wireframe** and to fit the stock to the screen. (Fit Screen)



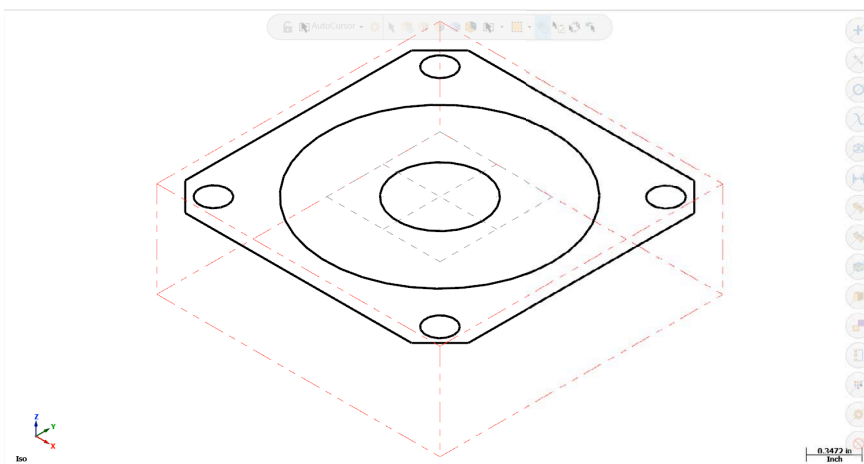
*Note: The stock model that you create is displayed when viewing the file or the toolpaths, during backplot, and while verifying toolpaths.*

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

- ◆ Right mouse click in the graphics window and select the **Isometric** view to see the stock.



- ◆ Press **Alt + F1** to fit the drawing to the screen.
- ◆ The stock model will appear as shown.



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

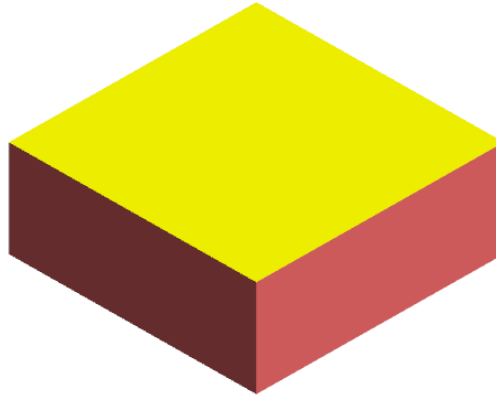
- ◆ Right mouse click in the graphics window and select the **Top** view to see the stock and if needed, press **Alt + F1** to fit the drawing to the screen.



## STEP 2: FACE THE PART

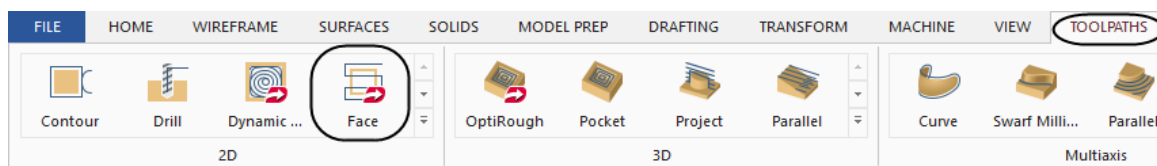
A **Facing** toolpath quickly removes material from the top of the part to create an even surface for future operations.

*Toolpath Preview:*



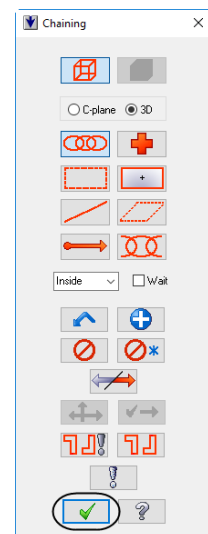
### TOOLPATHS

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



- ◆ When the **Chaining** dialog box appears, choose the **OK** button to use the defined stock and exit the Chaining dialog box.

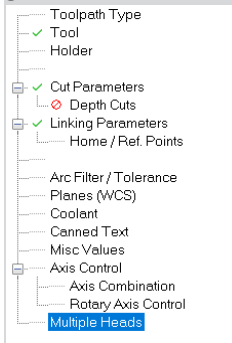
*Note: Mastercam will create the **Facing** toolpath defined from the stock setup.  
For more information on the **Chaining** button and **Options**, click on the **Help** button.*



- ◆ In the **Toolpath Type** page, the **Facing** icon will be automatically selected.

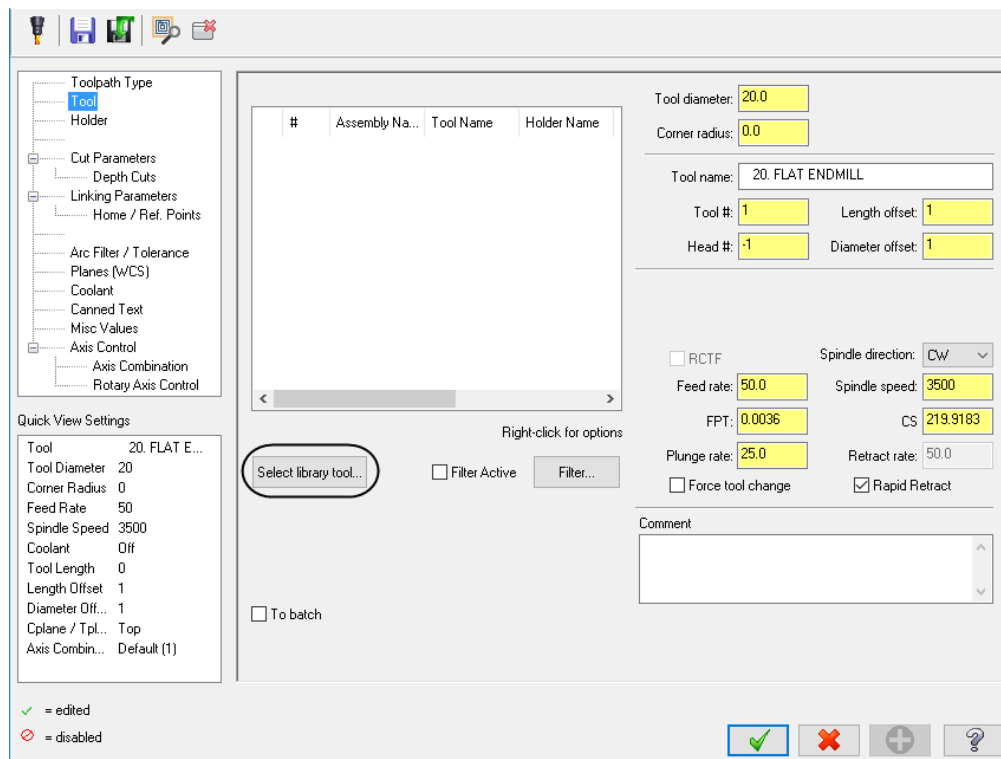


*Note: Mastercam updates the pages as you modify them and then marks them, in the **Tree View** list, with a green check mark. Pages that are not enabled are marked with a red circle and slash.*



### 2.1 Select a 50mm Face Mill from the library and set the Tool parameters

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



# 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.



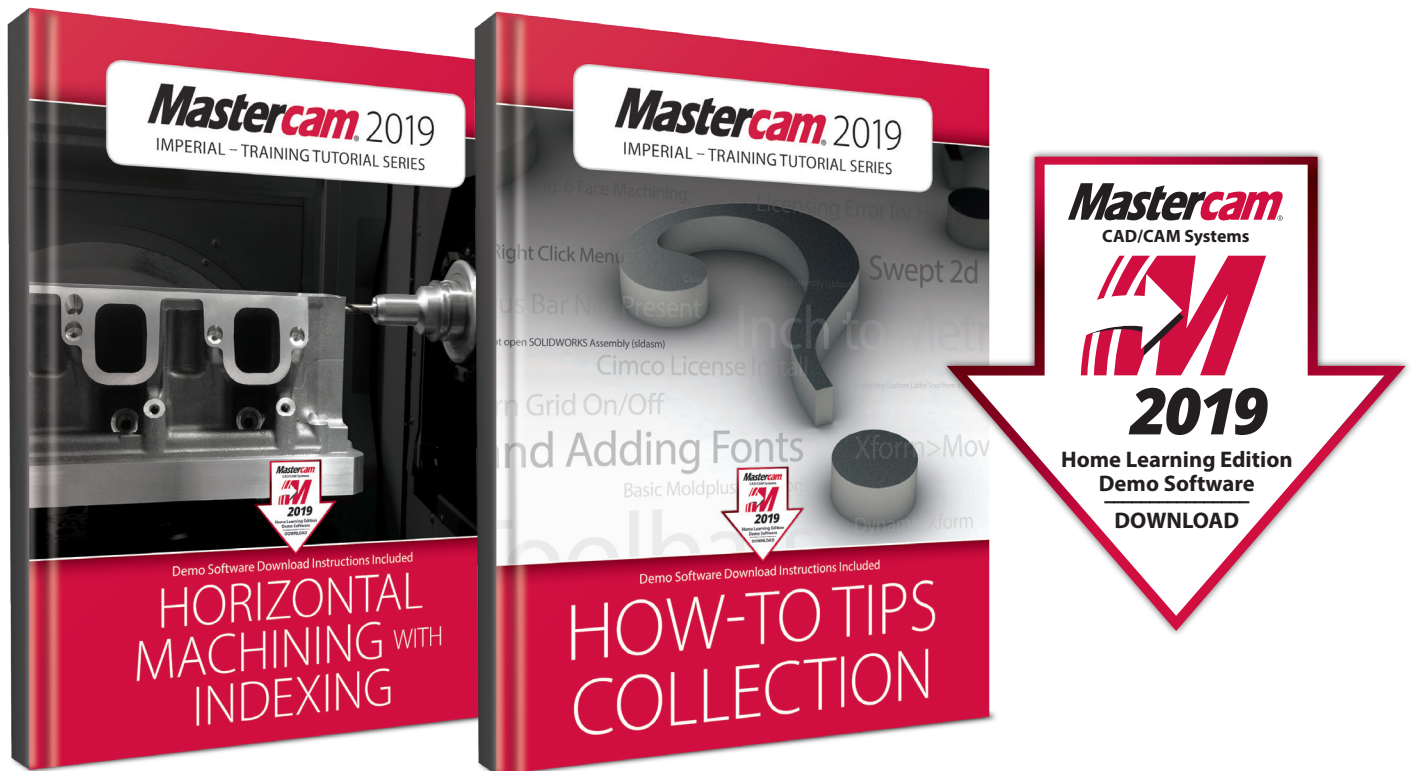
## We speak your machine's language.

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)



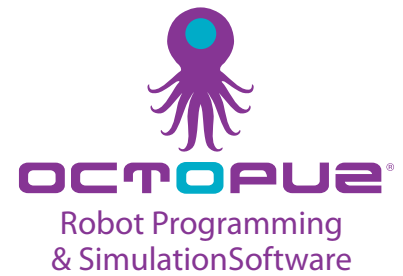
# DOES YOUR SCHOOL HAVE ROBOTS? YOU SHOULD CHECK OUT OCTOPUZ.

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.



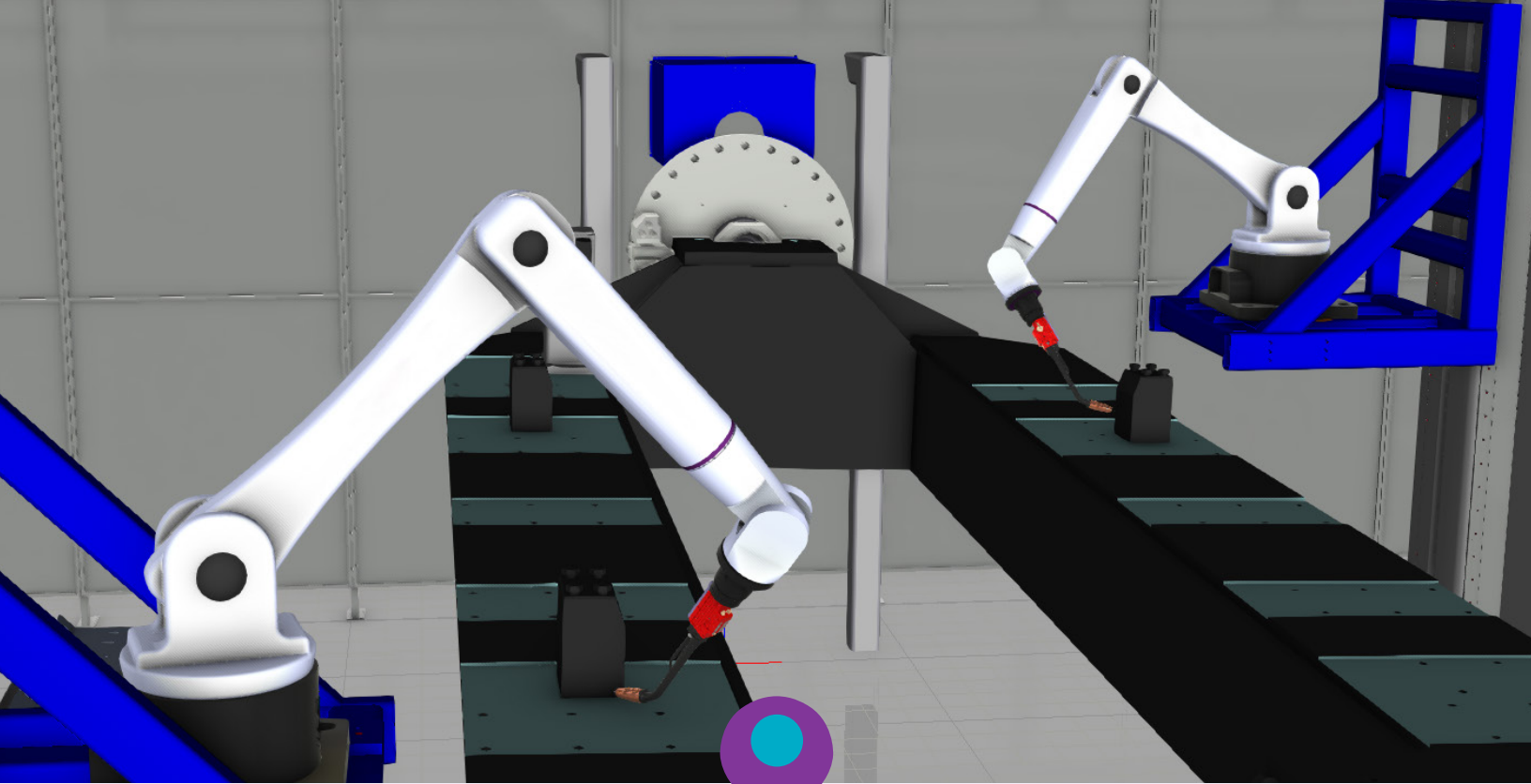
Protect your investment!  
OCTOPUZ can detect  
collisions offline.



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)