

Mastercam® 2020

IMPERIAL | TRAINING TUTORIAL SERIES

HLE
DOWNLOAD
INSTRUCTIONS INSIDE

M
2020



BEGINNER

Mastercam[®] 2020

BEGINNER TRAINING TUTORIAL

To order more books:

Call 1-800-529-5517 or

Visit www.emastercam.com or

Contact your Mastercam dealer



HOW TO DOWNLOAD:

Mastercam Home Learning Edition (HLE)
Demo Software

Visit [**www.eMastercam.com/files**](http://www.eMastercam.com/files)

Click on Mastercam Demo Software
from the categories on the right.

Proceed to download!

Mastercam 2020 Beginner Training Tutorial

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

Software: Mastercam 2020

Authors: Mariana Lendel

ISBN: 978-1-77146-837-4

Date: July 4, 2019

Notice

In-House Solutions Inc. reserves the right to make improvements to this manual at any time and without notice.

Disclaimer Of All Warranties And Liability

In-House Solutions Inc. makes no warranties, either express or implied, with respect to this manual or with respect to the software described in this manual, its quality, performance, merchantability, or fitness for any particular purpose. In-House Solutions Inc. manual is sold or licensed "as is." The entire risk as to its quality and performance is with the buyer. Should the manual prove defective following its purchase, the buyer (and not In-House Solutions Inc., its 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 Contents

Mill Essentials Training Tutorials	17
Getting Started	19
Objectives	20
Step 1: Starting Mastercam	20
Step 2: GUI - Graphical User Interface	21
Step 3: Navigate Through Mastercam	22
Step 4: Set The Attributes	24
Step 5: Manager Panels	25
Step 6: Setting Mastercam To Imperial	27
Step 7: Set The Grid	29
Conventions Used In This Book:	30
Mastercam® Work Flow	31
Tutorial 1: Geometry Creation	35
Tutorial #1 Drawing	37
Step 1: Setting Up The Graphical User Interface	38
Step 2: Create One Rectangle	39
Step 3: Create The 1/4" Diameter Circles	42
Step 4: Create The 2.0" And 0.75" Diameter Circles	45
Step 5: Create The Chamfers	48
Step 6: Save The File	50
Tutorial #1 Review Exercise	51
Create The Geometry For Tutorial #1 Exercise	52
Tutorial 1: Toolpath Creation	55
Suggested Fixture	57
Setup Sheet	58
Step 1: Select The Machine And Set Up The Stock	59

Step 2: Face The Part	64
Step 3: Circle Mill The Large Hole	71
Step 4: Backplot The Toolpaths	78
Step 5: Simulate The Toolpath In Verify	80
Step 6: Circle Mill The Inside Hole	81
Step 7: Spot Drill The 0.25" Holes	90
Step 8: Drill The 0.25" Holes	100
Step 9: Chamfer The Large Hole	106
Step 10: Machine The Chamfers At The Corners Using Contour Toolpath	115
Step 11: Post The File	122
Step 12: Save The Updated MCAM File	123
Create The Toolpaths For Tutorial #1 Exercise	124
Tutorial 2: Geometry Creation	127
Tutorial #2 Drawing	129
Step 1: Setting Up The Graphical User Interface	130
Step 2: Create A Rectangle	131
Step 3: Create Two Obround Shapes	134
Step 4: Create A Circle	138
Step 5: Use Trim Divide To Clean The Circle	140
Step 6: Create Parallel Lines	143
Step 7: Use Trim Divide Delete To Clean Up The Geometry	146
Step 8: Create Angular Lines	148
Step 9: Create Polygon	151
Step 10: Create Fillets	153
Step 11: Rotate The Part	158
Step 12: Save The File	160
Tutorial #2 Review Exercise	161
Create The Geometry For Tutorial #2 Exercise	162

Tutorial 2: Toolpath Creation	165
Toolpath Creation - Setup #1	167
Suggested Fixture	167
Setup Sheet	167
Step 1: Select The Machine And Set Up The Stock	168
Step 2: Slot Milling	173
Step 3: Backplot The Toolpaths	183
Step 4: Simulate The Toolpath In Verify	185
Step 5: Machine The Cutout Pockets	186
Step 6: Finish The Inside Shapes Using 2D HS Dynamic Contour	194
Step 7: Rough The Outside Using High Speed Dynamic Mill	200
Step 8: Finish The Outside Profile Using Contour Toolpath	211
Suggested Fixture:	217
Setup Sheet:	217
Step 9: Creating And Renaming Toolpath Groups	218
Step 10: Create And Set WCS To Bottom	220
Step 11: Face The Part	226
Step 12: Rename The NC File	232
Step 13: Post The File	233
Step 14: Save The Updated MCAM File	235
Create The Toolpaths For Tutorial #2 Exercise	236
Tutorial 3: Geometry Creation	239
Tutorial #3 Drawing	241
Step 1: Setting Up The Graphical User Interface	242
Step 2: Create A Rectangle	242
Step 3: Create Arcs Knowing Endpoints	244
Step 4: Delete The Rectangle	248

Step 5: Mirror The Geometry	249
Step 6: Create A Fillet	252
Step 7: Create The Ellipses	254
Step 8: Offset The Ellipse	257
Step 9: Create Lines Knowing The Endpoints	259
Step 10: Add The Text	261
Step 11: Break At Intersection	267
Step 12: Translate	268
Step 13: Change The Main Level To 2	274
Create The Solid Body	274
Step 14: Extrude The Base Geometry	274
Step 15: Extrude Cut The Body	278
Step 16: Add The Ellipse As A Boss Shape	280
Step 17: Extrude Cut The Ellipse	282
Step 18: Create The Bottom Cylinder	284
Step 19: Save The File	288
Tutorial #3 Review Exercise	289
Create The Geometry For Tutorial #3 Exercise	290
Tutorial 3: Toolpath Creation	293
Suggested Fixture	295
Setup Sheet	295
Step 1: Select The Machine And Set Up The Stock	296
Step 2: 2D High Speed Area Mill	299
Step 3: Backplot The Toolpaths	309
Step 4: Simulate The Toolpath In Verify	311
Step 5: Pocket With Islands	312
Step 6: Pocket Remachining	321
Step 7: 2D HS Dynamic Mill - Outside Profile	325

Step 8: 2D HS Dynamic Mill - Open Pockets	332
Step 9: Post The File	340
Step 10: Save The Updated MCAM File	341
Create The Toolpaths For Tutorial #3 Exercise	342
Tutorial 4: Geometry Creation	345
Tutorial #4 Drawing	347
Step 1: Setting Up The Graphical User Interface	348
Step 2: Create Two Arcs	348
Step 3: Create Vertical Lines	357
Step 4: Create Arc Using Arc Polar Endpoints	359
Step 5: Rotate Geometry	361
Step 6: Mirror Geometry	363
Step 7: Create An Arc Tangent	365
Step 8: Trim Geometry	367
Step 9: Mirror Geometry To Complete Arms	370
Step 10: Create A Construction Line	373
Step 11: Create A 0.5" Diameter Circle	375
Step 12: Delete Construction Geometry	377
Step 13: Create Tangent Lines	378
Step 14: Create Arc Polar	383
Step 15: Create Fillets	385
Step 16: Trim The Arc	387
Step 17: Rotate	389
Step 18: Translate	391
Step 19: Change The Main Level To 2	396
Step 20: Create The Solid Body By Extruding A Closed Chain	396
Step 21: Extrude Cut The Part Pockets And The Holes	404

Step 22: Chamfer The Part	412
Step 23: Save The File	417
Tutorial #4 Review Exercise	418
Create The Geometry For Tutorial #4 Exercise	419
Create The Solid Geometry For Tutorial #4 Exercise	419
Tutorial 4: Toolpath Creation	421
Suggested Fixture	423
Setup Sheet	424
Step 1: Select The Machine And Set Up The Stock	425
Step 2: 2D High Speed Dynamic Mill	429
Step 3: Backplot The Toolpaths	441
Step 4: Simulate The Toolpath In Verify	442
Step 5: Finish The Walls Using Contour Toolpath	444
Step 6: Area Mill Toolpath	451
Step 7: Transform-Rotate Toolpath	462
Step 8: Finish The Pocket Walls Using Contour Toolpath	466
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:	495
Setup Sheet 2:	495
Step 13: Creating And Renaming Toolpath Groups	496
Step 14: Set The WCS To Bottom	498
Step 15: 2D HS Dynamic Mill	501
Step 16: Finish The Pocket Wall Using Contour Toolpath	510
Step 17: Rename The NC File	516
Step 18: Post The File	517

Step 19: Save The Updated MCAM File	518
Create The Toolpaths For Tutorial #4 Exercise	519
Lathe Training Tutorials	525
Getting Started	527
Objectives	528
Step 1: Starting Mastercam	528
Step 2: GUI - Graphical User Interface	529
Step 3: Navigate Through Mastercam	530
Step 4: Set The Attributes	532
Step 5: Manager Panels	533
Step 6: Setting Mastercam To Imperial	535
Step 7: Set The Grid	537
Conventions Used In This Book:	538
Step 8: Step Titles	538
Mastercam® Work Flow	539
Tutorial 1: Geometry Creation	541
Overview Of Steps Taken To Create The Part Geometry	542
Tutorial #1 Drawing	543
Step 1: Setting Up The Graphical User Interface	544
Step 2: Create A Rectangle	546
Step 3: Create The Parallel Lines	549
Step 4: Create Line Endpoint	557
Step 5: Create The FilletS	558
Step 6: Trim The Geometry	561
Step 7: Save The File	564
Tutorial #1 Review Exercise	565
Create The Geometry For Tutorial #1 Exercise	566

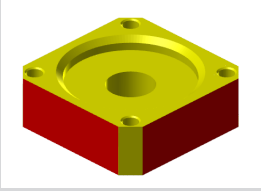
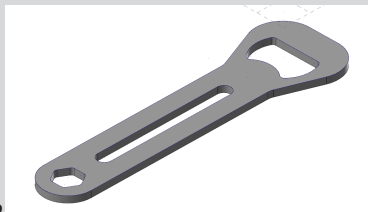
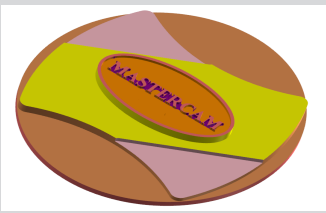
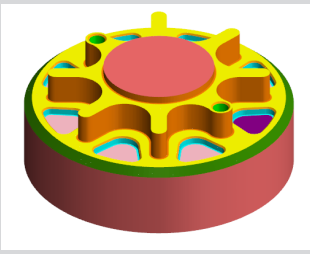
Tutorial #1 Geometry Creation Quiz	568
Tutorial 1: Toolpath Creation	569
Overview Of Steps Taken To Create The Final Part:	570
Part Setup:	571
Setup Sheet:	571
Step 1: Select The Machine And Set Up The Stock	572
Step 2: Face The Part	579
Step 3: Backplot The Toolpaths	581
Step 4: Simulate The Toolpath In Verify	583
Step 5: Rough Out The Part	585
Step 6: Finish The Part	592
Step 7: Post The File	597
Step 8: Save The Updated MCAM File	598
Create The Toolpaths For Tutorial #1 Exercise	599
Tutorial #1 Toolpath Creation Quiz	602
Tutorial 2: Geometry Creation	603
Overview Of Steps Taken To Create The Part Geometry	604
Tutorial #2 Drawing	605
Step 1: Setting Up The Graphical User Interface	606
Step 2: Select The Lathe Default	606
Step 3: Set Up The Lathe Plane +D +Z	606
Step 4: Create A Rectangle	608
Step 5: Create Parallel Lines	610
Step 6: Create The Fillets	615
Step 7: Trim The Geometry	618
Step 8: Create Additional Lines	620
Step 9: Trim The Geometry	625
Step 10: Save The File	628

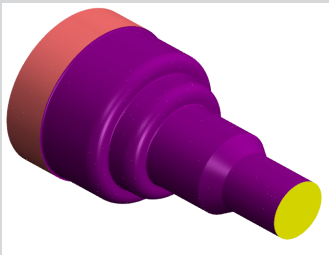
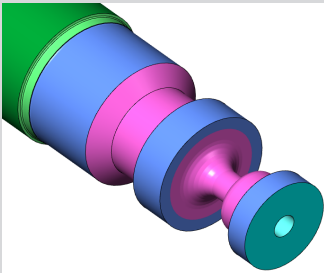
Tutorial #2 Review Exercise	629
Tutorial #2 Geometry Creation Quiz	632
Tutorial 2: Toolpath Creation	633
Overview Of Steps Taken To Create The Final Part:	634
Part Setup:	635
Setup Sheet:	635
Step 1: Select The Tool Settings And Set Up The Stock	636
Step 2: Face The Part	642
Step 3: Backplot The Toolpath	644
Step 4: Simulate The Toolpath In Verify	646
Step 5: Rough The Part	648
Step 6: Finish The Part	653
Step 7: Groove The Part Using The Multiple ChainS Method	659
Step 8: Center Drill The Part	667
Step 9: Drill The Part	670
Step 10: Run The Post Processor To Obtain The G-code File	674
Step 11: Save The Updated MCAM File	675
Create The Toolpaths For Tutorial #2 Exercise	677
Tutorial #2 Toolpath Creation Quiz	680
Solids Training Tutorials	681
Tutorial 1	683
Overview Of Steps Taken To Create The Part Geometry	684
Tutorial #1 Drawing	685
Step 1: Setting Up The Graphical User Interface	686
Step 2: Set The Construction Plane And Change Wireframe Color	687
Step 3: Create Rectangles Given The Size And Anchor Point	688
Step 4: Create The Parallel Lines	691

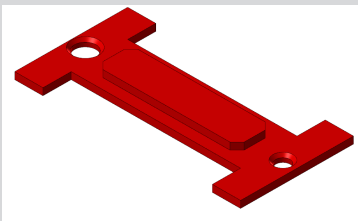
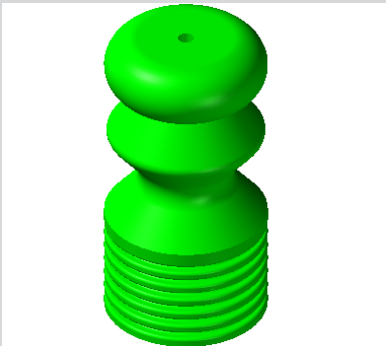
Step 5: Trim The Geometry	695
Step 6: Remove The Leftover Lines Using Divide Command	697
Step 7: Chamfer The Inside Rectangle	699
Step 8: Save The File	702
Step 9: Change The Solid Color To Red	703
Step 10: Change The Main Level To 2	703
Step 11: Create The Solid Body	704
Step 12: Create Two Holes	708
Step 13: Using Solids Manager To Modify The Holes	713
Step 14: Add A Boss	717
Step 15: Chamfer The Top Of The Boss	719
Step 16: Save The File	722
Tutorial #1 Review Exercise	723
Create The Geometry For Tutorial #1 Exercise	724
Tutorial 2	729
Overview Of Steps Taken To Create The Part Geometry	730
Tutorial #2 Drawing	731
Step 1: Setting Up The Graphical User Interface	732
Step 2: Set The Construction Plane And Change Wireframe Color	732
Step 3: Create Rectangles Given The Size And Anchor Point	733
Step 4: Create The Parallel Lines	735
Step 5: Create Tangent Arcs	741
Step 6: Create Lines Knowing Their Endpoints	744
Step 7: Trim The Geometry	746
Step 8: Translate Geometry	749
Step 9: Clean The Geometry Using Divide	751
Step 10: Fillet The Part	753
Step 11: Create The Circles In The Top Plane	759

Step 12: Save The File	763
Step 13: Change The Solid Color To Red	763
Step 14: Change The Main Level To 2	763
Step 15: Create The Solid Body By Revolving A Closed Chain	764
Step 16: Extrude Cut The 0.25" Circle	766
Step 17: Extrude Cut The 1.0" Hole	770
Step 18: Extrude Cut The 1.25" Hole	773
Step 19: Save The File	775
Tutorial #2 Review Exercise	776
Create The Geometry For Tutorial #2 Exercise	777
Quiz Answers	781
Mill Essentials Tutorial Quiz Answers	782
Lathe Tutorial Quiz Answers	785
Solids Tutorial Quiz Answers	787

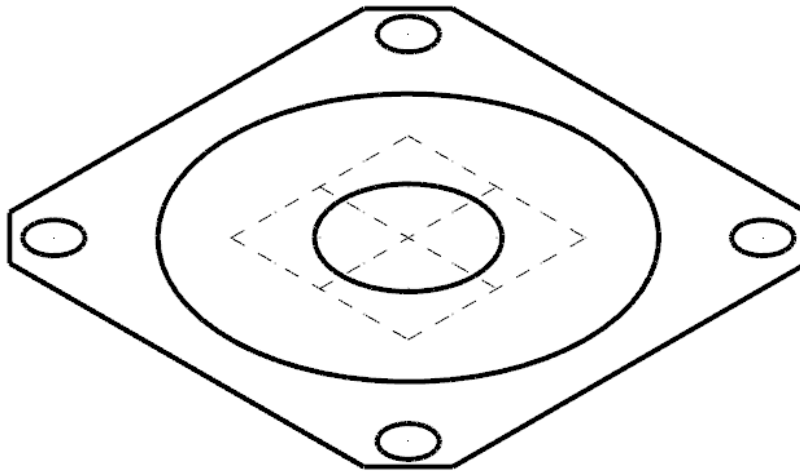
Beginner Projects

Mill Tutorials	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. Fillet. 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>

Lathe Tutorials	Geometry Functions	Toolpath Creation
<p>#5</p> 	<p>Rectangle. Line Parallel. Chamfer Entities. Fillet Entities. Trim Entities.</p>	<p>Face. Roughing. Finish.</p>
<p>#6</p> 	<p>Line Endpoints (Polar Line). Line Parallel. Line Endpoints (Horizontal). Divide. Trim 2 Entities. Fillet.</p>	<p>Face. Roughing. Finish Groove - Multiple Chains. Drilling.</p>

Solids Tutorials	Geometry Functions
<p>#7</p> 	<p>Create Rectangle. Chamfer Outside Profile. Solid Extrude Create Body. Solid Extrude Add Boss. Solid Hole. Constant Radius Fillet. One Distance Chamfer.</p>
<p>#8</p> 	<p>Create Geometry in Front Plane. Create Rectangle. Create Parallel Lines. Create Tangent Arcs. Create Lines. Translate Geometry. Create Fillets. Solid Revolve Create Body. Solid Extrude Cut Body</p>

Tutorial 1: Geometry Creation



OVERVIEW OF STEPS TAKEN TO CREATE THE PART GEOMETRY:

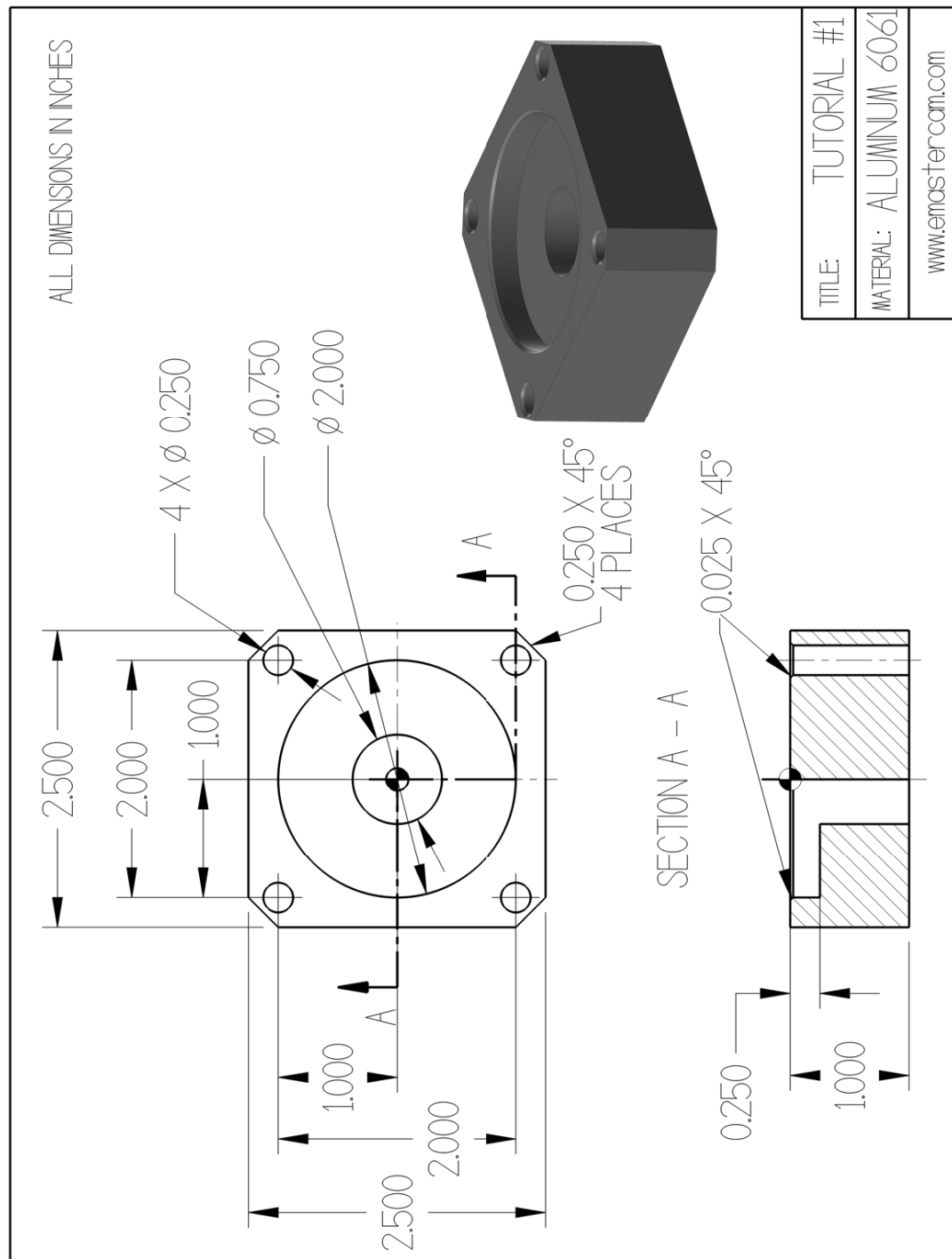
From Drawing to CAD Model:

- ◆ 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.

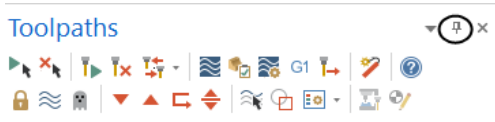
TUTORIAL #1 DRAWING



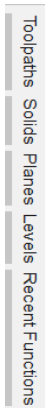
STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

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.

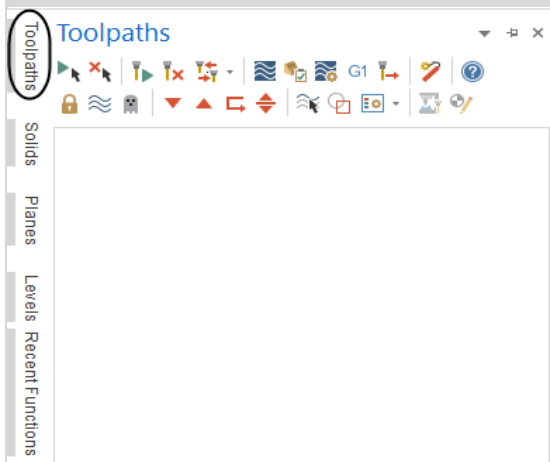
- ◆ Use **Auto Hide** icon to hide all **Manager** panels.



- ◆ 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.

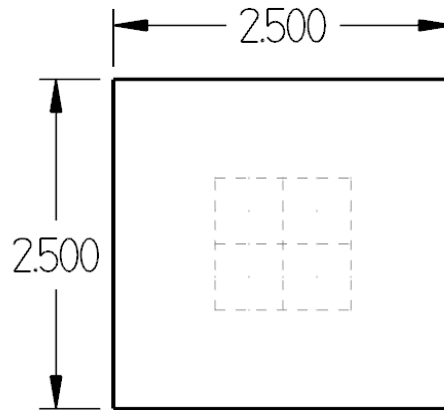


While creating the geometry, keep the Manager panels hidden. This ensures more space in the graphics window for the geometry.

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 2.5" by 2.5" rectangle with the center anchor in the Origin.

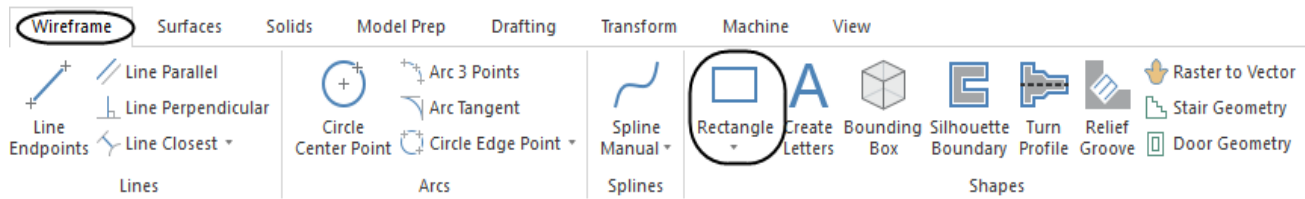
Step Preview:



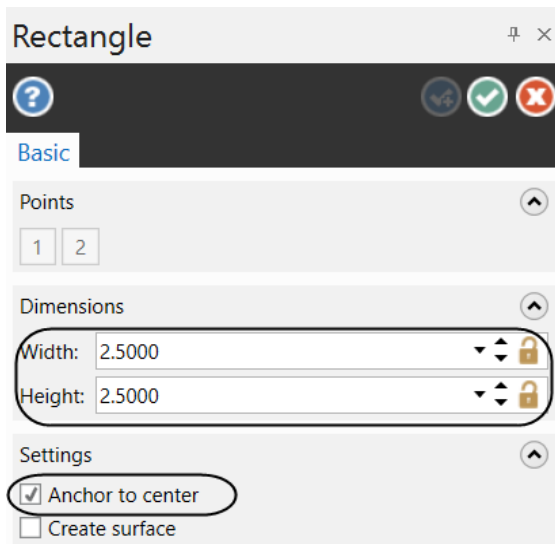
2.1 Create a 2.5" by 2.5" Rectangle

Wireframe

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

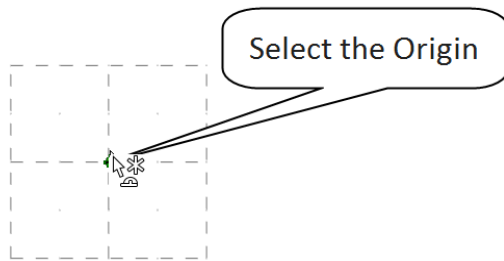


- ◆ In the **Rectangle** panel, enter the **Width** and **Height** and enable **Anchor to center** as shown.

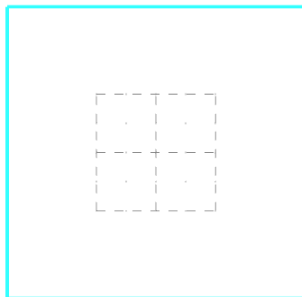


*Note: Make sure that **Create surface** is not selected. **Anchor to center** sets the base point of the rectangle to its center and draws the rectangle outward from the center. **Create surface** creates a surface inside of the rectangle. Surface creation and Surface toolpath are covered in Mill Advanced.*

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



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

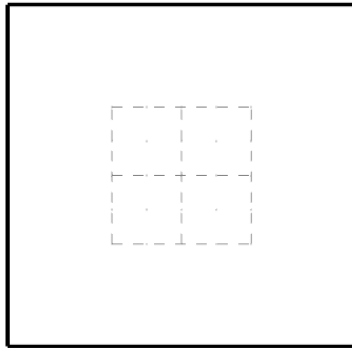




Note: The geometry should appear in cyan blue color which is the color for the live entities. While the rectangle is live, you can adjust the dimensions or select a new base point.

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



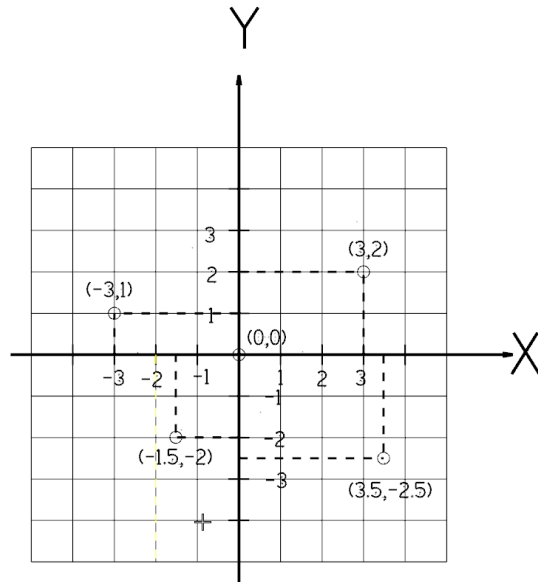
- ◆ The geometry should look as shown.



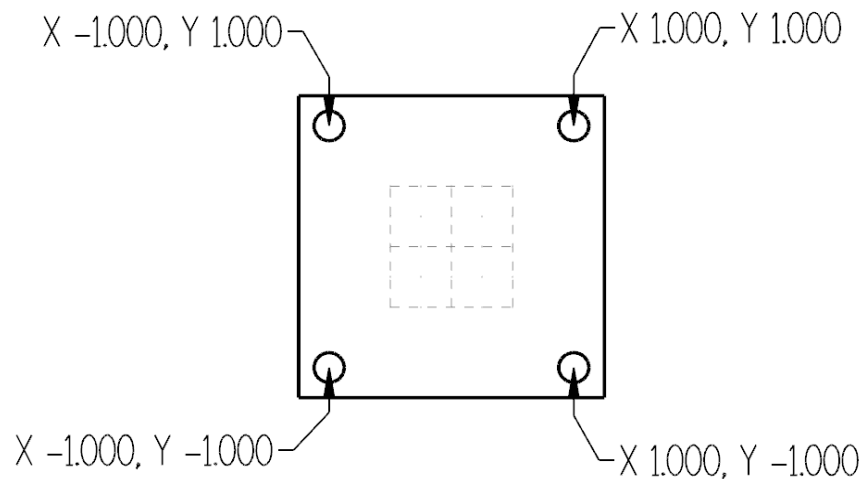
*Note: While creating geometry for this tutorial, if you make a mistake, you can undo the last step using the **Undo** icon.  You can undo as many steps as needed. If you delete or undo a step by mistake, just use the **Redo** icon.  To delete unwanted geometry, select the geometry first and then press **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 1/4" 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.

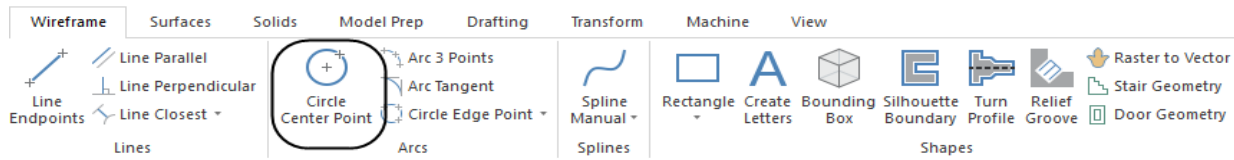


Step Preview:

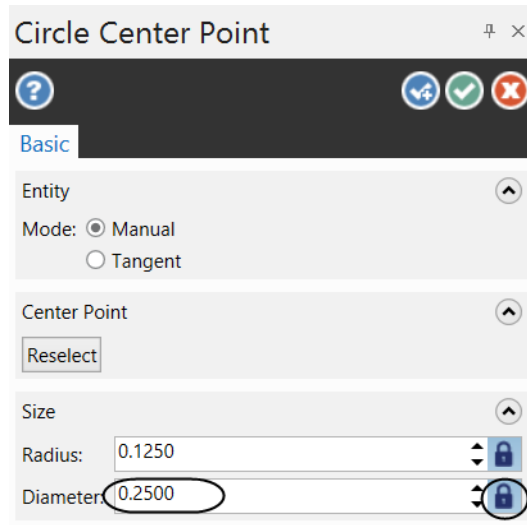


Wireframe

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



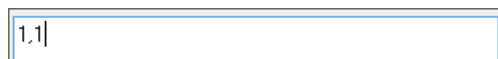
- ◆ Enter a **Diameter** of **0.25** in the panel as shown.
- ◆ To create all four circles, click on the locker icon to lock the value.



- ◆ [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.

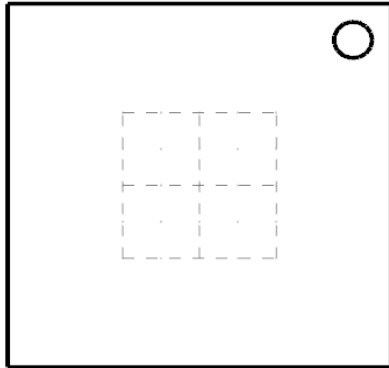


- ◆ Type **1, 1** as shown.

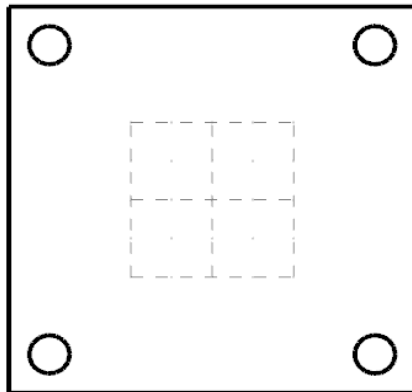


*Note: When entering the coordinates for the center point, the first value is the **X** coordinate value, then the **Y** value followed by the **Z** value only if it is different from zero. The coordinate values are separated with commas. You do not need to use the coordinate labels if you enter the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

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



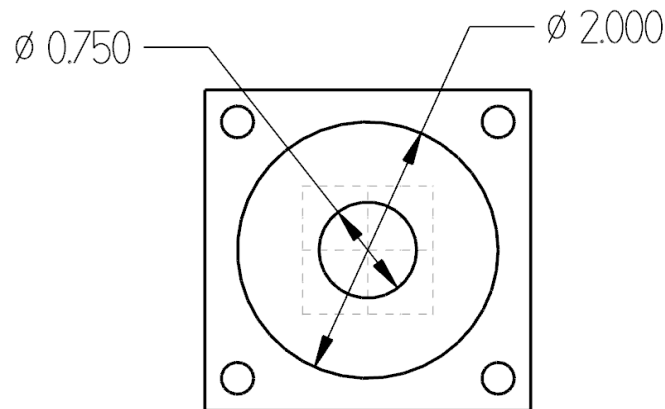
- ◆ [Enter the center point]: Select the **AutoCursor Fast Point** icon again and enter **1, -1**.
- ◆ Press **Enter** to place the circle.
- ◆ [Enter the center point]: Select the **AutoCursor Fast Point** icon again and enter **-1, 1**.
- ◆ Press **Enter** to place the circle.
- ◆ [Enter the center point]: Select the **AutoCursor Fast Point** icon again and enter **-1, -1**.
- ◆ 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 2.0" AND 0.75" DIAMETER CIRCLES

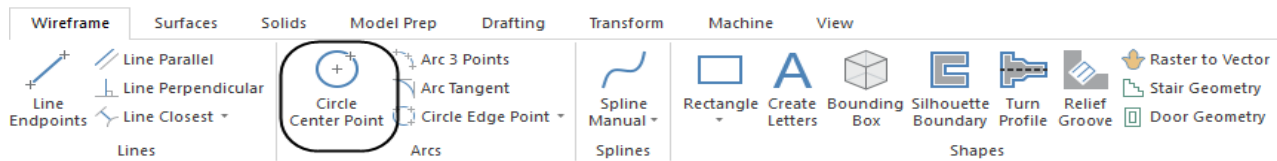
In this step, you will use the same Circle Center Point to create circles that you know the diameters and the locations.

Step Preview:

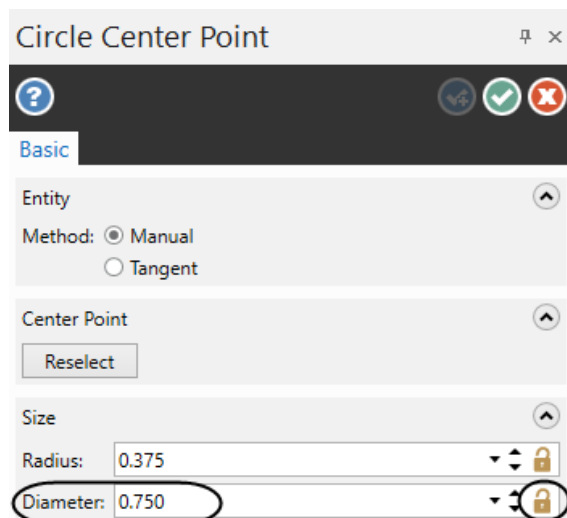


Wireframe

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



- ◆ Enter the **Diameter 0.75** in the panel and disable the locker icon as shown.

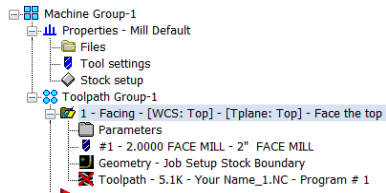


- ◆ Press **Enter** to see the circle preview.

- ◆ Select the **OK** button to exit the **Facing Parameters**.



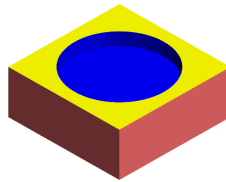
*Note: If you exit the toolpath in the middle of setting the parameters, in the **Toolpaths Manager**, you will have a red **X** on the **Face Toolpath** as shown in. This shows that you modified the toolpath and you need to update it. You will have to select the **Regenerate all dirty operations** icon each time you change something in the toolpath parameters.*



STEP 3: CIRCLE MILL THE LARGE HOLE

Circle Mill Toolpaths remove circular pockets based on a single point. You can select either point entities or center points of arcs. Mastercam will then pocket out a circular area of the diameter to the depth that you specify.

Toolpath Preview:



3.1 Drill Point Selection

- ◆ Press **Alt + T** to remove the toolpath display.

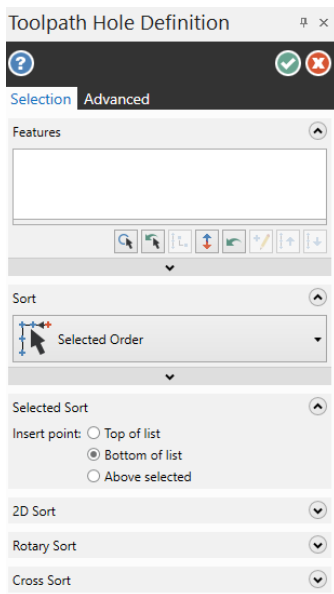
Toolpaths

- ◆ From the **2D** group, click on the drop down arrow until the **Circle Mill** toolpath appears as shown.
- ◆ Click on the **Circle Mill** icon.

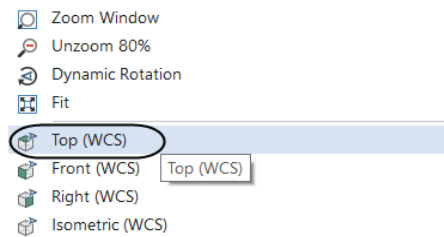


*Note: Move the cursor to the center of the graphics window to see the **Toolpath Hole Definition** panel that appears below the **Toolpaths Manager**.*

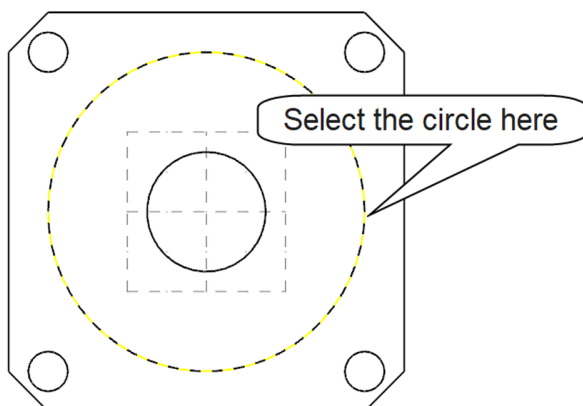
- ◆ The **Toolpath Hole Definition** should appear as shown.



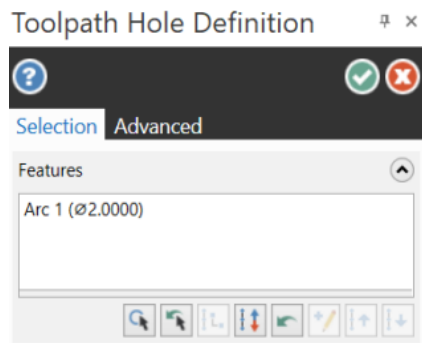
- ◆ Right click and select the **Top** view.



- ◆ [Select one or more entities to add or remove from the features list]: Select the center of the **2.0"** diameter circle as shown.



- ◆ The Point will be displayed in the **Features** list as shown.

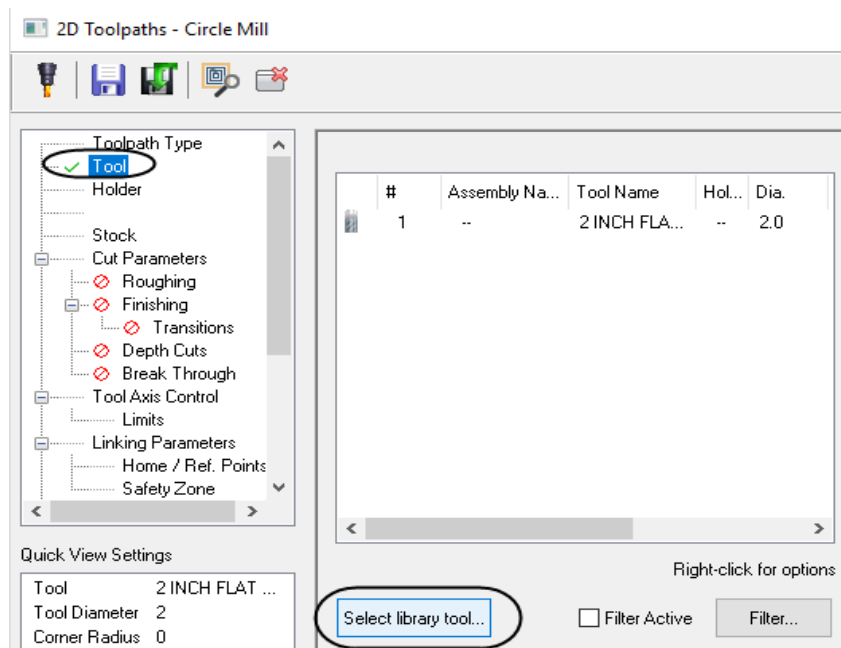


- ◆ Select the **OK** button to finish the selection and exit **Toolpath Hole Definition**.
- ◆ In the **Toolpath Type** page, the **Circle Mill** icon will be selected.

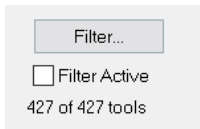


3.2 Select a 3/4" Flat Endmill from the library and set the Tool parameters

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



- ◆ To be able to see all the tools from the library, disable **Filter Active**.



- ◆ Scroll down and select the **3/4" Flat Endmill (#294)** as shown.

289	—	7/16 FLAT ENDMILL	—	0.4375	0.0	0.8	4	End...	None
290	—	1/2 FLAT ENDMILL	—	0.5	0.0	1.0	4	End...	None
291	—	17/32 FLAT ENDMILL	—	0.5312	0.0	1.0	4	End...	None
292	—	5/8 FLAT ENDMILL	—	0.625	0.0	1.5	4	End...	None
293	—	23/32 FLAT ENDMILL	—	0.71...	0.0	1.5	4	End...	None
294	—	3/4 FLAT ENDMILL	—	0.75	0.0	2.0	4	End...	None
295	—	13/16 FLAT ENDMILL	—	0.8125	0.0	2.0	4	End...	None
296	—	7/8 FLAT ENDMILL	—	0.875	0.0	2.0	4	End...	None
297	—	1 INCH FLAT ENDMILL	—	1.0	0.0	2.0	4	End...	None
298	—	1-3/16 FLAT ENDMILL	—	1.1875	0.0	2.0	4	End...	None

- ◆ Select the tool in the **Tool Selection** page and then select the **OK** button to exit. 

- ◆ Input a comment and make all the necessary changes, as shown.

A screenshot of the 'Tool Selection' dialog box. On the left, a table lists tools, with tool #2 (3/4 FLAT E...) selected. On the right, various parameters are configured:

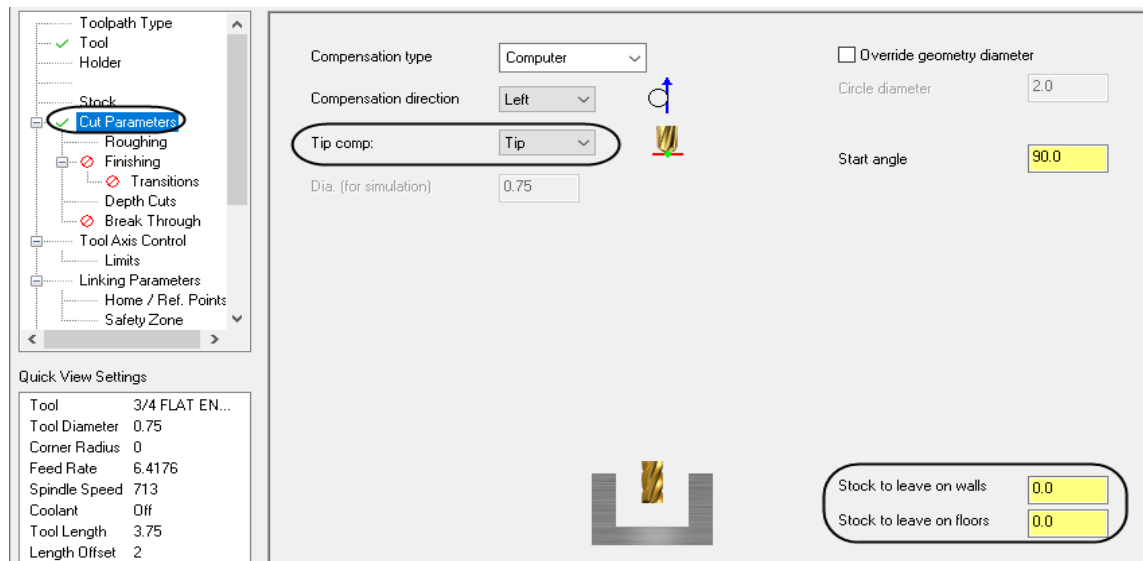
- Tool diameter: 0.75
- Corner radius: 0.0
- Tool name: 3/4 FLAT ENDMILL
- Tool #: 2
- Length offset: 2
- Head #: 0
- Diameter offset: 2
- ☐ RCTF
- Spindle direction: CW
- Feed rate: 6.4176
- Spindle speed: 713
- FPT: 0.0023
- SFM: 139.9869
- Plunge rate: 6.4176
- Retract rate: 6.4176
- ☐ Force tool change
- ☒ Rapid Retract
- Comment: Circle Mill the largest hole
- ☐ To batch

Buttons at the bottom include 'Select library tool...', 'Filter Active' (unchecked), and 'Filter...'.

*Note: The **Feed rate**, **Plunge rate**, **Retract rate**, and **Spindle speed** are based on the tool definition as set in the **Tool Settings**. You may change these values as per your part material and tools.*

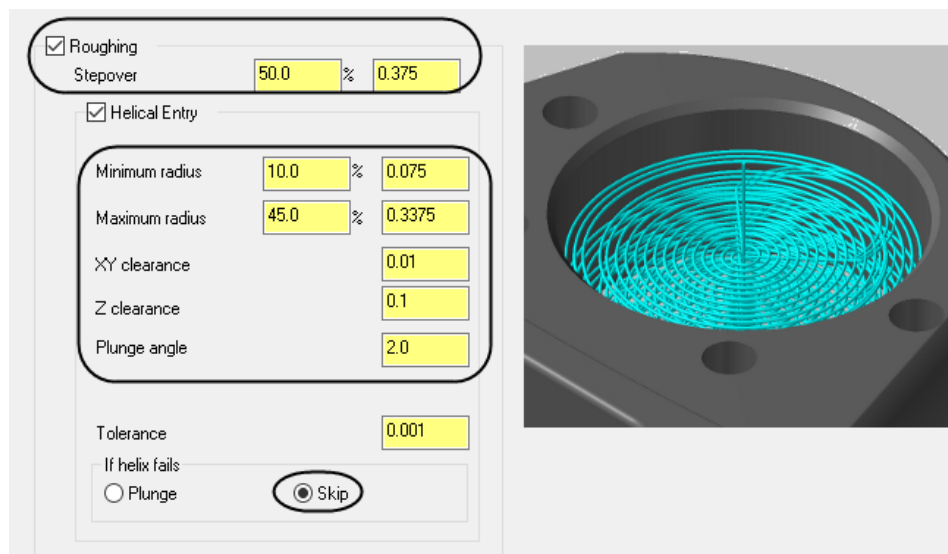
3.3 Cut Parameters

- ◆ From the **Tree View** list, select **Cut Parameters** and ensure the settings appear as shown.



3.4 Roughing

- ◆ From the **Tree View** list, select **Roughing** and enable it. Set the **Stepover** to **50%**, enable **Helical Entry**, and specify the other parameters as shown.



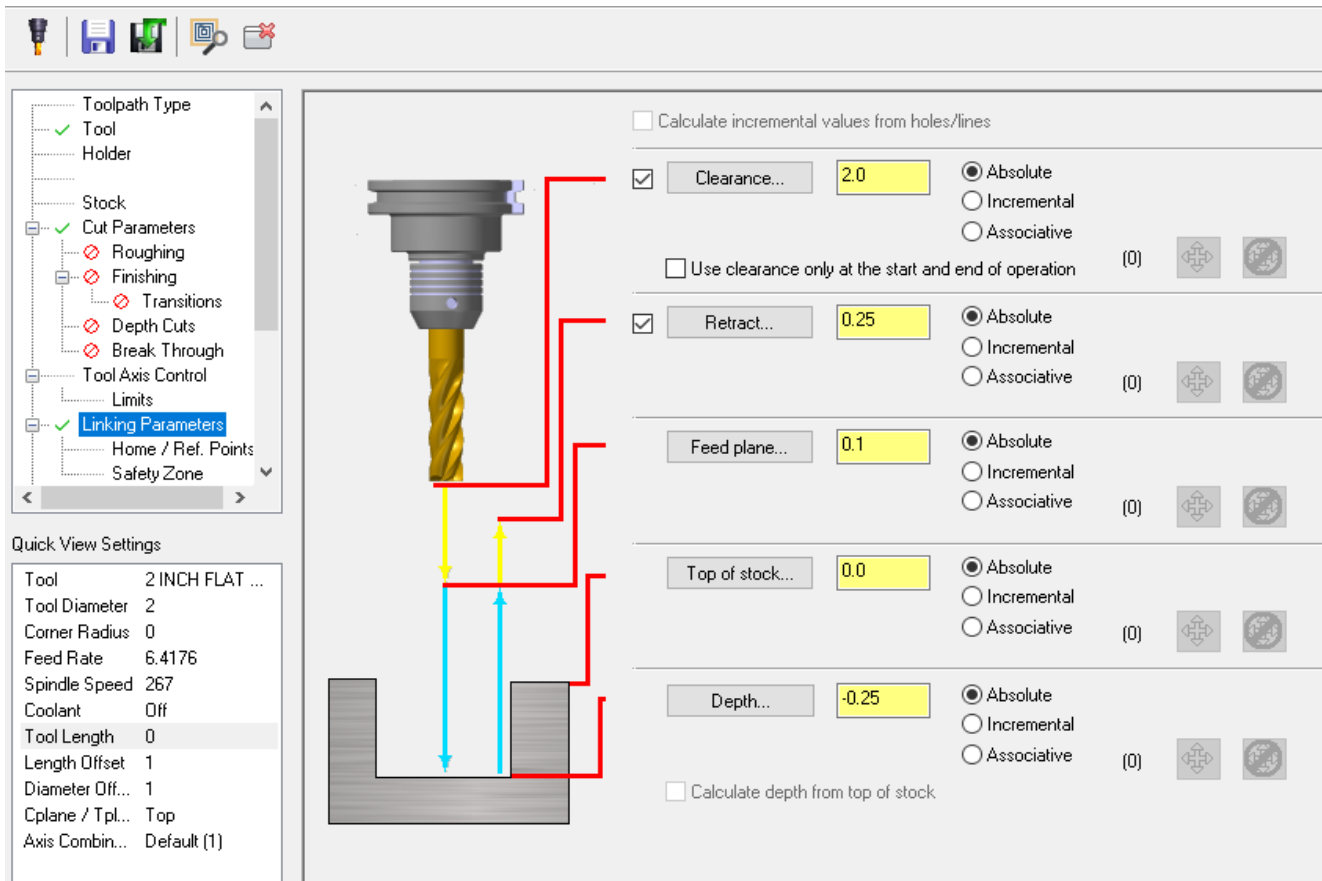
Stepover sets the distance between cutting passes in the **X** and **Y** axes as a percentage of the tool diameter.

Helical Entry creates a helix at the center of the circle to begin the roughing motion. If this option is turned off, the tool plunges to start the toolpath.

Note: The images in the toolpaths change depending on the parameter that you last selected in the page.

3.5 Linking Parameters

- ◆ Select **Linking Parameters** from the **Tree View** list.
- ◆ Change the **Top of stock** to **0.0** and set the **Depth** to **-0.25**. Ensure all the values are set the same as shown.



Absolute values are always measured from the origin 0,0,0.

Incremental values are relative to other parameters or chained geometry.

Associative option allows you to select points from the existing geometry from where the values will be measured.

3.6 Preview the Toolpath

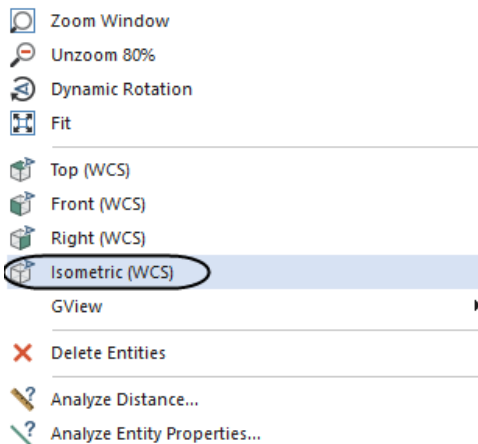
- ◆ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



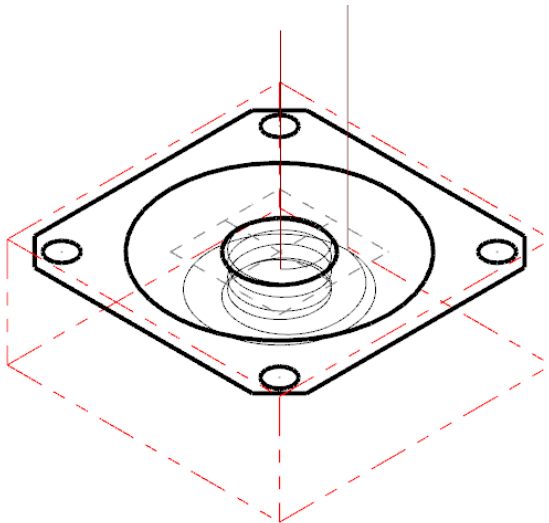
- ◆ To hide the dialog box, click on the **Hide dialog** icon as shown.



- ◆ To see the part from an Isometric view, right mouse click in the graphics window and select Isometric as shown.



- ◆ The toolpath should look as shown.



- ◆ Press **Esc** key to exit the preview.

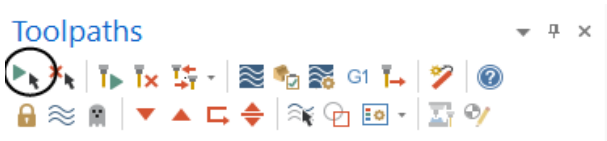
Note: If the toolpath does not look as shown in the preview, check your parameters again.

- ◆ Select the **OK** button to exit the **2D Toolpaths - Circle Mill** parameters.

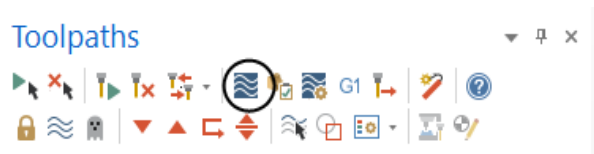
STEP 4: BACKPLOT THE TOOLPATHS

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.

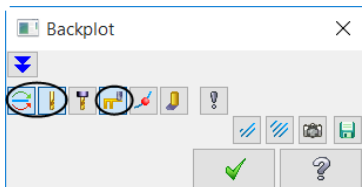
- ◆ Make sure that the toolpaths are selected (signified by the green check mark on the folder icon). If both operations are not selected, choose the **Select all operations** icon.



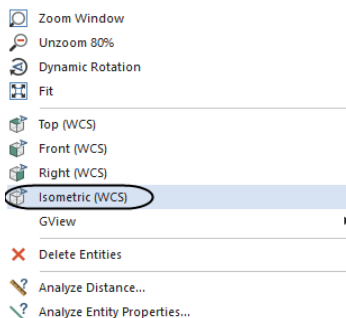
- ◆ Select the **Backplot selected operations** button.



- ◆ In the **Backplot** panel, enable **Display with color codes**, **Display tool** and **Display rapid moves** icons as shown.



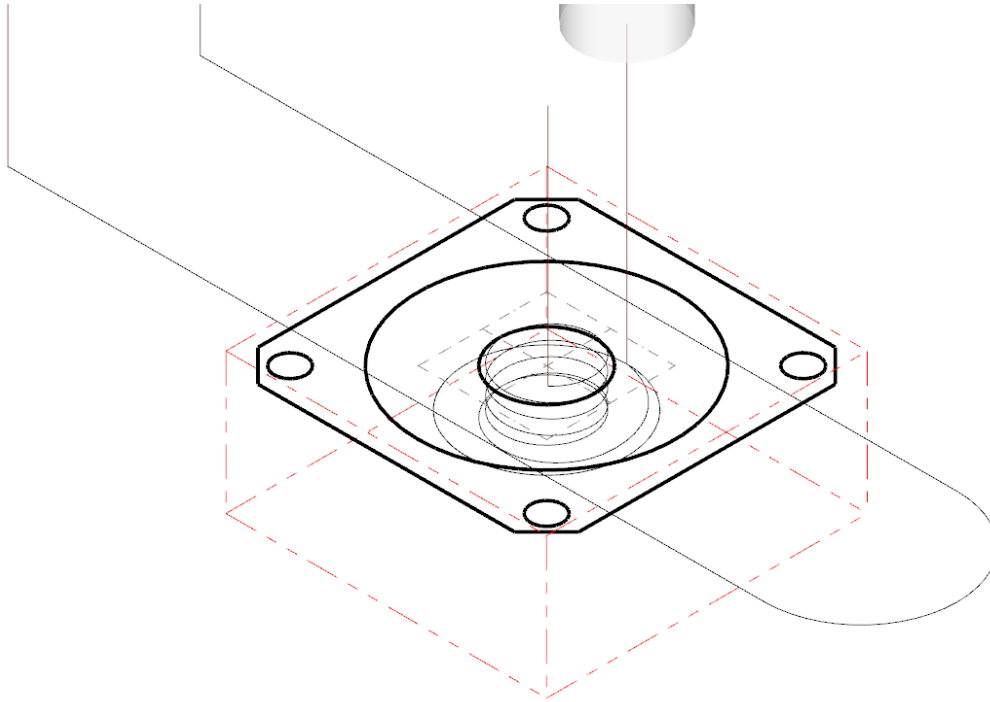
- ◆ To see the part from an **Isometric** view, right mouse click in the graphics window and select Isometric as shown.




- ◆ To fit the workpiece to the screen, if needed, right mouse click in the graphics window again and select the Fit.
- ◆ You can step through the **Backplot** by using the **Step forward** (right arrow) or **Step back** (left arrow) buttons.
- ◆ You can adjust the speed of the backplot.
- ◆ Select the **Play** button to run **Backplot**.



- ◆ The toolpath should look as shown.



- ◆ Select the **OK** button to exit **Backplot**. 

STEP 5: SIMULATE THE TOOLPATH IN VERIFY

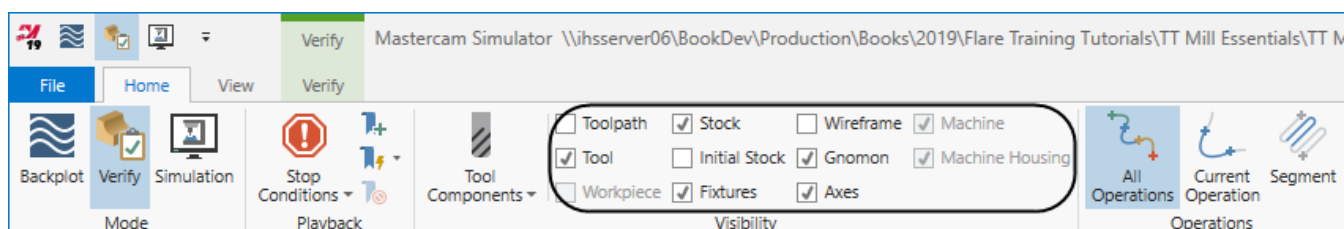
Verify shows the path the tools take to cut the part with material removal. This display lets you spot errors in the program before you machine the part. As you verify toolpaths, Mastercam displays additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates,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

- ◆ From the **Toolpaths Manager**, select **Verify selected operations** icon as shown.



*Note: Mastercam launches a new window that allows you to check the part using **Backplot** or **Verify**.*

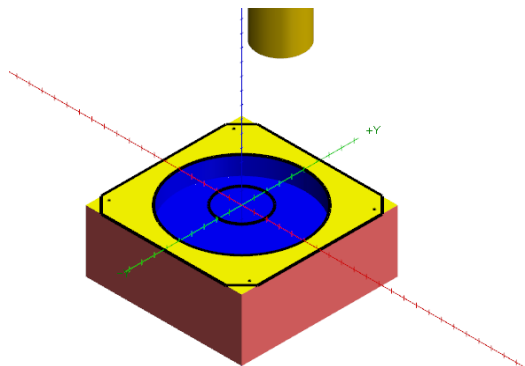
- ◆ In **Mastercam Simulator**, **Verify** should be enabled and change the settings as shown.



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



- ◆ The part should appear as shown.

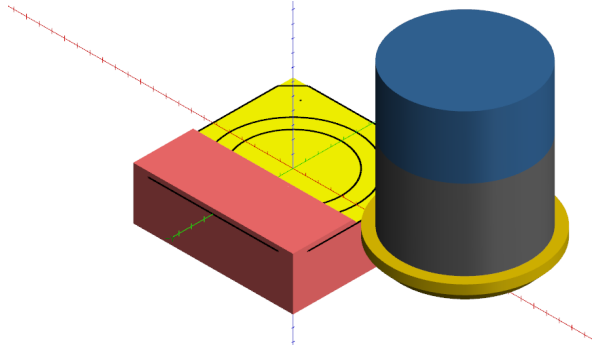


Note: To rotate the part, move the cursor to the center of the part and click and hold the mouse wheel and slowly move it in one direction. To zoom in or out, hold down the mouse wheel and scroll up or down as needed.

- ◆ Right mouse click in the graphics window and select **Isometric**. Then right mouse click again and select **Fit** to see the part in the original position.
- ◆ To check the part step-by-step, click first on the **Start** button.



- ◆ Click on the **Step Forward** to see the tool moving one step at a time.
- ◆ The part should look as shown after several steps.



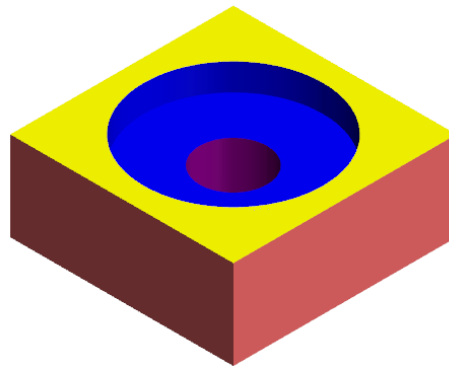
- ◆ Click on the **Step Forward** until the toolpath is completed.
- ◆ To go back to Mastercam window, minimize **Mastercam Simulator** window as shown.



STEP 6: CIRCLE MILL THE INSIDE HOLE

Circle Mill Toolpaths remove circular pockets based on a single point. You can select either point entities or center points of arcs. Mastercam will then pocket out a circular area of the diameter to the depth that you specify.

Toolpath Preview:

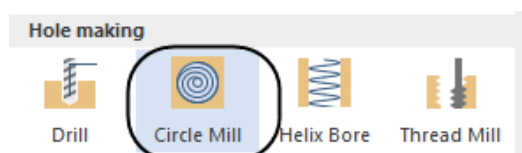


6.1 Drill Point Selection

- ◆ Hover the cursor in the **Toolpaths Manager** and press **T** or press **Alt + T** to remove the toolpath display.

Toolpaths

- ◆ From the **2D** group, click on the **Circle Mill** icon.





In-House Solutions **Mastercam**® 2020



TRAINING SOLUTIONS

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

TRAINING TUTORIALS

Our popular line of Mastercam Training Tutorials cover every Mastercam application, offering the most extensive collection of Mastercam training solutions available. Each book contains a series of tutorials that guide you through creating geometry, applying then verifying toolpaths and generating G-Code. Helpful hints, review exercises and quizzes complete the learning experience.



The Home Learning Edition (HLE) Demo Software is available for download with each eBook, and can be purchased on DVD for \$5.

ARE YOU AN INSTRUCTOR?

Buy 10 or more of the Mill Essentials, Mill Advanced or Lathe Training Tutorials, get the corresponding Instructor Guide for Free! (see page 5)



Mill Essentials

The Mill Essentials Training Tutorial takes a very comprehensive look at 2D machining with Mastercam.

This book begins with the absolute basics by introducing the Mastercam user interface and will have you creating geometry, mastering 2D toolpaths and posting code before you know it.

- 7 Step-by-Step Tutorials with Review Exercises
- Includes Video Training DVD and Mastercam HLE Demo Software

Price \$65 (eBook)

ISBN: 978-1-77146-870-1

Price \$90 (Print)

ISBN: 978-1-77146-833-6



Lathe

The Lathe Training Tutorial is ideal for both beginners and current Mastercam users.

This book introduces 2D geometry creation and lathe 2D toolpaths providing an excellent foundation for many lathe applications.

- 7 Step-by-Step Tutorials with Review Exercises
- Includes Video Training DVD and Mastercam HLE Demo Software

Price \$45 (eBook)

ISBN: 978-1-77146-872-5

Price \$65 (Print)

ISBN: 978-1-77146-835-0



Mill Advanced

The Mill Advanced Training Tutorial picks up where Mill Essentials leaves off, introducing 3D concepts including surface high speed toolpaths.

It is strongly recommended that you have a good understanding of 2D machining before beginning this book.

- 6 Step-by-Step Tutorials with Review Exercises
- Includes Video Training DVD and Mastercam HLE Demo Software

Price \$60 (eBook)

ISBN: 978-1-77146-871-8

Price \$80 (Print)

ISBN: 978-1-77146-834-3



Multiaxis Essentials

The Multiaxis Essentials Training Tutorial is intended for the advanced Mastercam user looking to learn Mastercam multiaxis programming.

A firm grasp on both 2D and 3D machining is required which can be attained by completing the Mill Essentials and Mill Advanced Training Tutorials.

- 10 Step-by-Step Tutorials
- Includes Video Training DVD and Mastercam HLE Demo Software

Price \$85 (eBook)

ISBN: 978-1-77146-878-7

Price \$90 (Print)

ISBN: 978-1-77146-841-1



Multiaxis Advanced

The Multiaxis Advanced Training Tutorial is intended for the advanced Mastercam user.

This book contains projects demonstrating a variety of Multiaxis Advanced milling applications such as pattern application toolpaths.

- 6 Step-by-Step Tutorials
- Includes Mastercam HLE Demo Software

Price \$75 (eBook)

ISBN: 978-1-77146-879-4

Price \$80 (Print)

ISBN: 978-1-77146-842-8



Lathe C & Y Axis

The Mastercam 2020 Lathe C & Y Axis Training Tutorial is intended for the advanced lathe user that wants to learn how to program lathe parts that include milling operations.

Advanced C-Axis toolpaths as well as the Y-Axis rotation will be described in this book. Examples on how to use the Mill toolpaths on a Lathe with Live Tooling are also incorporated.

Price \$65 (eBook)

ISBN: 978-1-77146-887-9

Price \$85 (Print)

ISBN: 978-1-77146-848-0



Solids

The Solids Training Tutorial provides concise step-by-step instructions on creating and manipulating 3D wireframe and solid geometry.

This book details commands such as extrude, loft, revolve, sweep, boolean add, fillet, chamfer and more.

- 9 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price \$40 (eBook)

ISBN: 978-1-77146-873-2

Price \$60 (Print)

ISBN: 978-1-77146-836-7



Mastercam for SOLIDWORKS

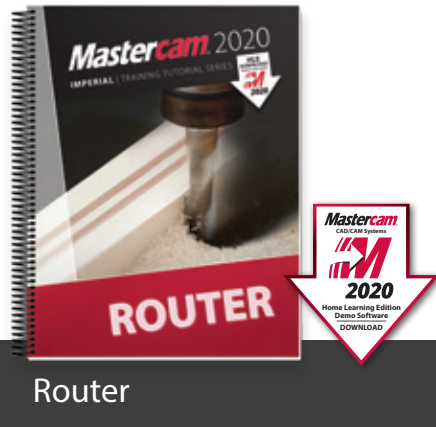
The Mastercam 2020 for SOLIDWORKS Programming Exercises book provides a comprehensive "hands on" method of learning Mastercam for SOLIDWORKS. You will learn how to program a variety of different parts that require most of the toolpath types available in Mastercam for SOLIDWORKS. Extensive emphasis is put on making parametric changes and toolpath updates to match the SOLIDWORKS model changes. Primary focus is on toolpath creation on SOLIDWORKS models.

Price \$65 (eBook)

ISBN: 978-1-77146-881-7

Price \$85 (Print)

ISBN: 978-1-77146-844-2



Router

The Router Training Tutorial offers detailed coverage of 2D geometry creation, drilling, contouring, pocketing, nesting, block drilling and importing graphics to machine.

Instruction on tool settings, stock setup and custom profile tool creation are also included.

- 8 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price \$65 (eBook)

ISBN: 978-1-77146-876-3

Price \$85 (Print)

ISBN: 978-1-77146-839-8



Beginner

The Beginner Training Tutorial provides a comprehensive step-by-step approach to learning the basics of three Mastercam modules: Mill Essentials (2D), Lathe and Solids.

It is an excellent choice for new Mastercam users looking to get a broader overview of the software.

- 8 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price \$65 (eBook)

ISBN: 978-1-77146-874-9

Price \$85 (Print)

ISBN: 978-1-77146-837-4



Design

The Design Training Tutorial provides a comprehensive step-by-step approach to learning geometry creation within Mastercam.

Its focus is 2D and 3D geometry creation with explanations given on how to dimension the parts.

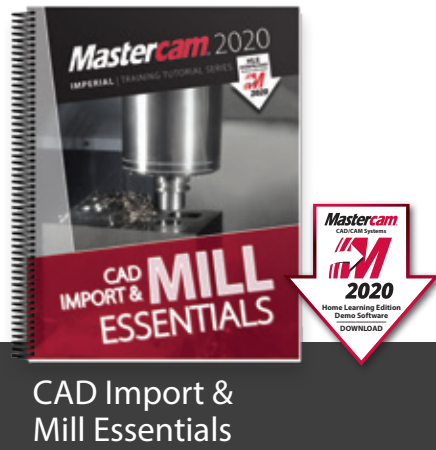
- 12 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price \$45 (eBook)

ISBN: 978-1-77146-875-6

Price \$65 (Print)

ISBN: 978-1-77146-838-1



CAD Import & Mill Essentials

The CAD Import & Mill Essentials Toolpaths Training Tutorial is intended for anyone looking to understand the ins and outs of Mastercam Mill Essentials toolpaths, while learning the best practices for importing geometry from various CAD software packages.

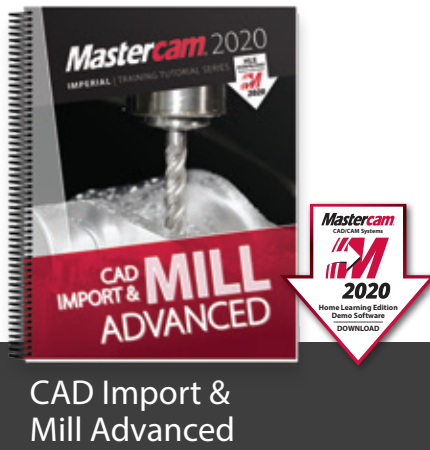
- 7 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price \$70 (eBook)

ISBN: 978-1-77146-878-7

Price \$90 (Print)

ISBN: 978-1-77146-846-6



CAD Import & Mill Advanced

The CAD Import & Mill Advanced Toolpaths Training Tutorial consists of 6 projects and 6 accompanying practice exercises. It is intended for intermediate to experienced Mastercam users who are primarily importing 3D geometry from another CAD software package.

- 6 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price \$60 (eBook)

ISBN: 978-1-77146-884-8

Price \$80 (Print)

ISBN: 978-1-77146-847-3



Wire

The Wire Training Tutorial provides users with an excellent resource for learning how to use Mastercam to program wire EDM machines.

In addition to geometry creation, the book focuses on wirepaths for dies, taper angle projects and more.

- 6 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price \$40 (eBook)

ISBN: 978-1-77146-877-0

Price \$60 (Print)

ISBN: 978-1-77146-840-4



Project Workbook

The Mastercam 2020 Project Workbook is ideal for High School students, hobbyists and those who prefer engaging in project-based learning. The Workbook includes an overview of CAD/CAM and basic machining followed by a series of step-by-step projects for both mills and lathes. *Note: no machining instructions included.*

- Five projects included: Art Lithophane, Bowling Pin, Shield, F1 Car, and Jewelry Box
- Includes Mastercam HLE Demo Software

Price \$60 (eBook)

ISBN: 978-1-77146-880-0

Price \$75 (Print)

ISBN: 978-1-77146-843-5



Nesting eBook

This tutorial includes a variety of projects that are using Nesting options in Mastercam to fit parts onto a sheet of material for best yield.

You will learn how Nesting operates on geometry and how you can create and use tabs to assist in holding down the nested parts. You will also learn how Nesting operates on toolpaths and how you can use WCS (Work Coordinate System) to set the parts of an assembly in the proper view for machining.

Price \$35 (eBook only)

ISBN: 978-1-77146-885-5

SITE LICENSE

An eBook site license allows an educational facility to purchase one or several of our Training Solutions in eBook format (PDF). This in turn grants lifetime access to the content from any workstation on campus for unlimited users.

For more information on eBook Site Licenses, email EBooks@inhousesolutions.com or call **1.800.529.5517**.



INSTRUCTOR KITS

These are the ultimate toolkits for instructors looking to enhance their 2D mill, 3D mill and lathe classes while minimizing prep time. Each kit contains tests, quizzes, mid-terms, finals, powerpoint presentations, lesson plans, answer sheets, and more.



Instructor Guide & Training Tutorial are included in each kit. Each print kit includes an Instructor DVD, Video Training DVD, and Mastercam HLE Demo Software. Download links are provided for eBook kits.

eBook

Bundle of all three:

\$200

ISBN: 978-1-77146-882-4

Print

\$100 Mill Essentials Kit

ISBN: 978-1-77146-864-0

\$100 Mill Advanced Kit

ISBN: 978-1-77146-865-7

\$100 Lathe Kit

ISBN: 978-1-77146-866-4

Bundle of all three:

\$250

ISBN: 978-1-77146-845-9

MASTERCAM PROFESSIONAL CERTIFICATION CURRICULUM

This 3-part series is designed to help you prepare for your Professional Level Mastercam Certification™ Test. The skills exam must be administered by a Mastercam Certified Instructor or a Mastercam Certified Reseller.



Part 1 –
Mill 2D

Part 1 – Mill 2D begins at the absolute basics by introducing the Mastercam user interface. It will have you creating geometry, drilling and creating contour toolpaths before you know it. Its gradual progression leads to more advanced concepts such as multiple setup scenarios and 2D High Speed Toolpaths. All of the parts within the book are designed with machinability in mind.

A multitude of topics are covered including 2D geometry, tool settings, stock setup, drilling, tapping, contouring, pocketing, circle milling. You will learn the 2D High Speed Toolpaths such as dynamic mill, area mill, dynamic rest mill, how to import a solid and machine it and how to use the WCS in multiple fixture applications.

Price \$100

ISBN: 978-1-77146-867-1



Part 2 –
Solids

Part 2 – Solids will teach a variety of useful solids commands, that include extrude, loft, revolve, sweep, boolean add and remove, fillet, chamfer, shell, trimming, and creating solid patterns. You will learn how to change a solids color and create a solid geometry from surfaces, as well as how to manipulate a solid by using commands from the Model Prep menu such as Push-Pull, Move and Split Solid Face. You will use create Layout and learn how to dimension parts to create blueprints.

Price \$100

ISBN: 978-1-77146-868-8



Part 3 –
Advanced
Mill 2D

Part 3 – Advanced Mill 2D is dedicated to the power user. You will learn how to machine an imported solid model using the WCS for multiple setups. To better organize the parts, you will master how to use levels and view sheets. You will practice indexing a 4-axis part using the tool planes defined in the Plane manager. Fixtures will be used and the part will be align to their faces. You will learn how to save operations to a library and how to import them for different parts. A brief introduction to the basic 3D machining will be also covered.

Price \$100

ISBN: 978-1-77146-869-5



Part 1, 2 & 3
Bundle

This part 1, 2 & 3 bundle provides you all three Mastercam Professional Certification Curriculum at a better price.

Price \$250

ISBN: 978-1-77146-904-3

HANDBOOKS

The Mastercam 2020 Handbooks provide an academic approach to teaching the theory and application of Mastercam. The Handbook series is designed to teach the fundamentals of Mastercam, gradually working up to more complex material with each volume. Each Handbook comes with a Student DVD that contains files referenced within the book, and the Mastercam 2020 HLE Demo Software.



Handbook Volume 1 Mill 2D & Solids

The Mastercam 2020 Handbook Volume 1 is an excellent resource that teaches the theory of Mastercam Mill and Solids. The material intertwines conceptual subject matter with practical applications suitable for beginners and intermediates alike. You not only learn how to use Mastercam but why things work as they do.

Price \$90 (eBook)

ISBN: 978-1-77146-893-0

Price \$98 (Print)

ISBN: 978-1-77146-855-8



Handbook Volume 2 3D Modeling & Machining

The Mastercam 2020 Handbook Volume 2 takes an academic approach to teaching Mastercam 3D modeling and machining. The material is most suitable for intermediates (including individuals that have completed Volume 1). The book teaches more advanced CAD modeling techniques and explains surface creation.

Price \$90 (eBook)

ISBN: 978-1-77146-894-7

Price \$98 (Print)

ISBN: 978-1-77146-856-5



Handbook Volume 3 Multiaxis & Machining

The Mastercam 2020 Handbook Volume 3 is an excellent resource that teaches the theory of multiaxis machining with Mastercam. This book covers the classic family of multiaxis toolpaths including drill 5-axis, curve 5-axis, swarf 5-axis, multiaxis flowline and multiaxis multi-surfaces as well as the drill & circle mill family.

Price \$90 (eBook)

ISBN: 978-1-77146-895-4

Price \$98 (Print)

ISBN: 978-1-77146-857-2

FREE eBooks



How-To Tips Collection

The In-House Solutions support specialists take note of the questions that are asked. We realize that if you're asking a question there are ten more people who want to know the answer but haven't asked. Our goal is to make your life easier, so we've collected the answers to your most popular questions here, in one convenient eBook.

Price – FREE!
(eBook only)



Horizontal Machining with Indexing

The Indexing Training Tutorial explains how to use the Plane Manager for working with tool planes and work offsets.

Examples include horizontal machining using the WCS and a multiple fixtured tombstone project.

Price – FREE!
(eBook only)

Download at www.eMastercam.com/freebooks

Mastercam® @Courses



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



LOG IN & LEARN TODAY!

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

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

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

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

Highlights:

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

\$149 each



Mill Essentials eCourse

The Mill Essentials eCourse introduces students to 2D CAD and milling toolpaths. It covers wireframe and solids creation as well as 2D mill toolpaths such as contour, drilling, blend, peel, dynamic area, transform, Feature Based Drilling, and more. This course serves as an excellent introduction to Mastercam.



Mill Advanced eCourse

The Mill Advanced eCourse builds on what students have learned in the Mill Essentials eCourse. It moves into more advanced CAD and demonstrated 3D wireframe, solid, and surface creation commands. 3 axis toolpaths such as Area Roughing, Dynamic OptiRough, Scallop, Pencil, Waterline, Radial, Hybrid, and more are covered.



Lathe eCourse

The Mastercam Lathe eCourse covers wireframe creation, working with imported part files, stock setup, facing, roughing, finishing, grooving, drilling, and cutoff toolpaths. Stock operations such as advance, flip, and tailstock are also covered. You will also learn how to program parts in a VTL.



Multiaxis Essentials eCourse

The Multiaxis Essentials eCourse covers 4 & 5 axis toolpaths. Toolpaths include contour with axis substitution, drilling with axis substitution, drilling with rotary axis positioning, rotary 4-axis, curve 5-axis, swarf 5-axis, drill 5-axis, circle mill 5-axis, flow 5-axis, and multisurface 5-axis. This course skips most CAD in favor of focusing on toolpaths.

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

PROFESSIONAL COURSEWARE

The Mastercam Professional Courseware titles are intended for industrial training settings. Instead of step-by-step instructions, these books introduce concepts through sequences of specialized training exercises followed by parts the users are expected to produce with minimal guidelines.



Professional Courseware Mill Essentials

The Mastercam 2020 Mill Essentials Professional Courseware provides in-depth coverage of 2D wireframes and solids geometry, as well as contour, pocket, drilling, circle milling and slot milling toolpaths. More advanced exercises explain the use of the Work Coordinate System (WCS), 2D high speed toolpaths, Feature Based Machining (FBM) and more.

Price \$60 (eBook)

ISBN: 978-1-77146-896-1

Price \$70 (Print)

ISBN: 978-1-77146-858-9



Professional Courseware Mill Advanced

The Mastercam 2020 Mill Advanced Professional Courseware covers a multitude of features that teach a user to create 3D wireframes, surfaces and solids for 3D modeling and toolpaths. Interactive training exercises introduce 3D geometry functionality, while newer surface high speed toolpaths are thoroughly investigated along with their various parameter settings.

Price \$50 (eBook)

ISBN: 978-1-77146-897-8

Price \$60 (Print)

ISBN: 978-1-77146-859-6



Professional Courseware Lathe

The Mastercam 2020 Lathe Professional Courseware offers an in-depth look at Mastercam Lathe geometry and toolpath creation. Advanced toolpaths such as Misc Ops and C-Axis toolpaths are also described. Additional Mastercam files are provided along with guidelines for creating the toolpaths to machine each part.

Price \$55 (eBook)

ISBN: 978-1-77146-898-5

Price \$65 (Print)

ISBN: 978-1-77146-860-2



Professional Courseware Multiaxis

The Mastercam 2020 Multiaxis Professional Courseware details numerous toolpaths that allow a user to successfully machine 4-axis and 5-axis parts. Multiaxis Advanced toolpaths have been included with more complex parts along with instructions on how to machine them. Machine Simulation is used to check for any collisions between the part, the tool and any of the machine's components.

Price \$65 (eBook)

ISBN: 978-1-77146-899-2

Price \$75 (Print)

ISBN: 978-1-77146-861-9

Join the eMastercam community!

eMastercam is one of the largest and oldest online forums for swapping knowledge of CNC machines, tools, manufacturing processes and technology. After hours, eMastercam continues to be a place for Machinists, Engineers and others in the manufacturing industry to connect, share stories, opinions and get to know each other beyond the parts we make and the tools we use.

- Active user forums with an education-specific sub forum
- Free downloads and educational resources
- Free post processors and advanced development requests
- Overviews of Mastercam 2020 and related products
- The place to find our Mastercam Training Solutions
- Industry insights and much more!

www.eMastercam.com/forums

eMastercam.com
YOUR ONLINE MASTERCAM SOURCE

We don't just educate,
we inspire!



eMastercam

We appreciate
your feedback

TELL US WHAT YOU THINK

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

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

www.eMastercam.com/feedback



240 Holiday Inn Drive, Unit A
Cambridge, Ontario, Canada N3C 3X4
T: 800.529.5517 F: 519.658.1335
books@inhousesolutions.com
www.inhousesolutions.com

