

Mastercam® 2020

IMPERIAL | TRAINING TUTORIAL SERIES

HLE
DOWNLOAD
INSTRUCTIONS INSIDE



2020



MILL ESSENTIALS

Mastercam® 2020

MILL ESSENTIALS 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

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

Proceed to download!

Mastercam 2020 Mill Essentials Tutorial

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

Software: Mastercam 2020

Authors: Mariana Lendel

ISBN: 978-1-77146-833-6

Date: July 15, 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 Projects	13
Getting Started	17
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 Imperial	25
Step 7: Set The Grid	27
Conventions Used In This Book:	28
Mastercam® Work Flow	29
Tutorial 1: Geometry Creation	33
Tutorial #1 Drawing	35
Step 1: Setting Up The Graphical User Interface	36
Step 2: Create One Rectangle	37
Step 3: Create The 1/4" Diameter Circles	40
Step 4: Create The 2.0" And 0.75" 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
Tutorial 1: Toolpath Creation	53
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	76
Step 5: Simulate The Toolpath In Verify	78
Step 6: Circle Mill The Inside Hole	79
Step 7: Spot Drill The 0.25" Holes	88
Step 8: Drill The 0.25" Holes	98
Step 9: Chamfer The Large Hole	104
Step 10: Machine The Chamfers At The Corners Using Contour Toolpath	113
Step 11: Post The File	120
Step 12: Save The Updated MNCAM File	121
Create The Toolpaths For Tutorial #1 Exercise	122
Tutorial 2: Geometry Creation	125
Tutorial #2 Drawing	127
Step 1: Setting Up The Graphical User Interface	128
Step 2: Create A Rectangle	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: Rotate The Part	156
Step 12: Save The File	158
Tutorial #2 Review Exercise	159
Create The Geometry For Tutorial #2 Exercise	160

Tutorial 2: Toolpath Creation	163
Toolpath Creation - Setup #1	165
Suggested Fixture	165
Setup Sheet	165
Step 1: Select The Machine And Set Up The Stock	166
Step 2: Slot Milling	171
Step 3: Backplot The Toolpaths	181
Step 4: Simulate The Toolpath In Verify	183
Step 5: Machine The Cutout Pockets	184
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	209
Suggested Fixture:	215
Setup Sheet:	215
Step 9: Creating And Renaming Toolpath Groups	216
Step 10: Create And Set WCS To Bottom	218
Step 11: Face The Part	224
Step 12: Rename The NC File	230
Step 13: Post The File	231
Step 14: Save The Updated MCAM File	233
Create The Toolpaths For Tutorial #2 Exercise	234
Tutorial 3: Geometry Creation	237
Tutorial #3 Drawing	239
Step 1: Setting Up The Graphical User Interface	240
Step 2: Create A Rectangle	240
Step 3: Create Arcs Knowing Endpoints	242
Step 4: Delete The Rectangle	246

Step 5: Mirror The Geometry	247
Step 6: Create A Fillet	250
Step 7: Create The Ellipses	252
Step 8: Offset The Ellipse	255
Step 9: Create Lines Knowing The Endpoints	257
Step 10: Add The Text	259
Step 11: Break At Intersection	265
Step 12: Translate	266
Step 13: Change The Main Level To 2	272
Create The Solid Body	272
Step 14: Extrude The Base Geometry	272
Step 15: Extrude Cut The Body	276
Step 16: Add The Ellipse As A Boss Shape	278
Step 17: Extrude Cut The Ellipse	280
Step 18: Create The Bottom Cylinder	282
Step 19: Save The File	286
Tutorial #3 Review Exercise	287
Create The Geometry For Tutorial #3 Exercise	288
Tutorial 3: Toolpath Creation	291
Suggested Fixture	293
Setup Sheet	293
Step 1: Select The Machine And Set Up The Stock	294
Step 2: 2D High Speed Area Mill	297
Step 3: Backplot The Toolpaths	307
Step 4: Simulate The Toolpath In Verify	309
Step 5: Pocket With Islands	310
Step 6: Pocket Remachining	319
Step 7: 2D HS Dynamic Mill - Outside Profile	323

Step 8: 2D HS Dynamic Mill - Open Pockets	330
Step 9: Post The File	338
Step 10: Save The Updated MCAM File	339
Create The Toolpaths For Tutorial #3 Exercise	340
Tutorial 4: Geometry Creation	343
Tutorial #4 Drawing	345
Step 1: Setting Up The Graphical User Interface	346
Step 2: Create Two Arcs	346
Step 3: Create Vertical Lines	355
Step 4: Create Arc Using Arc Polar Endpoints	357
Step 5: Rotate Geometry	359
Step 6: Mirror Geometry	361
Step 7: Create An Arc Tangent	363
Step 8: Trim Geometry	365
Step 9: Mirror Geometry To Complete Arms	368
Step 10: Create A Construction Line	371
Step 11: Create A 0.5" Diameter Circle	373
Step 12: Delete Construction Geometry	375
Step 13: Create Tangent Lines	376
Step 14: Create Arc Polar	381
Step 15: Create Fillets	383
Step 16: Trim The Arc	385
Step 17: Rotate	387
Step 18: Translate	389
Step 19: Change The Main Level To 2	394
Step 20: Create The Solid Body By Extruding A Closed Chain	394
Step 21: Extrude Cut The Part Pockets And The Holes	402

Step 22: Chamfer The Part	410
Step 23: Save The File	415
Tutorial #4 Review Exercise	416
Create The Geometry For Tutorial #4 Exercise	417
Create The Solid Geometry For Tutorial #4 Exercise	417
Tutorial 4: Toolpath Creation	419
Suggested Fixture	421
Setup Sheet	422
Step 1: Select The Machine And Set Up The Stock	423
Step 2: 2D High Speed Dynamic Mill	427
Step 3: Backplot The Toolpaths	439
Step 4: Simulate The Toolpath In Verify	440
Step 5: Finish The Walls Using Contour Toolpath	442
Step 6: Area Mill Toolpath	449
Step 7: Transform-Rotate Toolpath	460
Step 8: Finish The Pocket Walls Using Contour Toolpath	464
Step 9: Spot Drill The Holes	469
Step 10: Drill All Holes	475
Step 11: Chamfer The Outside Diameter	479
Step 12: Chamfer The Pockets	486
Suggested Fixture 2:	493
Setup Sheet 2:	493
Step 13: Creating And Renaming Toolpath Groups	494
Step 14: Set The WCS To Bottom	496
Step 15: 2D HS Dynamic Mill	499
Step 16: Finish The Pocket Wall Using Contour Toolpath	508
Step 17: Rename The NC File	514
Step 18: Post The File	515

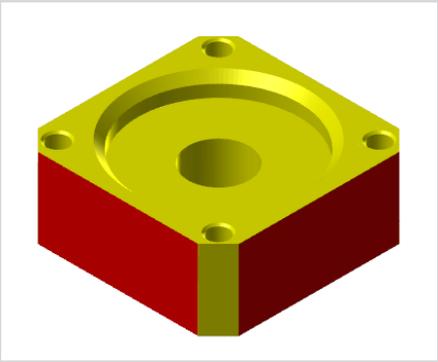
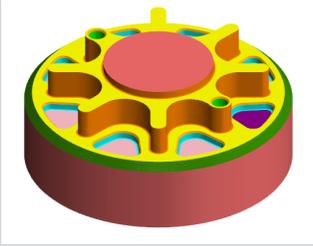
Step 19: Save The Updated MCAM File	516
Create The Toolpaths For Tutorial #4 Exercise	517
Tutorial 5: Geometry Import	523
Step 1: Setting Up The Graphical User Interface	524
Step 2: Importing The Solidworks File Geometry	524
Step 3: Save The File	526
Tutorial #5 Review Exercise	527
Tutorial 5: Toolpath Creation	529
Suggested Fixture	531
Setup Sheet	532
Step 1: Select The Machine And Set Up The Stock	533
Step 2: 2D High Speed Area Mill	537
Step 3: Backplot The Toolpaths	550
Step 4: Simulate The Toolpath In Verify	551
Step 5: 2D High Speed Area Mill	552
Step 6: Remachine The Remaining Material Using Area Mill	564
Step 7: Spot Drill The Holes	574
Step 8: Drill All Holes	582
Step 9: Tap The Holes	588
Suggested Fixture 2:	593
Setup Sheet 2:	594
Step 10: Creating And Renaming Toolpath Groups	595
Step 11: Set WCS To Front	597
Step 12: Spot Drill All 3 Holes	600
Step 13: Drill The Two 3/8" Tapped Holes	608
Step 14: Tap The Two Holes	615
Step 15: Drill The 1/2" Hole	621

Step 16: Counterbore The 1/2" Hole	626
Step 17: Rename The NC File	632
Setup Sheet 3:	634
Step 18: Creating And Renaming Toolpath Groups	635
Step 19: Set The WCS To Left Side	636
Step 20: Machine The Slot	638
Step 21: Rename The NC File	646
Step 22: Post The File	647
Step 23: Save The Updated MCAM File	648
Create The Toolpaths For Tutorial #5 Exercise	649
Tutorial 6: Geometry Creation	661
Tutorial #6 Drawing	663
Step 1: Setting Up The Graphical User Interface	664
Step 2: Create Rectangles	664
Step 3: Create Circles And Arc Tangent	667
Step 4: Trim The Arcs	672
Step 5: Create Line Parallel	677
Step 6: Create Chamfers	679
Step 7: Create Polar Lines	681
Step 8: Create Line Parallel	684
Step 9: Delete Entities	686
Step 10: Trim Entities	687
Step 11: Save The File	690
Tutorial #6 Review Exercise	691
Create The Geometry For Tutorial #6 Exercise	692
Tutorial 6: Toolpath Creation	695
Suggested Fixture	697
Setup Sheet	697

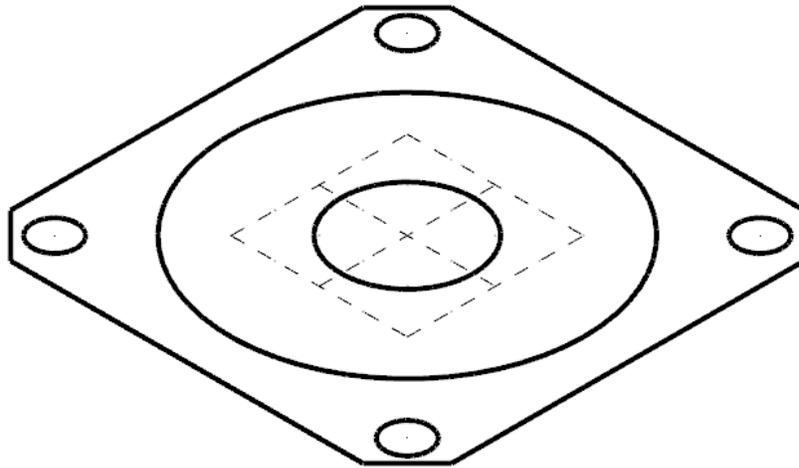
Step 1: Select The Machine And Set Up The Stock	698
Step 2: 2D High Speed Dynamic Mill	703
Step 3: Backplot The Toolpaths	715
Step 4: Simulate The Toolpath In Verify	716
Step 5: 2D High Speed Area Mill	717
Step 6: 2D High Speed Blend Mill	727
Step 7: 2D High Speed Peel Mill	738
Step 8: 2D High Speed Peel Mill	744
Step 9: Post The File	751
Step 10: Save The Updated MCAM File	752
Create The Toolpaths For Tutorial #6 Exercise	753
Tutorial 7: Geometry Creation	755
Tutorial #7 Drawing	757
Step 1: Setting Up The Graphical User Interface	758
Step 2: Create Circle Center Point	758
Step 3: Create Line Tangent	760
Step 4: Create A Line Parallel	762
Step 5: Trim The Entities	764
Step 6: Create Rectangular Shape	767
Step 7: Trim Divide	771
Step 8: Fillet Chains	773
Step 9: Create The Obround Shapes	776
Step 10: Create Circle Center Point	779
Step 11: Set The Level	783
Step 12: Extrude The Base Of The Solid	785
Step 13: Chamfer The Holes	795
Step 14: Fillet The Edges	798

Step 15: Save The File	801
Tutorial #7 Review Exercise	802
Create The Geometry For Tutorial #7 Exercise	803
Create The Solid Geometry For Tutorial #7 Exercise	803
Tutorial 7: Toolpath Creation	805
Suggested Fixture	807
Setup Sheet	807
Step 1: Select The Machine And Set Up The Stock	809
Step 2: 2D High Speed Dynamic Mill	814
Step 3: Backplot The Toolpaths	825
Step 4: Simulate The Toolpath In Verify	826
Step 5: Drill The Holes Using FBM Drill	827
Step 6: Machine The Outside Of The Part Using Dynamic Milling	836
Step 7: Finish The Pockets Using A Pocket Toolpath	848
Step 8: Contour Toolpath	858
Step 9: Post The File	866
Step 10: Save The Updated MCAM File	867
Create The Toolpaths For Tutorial #7 Exercise	868
Creating/Editing A Mill Tool Library	873
Quiz Answers	885
Mill Essentials Tutorial Quiz Answers	886

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. 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>

Tutorial 1: Geometry Creation



OVERVIEW OF STEPS TAKEN TO CREATE THE PART GEOMETRY:

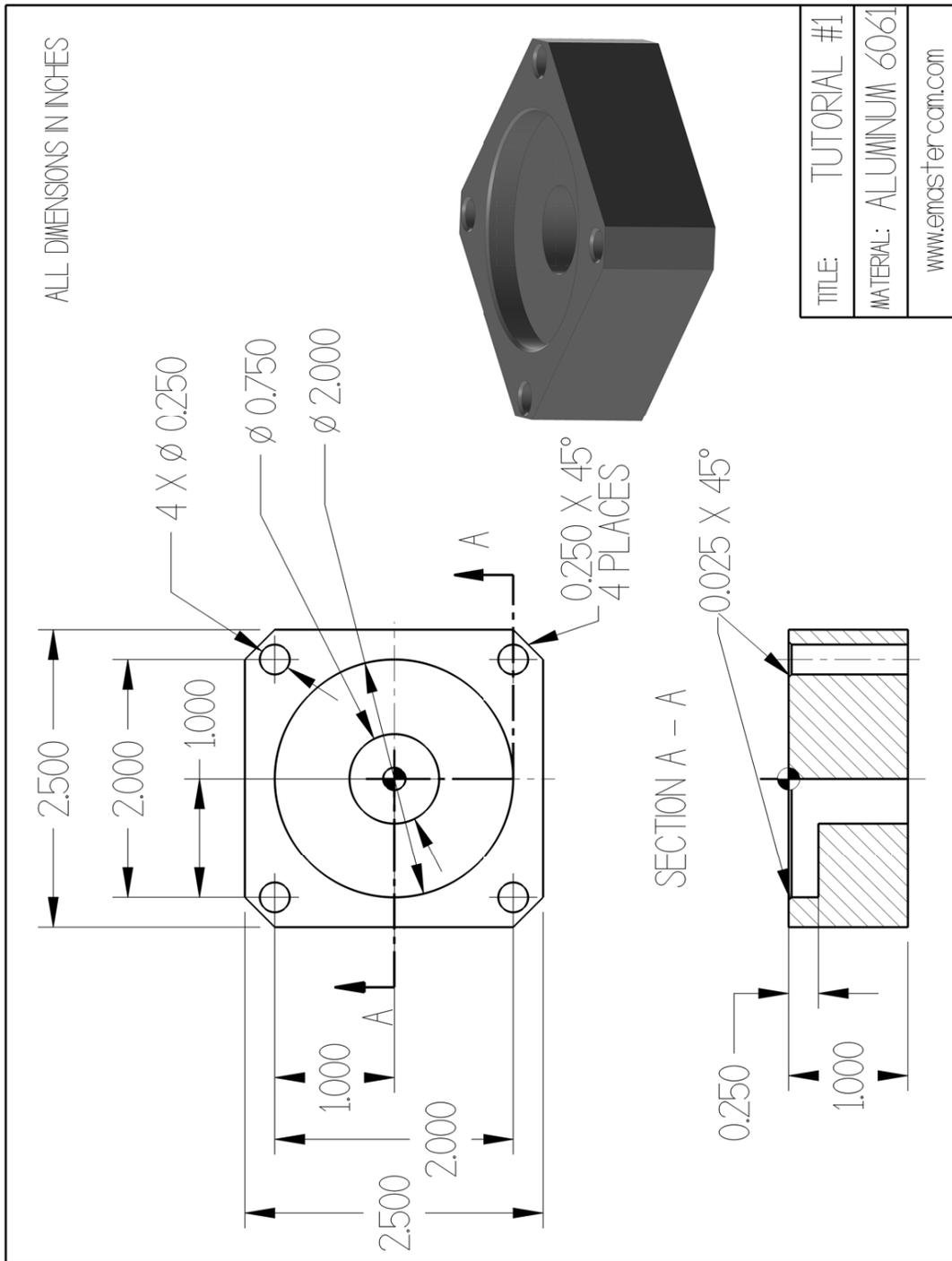
From Drawing to CAD Model:

- ◆ 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.

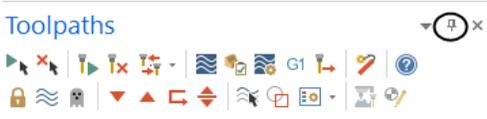
TUTORIAL #1 DRAWING



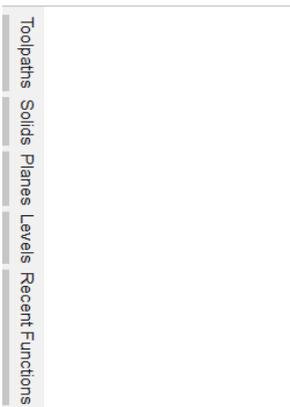
STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

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.

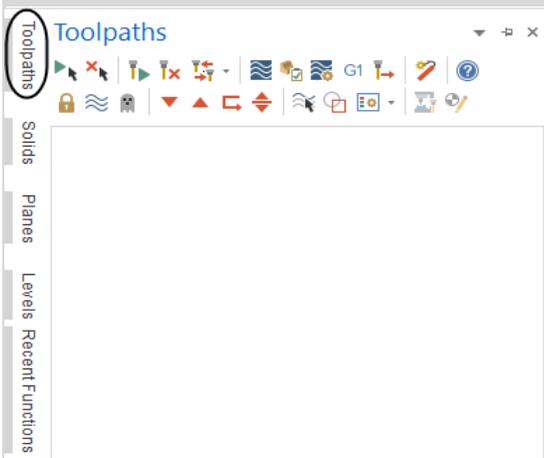
- ◆ Use **Auto Hide** icon to hide all **Manager** panels.



- ◆ 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.

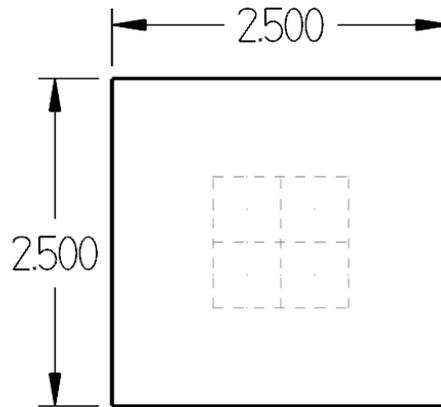


While creating the geometry, keep the Manager panels hidden. This ensures more space in the graphics window for the geometry.

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.

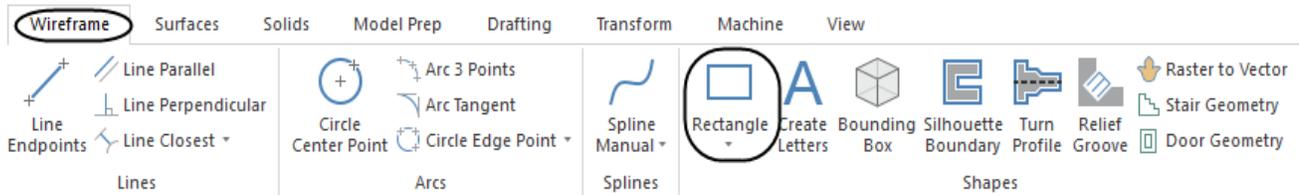
Step Preview:



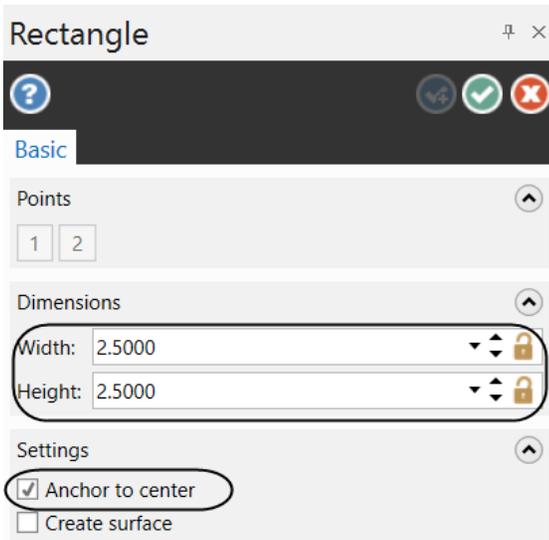
2.1 Create a 2.5" by 2.5" Rectangle

Wireframe

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

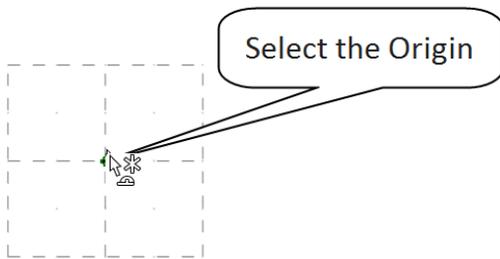


- ◆ In the **Rectangle** panel, enter the **Width** and **Height** and enable **Anchor to center** as shown.

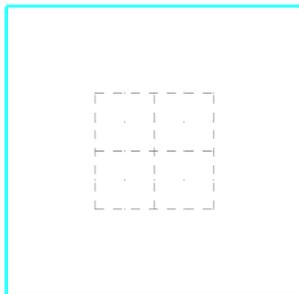


*Note: Make sure that **Create surface** is not selected. **Anchor to center** sets the base point of the rectangle to its center and draws the rectangle outward from the center. **Create surface** creates a surface inside of the rectangle. Surface creation and Surface toolpath are covered in Mill Advanced.*

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



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

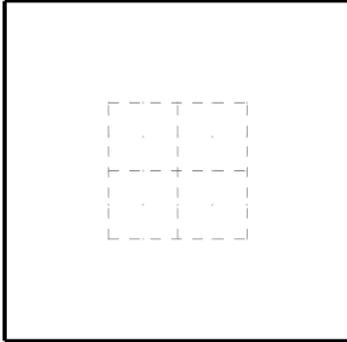


Note: The geometry should appear in cyan blue color which is the color for the live entities. While the rectangle is live, you can adjust the dimensions or select a new base point.

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



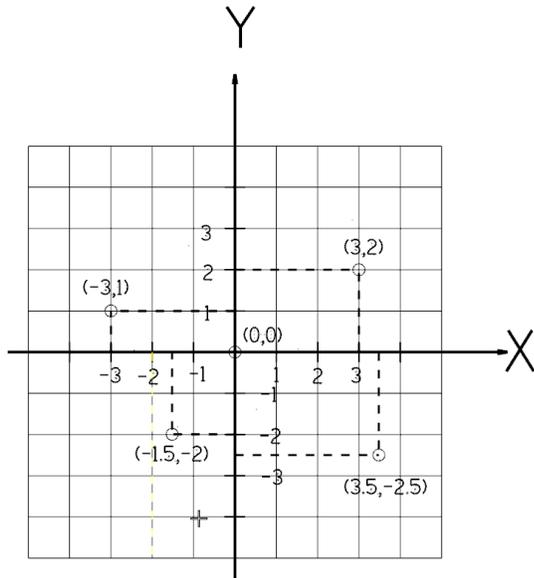
- ◆ The geometry should look as shown.



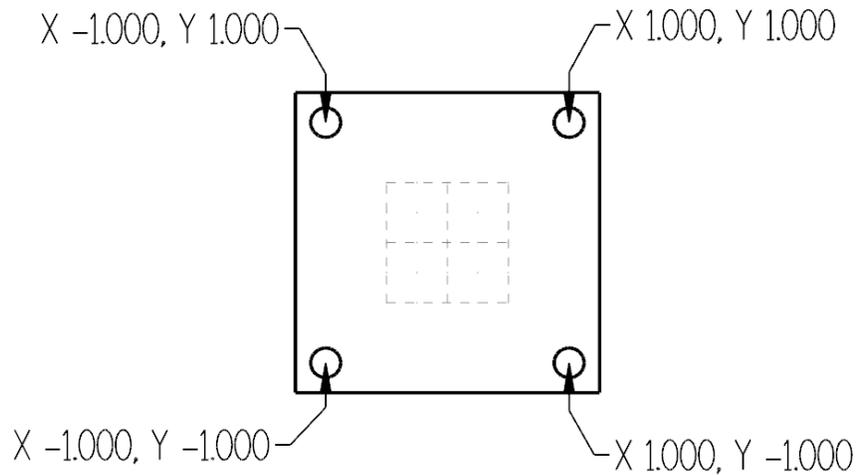
*Note: While creating geometry for this tutorial, if you make a mistake, you can undo the last step using the **Undo** icon.  You can undo as many steps as needed. If you delete or undo a step by mistake, just use the **Redo** icon.  To delete unwanted geometry, select the geometry first and then press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

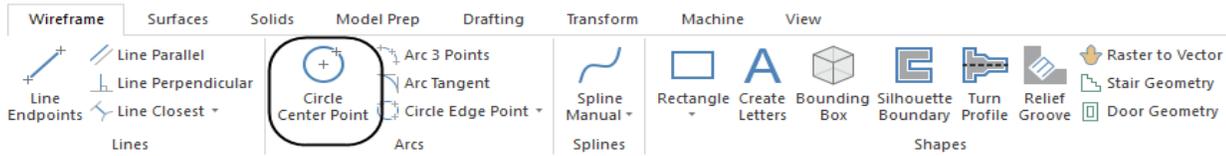


Step Preview:

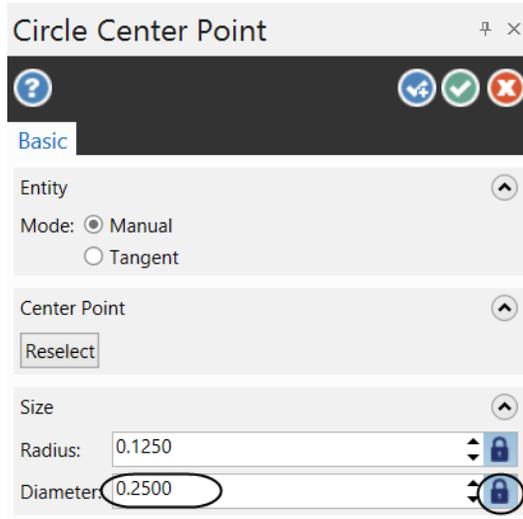


Wireframe

- ◆ 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.

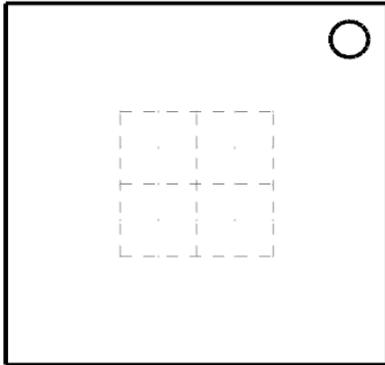


- ◆ Type **1, 1** as shown.

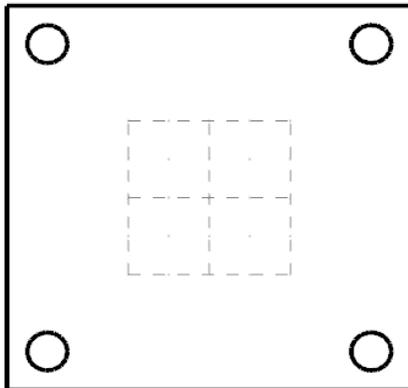


Note: When entering the coordinates for the center point, the first value is the X coordinate value, then the Y value followed by the Z value only if it is different from zero. The coordinate values are separated with commas. You do not need to use the coordinate labels if you enter the values in this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

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



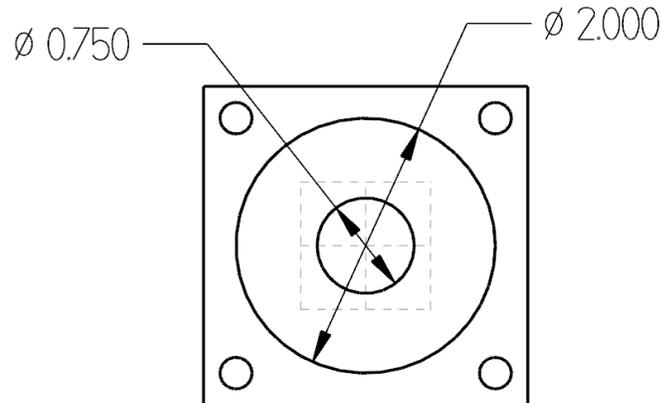
- ◆ [Enter the center point]: Select the **AutoCursor Fast Point** icon again and enter **1, -1**.
- ◆ Press **Enter** to place the circle.
- ◆ [Enter the center point]: Select the **AutoCursor Fast Point** icon again and enter **-1, 1**.
- ◆ Press **Enter** to place the circle.
- ◆ [Enter the center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

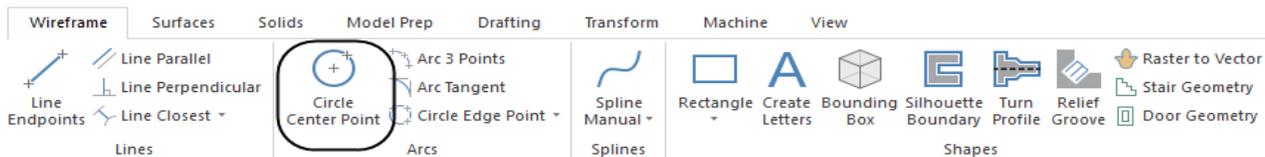
In this step, you will use the same Circle Center Point to create circles that you know the diameters and the locations.

Step Preview:

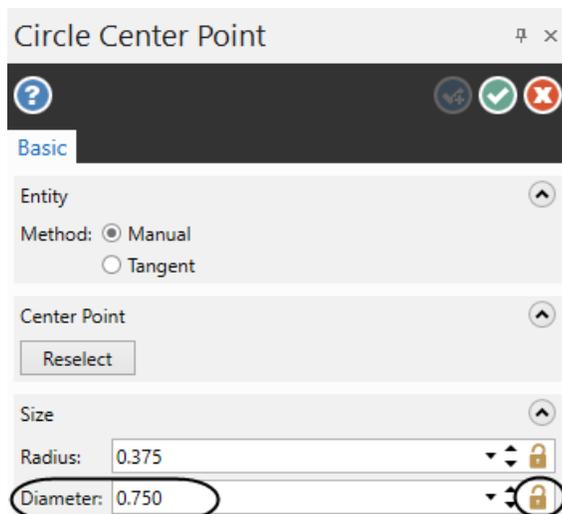


Wireframe

- ◆ 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**. 

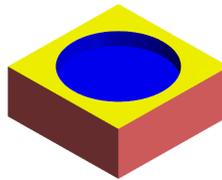
*Note: If you exit the toolpath in the middle of setting the parameters, in the **Toolpaths Manager**, you will have a red X on the **Face Toolpath** as shown in. This shows that you modified the toolpath and you need to update it. You will have to select the **Regenerate all dirty operations** icon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

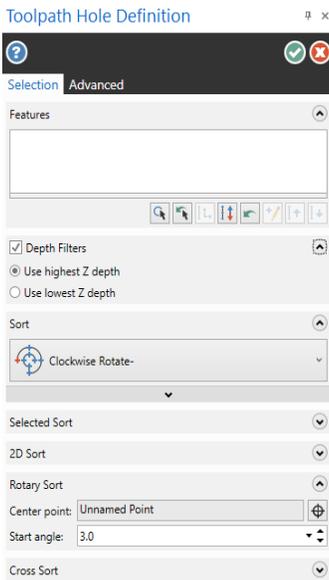
Toolpaths

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

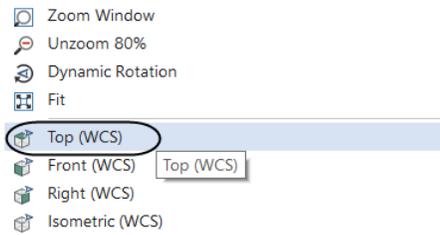


*Note: Move the cursor to the center of the graphics window to see the **Toolpath Hole Definition** panel that appears below the **Toolpaths Manager**.*

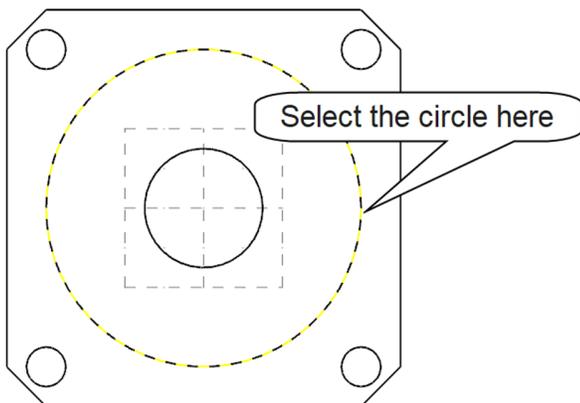
- ◆ The **Toolpath Hole Definition** should appear as shown.



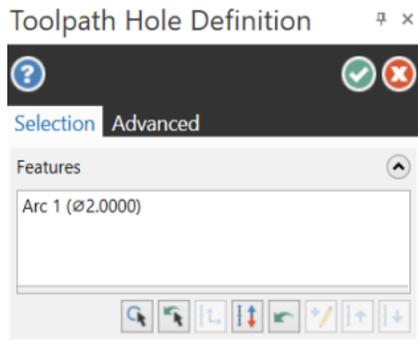
- ◆ Right click and select the **Top** view.



- ◆ [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.

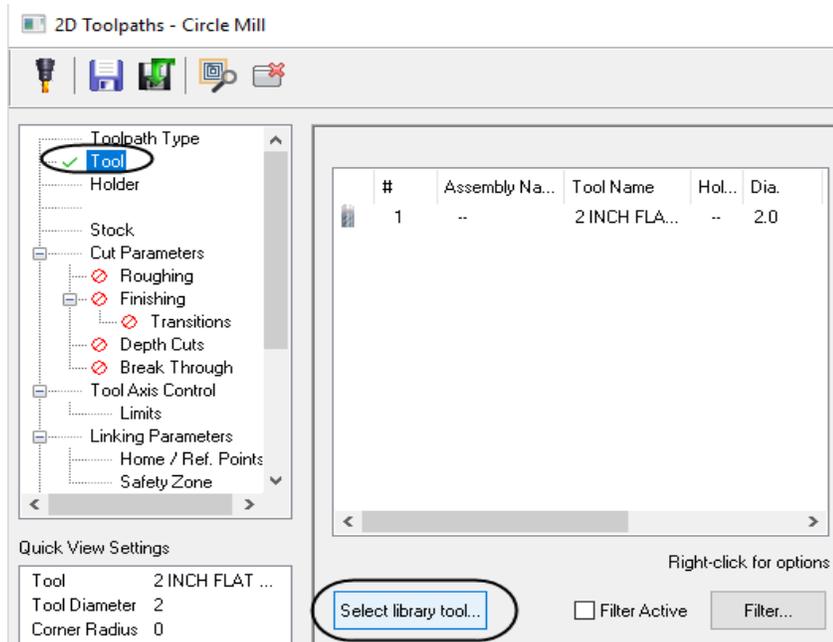


- ◆ Select the **OK** button to finish the selection and exit **Toolpath Hole Definition**.
- ◆ In the **Toolpath Type** page, the **Circle Mill** icon will be selected.



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**.

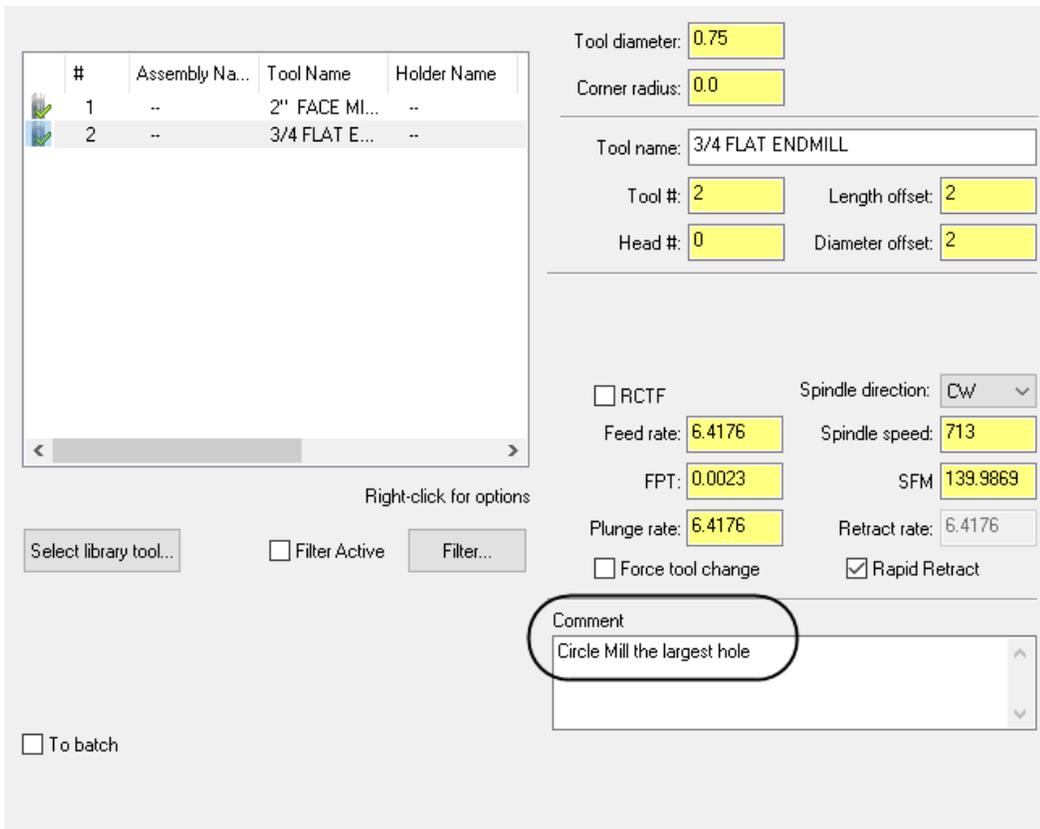


- ◆ Scroll down and select the **3/4" Flat Endmill (#294)** as shown.

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.

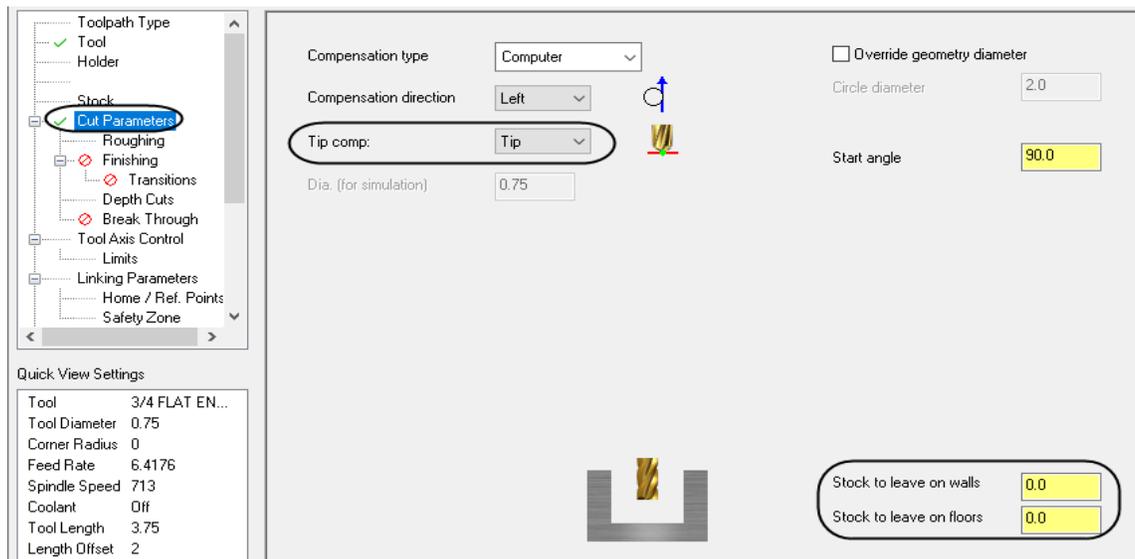
- ◆ Input a comment and make all the necessary changes, as shown.



*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.*

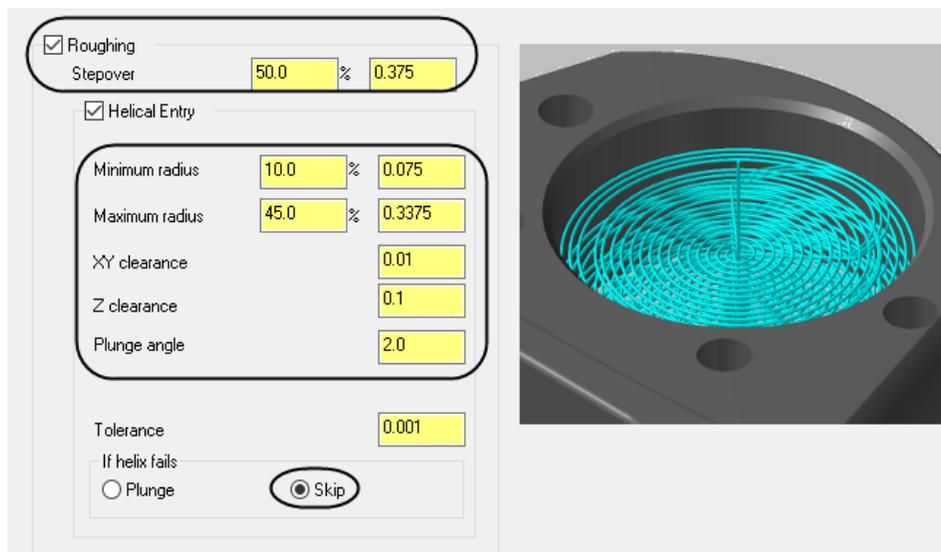
3.3 Cut Parameters

- ◆ 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.



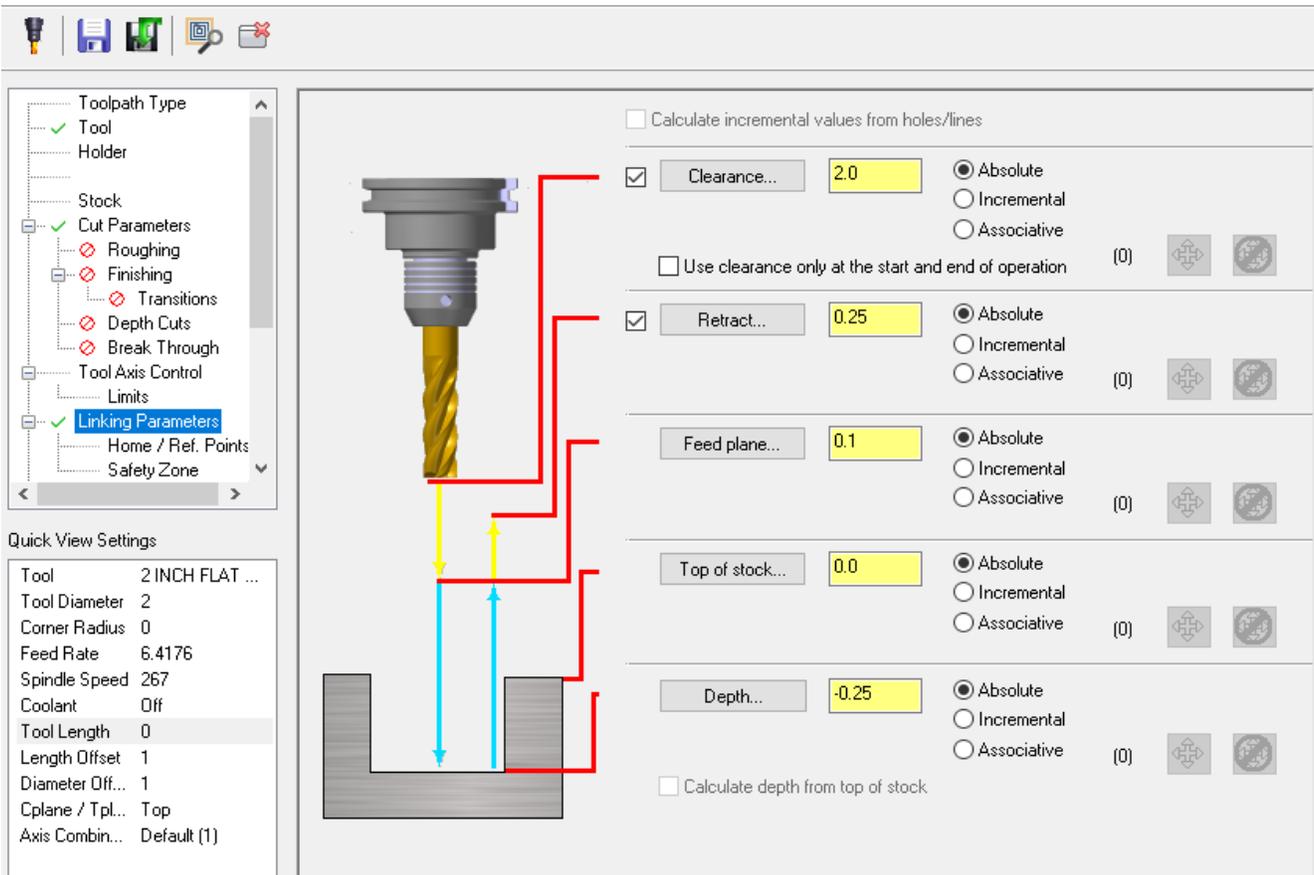
Stepover sets the distance between cutting passes in the **X** and **Y** axes as a percentage of the tool diameter.

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.

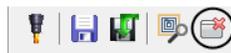
Associative option allows you to select points from the existing geometry from where the values will be measured.

3.6 Preview the Toolpath

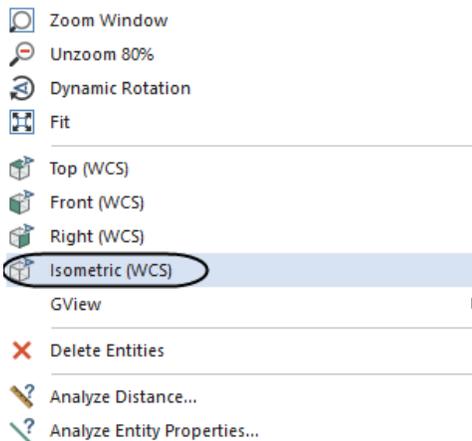
- ◆ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



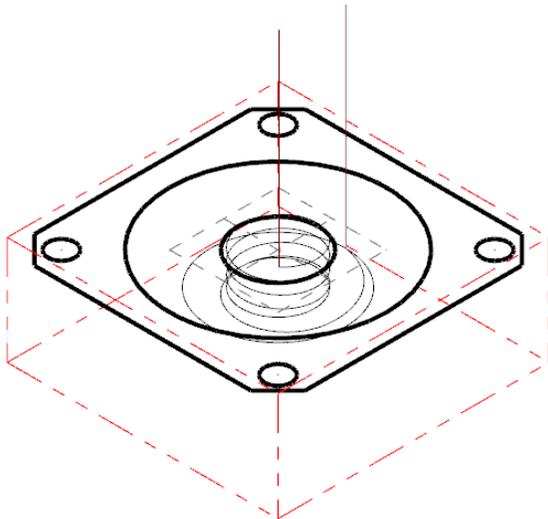
- ◆ To hide the dialog box, click on the **Hide dialog** icon as shown.



- ◆ 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.

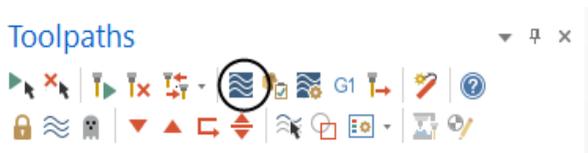
STEP 4: BACKPLOT THE TOOLPATHS

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.

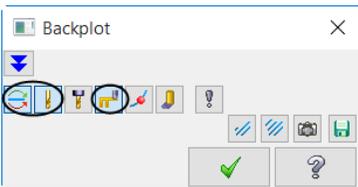
- ◆ Make sure that the toolpaths are selected (signified by the green check mark on the folder icon). If both operations are not selected, choose the **Select all operations** icon.



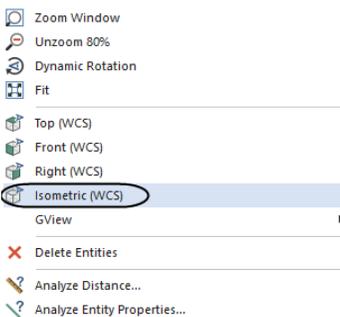
- ◆ Select the **Backplot selected operations** button.



- ◆ 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.



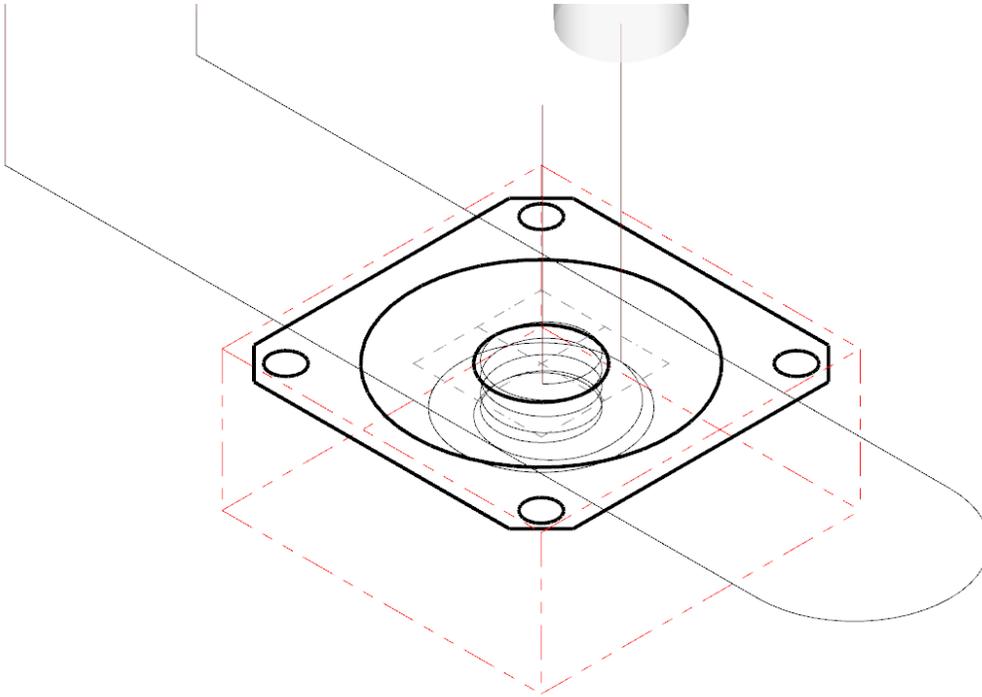
- ◆ To fit the workpiece to the screen, if needed, right mouse click in the graphics window and select **Fit**.

- ◆ You can step through the **Backplot** by using the **Step forward**  or **Step back**  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

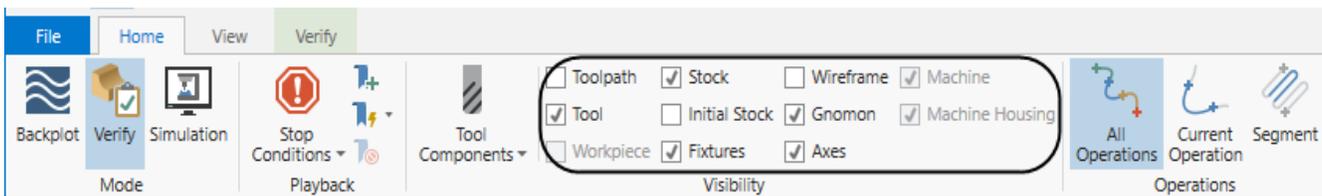
Verify shows the path the tools take to cut the part with material removal. This display lets you spot errors in the program before you machine the part. As you verify toolpaths, Mastercam displays additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates,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

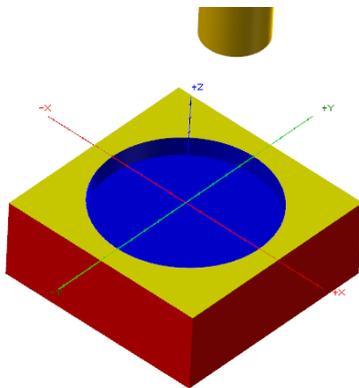
- ◆ In **Mastercam Simulator**, **Verify** should be enabled and change the settings as shown.



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



- ◆ The part should appear as shown.



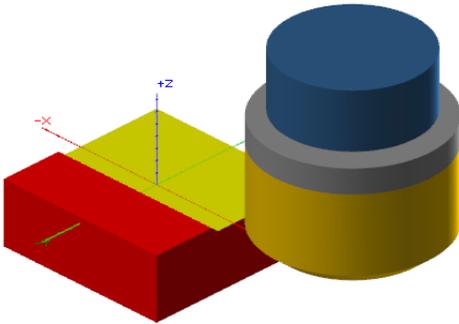
Note: To rotate the part, move the cursor to the center of the part and click and hold the mouse wheel and slowly move it in one direction. To zoom in or out, hold down the mouse wheel and scroll up or down as needed.

- ◆ 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.



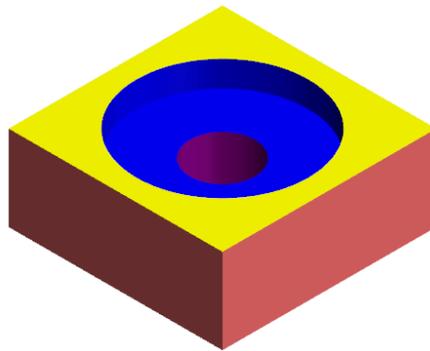
- ◆ Click on the **Step Forward** until the toolpath is completed.
- ◆ To go back to Mastercam window, minimize **Mastercam Simulator** window as shown.



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)



ORDER: 1.800.529.5517 • books@inhousesolutions.com • www.eMastercam.com



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

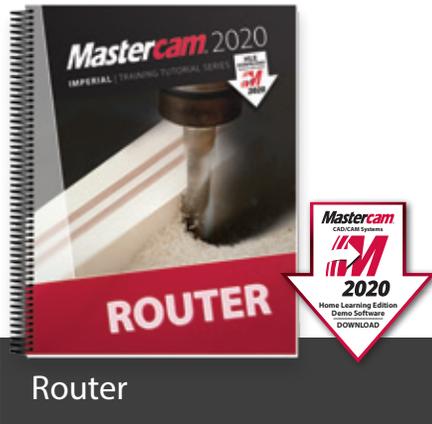
The Mastercam 2020 for SOLIDWORKS Programming Exercises book provides a comprehensive "hands on" method of learning Mastercam for SOLIDWORKS. You will learn how to program a variety of different parts that require most of the toolpath types available in Mastercam for SOLIDWORKS. Extensive emphasis is 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.

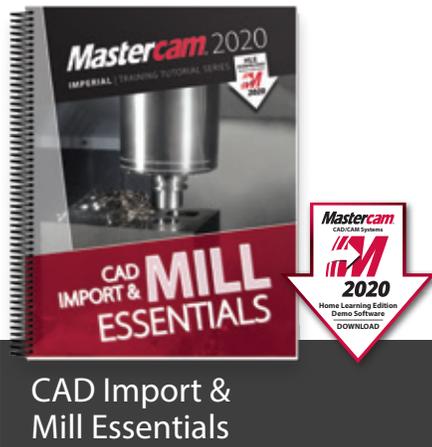
- 12 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

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.

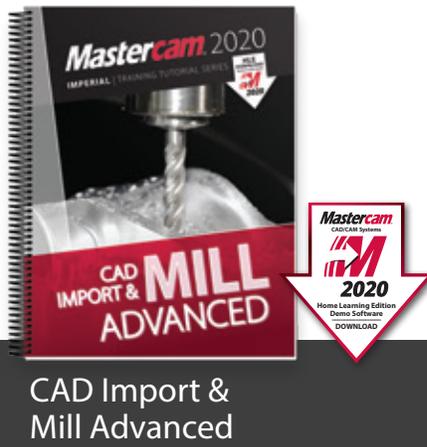
- 7 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

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

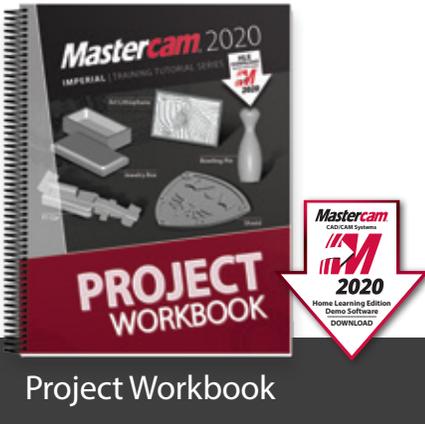
Price \$60 (Print)

ISBN: 978-1-77146-840-4

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**.



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

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.

ORDER: 1.800.529.5517 • books@inhousesolutions.com • www.eMastercam.com



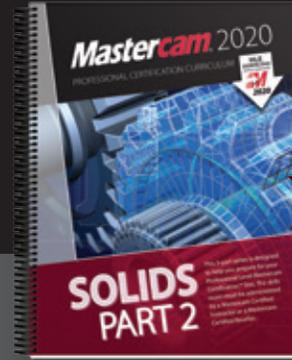
Part 1 –
Mill 2D

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.

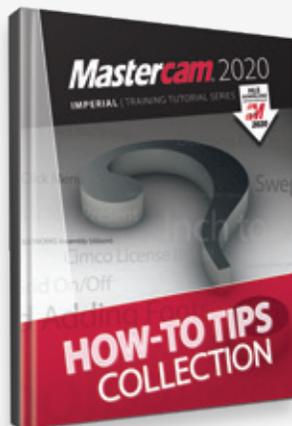
Price \$90 (eBook)

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.

ORDER: 1.800.529.5517 • books@inhousesolutions.com • 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)



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

