

Mastercam® 2020

IMPERIAL | TRAINING TUTORIAL SERIES

HLE
DOWNLOAD
INSTRUCTIONS INSIDE



2020



WIRE

Mastercam 2020

WIRE 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 Wire Training Tutorial

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

Software: Mastercam 2020

Authors: Mariana Lendel

ISBN: 978-1-77146-840-4

Date: September 5, 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.

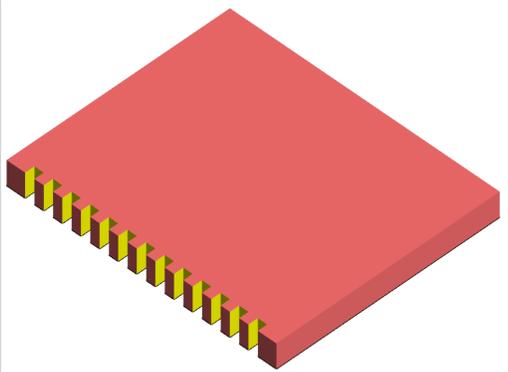
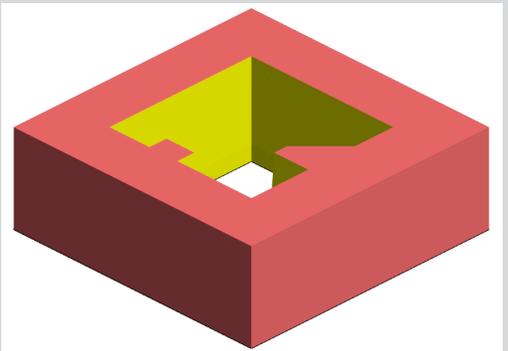
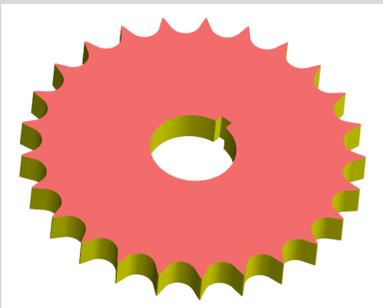
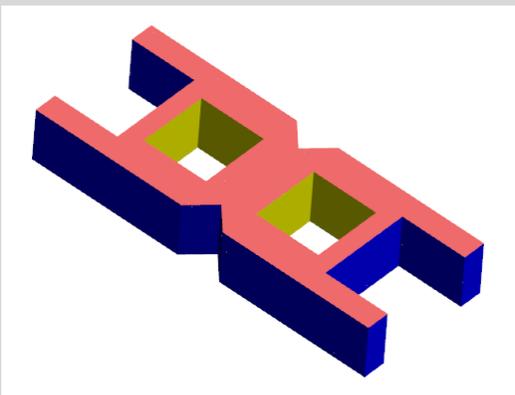
Trademarks

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.

Wire Projects

| Tutorial | Geometry Functions | Toolpath Creation |
|---|---|--|
| <p>#1</p>  | <p>Create Rectangle. Line Parallel. Trim. Translate. Create Thread Point. Create Cut Points.</p> | <p>Contour Wirepath.</p> |
| <p>#2</p>  | <p>Create Rectangle. Line Parallel. Line Polar. Trim. Create Thread Point.</p> | <p>Contour Wirepath. Edit the wirepath with Change at Point.</p> |
| <p>#3</p>  | <p>Create Rectangle. Circle Center Point. Line Polar. Arc Tangent. Line Tangent. Trim. Mirror. Rotate. Create Thread Point.</p> | <p>Contour Wirepath to cut the inner profile. Contour Wirepath to cut the outer profile.</p> |
| <p>#4</p>  | <p>Create Rectangle. Circle Center Point. Line Polar. Arc Tangent. Line Tangent. Trim. Mirror. Rotate. Create Thread Point.</p> | <p>Contour Wirepath to cut the inner profile. Contour Wirepath to cut the outer profile.</p> |

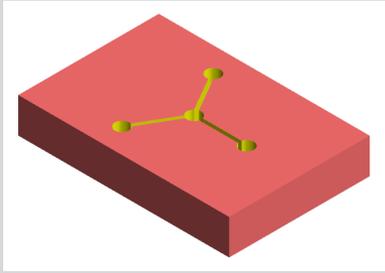
| Tutorial | Geometry Functions | Toolpath Creation |
|--|---|-------------------------------|
| <p>#5</p>  | <p>Create Rectangle. Circle Center Point. Arc Tangent. Fillet. Trim. Mirror. Create Thread Point.</p> | <p>4-Axis Wirepath</p> |
| <p>#6</p>  | <p>Create Rectangle. Circle Center Point. Line Polar. Line Parallel. Trim. Bolt Circle.</p> | <p>Wire No Core Wirepath.</p> |

Table Of Contents

| | |
|--|-----------|
| Wire Projects | 3 |
| Getting Started | 13 |
| Objectives | 14 |
| Step 1: Starting Mastercam | 14 |
| Step 2: GUI - Graphical User Interface | 15 |
| Step 3: Navigate Through Mastercam | 16 |
| Step 4: Set The Attributes | 19 |
| Step 5: Manager Panels | 21 |
| Step 6: Setting Mastercam To Imperial | 23 |
| Step 7: Set The Grid | 25 |
| Tutorial 1 | 27 |
| Overview Of Steps Taken To Create The Part Geometry: | 28 |
| Tutorial #1 Drawing | 29 |
| Step 1: Setting Up The Graphical User Interface | 30 |
| Step 2: Create The 3.625" By 3.0" Rectangle | 30 |
| Step 3: Create Parallel Lines | 32 |
| Step 4: Trim The Lines | 37 |
| Step 5: Translate/Copy The Shape | 41 |
| Step 6: Trim The Lines Using Trim 2 Entities Command | 43 |
| Step 7: Create The Thread And The Cut Points | 45 |
| Step 8: Save The File | 50 |
| Step 9: Select The Machine And Set Up The Stock | 51 |
| Step 10: Machine The Open Profile Using Contour Wirepath | 51 |
| Step 11: Backplot The Wirepath | 65 |
| Step 12: Verify The Wirepath | 68 |
| Step 13: Post The File To Generate The NC Code | 75 |

| | |
|---|------------|
| Step 14: Save The Updated MCAM File | 76 |
| Review Exercise - Student Practice | 77 |
| Create The Geometry For Tutorial #1 Exercise | 78 |
| Tutorial 2 | 83 |
| Geometry Creation | 86 |
| Step 1: Setting Up The Graphic User Interface | 86 |
| Step 2: Create A Rectangle | 86 |
| Step 3: Create Parallel Lines | 89 |
| Step 4: Create A Polar Line | 96 |
| Step 5: Trim Three Entities And Divide/Delete | 98 |
| Step 6: Create The Thread Point | 103 |
| Step 7: Save The File | 105 |
| Step 8: Select The Machine And Set Up The Stock | 106 |
| Step 9: Machine The Blanking Die Using Contour Wirepath | 106 |
| Step 10: Backplot The Wire Contour Wirepath | 119 |
| Step 11: Verify The Wirepath | 124 |
| Step 12: Post The File To Generate The NC Code | 132 |
| Step 13: Save The Updated MCAM File | 133 |
| Reveiw Exercise - Student Practice | 134 |
| Create The Geometry For Tutorial #2 Exercise | 135 |
| Create The Toolpaths For Tutorial #2 Exercise | 136 |
| Notes: | 137 |
| Tutorial #2 Quiz | 138 |
| Tutorial 3 | 139 |
| Geometry Creation | 142 |
| Step 1: Setting Up The Graphic User Interface | 142 |
| Step 2: Create A Rectangle | 142 |
| Step 3: Create 2 Circles Knowing The Diameter And The Center Location | 144 |

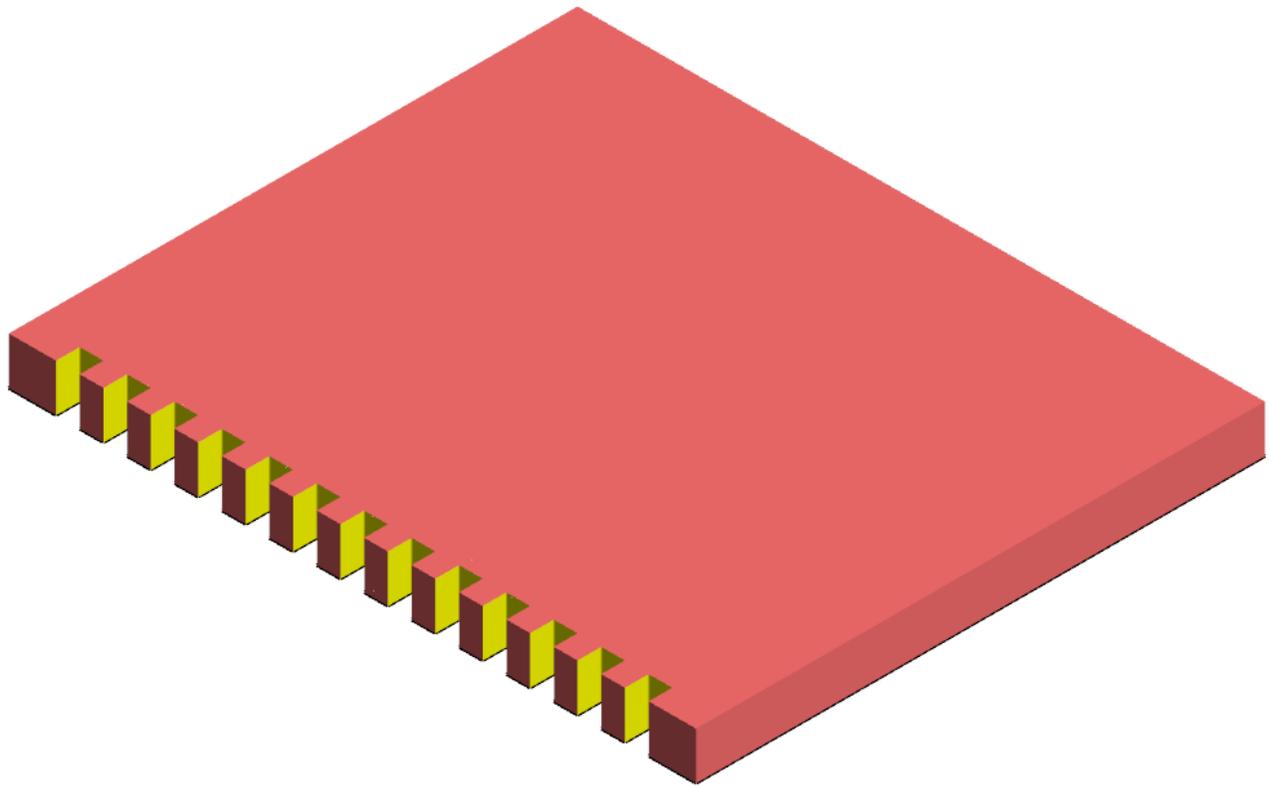
| | |
|---|-----|
| Step 4: Create A Polar Line | 146 |
| Step 5: Create An Arc Tangent To One Entity | 148 |
| Step 6: Create The Thread Point | 152 |
| Step 7: Trim Two Entities | 155 |
| Step 8: Mirror The 0.1875" Radius Arc And The Tangent Line About A Line | 157 |
| Step 9: Create An Arc Tangent To Three Entities | 160 |
| Step 10: Delete The Construction Circles And The Line | 162 |
| Step 11: Trim Three Entities | 163 |
| Step 12: Rotate/Copy The Tooth | 165 |
| Step 13: CREATE THE 1.0" DIAMETER CIRCLE | 169 |
| Step 14: Create Parallel Lines | 170 |
| Step 15: Delete The Construction Lines | 174 |
| Step 16: Use Trimming Commands To Clean The Keyway | 175 |
| Step 17: Create The Thread Points | 179 |
| Step 18: Save The File | 183 |
| Step 19: Select The Machine And Set Up The Stock | 184 |
| Step 20: Machine The Inner Profile Using Contour Wirepath | 184 |
| Step 21: Backplot The Wire Contour Wirepath | 197 |
| Step 22: Machine The Outer Profile Using Contour Wirepath | 200 |
| Step 23: Verify The Toolpath | 214 |
| Step 24: Post The File To Generate The NC Code | 220 |
| Step 25: Save The Updated MCAM File | 221 |
| Review Exercise - Student Practice | 222 |
| Create The Geometry For Tutorial #3 Exercise | 223 |
| Create The Toolpaths For Tutorial #3 Exercise | 224 |
| Notes: | 226 |
| Tutorial #3 Quiz | 227 |

| | |
|--|------------|
| Tutorial 4 | 229 |
| Geometry Creation | 232 |
| Step 1: Setting Up The Graphic User Interface | 232 |
| Step 2: Create A Rectangle | 232 |
| Step 3: Create Parallel Lines For The Outside Profile | 234 |
| Step 4: Create A Line | 240 |
| Step 5: Clean Up The Outside Profile Using Trim Two Entities | 242 |
| Step 6: Create Parallel Lines For The Inside Profile | 245 |
| Step 7: Clean The Inside Profile Using Trim Three Entities | 249 |
| Step 8: Delete The Construction Lines | 251 |
| Step 9: Mirror The Geometry About X-axis And Y-axis | 252 |
| Step 10: Create The Thread Points | 256 |
| Step 11: Save The File | 262 |
| Step 12: Select The Machine And Set Up The Stock | 263 |
| Step 13: Machine Multiple Contours Using Contour Wirepath | 266 |
| Step 14: Backplot The Wire Contour Wirepath | 278 |
| Step 15: Machine The Outer Profile With Multiple Tabs | 280 |
| Step 16: Verify The Wirepath | 296 |
| Step 17: Post The File To Generate The NC Code | 300 |
| Step 18: Save The Updated MCAM File | 301 |
| Review Exercise - Student Practice | 302 |
| Create The Geometry For Tutorial #3 Exercise | 303 |
| Create The Toolpaths For Tutorial #4 Exercise | 304 |
| Notes: | 306 |
| Tutorial #4 Quiz | 307 |
| Tutorial 5 | 309 |
| Geometry Creation | 312 |
| Step 1: Setting Up The Graphic User Interface | 312 |

| | |
|--|------------|
| Step 2: Change The Z Depth Of The Cplane And Switch To 2d Mode | 312 |
| Step 3: Create The 0.5625" Radius Circles | 312 |
| Step 4: Create Fillets With The Radius 0.025" | 315 |
| Step 5: Change The Z Depth Of The Cplane To Z 0.0 | 318 |
| Step 6: Create The 0.3625" Radius Circle | 318 |
| Step 7: Create The 1.25" Radius Tangent Through A Point Arc | 320 |
| Step 8: Clean The Geometry Using Trim Divide/delete | 322 |
| Step 9: Mirror The Geometry About Y-axis | 324 |
| Step 10: Create Fillets With The Radius 0.025" And 0.05" | 326 |
| Step 11: Create Branch Lines | 330 |
| Step 12: Create The Stock Rectangle | 341 |
| Step 13: Create The Thread Point | 343 |
| Step 14: Save The File | 346 |
| Step 15: Select The Machine And Set Up The Stock | 347 |
| Step 16: Machine The Part Using 4-Axis Wirepath | 347 |
| Step 17: Backplot The Wire Contour Wirepath | 359 |
| Step 18: Verify The Wirepath | 361 |
| Step 19: Post The File To Generate The NC Code | 366 |
| Step 20: Save The Updated MCAM File | 367 |
| Review Exercise - Student Practice | 368 |
| Create The Geometry For Tutorial #5 Exercise | 369 |
| Create The Toolpaths For Tutorial #5 Exercise | 370 |
| Notes: | 371 |
| Tutorial #5 Quiz | 372 |
| Tutorial 6 | 373 |
| Geometry Creation | 376 |
| Step 1: Setting Up The Graphic User Interface | 376 |

| | |
|---|------------|
| Step 2: Create The 3.0" By 2.0" Rectangle | 376 |
| Step 3: Create The 0.2" Diameter Circle | 378 |
| Step 4: Create The Three 0.2" Diameter Circles On A Bolt Circle | 380 |
| Step 5: Create A Line Given The Endpoint | 382 |
| Step 6: Create Parallel Lines | 383 |
| Step 7: Delete The Construction Line | 385 |
| Step 8: Rotate/Copy | 387 |
| Step 9: Clean The Geometry Using Divide/Delete | 389 |
| Step 10: Save The File | 394 |
| Step 11: Select The Machine And Set Up The Stock | 395 |
| Step 12: Machine The Part Using No Core Wirepath | 395 |
| Step 13: Backplot The Wire Contour Wirepath | 402 |
| Step 14: Verify The Wirepath | 404 |
| Step 15: Post The File To Generate The NC Code | 409 |
| Step 16: Save The Updated MCAM File | 410 |
| Review Exercise - Student Practice | 411 |
| Create The Geometry For Tutorial #5 Exercise | 412 |
| Create The Toolpaths For Tutorial #6 Exercise | 413 |
| Notes: | 414 |
| Tutorial #6 Quiz | 415 |
| Quiz Answers | 417 |

Tutorial 1



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 used to generate Wirepaths from:

- ◆ The student will create the Top 2D geometry needed to create the wirepaths.
- ◆ Geometry creation commands such as Create Rectangle, Line Parallel, Trim, Translate, Create Thread Point, and Create Cut Points will be used.

Create the necessary Wirepaths to machine the part:

- ◆ A Contour wirepath for open contour will be created to remove the material.

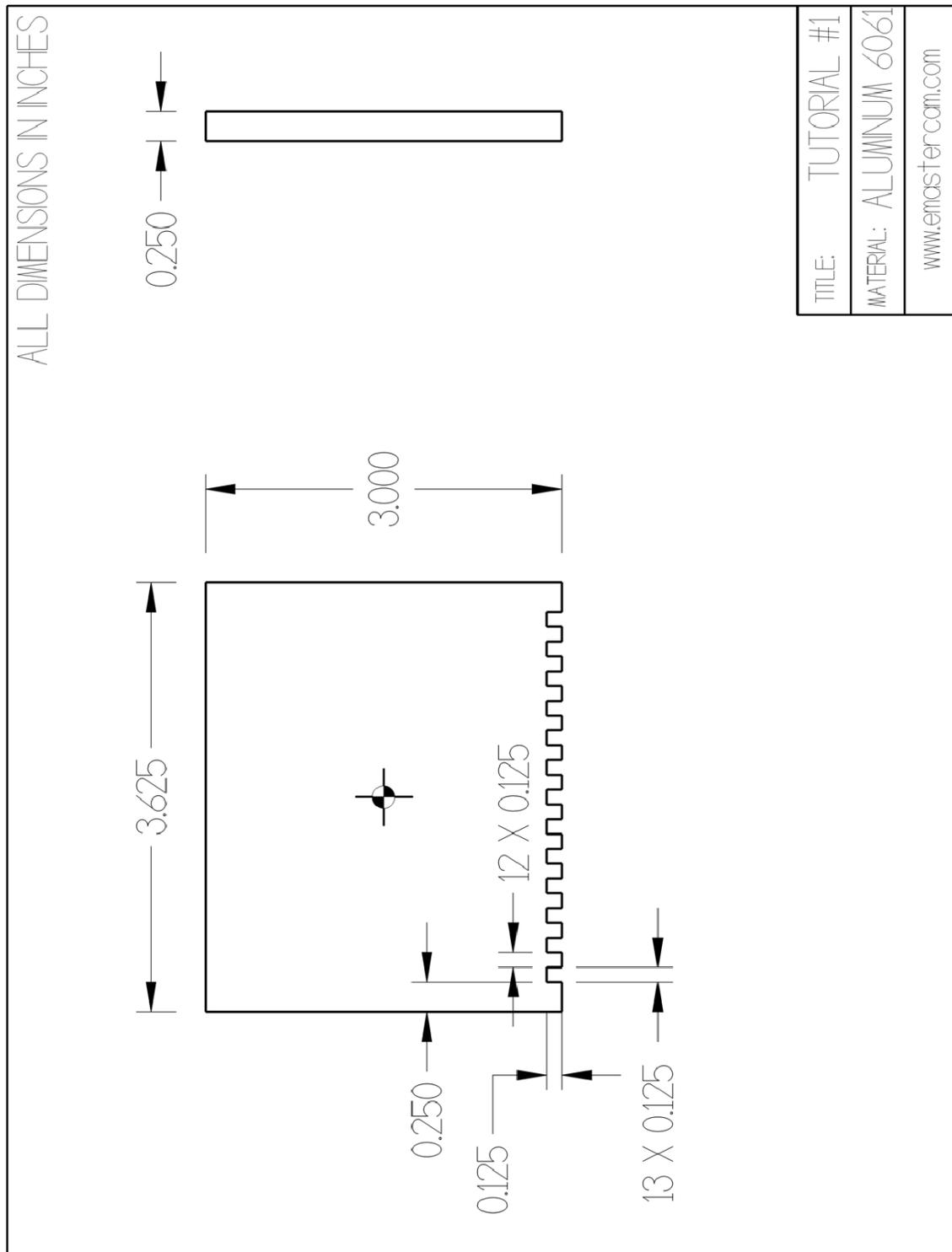
Backplot and Verify the file:

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

Post Process the file to generate the G-code:

- ◆ The student will post process the file to obtain an NC file containing the necessary code for the machine.

TUTORIAL #1 DRAWING



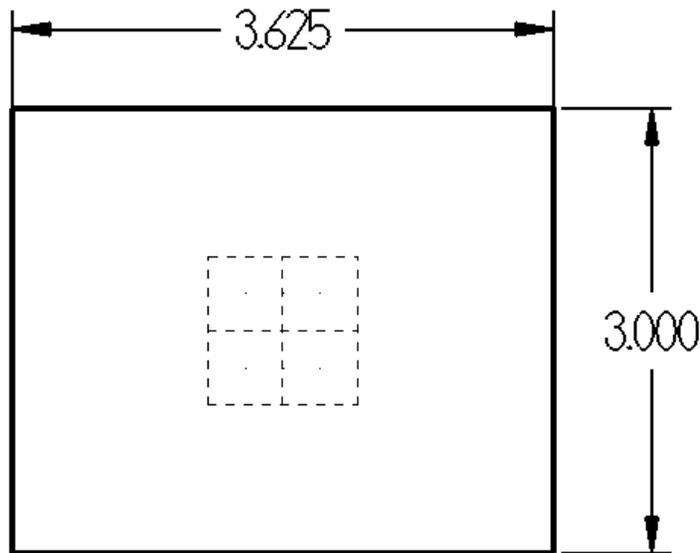
STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

Please refer to the **Getting Started** section for more info on how to set up the graphical user interface.

STEP 2: CREATE THE 3.625" BY 3.0" RECTANGLE

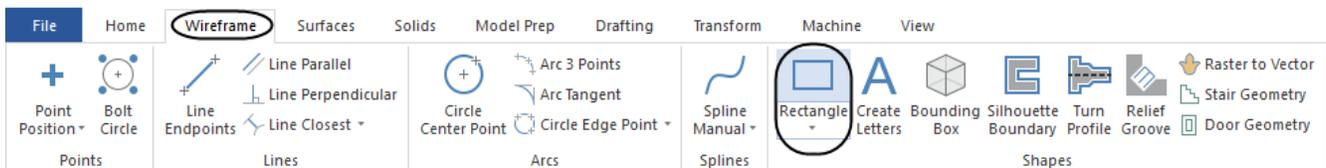
In this step, you will learn how to create a rectangle given the width, the height, and the anchor position.

Step Preview:

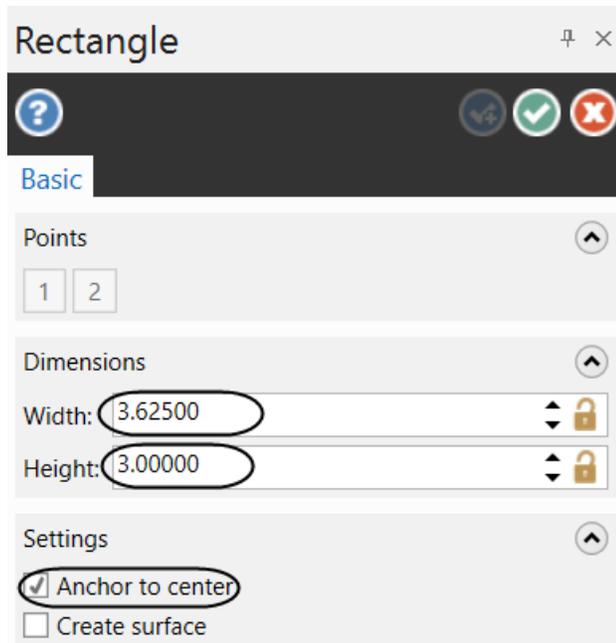


Wireframe

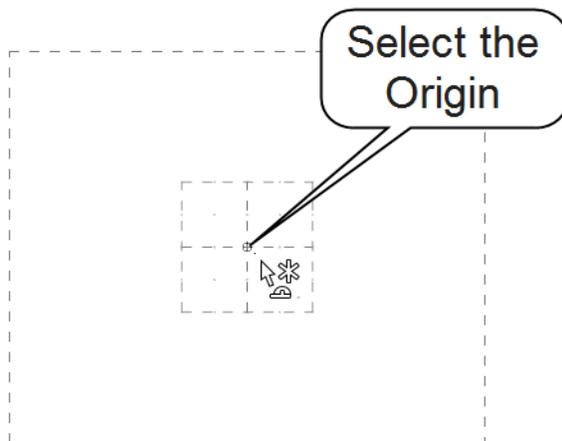
◆ From the **Shapes** group, select the **Rectangle** icon.



- ◆ Enter the **Width** and **Height** and enable **Anchor** to center as shown.



- ◆ [Select position of base point]: Select the **Origin** as shown.



- ◆ Select the **OK** button to exit the **Rectangle** command.
- ◆ Press **Alt+F1** to fit the rectangle to the screen.

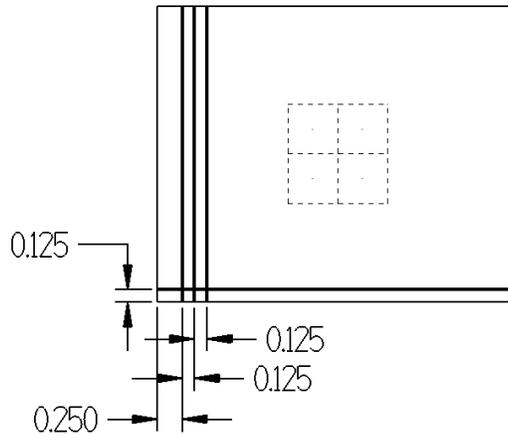
*Note: While creating the geometry for this tutorial, if you make a mistake, you can undo the last step by using the **Undo** icon. You can undo as many steps as needed. If you delete or undo a step by mistake, just use the **Redo** icon. To delete unwanted geometry, select the geometry first and then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

Note: In the next few steps, you will create the profile with 13 identical shapes. You will create one shape first, and then, by using the **Translate** command with the **Copy** option, generate the other twelve shapes.

STEP 3: CREATE PARALLEL LINES

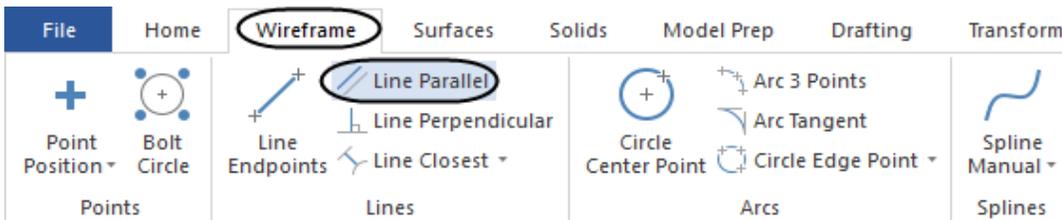
In this step you will create the four parallel lines that will be used to create one of the shapes.

Step Preview:

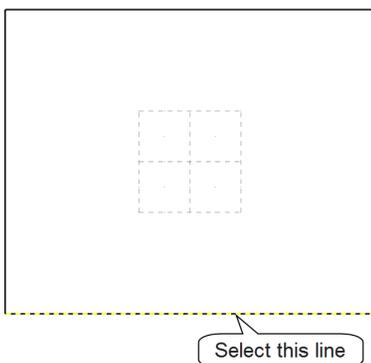


Wireframe

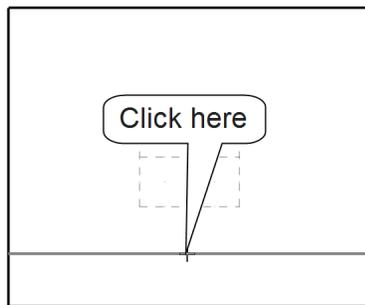
◆ From the **Lines** group, select **Line Parallel**.



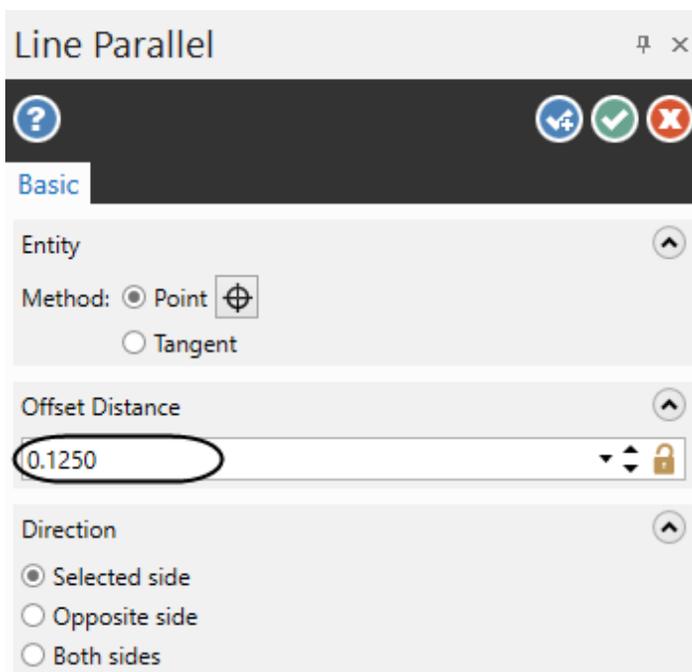
◆ [Select a line]: Select the lower horizontal line of the rectangle as shown.



- ◆ [Select the point to place a parallel line through]: Select a point above the line as shown.



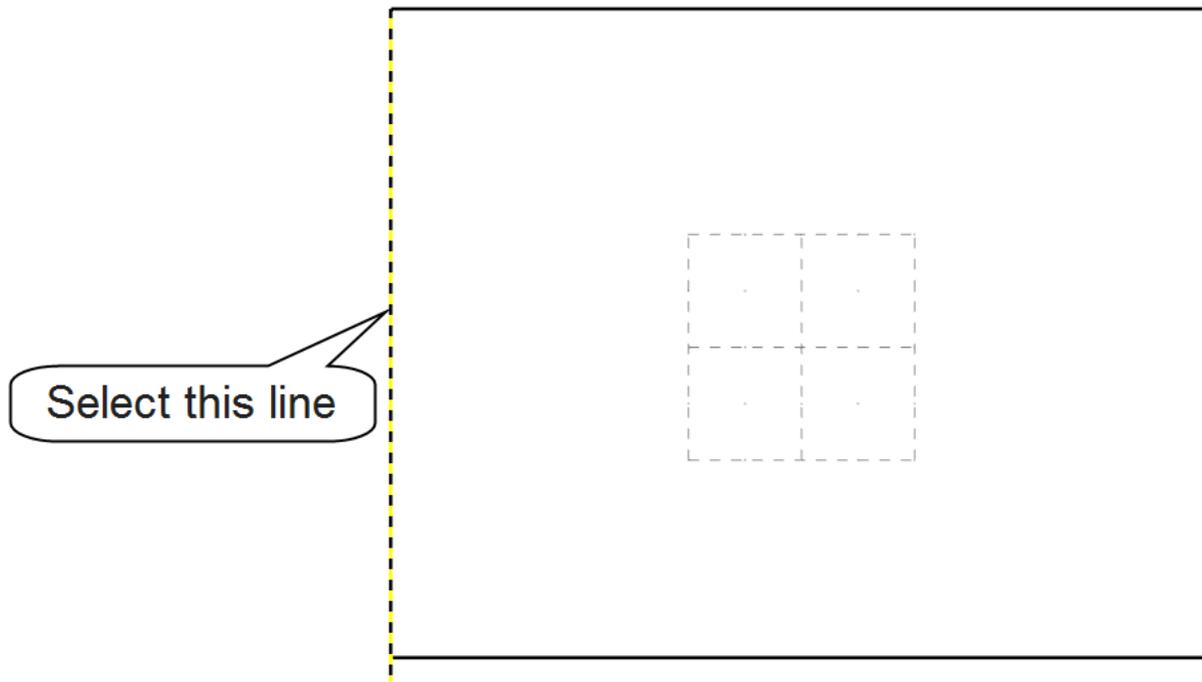
- ◆ Under **Offset Distance**, enter **0.125** as shown. Press **Enter**.



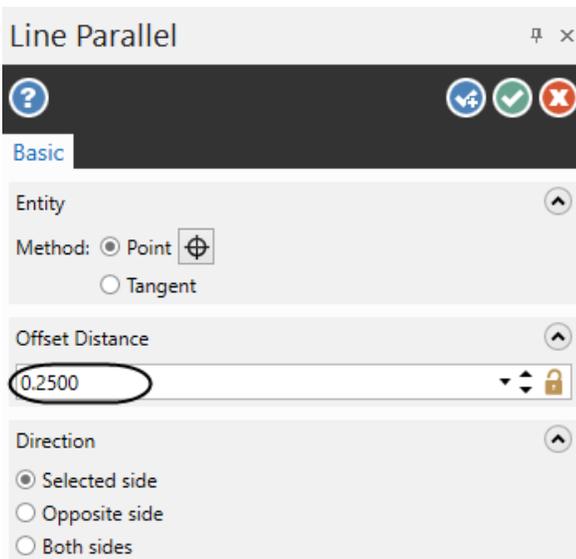
- ◆ Select **OK and Create New Operation** to stay in the command.



- ◆ [Select a line]: Select the left side vertical line of the rectangle as shown.



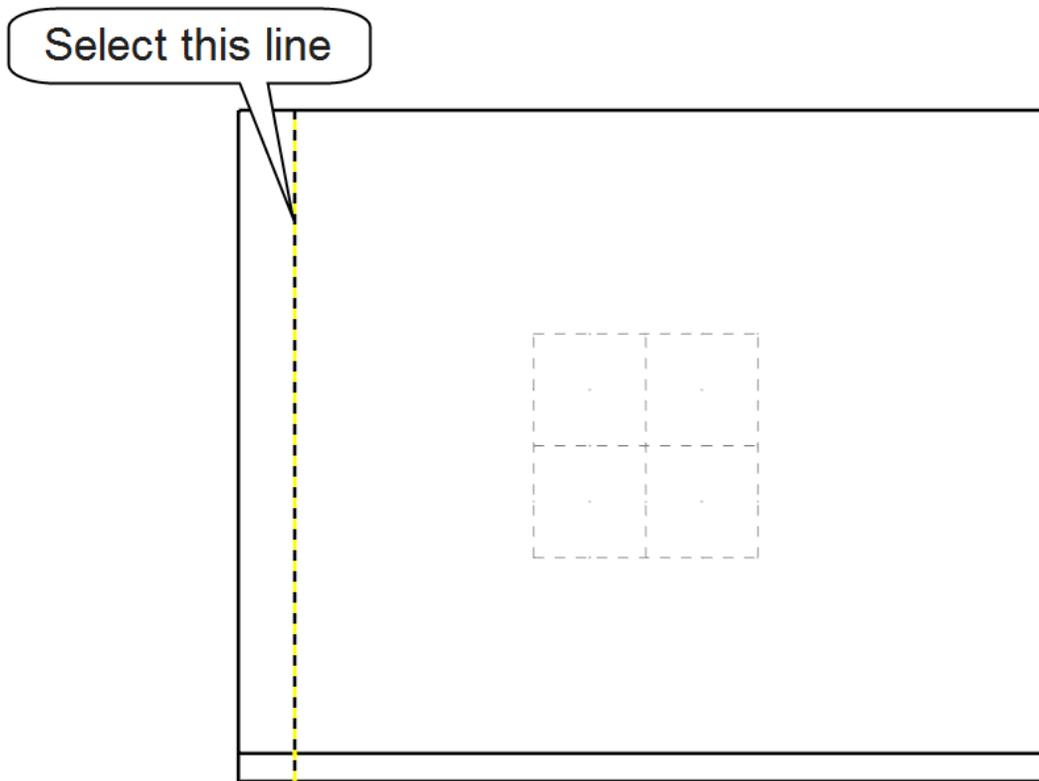
- ◆ [Select the point to place a parallel line through]: Select a point to the right of the selected line.
- ◆ Under **Offset Distance**, enter **0.25** as shown. Press **Enter**.



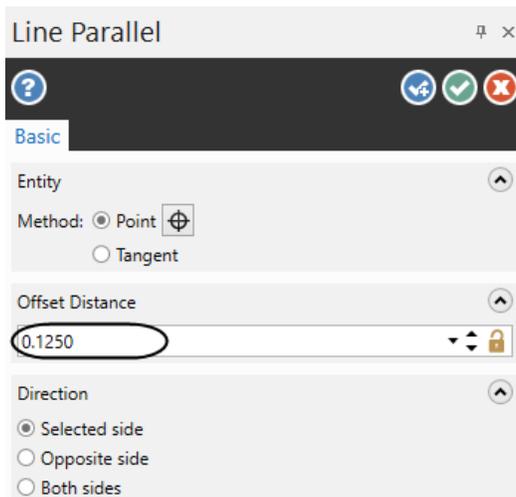
- ◆ Select **OK and Create New Operation** to stay in the command.



- ◆ [Select a line]: Select the vertical line that you just created as shown.



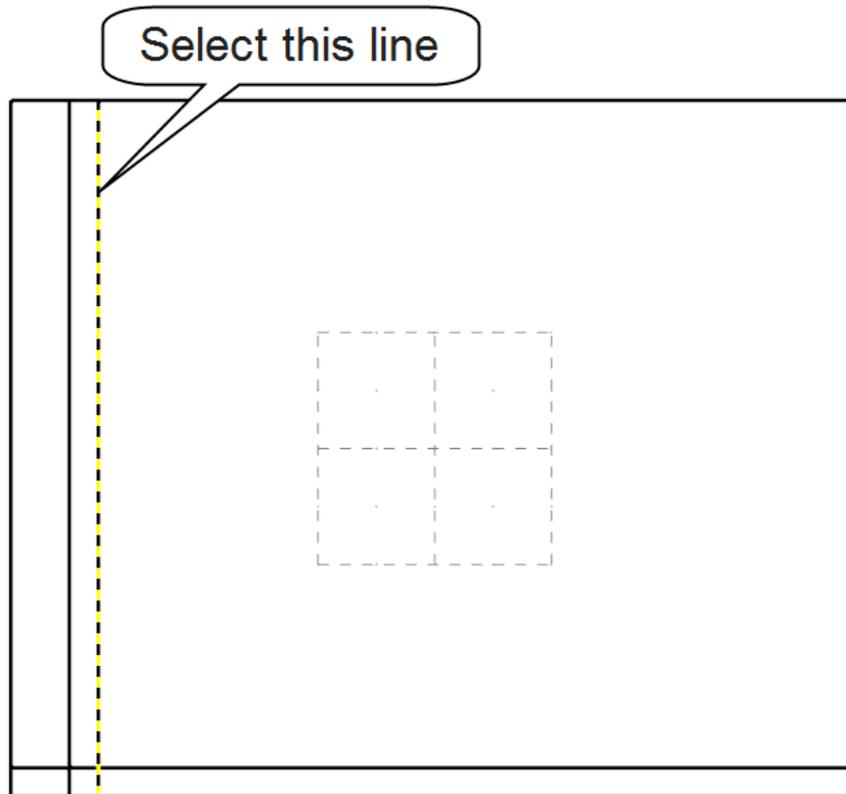
- ◆ [Select the point to place a parallel line through]: Select a point to the right of the selected line.
- ◆ Under **Offset Distance**, enter **0.125** as shown. Press **Enter**.



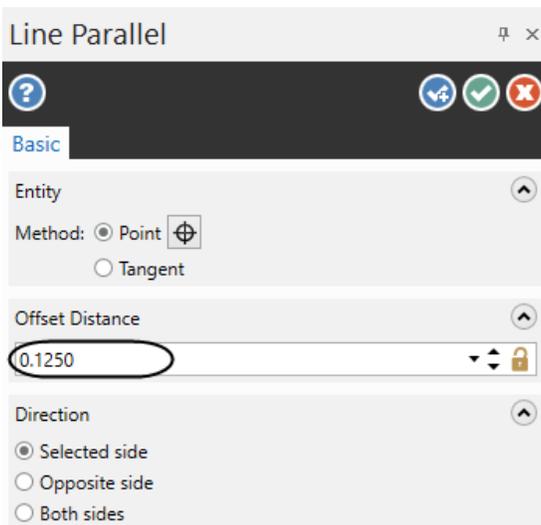
- ◆ Select **OK** and **Create New Operation** to stay in the command.



- ◆ [Select a line]: Select the vertical line that you just created as shown.



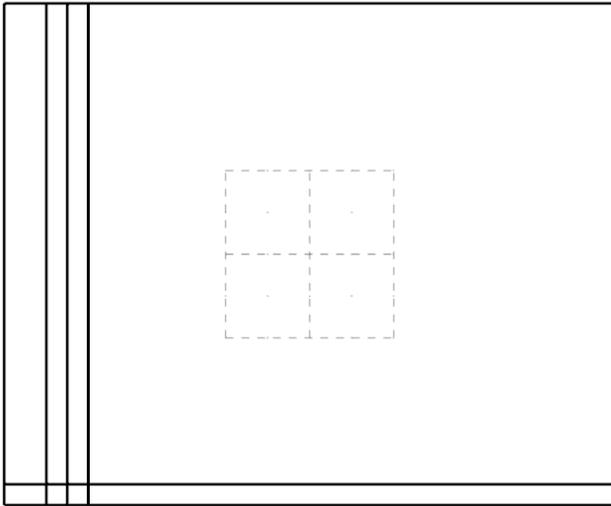
- ◆ [Select the point to place a parallel line through]: Select a point to the right of the selected line.
- ◆ Under **Offset Distance**, enter as shown. Press **Enter**.



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



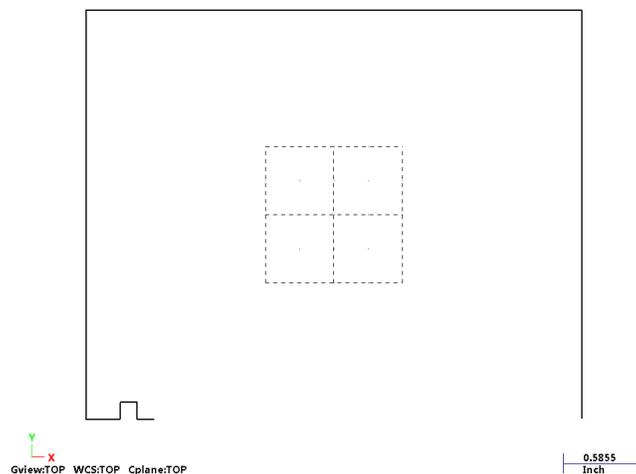
- ◆ The geometry should look as shown.



STEP 4: TRIM THE LINES

This step shows how to use the **Trim 3 Entities** and the **Divide/delete** commands to clean up the shape.

Step Preview:



Note: During the trimming process it is very important to select the entities exactly in the order and at the locations as shown in the graphics on the following pages.

Trim 3 entities command requires a special selection. The first two entities that you select are trimmed to the third, which acts as a trimming curve. The third entity is then trimmed to the first two.

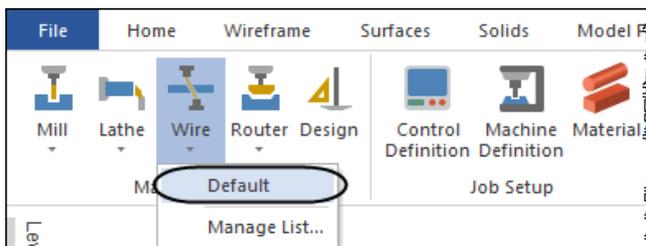
STEP 9: SELECT THE MACHINE AND SET UP THE STOCK

In Mastercam, you select a Machine Definition before creating any toolpaths. The Machine Definition is a model of your machine tool's capabilities, features, and it acts like a template for setting up machining jobs. The machine definition ties together three main components: the schematic model of your machine tool's components, the control definition that models your control unit's capabilities, and the post processor that will generate the required machine code (G-code). For the purpose of this tutorial, we will be using the Wire Default machine.

*Note: If you already have a wire in the **Toolpaths Manager**, do not select another machine. Expand **Properties** and select **Files**. When the **Machine Group Properties** dialog box appears, on the **File** tab select **Replace** button under the **Machine - Toolpath Copy** section, and open the file of WIRE DEFAULT MM.MCAM-WMD/WIRE DEFAULT.MCAM-WMD. Once finished, select OK to exit the dialog box. Otherwise, please follow next step.*

Machine

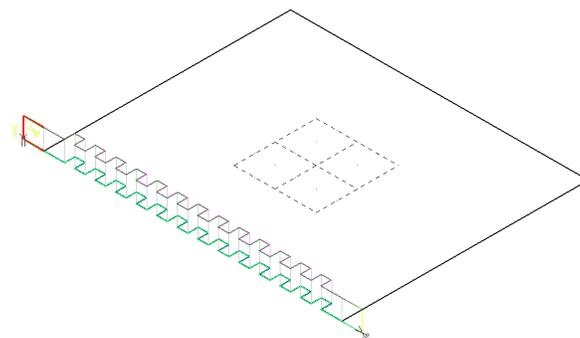
- ◆ From the **Machine Type** group, click on the pull down arrow under **Wire** and select **Default** as shown.



STEP 10: MACHINE THE OPEN PROFILE USING CONTOUR WIREPATH

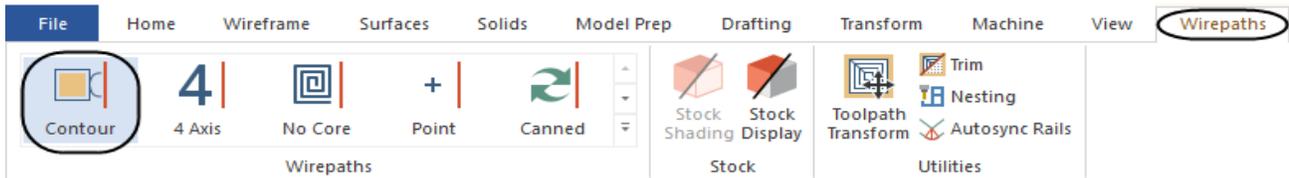
Contour Wirepaths have the same general shape in both the XY plane (the lower contour) and the UV plane (the upper contour). Contour wirepaths can taper inward or outward, and you can specify the location of the land, the point at which the taper begins. You can further modify the shape of the contour wirepath by specifying how Mastercam handles sharp and smooth corners. A contour wirepath can also be based on an open boundary and used for cutting off or trimming a part.

Toolpath Preview:

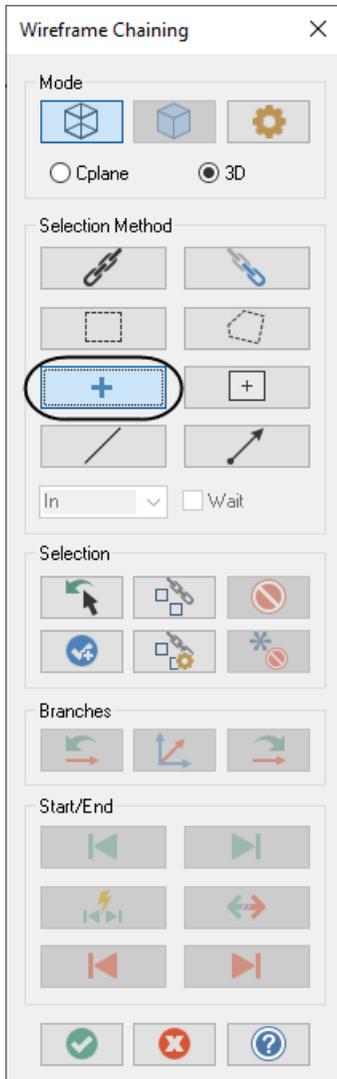


Wirepaths

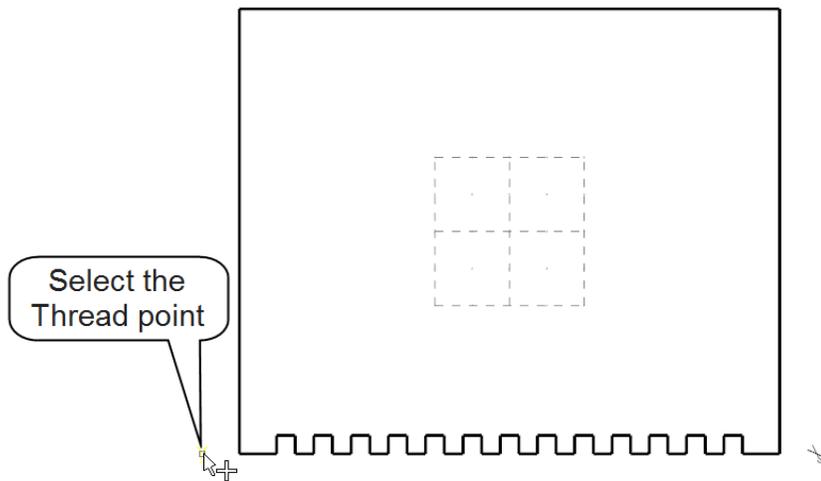
- ◆ From the **Wirepaths** group, select **Contour**.



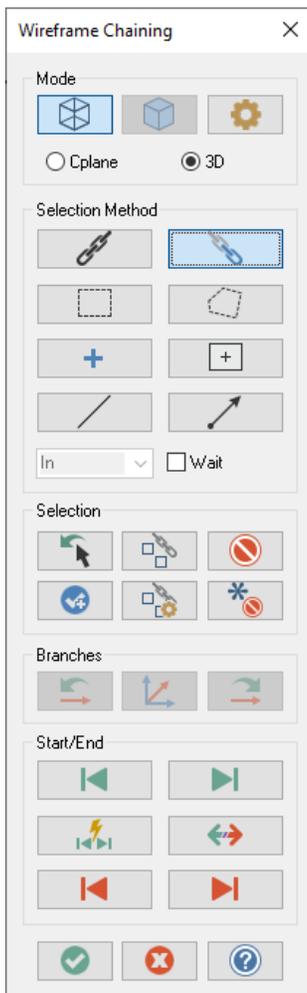
- ◆ From the **Chaining** dialog box, select the **Point** button.



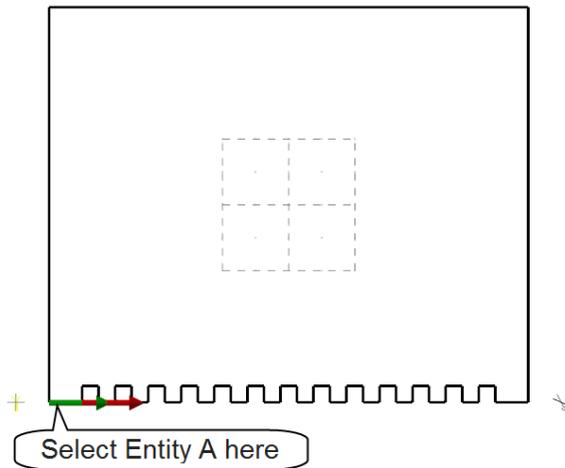
- ◆ [Contour: select chain 1]: Select the thread point as shown.



- ◆ From the **Chaining** dialog box, select the **Partial Chain** button.



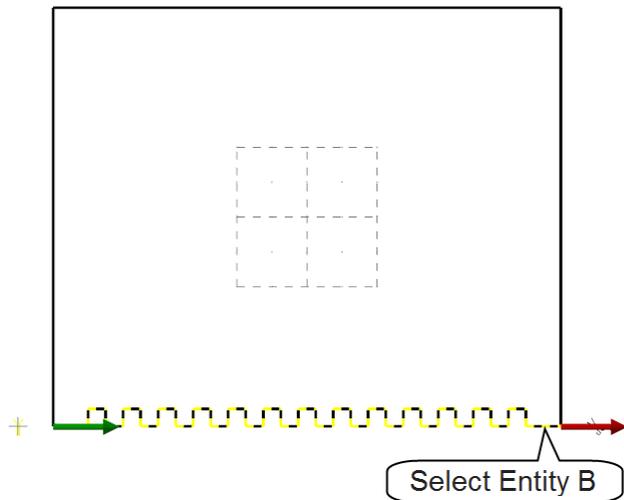
- ◆ [Select the first entity]: Select Entity A as shown.



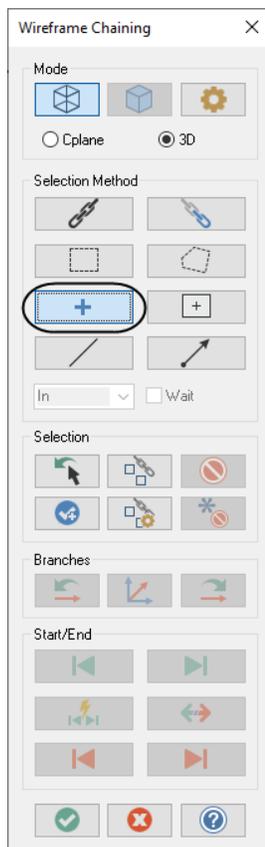
- ◆ The direction of the chain should be toward the right. Otherwise use the **Reverse** button.



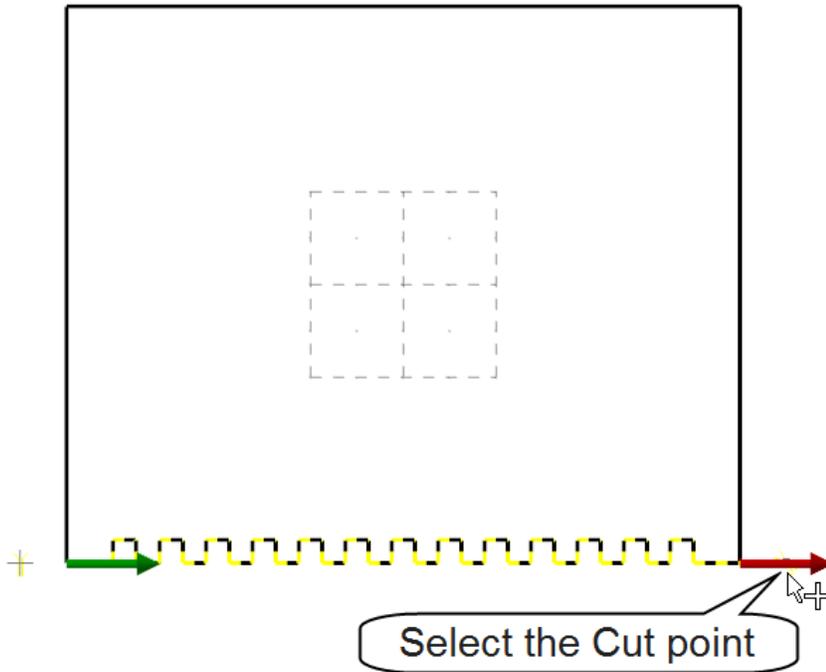
- ◆ [Select the last entity]: Select Entity B as shown.



- ◆ From the **Chaining** dialog box, select the **Point** button.



- ◆ [Contour: select chain 3]: Select the **Cut point** as shown.

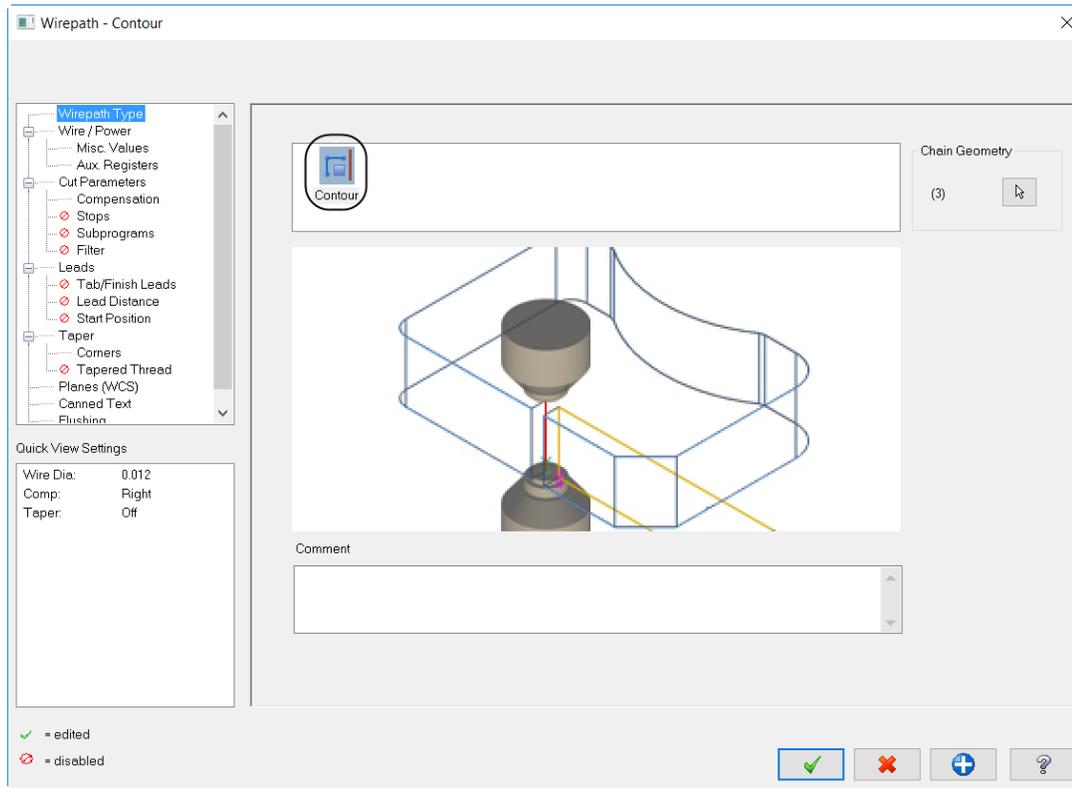


- ◆ Select the **OK** button to exit from the **Chaining** dialog box.

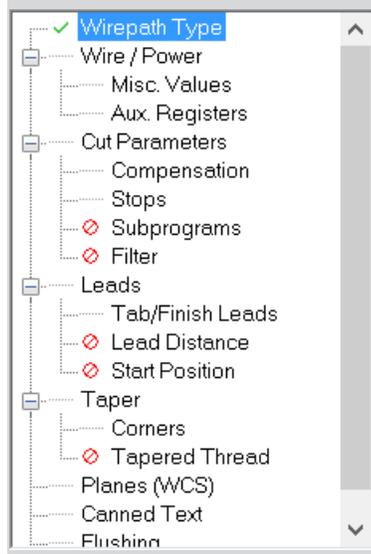


10.1 Wirepath Type

◆ In the **Wirepath Type** page, the **Contour** toolpath should already be selected.



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



10.2 Set the Wire/Power parameters

The **Wire/Power** page allows you to choose a wire power settings library file and use its values for the wirepath operation. You can also set the pass number to use for the first pass of the wirepath, and specify the settings for a single pass. To edit the power settings for each pass you have to disassociate the wirepath from the power settings library by unselecting the **Associate to library** check box.

- ◆ From the **Tree View** area, select **Wire/Power**.
- ◆ Set the parameters for **Pass #1** (Roughing Pass) as shown below.

Power library
C:\Users\Public\Documents\Shared Mastercam 2020\wire\Power\WIRE_INCH.po

Associate to library

Starting Pass # 1

Options
 Wire
 Power
 Fill tank

Pass # 1

Offset 1 Condition code: 101 Feed rate: 0.0

Wire Diameter 0.012
Wire Radius 0.006
Wire Overburn 0.0

Stock to Leave 0.0

Apply additional offset to:
 Program coordinates
 Machine offset register

Total Offset 0.006

Pass Comment
Roughing Pass

Pass # is used to enter the pass number to edit the settings for a single pass in the current library.

Offset allows you to set the wire offset register number. You may need to refer to your wire machine documentation for this information.

Condition Code is a wire machine-specific value that corresponds to a register number. Refer to your wire machine documentation codes for material and wire type and thickness.

Feed rate is available if needed by the controller. Most EDMs do not use a feed rate.

Wire diameter/radius sets the wire diameter/radius value and is automatically updated when one of the values is entered.

Wire overburn sets the amount of material beyond the wire diameter that can be removed.

Stock to Leave sets the amount of stock to leave for all passes. Stock to Leave offsets the wire in the direction specified by the Compensation Direction. In our case is set to 0.0 because this parameter is commonly set at the controller.

- ◆ Select the upper arrow and change the **Pass #** to **2**.
- ◆ Set the parameters for **Pass #2** (Skim cut) as shown.

Power library
C:\Users\Public\Documents\Shared Mastercam 2020\wire\Power\WIRE_INCH.po

Associate to library

Starting Pass # 1

Options
 Wire
 Power
 Fill tank

Pass # 2

Offset 2 Condition code: 102 Feed rate: 0.0

Wire Diameter 0.012
Wire Radius 0.006
Wire Overburn 0.0

Stock to Leave 0.0 Apply additional offset to:
 Program coordinates
 Machine offset register

Total Offset 0.006
Pass Comment
Skim cut

*Note: If you want to save this power library for other jobs, you can select the **Save** icon as shown in the image below. For the purpose of this tutorial, we will not save the power library. The Power folder will automatically be opened, and in the File name field you can enter the name of the library and then save it.*

Power library
C:\Users\Public\Documents\Shared Mastercam 2020\wire\Power\WIRE_INCH.po

Associate to library

Starting Pass # 1

Options
 Wire
 Power
 Fill tank

Save

Offset 2 Condition code: 102 Feed rate: 0.0

Wire Diameter 0.012
Wire Radius 0.006
Wire Overburn 0.0

Stock to Leave 0.0 Apply additional offset to:
 Program coordinates
 Machine offset register

Total Offset 0.006
Pass Comment

10.3 Set the Cut Parameters

The **Cut Parameters** page is used to define the type and the number of cuts you will make in the wirepath, to control the stop codes, to suppress thread or cut flags. You can also divide the cuts into separate operations.

Mastercam Wire supports three main categories of cuts: rough and skim cuts before the tab, tab cuts, and skim cuts after the tab. The **Cut Parameters** page lists the programmed cuts and indicates their grouping as operations.

- ◆ From the **Tree View** area, select **Cut Parameters**.
- ◆ Change the parameters to make one rough pass and one finish pass.

The screenshot shows the 'Cut Parameters' dialog box with the following settings:

- Cuts before tabs:**
 - Perform rough cut
 - Additional skim cuts (before tab): 1
- Cutting method:**
 - One Way
 - Reverse
- Reset pass number on tab cuts:**
- Expand operation:**
- Suppress all wire threads:**
- Suppress all wire cuts:**
- Linearization tolerance:** 0.0001
- Chain 1:**
 - Rough
 - Skim
- Total cuts for each chain:** 2
- Additional settings:**
 - Tab:** Tab Width: 0.1, Number of tab cuts: 1
 - Cuts after tabs:** Skim cuts after tab, Together: 1, Separate: 0

Perform rough cut is used to create a rough cut for the contour. The rough cut uses the Starting Pass # from the Wire power settings library.

Additional skim cuts (before tab) allows you to enter the number of passes before cutting the tab.

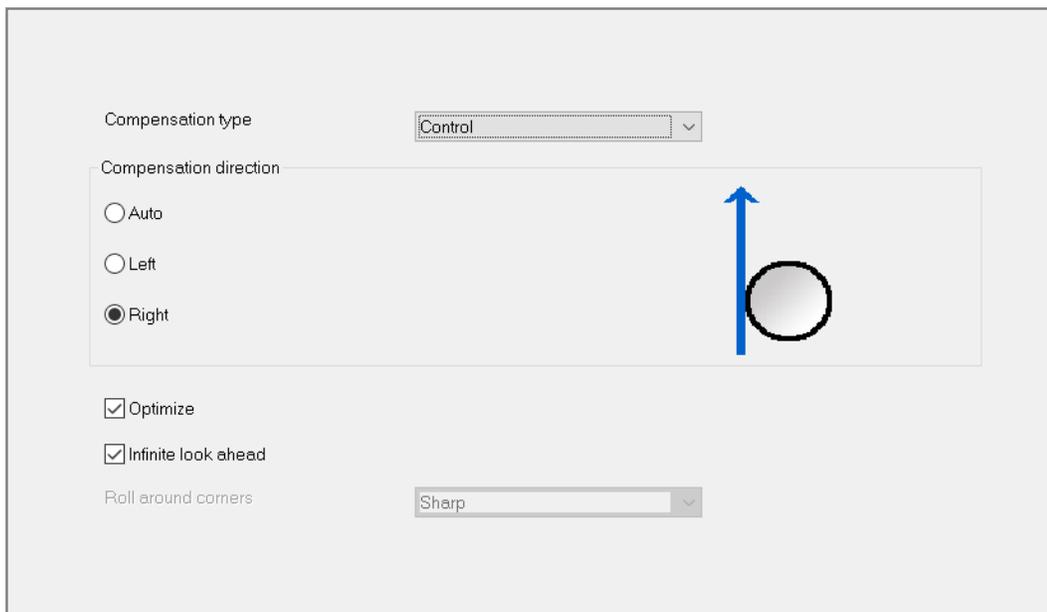
Cutting method set to **Reverse** is used when multiple passes are set for the toolpath. With Reverse enabled, Mastercam reverses the direction of the previous pass at the end of the contour or at the tab.

Note: For this part you do not need to add a tab on the contour as the part can be clamped from the opposite side, which is not machined in this operation.

10.4 Set the Compensation parameters

Compensation page allows you to set the compensation method by which Mastercam Wire offsets the wire from the wirepath. Choose to offset the wire to the right or left of the wirepath. You can eliminate arcs in the wirepath that are less than or equal to the radius of the wire, or insert arc moves around corners in the wirepath. It also allows you to check for and eliminate unwanted intersections in the wirepath.

- ◆ From the **Tree View** area, select **Compensation**.
- ◆ Select the **Cut Parameters** page and change the **Compensation** direction to **Right** as shown.



Compensation type sets to **Control** outputs control codes for compensation and does not compute the compensated wirepath.

Compensation direction allows you to offset the wire to the right or left of the wirepath. **Auto** sets the compensation direction depending on the location of the thread point inside or outside of the contour.

Optimize eliminates arcs in the wirepath that are less than or equal to the radius of the wire.

Infinite look ahead checks for wirepath self-intersections along the entire contour before creating the wirepath. If Mastercam Wire finds a wirepath intersection, it modifies the wirepath so that it does not cut the portion of the part that comes after the intersection.

Roll around corners inserts arc moves around corners in the wirepath. The radius of the arc moves equals the radius of the wire. Not available with compensation type set to **Control**.