Mastercam® 2022

VIDEO ECOURSE PROMO CODE INCLUDED

MILL ESSENTIALS

TRAINING TUTORIAL SERIES | **IMPERIAL**

2022

HLE DOWNLOAD
INSTRUCTIONS INSIDE

Mastercam 2022

Mill Essentials Training Tutorial

To order more books:

Call 1-800-529-5517 or

Visit www.emastercam.com or

Contact your Mastercam dealer

Mastercam 2022 Mill Essentials Tutorial

Software: Mastercam 2022

Authors: Mariana Lendel

ISBN: 978-1-77146-943-2

Date: July 21, 2021

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

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 Tutorial **1**

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: Setting the attributes 22

Step 5: About Manager Panels 24

Step 6: Setting Mastercam to Imperial 26

Step 7: Setting the Grid 28

Conventions used in this book: 29

Mastercam® Workflow 30

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 39

Step 4: Create The 3/4" & 2.0" Diameter Circles 42

Step 5: Create The Chamfers 45

Step 6: Save The File 47

Tutorial #1 Review Exercise 48

Create The Geometry For Tutorial #1 Exercise 49

Tutorial 1: Toolpath Creation **51**

Suggested Fixture 53

Setup Sheet 54

Step 1: Select The Machine And Set Up The Stock 55

Step 2: Face The Part 58

Step 3: Circle Mill The Large hole 65

Step 4: Backplot The Toolpaths 71
Step 5: Simulate the toolpath in Verify 73
Step 6: Circle Mill The Inside hole 74
Step 7: spot drill the 0.25" holes 81
Step 8: Drill The 0.25" holes 88
Step 9: Chamfer The Large hole 93
Step 10: Machine the Corners using Contour toolpath 100
Step 11: Post The File 107
Step 12: Save The Updated MCAM File 108
Create The Toolpaths For Tutorial #1 Exercise 109
Tutorial 2: Geometry Creation 111
Tutorial #2 Drawing 113
Step 1: Setting Up The Graphical User Interface 114
Step 2: Create A Rectangle 115
Step 3: Create Two Obround Shapes 117
Step 4: Create A Circle 120
Step 5: Use Divide To Clean The Circle 121
Step 6: Create Parallel Lines 123
Step 7: use divide delete to clean up the geometry 126
Step 8: create angular lines 128
Step 9: Create A polygon 131
Step 10: create fillets 133
Step 11: Rotate the Part 138
Step 12: Create the Solid 140
Step 13: Create the stock using bounding box 143
Step 14: Save The File 145
Tutorial #2 Review Exercise 146
Create The Geometry For Tutorial #2 Exercise 147
Tutorial 2: Toolpath Creation 149
Toolpath Creation - Setup #1 151

Suggested Fixture 151
Setup Sheet 151
Step 1: Open the Vise and Merge the part 152
Step 2: Set the part Origin 158
Step 3: Use Levels Manager to make the vise invisible 159
Step 4: Select The Machine And Set Up The Stock 160
Step 5: Slot Milling 163
Step 6: Backplot The Toolpaths 171
Step 7: Simulate the toolpath in Verify 172
Step 8: Machine the Cutout pockets 174
Step 9: Finish the Inside shapes - Dynamic contour 182
Step 10: Rough the outside using High Speed Dynamic Mill 189
Step 11: Finish the Outside profile using contour toolpath 196
Step 12: Create a Stock Model 200
Toolpath Creation - Setup 2 203
setup Sheet: 203
Step 13: Creating And Renaming Toolpath Groups 204
Step 14: Create and Set WCS To bottom 206
Step 15: Merge the soft jaw vise 210
Step 16: Use Translate to Align the soft jaw vise To The Part 213
Step 17: Make The Vise Invisible 216
Step 18: Face The Part 217
Step 19: Rename The NC File 223
Step 20: Post The File 224
Step 21: Save The Updated MCAM File 226
Create The Toolpaths For Tutorial #2 Exercise 227
Tutorial 3: Geometry Creation 235
Tutorial #3 Drawing 237
Step 1: Setting Up The Graphical User Interface 238
Step 2: Create Two Arcs 238

Step 3: Create A Vertical Line 247
Step 4: Create arc using Arc Polar Endpoints 249
Step 5: Rotate The Geometry 251
Step 6: Mirror Geometry 253
Step 7: Create Tangent Arcs 255
Step 8: Trim geometry 257
Step 9: Break and delete the small circle at quadrant point 259
Step 10: Mirror Geometry to complete arms 262
Step 11: Join the half arcs 264
Step 12: Create A Construction Line 265
Step 13: Create a 0.5" Diameter Circle 266
Step 14: Delete Construction Geometry 268
Step 15: Create Tangent Lines 269
Step 16: Create an Arc Polar 272
Step 17: Create Fillets 274
Step 18: Trim the arc 276
Step 19: Rotate 277
Step 20: Translate 279
Step 21: Change The Main Level To 2 283
Step 22: Create The Solid Body By Extruding A Closed Chain 283
Step 23: Extrude Cut The Pockets and the Holes 291
Step 24: Chamfer The Part 298
Step 25: Save The File 302
Tutorial #3 Review Exercise 303
Create The Geometry For Tutorial #3 Exercise 304
Create The Solid Geometry For Tutorial #3 Exercise 304
Tutorial 3: Toolpath Creation 307
Suggested Fixture 309
Step 1: Select The Machine And Set Up The Stock 311
Step 2: 2D High Speed Dynamic Mill 314

Step 3: Backplot The Toolpaths 322
Step 4: Simulate the toolpath in Verify 324
Step 5: Finish the walls using Contour toolpath 325
Step 6: Area Mill Toolpath 330
Step 7: Transform-Rotate Toolpath 338
Step 8: Finish the pocket walls using Contour toolpath 341
Step 9: Chamfer Drill to Spot Drill The Holes 346
Step 10: Drill all Holes 350
Step 11: Chamfer The Outside Diameter 354
Step 12: Chamfer The Pockets 359
Setup Sheet 2: 365
Step 13: Creating And Renaming Toolpath Groups 366
Step 14: set The WCS To Bottom 367
Step 15: 2D HS Dynamic Mill 370
Step 16: Finish the Pocket wall using Contour toolpath 377
Step 17: Rename The NC File 382
Step 18: Post The File 383
Step 19: Save The Updated MCAM File 384
Create The Toolpaths For Tutorial #3 Exercise 385
Tutorial 4: Geometry Import 391
Step 1: Setting Up The Graphical User Interface 392
Step 2: Importing the Solidworks File geometry 392
Step 3: Save The File 394
Tutorial #4 Review Exercise 395
Tutorial 4: Toolpath Creation 399
Suggested Fixture 401
Setup Sheet 1 402
Step 1: Select The Machine And Set Up The Stock 403
Step 2: 2D High Speed Area Mill 405
Step 3: Backplot The Toolpaths 411

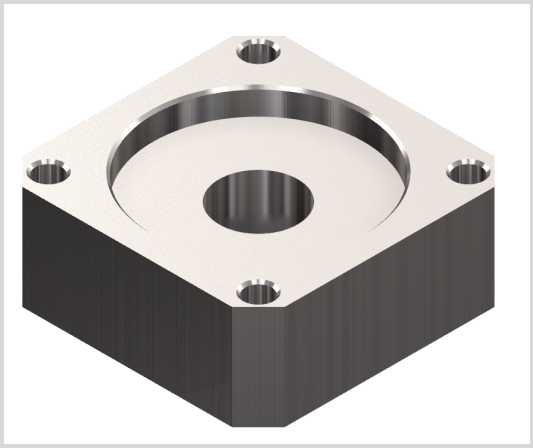
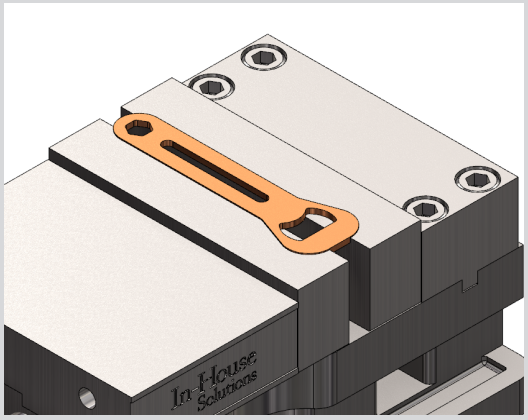
Step 4: Simulate the toolpath in Verify 412
Step 5: 2D High Speed Area Mill 412
Step 6: Remachine the remaining material using Area mill 422
Step 7: Drill all Holes 429
Step 8: Chamfer Drill The Holes 434
Step 9: Tap The Holes 437
Suggested Fixture 2: 442
Setup Sheet 2: 443
Step 10: Creating And Renaming Toolpath Groups 444
Step 11: set WCS To Front 445
Step 12: Chamfer Drill The Holes 448
Step 13: Drill The Two 3/8" Tap Holes 452
Step 14: Tap The Two Holes 455
Step 15: Drill The 5/8 -11 Tap Hole 459
Step 16: Create a 1/4" Thread Mill 462
Step 17: Set the Thread Mill Cut Parameters 466
Step 18: Rename The NC File 469
Setup Sheet 3: 470
Step 19: Creating And Renaming Toolpath Groups 471
Step 20: set The WCS To Left Side 472
Step 21: Machine The Slot 473
Step 22: Rename The NC File 478
Step 23: Post The File 479
Step 24: Save The Updated MCAM File 480
Create The Toolpaths For Tutorial #4 Exercise 481
Tutorial 5: Geometry Creation 495
Tutorial #5 Drawing 497
Step 1: Setting Up The Graphical User Interface 498
Step 2: Open Tutorial #5 Wireframe 498
Step 3: Create the Solid Body 498

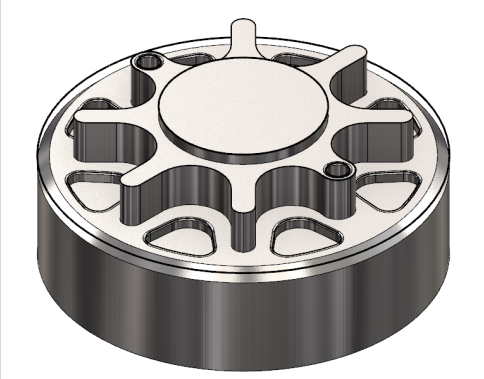
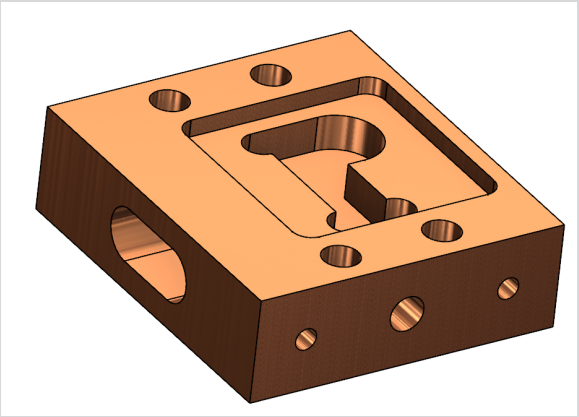
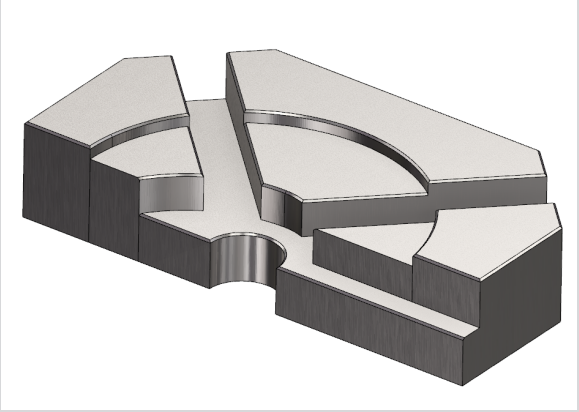
Step 4: Extrude Cut the solid Body at 0.125" 501
Step 5: Extrude Cut the solid with a 0.5" Distance 502
Step 6: Move The Solid and the Rectangle on level 2 503
Step 7: Chamfer the solid 505
Step 8: Save The File 508
Tutorial #5 Review Exercise 509
Create The Geometry For Tutorial #5 Exercise 510
Tutorial 5: Toolpath Creation 513
Suggested Fixture 515
Setup Sheet 515
Step 1: Select The Machine And Set Up The Stock 516
Step 2: 2D High Speed Dynamic Mill 518
Step 3: Backplot The Toolpath 524
Step 4: Simulate the toolpath in Verify 524
Step 5: 2D High Speed Area Mill 525
Step 6: 2D High Speed Blend Mill 533
Step 7: 2D High Speed Peel Mill 539
Step 8: 2D High Speed Peel Mill 544
Step 9: Use Model Chamfer toolpath to chamfer the solid 549
Step 10: Post The File 554
Step 11: Save The Updated MCAM File 555
Create The Toolpaths For Tutorial #5 Exercise 556
Tutorial 6: Geometry Creation 561
Tutorial #6 Drawing 563
Step 1: Setting Up The Graphical User Interface 564
Step 2: Create The Circles 564
Step 3: Create A Line Tangent 565
Step 4: Create A Line Parallel 567
Step 5: Trim The Entities 568
Step 6: Create Rectangular Shape 570

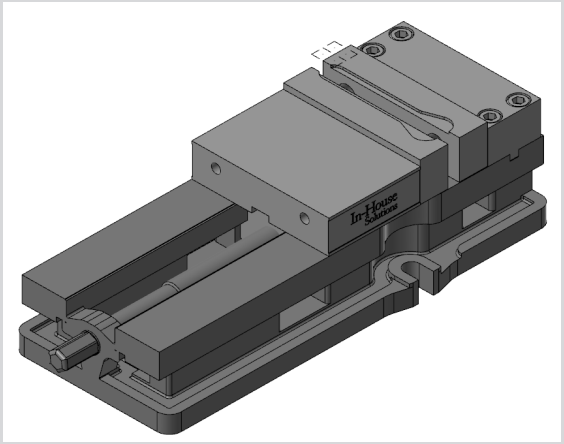
Step 7: Trim Divide 572
Step 8: Fillet Chains 574
Step 9: Create the Obround Shapes 576
Step 10: Create Circle Center Point 578
Step 11: Set the Solid Level and Color 581
Step 12: Extrude The Base Of The Solid 583
Step 13: Chamfer The Holes 591
Step 14: Fillet the Edges 593
Step 15: Create the Letters 595
Step 16: Save The File 598
Tutorial #6 Review Exercise 599
Create The Geometry For Tutorial #6 Exercise 600
Create The Solid Geometry For Tutorial #6 Exercise 600
Create The Letters For Tutorial #6 Exercise 601
Tutorial 6: Toolpath Creation 603
Setup Sheet 605
Step 1: Select The Machine And Set Up The Stock 606
Step 2: 2D High Speed Dynamic Mill 609
Step 3: Backplot The Toolpaths 616
Step 4: Simulate the toolpath in Verify 617
Step 5: Drill the holes using FBM Drill 618
Step 6: Machine the outside of the part using Dynamic Milling 625
Step 7: Finish the pockets using a Pocket toolpath 633
Step 8: Finish the Outside Profile - Contour Toolpath 638
Step 9: Engrave the letters using Contour toolpath 646
Step 10: Post The File 651
Step 11: Save The Updated MCAM File 652
Create The Toolpaths For Tutorial #6 Exercise 653
Tutorial 7: Soft Jaw Vise 659
Create A Soft jaws Vise for Fixture 661

Step 1: Modify the jaws using Push-Pull command 661
Step 2: Move the movable jaw using Dynamic command 664
Step 3: Merge the solid part 667
Step 4: Remove the Solid History 668
Step 5: Remove the Cutouts Using Modify Feature 669
Step 6: Align the part Using Transform Dynamic 670
Step 7: Duplicate the solid part 674
Step 8: Boolean remove the solid from the jaws 675
Step 9: Move the soft jaw vise to Level 1000 678
Creating/Editing A Mill Tool 687
Create And Editing A Mill Tool 688
Step 1: Create A New Tool 688
Step 2: Editing An Existing Tool 695
Quiz Answers 699
Mill Essentials Tutorial Quiz Answers 700

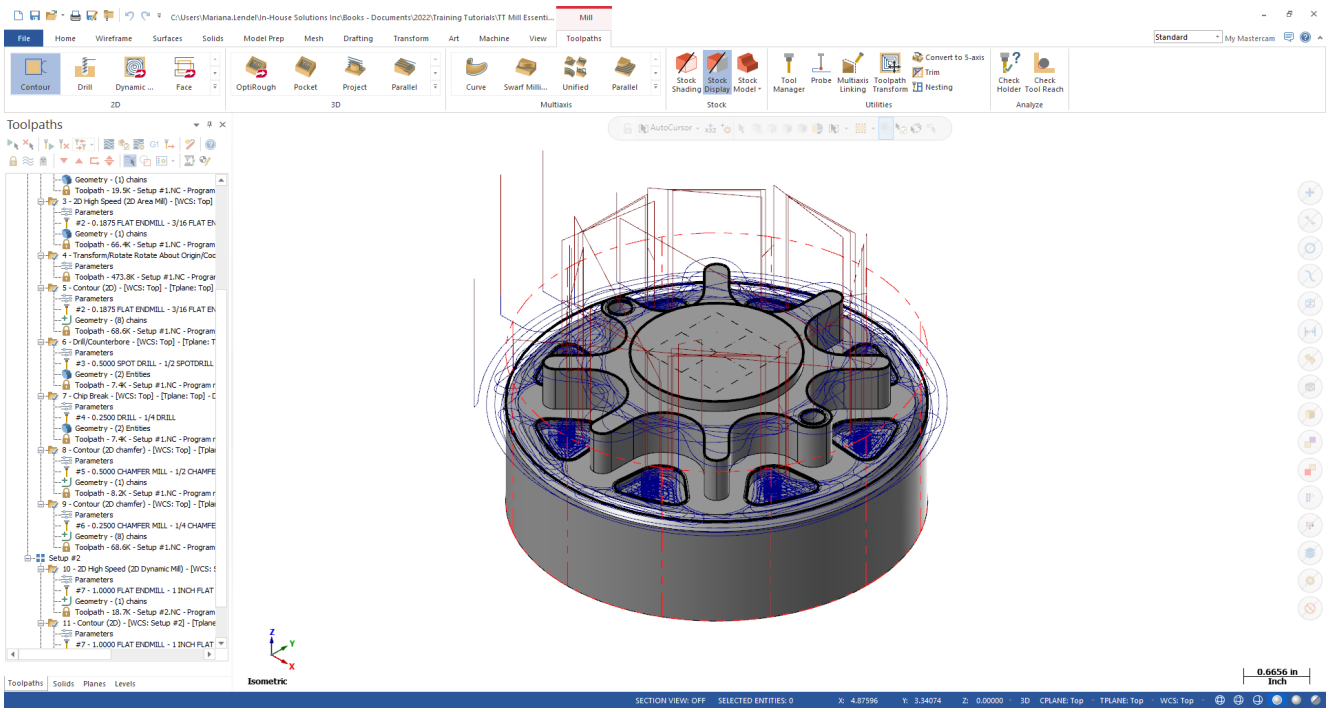
Mill Essentials Projects

Tutorial	Geometry Functions	Toolpath Creation
#1 	Rectangle. Circle Center Point. Chamfer Entities.	Facing Toolpath. Circle Mill Toolpath. Contour Toolpath. Spot Drill Toolpath. Drill Toolpath. 2D Contour (Chamfer Toolpath).
#2 	Rectangle. Rectangular Shapes. Polygon. Fillet Entities. Fillet Chains. Line Endpoints. Trim Divide. Solid Extrude. Bounding Box.	Setup 1 Open Vise and Merge Part. Slot Mill Toolpath. Pocket Toolpath. 2D HS Dynamic Contour Toolpath. 2D HS Dynamic Mill Toolpath. Contour Toolpath. Setup 2 Toolpath Groups. Create and set a new WCS. Merge Soft Jaw Vise. Align the Soft Jaw Vise to the Part. Facing Toolpaths. Stock Model.

Tutorial	Geometry Functions	Toolpath Creation
#3 	Circle Center Point. Line Vertical. Arc Polar Endpoints. Rotate. Mirror. Arc Tangent. Trim. Break two Pieces Fillets. Translate. Solids Extrude. Solid Chamfer.	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. Contour Toolpath.
#4 	Import a SolidWorks file. Translate 3D.	Setup 1 - Top Tool Planes. 2D HS Area Mill Toolpath. 2D HS Area Mill Rest Toolpath. Drill Toolpath. Setup 2 - Front Tool Plane. Drill Toolpath. Thread Mill Toolpath. Setup 3 - Left Tool Plane. Slot Mill Toolpath.
#5 	Solid Extrude Create Body. Solid Extrude Cut Body. Solid Chamfer.	2D HS Dynamic Mill Toolpath. 2D HS Area Mill Toolpath. 2D HS Blend Mill Toolpath. 2D HS Peel Mill Toolpath. Model Chamfer Toolpath.

Tutorial	Geometry Functions	Toolpath Creation
#6 	Circle Center Point. Line Tangent. Line Parallel. Rectangular Shapes. Trim. Fillet Chains. Solids Extrude. Solids Chamfer. Solids Fillet. Create Letters	2D HS Dynamic Mill Toolpath. Feature Based Drilling Toolpath. 2D HS Area Mill Toolpath. Pocket Toolpath. 2D Contour Toolpath. 2D Contour Toolpath (Engrave).
#7 	Push-Pull. Transform Dynamic . Merge/Pattern. Remove History Modify Feature. Align it in the vise. Duplicate Solid. Boolean Remove. Move the vise to Level 1000.	

Getting Started



OBJECTIVES

- Starting Mastercam
- The student will learn about the Graphical User Interface.
- The student will learn how to navigate through Mastercam.
- Setting the System Configuration to Imperial.
- Setting the Grid.
- Conventions used in the book.
- Mastercam Workflow.

STEP 1: STARTING MASTERCAM

1.1 For Windows 7

- Select the **Start** button.
- Select **All Programs** and click on Mastercam 2022.

1.2 For Windows 8

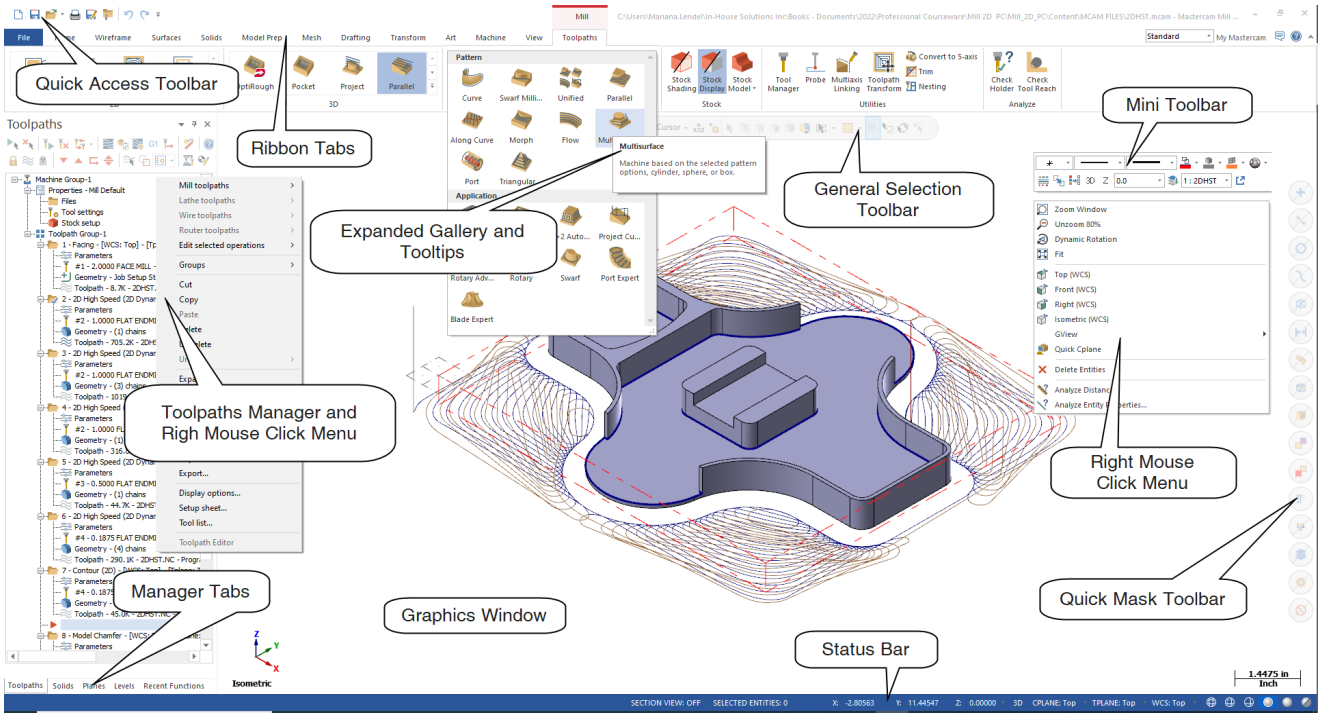
- Select the **Start** button.
- Click on the drop down arrow to open Apps.
- Find and click on Mastercam 2022.

1.3 For Windows 10

- Select the **Start** button.
- Click on the drop down arrow to open Apps.
- Find and click on Mastercam 2022.

- To start the software, from Desktop, click on the shortcut icon as shown.

STEP 2: GUI - GRAPHICAL USER INTERFACE



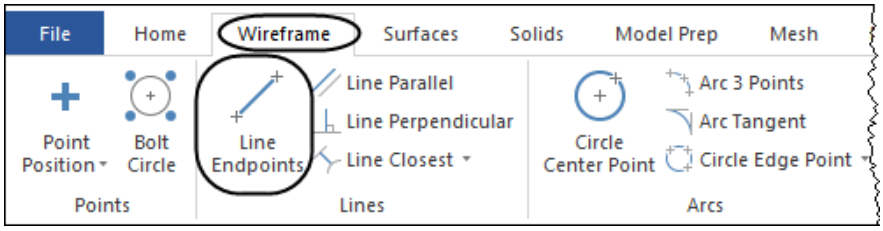
Quick Access Toolbar	**QAT** contains a fully customizable set of functions that can be quickly accessed by the user.
Backstage (File)	Allows you to manage files. You can insert information about files, start a new file, open an existing one or merge files together. You can also save, convert or print files as well as access the help resources.
Tabs	Contains all the functionality within Mastercam.
Ribbon	Displays the commands available for a selected Tab.
Selection Bar	Allows you to set the **AutoCursor** modes and to switch between wireframe or solid selections.
Quick Mask Buttons	Lets you select all entities of a specific type. Clicking on the left side of the button or right side of the button toggles between select all or only.
Right Click Menu	Right click menu allows quick access to functions such as zoom, graphic views or recent functions used. A mini toolbar will also appear that allows you to quickly change the attributes.
Toolpaths/Solids/Planes Manager	Lists the history of the toolpath operations and solids.
Graphics Window	Workspace area in Mastercam where the geometry is displayed.
Scale	Shows you a scale of the object on the screen.
WCS: TOP T/Cplane:	Displays the current **WCS** and **T/Cplane** information.

STEP 3: NAVIGATE THROUGH MASTERCAM

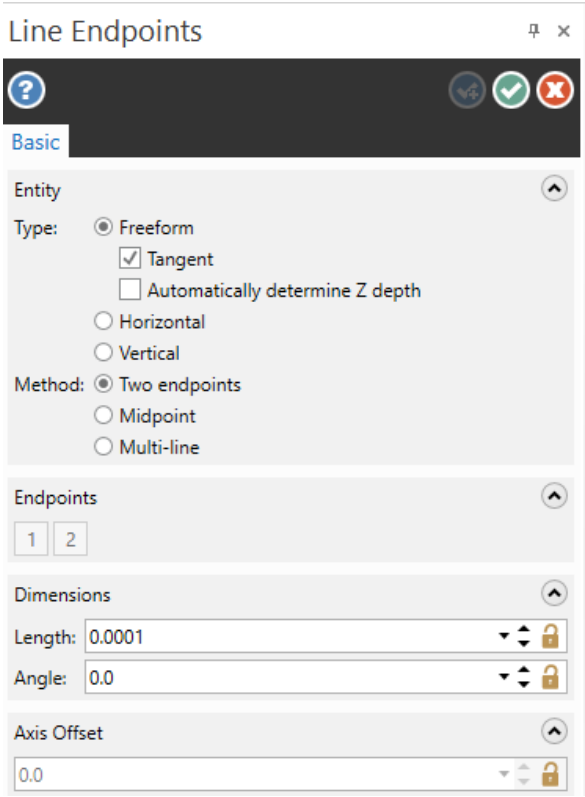
In this step, you will learn how to use the menu functions in Mastercam to create geometry.

Start Line Endpoints command

- Select the **Wireframe** tab (left click).
- Left click on the **Line Endpoints** icon as shown.



- Once you select **Line Endpoints**, the **Line Endpoints** panel appears on the screen as shown.



Function Prompt

Prompts the user to execute a command.

Sketching a line

- To sketch a line, left click on two locations on the screen between which the line will be generated.

Creating a line knowing the endpoint coordinates

- To make a line knowing the two endpoint coordinates, select the **AutoCursor Fast Point** icon from the **General Selection** toolbar.

- In the coordinates field that opens in the upper left corner enter the coordinates of the first endpoint as shown.
- Press **Enter** to continue.

Note: Enter the X value then the Y value and if needed the Z value separated by comma (,) .

- Select the **AutoCursor Fast Point** icon again and enter in the coordinates of the second endpoint and then press **Enter**.

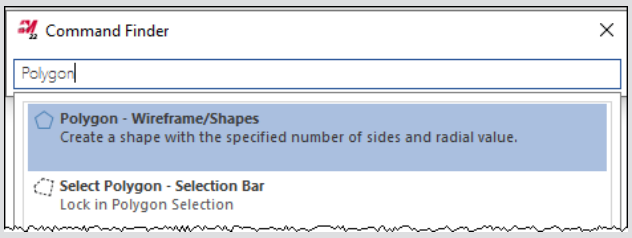
Note: You do not need click on the **AutoCursor Fast Point** icon. Once Mastercam promp you to enteran endpoint, you can just start typing the values.

Creating a line knowing an endpoint, the length, and the angle

- You can also enter the coordinates of the first endpoint, then enter the **Length** and **Angle** if necessary.
- To continue making lines, choose the **OK and Create New Operation** button from the dialog box or press **Enter**. 
- To exit the current command, select the **OK** button or press the **Esc** button. 
- To undo the last command, from the **QAT** (**Q**uick **A**ccess **T**oolbar) select the **Undo** button.  The Undo button can be used to go back to the beginning of geometry creation or to the last point of the saved file. Mastercam also has a **Redo** button for your convenience. 

Example: this prompt is used in the **Line Endpoints** command. Specify the first endpoint

Note: To find a command, from the **Home** ribbon, select the **Command Finder** icon and type the function name in the field that opens up.
For example, to find the **Polygon** command, type "polygon" in the text field. From the list, select the desired command.



STEP 4: SETTING THE ATTRIBUTES

Mastercam attributes are point style, line style, line thickness, color and levels. Before starting to create geometry, you should set the attributes.

4.1 Attributes Group

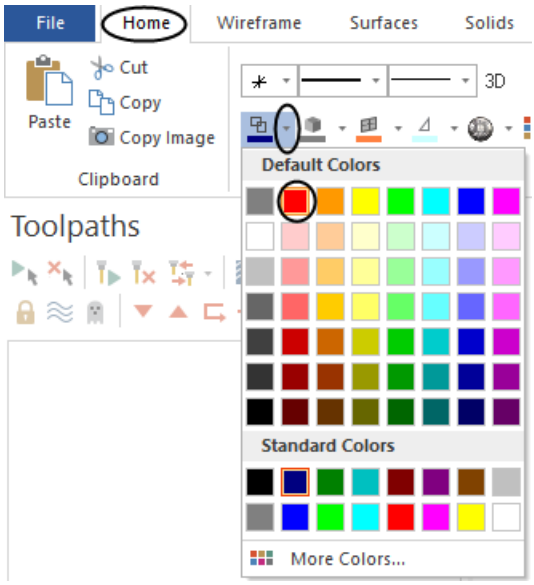
Point Style	Displays and sets the system's point style.
Line Style	Displays and sets the system's line style.
Line Width	Displays and sets the current system's line width.
Color	Assigns the current color to wireframe, solid and surface entities. To change the current color, click in the specific color field and select a color from the color palette. To change an existing geometry color, select the geometry first and then click in the color field and select a color from the color palette.
Clear Color	When performing a transform function (Transform), Mastercam creates a temporary group from the originals (red) and a result (purple) from the transformed entities. These system groups appear in the Groups dialog box. However, they stay in effect only until you use the Clear Colors function or perform another transform function.
2D / 3D Construction Mode	Toggles between 2D and 3D construction modes. In 2D mode, all geometry is created parallel to the current Cplane at the current system Z depth. In 3D mode, you can work freely in various Z depths, unconstrained by the current system Z depth and Cplane setting.

4.2 Organize Group

Z Depth	Sets the current construction depth. To set this, click the drop down arrow and pick one from the most recently used list or click the Z: label and pick a point in the graphics window to use the Z depth values based on the selected entity.
Level	Sets the main level you want to work with in the graphics window. To change the current working level. Type the level number in the box.

Set the Wireframe Color

- In the **Home** tab, **Attributes** group, click on the drop down arrow next to the **Wireframe Color** field as shown.
- Select the desired color from the dialog box as shown.



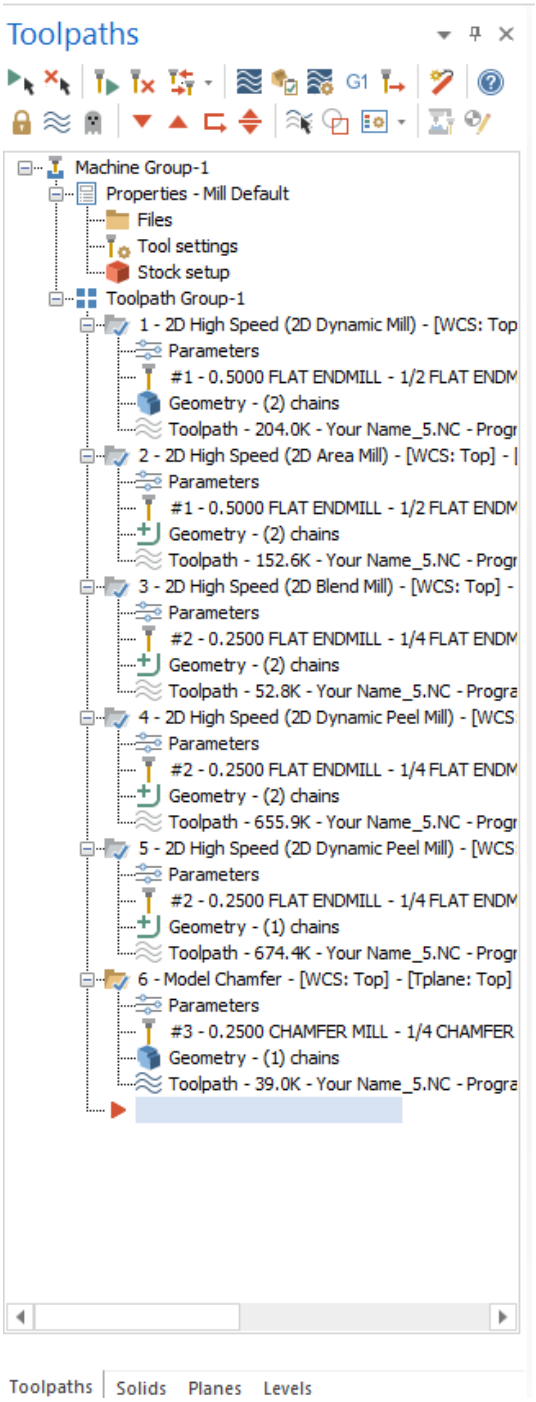
Note: Any geometry on your screen will remain in the previous system color. This change will only affect the geometry you create going forward.

To change the color of existing geometry, select the entities first and then click on the drop down arrow next to the Wireframe Color and select the desired color. The same method can be applied for any other attribute that you want to set or change.

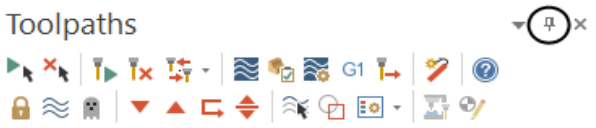
STEP 5: ABOUT MANAGER PANELS

5.1 The Toolpaths Manager

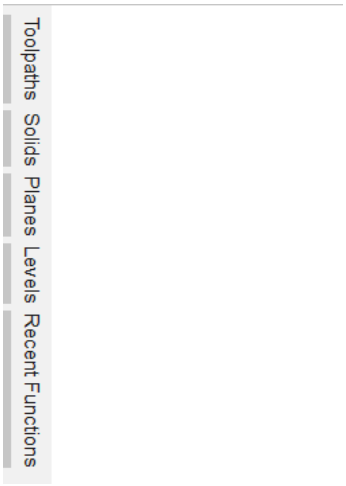
The **Toolpaths Manager** displays all the operations for the current part. You can sort, edit, regenerate, verify and post any operation as shown. For more information on the **Toolpaths Manager**, please click on the **Help** icon.



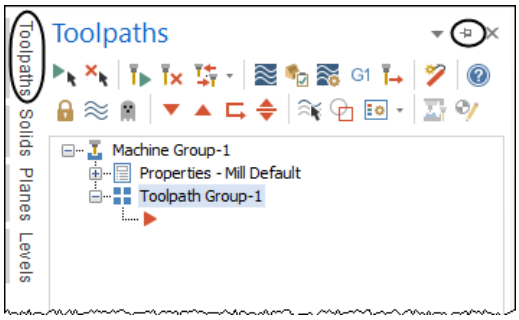
- The **Toolpaths Manager**, **Solids Manager**, or **Planes Manager** can be hidden to gain more space in the graphics area for creating geometry. Use **Auto Hide** icon to close all **Toolpaths**, **Solids**, **Planes** and **Levels Manager** panels.



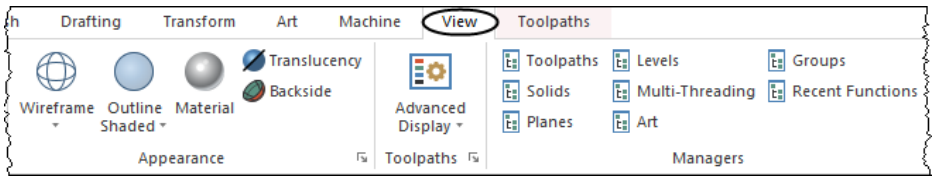
- The panels will be hidden to the left of the graphics window as shown or at the bottom of the manager as shown previously.



- To un-hide them, click on one of the managers to open it and then click again on the **Auto Hide** icon a shown.
- Selecting the **X** (**Close** icon) instead of the **Auto Hide** will close the manager panel.



- To re-open them, from the **View** tab, select **Toolpaths**, **Solids**, **Planes** or **Levels** as shown.



STEP 6: SETTING MASTERCAM TO IMPERIAL

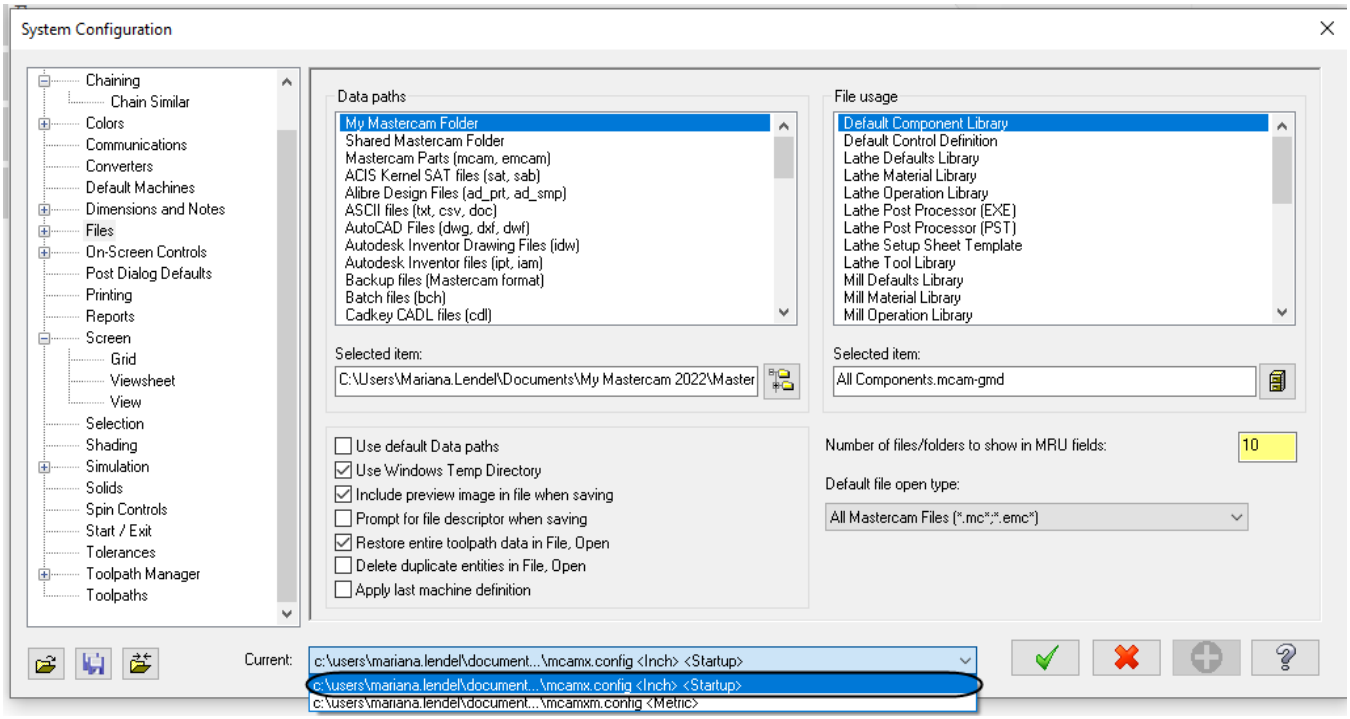
In this step you will learn how to set the imperial system as your default. You will have to select the **Backstage** options and select the system configuration.

6.1 Setting Mastercam to inch for the current session only

Note: You may need to switch Mastercam to run in Inch mode.

File

- **Configuration.**
  - Select the drop down arrow beside **Current** as shown.
  - Select **mcamx.config <Inch>** as shown.



- Select the **OK** button to exit the **System Configuration** dialog box.

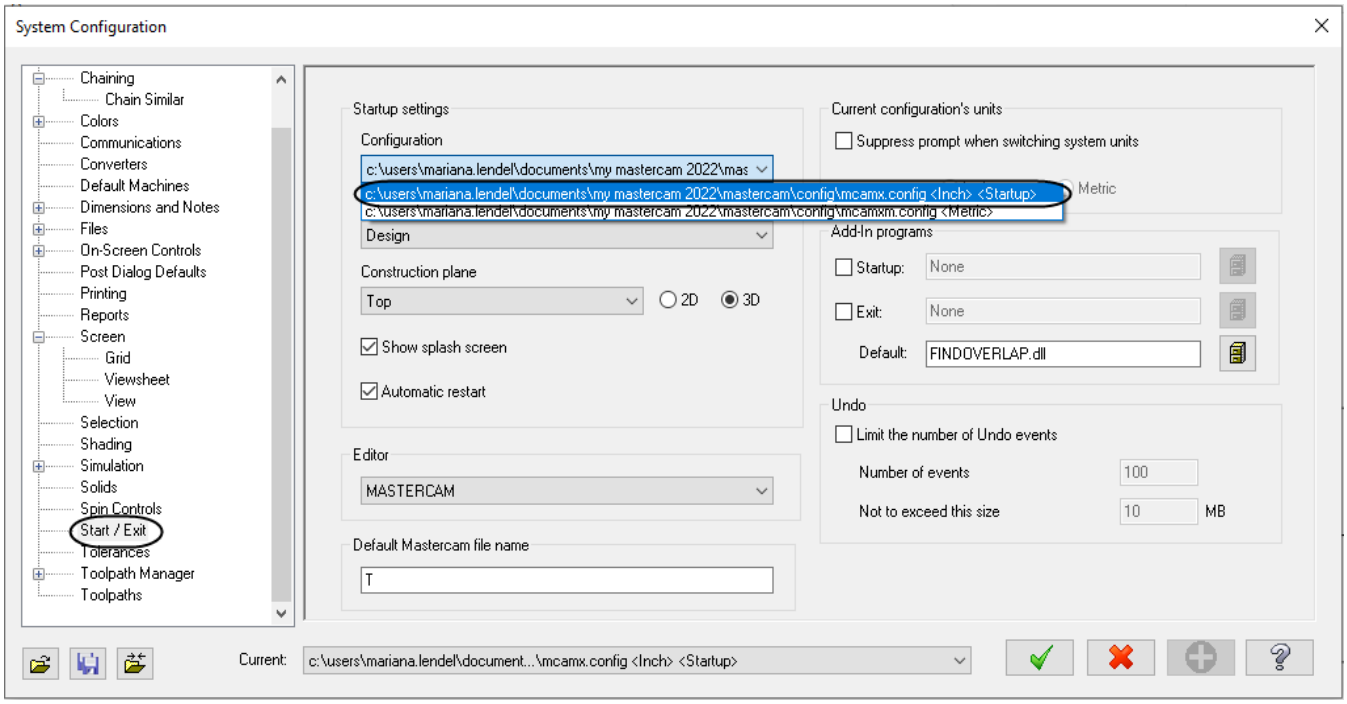
Note: If you have open a drawing done in metric on the screen it may ask you to scale the current part to imperial. Choose Yes if you wish to do this.

6.2 Setting Mastercam to Imperial as a default

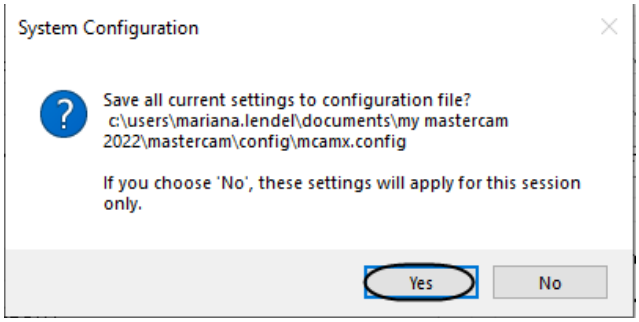
Note: If you wish to always work in Imperial mode, follow these steps to save Imperial as your current configuration file.

File

- **Configuration.**
  - Select **Start/Exit** from the configuration topics.
  - Select the drop down arrow below **Configuration** in the **Startup** settings area as shown
  - Select **mcamx.config <Inch>** as shown.



- Select the **OK** button to exit the **System Configuration** dialog box.
- Mastercam will then prompt you to save these settings to your current configuration file, select **Yes**.

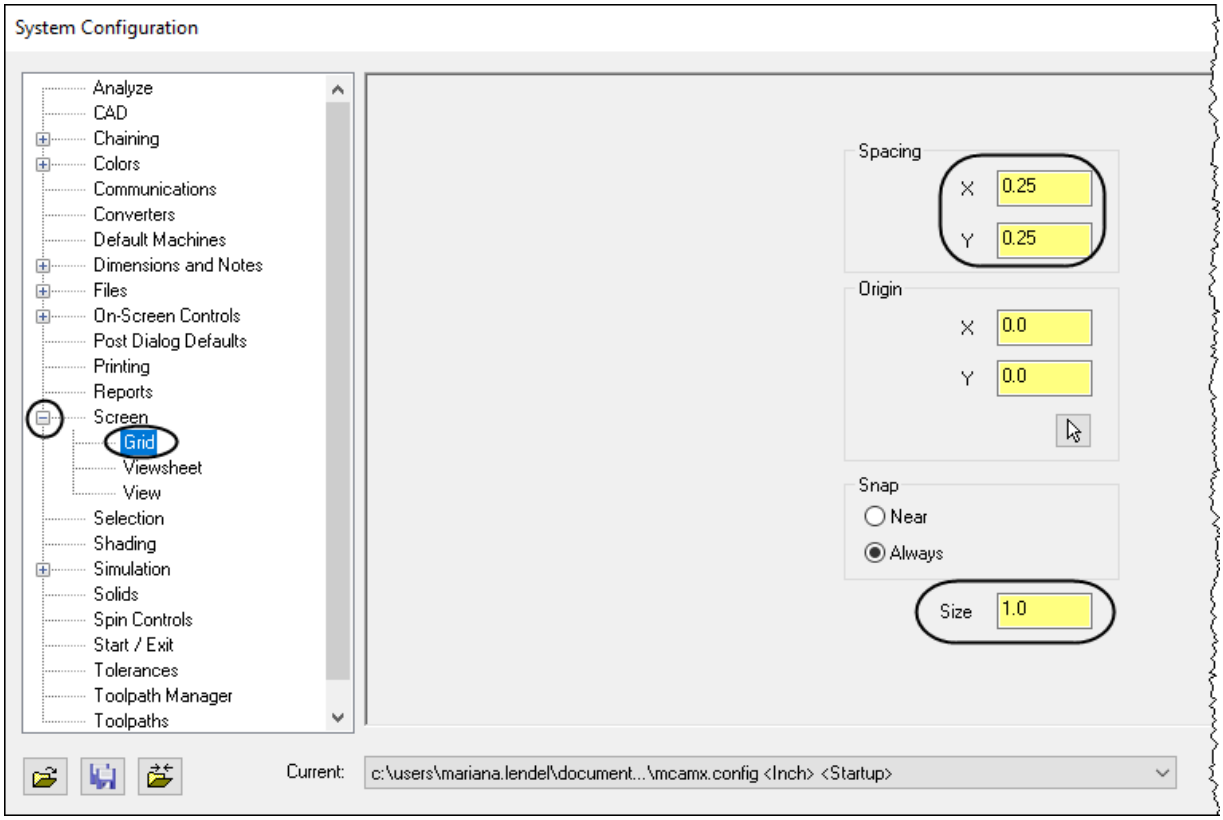


STEP 7: SETTING THE GRID

Before beginning to create geometry, it is highly recommended to enable the Grid. The Grid will show you where the origin is and the orientation of the Grid gives you a quick preview of the plane you are working in.

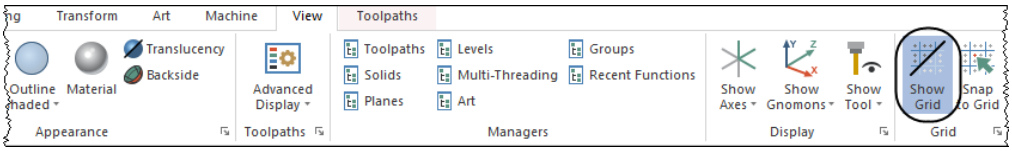
File

- **Configuration.**
  - Select **Screen** from the configuration **Topics**.
  - Select the plus sign (**+**) beside **Screen** as shown.



  - In **Grid Setting**s, change the **Spacing** to **X = 0.25** and **Y = 0.25**.
  - Set the **Size** to **1.0**.
  - Choose the **OK** button to exit. 
  - Select the **Yes** button to save the settings in the **System Configuration**.

- To see the **Grid** in the graphics window, from the **View** tab, enable **Show Grid** as shown.



- The grid should look as shown.

CONVENTIONS USED IN THIS BOOK:

We have attempted to make this manual as uncluttered as possible and provide you with reference information when it is appropriate. It is not intended to be a Reference Guide or all-encompassing user manual.

The Text Styles Used Are The Following:

Standard Text - Represents normal wording needed to provide you the instructions.

STEP 8: STEP TITLES

8.1 Sub step titles

Information about the current step, terms or parameter definitions describing the parameters and description.

Bold Text - Represents menu commands, dialog box settings or other similar items from the screen.

Note: Represents information about the process/step that is important or may require an explanation.

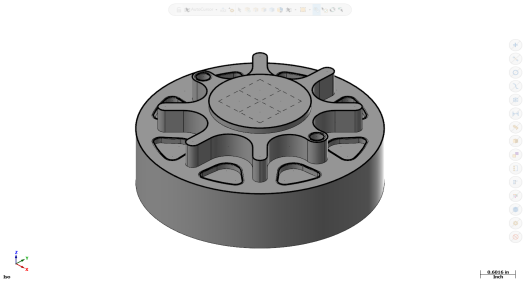
- Bulleted text are step by step instructions that are to be followed.

The files used in this book are available for download at http://www.emastercam.com/files/.

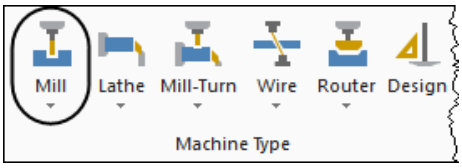
MASTERCAM® WORKFLOW

The process to create or import the geometry and to generate a toolpath will be repeated over and over through the tutorials in this book. You will find the process simple and straightforward once you have programmed a few parts. The following is an outline of the process we will follow to create programs:

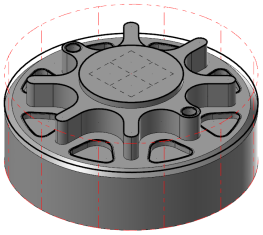
1. Create or import the part geometry.



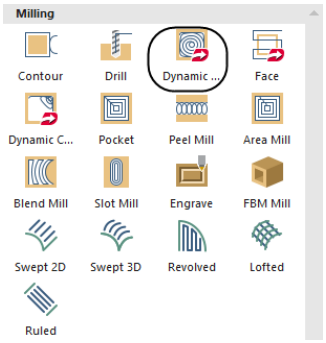
2. Select the Machine type.



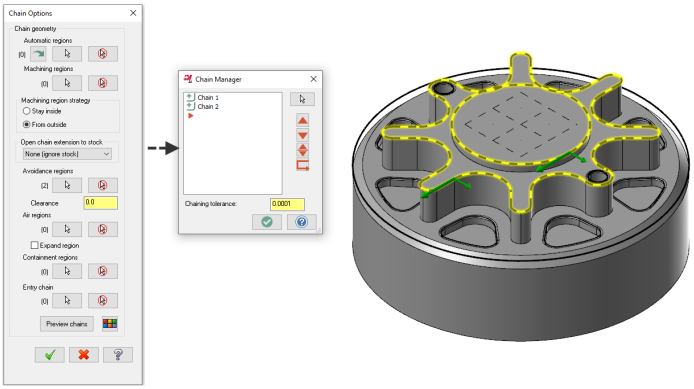
3. Define the stock size that your part will be cut from and set tool information.



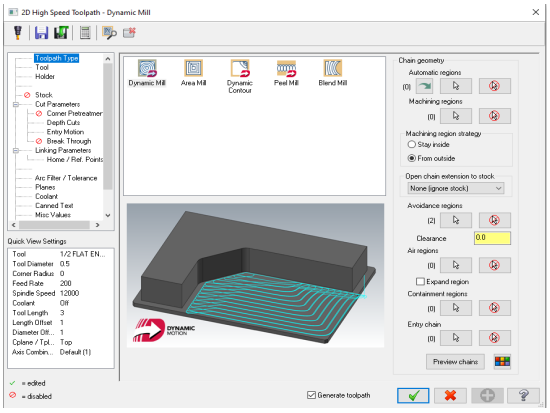
4. Select a toolpath type such as 2D High Speed Dynamic.



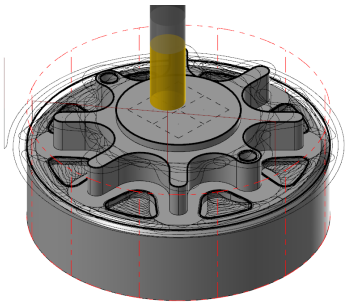
5. Select the geometry of the part you will cut with the different selection options.



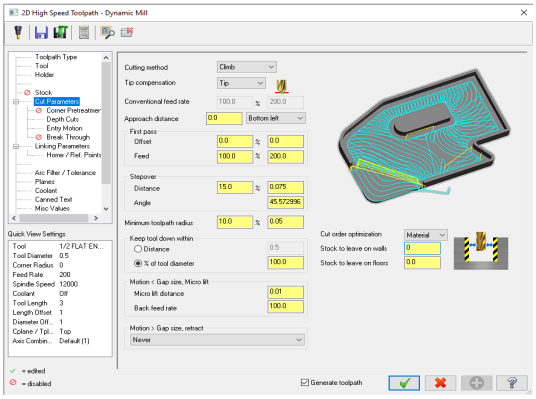
6. Fill in the necessary information on the parameter pages that appear for the toolpath type you selected.



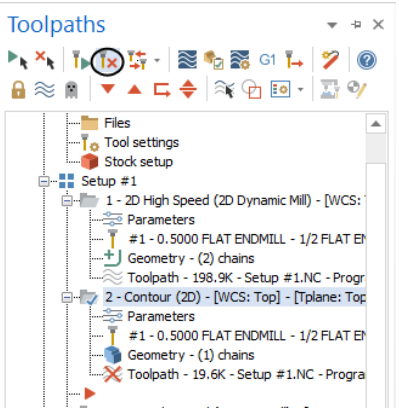
7. Verify the toolpath on your computer screen to confirm the results are as you expected, using Backplot and/or Solid Verify.



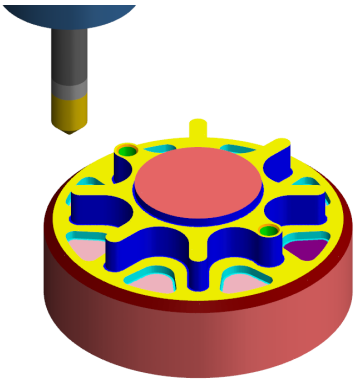
8. Make any changes as required by changing parameters.



9. If the **Generate toolpath** is selected in the toolpath parameters, you can skip this step as the toolpath will be automatically updated. Otherwise, **Regenerate** the "**Dirty**" operation to update the parameter changes.

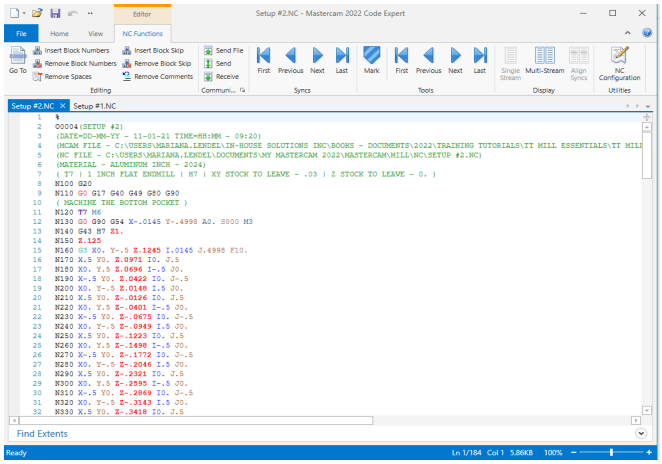


10. Verify again to make sure the toolpath is correct.

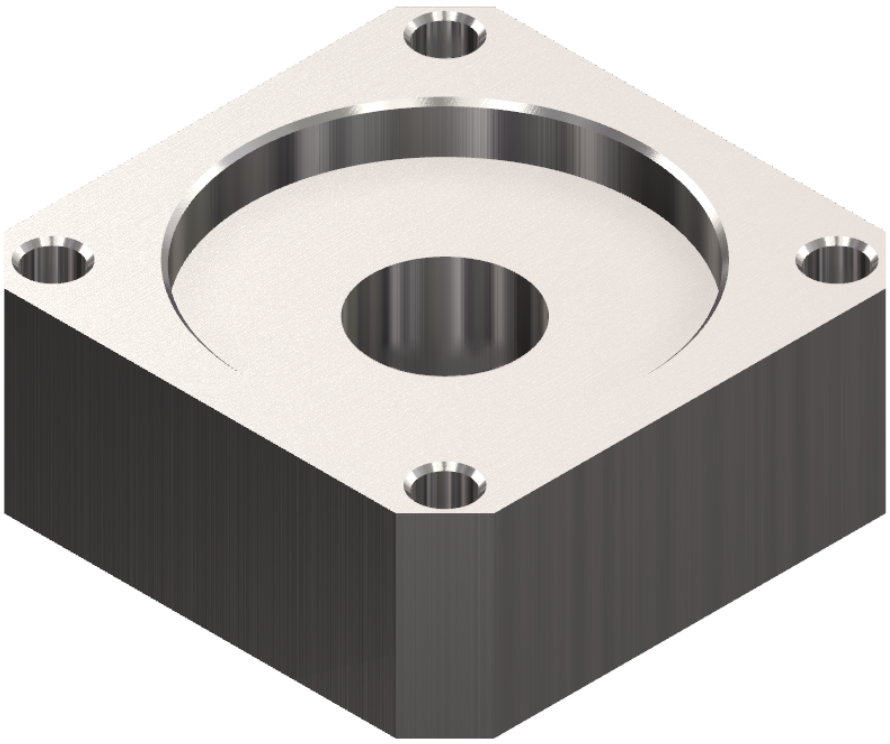


11. Convert the graphical toolpath information into machine code by Post Processing and sending it to the CNC machine.

Note: Mastercam HLE does not support post processing.



Tutorial 1:
Geometry Creation



Overview Of Steps Taken To Create The Part Geometry:

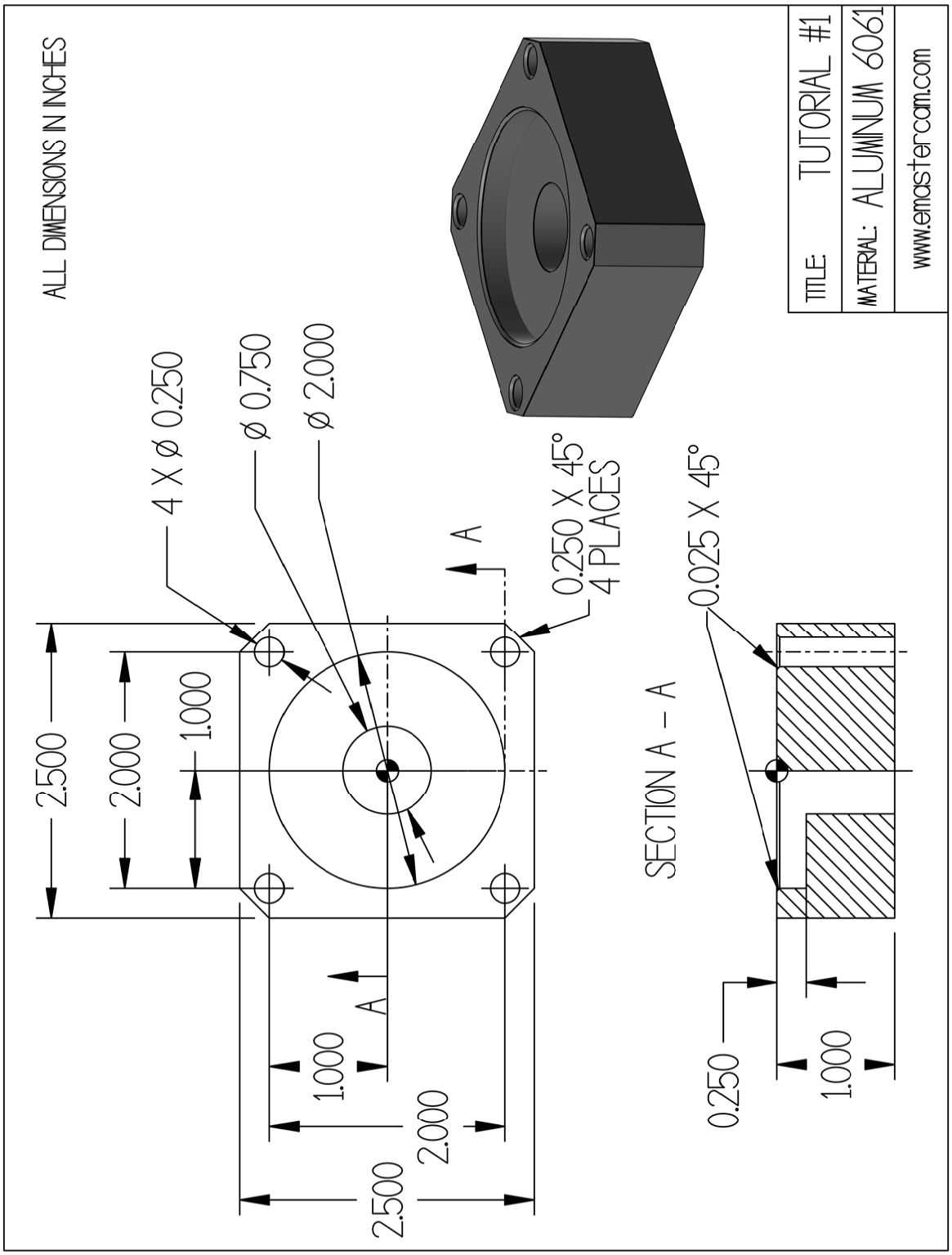
From Drawing to CAD Model:

- 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.

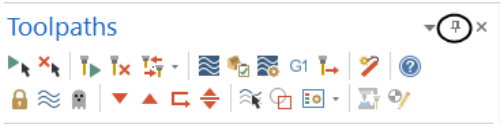
TUTORIAL #1 DRAWING



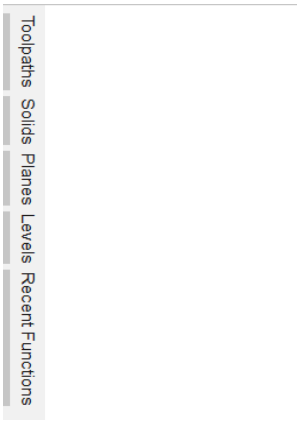
STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

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.

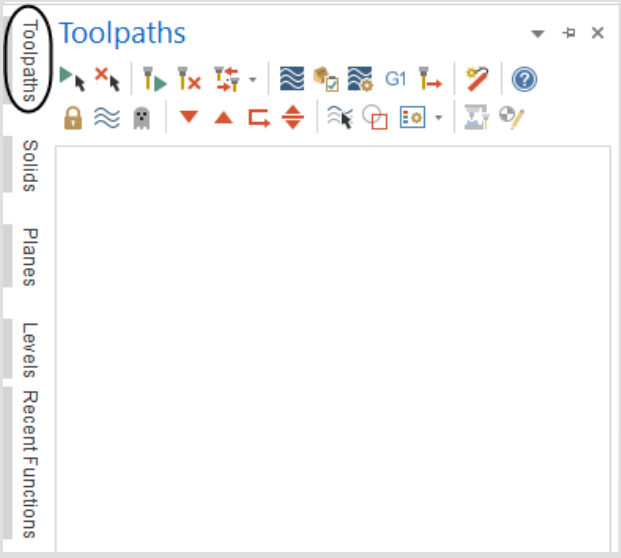
- Use the **Auto Hide** icon to hide all **Manager** panels.



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



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

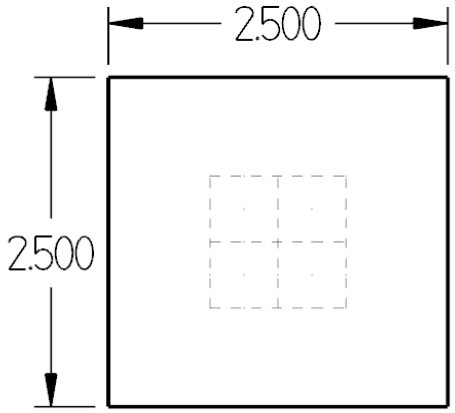


While creating the geometry, keep the Manager panels hidden. This ensures more space in the graphics window for the geometry.

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 anchored to the Origin.

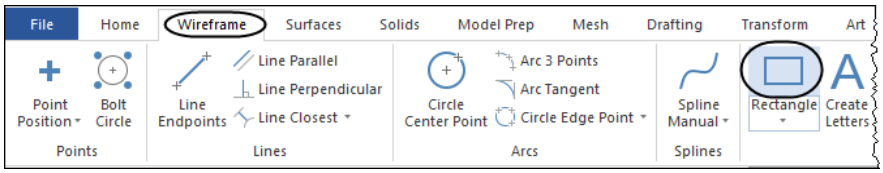
Step Preview:



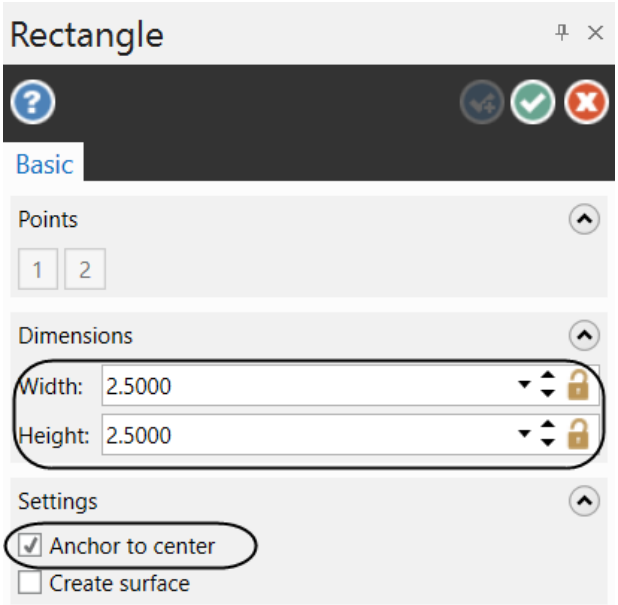
2.1 Create a 2.5" by 2.5" Rectangle

Wireframe

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

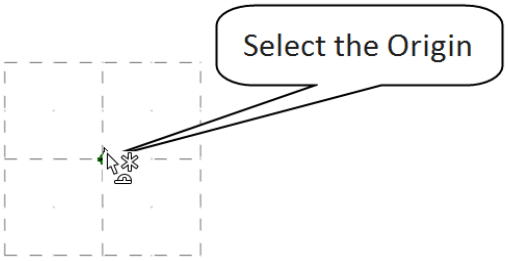


- In the **Rectangle** panel, enter the **Width** and **Height** and enable **Anchor to center** as shown.

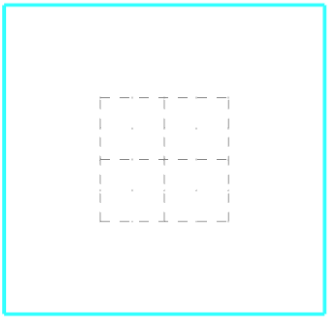


Note: Make sure that **Create surface** is not selected. **Anchor to center** sets the base point of the rectangle to its center and draws the rectangle outward from the center. **Create surface** creates a surface inside of the rectangle. Surface creation and Surface toolpath are covered in Mill Advanced.

- Select the position of the base point as shown.



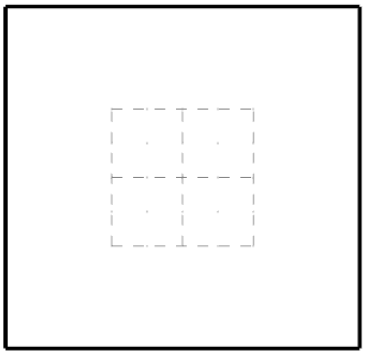
- A preview of the geometry should look as shown.



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

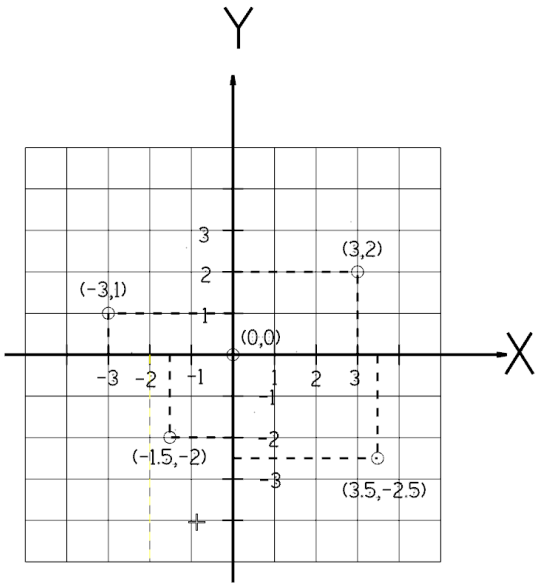
- The geometry should look as shown.



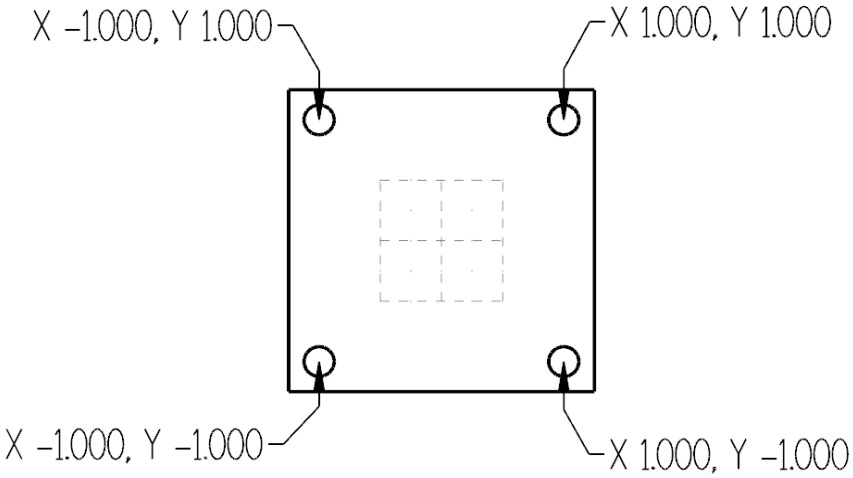
Note: While creating geometry for this tutorial, if you make a mistake, you can undo the last step using the **Undo** icon. You can undo as many steps as needed. If you delete or undo a step by mistake, just use the **Redo** icon. To delete unwanted geometry, select the geometry first and then press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

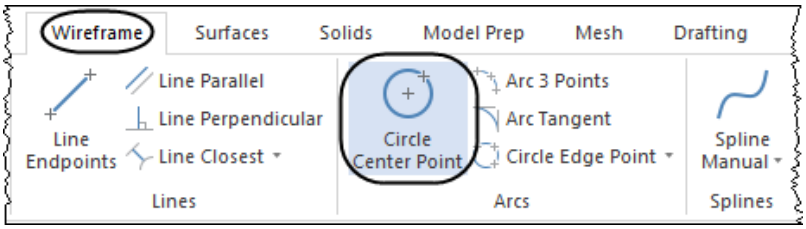


Step Preview:

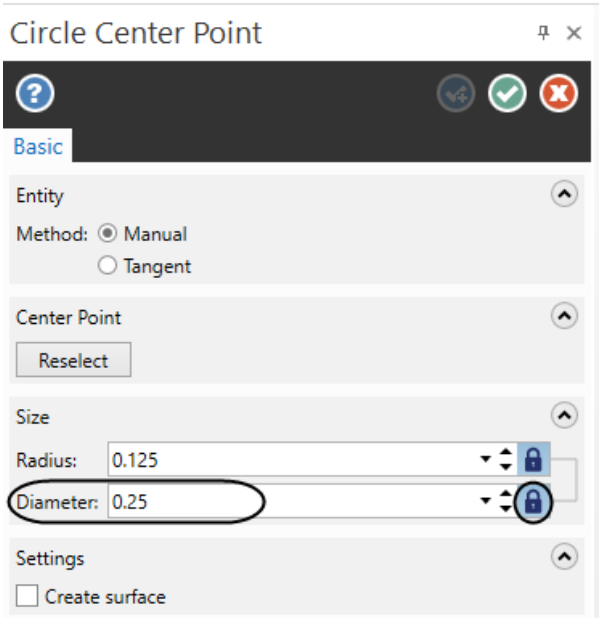


Wireframe

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

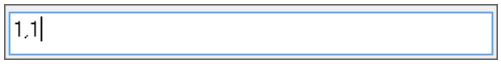


- Enter a **Diameter** of **0.25** in the panel as shown.
- To create all four circles, click on the lock icon to lock the value.



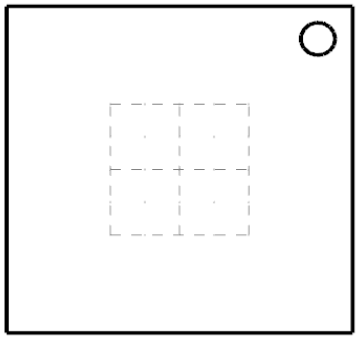
- [Enter the center point]: Select the **AutoCursor Fast Point** icon from the **General Selection** toolbar and the field where you can type the coordinates will open at the upper left side of the graphics window as shown.

- Type **1, 1** as shown.



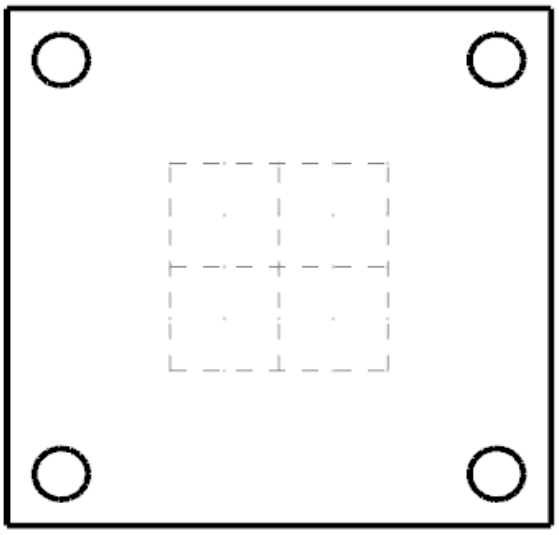
Note: When entering the coordinates for the center point, the first value is the **X** coordinate value, then the **Y** value followed by the **Z** value only if it is different from zero. The coordinate values are separated with commas. You do not need to use the coordinate labels if you enter the values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

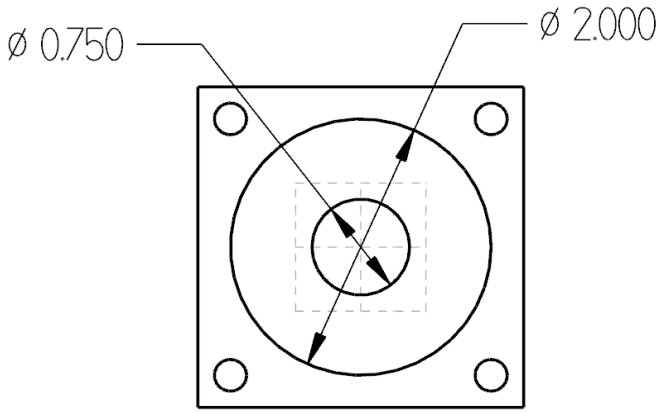
- The geometry should look as shown.



STEP 4: CREATE THE 3/4" & 2.0" DIAMETER CIRCLES

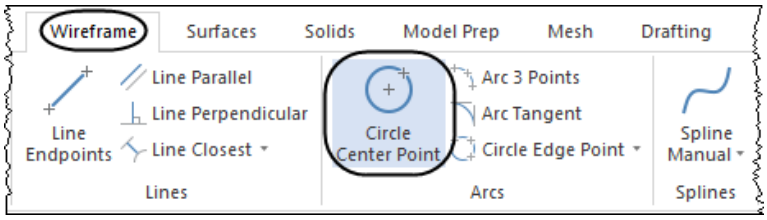
In this step, you will use the same **Circle Center Point** command to create circles for which you know the diameters and the locations.

Step Preview:

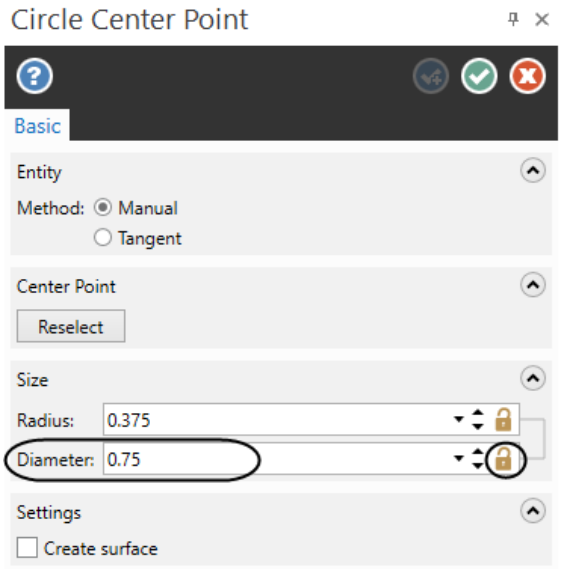


Wireframe

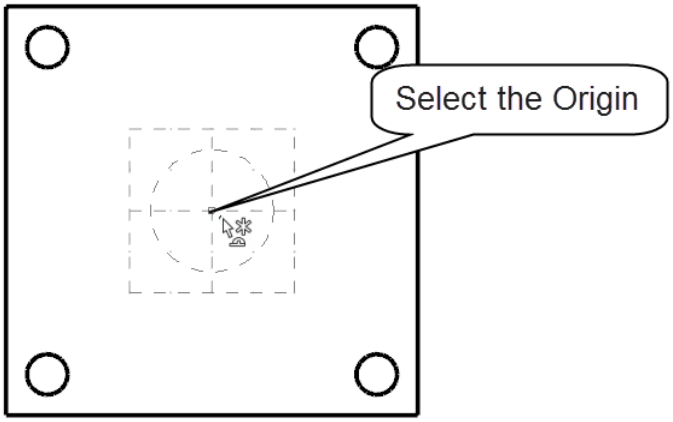
- From **Arcs** group, select **Circle Center Point**.



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



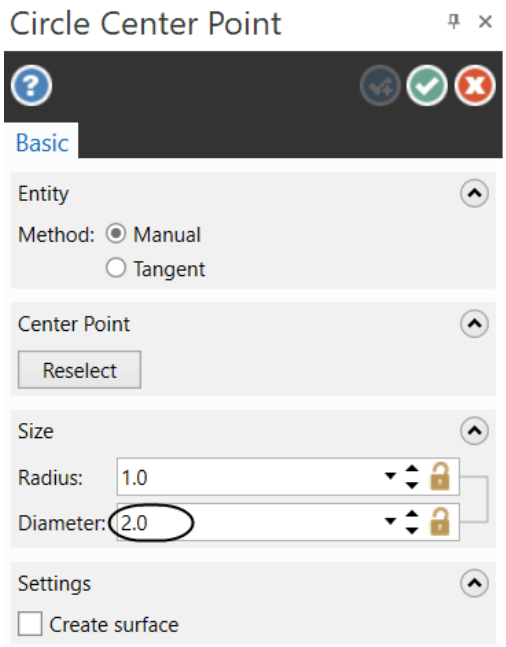
- [Enter the center point]: Move the cursor to the center of the rectangle until the cursor cue tip changes to the Origin as shown.
- Click to select the Origin.



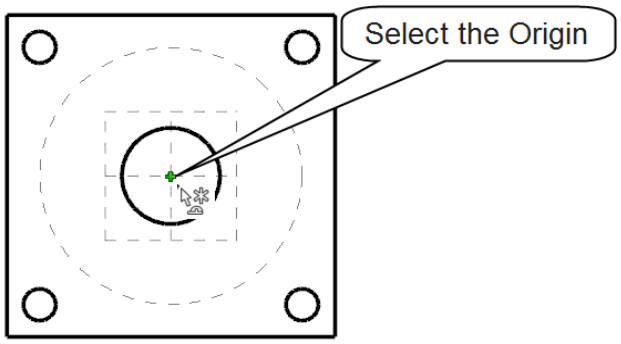
- Press **Enter** to see the circle preview.
- Press **Enter** again to finish the circle.

Note: While the circle is live, cyan color, the circle diameter and its location can be modified. To avoid this, you need to press **Enter** to finish the circle.

- In the **Diameter** field of the **Circle Center Point** panel, type **2.0** and press **Enter**.
- The panel should look as shown.



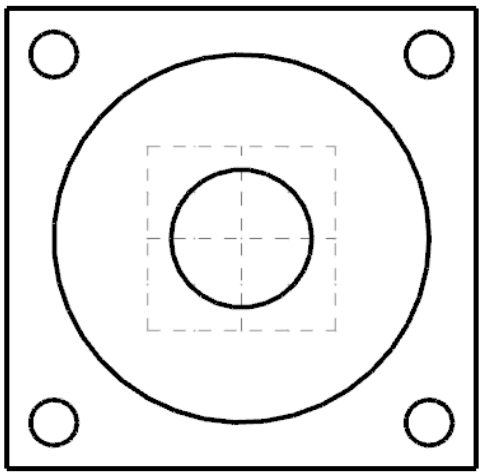
- [Enter the center point]: Select the Origin as shown.



Note: Because the center of the 0.75" diameter circle is in the Origin, you could also select the point when the cursor center cue tip appears as shown.

- Once complete, choose the **OK** button to exit the command. 

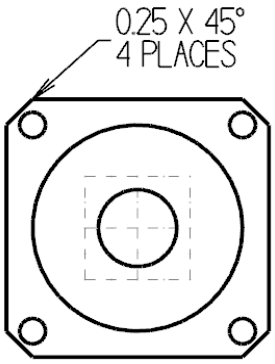
- The geometry should look as shown.



STEP 5: CREATE THE CHAMFERS

In this step, you will create 45 degree chamfers at the corners of the rectangle. You will use the **Chamfer Entities** command.

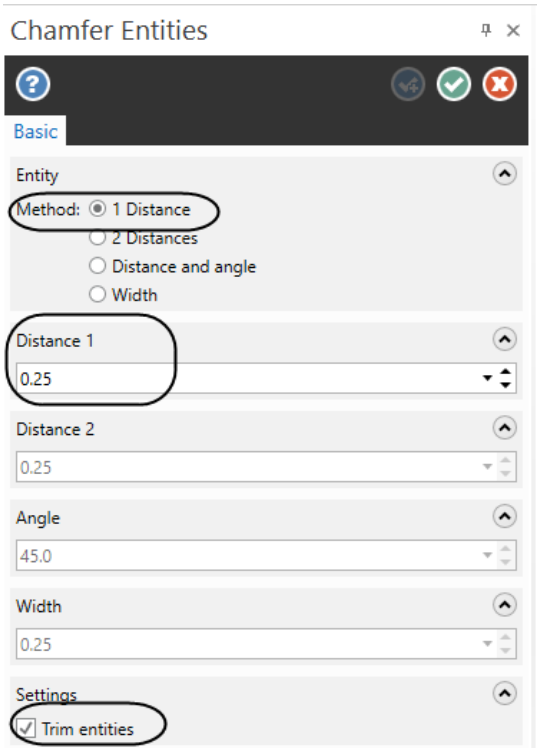
Step Preview:



Wireframe

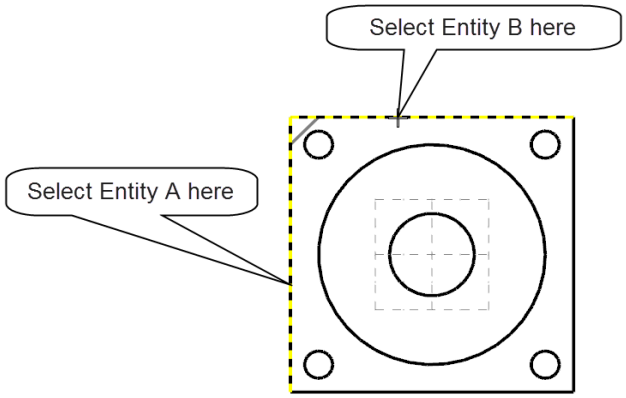
- From the **Modify** group, select **Chamfer Entities**.

- In the **Chamfer Entities** panel, make sure that **1 Distance** and **Trim entities** are enabled and **Distance 1** is set to **0.25** as shown.

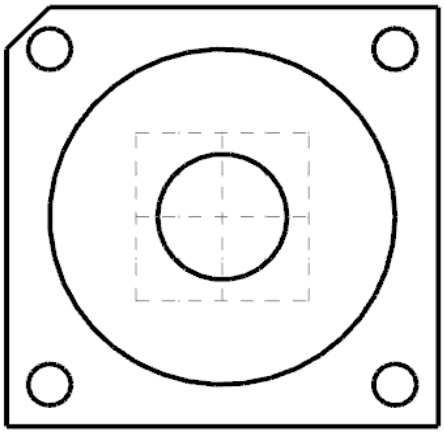


- Select the lines as shown.

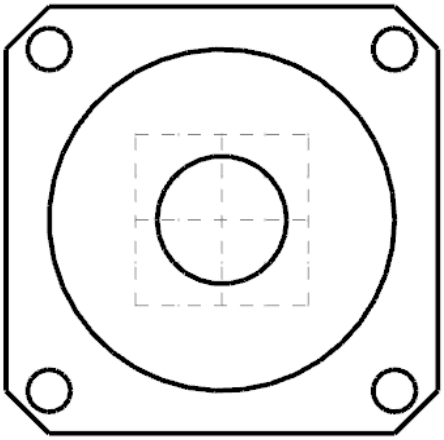
Note: A preview of the chamfer should appear when you hover the cursor above the second line (Entity B).



- The geometry should look as shown.



- Follow the same steps to chamfer the rest of the corners.
- The geometry should look as shown when completed.



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