Mastercam® 2018

IMPERIAL – TRAINING TUTORIAL SERIES

WIRE

Mastercam 2018

Wire Training Tutorial

To order more books:

Call 1-800-529-5517 or

Visit www.emastercam.com or

Contact your Mastercam dealer

Mastercam 2018 Wire Training Tutorial

Copyright: 1998 - 2018 In-House Solutions Inc. All rights reserved

Software: Mastercam 2018

Author: Mariana Lendel

ISBN: 978-1-77146-651-6

Revision Date: May 30, 2017

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 distributer, 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.

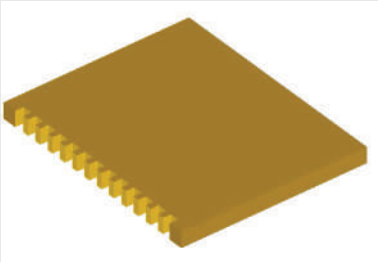
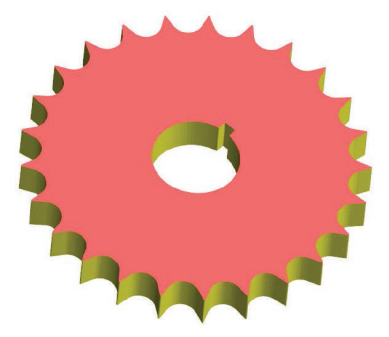
Copyrights

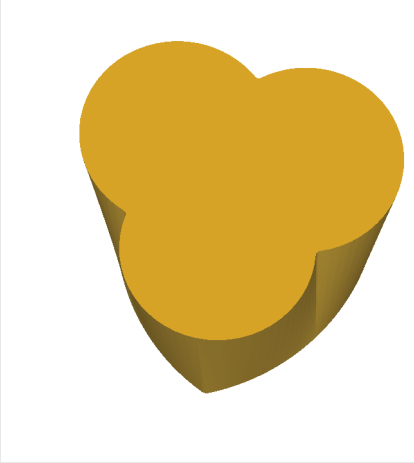
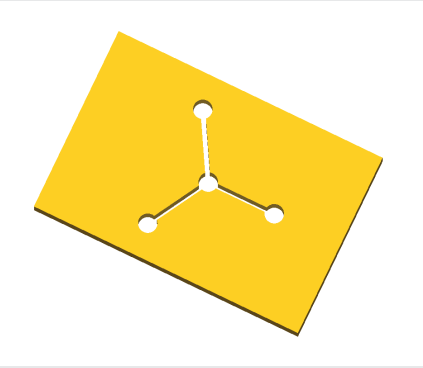
This manual is protected under International copyright laws. All rights are reserved. This document may not, in whole or part, be copied, photographed, reproduced, translated or reduced to any electronic medium or machine readable form without prior consent, in writing, from In-House Solutions Inc.

Trademarks

Mastercam is a registered trademark of CNC Software, Inc.

Microsoft, the Microsoft logo, MS, and MS-DOS are registered trademarks of Microsoft Corporation; N-See is a registered trademark of Microcompatibles, Inc.; Windows 7 and Windows 8 are registered trademarks of Microsoft Corporation.

Tutorial	Geometry Functions	Surface and Toolpath Creation
#1 	Create Rectangle. Create Parallel Line. Edit Trim 3 entities. Xform Translate/ Copy. Edit Trim 2 entities. Create Thread Point. Create Cut Point.	Contour wirepath for open contour.
#2	Create Rectangle. Create Line Parallel. Create Line Endpoint (polar line). Edit Trim 3 entities. Edit Trim 2 entities. Create Thread Point.	Contour wirepath to cut a die with taper angle and land. Edit the wirepath using Change at a point.
#3 	Create Rectangle. Create Circle Center Point. Create Line Endpoint (polar line). Create Arc Tangent to 1 Entity. Create Line Tangent at an angle. Xform Mirror. Create Arc Tangent to 3 Entities. Edit Trim 3 entities. Edit Trim Divide. Edit Trim 2 entities. Xform Rotate Create Line Parallel. Create Thread Point.	Contour wirepath to cut the inner profile. Contour wirepath to cut the outer profile.

Tutorial	Geometry Functions	Surface and Toolpath Creation
#4 	Create Rectangle. Create Line Parallel. Create Line Endpoint (polar line). Edit Trim 2 entities. Mirror about X axis. Mirror about Y axis. Create Thread Point.	Contour wirepath to cut multiple contours. Contour wirepath with multiple tabs.
#5 	Change the Z depth. Change the construction mode to 2D. Create Circle Center Point. Create Fillet Entities. Create Arc Tangent through Point. Trim Divide. Mirror about Y axis. Change the construction mode to 3D. Break 2 Pieces. Create Point Endpoints. Create Line Endpoint Create Rectangle	4 Axis wirepath.
#6 	Create Rectangle. Create Circle Center Point. Create Bolt Circle Create Line Endpoint Create Line Parallel. Xform Rotate Edit Trim Divide.	No core wirepath.

MASTERCAM SHORTCUTS

Function	Keyboard Shortcut	Function	Keyboard Shortcut
Analyze entities	F4	Mastercam version, SIM serial number	Alt+V
AutoSave	Alt+A	Motion controller rotation point	Alt+F12
C-Hook or user app	Alt+C	Pan	Arrow keys
Configure Mastercam	Alt+F8	Paste from clipboard	Ctrl+V
Copy to clipboard	Ctrl+C	Redo an event that has been undone	Ctrl+Y
Cut to clipboard	Ctrl+X	Repaint	F3
Delete entities	F5	Rotate	Alt+Arrow keys
Drafting global options	Alt+D	Select all	Ctrl+A
Exit Mastercam	Alt+F4	Selection grid parameters	Alt+G
Fit geometry to screen	Alt+F1	Shading on/off	Alt+S
Gview–Back	Alt+3	Show/hide all axes (WCS, Cplane, Tplane)	Alt+F9
Gview–Bottom	Alt+4	Show/hide coordinate axes	F9
Gview–Front	Alt+2	Show/hide displayed toolpaths	Alt+T
Gview–Isometric	Alt+7	Show/hide Operations Manager panel	Alt+O
Gview–Left	Alt+6	Undo the last creation or event	Ctrl+U, Ctrl+Z
Previous Plane	Alt+P	Unzoom to 80% of original	Alt+F2
Gview–Right	Alt+5	Unzoom to previous or 50% of original	F2
Gview–Top	Alt+1	Zoom around target point	Ctrl+F1
Help	Alt+H	Zoom with window selection	F1
Hide entities	Alt+E	Zoom/unzoom by 5%	Page Up/Page Down
Level Manager	Alt+Z		
Main attributes, set from entity	Alt+X		

CUSTOMIZE MASTERCAM

Create Your Own Keyboard Shortcuts

- Right mouse click in the **Ribbon.**
- **Customize the Ribbon.**
- **Keyboard shortcuts,** select **Customize** button.
- Select the **Category**.
- Select a command and under **Press new shortcut key** enter the key combinations you want to assign to it.

Customize Quick Access Toolbar

- Right mouse click on a command in the **Ribbon.**
- Select **Add to Quick Access Toolbar.**

Customize the Ribbon

- Right mouse click in the **Ribbon.**
- Select **Customize the Ribbon.**
- Select a Mastercam command.
- Select a **Tab** or create a **New Tab**.
- Press **Add** button.

Customize the right-click menu

- Right mouse click in the **Ribbon.**
- Select **Customize the Ribbon.**
- Select **Context Menu.**
- Select the **Category** and then the command that you want to add.
- Once you click on the **Add** button the function will be added to the **Right mouse button menu**.

WAYS TO GET THE MOST FROM MASTERCAM

Mastercam Training

In-House Solutions offers unsurpassed industrial training for Mastercam and Octopuz. We have training facilities in a number of cities across Canada and some of our courses can also be offered onsite, depending on trainer availability. Learn more at **eMastercam.com/store**.

Our library of **Mastercam Training Solutions** consists of several product lines that cater to any learning style. Learn Mastercam at your own pace with our **Training Tutorials**, teach your students with the help of our **Instructor Kits**, learn the theory behind Mastercam with our **Handbooks**, get projects à-la-carte with our **Single Projects**, let our instructors show you best practices with our **Video Training** or go digital with our **eBooks**.

Mastercam Community

eMastercam is the one-stop web resource for Mastercam users. People from all over the world visit the site whether they are teaching, learning or working with Mastercam daily. Members can post questions, comments or share projects and success stories. Visit eMastercam.com and sign up for your free account today!

For downloaded pdf please visit

www.emastercam.com/qrc

Table of Contents

Getting Started **1**

Tutorials:

Tutorial #1 - Open Contour Wirepath 15

Tutorial #2 - Contour Wirepath With Taper Angle And Land 71

Tutorial #3 - Contour Wirepaths For Both Inner And Outer Profile 127

Tutorial #4 - Multiple Contour Wirepaths and Multiple Tabs 215

Tutorial #5 - 4 Axis Wirepath 295

Tutorial #6 - No Core Wirepath 361

Quiz Answers **405**

General Notes **409**

TABLE OF CONTENTS

GETTING STARTED

Objectives:

✓ Starting Mastercam.
✓ The student will learn about the Graphical User Interface.
✓ The student will learn how to navigate through Mastercam.

STEP 1: STARTING MASTERCAM

For Windows 7

- Select the **Start** button.
- Select **All Programs** and click on **Mastercam 2018**.

For Windows 8

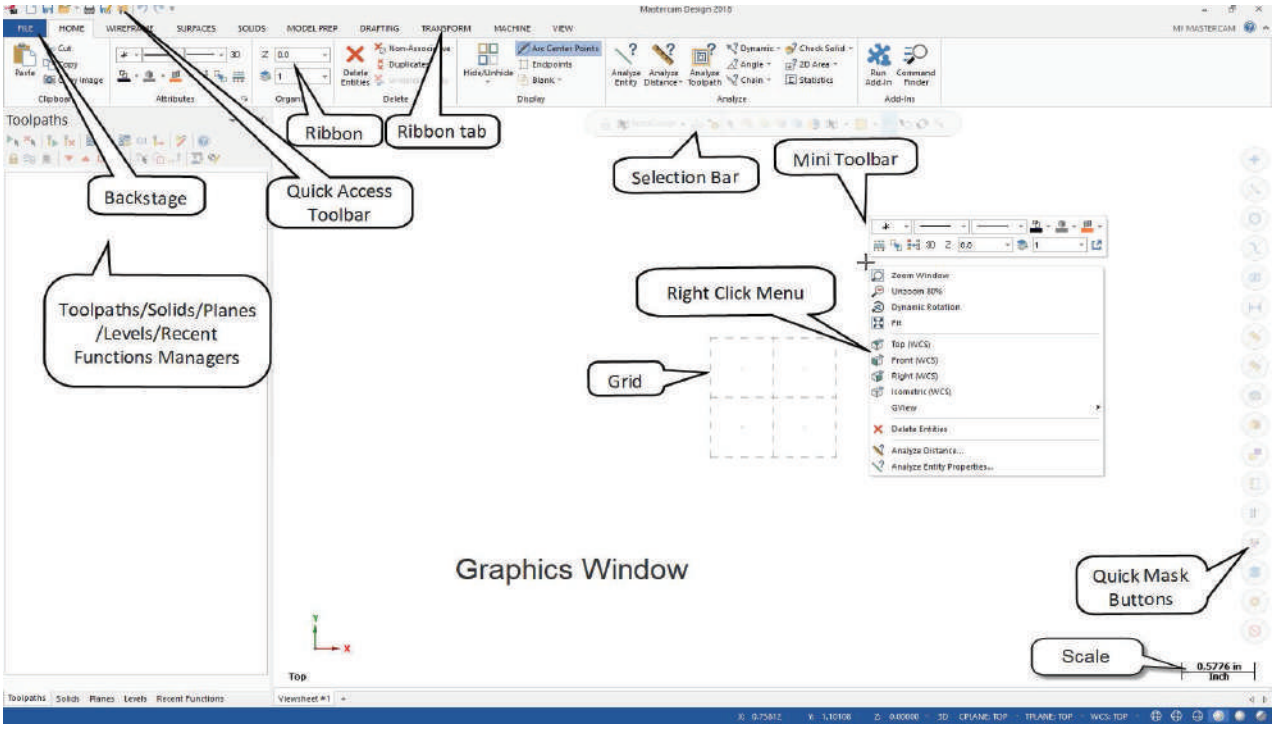
- Select the **Start** button.
- Click on the drop down arrow to open **Apps**.
- Find and click on **Mastercam 2018**.

For Windows 10

- Select the **Start** button.
- Click on the drop down arrow to open **Apps**.
- Find and click on **Mastercam 2018**.

- 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	Contain 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	Let 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.

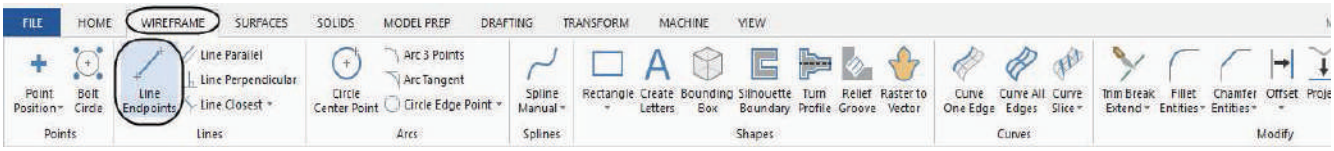
STEP 3: NAVIGATE THROUGH MASTERCAM

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

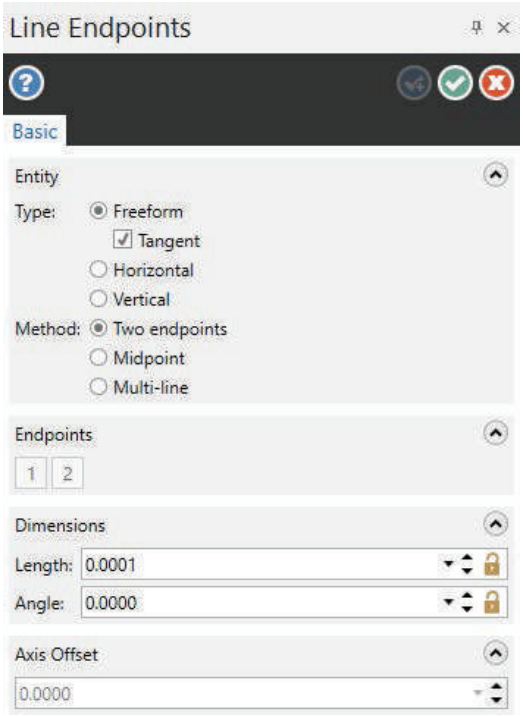
3.1 Using the Wireframe tab to select the command to create Line Endpoints

- Left click on **WIREFRAME.**
- Left click on the **Line Endpoints** icon as shown in Figure: 3.1.1.

Figure: 3.1.1



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



Sketching a line

- To sketch a line, left click on two locations on the screen.

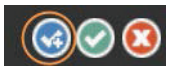
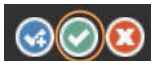
Creating a line knowing the endpoint coordinates

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

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** (Quick Access Toolbar) 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.

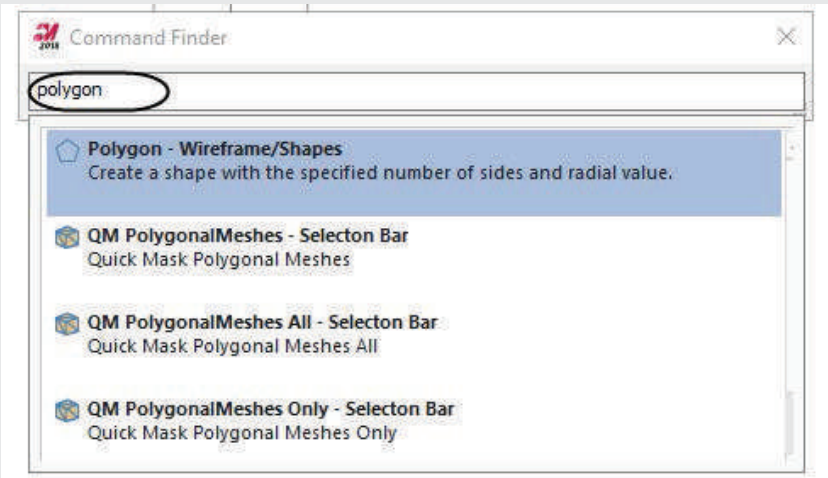
  Mastercam also has a **Redo** button for your convenience.

3.2 Function Prompt

Prompts the user to execute a command.

- 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: SET 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

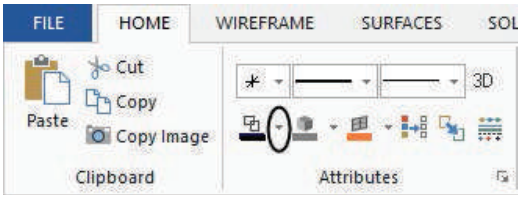
Point Style	Displays and sets the system's point style.
Line Style	Displays and sets the system's line style.
Line Width	Displays and sets the current system's line width.
Color	Assigns the current color to wireframe, solid and surface entities. To change the current color, click in the specific color field and select a color from the color pallet. To change an existing geometry color, select the geometry first and then click in the color field and select a color from the color pallet.
Clear Color	When performing a transform function (Xform), 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

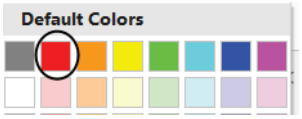
Z Depth	Sets the current construction depth. To set this, click the drop down arrow and pick one from the most recently used list or click the **Z:** label and pick a point in the graphics window to use the Z depth values based on the selected entity.
Level	Sets the main level you want to work with in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

Change the Wireframe Color

- Click on the drop down arrow next to the **Wireframe Color** field as shown.



- Select the desired color from the dialog box as shown.



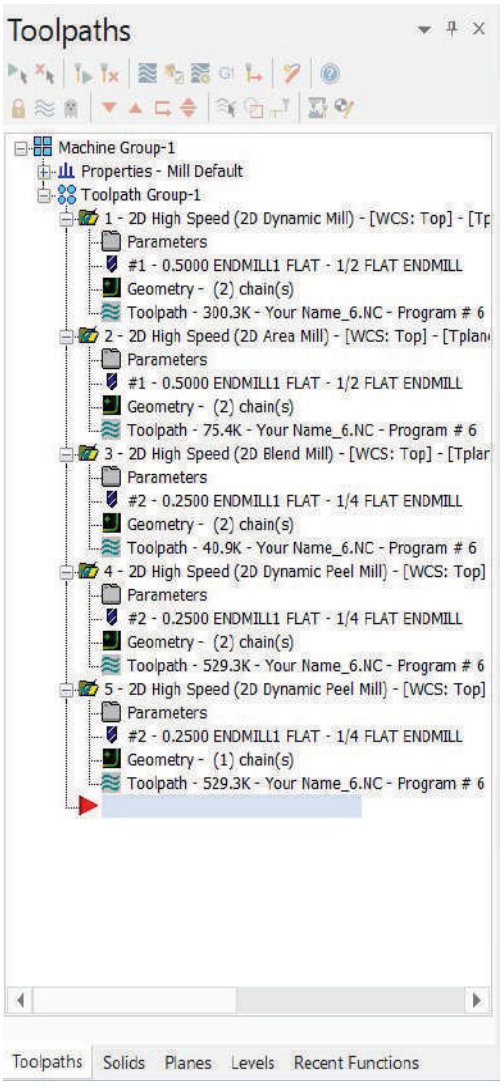
NOTE: Any geometry on your screen will remain in the previous system color. This change will only affect the geometry you create going forward.
To change the color of existing geometry, select the entities first and then click on the drop down arrow next to the **Wireframe Color** and select the desired color.
The same method can be applied for any other attribute that you want to set or change.

STEP 5: MANAGER PANELS

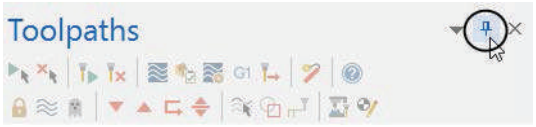
5.1 The Toolpaths Manager

The **Toolpaths Manager** displays all the operations for the current part. You can sort, edit, regenerate, verify and post any operation as shown in Figure: 5.1.1. For more information on the **Toolpaths Manager**, please refer to **General Notes** or click on the **Help** icon.

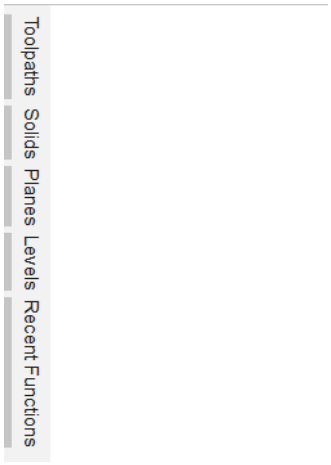
Figure: 5.1.1



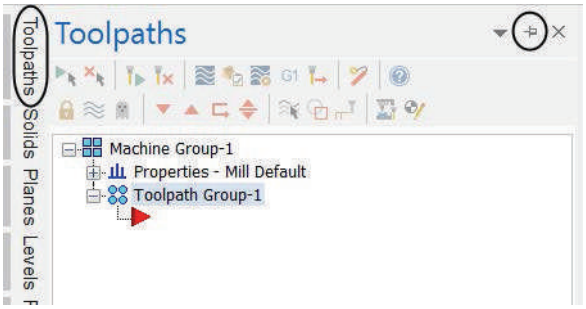
- The **Toolpaths Manager, Solids Manager**, or **Planes Manager** can be hidden to gain more space in the graphics area for creating geometry. Use **Auto Hide** icon to close all **Toolpaths, Solids, Planes** and **Levels Manager** panels.



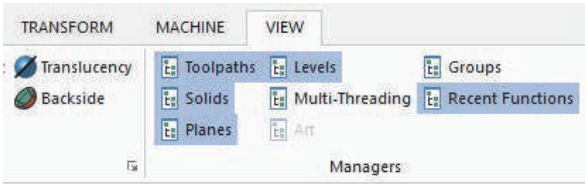
- The panels will be hidden to the left of the graphics window as shown.



- 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**, you 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 switch the system to imperial and how to set the imperial system as your default. You will have to select the **Backstage** options and select the system configuration.

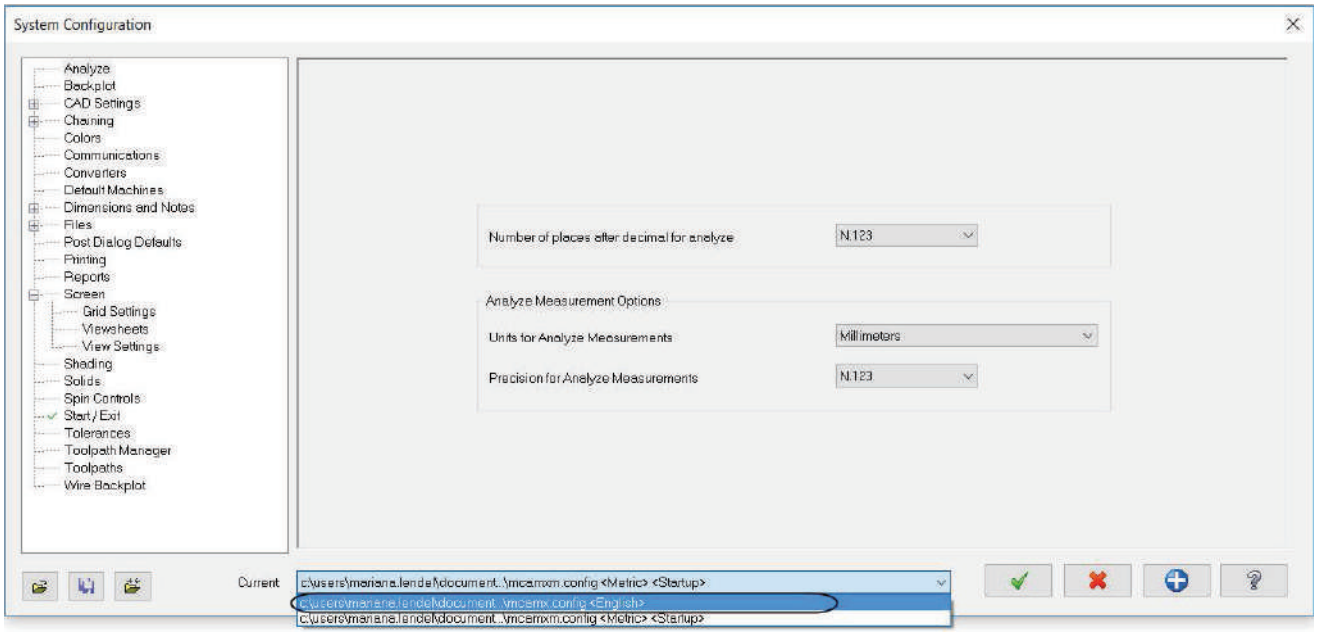
6.1 Setting Mastercam to imperial for the current session only

> **NOTE:** You may need to switch Mastercam to run in Imperial mode.

FILE

- **Configuration.**
- Select the drop down arrow beside **Current** as shown in Figure: 6.1.1.
- Select **mcamxm.config <English>** as shown.

Figure: 6.1.1



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

> **NOTE:** If you have a drawing on the screen it may ask you to scale the current part to imperial. Choose Yes if you wish to do this.

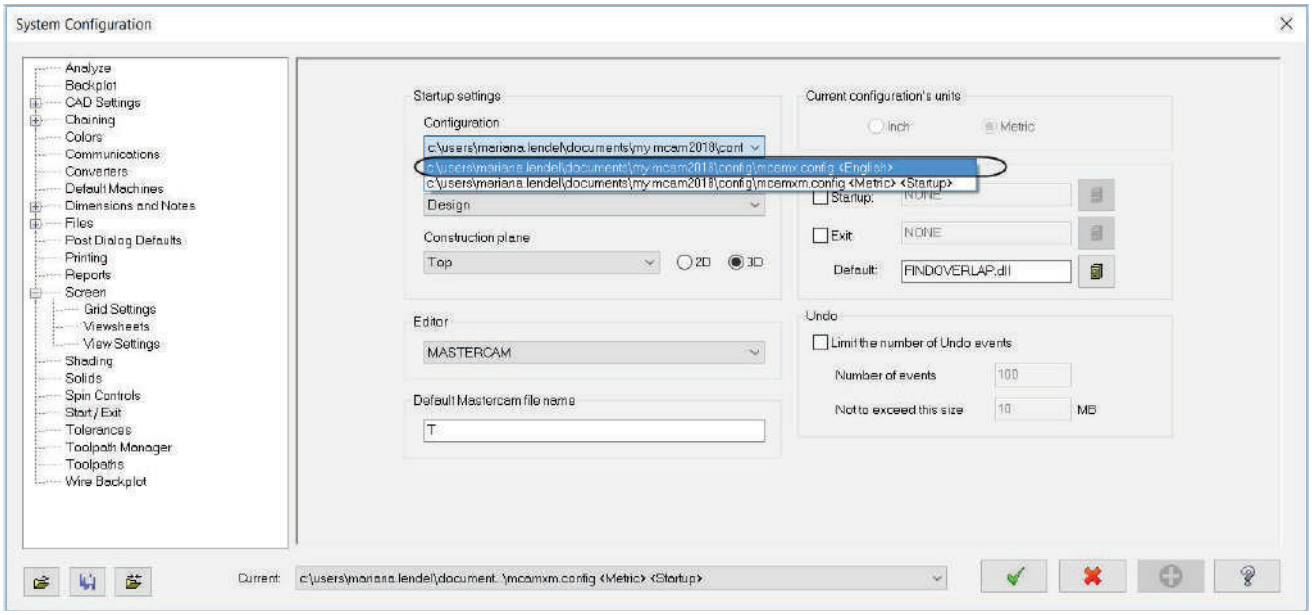
6.2 Setting Mastercam to imperial as a default

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

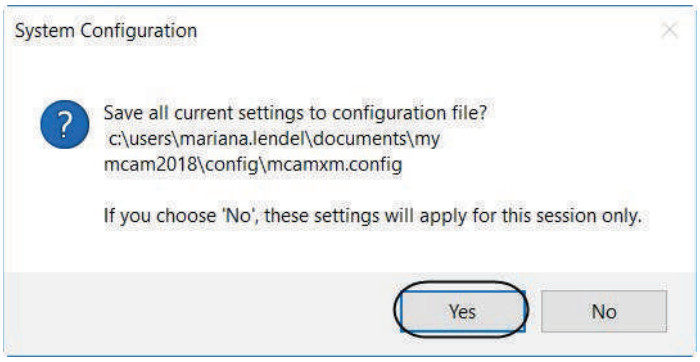
FILE

- **Configuration.**
- Select **Start/Exit** from the configuration topics.
- Select the drop down arrow below **Configuration** in the **Startup settings** area as shown in Figure: 6.2.1.
- Select **mcamxm.config <English>** as shown.

Figure: 6.2.1



- 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**.



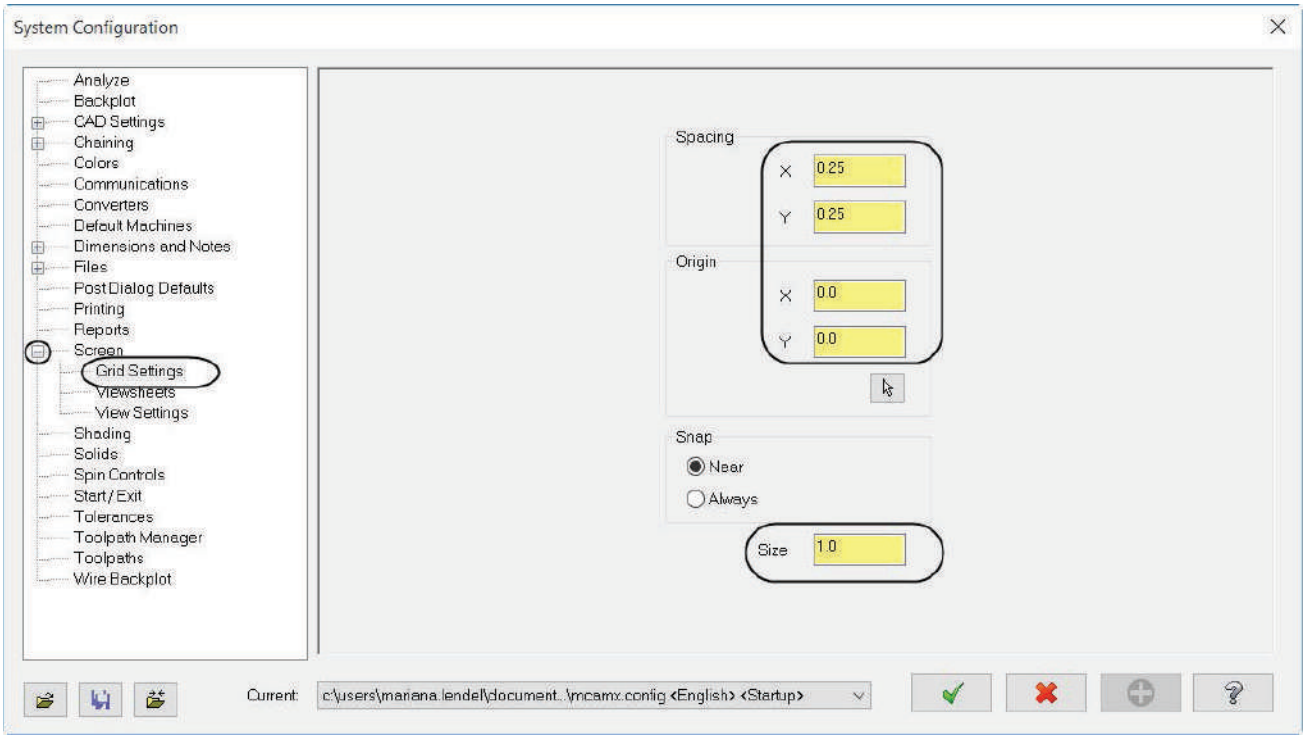
STEP 7: SET THE GRID

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.

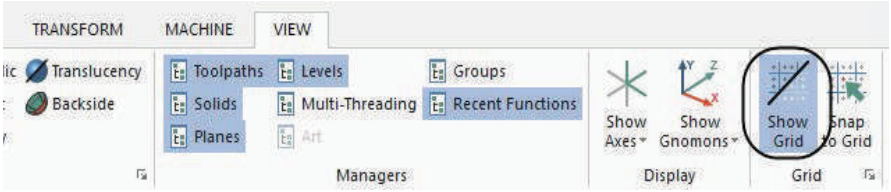
FILE

- **Configuration**.
- Select **Screen** from the configuration **Topics.**
- Select the plus sign (+) beside **Screen** as shown in Figure: 7.0.1.

Figure: 7.0.1

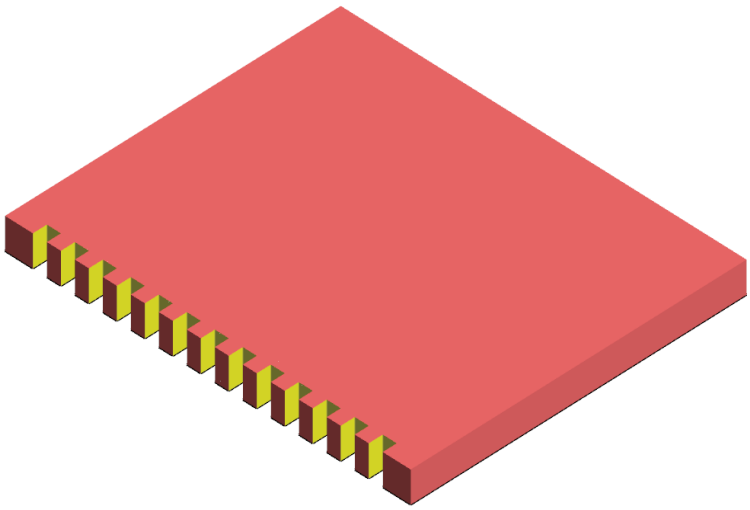


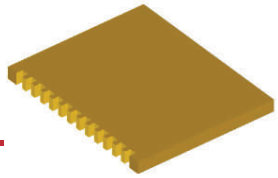
- In **Grid Settings**, change the **Spacing** to **X = 0.25** and **Y = 0.25**.
- Set the **Size** to **1.0.**
- Choose the **OK** button to exit.
- Select the **Yes** button to save the settings in the **System Configuration**.
- To see the **Grid** in the graphics window, from the **VIEW** tab, enable **Show Grid** as shown.



- The grid should look as shown.

TUTORIAL #1





OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

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.

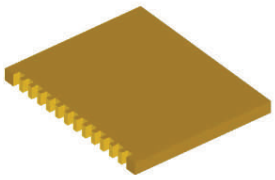
Backplot and Verify the file:

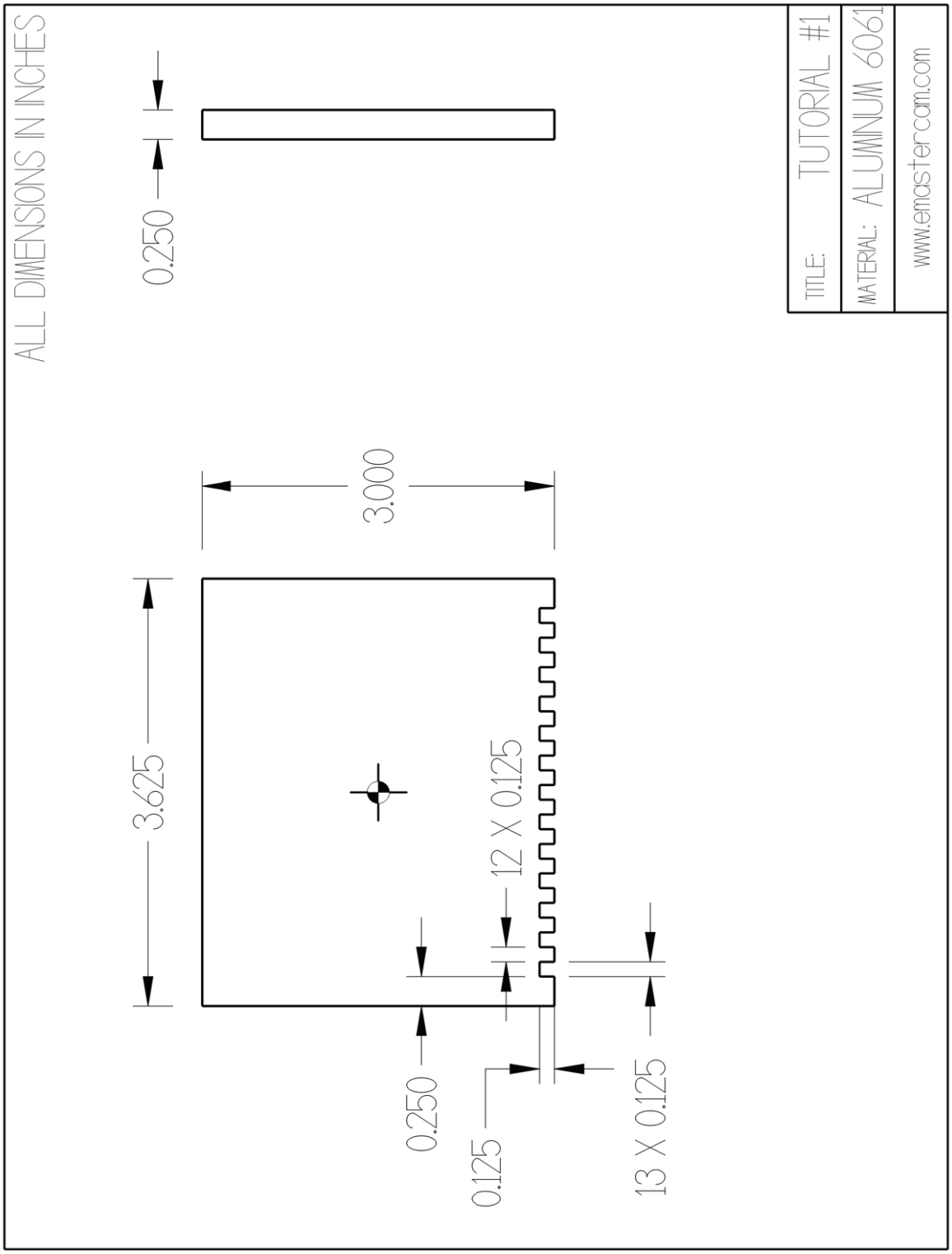
- The Backplot will be used to simulate a step-by-step process of the wire's movements.
- The Verify will be used to watch the wire machine the part out of a solid model.

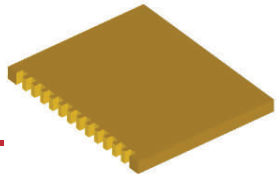
Post Process the file to generate the G-code:

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

This tutorial takes approximately 45 minutes to complete.




ALL DIMENSIONS IN INCHES
0.250
3.000
3.625
12 X 0.125
0.250
0.125
13 X 0.125
TITLE: TUTORIAL #1
MATERIAL: ALUMINUM 6061
www.emastercam.com



GEOMETRY CREATION

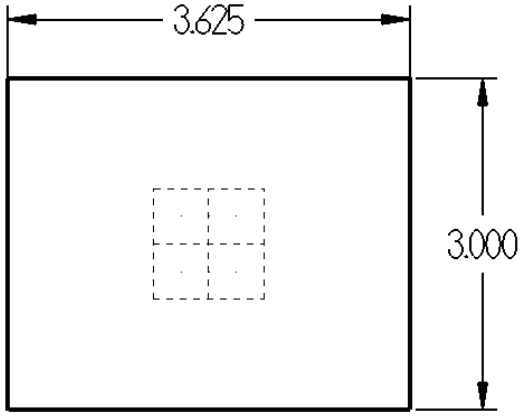
STEP 1: SETTING UP THE GRAPHIC USER INTERFACE

Please refer to the **Getting Started** section to set up the graphical user interface.

STEP 2: CREATE THE 3.625" BY 3.0" RECTANGLE

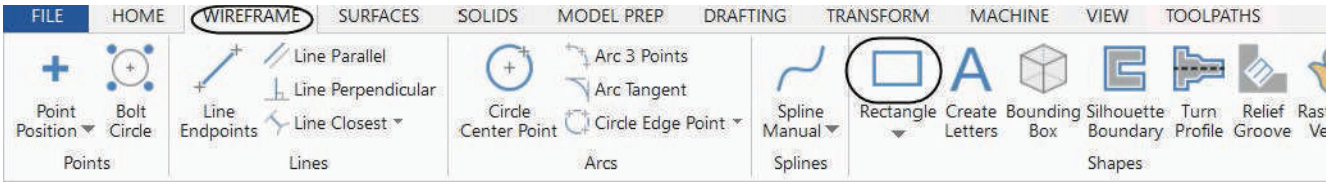
In this step you will learn how to create a rectangle given the width, the height, and the anchor location.

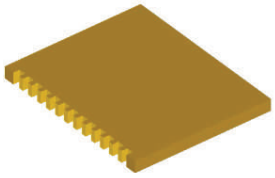
Step Preview:



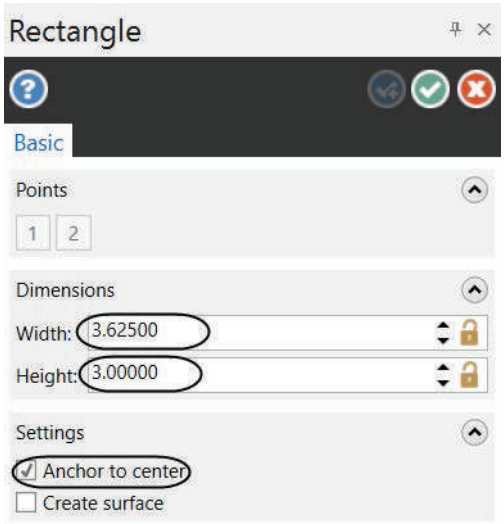
WIREFRAME

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



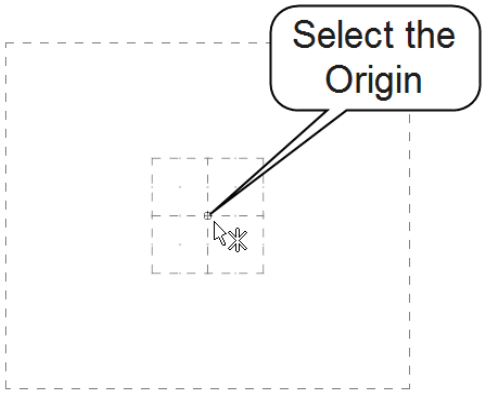


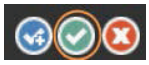
- Enter the **Width** and **Height** and enable **Anchor to center** as shown.



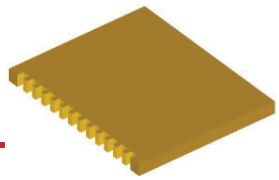
- [Select position of base point]: Select the Origin as shown in Figure: 2.0.1.

Figure: 2.0.1



- 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** from the keyboard.

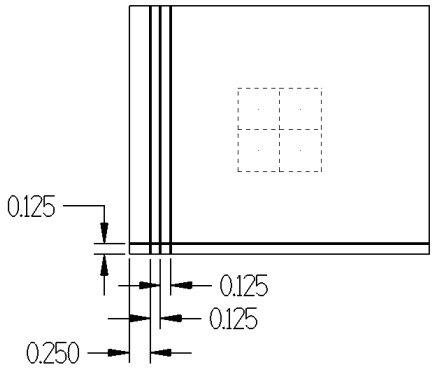


NOTE: In the next few steps, you will create the profile with 13 identical shapes. You will create one shape first, and then, by using the **Translate** command with the Copy option, generate the other twelve shapes.

STEP 3: CREATE PARALLEL LINES

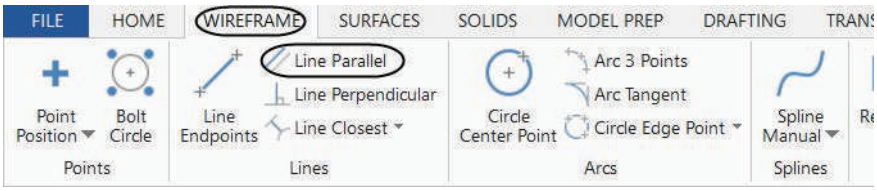
In this step you will create the four parallel lines that will be used to create one of the shapes.

Step Preview:



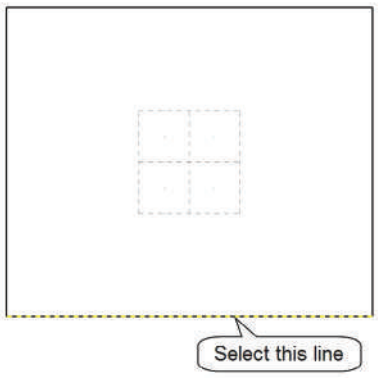
WIREFRAME

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

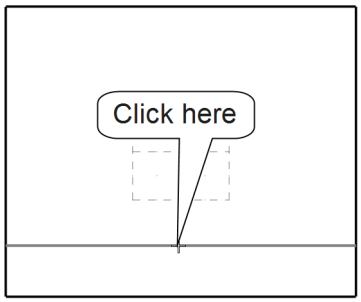


- [Select a line]: Select the lower horizontal line of the rectangle as shown in Figure: 3.0.1.

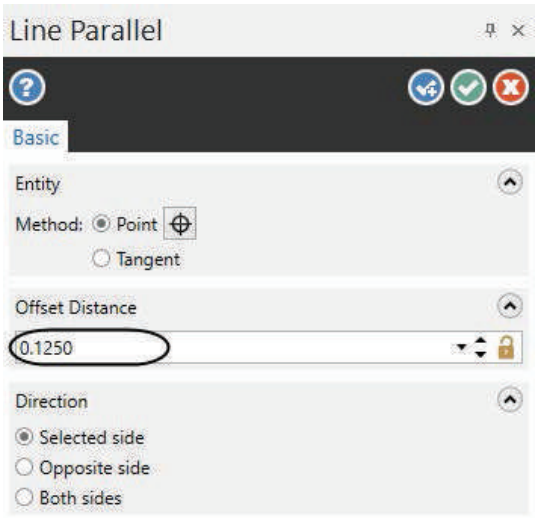
Figure: 3.0.1

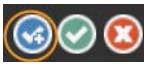


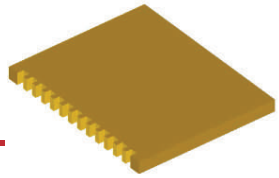
- [Select the point to place a parallel line through]: Select a point above the line as shown.



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

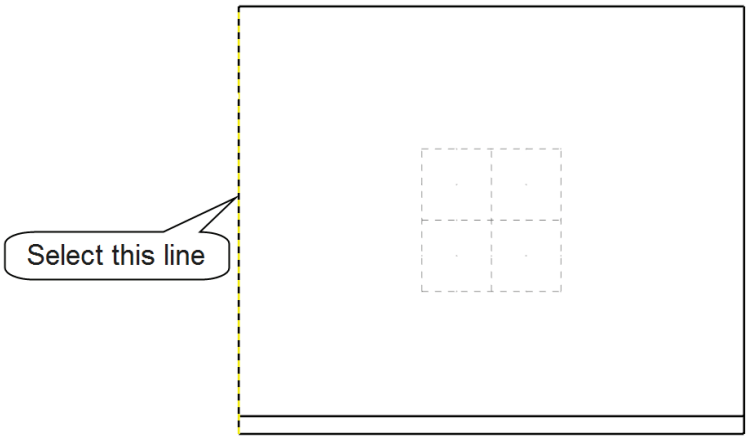


- Select **OK and Create New Operation** to stay in the command. 

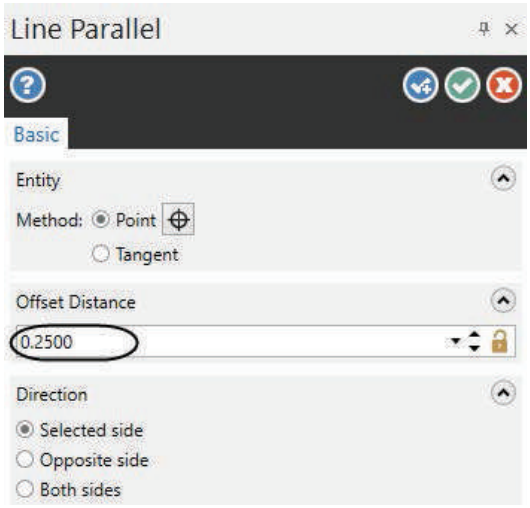


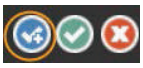
- [Select a line]: Select the left side vertical line of the rectangle as shown in Figure: 3.0.2.

Figure: 3.0.2



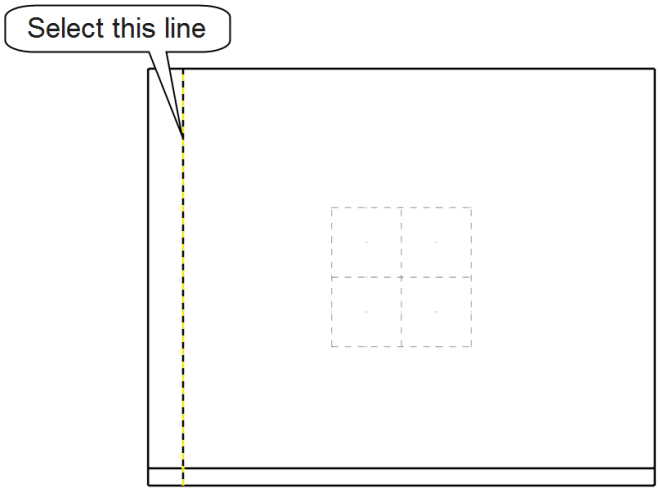
- [Select the point to place a parallel line through]: Select a point to the right of the selected line.
- Under **Offset Distance**, enter **0.25** as shown. Press **Enter.**



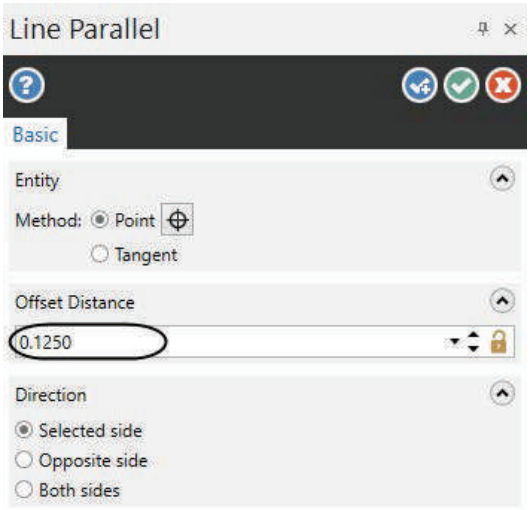
- Select **OK and Create New** to stay in the command. 

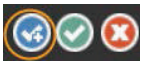
- [Select a line]: Select the vertical line that you just created as shown in Figure: 3.0.3.

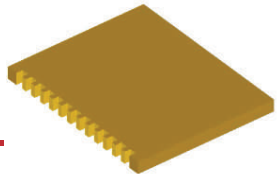
Figure: 3.0.3



- [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.**

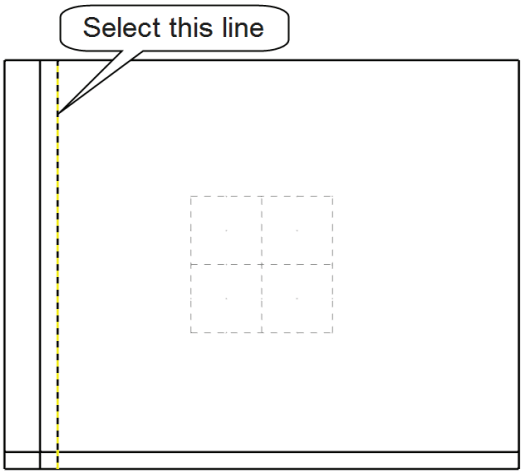


- Select **OK and Create New** to stay in the command. 

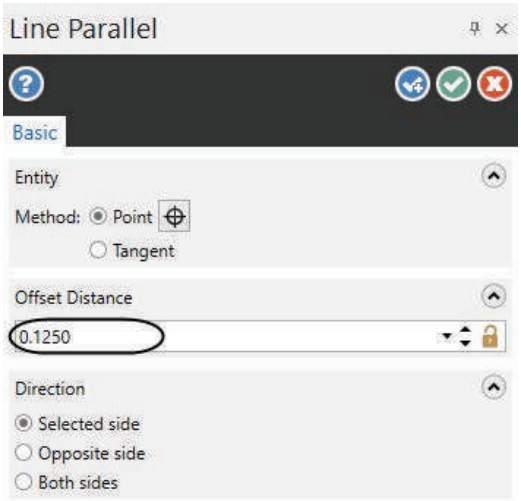


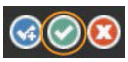
- [Select a line]: Select the vertical line that you just created as shown in Figure: 3.0.4.

Figure: 3.0.4

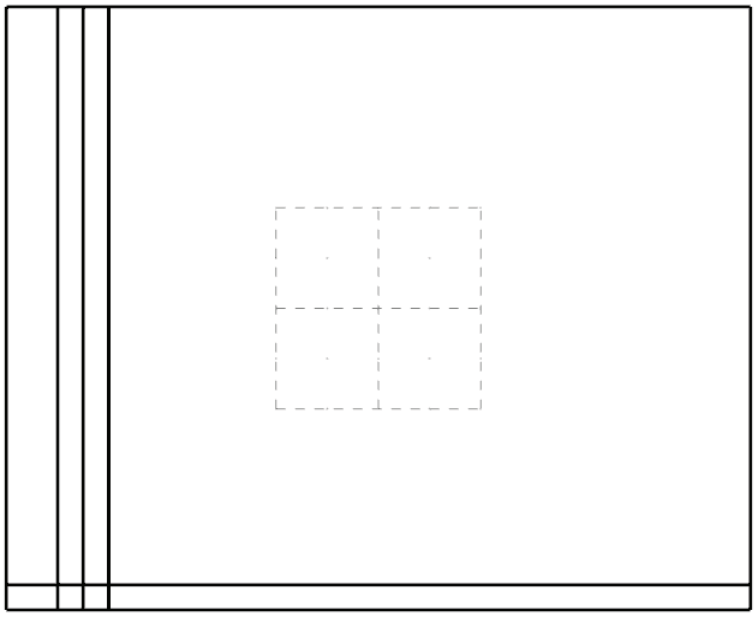


- [Select the point to place a parallel line through]: Select a point to the right of the selected line.
- Under **Offset Distance**, enter **0.125** as shown. Press **Enter**



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

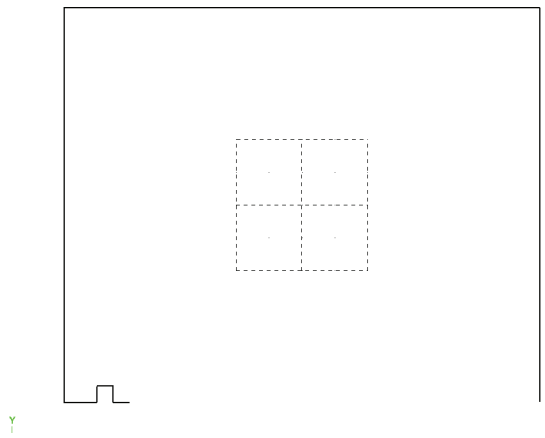
- The geometry should look as shown.



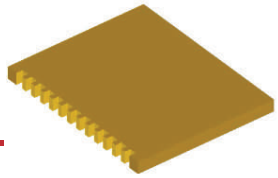
STEP 4: TRIM THE LINES

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

Step Preview:

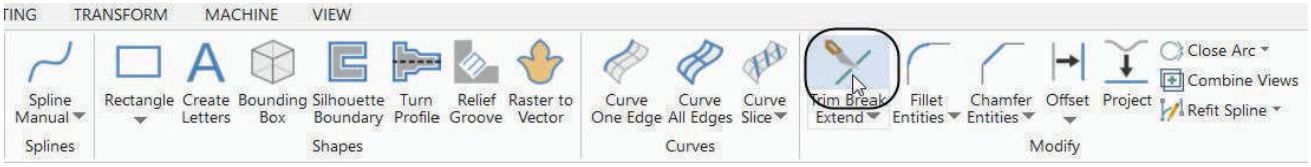


NOTE: During the trimming process it is very important to select the entities exactly in the order and at the locations as shown in the graphics on the following pages.
Trim 3 entities command requires a special selection. The first two entities that you select are trimmed to the third, which acts as a trimming curve. The third entity is then trimmed to the first two.

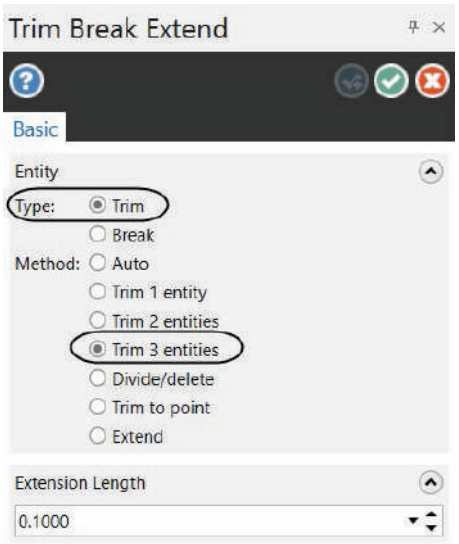


WIREFRAME

- From the **Modify** group, select the **Trim Break Extend** icon as shown.

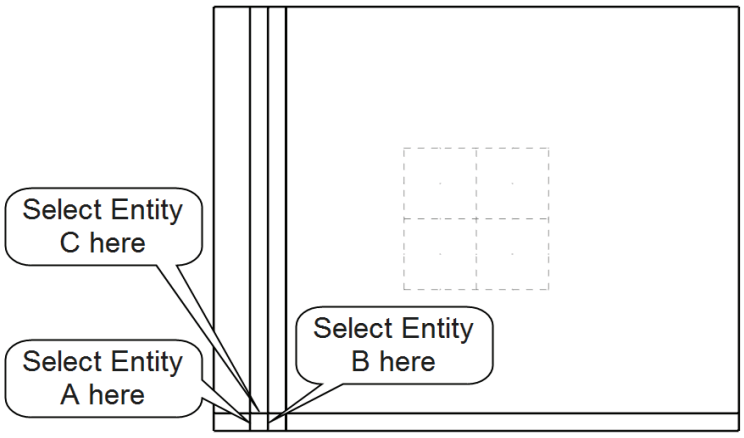


- In the **Trim Break Extend** panel, enable **Trim 3 entities** as shown.

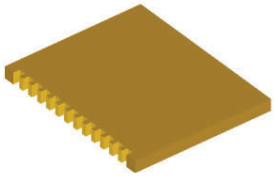


- [Select the first entity to trim/extend]: Click on Entity A as shown in Figure: 4.0.1.
- [Select the second entity to trim/extend]: Click on Entity B as shown in the same figure.
- [Select the entity to trim/extend to]:Click on Entity C as shown in the same figure.

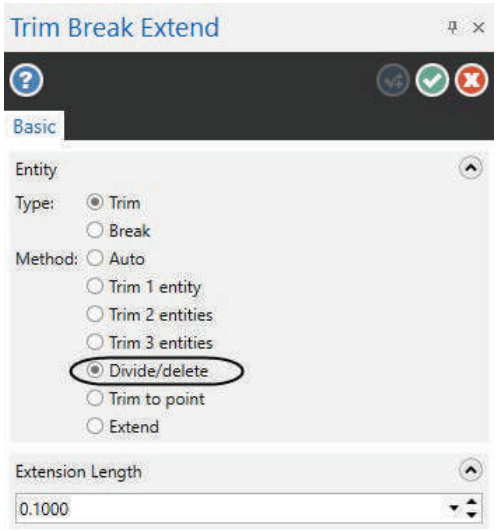
Figure: 4.0.1



NOTE: Mastercam's auto-preview feature displays the result of the selected function as you move the mouse to select the last entity. A solid line represents what will be created, while a dashed line represents what will be removed.

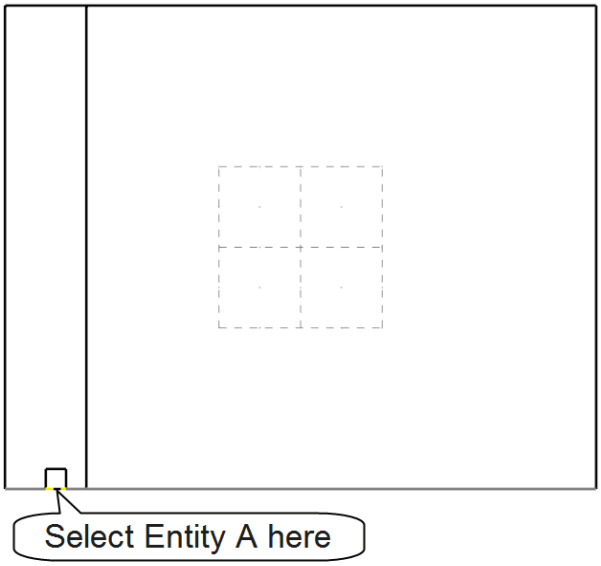


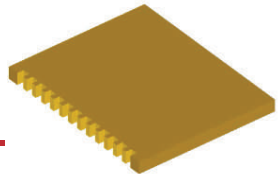
- Enable **Divide/Delete** as shown.



- [Select the curve to divide / delete]: Click on Entity A first at the location shown in Figure: 4.0.2.

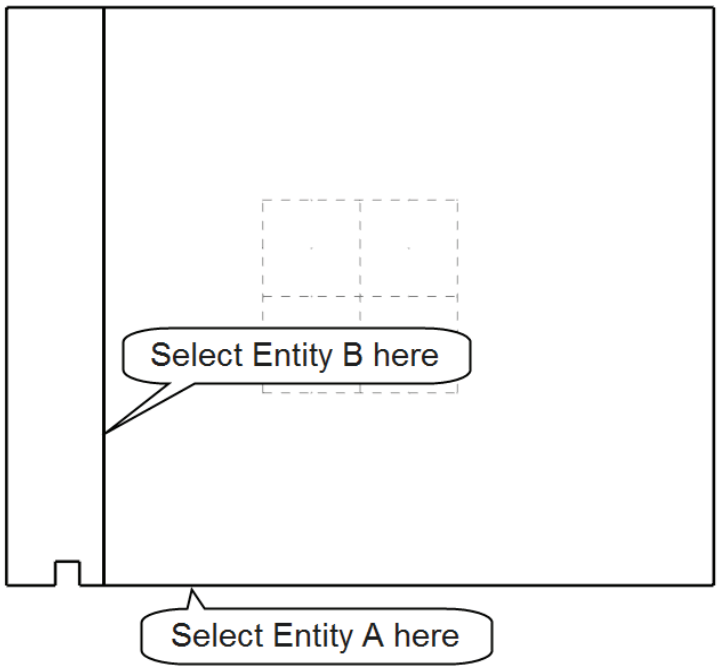
Figure: 4.0.2



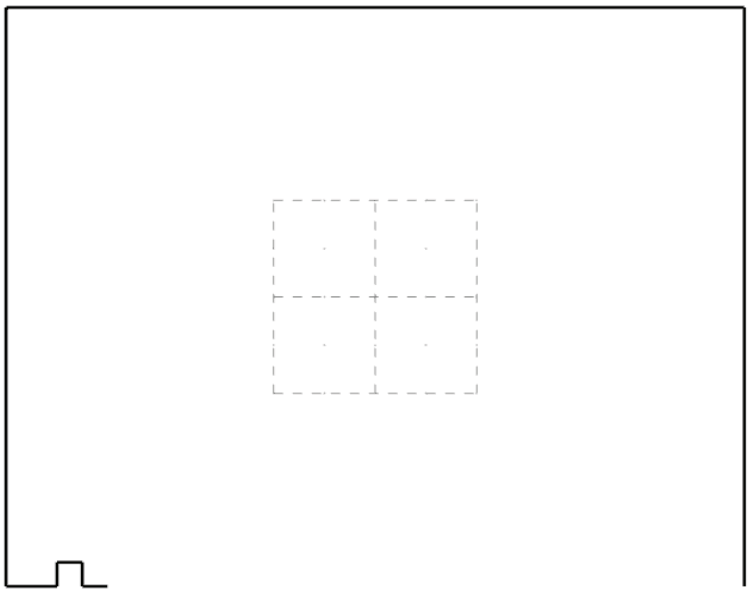


- [Select the curve to divide/delete]: Click on Entity A as shown in Figure: 4.0.3.

Figure: 4.0.3



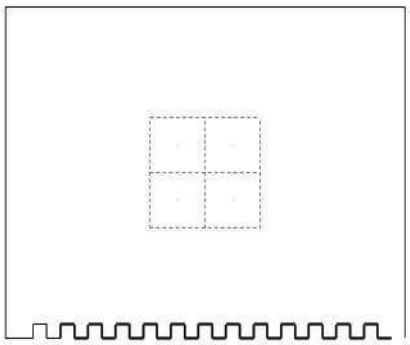
- [Select the curve to divide/delete]: Click on Entity B at the location as shown in Figure: 4.0.3.
- Select the **OK** button to exit the command.
- The drawing should look as shown.



STEP 5: TRANSLATE/COPY THE SHAPE

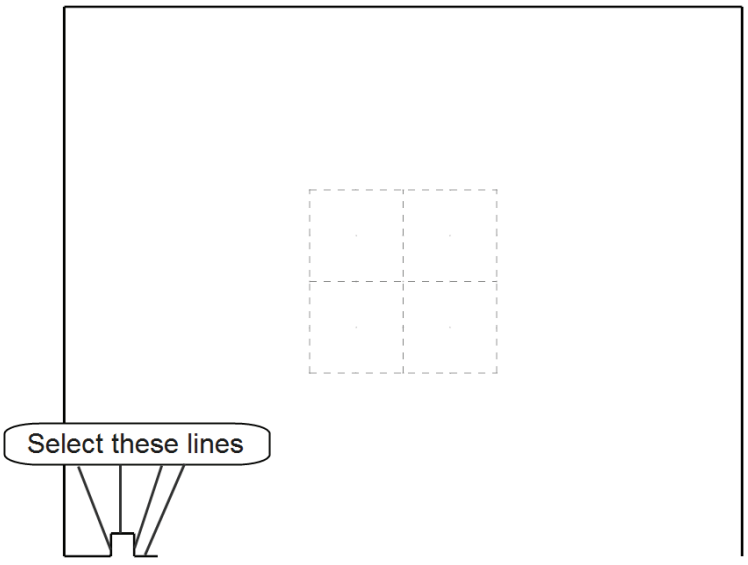
In this step you will learn how to create the rest of the shapes using the **Translate** command with the copy option enabled. The shape will be copied 12 times and translated along the X-Axis.

Step Preview:



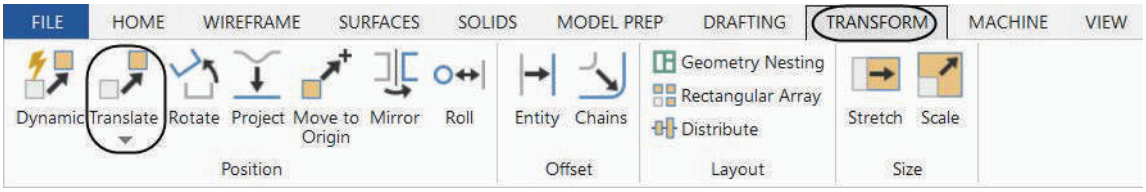
- Select the four lines to be translated as shown in Figure: 5.0.1.

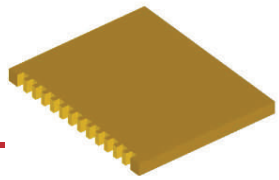
Figure: 5.0.1



TRANSFORM

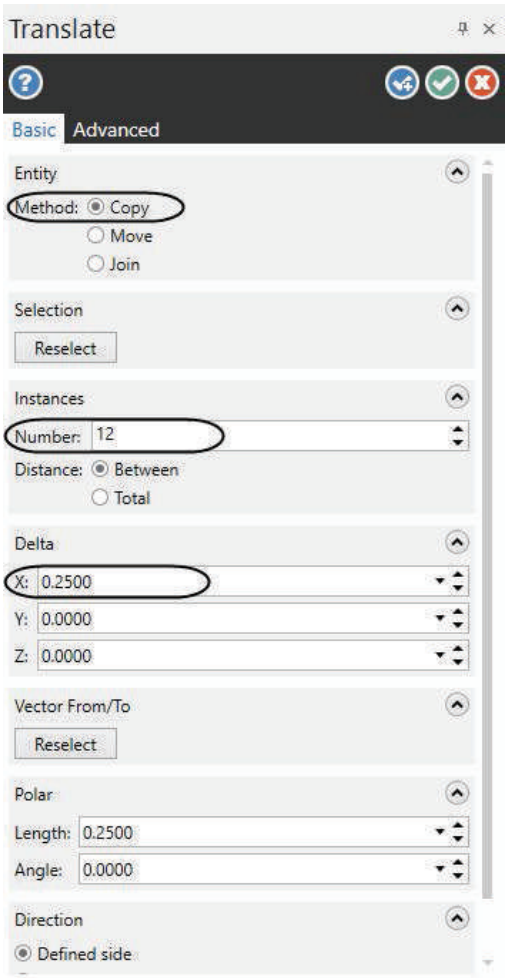
- From the **Position** group, select the **Translate** icon.



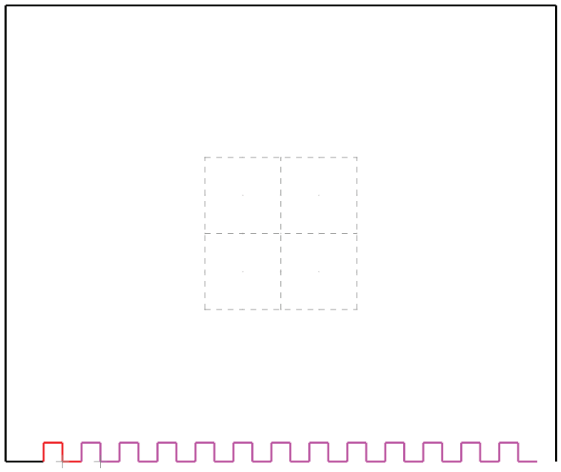


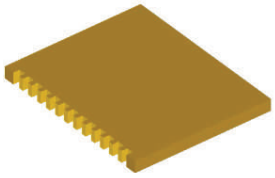
- Make sure that **Copy** is enabled in the **Translate** panel as shown in Figure: 5.0.2.
- The number of **Instances** should be set to **12** and the **Delta X** distance to **0.25** as shown in Figure: 5.0.2.

Figure: 5.0.2



- Press **Enter** and the preview of the generated geometry should look as shown.

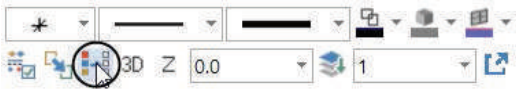




- Select the **OK** button to exit the **Translate** panel.

NOTE: When performing a transform operation, Mastercam creates a temporary group from the original, highlighted in red, and a result from the transformed entities, highlighted in magenta. These two colors will remain until you use the **Clear Colors** function or perform another Transform operation.

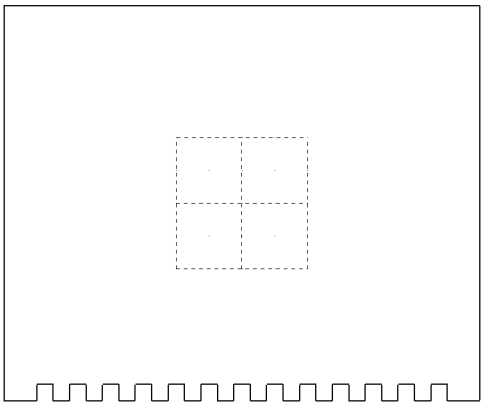
- Right mouse click on the graphics window and select **Clear Colors** as shown to reset the transformed entities to their default color.



STEP 6: TRIM THE LINES USING TRIM 2 ENTITIES COMMAND

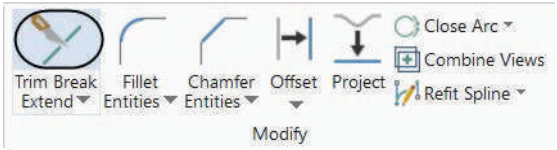
This step shows how to trim two intersecting entities using the Trim command. To trim both entities to their intersection, enable **Trim 2 entity** or single click the first entity and double click the second.

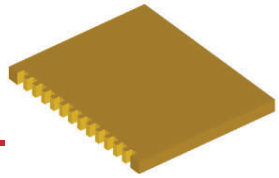
Step Preview:



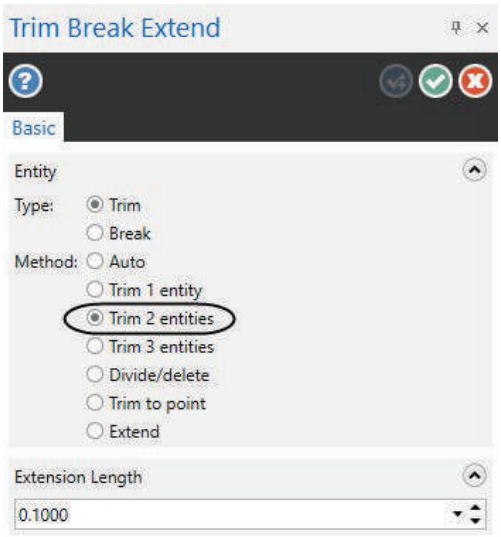
WIREFRAME

- From the **Modify** group, select **Trim Break Extend** icon as shown.



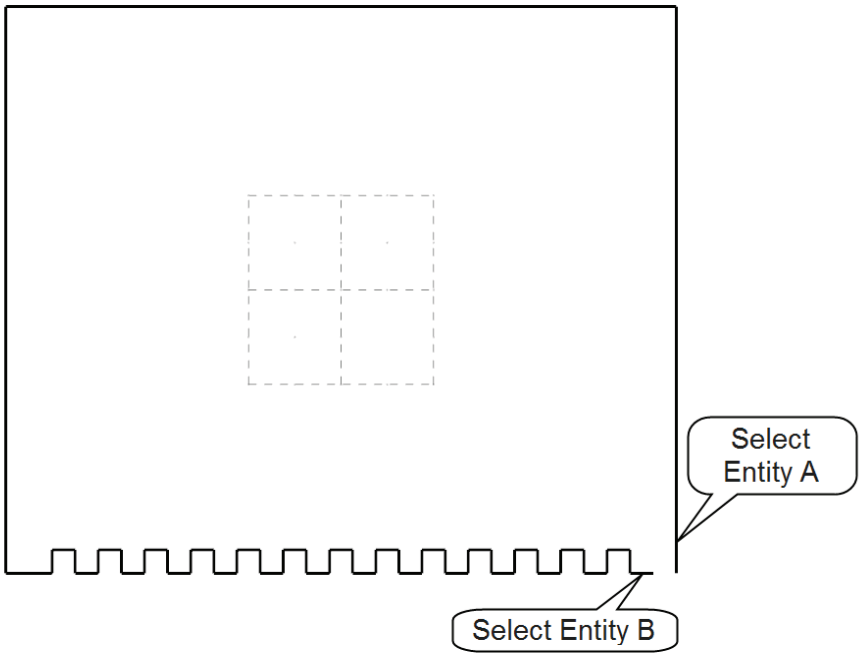


- Enable **Trim 2 entities** as shown.

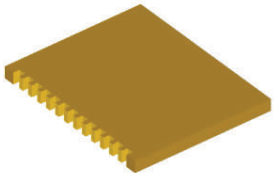


- [Select the entity to trim/extend]: Click on Entity A as shown in Figure: 6.0.1.
- [Select the entity to trim/extend to]: Click on Entity B as shown in Figure: 6.0.1.

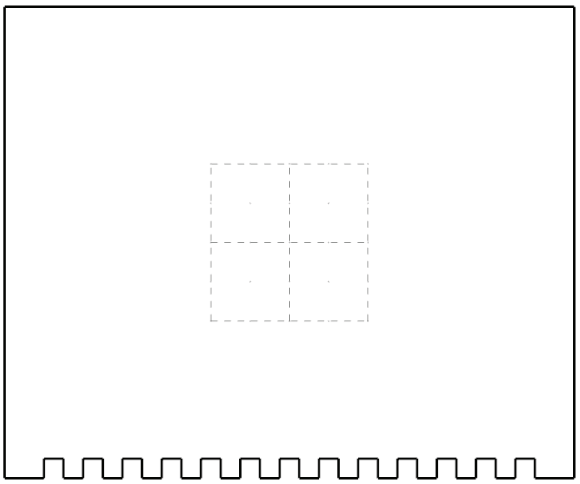
Figure: 6.0.1



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



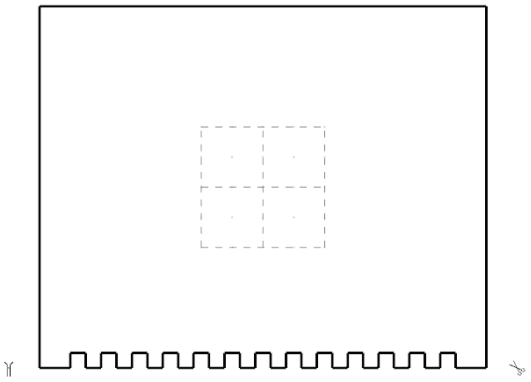
- The drawing should look as shown.

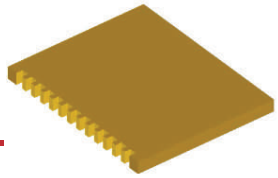


STEP 7: CREATE THE THREAD AND THE CUT POINTS

Thread Point is the point where the machine threads the wire, usually a pre-drilled hole in the material. The wire cuts from the thread position to the start of the chain based on the settings for **Lead in** and **Lead out**. **Cut Point** is the point where the wire machine cuts the wire before moving to the next thread point. See **Getting Started** on how to obtain the Wire utilities toolbar.

Step Preview:

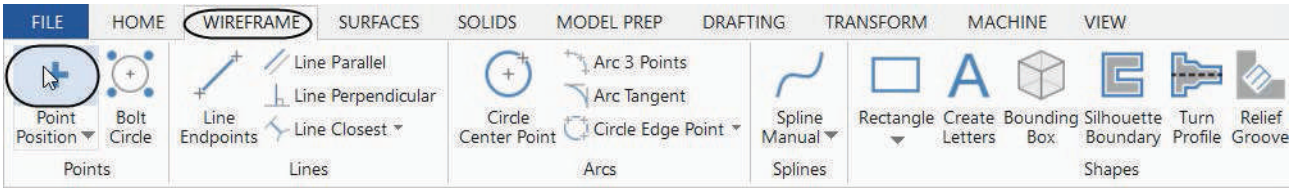




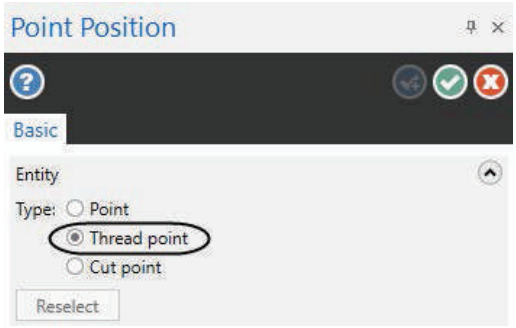
7.1 Create the Thread Point

WIREFRAME

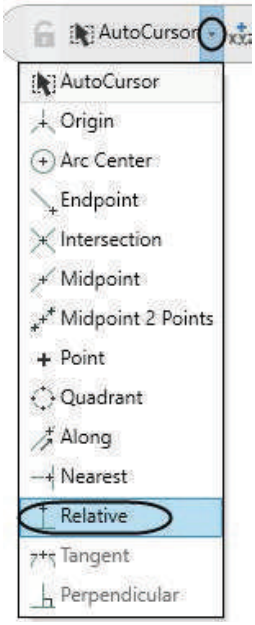
- From the **Points** group, select **Point Position** as shown.



- Select **Thread Point** in the **Point Position** panel as shown.

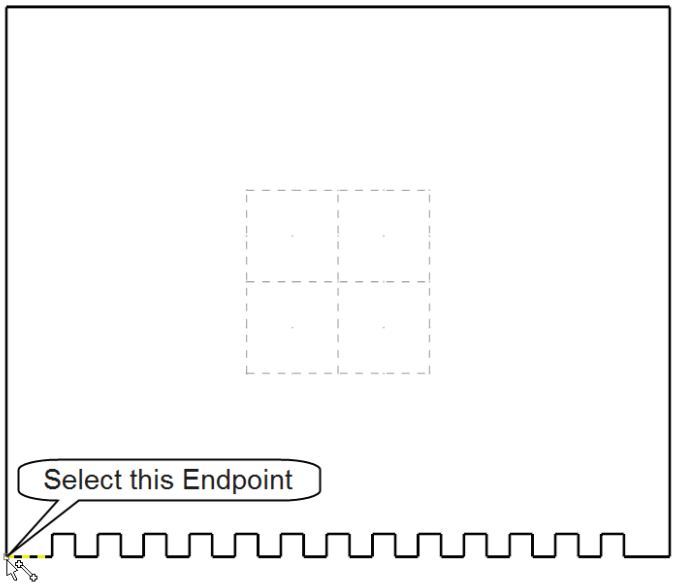


- From the **AutoCursor** drop down list, select **Relative**.

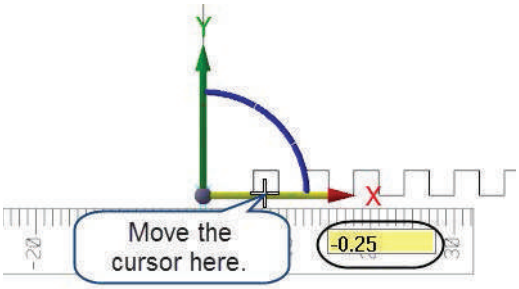


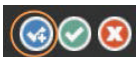
- [Pick a known point to use as a reference]: Select the Endpoint as shown in Figure: 7.1.1.

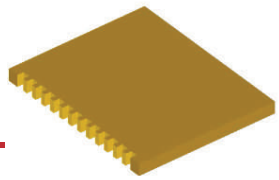
Figure: 7.1.1



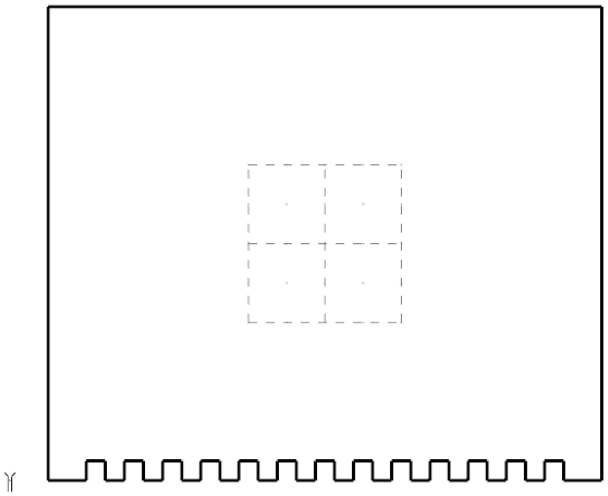
- Make sure that when selecting the endpoint, the visual cue of the cursor changes as shown.
- Hover the cursor above the **X axis** as shown and enter **-0.25** to create the thread point to the left of the reference point.



- Press **Enter** three times to create the point.
- Select **OK and Create New Operation** to stay within the command. 

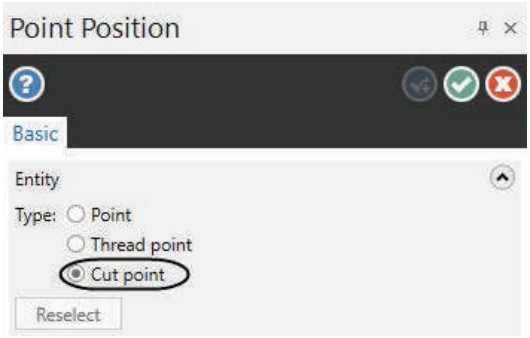


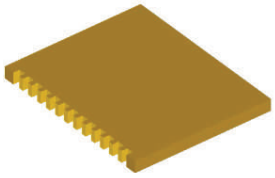
- The geometry should look as shown.



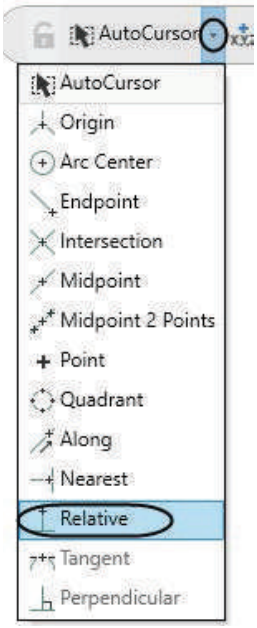
7.2 Create the Cut Point

- Select **Cut Point** in the **Point Position** panel as shown.



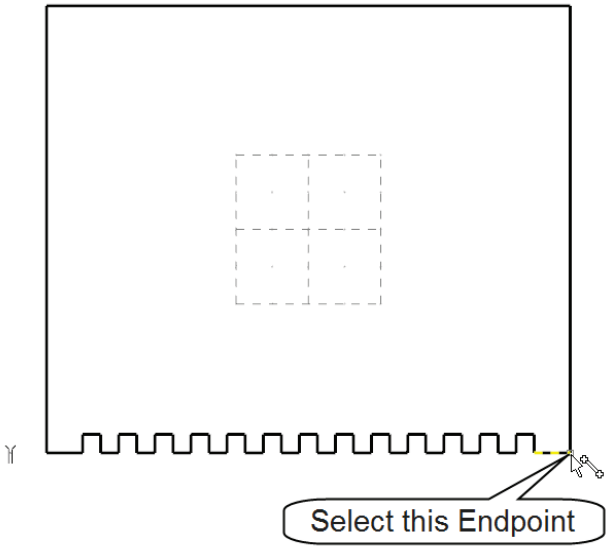


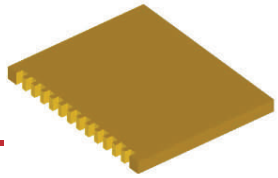
- From the **AutoCursor** drop down list, select **Relative**.



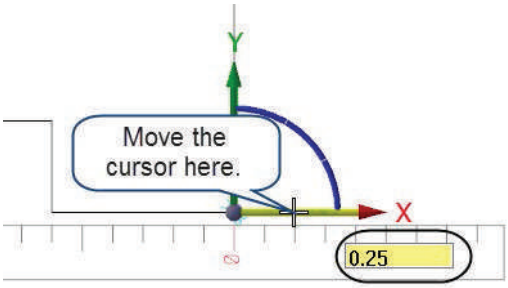
- [Pick a known point to use as a reference]: Select the Endpoint as shown in Figure: 7.2.1.

Figure: 7.2.1

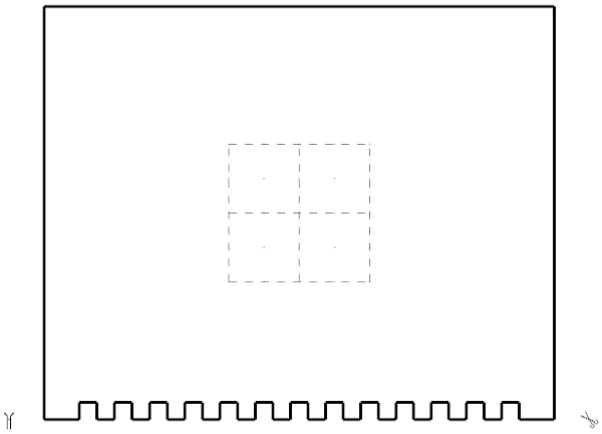




- Hover the cursor above the **X axis** as shown and when the scale appears, enter **0.25** to create a cut point to the right of the reference point and along the **X axis**.

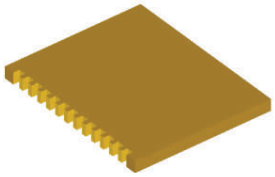


- Press **Enter** three times to create the point.
- Select the **OK** button to complete the command.
- The geometry should look as shown.



STEP 8: SAVE THE FILE

- From the **Quick Access Toolbar**, select the **Save As** icon.
- Save the file as "Tutorial 1_ Open Contour.mcam."



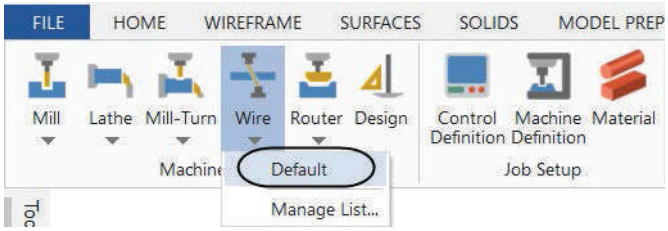
STEP 9: SELECT THE MACHINE AND SET UP THE STOCK

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.MCAM-WMD. Once finished, select OK to exit the dialog box. Otherwise, please follow next step.

MACHINE

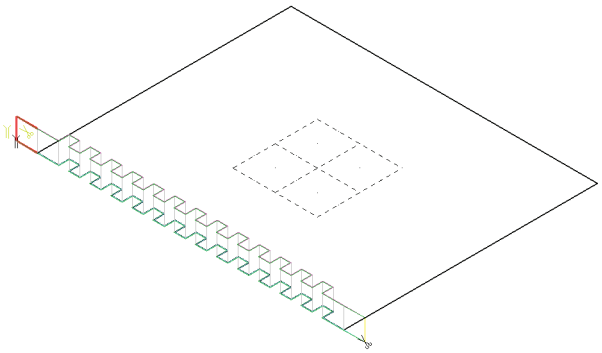
- From the **Machine Type** group, click on the pull down arrow under **Wire** and select **Default** as shown.

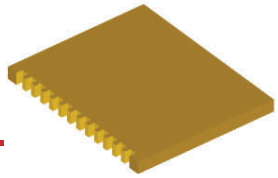


STEP 10: MACHINE THE OPEN PROFILE USING CONTOUR WIREPATH

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.

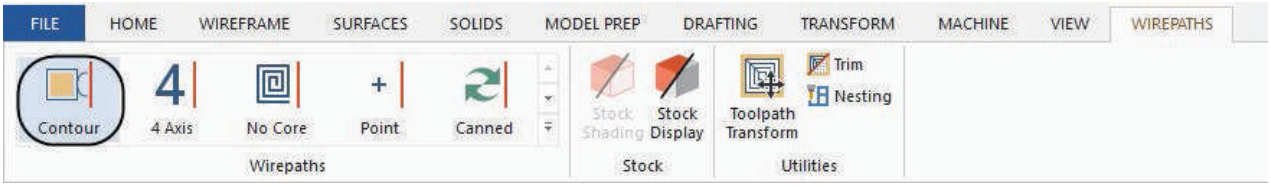
Toolpath Preview:



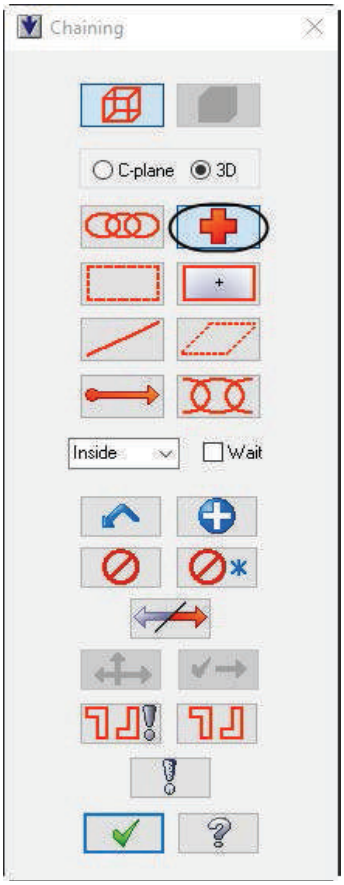


WIREPATHS

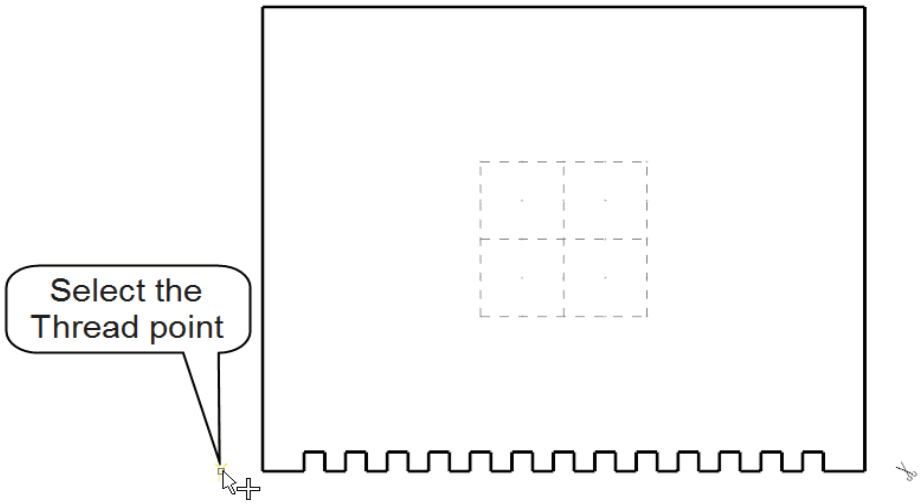
- From the **Wirepaths** group, select **Contour**.



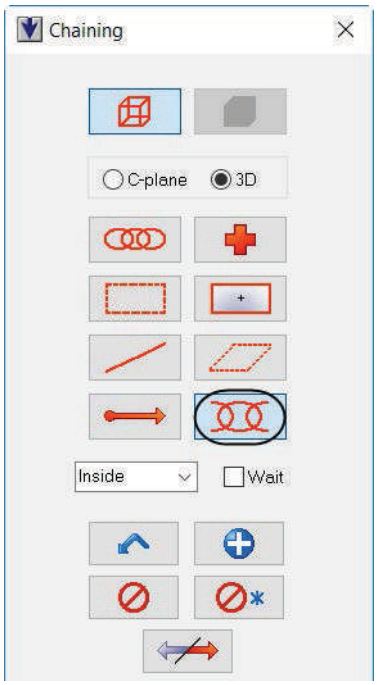
- From the **Chaining** dialog box, select the **Point** button.



- [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 in Figure: 10.0.1.

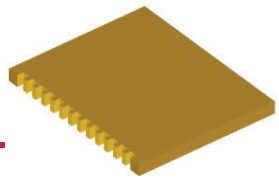
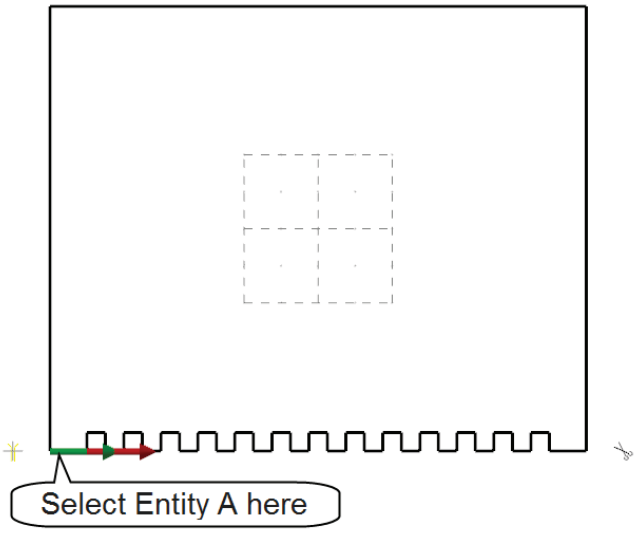


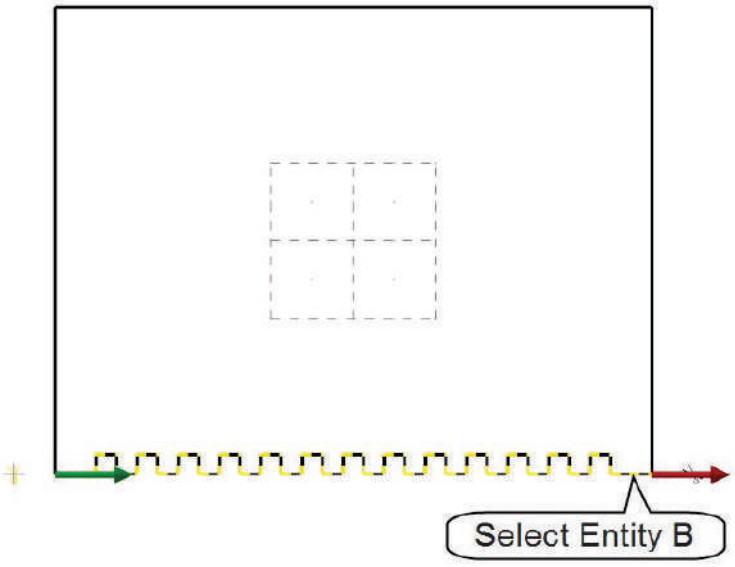
Figure: 10.0.1



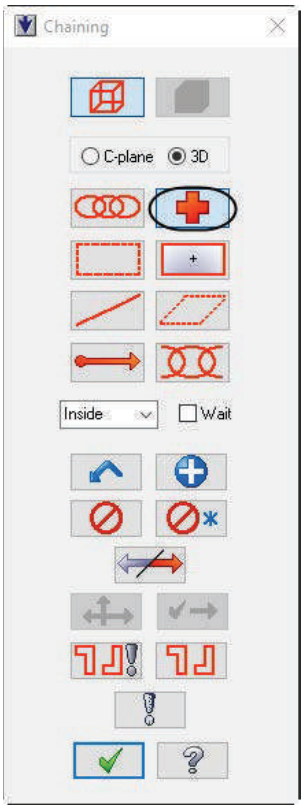
NOTE: The direction of the chain should be toward the right. Otherwise use the **Reverse** button.

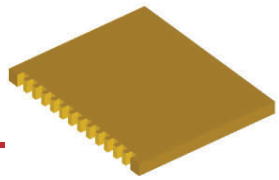
- [Select the last entity]: Select Entity B as shown in Figure: 10.0.2.

Figure: 10.0.2



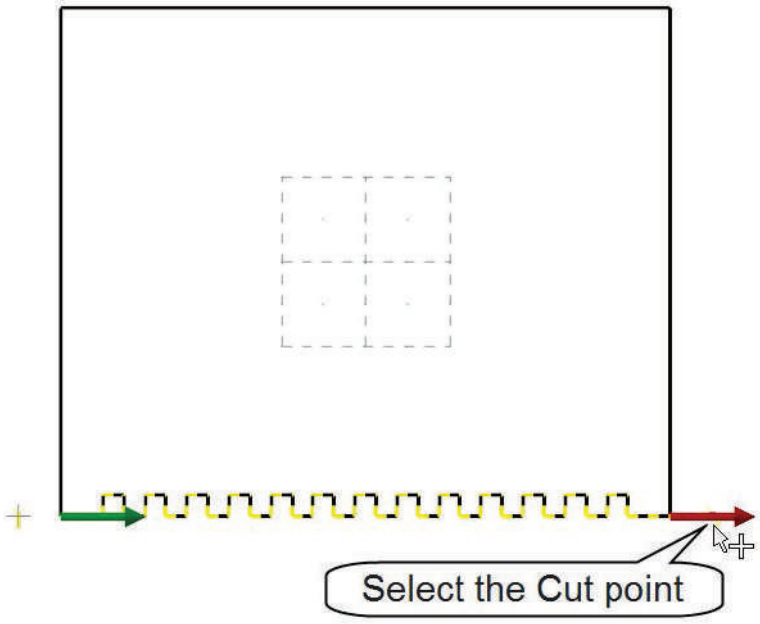
- From the **Chaining** dialog box, select the **Point** button.





- [Contour: select chain 3]: Select the **Cut point** as shown in Figure: 10.0.3.

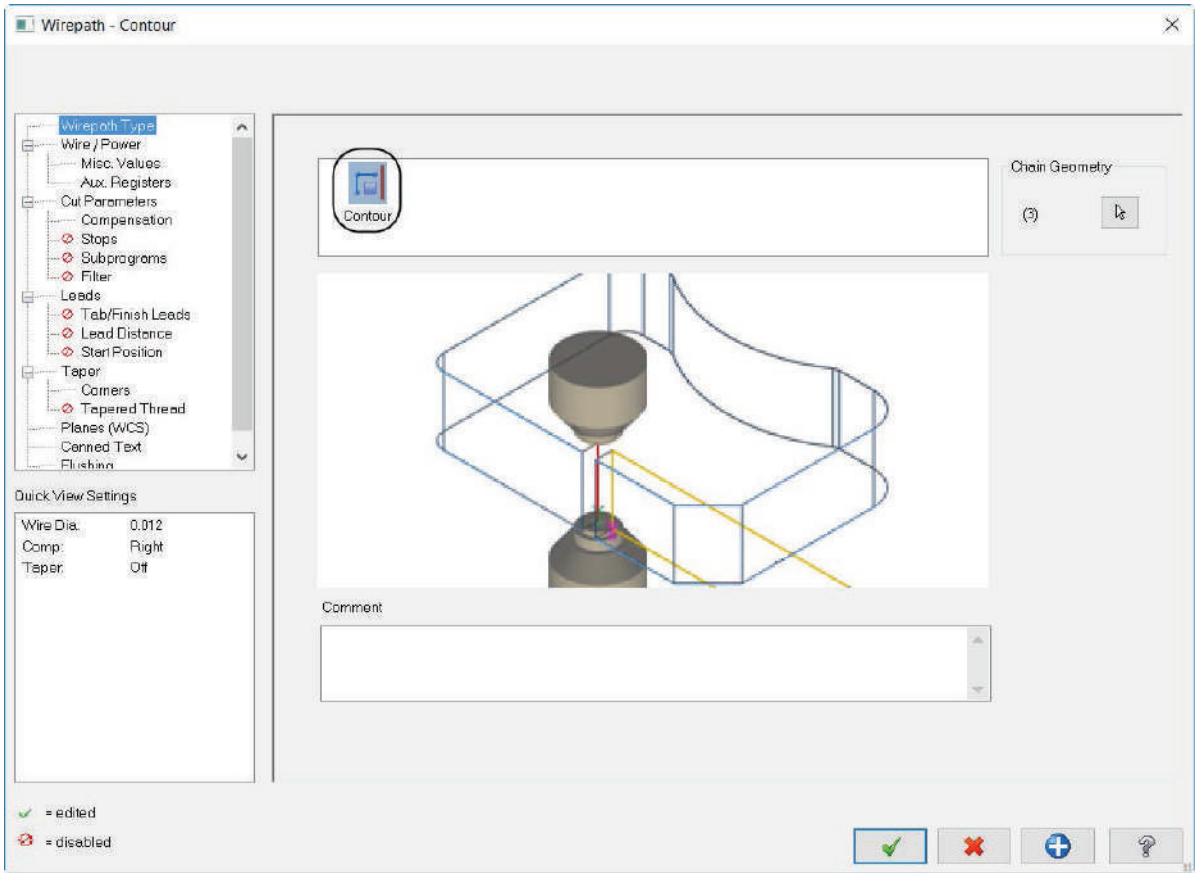
Figure: 10.0.3



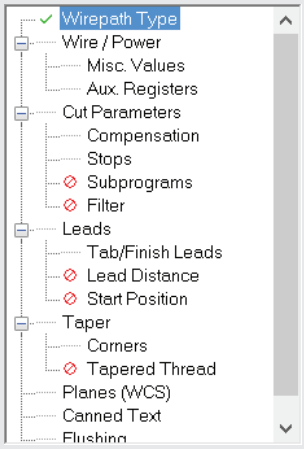
- Select the **OK** button to exit from the **Chaining** dialog box.

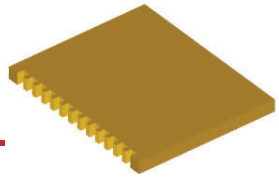
10.1 Wirepath Type

- 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.

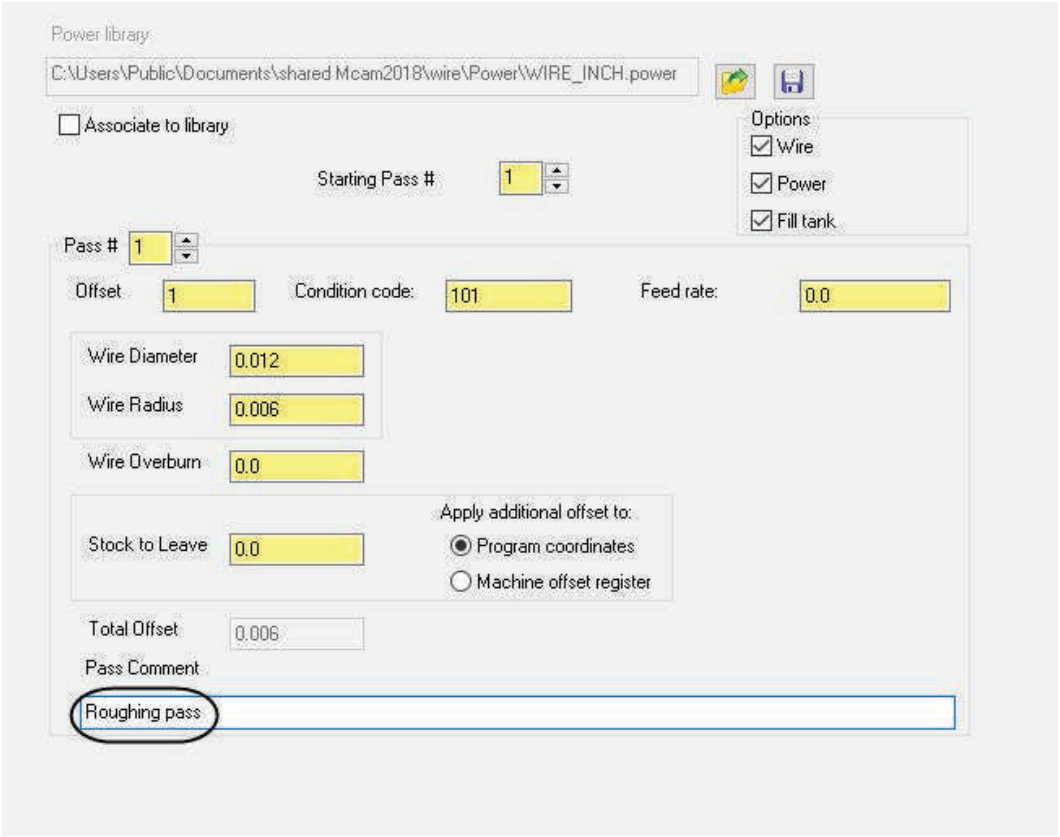




10.2 Set the Wire/Power parameters

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.



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 for condition 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.

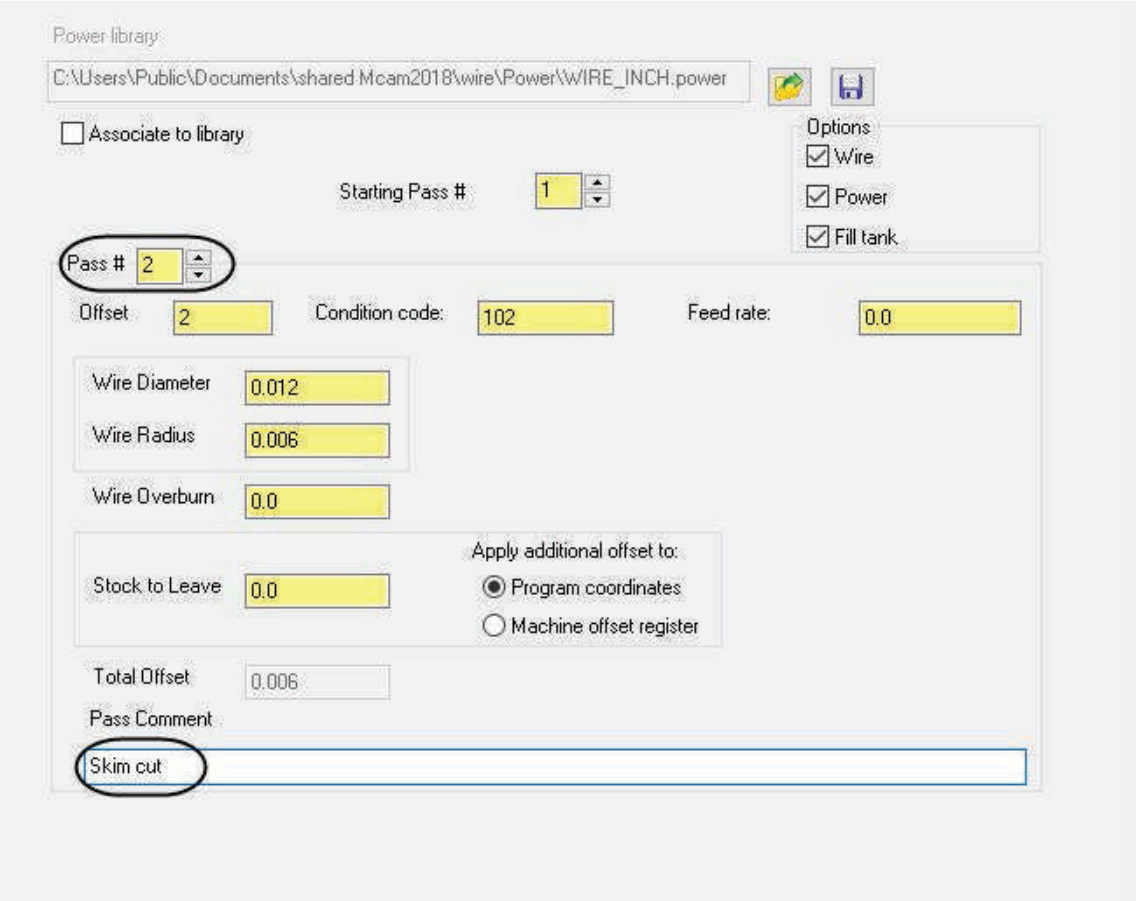
Wire diameter/radius sets the wire diameter/radius value and is automatically updated when one of the values is entered.

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.

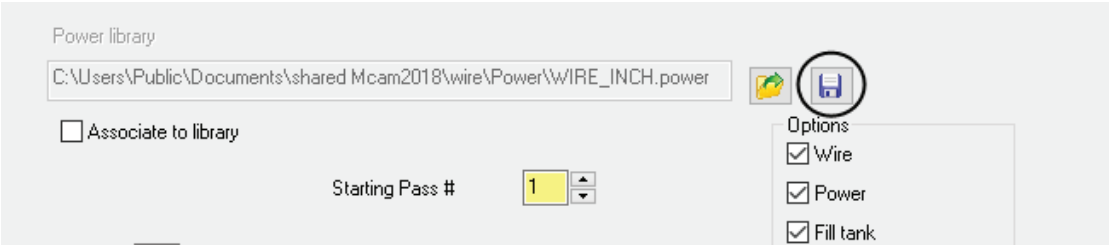
- Select the upper arrow and change the **Pass #** to **2.**
- Set the parameters for **Pass #2** (Skim cut) as shown in Figure: 10.2.1.

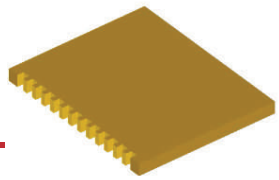
Figure: 10.2.1



NOTE: If you want to save this power library for other jobs, you can select the **Save** icon as shown in Figure: 10.2.2. 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.

Figure: 10.2.2

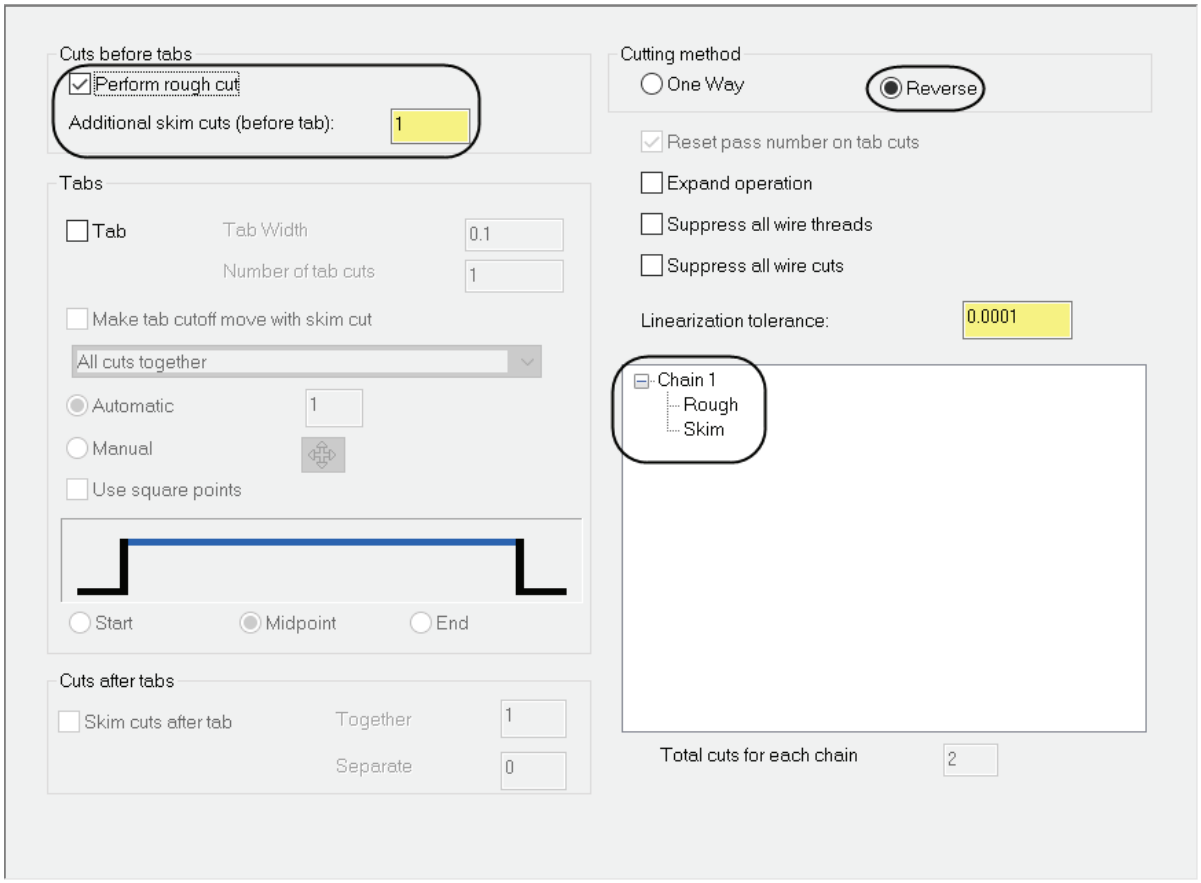




*10.3 Set the **Cut Parameters***

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.



Perform rough cut is used to create a rough cut for the contour. The rough cut uses the Start 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.

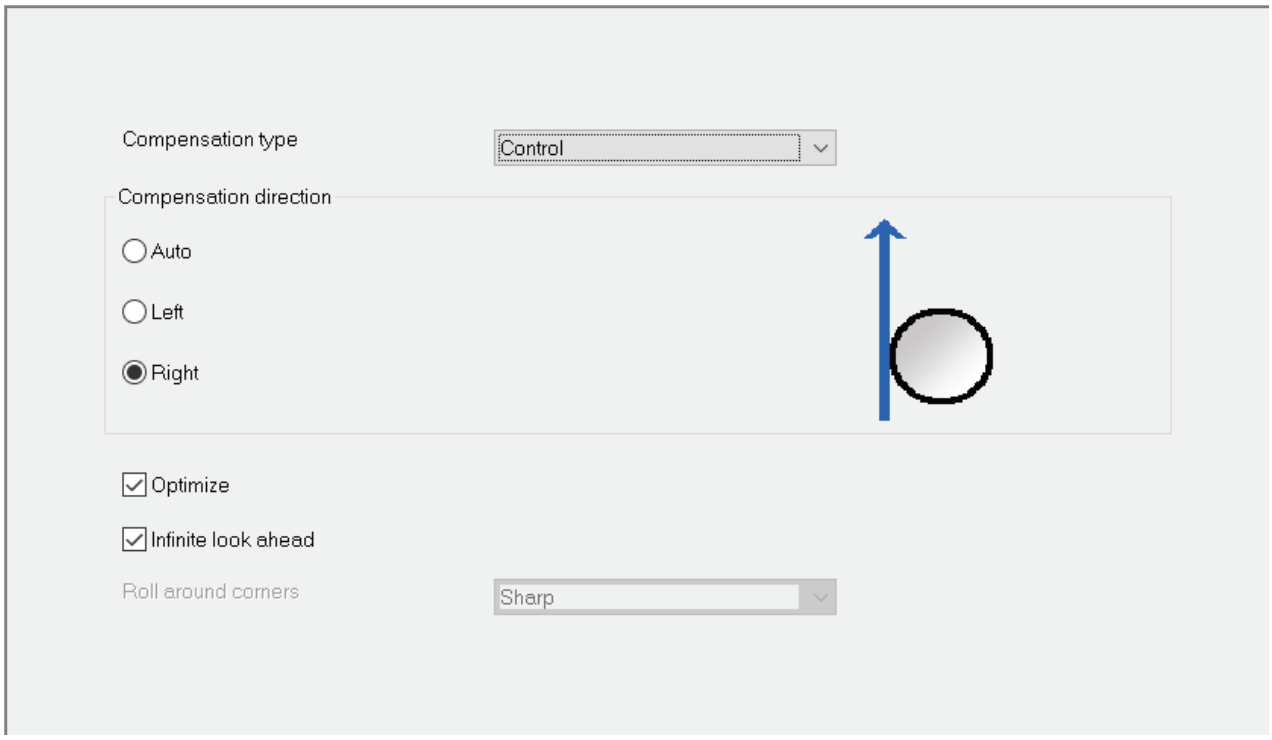
NOTE: For this part you do not need to add a tab on the contour as the part can be clamped from the opposite side, which is not machined in this operation.

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 in Figure: 10.4.1.

Figure: 10.4.1.



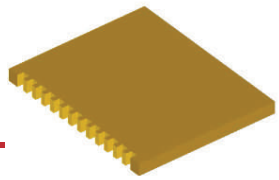
Compensation type sets to **Control** outputs control codes for compensation and does not compute the compensated wirepath.

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.

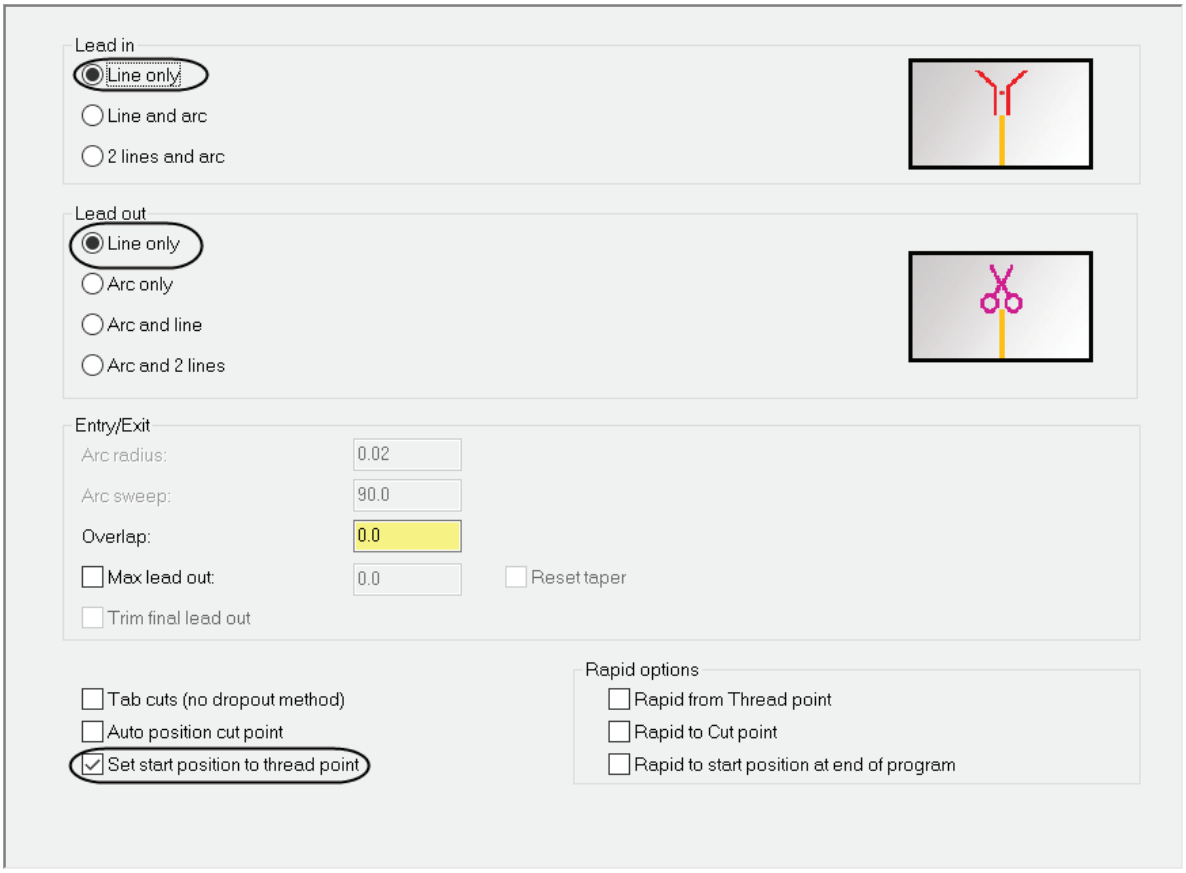
Roll around corners inserts arc moves around corners in the wirepath. The radius of the arc moves equals the radius of the wire. Not available with compensation type set to **Control**.



10.5 Leads

Leads page allows you to set the wire motion when entering or exiting a part.

- From the **Tree view** area, select **Leads**.
- Make sure that the **Lead in** and **Lead out** are set to **Line only** as shown.



Lead in sets the shape of the part entry motion. The lead in extends from the thread point to the start of contour.

Line only creates a line between the thread point and start of contour.

Line and arc adds a line and arc between the thread point and start of contour. Enter values for the **Arc radius** and **Arc sweep**.

Two lines and arc creates two lines and one arc between the thread point and start of contour. Enter values for the **Arc radius** and **Arc sweep**.

Lead out sets the shape of the part exit motion. Lead out extends from the end of contour to the cut point. You have the same options as in the Lead in plus the following extra options.

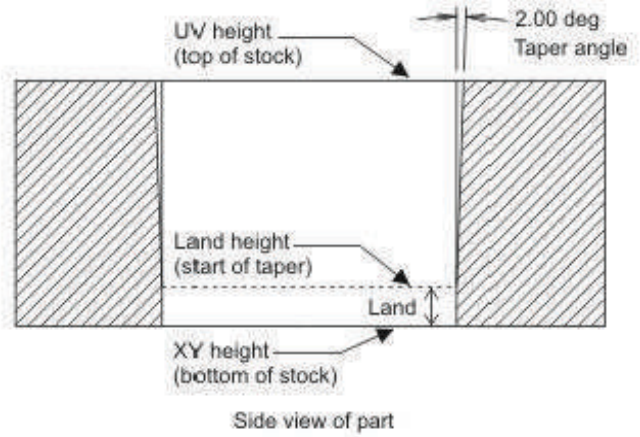
Arc only in which the wire arcs off the part. Enter values for the **Arc radius** and **Arc sweep**. A lead out radius uses the same **Arc radius** and **Arc sweep** values as the lead in.

Arc and two lines the wire arcs off the part, then creates two line moves to the cut point. Enter values for the **Arc radius** and **Arc sweep**. A lead out radius uses the same **Arc radius** and **Arc sweep** values as the lead in.

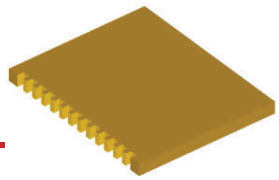
10.6 Taper

Taper page allows you to set the parameters to taper the contour wirepath inward or outward and to specify the location of the land (the point at which the taper begins). You can set the Z height of the upper wire guide for rapid moves, the location of the upper wire guide and lower wire guide, the height of the part, and the location of the XY plane as shown in Figure: 10.6.1.

Figure: 10.6.1

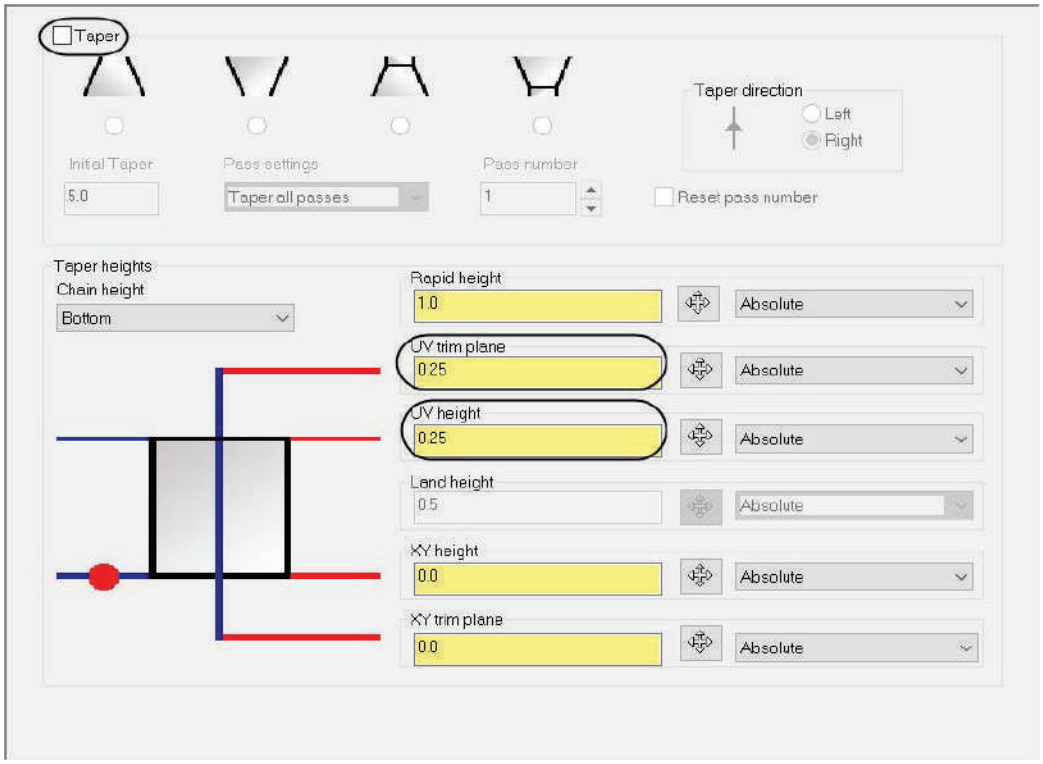


- From the **Tree view** area, select **Taper**.



- Make sure that the **Taper** is disabled and change the heights as shown in Figure: 10.6.2.

Figure: 10.6.2



Chain height options allows you to indicate where the geometry was chained (XY-**Bottom** or UV-**Top** height or, if there is a land, at the land height-**Middle**).

Rapid height sets the Z depth of a rapid move. Can be higher than the **UV height**, **UV trim**, or **UV extension planes** in order to clear any clamps.

UV trim plane can be used to set the location of the upper guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to Gcode 1015 in the NC file. It is usually set directly at the controller.

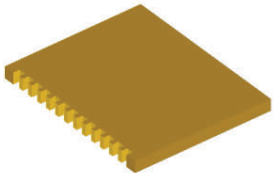
UV height sets the location of the XY plane that contains the geometry for the upper contour of a 4-Axis part.

Land height is available only when **Land up** or **Land down** is selected for **Taper** style, and sets the plane at which the wire pivots to the taper angle.

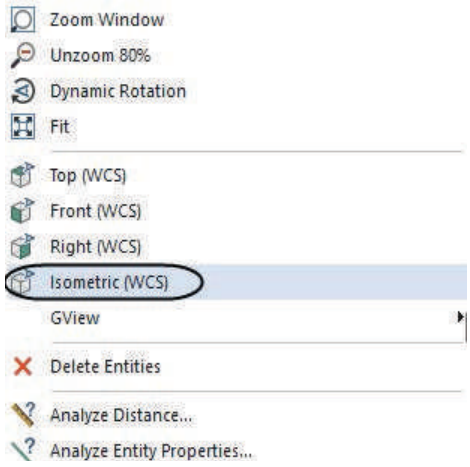
XY height sets the location of the XY plane that contains the geometry for the lower contour of a 4-Axis part.

XY trim plane can be used to set the location of the lower guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to Gcode 1015 in the NC file. It can be set directly at the controller. When not using the XY trim plane, enter the same value as the XY

- Select the **OK** button to exit the **Wirepath - Contour** parameters.
- Select the **OK** button to exit the **Chain Manager** dialog box.



- Right mouse click in the graphics window and select **Isometric** as shown.

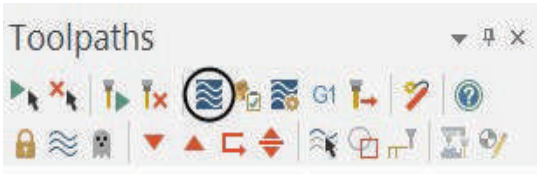


- Press **Alt + F1** to fit the drawing to the screen.

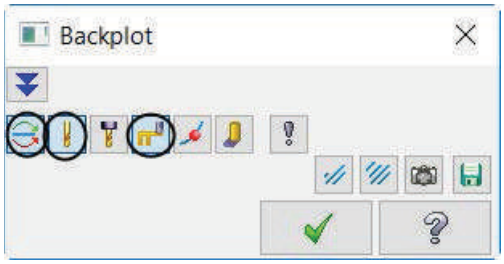
STEP 11: BACKPLOT THE WIREPATH

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 wirepaths, Mastercam displays the current X, Y, and Z coordinates in the lower left corner of the screen.

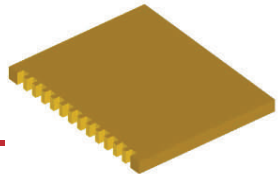
- Make sure that the toolpath is selected (signified by the check mark on the folder icon).
- Select the **Backplot selected operations** button.



- Make sure that you have the following buttons turned on to see the tool and the rapid moves.

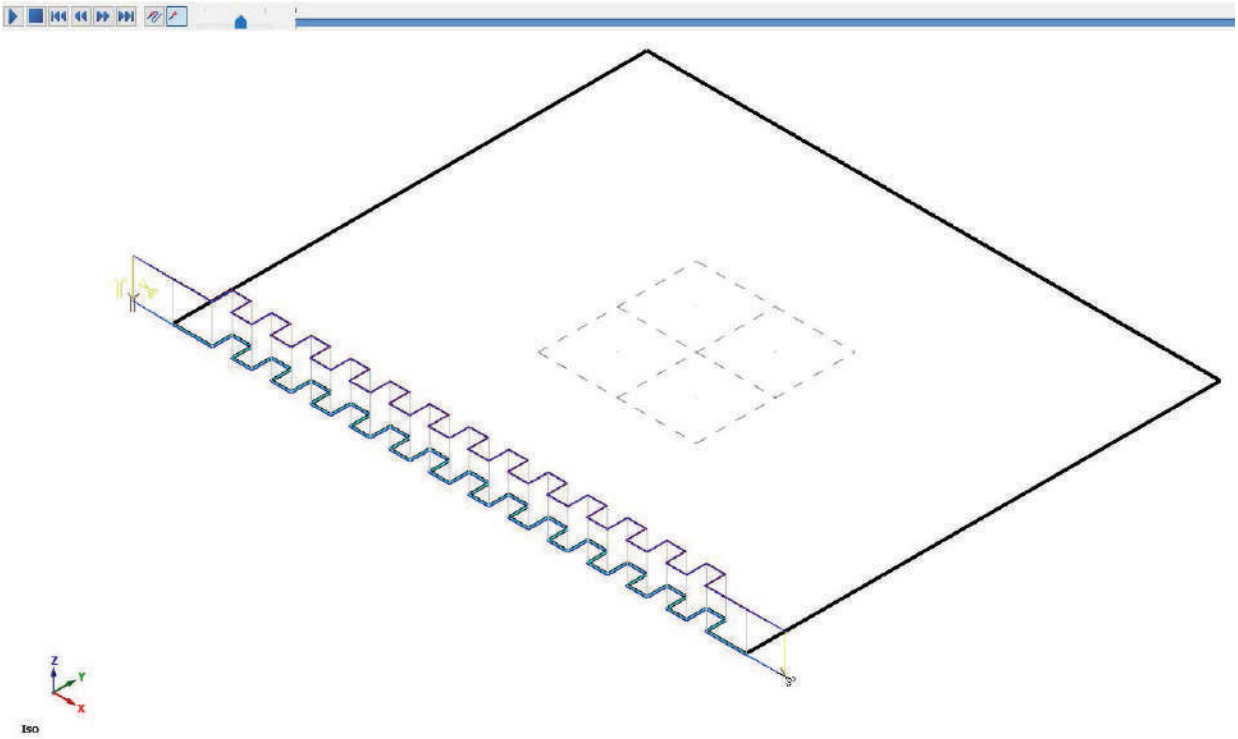


- Press **Alt+F1** to fit the drawing to the screen.
- You can adjust the speed of the backplot.
- You can step through the **Backplot** by using the **Step forward** or **Step back** buttons.
- Select the **Play** button to run **Backplot**.

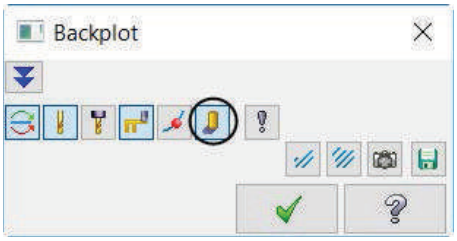


- The toolpath should look as shown Figure: 11.0.1.

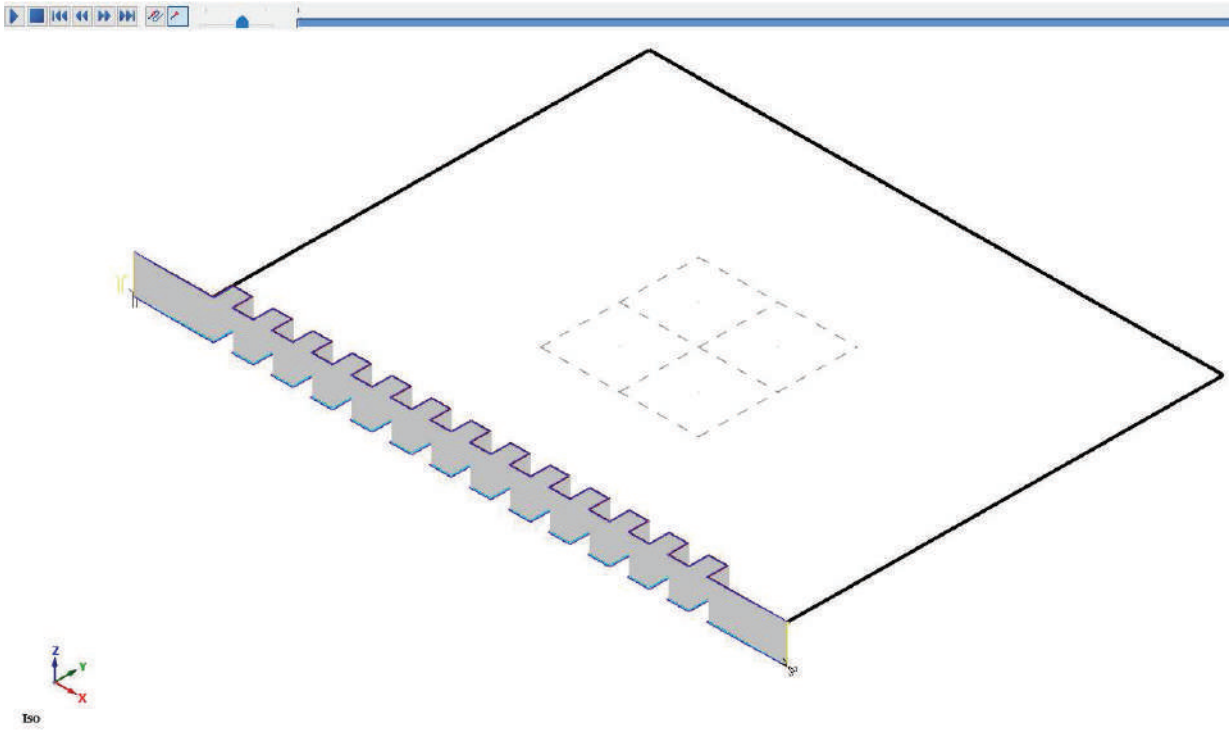
Figure: 11.0.1



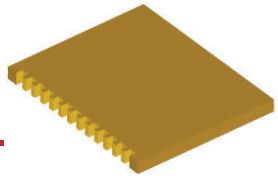
- In the **Backplot** dialog box enable **Quick verify** as shown.



- The toolpath should look as shown.



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

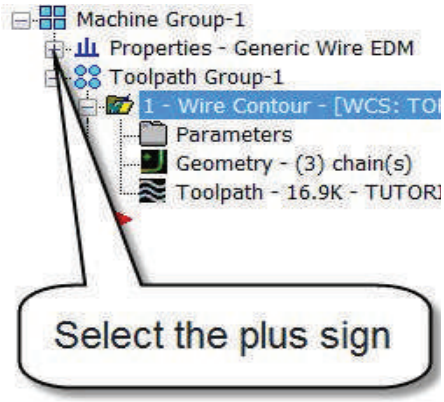


STEP 12: VERIFY THE WIREPATH

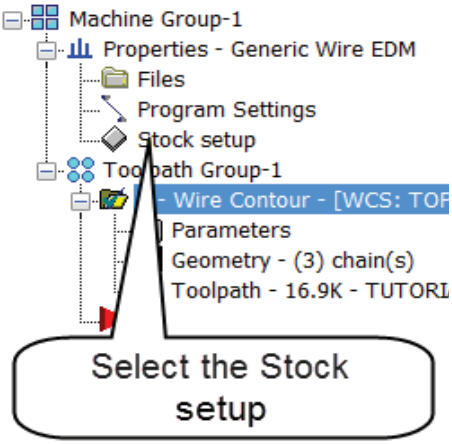
Verify simulates the machining of a part from a solid stock model display. The stock dimensions are based on the values that you can specified in the **Stock Setup**.

12.1 Set up the stock

- From the **Toolpaths Manager**, select the plus sign (+) in front of the **Properties** to expand the **Toolpaths Group Properties**.

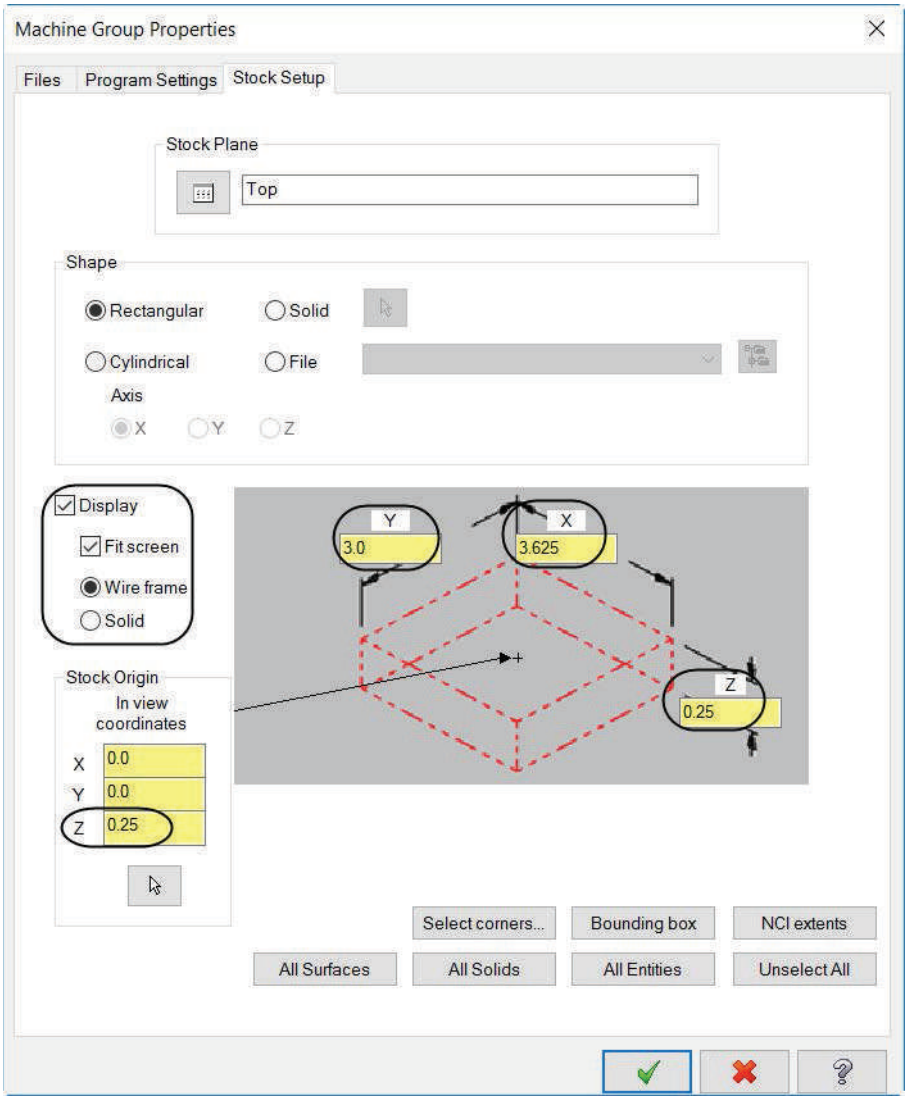


- Select the **Stock setup** to set the stock size.

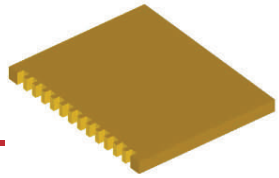


- Change the parameters to match Figure: 12.1.1.

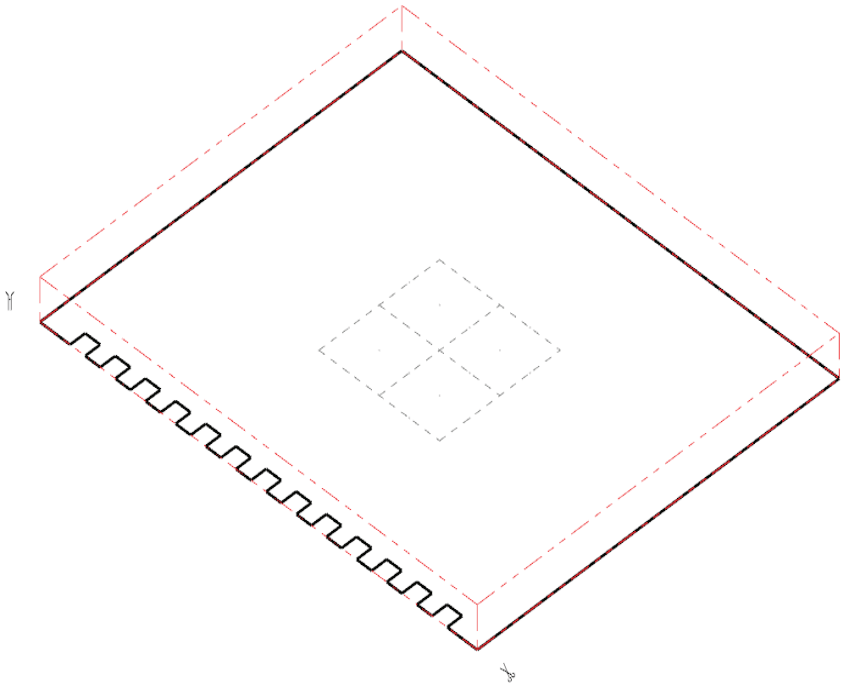
Figure: 12.1.1



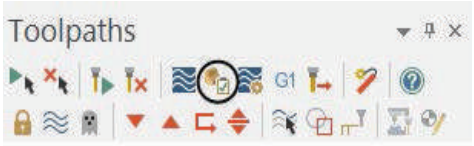
- Select the **OK** button to exit **Machine Group Properties**.



- The stock is displayed with phantom red lines and it should look as shown.

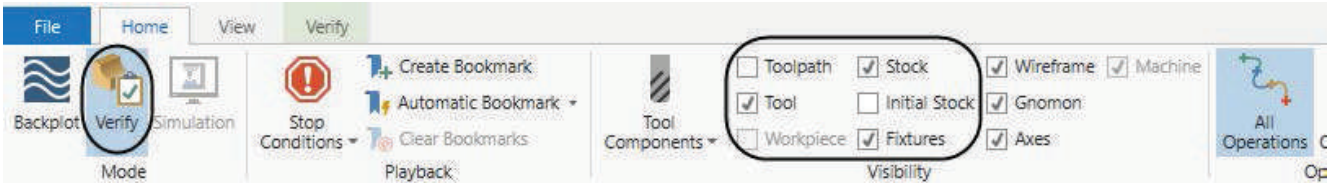


- Select the **Verify selected operations** button.

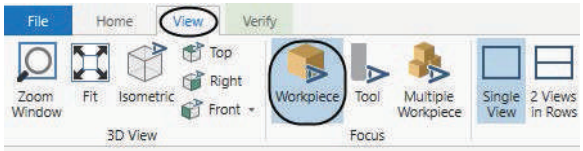


- From **Verify** change the settings for the **Visibility** as shown in Figure: 12.1.2.

Figure: 12.1.2



- Select **View** tab and enable **Workpiece** as shown.

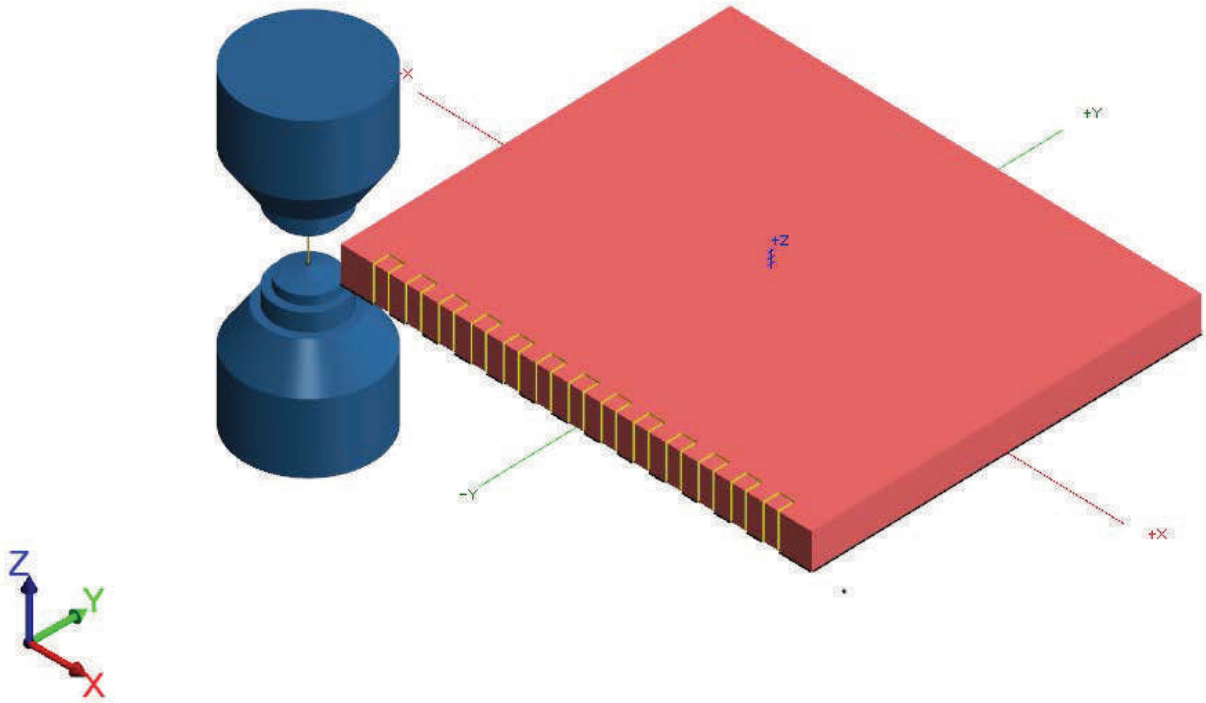


- Select **Verify** tab and enable **Color Loop** as shown.

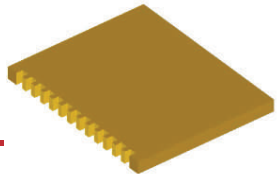
- Right click in the graphic area and select the **Isometric** option.
- Right click in the graphic area again and select the **Fit** option.
- Select the **Play** button to run **Verify**.

- The part should look as shown in Figure: 12.1.3.

Figure: 12.1.3



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.

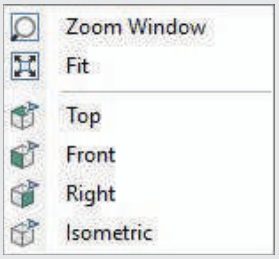


- Under the **View** tab, click on the **Isometric** icon

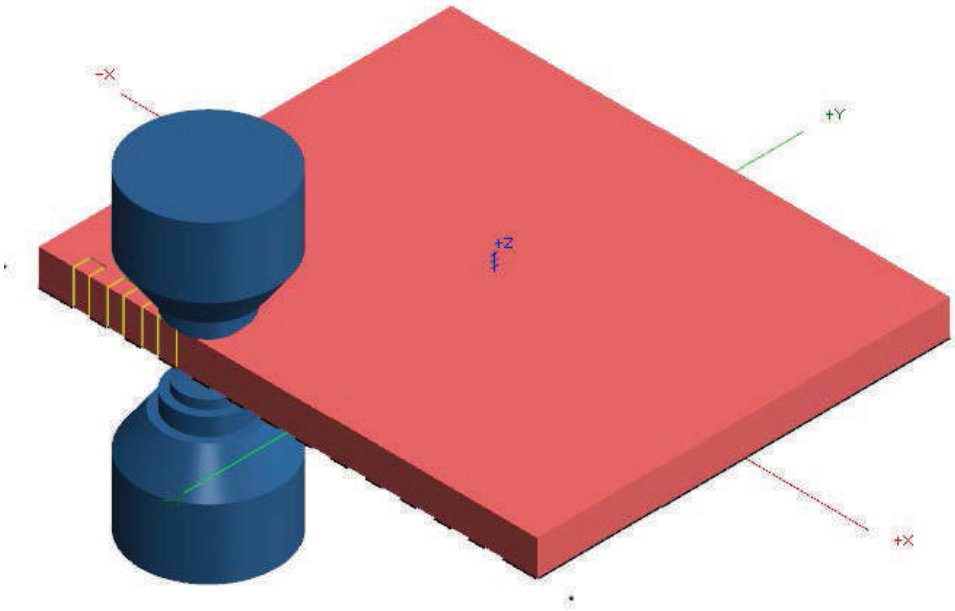
and then the **Fit** icon

to see the part in the original position if needed.

NOTE: You can also change the view or fit the part to the graphics window if you right mouse click in the graphics window.



- To check the part step by step, click first on the **Start (Home)** icon.
- Click on the **Step Forward** icon to see the tool moving one step at a time.
- The part should look as shown after several steps.

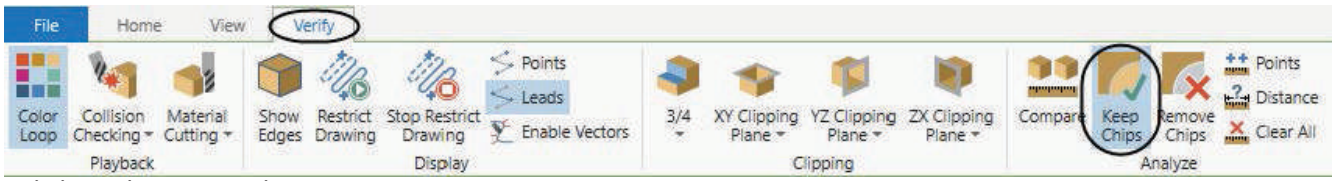


- Click on the **Step Forward** until the toolpath is completed.

12.2 Chip removal

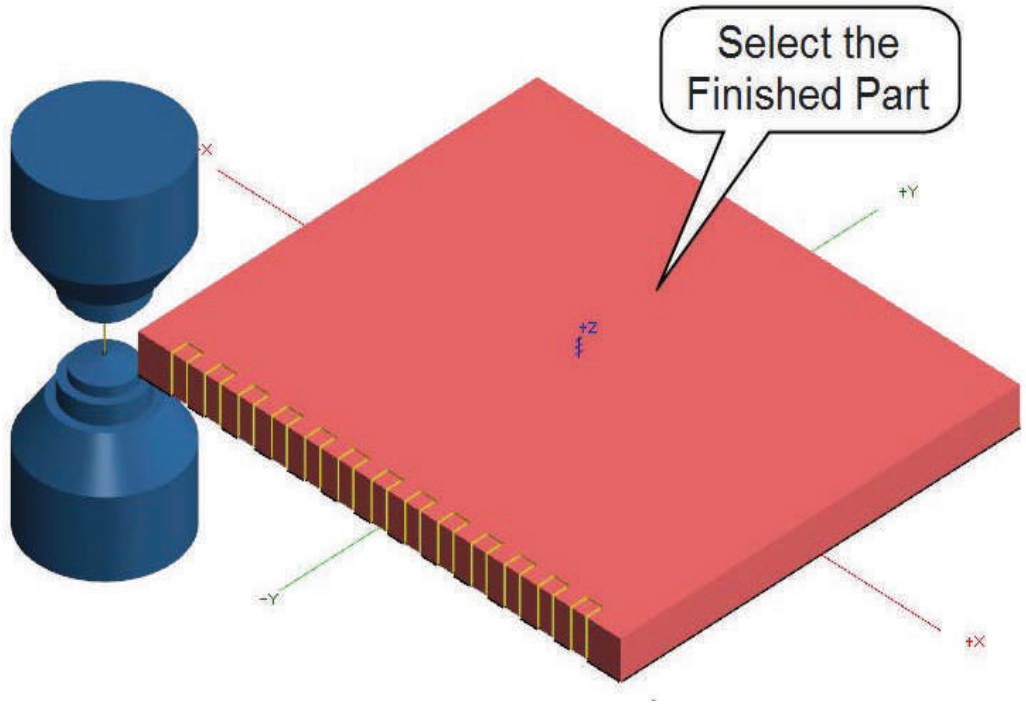
- Select the **Verify Tab** and select the **Keep Chips** icon as shown in Figure: 12.2.1.

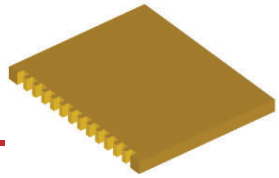
Figure: 12.2.1



- Click on the part as shown in Figure: 12.2.2.

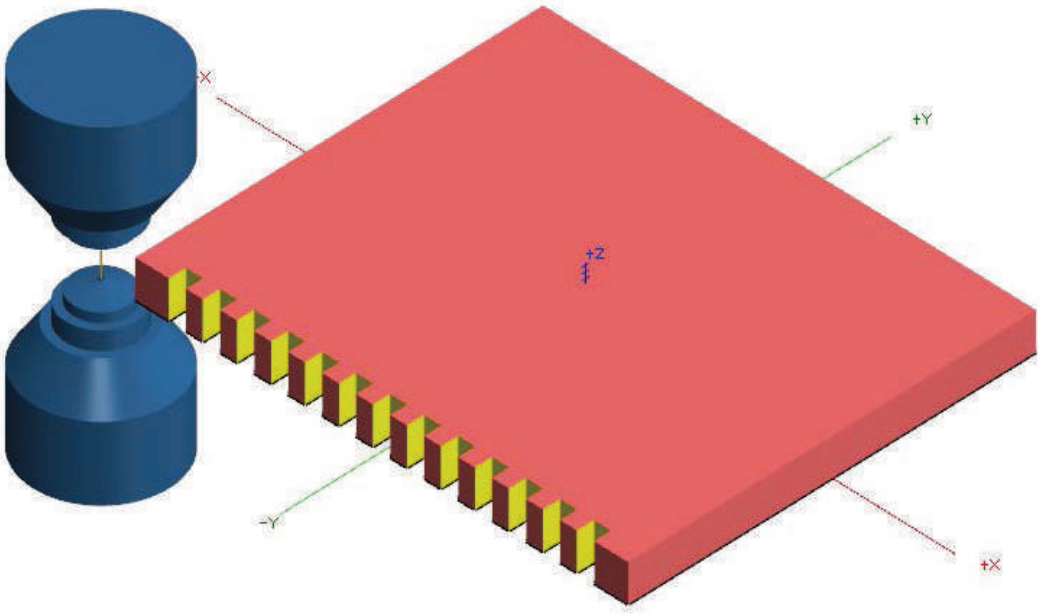
Figure: 12.2.2



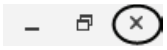


- The part will appear as shown in Figure: 12.2.3.

Figure: 12.2.3

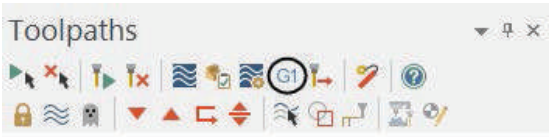


- To exit **Mastercam Simulator** click on the **Close** icon.



STEP 13: POST THE FILE TO GENERATE THE NC CODE

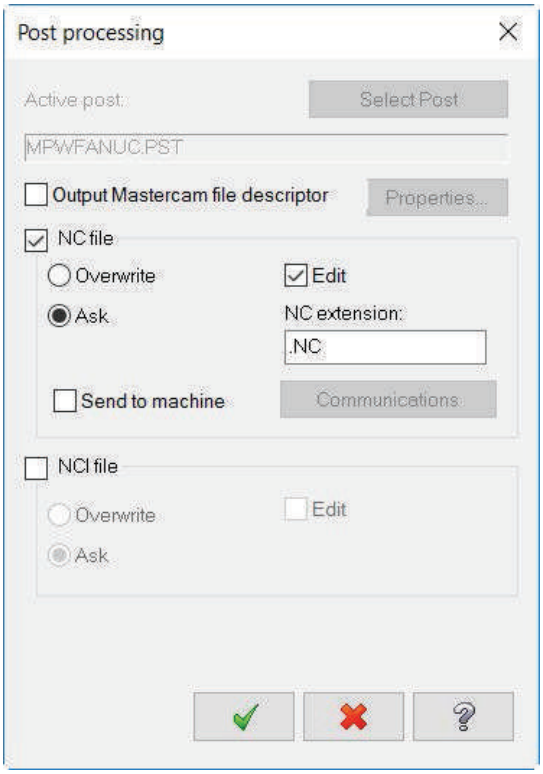
- Select **Post selected operations** button from **Toolpaths Manager**.



NOTE: The active **Post Processor** is a generic FANUC post processor.

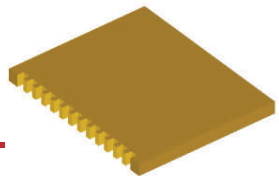
- In the **Post processing** window as shown in Figure: 13.0.1, if needed, make any necessary changes.

Figure: 13.0.1



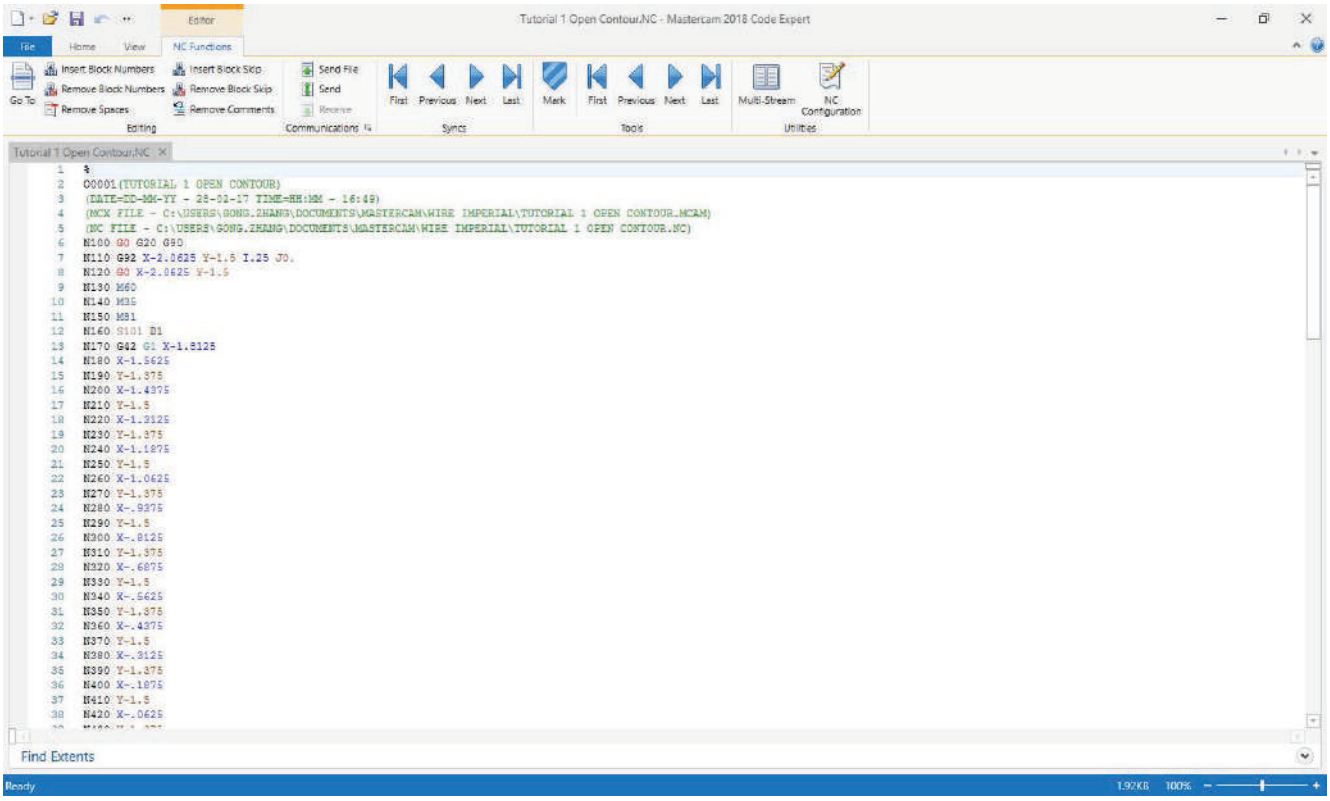
- Select the **OK** button to continue.
- Enter **Tutorial1_Open Contour** in the **File name** field.
- Save the NC file.

NOTE: The HLE/Demo version of Mastercam does not support post processing. G1 button does not work and no G-code can be created in the HLE/Demo version.



- The NC file will appear on the screen as shown in Figure: 13.0.2.

Figure: 13.0.2

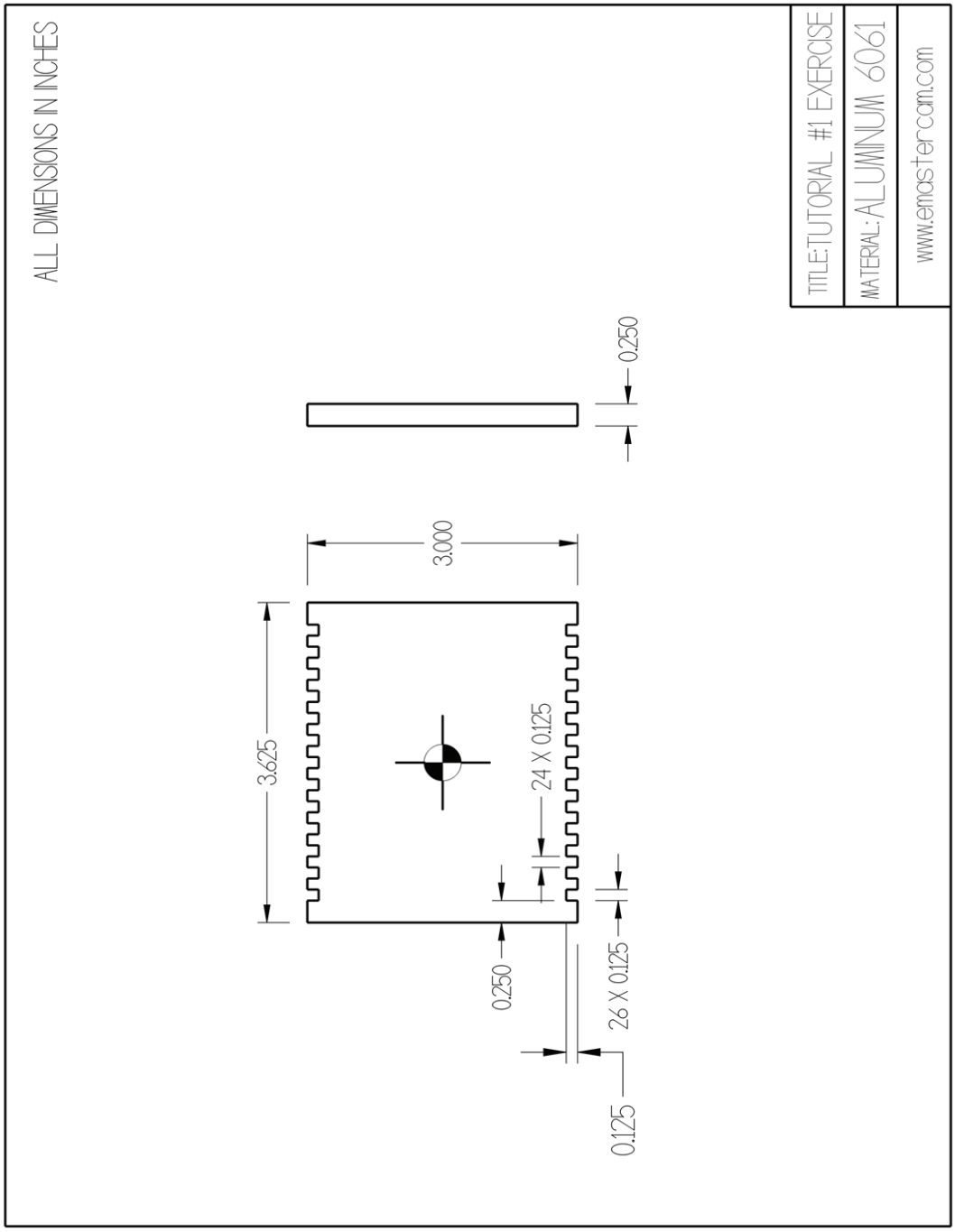


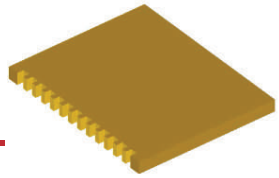
NOTE: The G-code that you have created will appear on the screen. If the code looks okay, you can shut down the window without saving it. If you need to change the code, make sure you save it before you close the window.
How the program is sent to the machine depends on the shop setup.

- Select the "**X**" box at the upper right corner to exit the **Mastercam Code Expert.**

STEP 14: SAVE THE UPDATED MCAM FILE

REVIEW EXERCISE -STUDENT PRACTICE

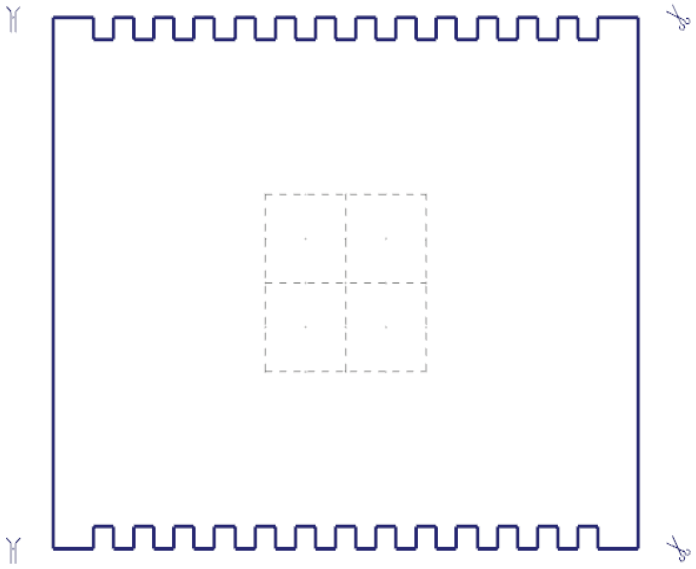


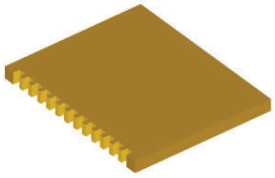


CREATE THE GEOMETRY FOR TUTORIAL #1 EXERCISE

Use these commands to create the geometry.

- Rectangle.
- Line Parallel.
- Trim Entities.
- Translate.
- Point Position - Thread Point.
- Point Position - Cut Points.



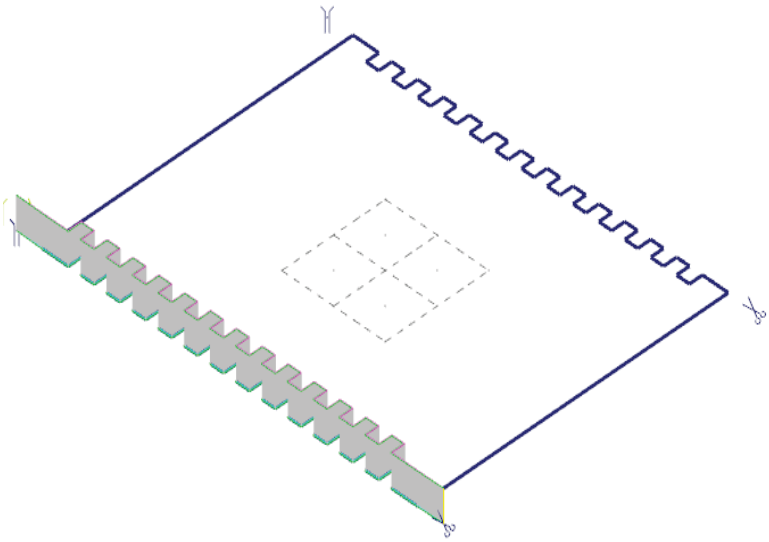


CREATE THE TOOLPATHS FOR TUTORIAL #1 EXERCISE

Create the Toolpaths for Tutorial #1 Exercise as per the instructions below.

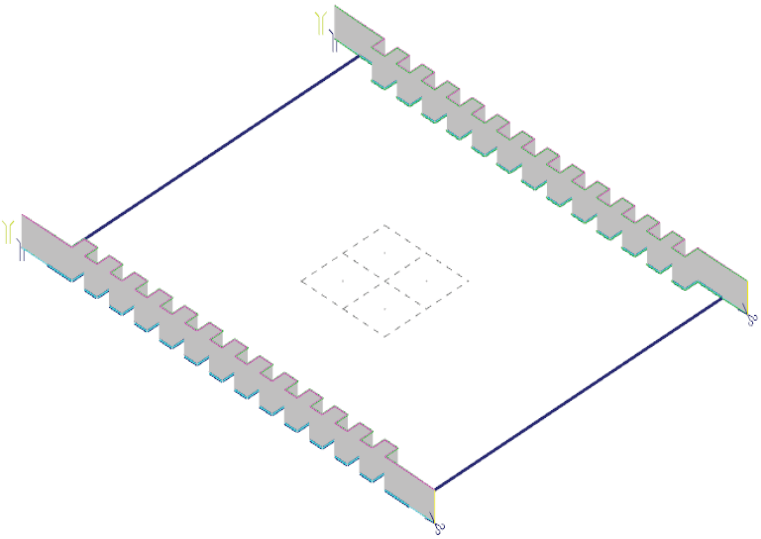
Machine The First Contour using Contour Toolpath.

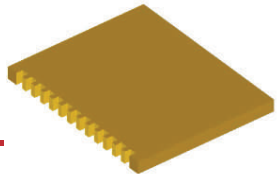
- In order to complete this operation follow **page 39** to **page 52.**



Machine The Second Contour using Contour Toolpath.

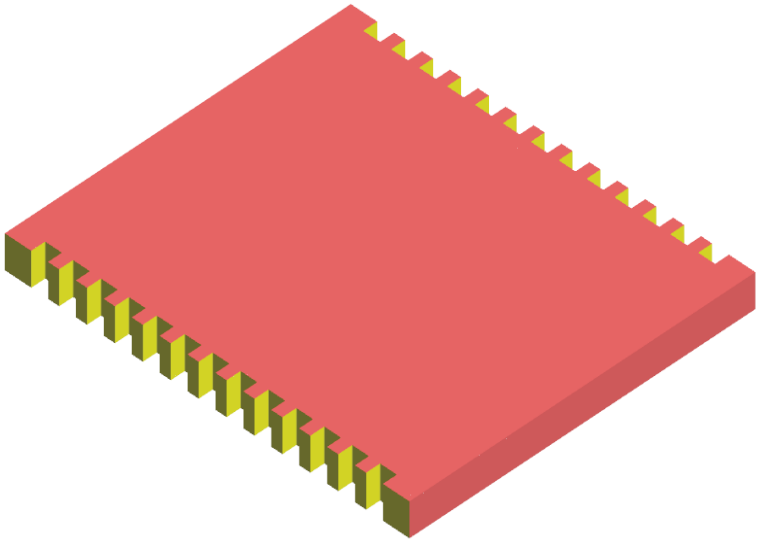
- In order to complete this operation follow **page 39** to **page 52.**

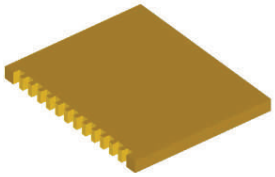




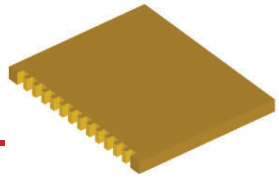
Verify The Wirepath.

- Set the **Stock** according to the part.
- In order to complete this operation Verify the part and follow **page 56** to **page 62**.





NOTES:



TUTORIAL #1 QUIZ

- What command allows you to copy a shape several times along the X-Axis?

- How can you remove the temporary group/result colors?

- What is the Tread Point?

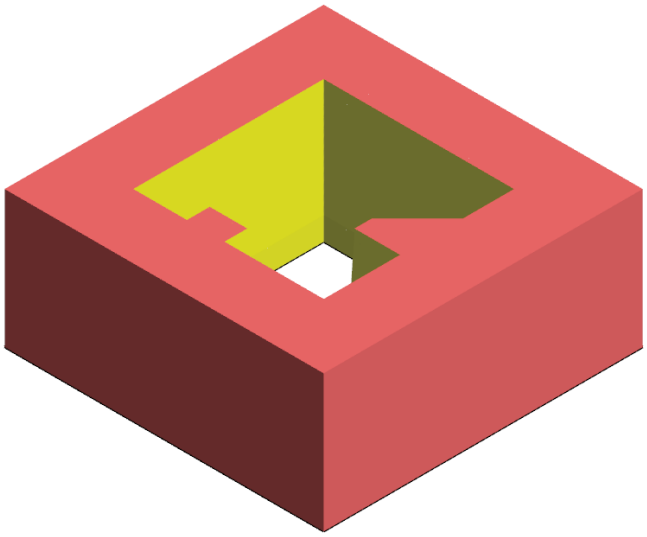
- What is the Cut Point?

- What does Backplot do?

☑ **Automatically sign me into this document in the future.**
(Do not select this when using a public computer)

Document ID: 0000-156C-2045D-00021DC2

TUTORIAL #2



OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

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, Line Polar, Trim, and Create Thread Point will be used.

Create the necessary Wirepaths to machine the part:

- A Contour wirepath to cut a punch with taper angle and land.
- Edit the wirepath using Change at point option.

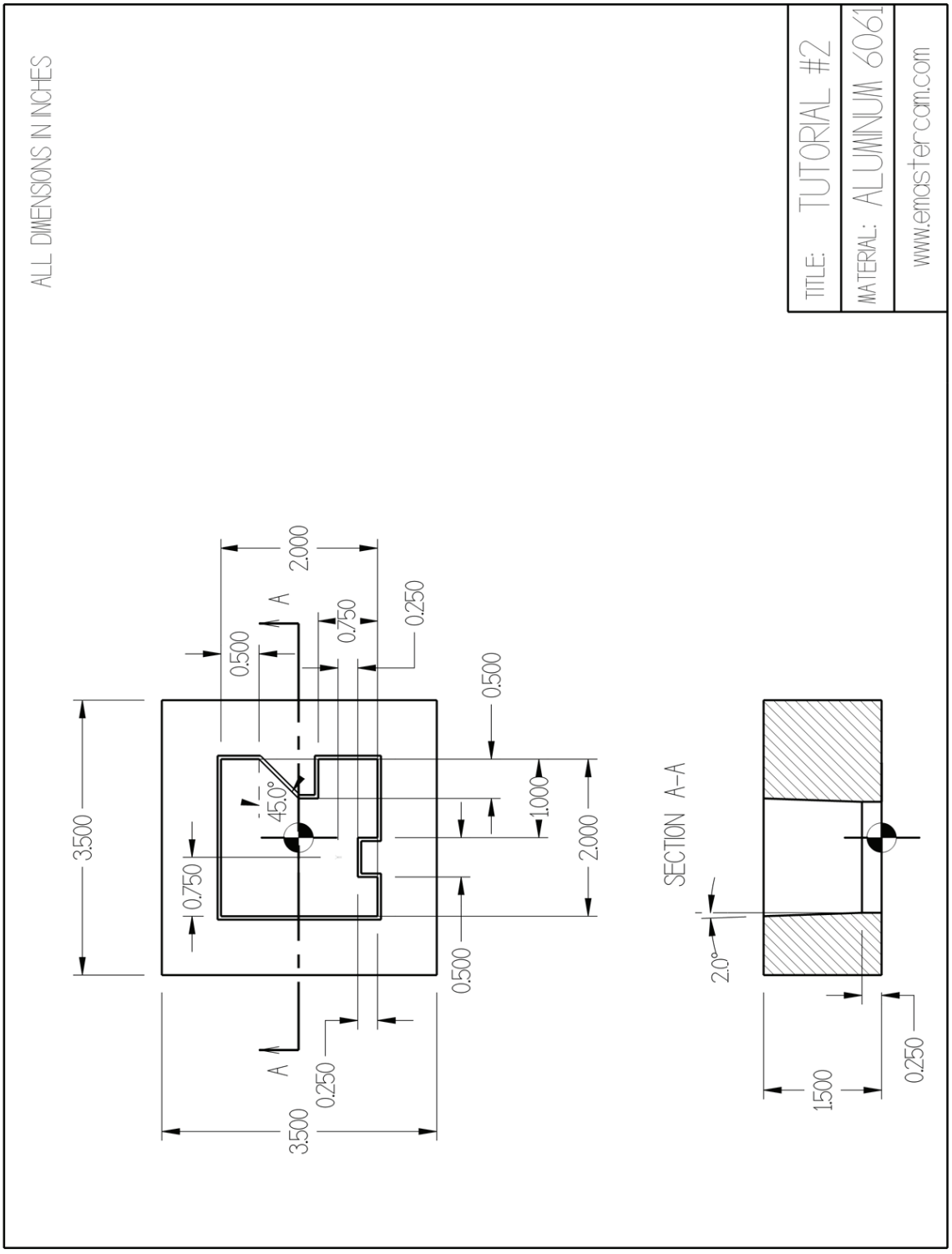
Backplot and the file:

- 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 then post process the file to obtain an NC file containing the necessary code for the machine.

This tutorial takes approximately 45 minutes to complete.


ALL DIMENSIONS IN INCHES
TITLE: TUTORIAL #2
MATERIAL: ALUMINUM 6061
www.emastercam.com
2.000
0.500
0.750
0.250
0.500
3.500
45.0°
1.000
2.000
0.750
0.500
0.250
3.500
SECTION A-A
2.0°
1.500
0.250

GEOMETRY CREATION

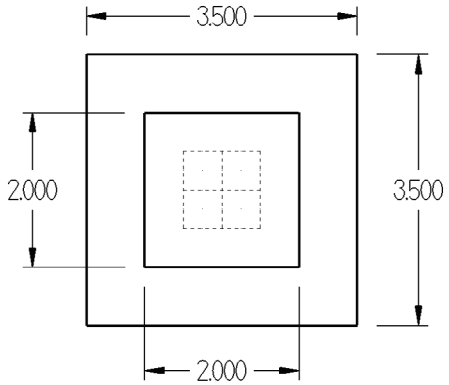
STEP 1: SETTING UP THE GRAPHIC USER INTERFACE

Please refer to the **Getting Started** section to set up the graphical user interface.

STEP 2: CREATE THE TWO RECTANGLES

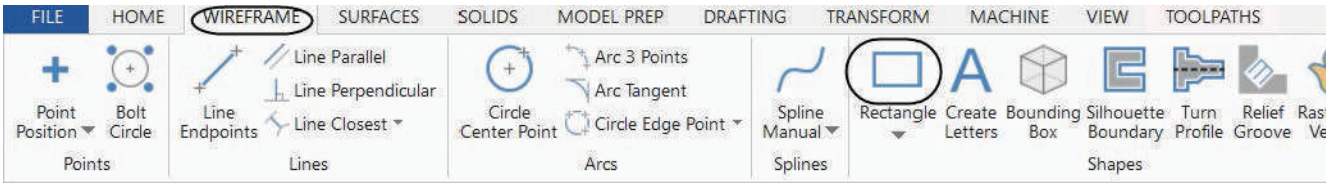
In this step you will learn how to create two rectangles given the width, the height, and the anchor location.

Step Preview:

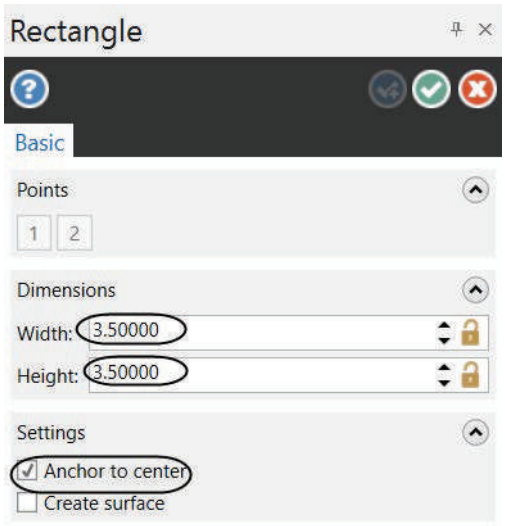


WIREFRAME

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

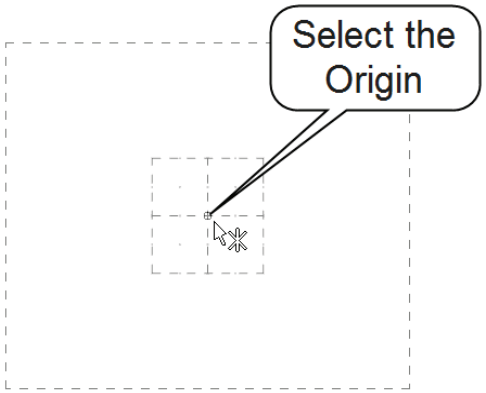


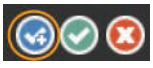
- Enter the **Width** and **Height** and enable **Anchor to center** as shown.

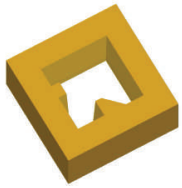


- [Select position of base point]: Select the Origin as shown in Figure: 2.0.1.

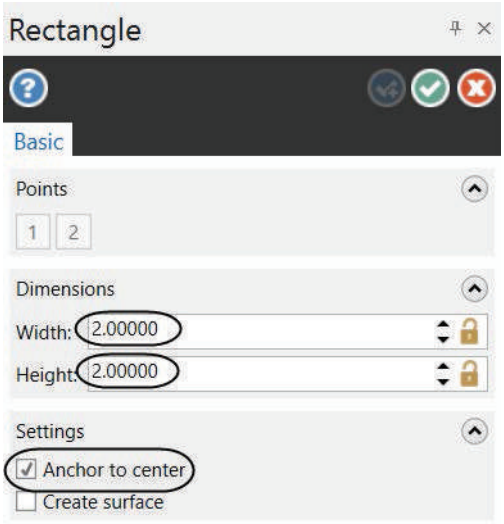
Figure: 2.0.1



- Select **OK and Create New Operation** to stay within the **Rectangle** command. 



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



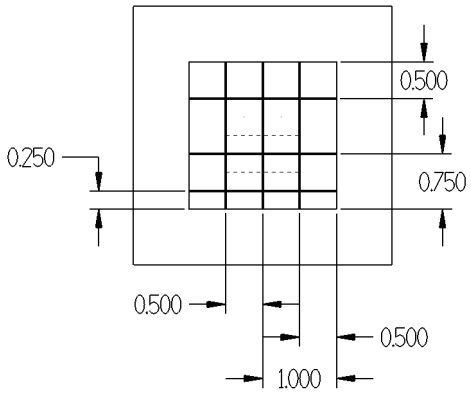
- [Select position of base point]: Select the Origin as shown in Figure: 2.0.1.
- Select the **OK** button to exit the **Rectangle** command.

NOTE: While creating the 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.

STEP 3: CREATE PARALLEL LINES

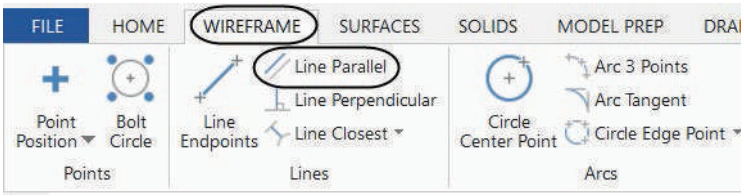
In this step you will create parallel lines used to define the die shape.

Step Preview:



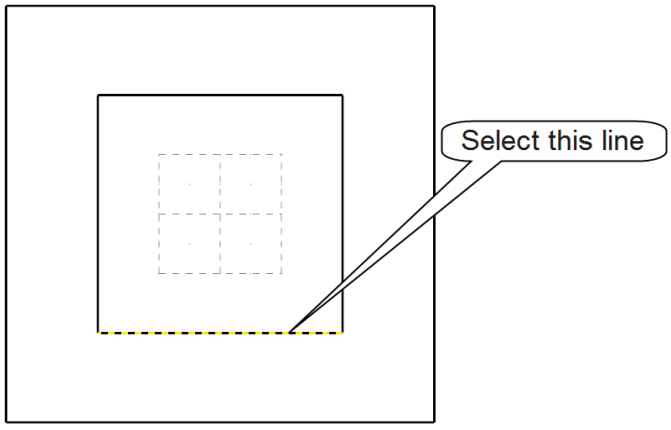
WIREFRAME

- From the **Lines** group, select the **Line Parallel** icon**.**



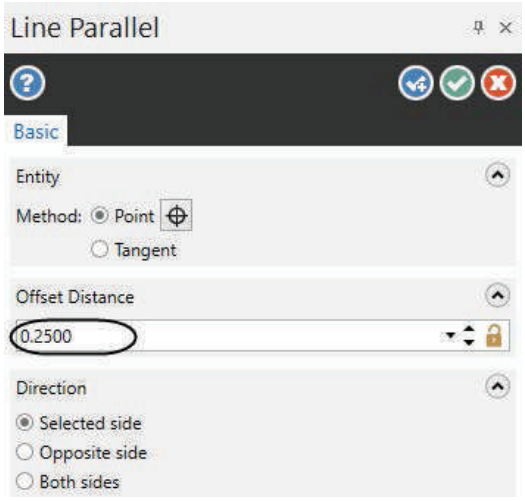
- [Select a line]: Select the lower horizontal line of the inside rectangle as shown in Figure: 3.0.1.

Figure: 3.0.1



- [Select the point to place a parallel line through]: Click somewhere above the selected line.

- In the **Line Parallel** panel, enter the **Offset Distance 0.25**. Press **Enter**.



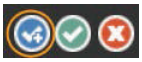
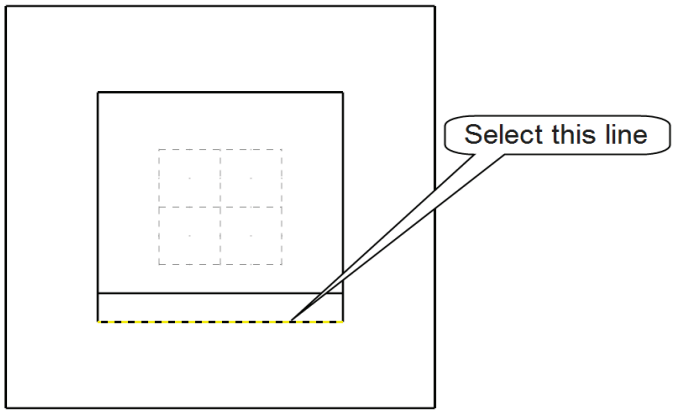
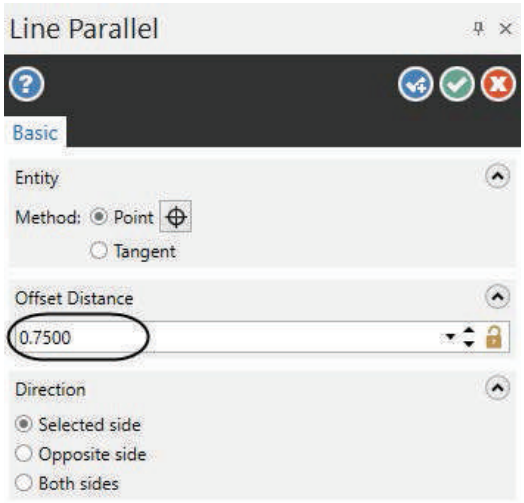
- Select **OK and Create New Operation** to continue with the same command. 
- [Select a line]: Select the same horizontal line as shown in Figure: 3.0.2.

Figure: 3.0.2



- [Select the point to place a parallel line through]: Click somewhere above the selected line.

- In the **Line Parallel** panel, enter the **Offset Distance 0.75**. Press **Enter**.



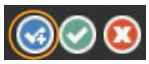
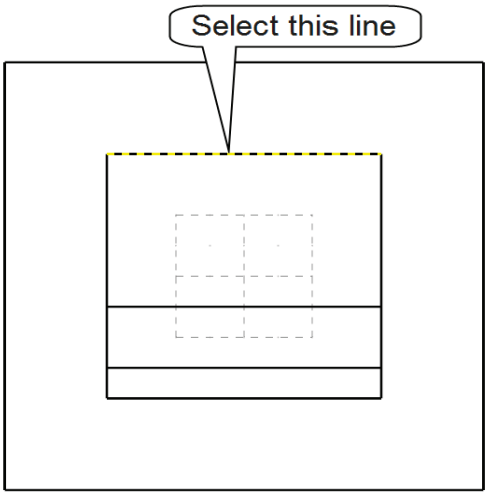
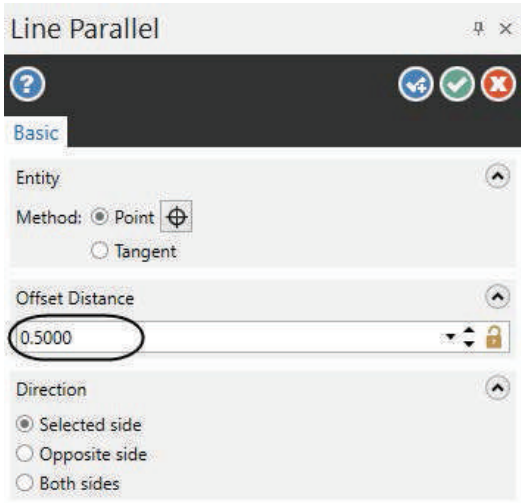
- Select **OK and Create New Operation** to continue with the same command. 
- [Select a line]: Select the upper horizontal line of the inside rectangle as shown in Figure: 3.0.3.

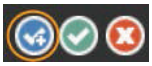
Figure: 3.0.3



- [Select the point to place a parallel line through]: Click somewhere below the selected line.

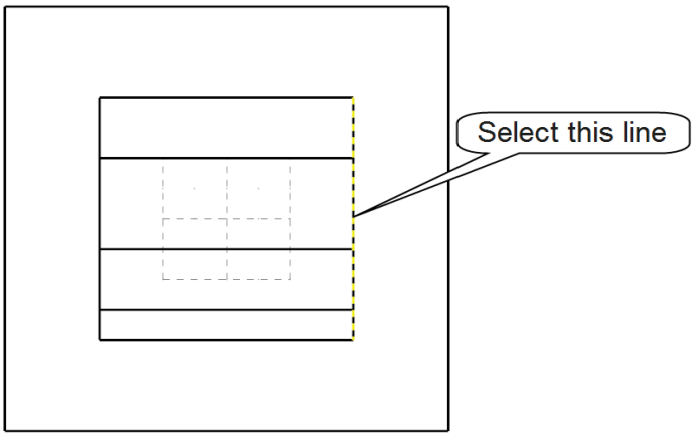
- In the **Line Parallel** panel, enter the **Offset Distance 0.5** (press **Enter**).



- Select **OK and Create New Operation** to continue with the same command. 

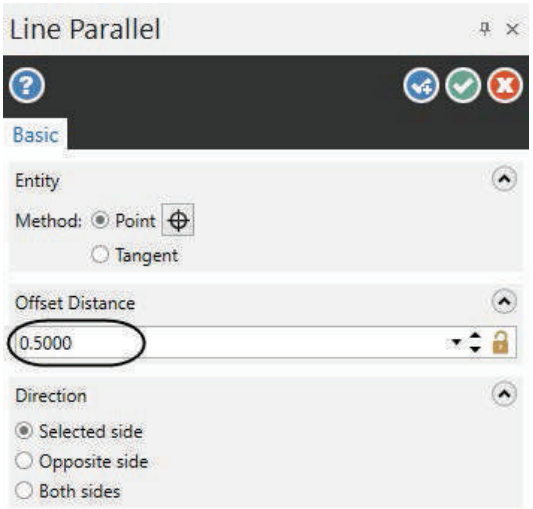
- [Select a line]: Select the right side vertical line of the inside rectangle as shown in Figure: 3.0.4.

Figure: 3.0.4



- [Select the point to place a parallel line through]: Click somewhere to the left of the selected line.

- In the **Line Parallel** panel, enter the **Offset Distance 0.5**. Press **Enter**.



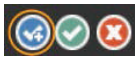
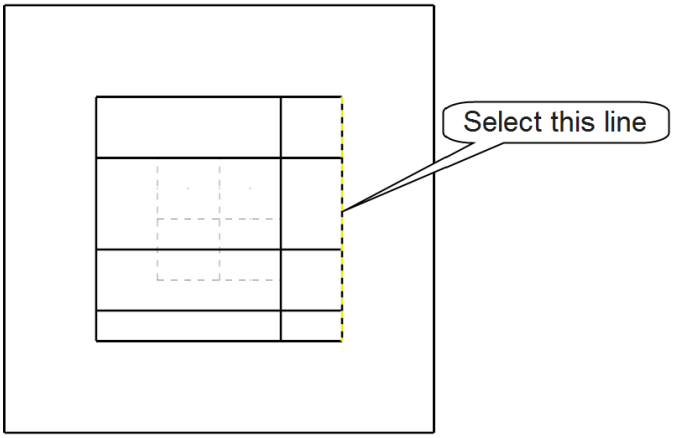
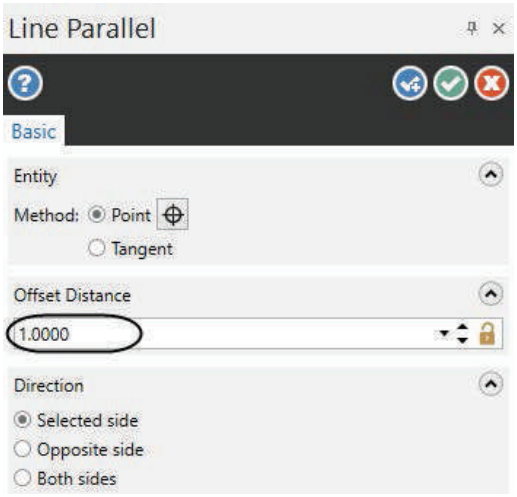
- Select **OK and Create New Operation** to continue with the same command. 
- [Select a line]: Select the right side vertical line of the inside rectangle as shown in Figure: 3.0.5.

Figure: 3.0.5



- [Select the point to place a parallel line through]: Click somewhere to the left of the selected line.

- In the **Line Parallel** panel, enter the **Offset Distance 1.0**. Press **Enter**.



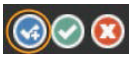
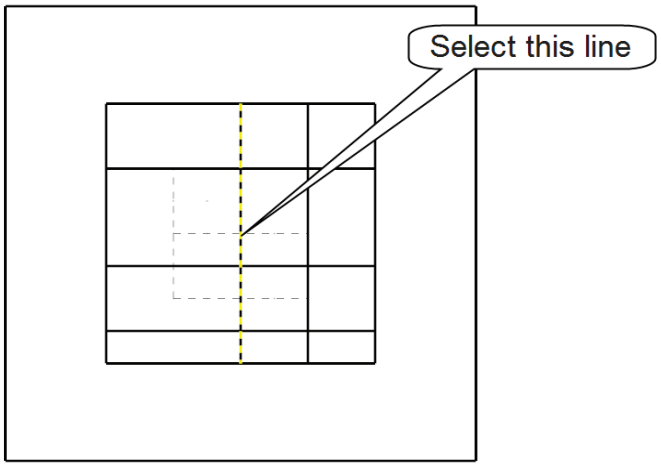
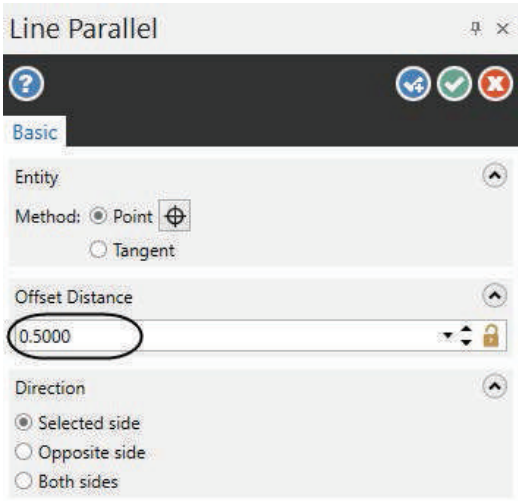
- Select **OK and Create New Operation** to continue with the same command. 
- [Select a line]: Select the last line that you created as shown in Figure: 3.0.6.

Figure: 3.0.6



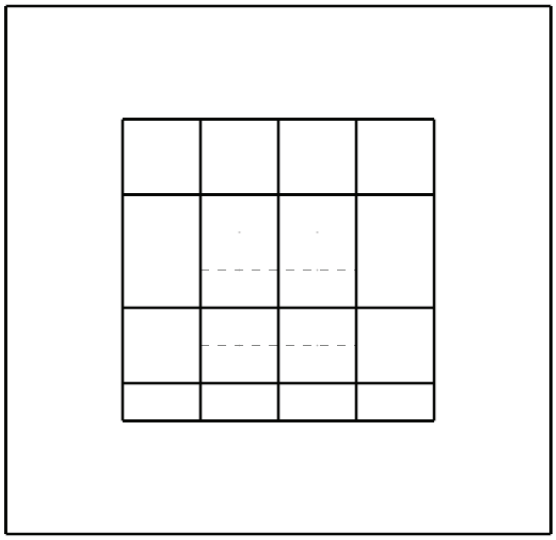
- [Select the point to place a parallel line through]: Click somewhere to the left of the selected line.

- In the **Line Parallel** panel, enter the **Offset Distance 0.5**.



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

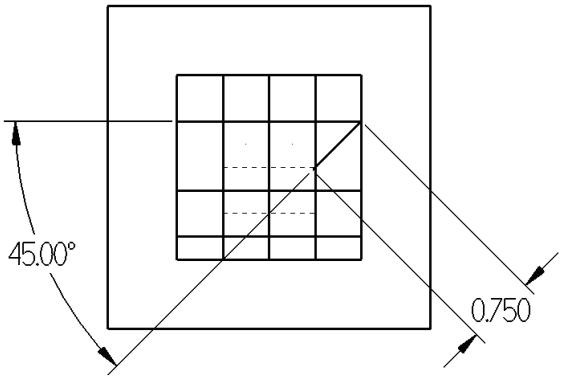
- The drawing should look as shown.



STEP 4: CREATE A POLAR LINE

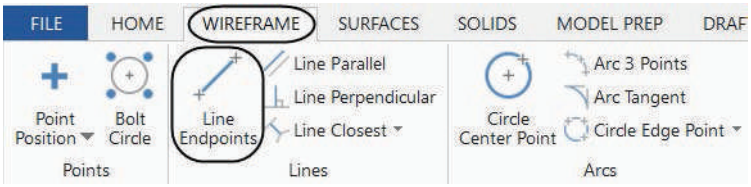
Using **Create Line Endpoint** command, you will learn how to create a line given the start point, the length, and the angle.

Step Preview:

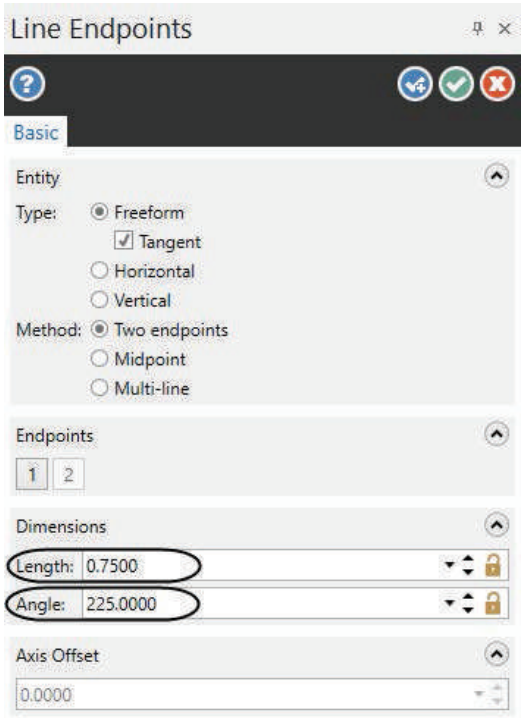


WIREFRAME

- From the **Lines** group, select **Line Endpoints**.

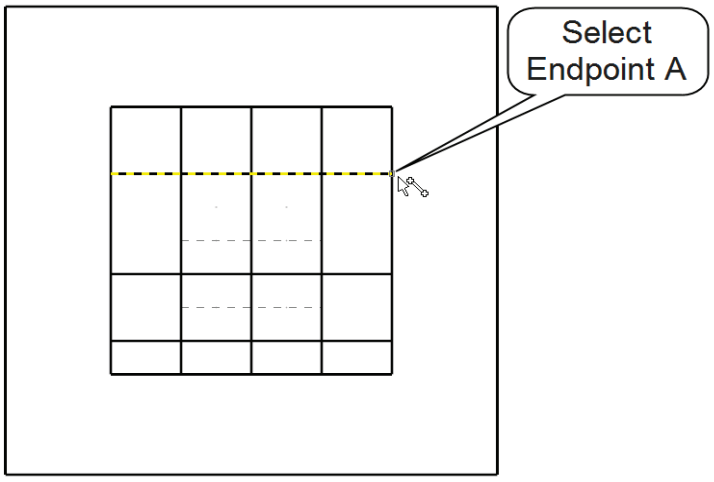


- Enter the **Length 0.75** and **Angle 225** as shown.

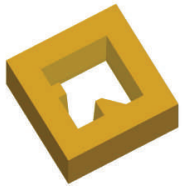


- [Specify the first endpoint]: Select the Endpoint A as shown in Figure: 4.0.1.

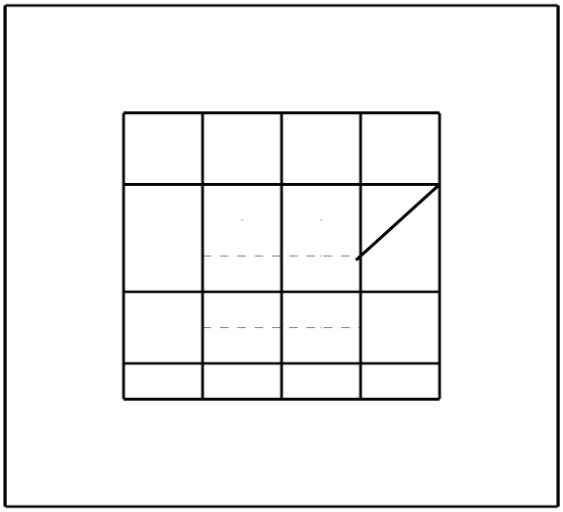
Figure: 4.0.1



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



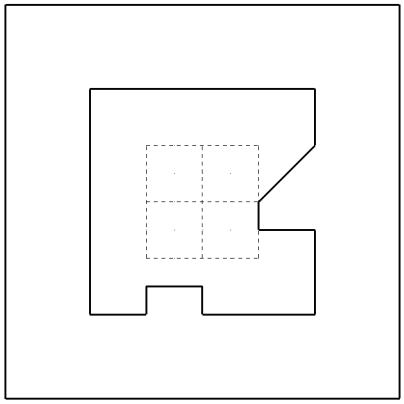
- The geometry should look as shown.



STEP 5: TRIM THREE ENTITIES AND DIVIDE

Clean up the geometry using the Trim 3 entity and the Divide commands.

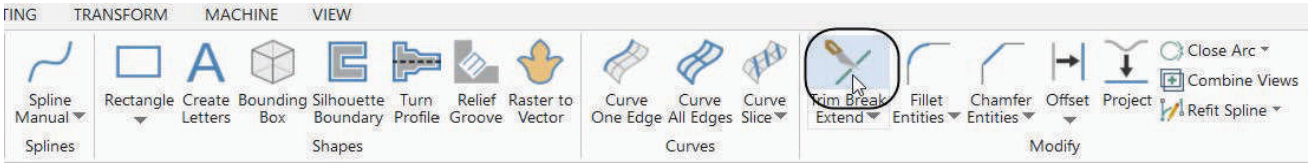
Step Preview:



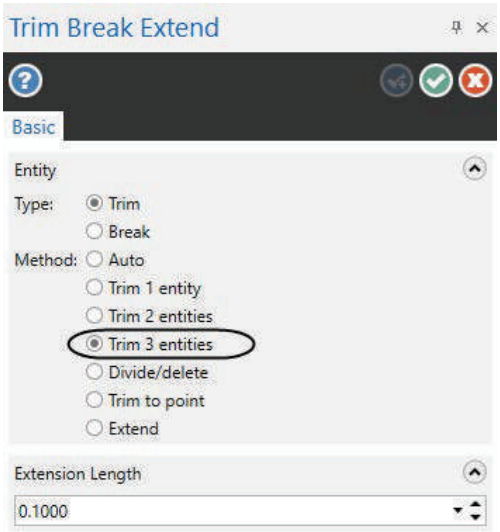
NOTE: During the trimming process it is very important to select the entities exactly in the order and at the locations that are shown in the graphics on the next few pages.

WIREFRAME

- From the **Modify** group, select **Trim Break Extend**.

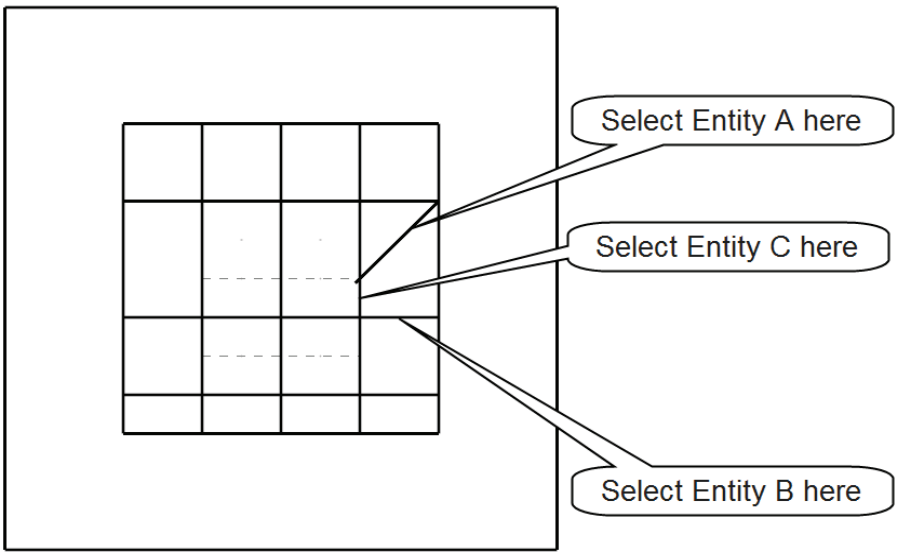


- In the **Trim Break Extend** panel, enable **Trim 3 entities** as shown.

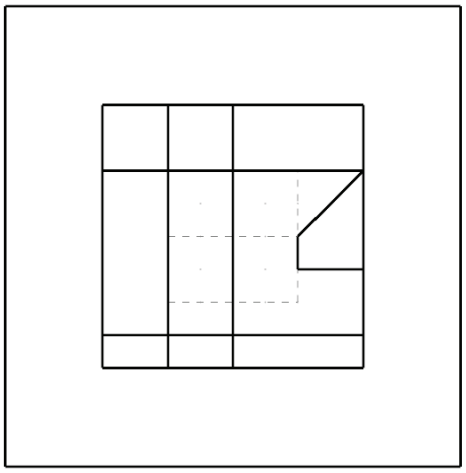


- [Select the first entity to trim/extend]: Click on Entity A as shown in Figure: 5.0.1.
- [Select the second entity to trim/extend]: Click on Entity B as shown in Figure: 5.0.1.
- [Select the entity to trim/extend to]: Click on Entity C as shown in Figure: 5.0.1.

Figure: 5.0.1

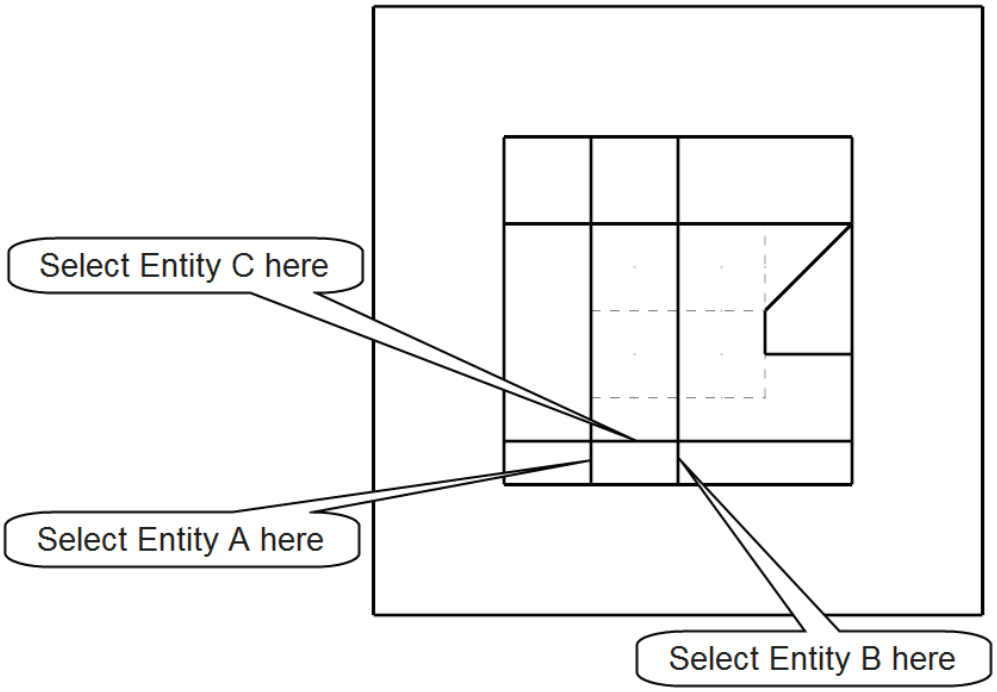


- The geometry should look as shown.

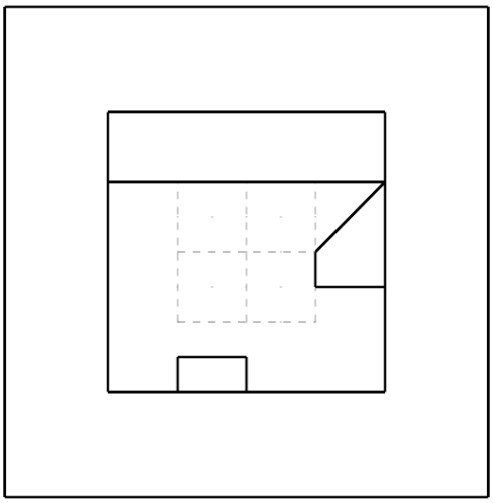


- [Select the first entity to trim/extend]: Click on Entity A as shown in Figure: 5.0.2.
- [Select the second entity to trim/extend]: Click on Entity B as shown in Figure: 5.0.2.
- [Select the entity to trim/extend to]: Click on Entity C as shown in Figure: 5.0.2.

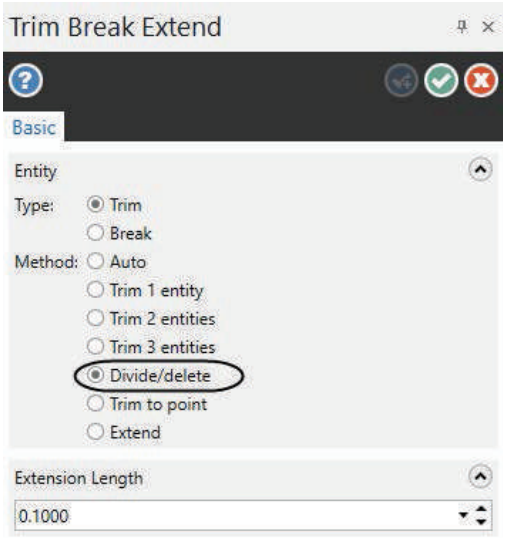
Figure: 5.0.2



- The geometry should look as shown.

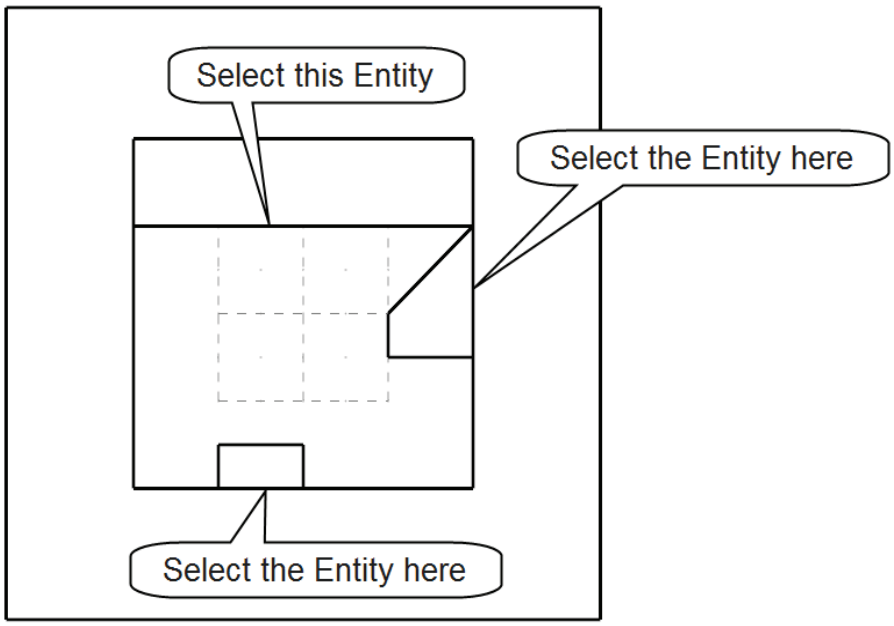


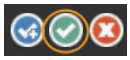
- From the **Trim Break Extend** panel**,** enable **Divide/Delete**.

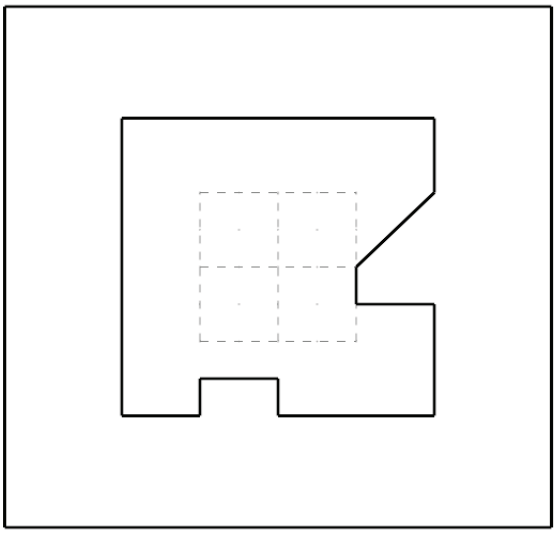


- Select the entities at the same locations as shown in Figure: 5.0.3.

Figure: 5.0.3



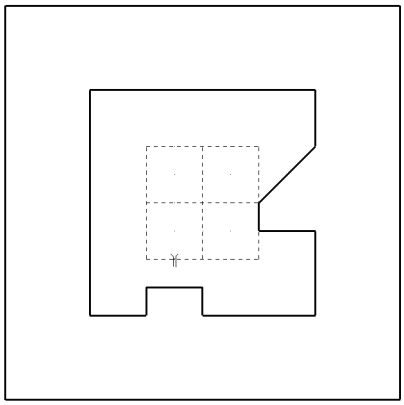
- Select the **OK** button to exit the command. 
- The geometry should look as shown.



STEP 6: CREATE THE THREAD POINT

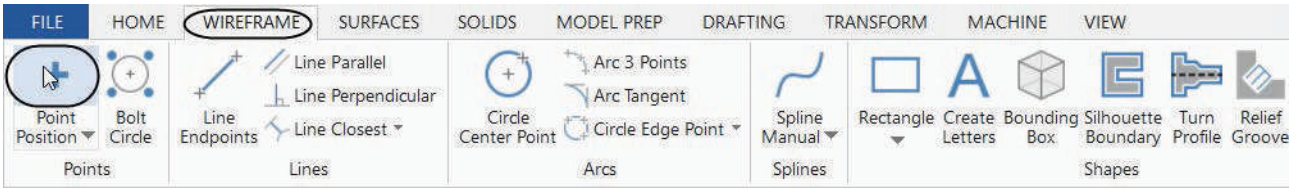
Thread Point is the point where the machine threads the wire, usually a pre-drilled hole in the material. The wire cuts from the thread position to the start of the chain based on the settings for lead in and lead out.

Step Preview:

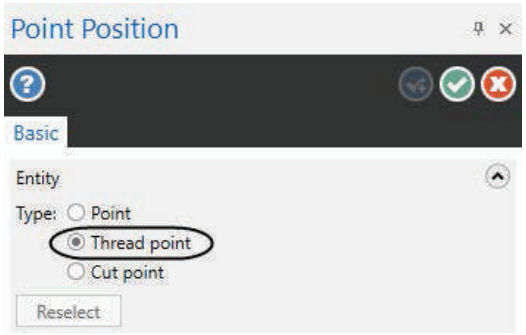


WIREFRAME

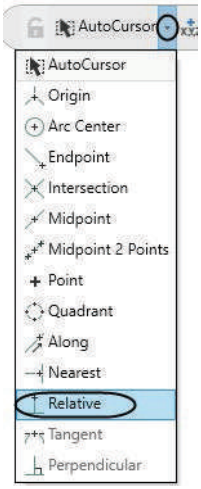
- From the **Points** group, select **Point Position** as shown.



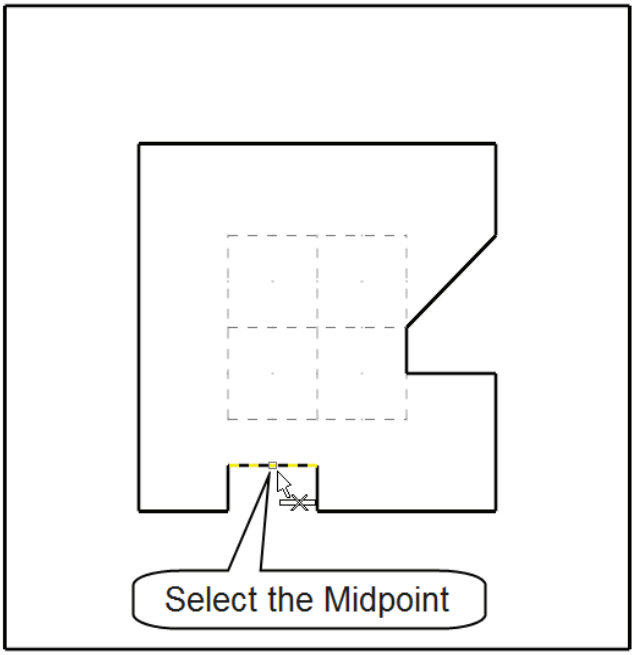
- Select **Thread Point** in the **Point Position** panel as shown.



- From the **AutoCursor** drop down list, select **Relative**.

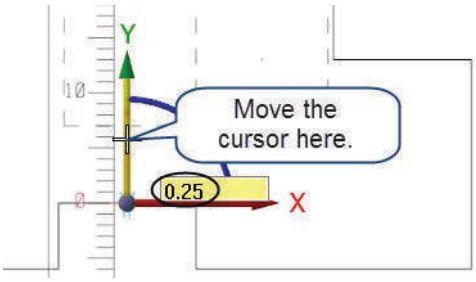


- [Pick a known point to use as a reference]: Select the **Midpoint** of the line as shown.

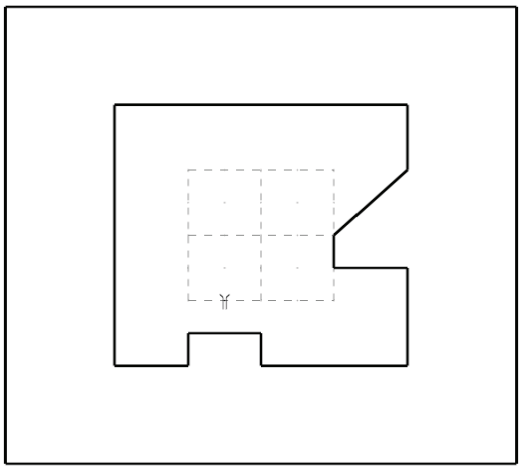


- Make sure that when selecting the **Midpoint**, the visual cue of the cursor changes as shown.

- Hover the cursor above the **Y axis** and enter **0.25** to create the point above the line as shown.



- Press **Enter** three times to create the point.
- Select the **OK** button to exit the command.
- The geometry should look as shown.



STEP 7: SAVE THE FILE

- From the **Quick Access Toolbar**, select the **Save As** icon.
- Save the file as "Tutorial 2_ Blanking Die.MCAM".

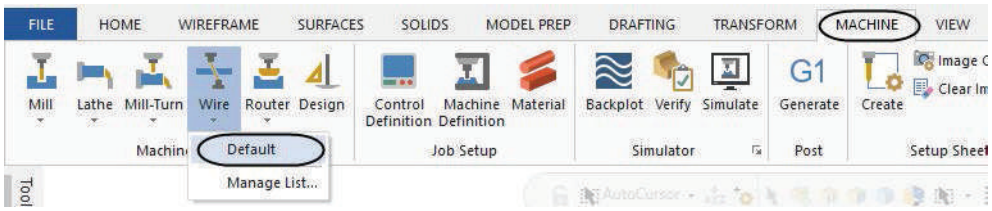
STEP 8: SELECT THE MACHINE AND SET UP THE STOCK

In Mastercam, you select a **Machine Definition** before creating any toolpaths. The **Machine Definition** is a model of your machine tool's capabilities and 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.WMD-9. Once finished, select OK to exit the dialog box. Otherwise, please follow next step.

MACHINE

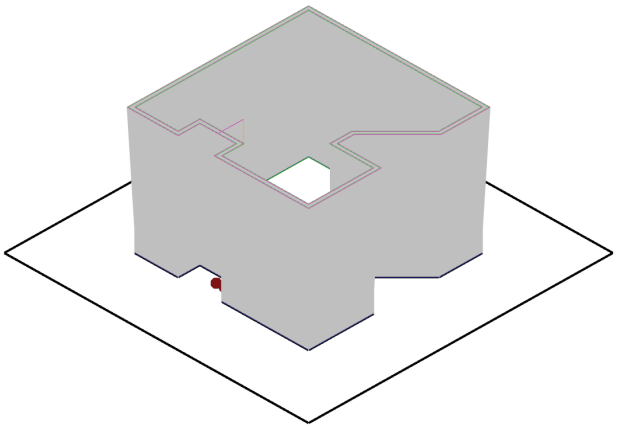
- From the **Machine Type** group, click on the pull down arrow under **Wire** and select the **Default** as shown.



STEP 9: MACHINE THE BLANKING DIE USING CONTOUR WIREPATH

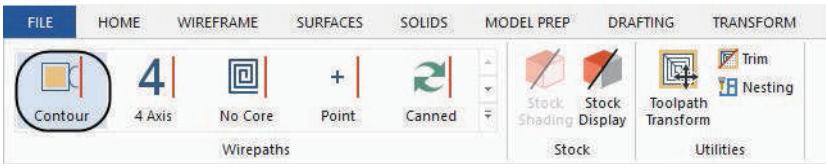
Contour wirepaths have the same general shape in both the XY plane (the lower contour) and the UV plane (the upper contour). In this case the Contour wirepath is tapered outward. You will specify the location of the land, the point at which the taper begins.

Toolpath Preview:

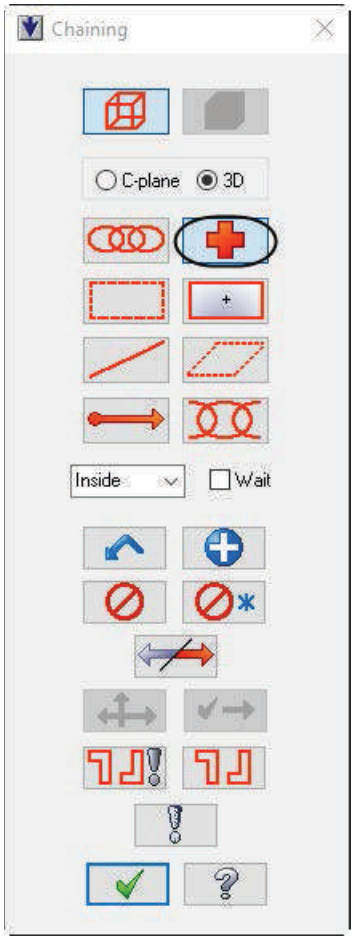


WIREPATHS

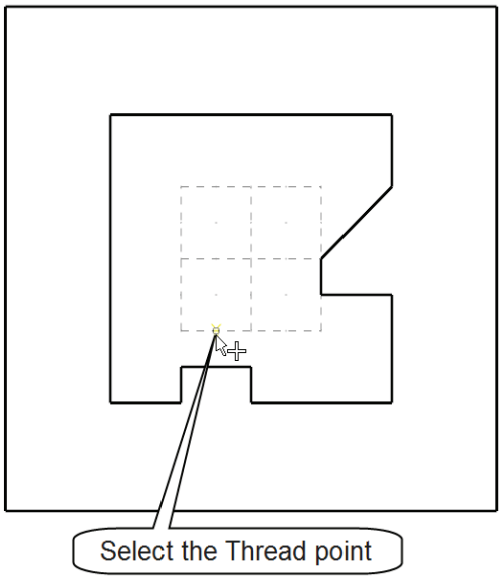
- From the **Wirepaths** group, select **Contour**.



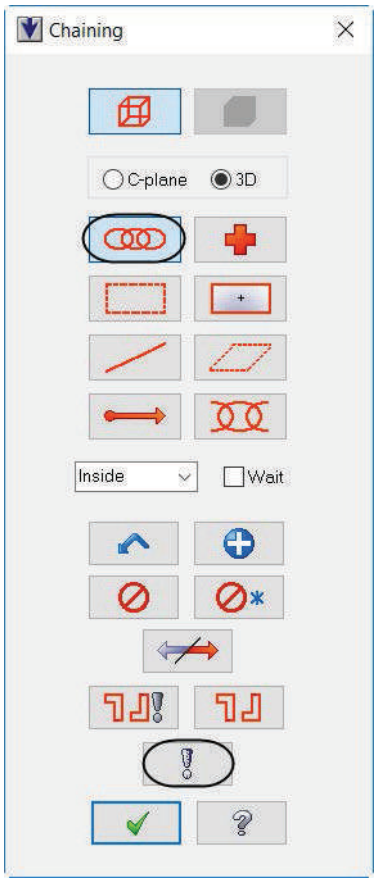
- From the **Chaining** dialog box, select the **Point** button.



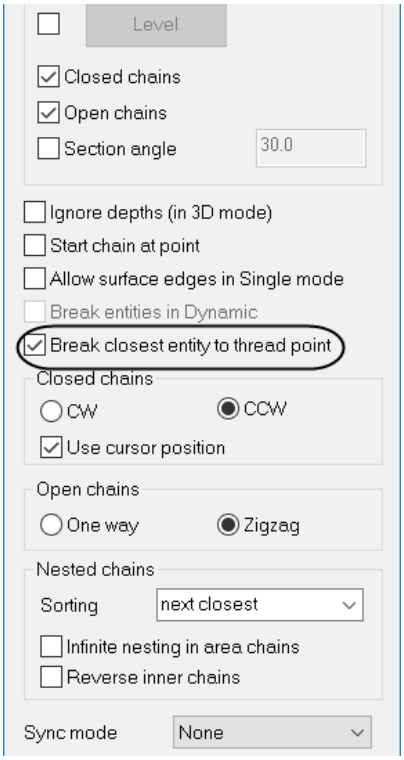
- [Contour: select chain 1]: Select the thread point as shown.



- From the **Chaining** dialog box, select the **Chain** button. Then select the **Options** button as shown.



- Enable **Break closest entity to thread point**.

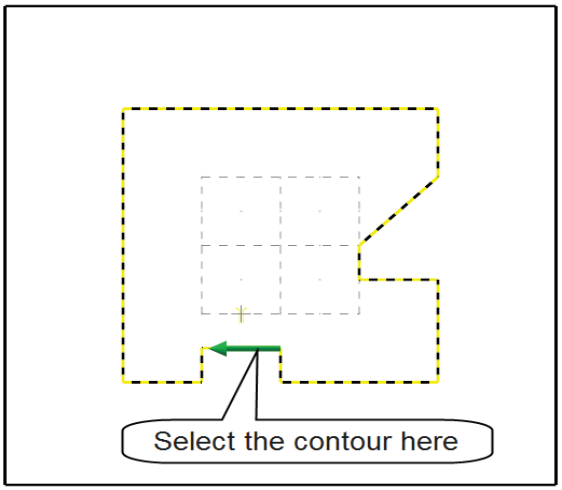


NOTE: Break closest entity to thread point establishes a wirepath start point by breaking the entity closest to the thread point at a position perpendicular to it. If a perpendicular position cannot be established, this setting is ignored.

- Select the **OK** button to exit **Chaining Options**.

- [Contour: select chain 2]: Select the contour in the CW direction as shown in Figure: 9.0.1.

Figure: 9.0.1

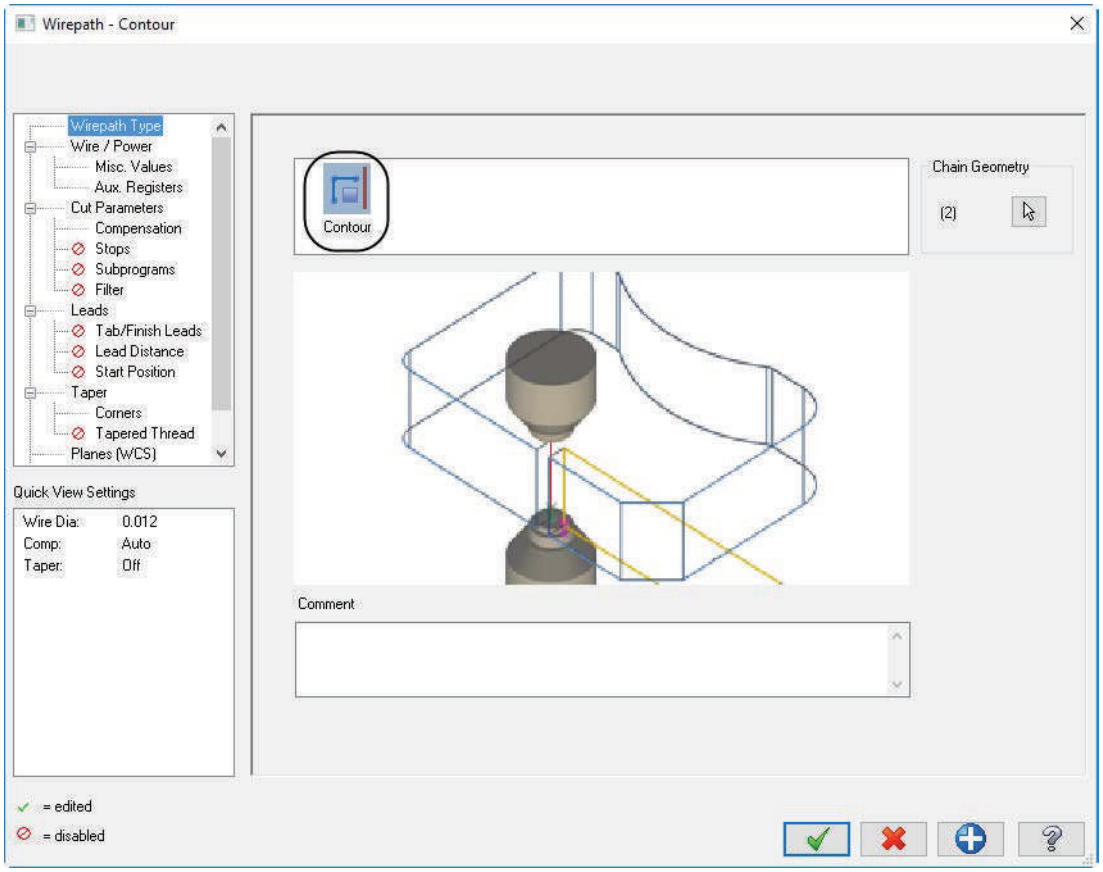


NOTE: The direction of the chain should be **Clockwise**. Otherwise select the **Reverse** button.

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

9.1 Wirepath Type

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



9.2 Set the Wire/Power parameters

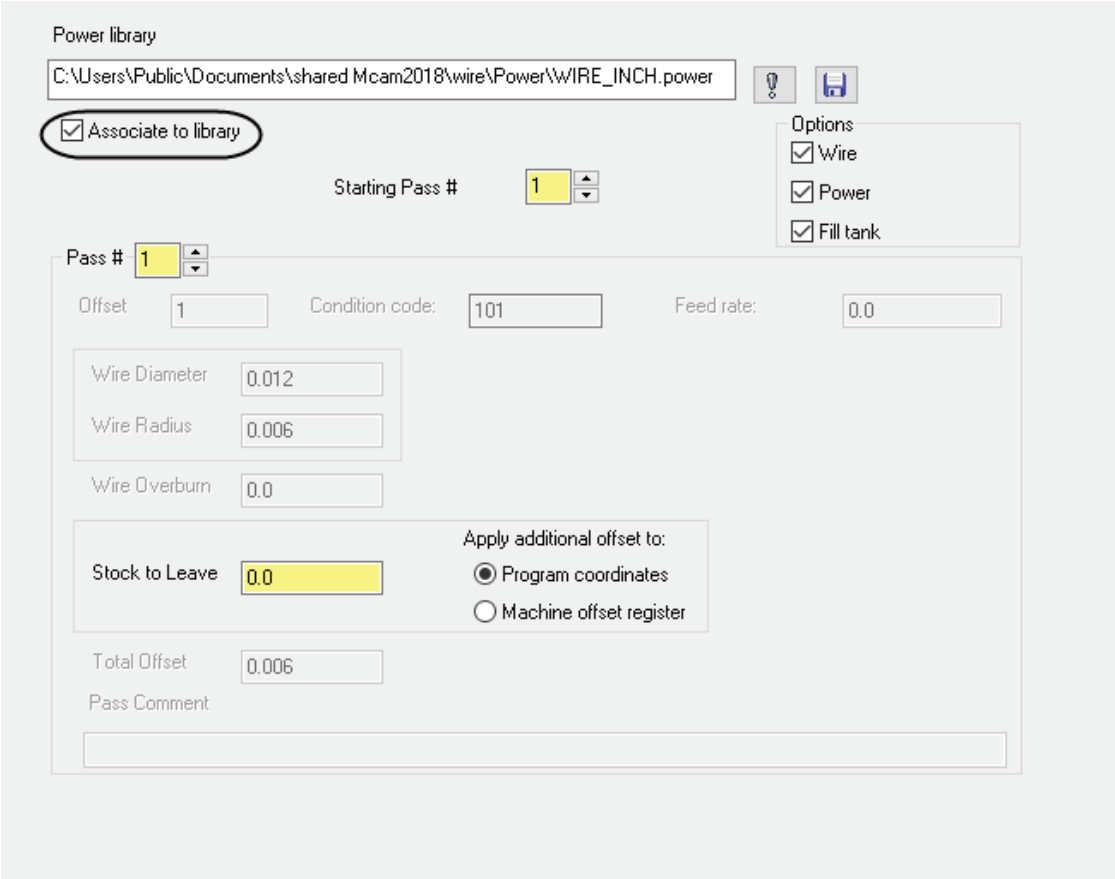
The **Wire/Power** page allows you to choose a wire power settings library file and use its values for the wirepath operation. You can also set the pass number to use for the first pass of the wirepath, and specify the settings for a single pass. To edit the power settings for each pass you have to disassociate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

- From the **Tree View** area, select **Wire/Power**.

NOTE: If you need to create your own wire power library, see **Tutorial #1 Step 10.2** for more information. To open the library saved in Tutorial #1, click on the **Open** icon and select the library from the Power library list. Otherwise, use the default WIRE_INCH.power library as shown in this tutorial.

- Enable **Associate to library** to use the default **WIRE_INCH.power** library as shown in Figure: 9.2.1.
- If an **Edit Power Settings** dialog box appears, select the option **No.**

Figure: 9.2.1

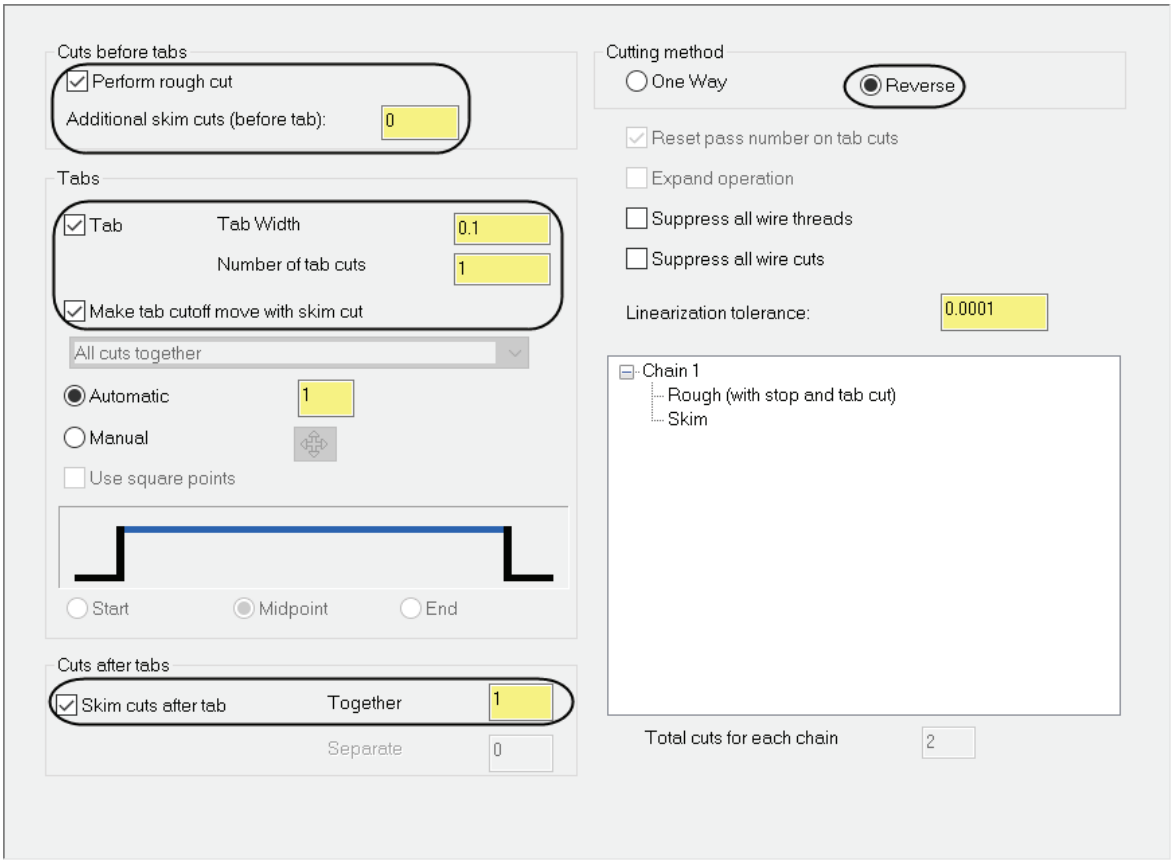


9.3 Set the Cut Parameters

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 , b 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, one tab cut, and one finish pass as shown in Figure: 9.3.1.

Figure: 9.3.1



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

Tab is the uncut material that secures the part to the material; can be cutoff with the skim cut pass or in a separate pass.

Skim cuts after tab allows you to enter the number of finish passes after cutting the tab.

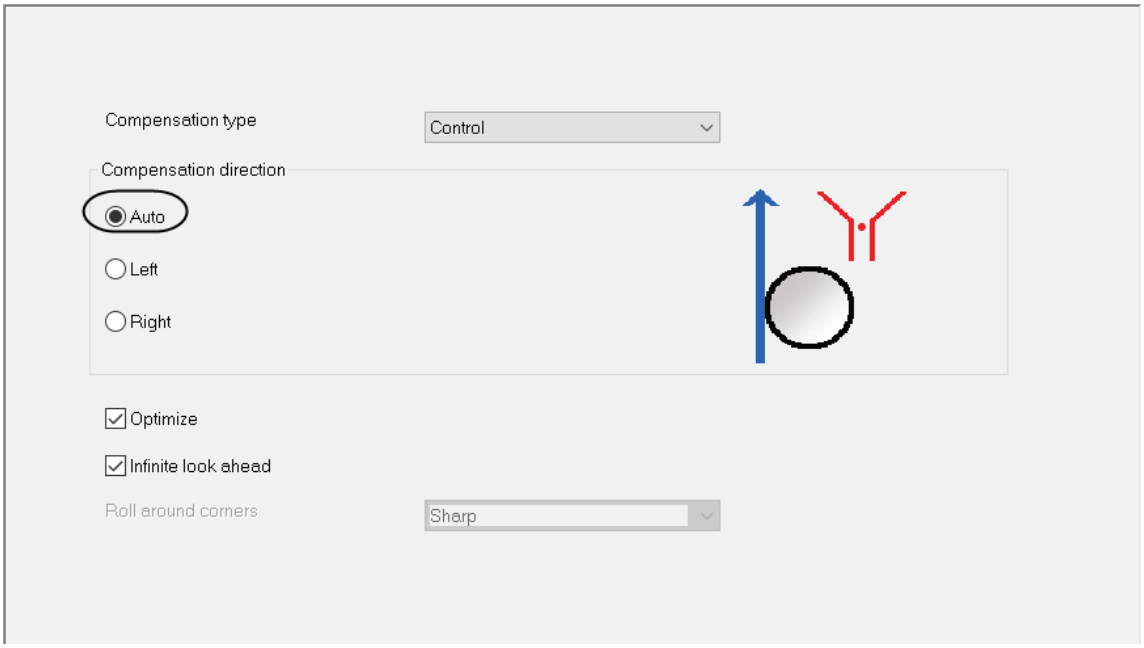
Cutting method set to **Reverse** is used when multiple passes are set for the toolpath. With **Reverse** enabled, Mastercam reverses the direction of the previous pass at the end of the contour or at the tab.

9.4 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**.
- Leave the **Compensation direction** set to **Auto** as shown in Figure: 9.4.1.

Figure: 9.4.1



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.

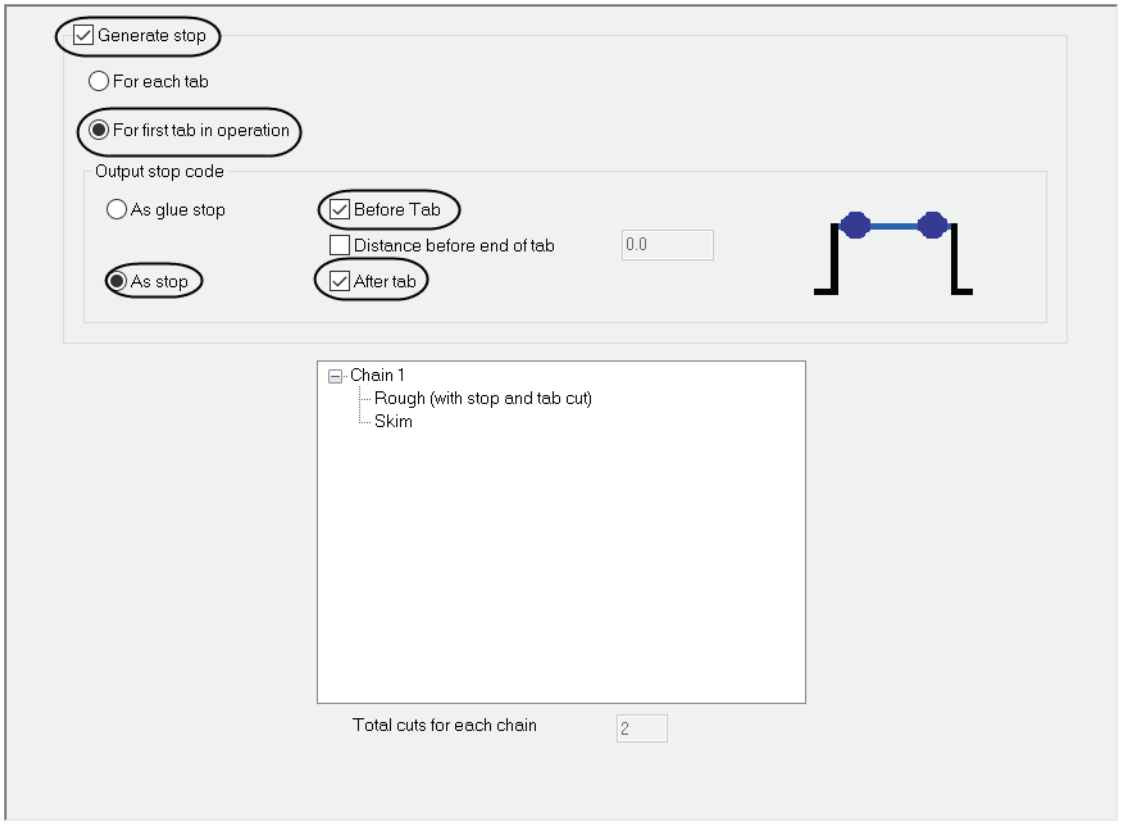
Roll around corners inserts arc moves around corners in the wirepath. The radius of the arc moves equals the radius of the wire. Not available with compensation type set to **Control**.

9.5 Stops

Stops page allows you to create optional stops (M01) or program stops (M00) in a wirepath. For contour wirepaths, you have the options to apply the stops before and/or after tabs.

- From the **Tree View** area, select **Stops**.
- Make sure that the parameters are set to generate the stop (**M00**) before and after the **Tab** as shown in Figure: 9.5.1.

Figure: 9.5.1



For first tab in operation outputs a stop code before the tab cut for the first tab cut on Chain 1 of the operation. Subsequent tab cuts throughout the operation do not include stop codes.

As stop outputs a program stop code (M00). Pauses the wire machine until the operator restarts it.

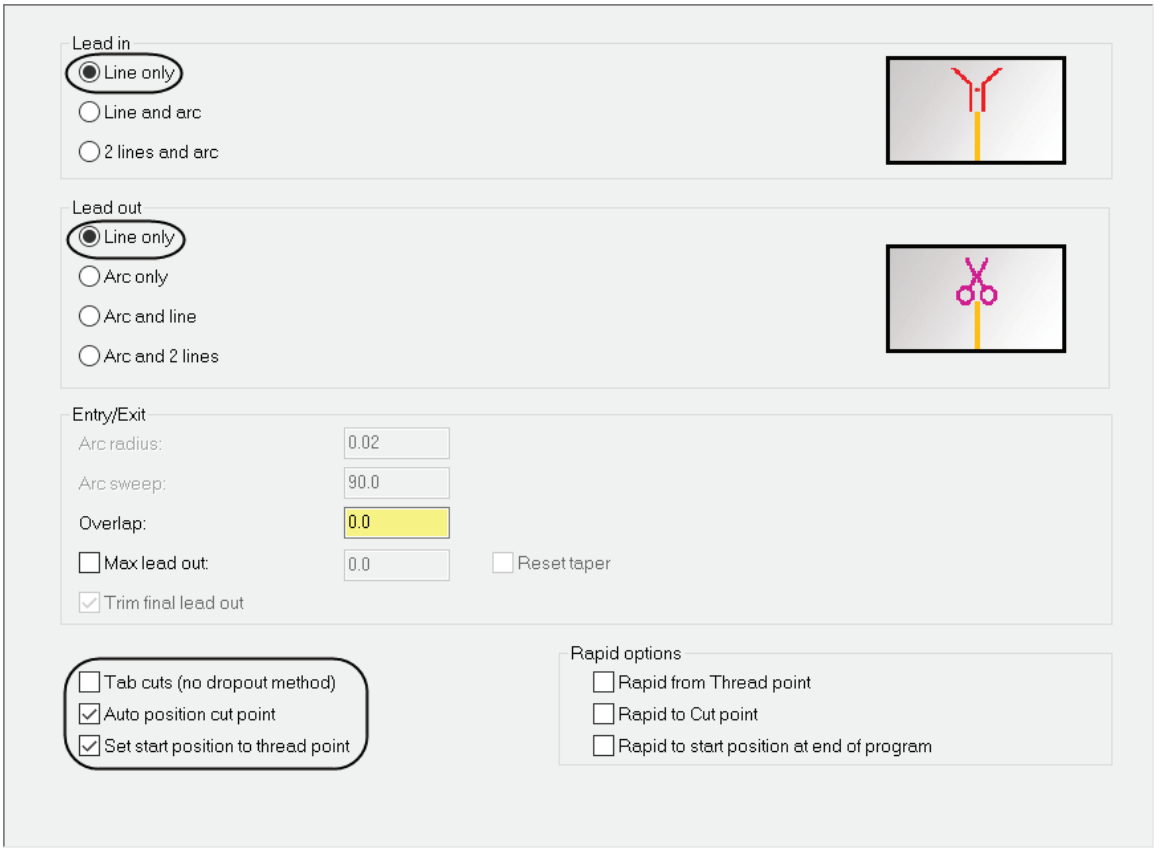
Before Tab and **After Tab** places the glue stop or the program stop (whichever stop code you select) either before or after the selected tab.

9.6 Leads

Leads page allows you to set the wire motion when entering or exiting a part.

- From the **Tree View** area, select **Leads**.
- Make sure that the **Lead in** and **Lead out** are set to **Line only** as shown in Figure: 9.6.1.

Figure: 9.6.1



Lead in sets the shape of the part entry motion. The lead in extends from the thread point to the start of contour.

Line only creates a line between the thread point and start of contour.

Lead out sets the shape of the part exit motion. Lead out extends from the end of contour to the cut point. You have the same options as in the Lead in plus the following extra options.

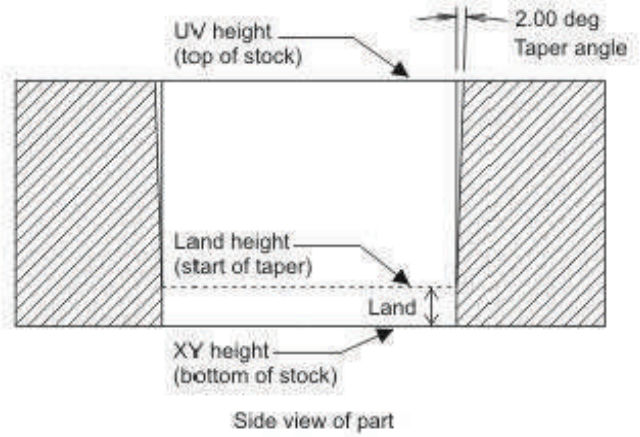
Tab cuts (No dropout method) eliminates slugs or slivers of material that result from the tab cut.

Set start position to thread point moves the job start point to the thread point of the first contour in the operation.

9.7 Taper

Taper page allows you to set the parameters to taper the contour wirepath inward or outward and to specify the location of the land (the point at which the taper begins). You can set the Z height of the upper wire guide for rapid moves, the location of the upper wire guide and lower wire guide, the height of the part, and the location of the XY plane as shown in Figure: 9.7.1.

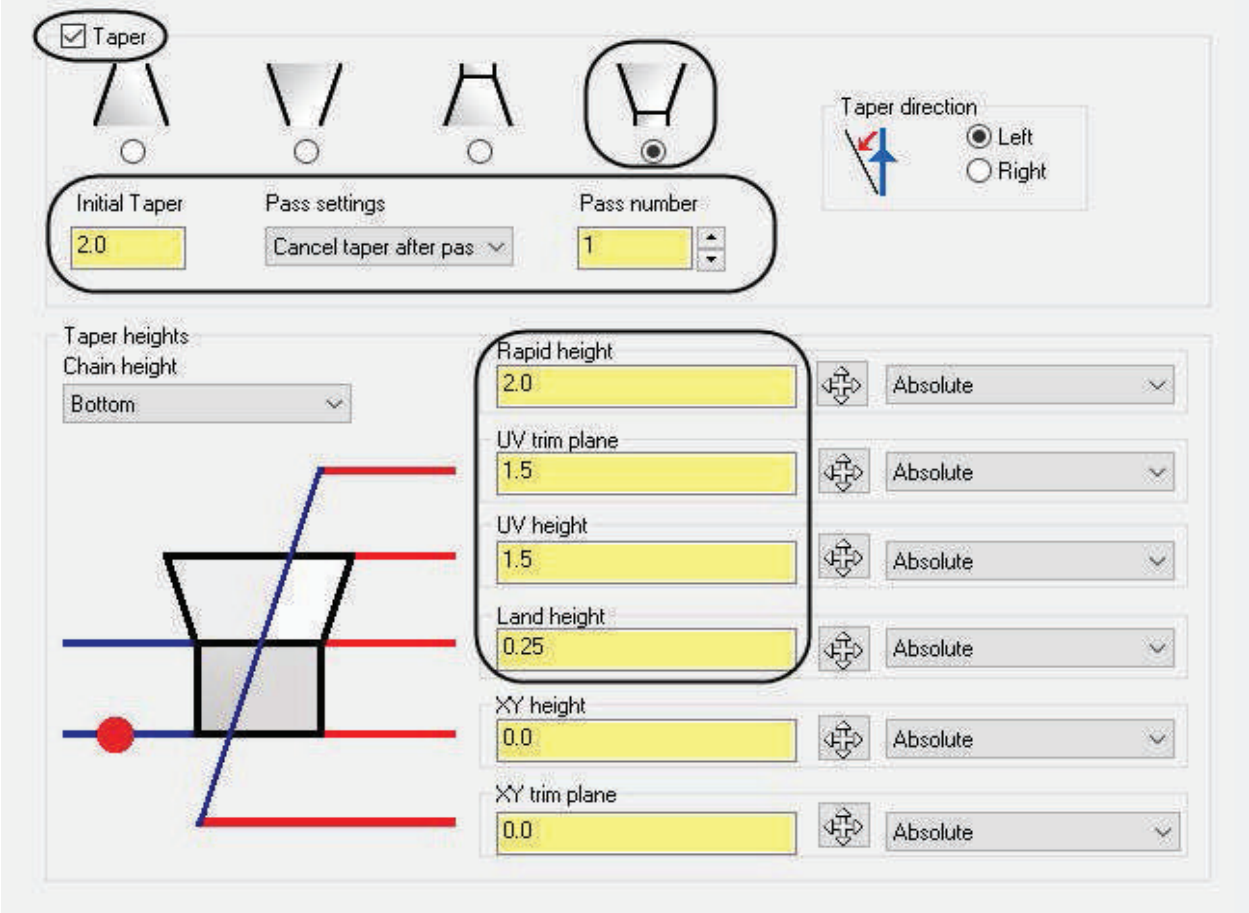
Figure: 9.7.1



- From the **Tree View** area, select **Taper**.

- Make sure that the **Taper** is enabled and change the heights as shown in Figure: 9.7.2.

Figure: 9.7.2



Chain height options allows you to indicate where the geometry was chained (XY-**Bottom** or UV-**Top** height or, if there is a land, at the land height-**Middle**).

Rapid height sets the Z depth of a rapid move. Can be higher than the **UV height**, **UV trim**, or **UV extension planes** in order to clear any clamps.

UV trim plane can be used to set the location of the upper guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to G code 1015 in the NC file. It is usually set directly at the controller.

UV height sets the location of the XY plane that contains the geometry for the upper contour of a 4-Axis part.

Land height is available only when **Land up** or **Land down** is selected for **Taper** style, and sets the plane at which the wire pivots to the taper angle.

XY height sets the location of the XY plane that contains the geometry for the lower contour of a 4-Axis part.

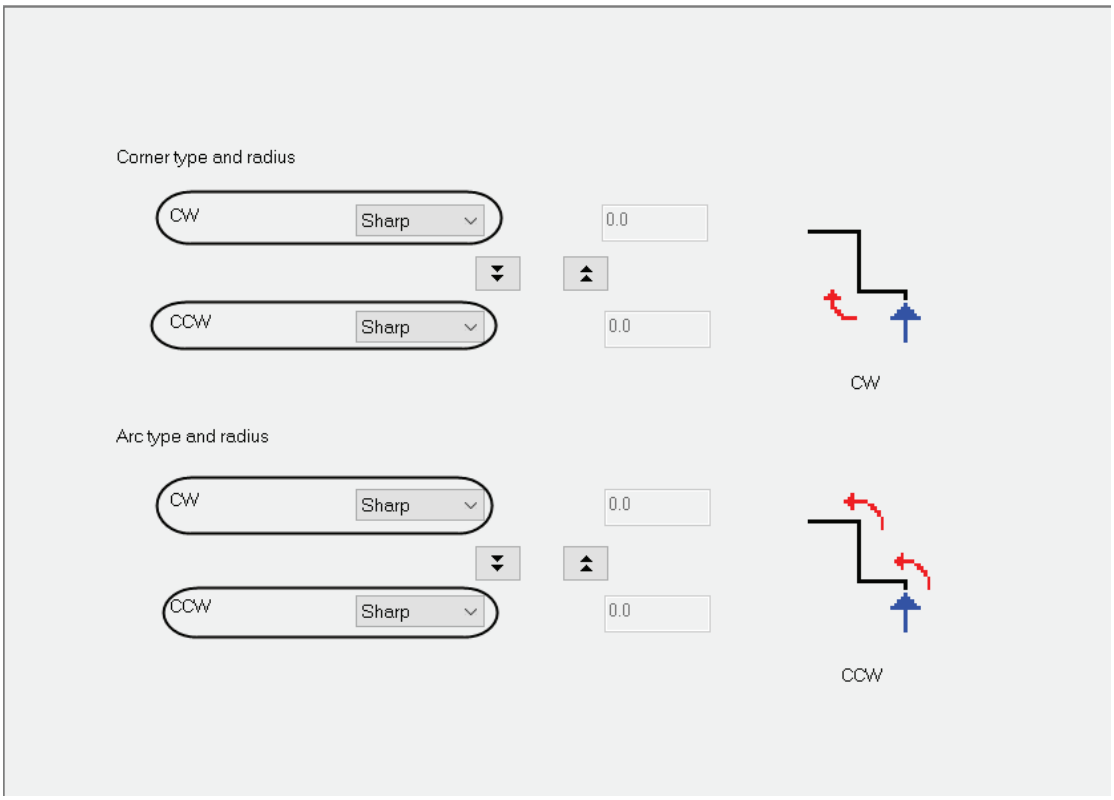
XY trim plane can be used to set the location of the lower guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to Gcode 1015 in the NC file. It can be set directly at the controller. When not using the XY trim plane, enter the same value as the XY height.

9.8 Corners

Corners page allows you to control how the wire cuts the corners in the offset contour (the contour that creates the taper). You have a choice between the following combinations: Conical, Sharp, Constant, Other, Fixed, and Fish Tail.

- From the **Tree View** area, select **Corners**.
- Make sure that the **Corner type and radius** and the **Arc type and radius** are set to **Sharp** to maintain the sharp edges from the original contour as shown in Figure: 9.8.1.

Figure: 9.8.1



Corner type and radius set to **Sharp** creates an angled move around the corner of the part.

Conical creates a cone-shaped move around the corner of the part. Conical results in a move that is bigger in the contour than in the geometry.

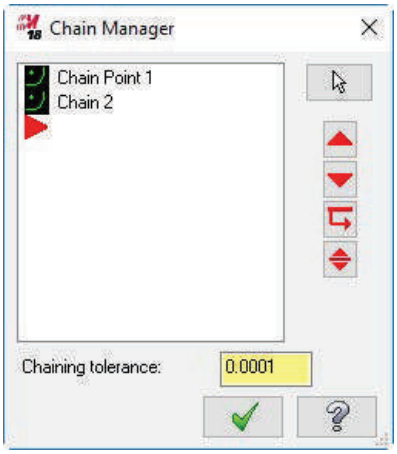
Constant keeps the corner or arc in the contour the same size as the corner or arc in the geometry.

Other allows you to specify a radius for a custom corner defined in the post processor.

Fixed applies the same radius to all corners. Fixed corners must be supported by the post processor for this selection to appear.

FishTail causes the wire to move slightly off the part, swing around in a loop, and then reenter the part. Fishtail corners create very sharp corners.

- Select the **OK** button to exit the **Wirepath - Contour** parameters.

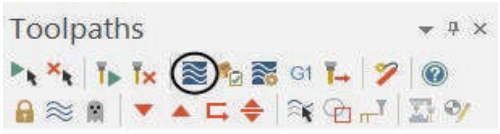


- Select the **OK** button to exit the **Chain Manager** dialog box.

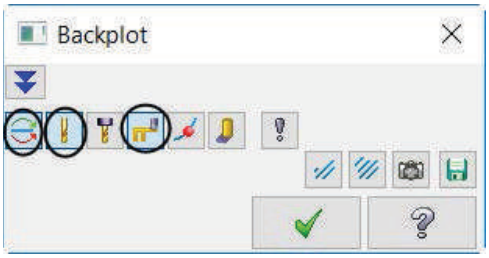
STEP 10: BACKPLOT THE WIRE CONTOUR WIREPATH

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 wirepaths, Mastercam displays the current X, Y, and Z coordinates in the lower left corner of the screen.

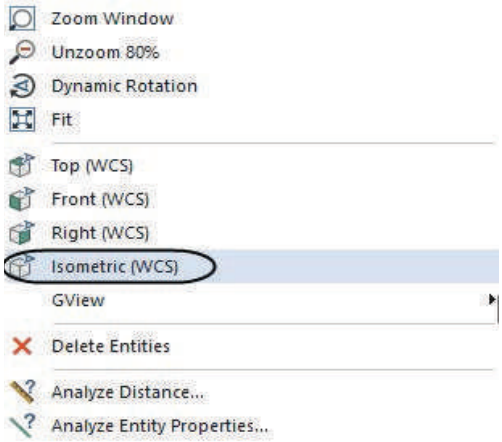
- Make sure that the toolpath is selected (signified by the green check mark on the folder icon).
- Select the **Backplot selected operations** button.



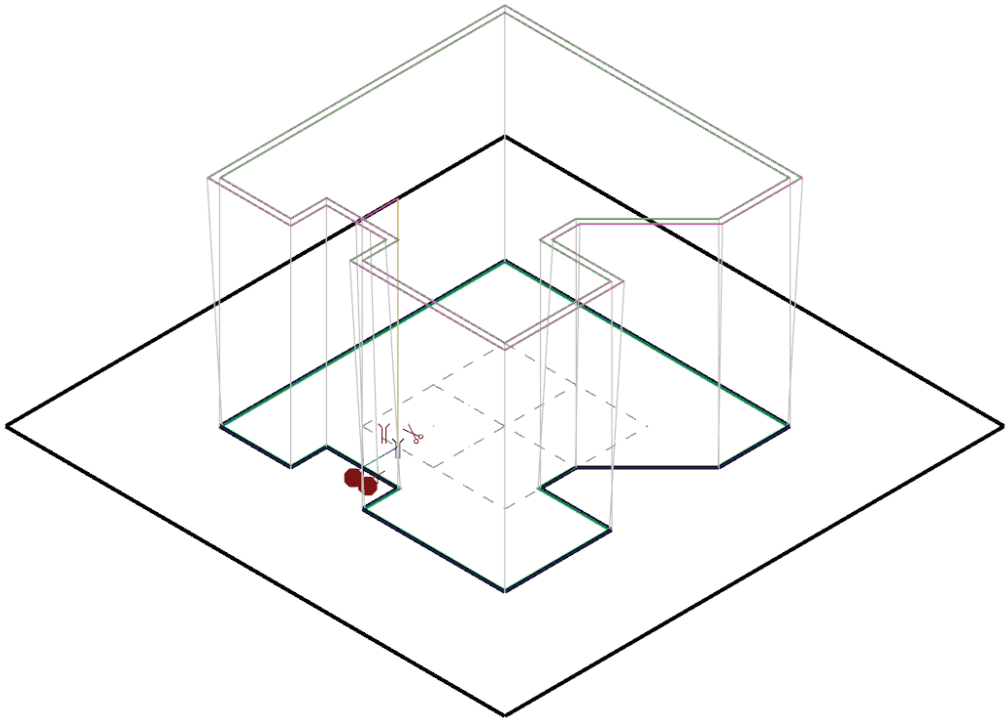
- Make sure that you have the following buttons turned on to see the tool and the rapid moves.



- Right mouse click in the graphics window and select **Isometric** as shown.

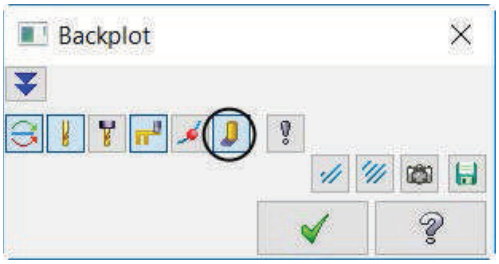


- Press **Alt+F1** to fit the drawing to the screen.
- You can adjust the speed of the backplot.
- You can step through the **Backplot** by using the **Step forward** or **Step back** buttons.
- Select the **Play** button to run **Backplot**.
- The toolpath should look as shown.

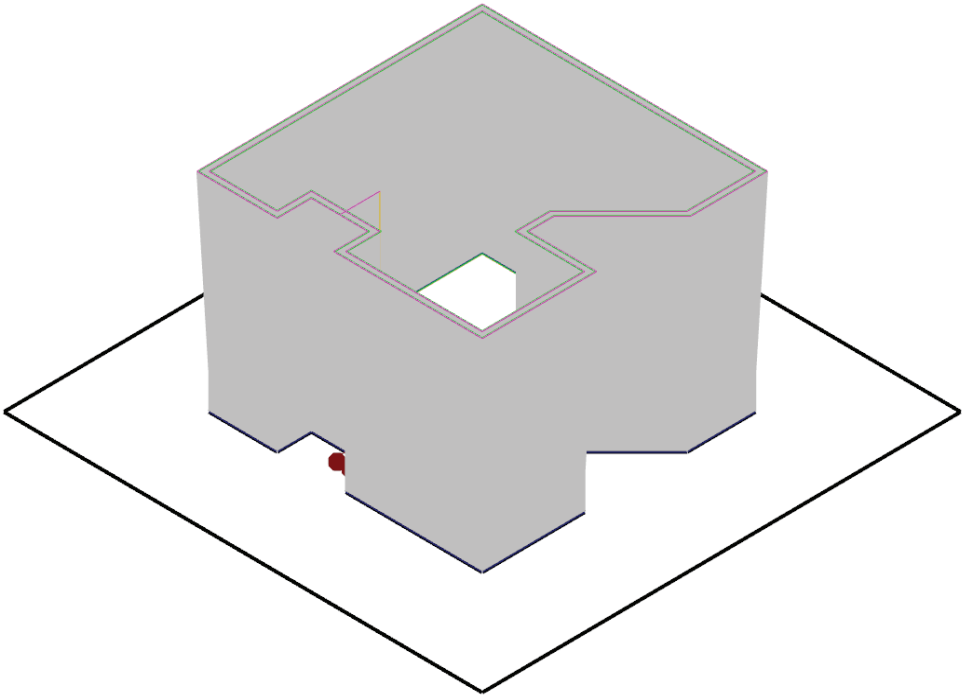


NOTE: The dots along the contour show where the machine stops are located.
The wire starts at the thread point, leads in along a line, and makes the roughing pass in the CW direction. It stops (M00) before the tab (0.1" from the starting location), allowing the operator to hold the part with a magnet. Once the operator starts the program again, the wire cuts the tab and retracts back to the thread point locations. Once the slug is removed, the operator can restart the program and make a skim cut.
You need to add another stop (M00) just after the tab is cut and slug is removed.

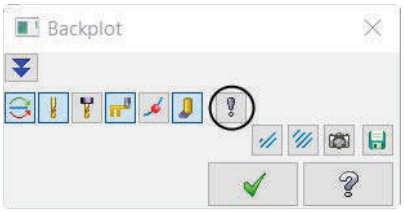
- In the **Backplot** dialog box enable **Quick** as shown.



- The toolpath should look as shown.



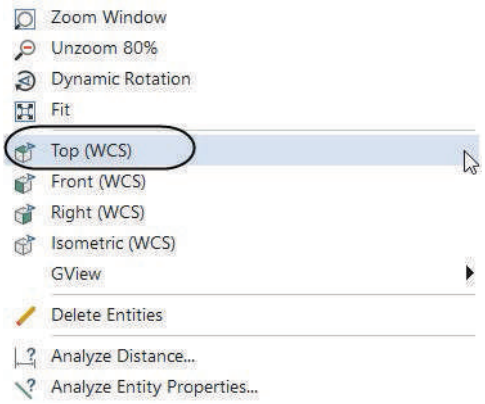
- To see if the wire is compensated on the proper side, in the **Backplot** dialog box, click on the **Options** icon as shown.



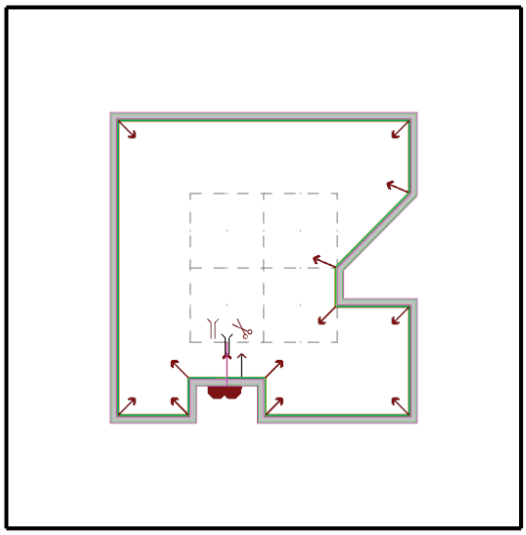
- In the **Backplot Options** dialog box, enable **Show wire comp vectors** as shown.

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

- Right mouse click in the graphics window and select the **Top** view as shown.



- Run the wirepath again and you should see the vectors as shown.

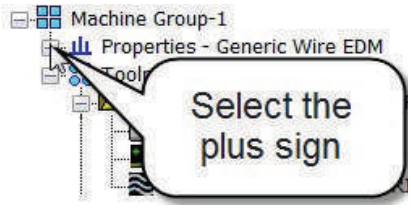


STEP 11: VERIFY THE WIREPATH

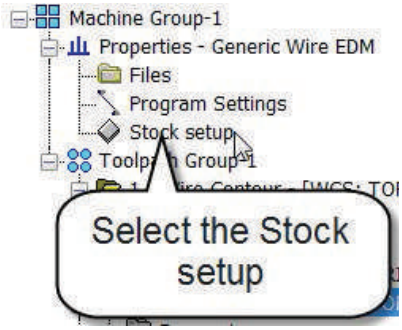
Verify simulates the machining of a part from a solid stock model display. The stock dimensions are based on the values that you can specify in the **Stock Setup**.

11.1 Set up the stock

- From the **Toolpaths Manager**, select the plus sign (+) in front of the **Properties** to expand the **Toolpaths Group Properties**.

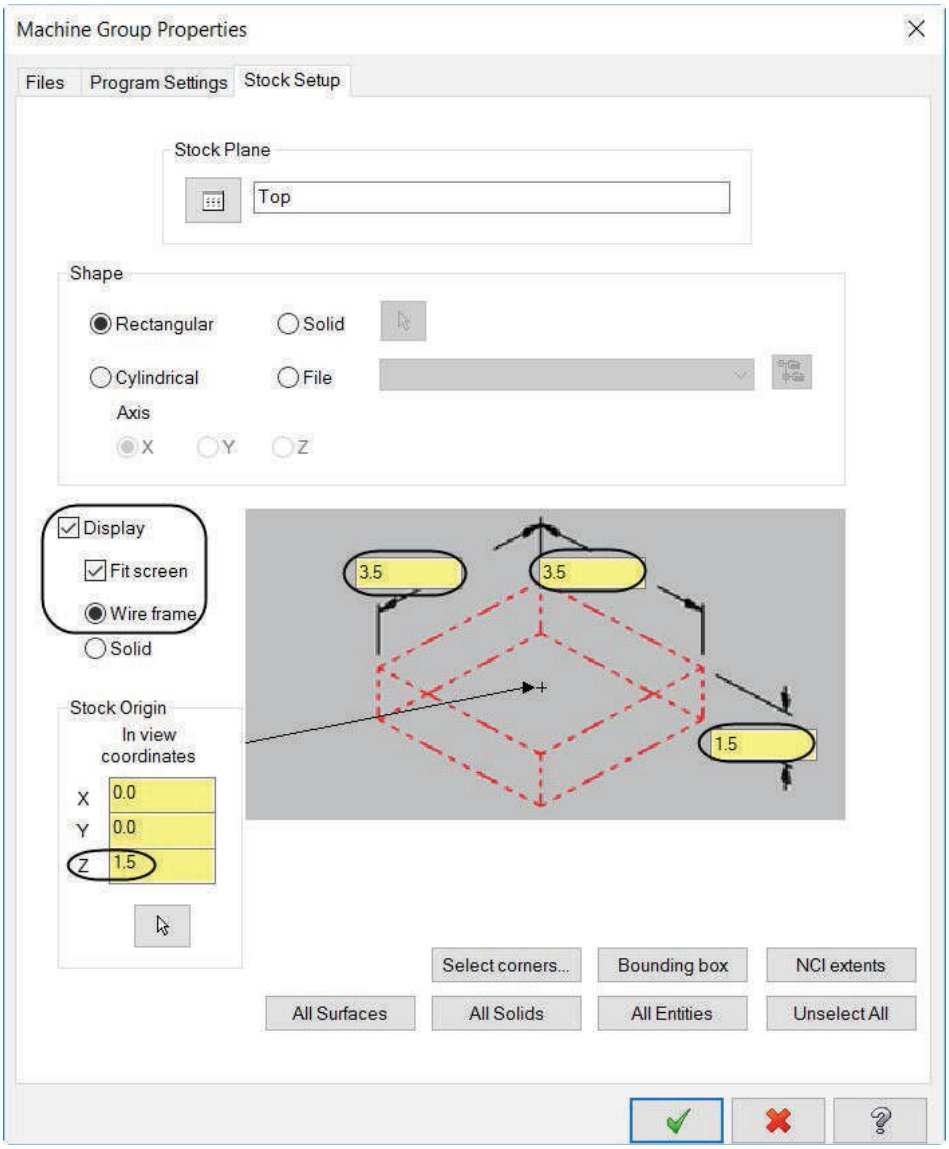


- Select the **Stock setup** to set the stock size.



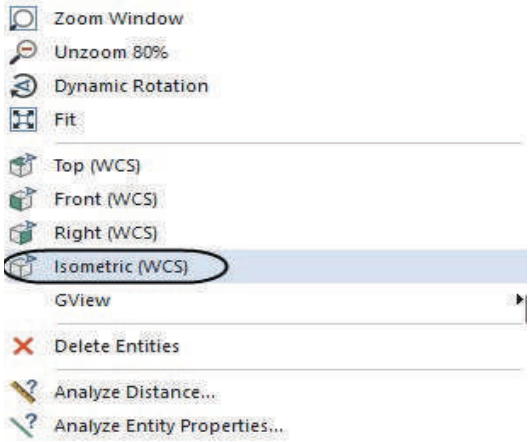
- Change the parameters to match the Figure: 11.1.1.

Figure: 11.1.1

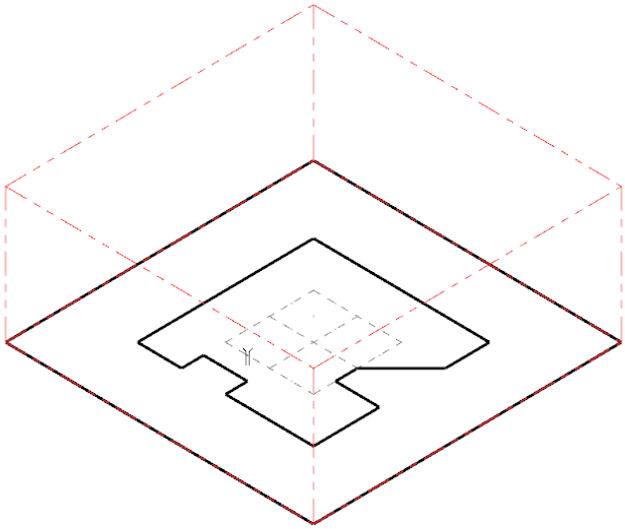


- Select the **OK** button to exit **Machine Group Properties**.

- Right mouse click in the graphics window and select **Isometric** as shown.

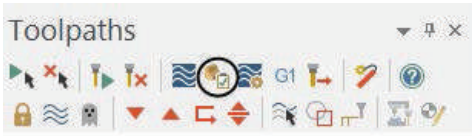


- Press **Alt+F1** to fit the drawing to the screen.
- The stock is displayed with phantom red lines and it should look as shown.



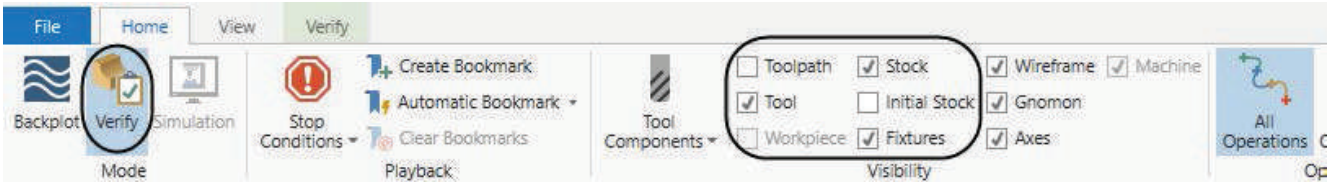
11.2 Verify the wirepaths

- Select the **Verify selected operations** button.

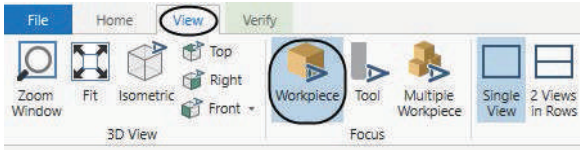


- From **Verify** change the settings for the **Visibility** as shown in Figure: 11.2.1.

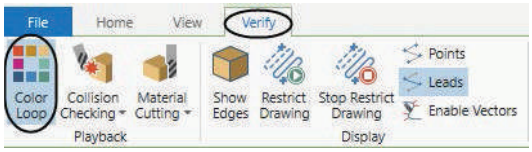
Figure: 11.2.1



- Select **View** tab and enable **Workpiece** as shown.



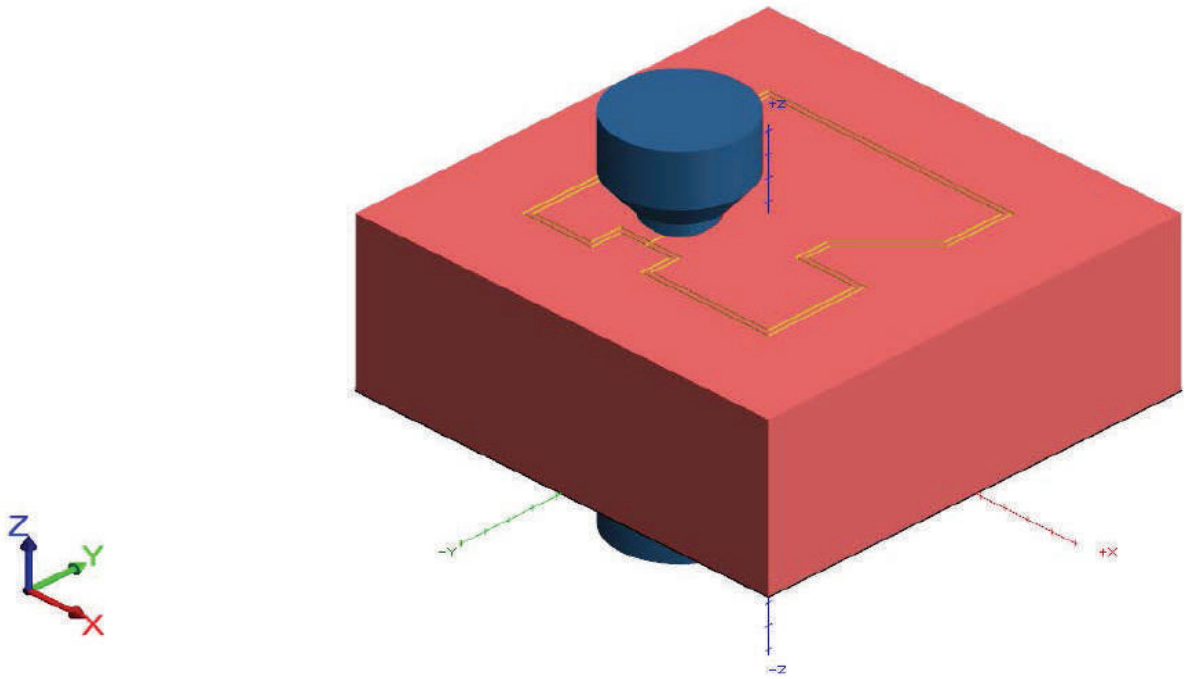
- Select **Verify** tab and enable **Color Loop** as shown.



- Right click and select the **Isometric** option.
- Right click and select the **Fit** option.
- Select the **Play** button to run **Verify**.

- The part should look as shown in Figure: 11.2.2.

Figure: 11.2.2



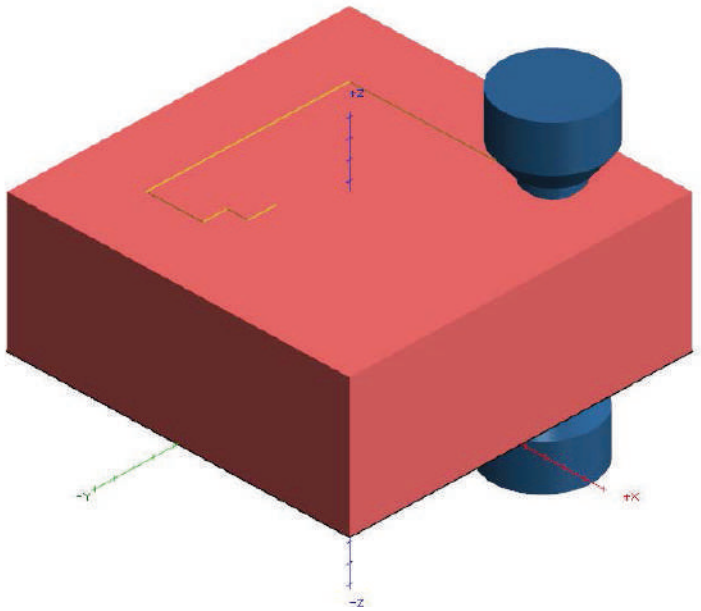
NOTE: To rotate the part, move the cursor to the center of the part, then click and hold the mouse wheel and slowly move it in one direction.
To **Zoom In** or **Out**, hold down the mouse wheel and scroll up or down as needed.

- Under the **View** tab, click on the **Isometric** icon

and then the **Fit** icon

to see the part in the original position if needed.
- To check the part step by step, click first on the **Start (Home)** icon.
- Click on the **Step Forward** icon to see the tool moving one step at a time.

- The part should look as shown after several steps.

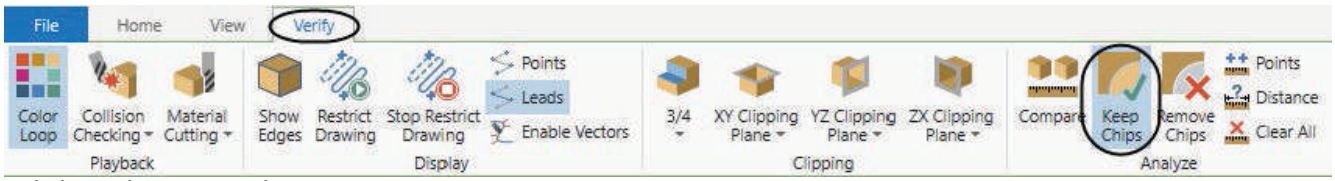


- Click on the **Step Forward** until the toolpath is completed.

11.3 Chip removal

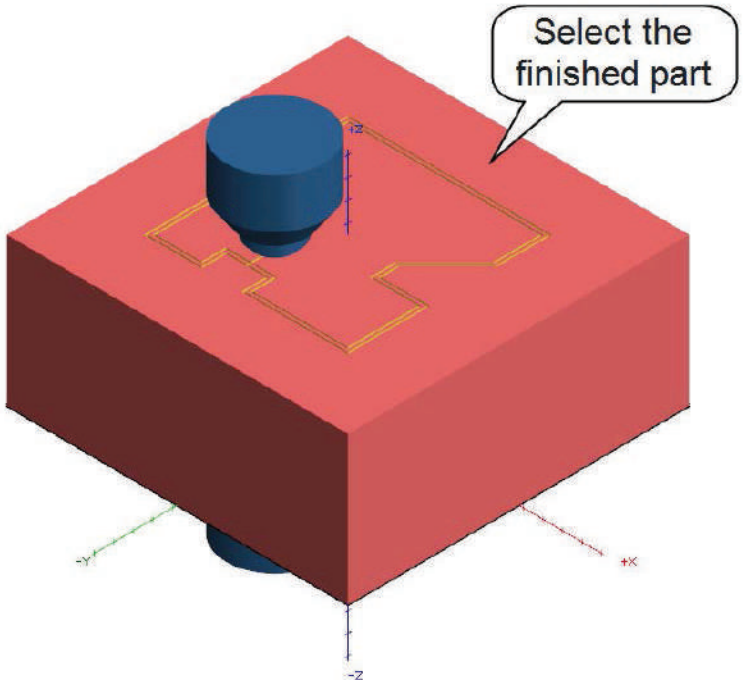
- Select the **Verify Tab** and select the **Keep Chips** icon as shown in Figure: 11.3.1.

Figure: 11.3.1



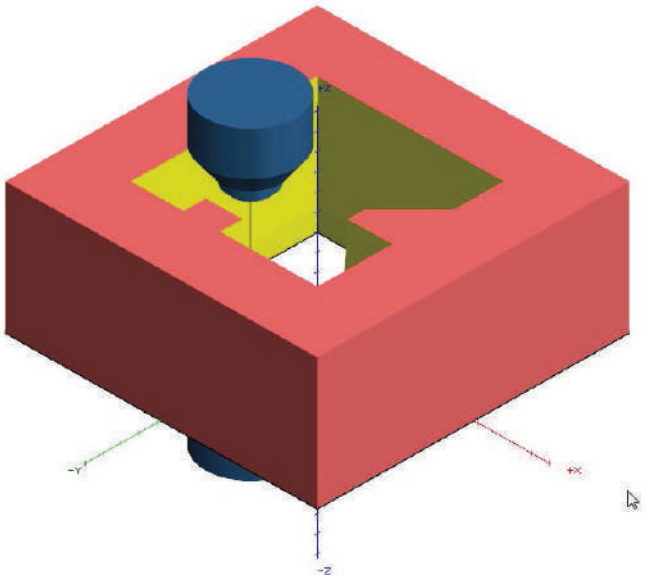
- Click on the part as shown in Figure: 11.3.2.

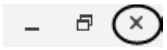
Figure: 11.3.2



- The part will appear as shown in Figure: 11.3.3.

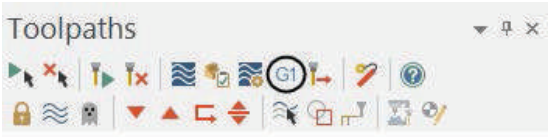
Figure: 11.3.3



- To exit **Mastercam Simulator** click on the **Close** icon. 

STEP 12: POST THE FILE TO GENERATE THE NC CODE

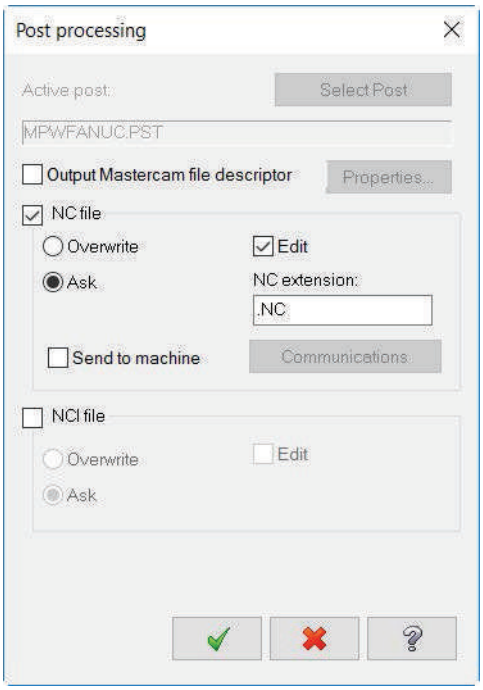
- Select **Post selected operations** button from **Toolpaths Manager**.



NOTE: The active **Post Processor** is a generic FANUC post processor.

- In the **Post processing** window, as shown in Figure: 12.0.1, if needed, make any necessary changes.

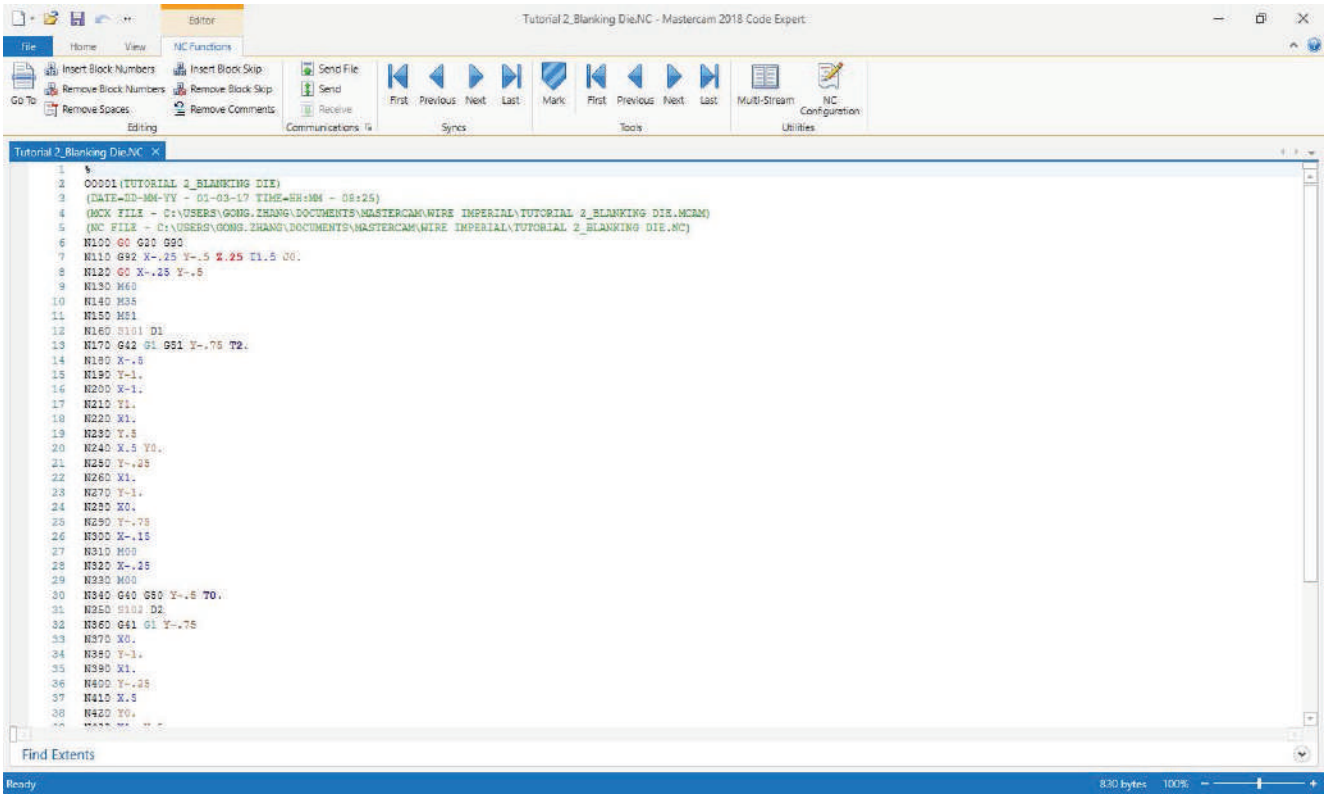
Figure: 12.0.1



- Select the **OK** button to continue.
- Save the NC file.

- The NC file will appear on the screen as shown in Figure: 12.0.2.

Figure: 12.0.2

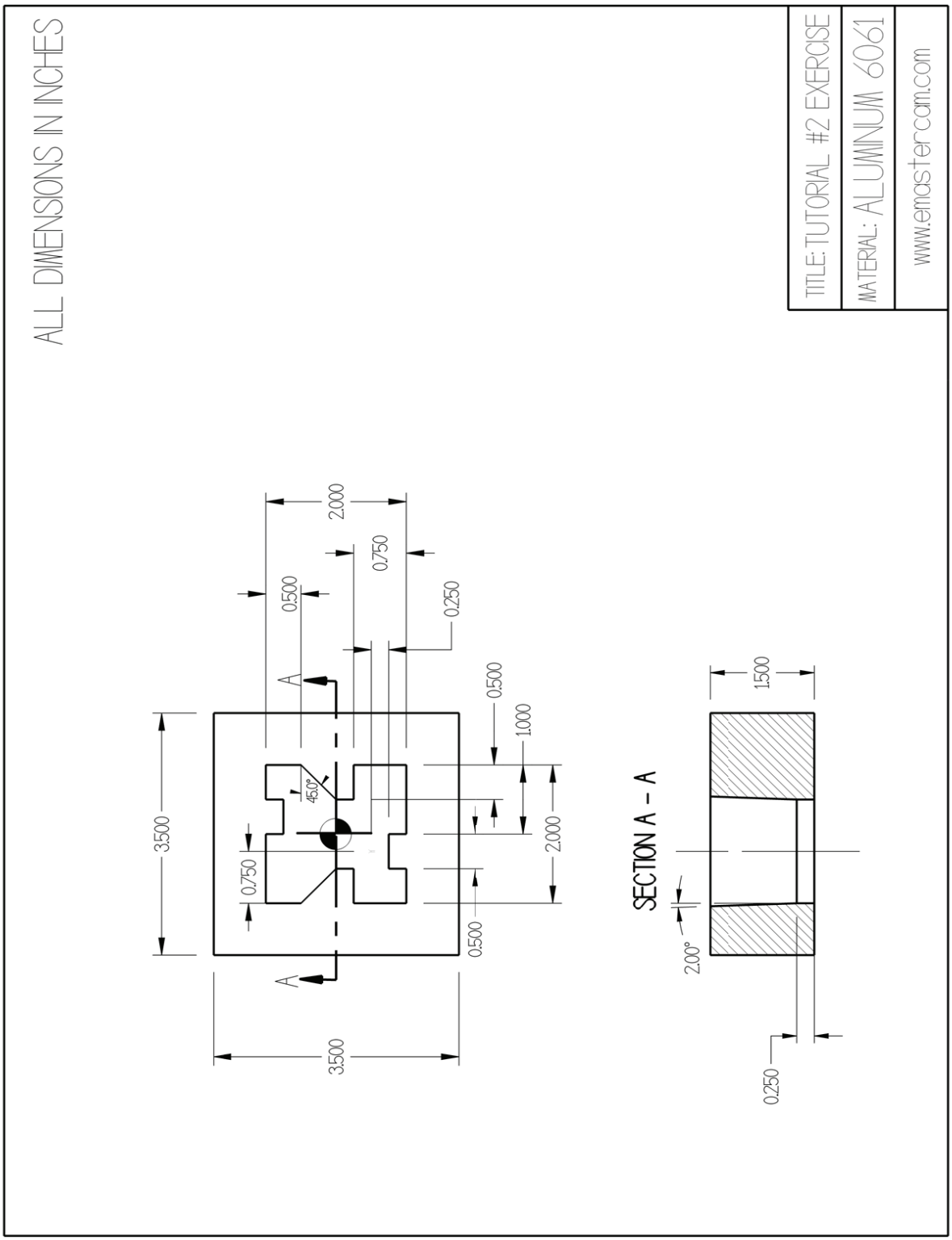


NOTE: The G code that you have created will appear on the screen. If the code looks okay, you can shut down the window without saving it. If you need to change the code, make sure you save it before you close the window.
How the program is sent to the machine depends on the shop setup.

- Select the "**X**" box at the upper right corner to exit the **Mastercam Code Expert.**

STEP 13: SAVE THE UPDATED MCAM FILE

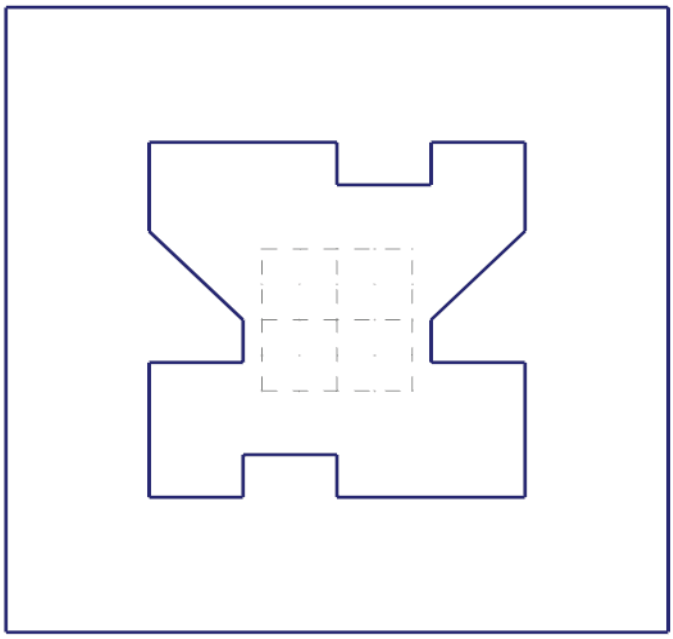
REVIEW EXERCISE -STUDENT PRACTICE

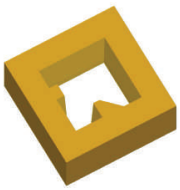


CREATE THE GEOMETRY FOR TUTORIAL #2 EXERCISE

Use these commands to create the geometry.

- Rectangle.
- Line Parallel.
- Line Endpoints.
- Trim Entities.
- Point Position - Thread Point.



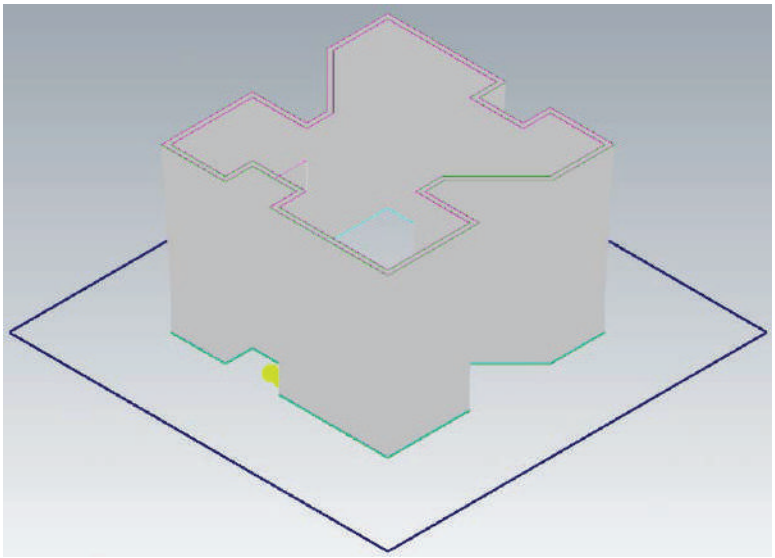


CREATE THE TOOLPATHS FOR TUTORIAL #2 EXERCISE

Create the Toolpaths for Tutorial #2 Exercise as per the instructions below.

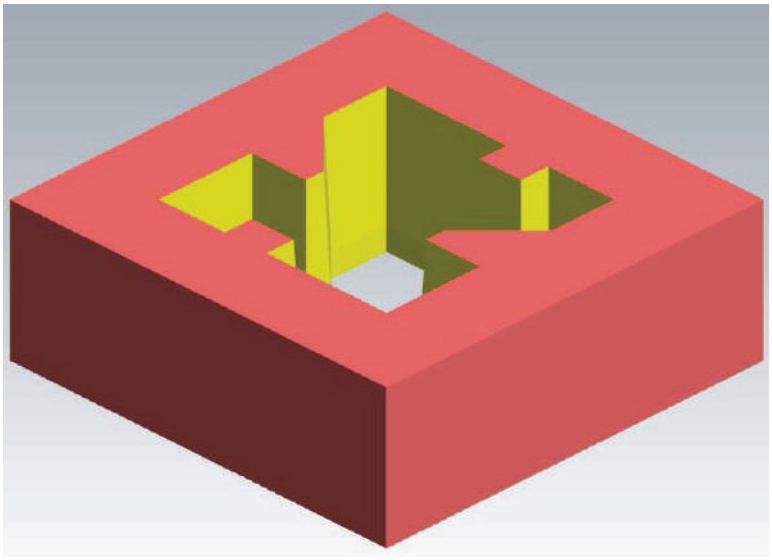
Machine The Die Using Contour Wirepath.

- Inorder to complete this operation follow **page 98** to **page 107.**



Verify The Wirepath.

- Set the **Stock** according to the part.
- Inorder to complete this operation **Verify** the part and follow **page 115** to **page 118**.



NOTES:

TUTORIAL #2 QUIZ

- What command allows you to create a line given one endpoint, the length, and the angle of the line?

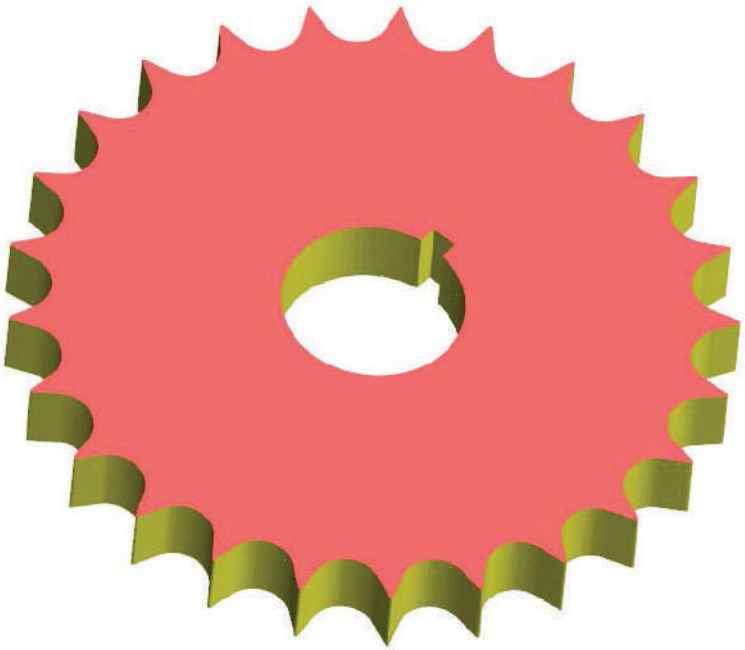
- In which order you have to select the three entities when using Trim 3 entities command?

- When should you use a Contour Wirepath?

- What does the Land height set?

- What does Verify do?

TUTORIAL #3



OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

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, Circle Center Point, Line Polar, Arc Tangent, Line Tangent, Trim, Mirror, Rotate, and Create Thread Point will be used.

Create the necessary Wirepaths to machine the part:

- Contour wirepath to cut the inner profile.
- Contour wirepath to cut the outer profile.

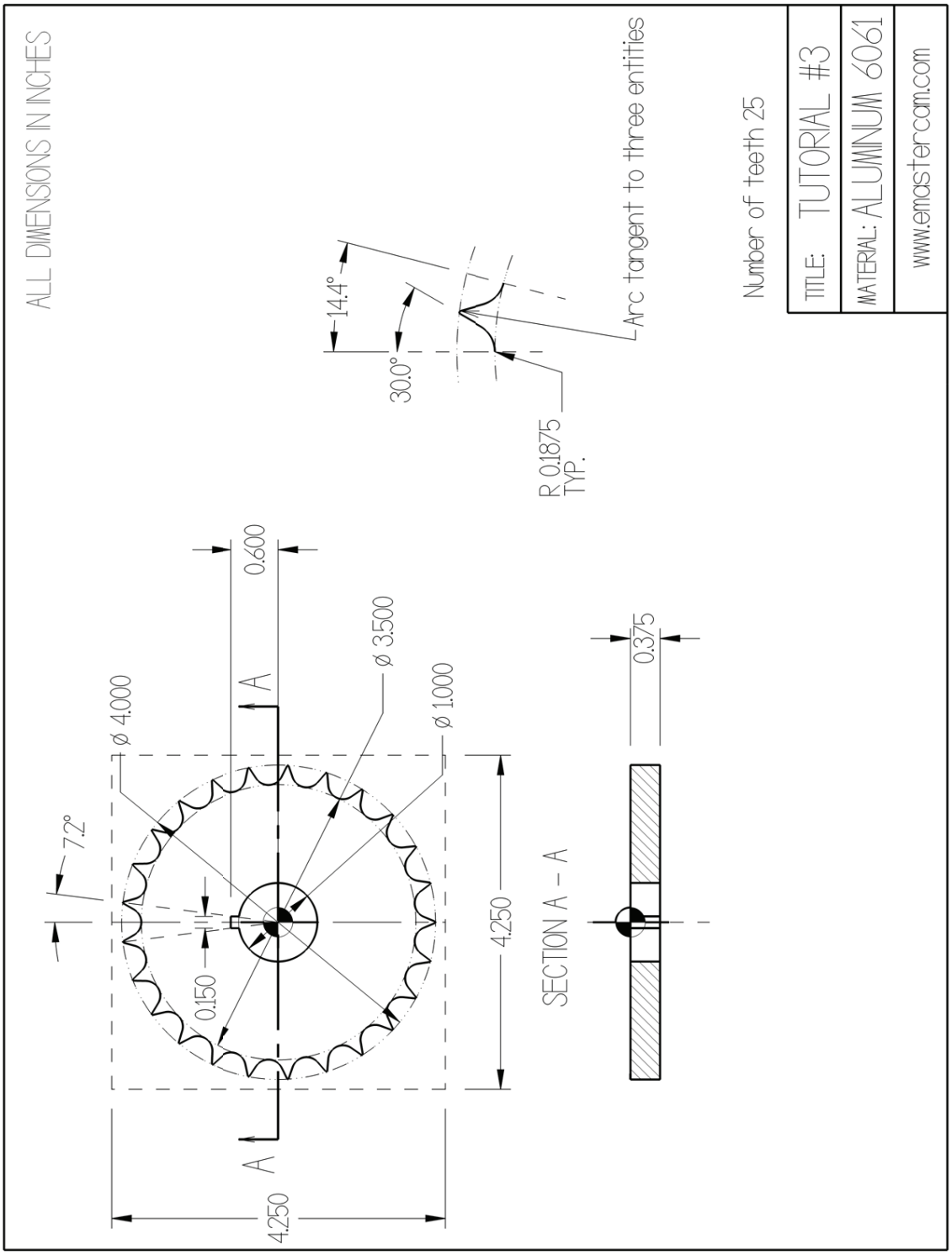
Backplot and the file:

- 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 then post process the file to obtain an NC file containing the necessary code for the machine.

This tutorial takes approximately one hour and 30 minutes to complete.


ALL DIMENSIONS IN INCHES
Arc tangent to three entities
Number of teeth 25
TITLE: TUTORIAL #3
MATERIAL: ALUMINUM 6061
www.emastercam.com
14.4°
30.0°
R 0.1875
TYP.
0.600
ø 3.500
ø 1.000
ø 4.000
A
7.2°
0.150
4.250
4.250
A
SECTION A – A
0.375

GEOMETRY CREATION

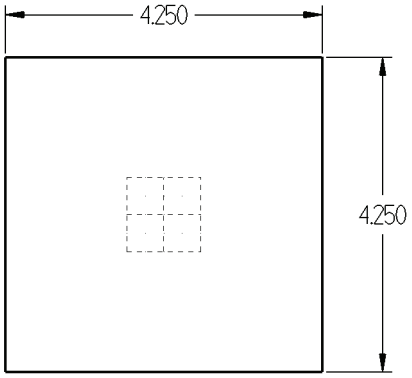
STEP 1: SETTING UP THE GRAPHIC USER INTERFACE

Please refer to the **Getting Started** section to set up the graphical user interface.

STEP 2: CREATE A RECTANGLE

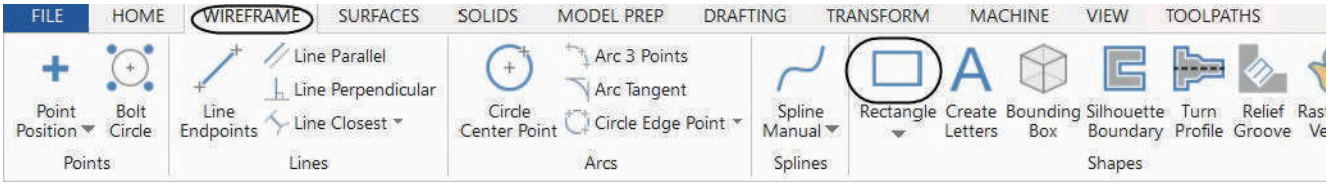
In this step you will learn how to create a rectangle given the width, the height, and the anchor location.

Step Preview:

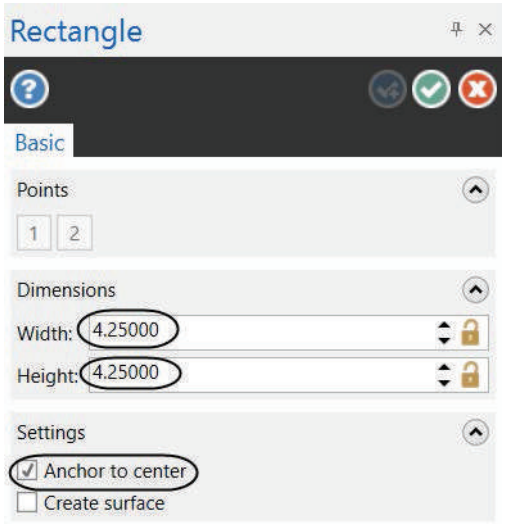


WIREFRAME

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

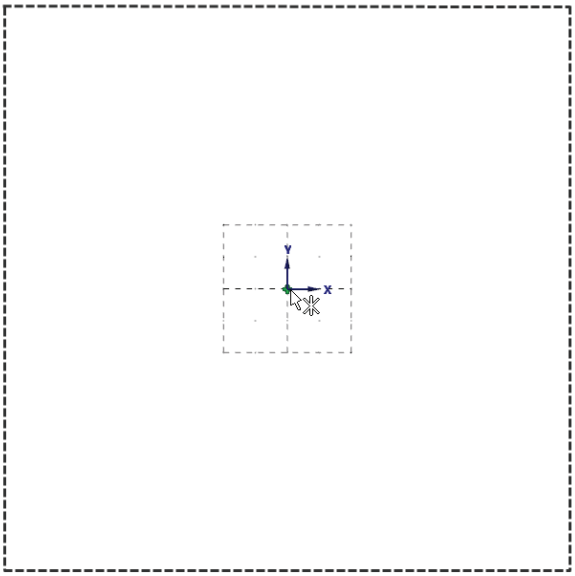


- Enter the **Width 4.25** and **Height 4.25** and enable **Anchor to center** as shown.

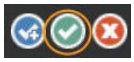


- [Select position of base point]: Select the Origin as shown in Figure: 2.0.1.

Figure: 2.0.1



- Make sure that when selecting the origin, the visual cue of the cursor changes as shown.
- Press **Alt+F1** to fit the drawing to the screen.

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

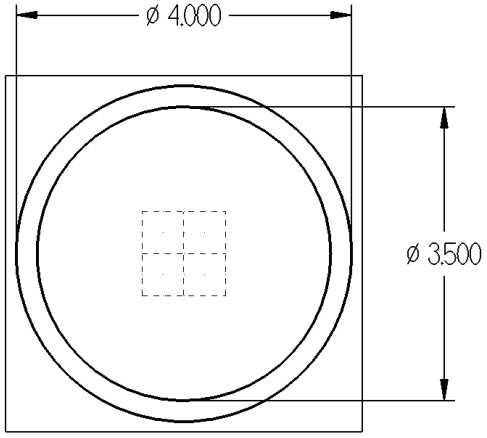
NOTE: While creating geometry for this tutorial, if you make a mistake, you can undo the last step using the **Undo** icon. You can undo as many steps as needed. If you delete or undo a step by mistake, just use the **Redo** icon. To delete unwanted geometry, select the geometry first and then press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

NOTE: In the next few steps, you will create the profile with 25 identical teeth. You will create one tooth and then, using the Rotate command with the copy option enabled, generate the rest of them.

STEP 3: CREATE 2 CIRCLES KNOWING THE DIAMETER AND THE CENTER LOCATION

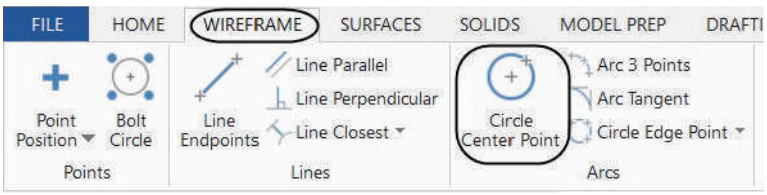
In this step you will create the circles with diameters of **4.0"** and **3.5"** with their center locations at the origin.

Step Preview:

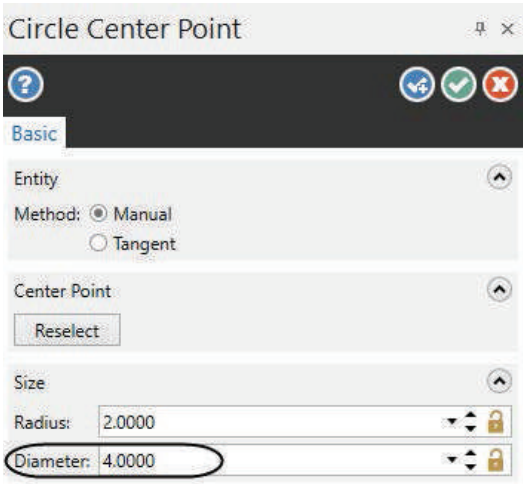


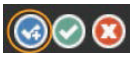
WIREFRAME

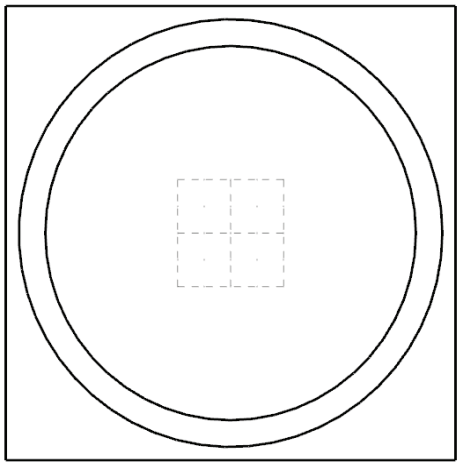
- From the **Arcs** group, select the **Circle Center Point** icon.



- Enter the **Diameter 4.0** as shown (press **Enter** to see a preview of the circle).



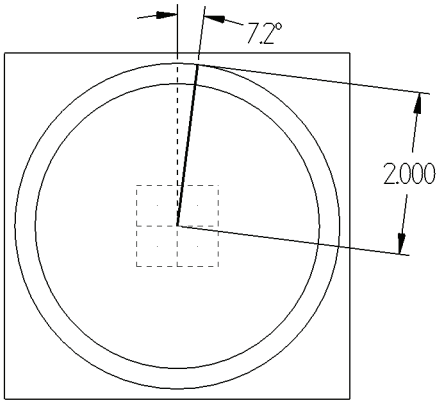
- [Enter the center point]: Select the Origin as shown previously in Figure: 2.0.1.
- Select **OK and Create New Operation** to continue with the same command. 
- Enter the **Diameter 3.5**.
- [Enter the center point]: Select the Origin.
- Select the **OK** button to exit the command.
- The geometry should look as shown.



STEP 4: CREATE A POLAR LINE

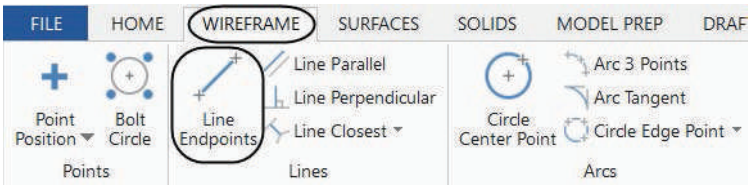
Using **Line Endpoint** command, you will learn how to create a line given the start point, the length, and the angle.

Step Preview:

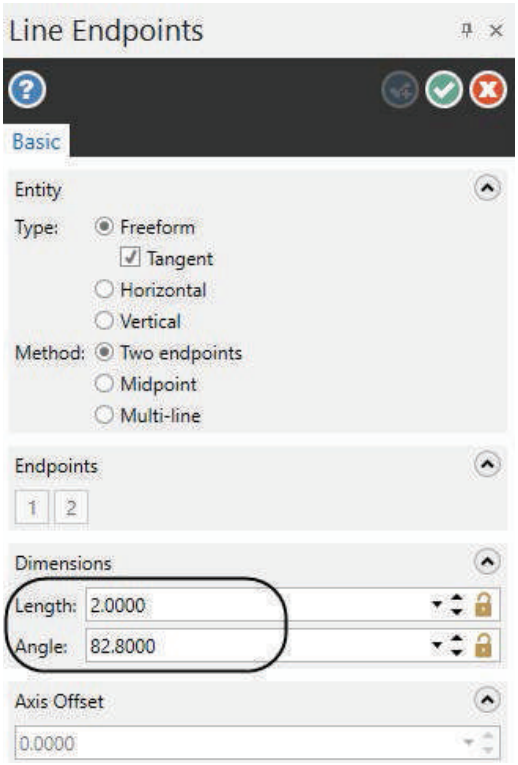


WIREFRAME

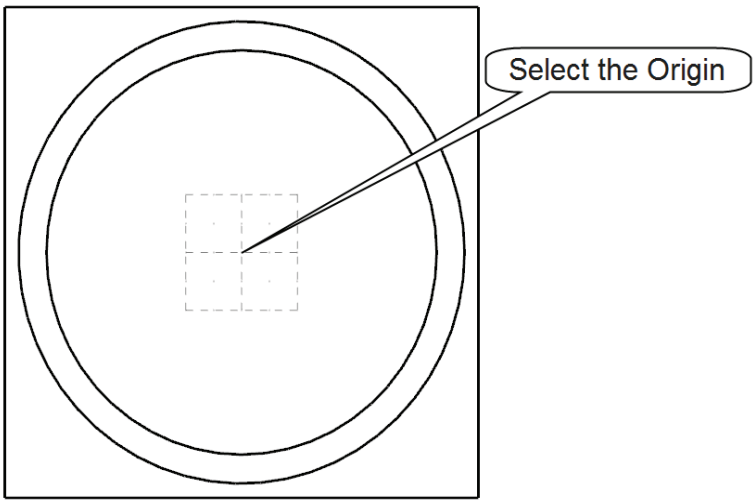
- From the **Lines** group, select the **Line Endpoint** icon.



- Enter the **Length 2.0** and the **Angle 82.8** as shown.



- [Specify the first endpoint]: Select the Origin.

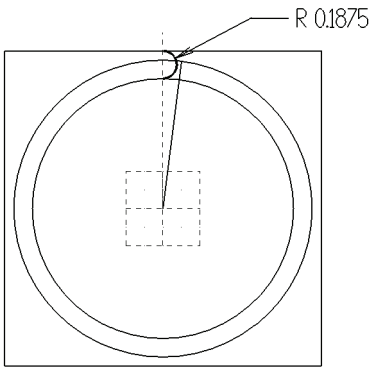


- Select the **OK** button to exit the command.

STEP 5: CREATE AN ARC TANGENT TO ONE ENTITY

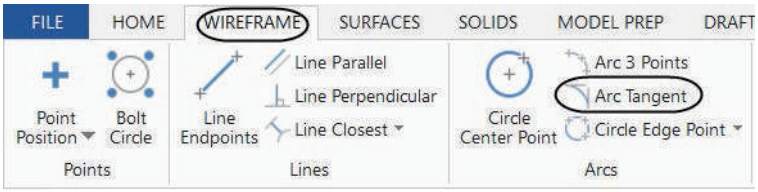
This step shows you how to create an arc tangent to one entity given the arc radius and the tangent point.

Step Preview:

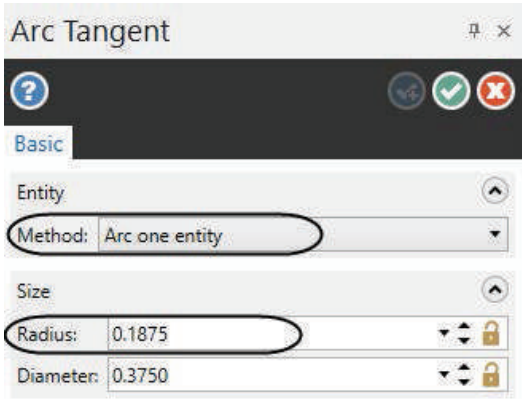


WIREFRAME

- From the **Arcs** group, select the **Arc Tangent** icon.

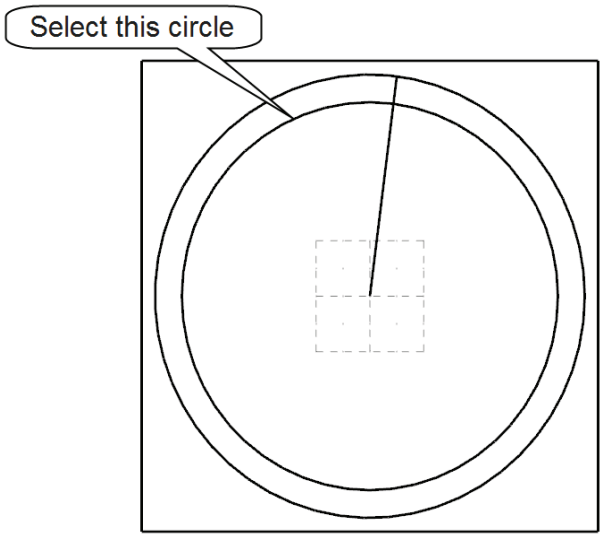


- Make sure that the **Method** is set to **Arc one entity** as shown. Then enter the **Radius 0.1875**.



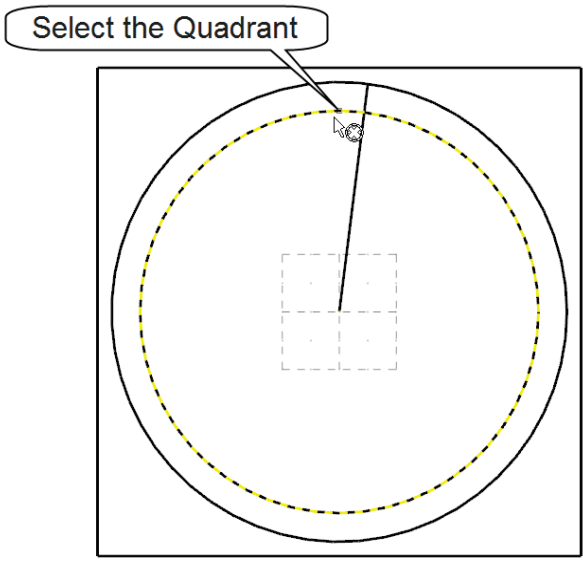
- [Select the entity that the arc is to be tangent to]: Select the **3.5"** diameter circle as shown in Figure: 5.0.1.

Figure: 5.0.1



- [Specify the tangent point]: Select the **Quadrant** of the circle as shown in Figure: 5.0.2.

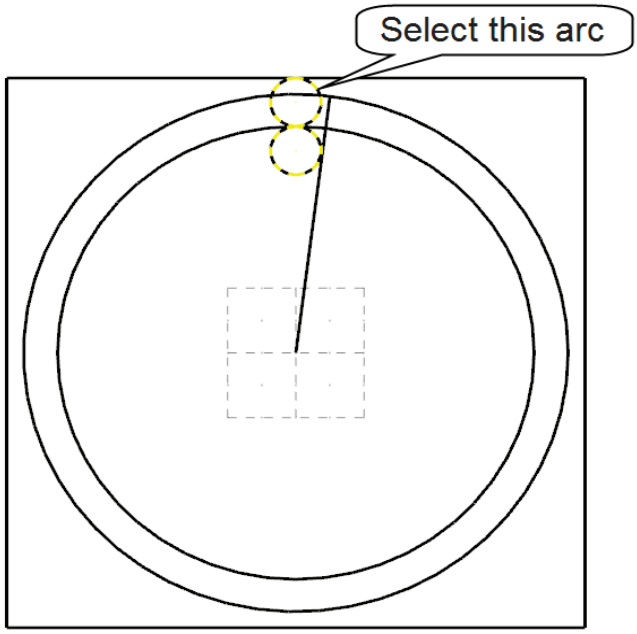
Figure: 5.0.2



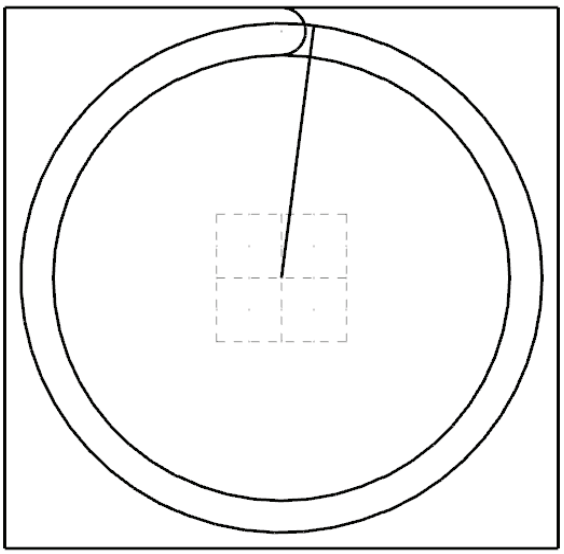
NOTE: Mastercam automatically creates four tangent arcs. You are prompted to select the arc that you want to keep.

- [Select an arc]: Select the arc as shown in Figure: 5.0.3.

Figure: 5.0.3



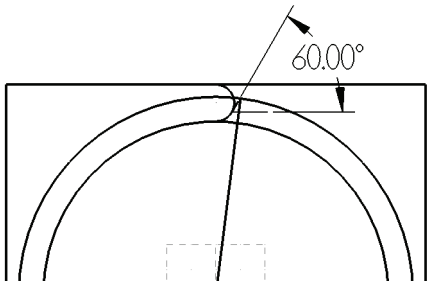
- Select the **OK** button to exit the command.
- The geometry should look as shown.



STEP 6: CREATE A LINE TANGENT AT AN ANGLE

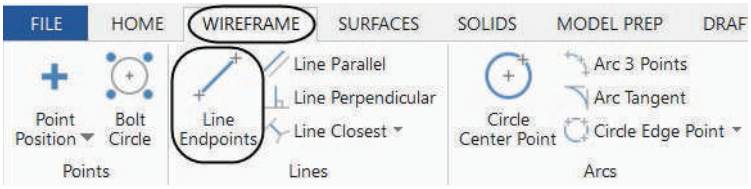
In this step you will create a line tangent to the 0.1875" arc that has a 60.0 degree angle.

Step Preview:

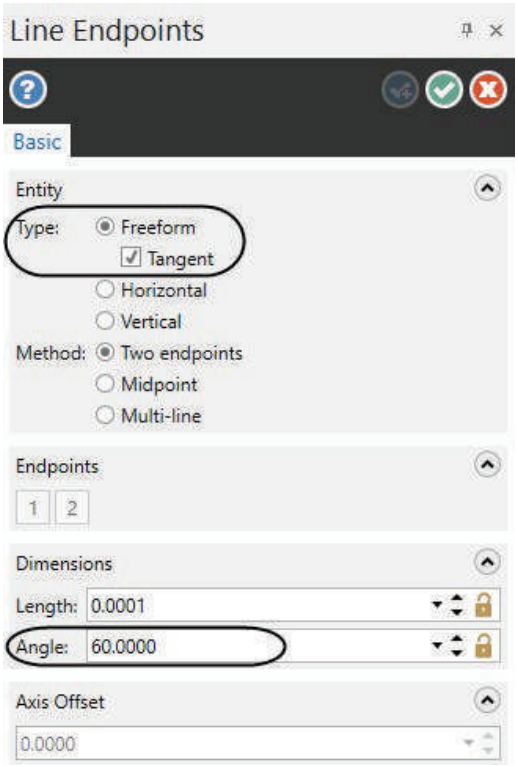


WIREFRAME

- From the **Lines** group, select the **Line Endpoints** icon.



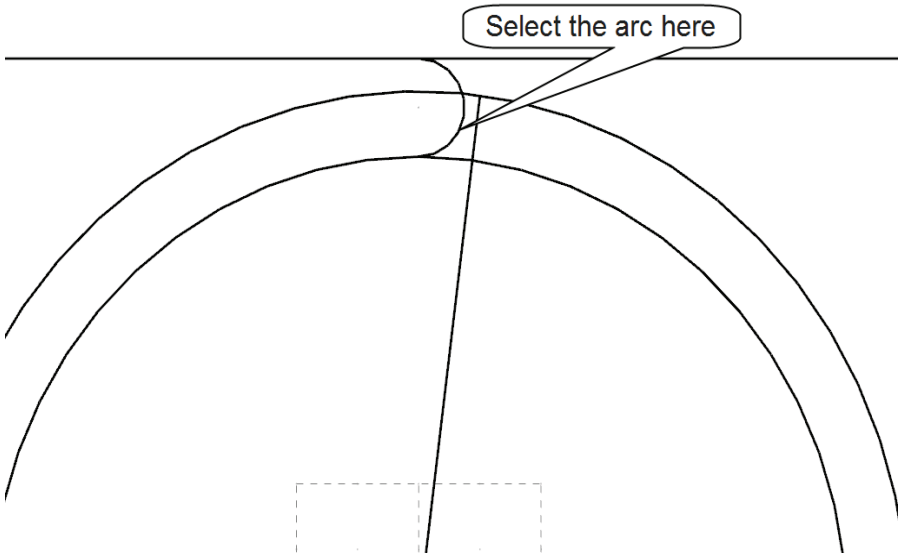
- Enter the **Angle 60.0**, maker sure that **Freeform** is selected and **Tangent** is enabled as shown.



NOTE: You need to select the arc, and make sure that you do not select the midpoint or the endpoints of the arc. In order to select the arc instead of other entities, you can hover the cursor above the center of the arc and scroll up the mouse wheel to Zoom in.

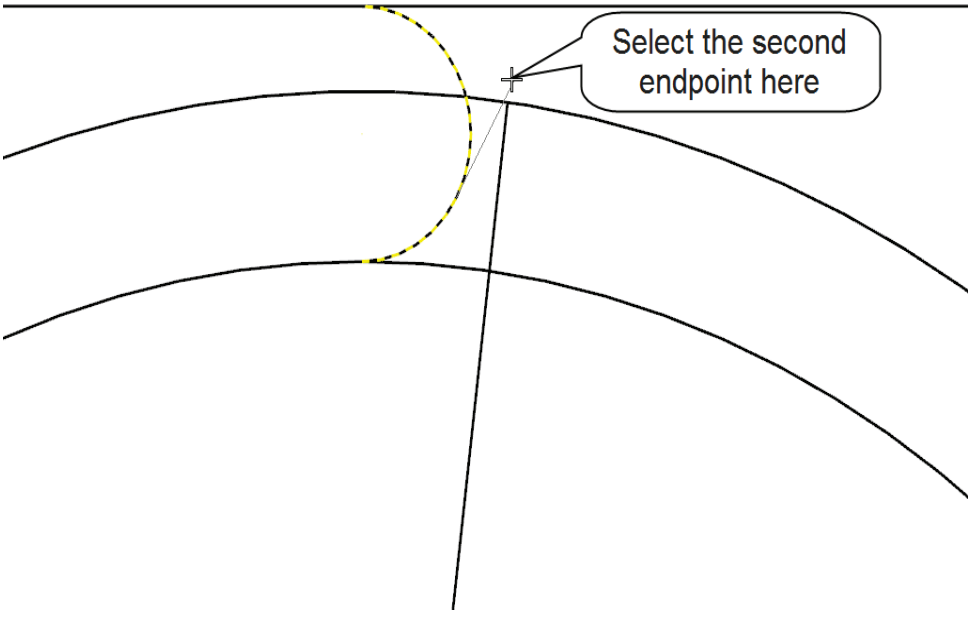
- [Specify the first endpoint]: Select the arc as shown in Figure: 6.0.1.

Figure: 6.0.1



- [Specify the second endpoint]: Select the second endpoint of the line above the 4.0" diameter circle as shown in Figure: 6.0.2.

Figure: 6.0.2

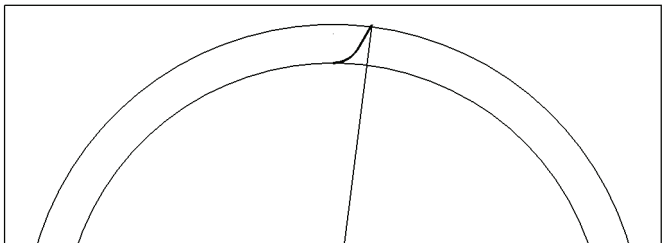


- Select the **OK** button to exit the command.

STEP 7: TRIM TWO ENTITIES

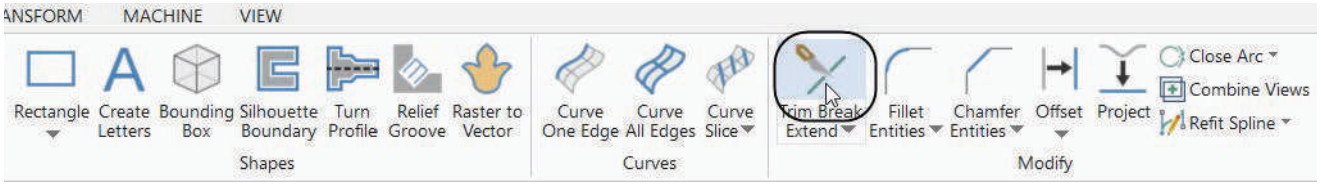
Clean up the geometry by trimming the entities to their intersection point.

Step Preview:

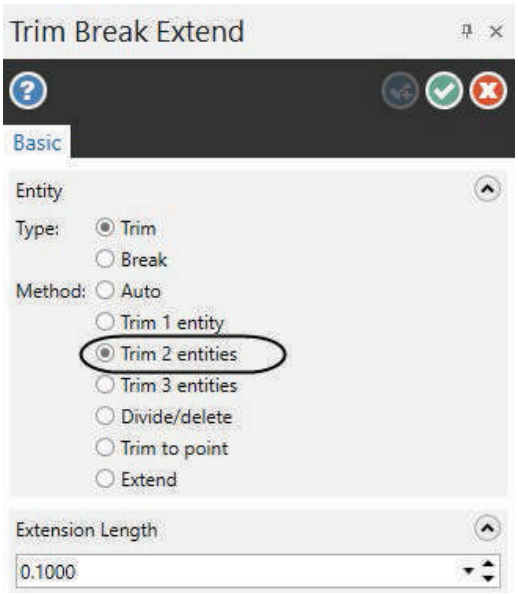


WIREFRAME

- From the **Modify** group, select the **Trim Break Extend** icon.

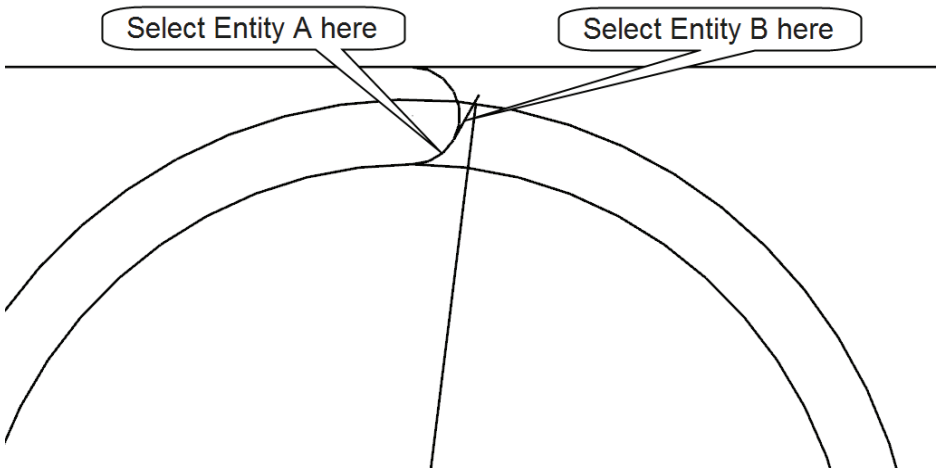


- Select **Trim 2 entities** as shown.



- [Select the entity to trim/extend]: Click on Entity A as shown in Figure: 7.0.1.
- [Select the entity to trim/extend to]: Click on Entity B as shown in the same figure Figure: 7.0.1.

Figure: 7.0.1



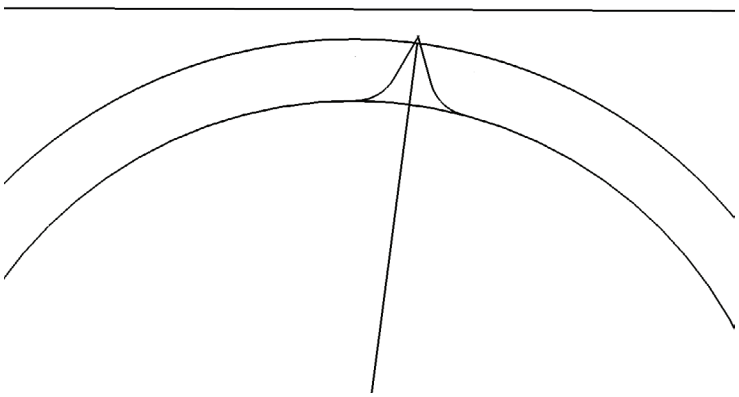
NOTE: Mastercam's auto-preview feature displays the result of the selected function as you move the mouse to select the last entity. A solid line represents what will be created, while a dashed line represents what will be removed.

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

STEP 8: MIRROR THE 0.1875" RADIUS ARC AND THE TANGENT LINE ABOUT A LINE

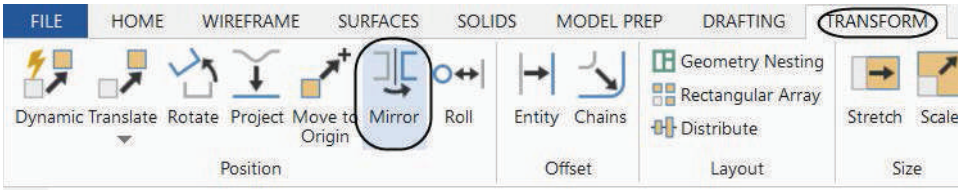
The **Mirror** command allows you to create mirror images of geometric and drafting entities by reflecting them symmetrically with respect to a defined axis or point.

Step Preview:



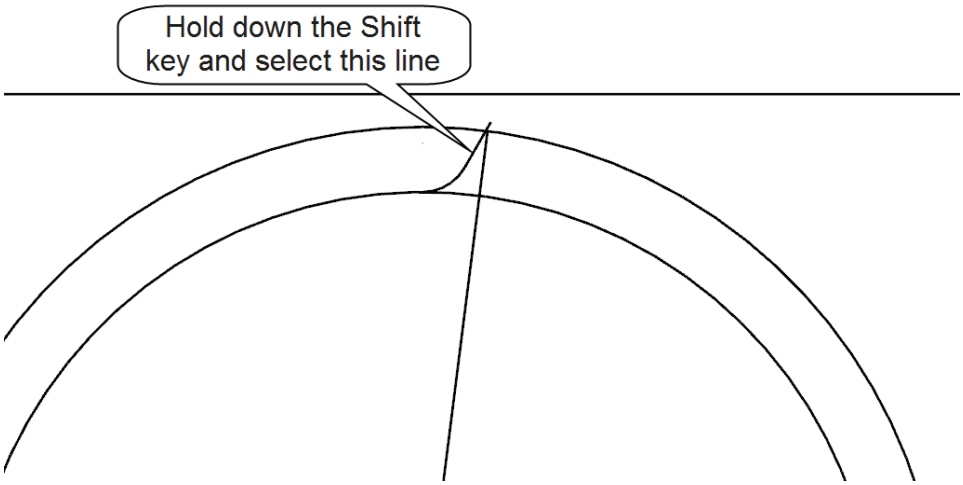
TRANSFORM

- From the **Position** group, select the **Mirror** icon.



- [Mirror: select entities to mirror]: Hold down the **Shift** key and select the line as shown in Figure: 8.0.1.

Figure: 8.0.1

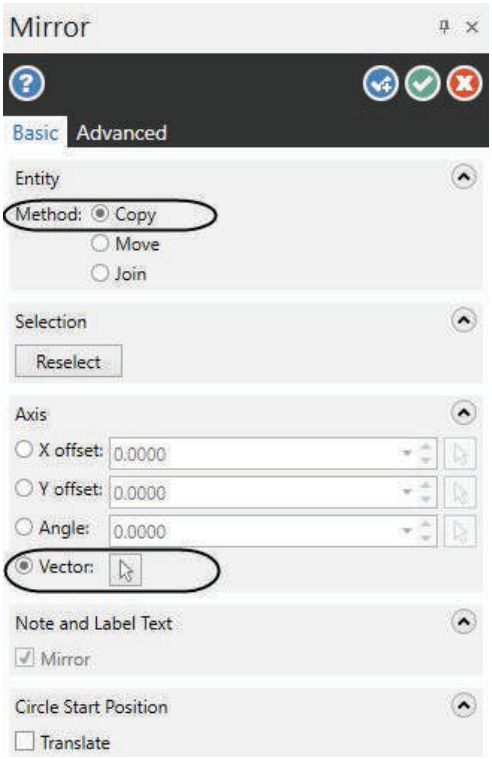


NOTE: Both the line and the arc are highlighted (selected). By holding down the **Shift** key a chain of entities are selected.

- Press **Enter** to continue.

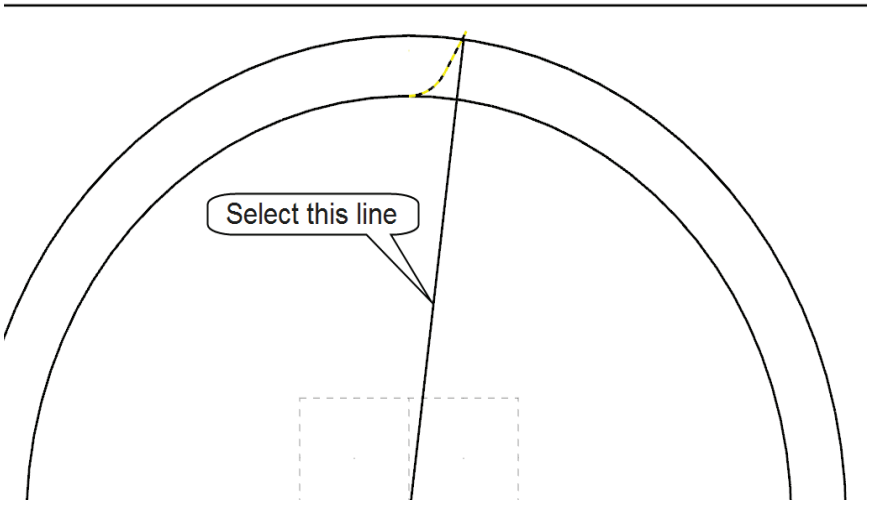
- In the **Mirror** panel, make sure that **Copy** is selected and then enable **Vector** option as shown in Figure: 8.0.2.

Figure: 8.0.2

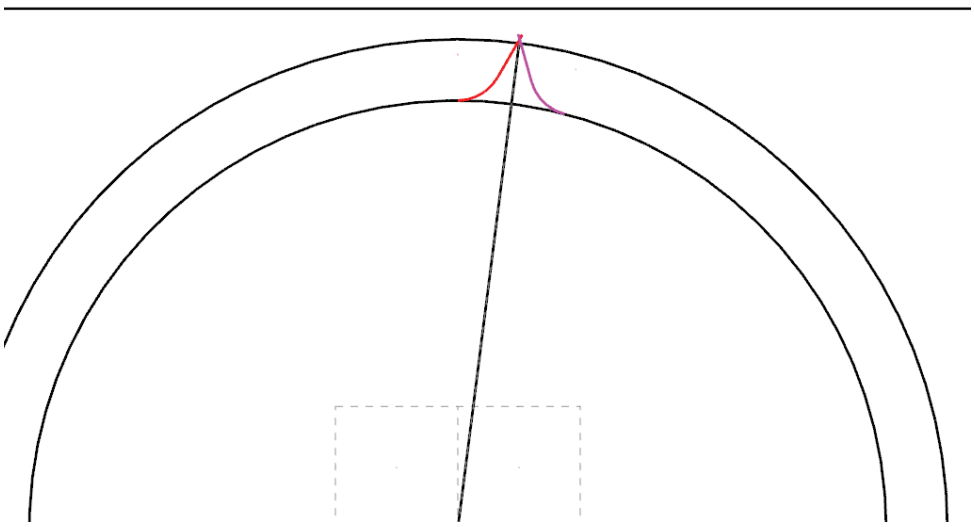


NOTE: The **Vector** option will bring you back into the graphics window where you can select the line about which to mirror.

- [Select line about which to mirror]: Select the line as shown.



- The preview should look as shown.

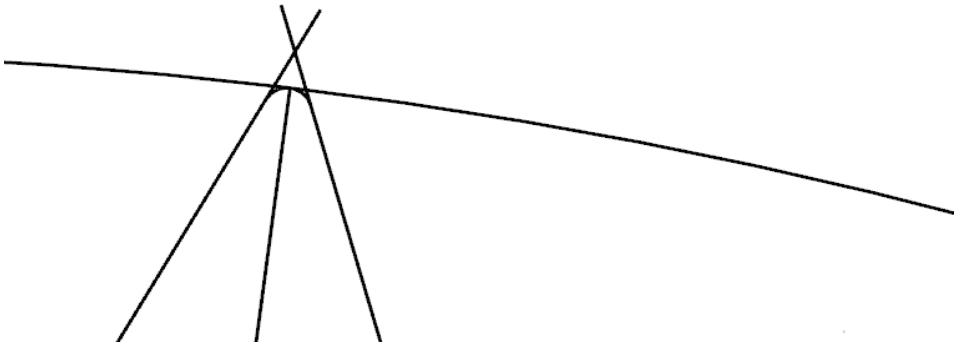


- Select the **OK** button to exit the **Mirror** dialog box.
- Right mouse click in the graphics window and select **Clear Colors** as shown.

STEP 9: CREATE AN ARC TANGENT TO THREE ENTITIES

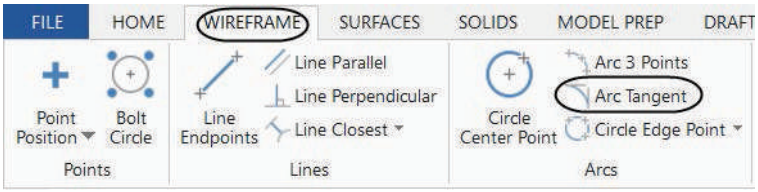
This step shows you how to create an arc tangent to three entities which you select from the graphics window.

Step Preview:

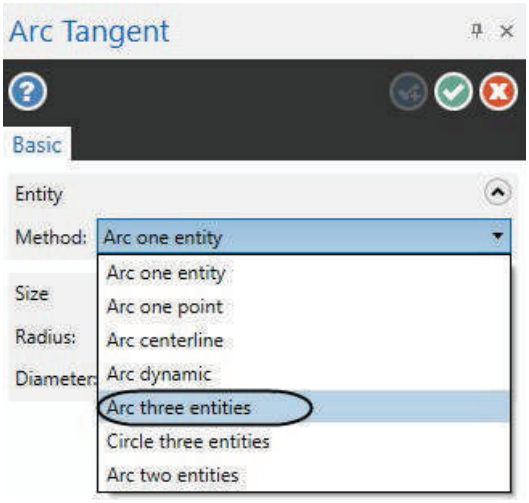


WIREFRAME

- From the **Arcs** group, select the **Arc Tangent** icon.



- Select **Arc three entities** as shown.

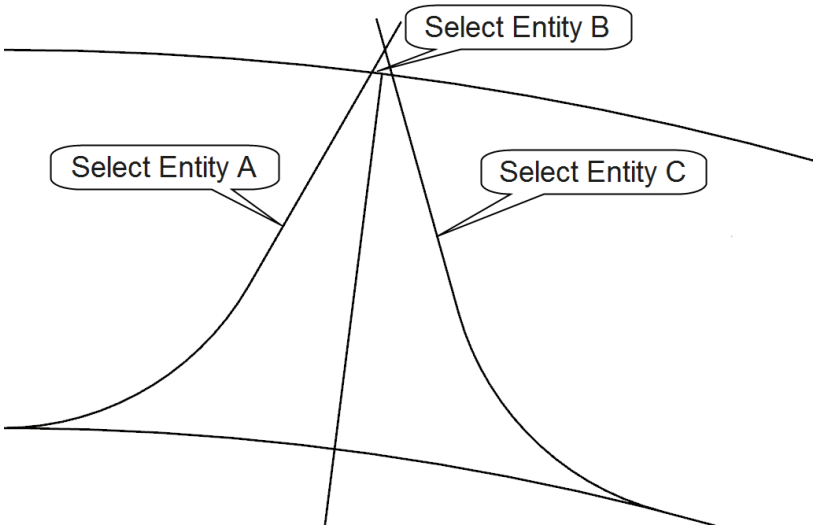


NOTE: The arc radius will be automatically calculated by Mastercam.

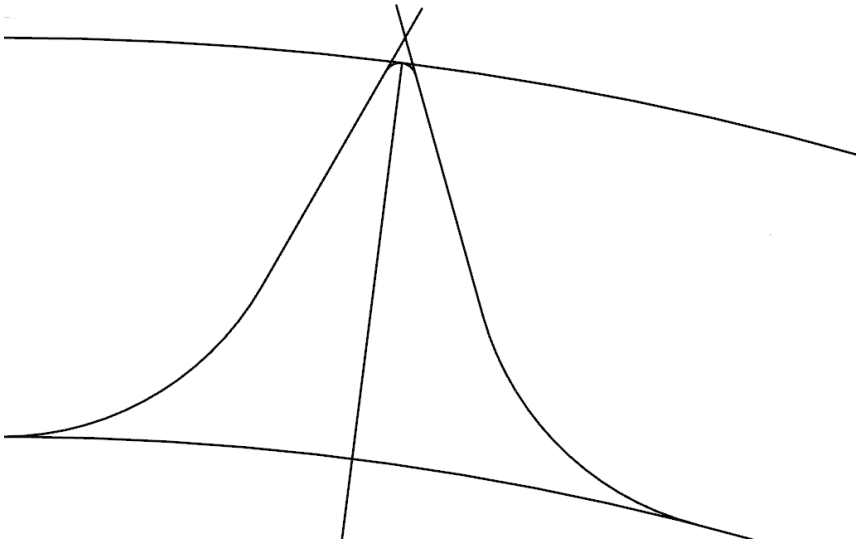
- Move the cursor to the tip of the tooth, and scroll up the mouse wheel to zoom in.

- [Select the entity that the arc is to be tangent to]: Select entity A as shown in Figure: 9.0.1.
- [Select the entity that the arc is to be tangent to]: Select entity B as shown in Figure: 9.0.1.
- [Select the entity that the arc is to be tangent to]: Select entity C as shown in Figure: 9.0.1.

Figure: 9.0.1



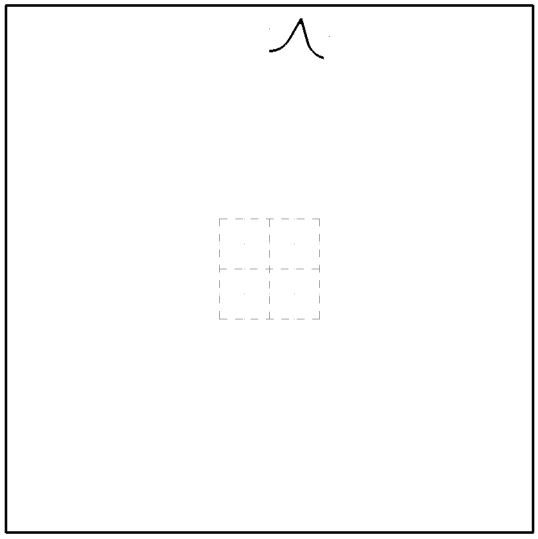
- The geometry should look as shown.



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

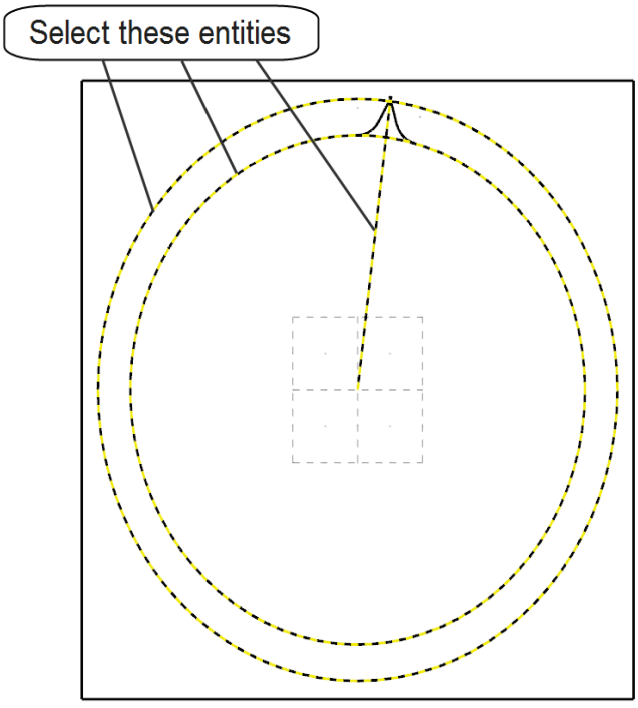
STEP 10: DELETE THE CONSTRUCTION CIRCLES AND THE LINE

Step Preview:



- Press **Alt+F1** to fit the drawing to the screen.
- Select the two circles and the line as shown in Figure: 10.0.1.

Figure: 10.0.1

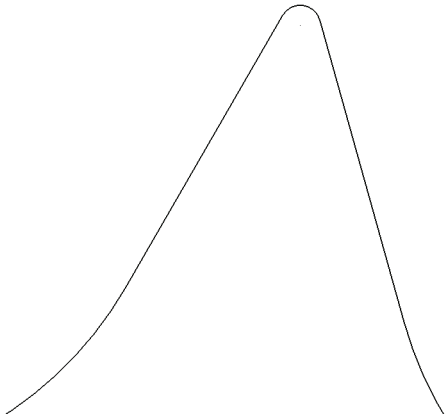


- Press **Delete** on the keyboard to delete the selected entities.

STEP 11: TRIM THREE ENTITIES

Clean the tip of the tooth using the trim three entities command.

Step Preview:



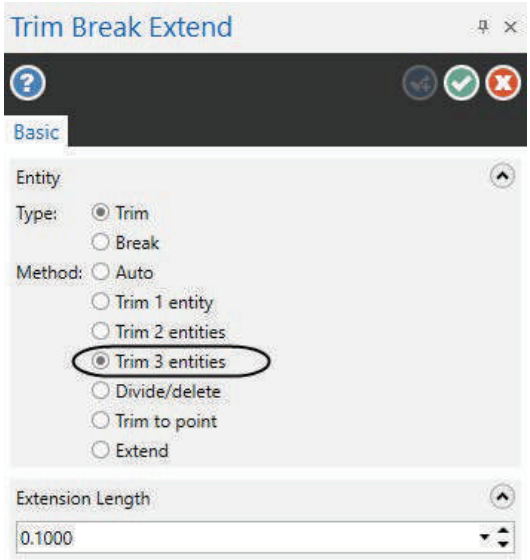
- Hover the cursor again above the tip of the tooth and scroll up the mouse wheel to zoom in.

NOTE: During the trimming process, it is very important to select the entities exactly at the locations that are shown in the graphics in the next few pages.

WIREFRAME

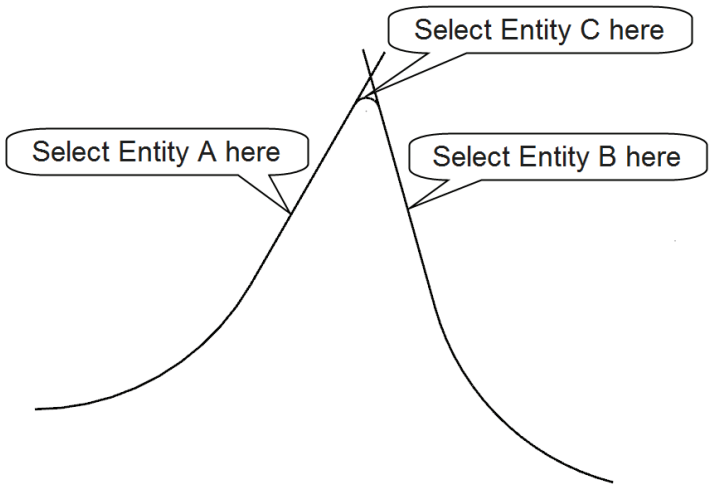
- From the **Modify** group, select the **Trim Break Extend** icon.

- Select **Trim 3 entities** as shown.



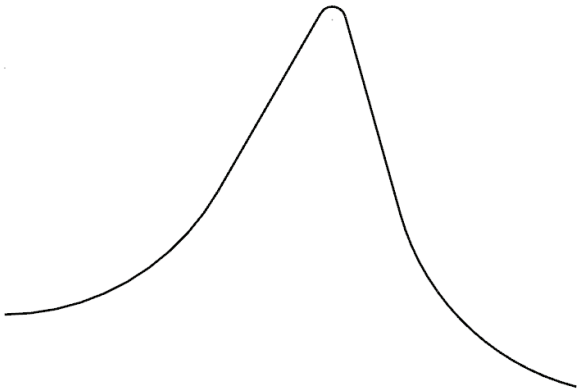
- Zoom in for a better view of the area you need to trim.
- [Select the first entity to trim/extend]: Click on Entity A as shown in Figure: 11.0.1.
- [Select the second entity to trim/extend]: Click on Entity B as shown in Figure: 11.0.1.
- [Select the second entity to trim/extend to]: Click on Entity C as shown in Figure: 11.0.1.

Figure: 11.0.1



NOTE: Mastercam's auto-preview feature displays the result of the selected function as you move the mouse to select the last entity. A solid line represents what will be created, while a dashed line represents what will be removed. If the other side of the arc is created, undo the command and use trim 2 entities twice to clean up the geometry.

- The geometry should look as shown.

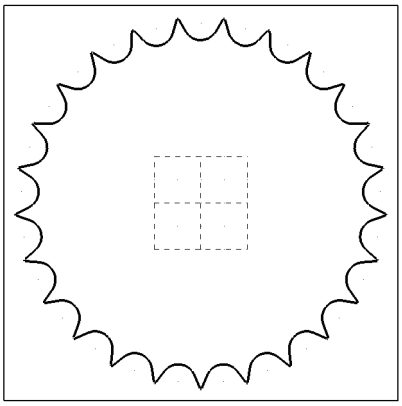


- Select the **OK** button to exit the command.

STEP 12: ROTATE/COPY THE TOOTH

In this step you will learn how to create the rest of the shapes using **Transform Rotate** command with the copy option enabled. The tooth will be copied 24 times and rotated around the origin.

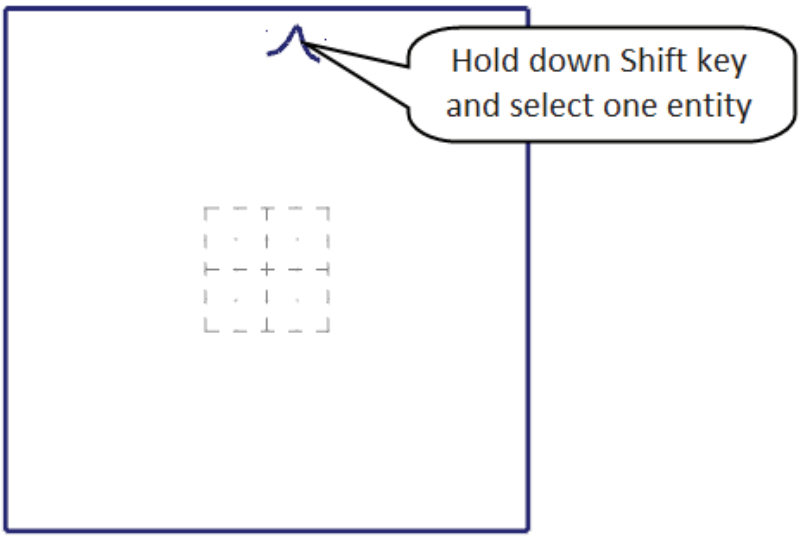
Step Preview:



- Press **Alt+F1** to fit the drawing to the screen.

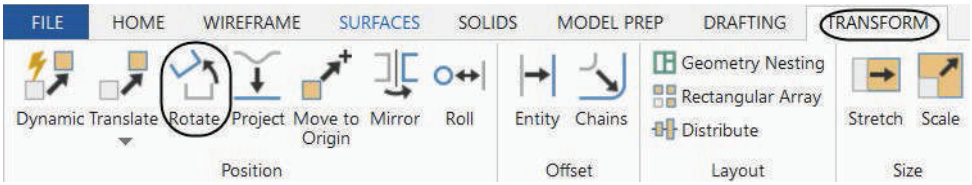
- To select all the entities of the tooth, hold down the **Shift** key and select one entity as shown in Figure: 12.0.1.

Figure: 12.0.1



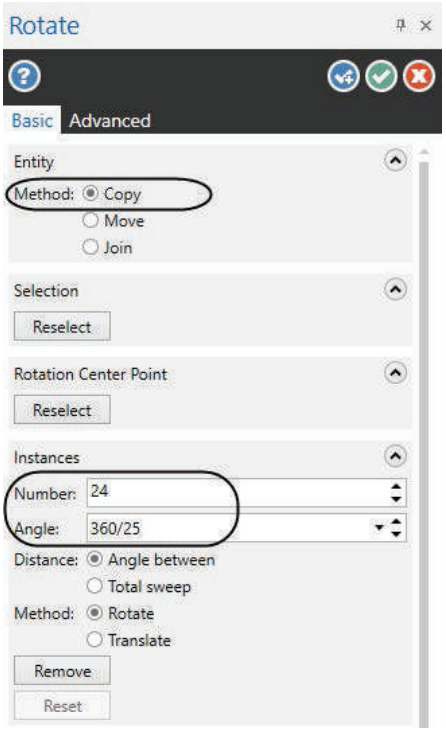
TRANSFROM

- In the **Position** group, select the **Rotate** icon.

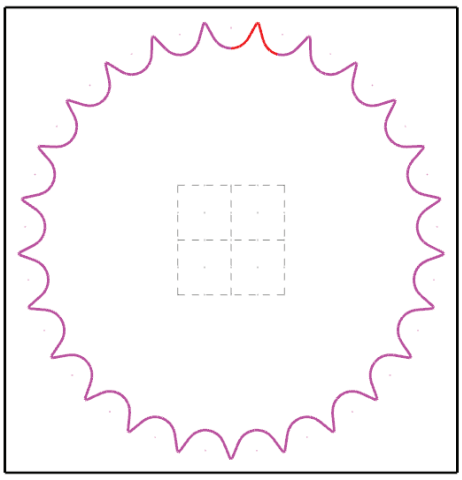


- Make sure that **Copy** is enabled in the **Rotate** panel and set the number of Instances to **24** and the **Angle** to **360/25** as shown in Figure: 12.0.2.

Figure: 12.0.2.



- Press **Enter** and the preview of the generated geometry should look as shown.



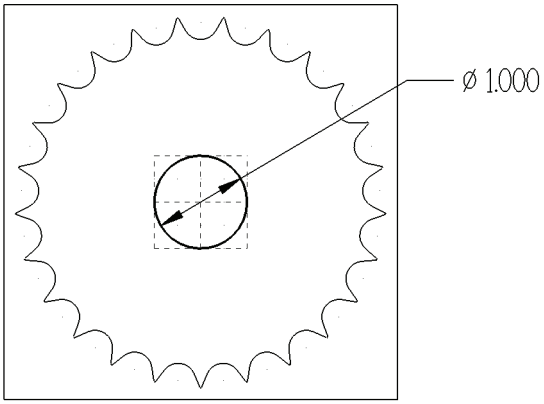
- Select the **OK** button to exit the **Rotate** dialog box.

NOTE: When performing a transform function, Mastercam creates a temporary group from the original, highlighted in red, and a result from the transformed entities, highlighted in magenta. These two colors will remain until you use the **Screen**, **Clear Colors** function or perform another transform function.

- Right mouse click in the graphics window and select **Clear Colors** as shown.

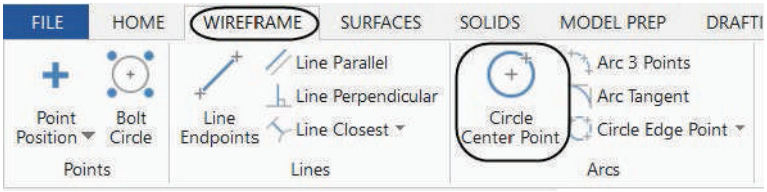
STEP 13: CREATE THE 1.0" DIAMETER CIRCLE

Step Preview:

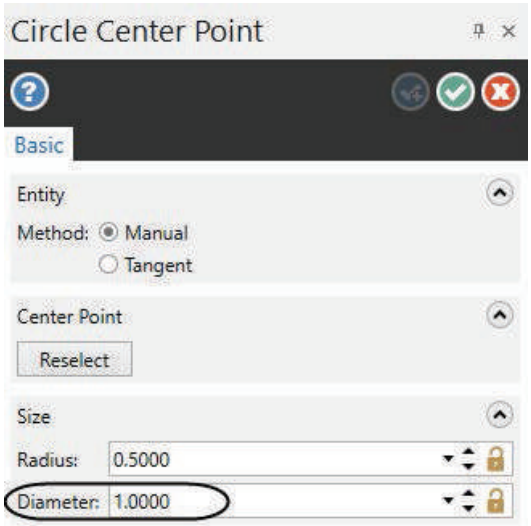


WIREFRAME

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



- Enter the Diameter **1.0**

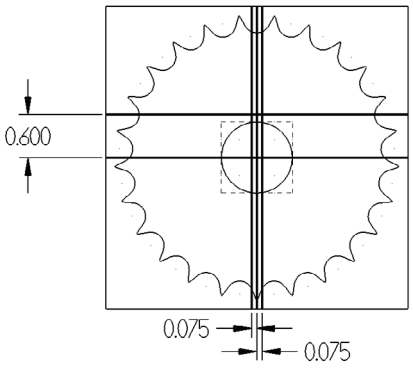


- [Enter the center point]: Select the Origin as shown previously.
- Select the **OK** button to exit the command.

STEP 14: CREATE PARALLEL LINES

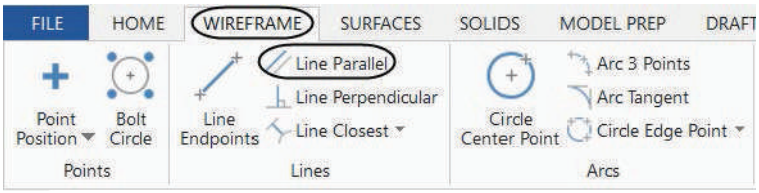
In this step you will create parallel lines that will be used to finish the keyway.

Step Preview:



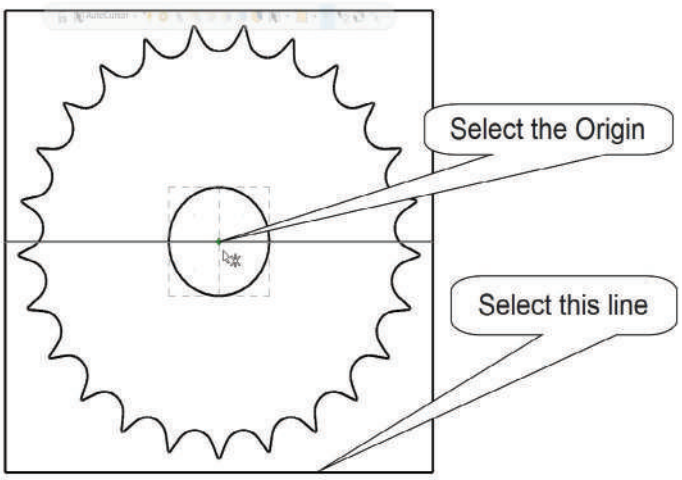
WIREFRAME

- From the **Lines** group, select the **Line Parallel** icon.



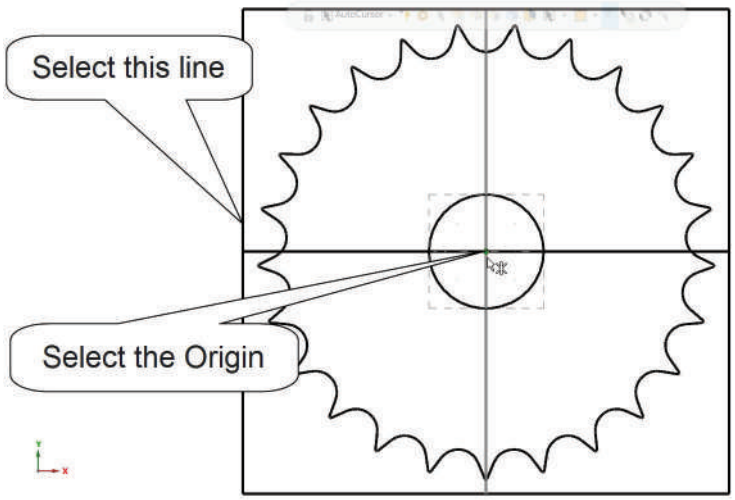
- [Select a line]: Select the lower horizontal line of the rectangle as shown in Figure: 14.0.1.
- [Select the point to place a parallel line through]: Select the Origin.

Figure: 14.0.1



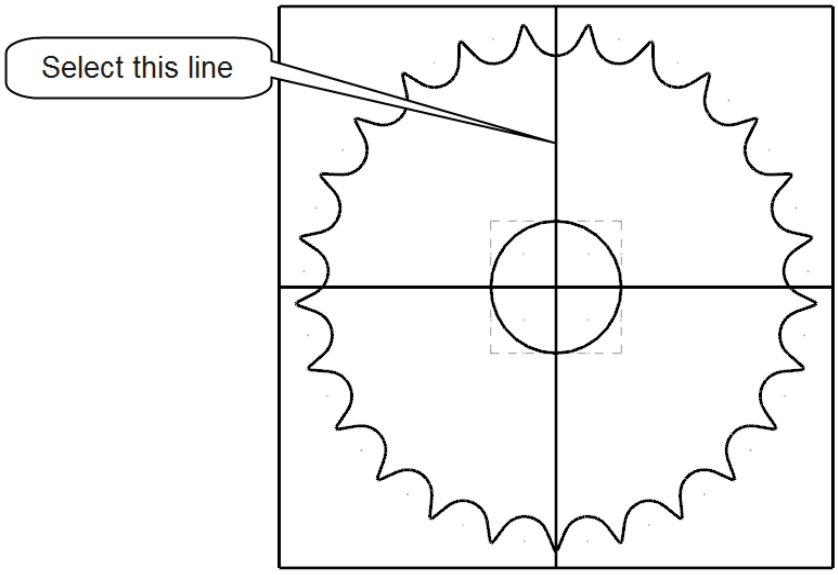
- Select **OK and Create New Operation** to continue with the same command.
- [Select a line]: Select the left side vertical line of the rectangle as shown in Figure: 14.0.2.

Figure: 14.0.2



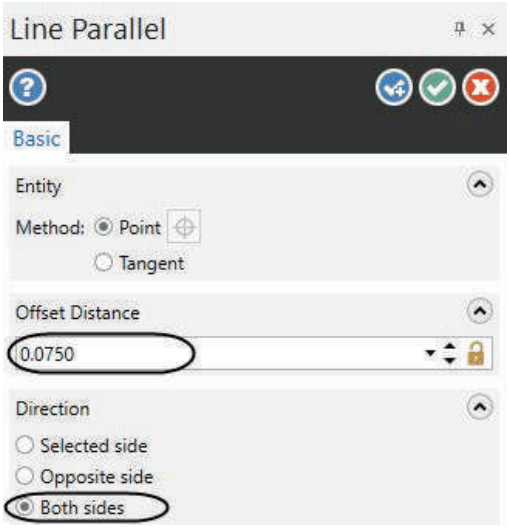
- [Select the point to place a parallel line through]: Select the Origin as shown above.
- Select **OK and Create New Operation** to continue with the same command.
- [Select a line]: Select the last vertical line that you created as shown in Figure: 14.0.3.

Figure: 14.0.3



- [Select the point to place a parallel line through]: Select a point to the right of the selected line.

- Enter the **Distance 0.075** and select **Both sides** as shown.



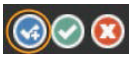
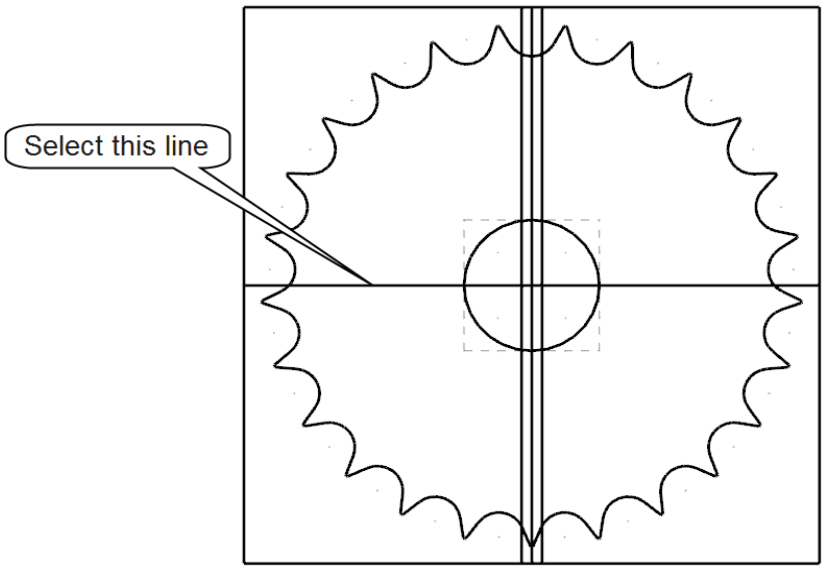
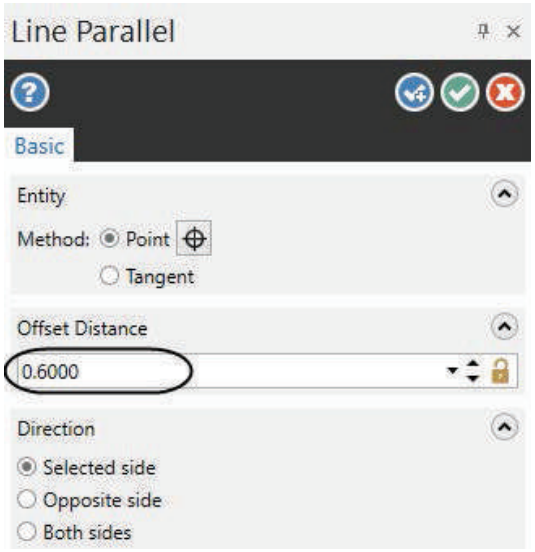
- Select **OK and Create New Operation** to continue with the same command. 
- [Select a line]: Select the horizontal line created through the Origin as shown in Figure: 14.0.4.

Figure: 14.0.4



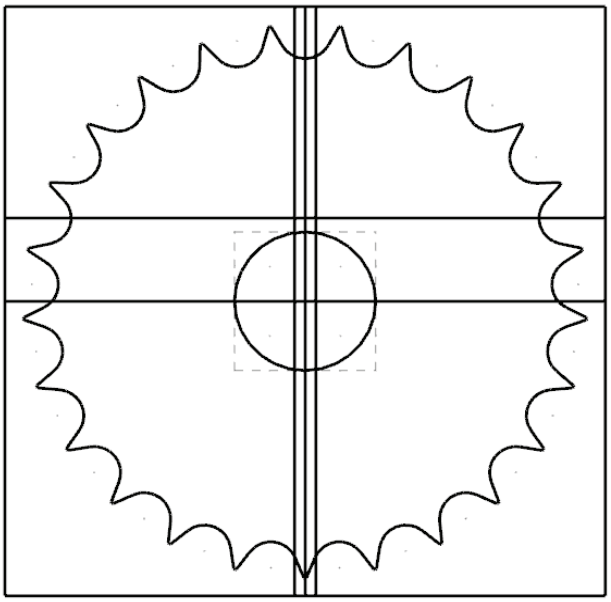
- [Select the point to place a parallel line through]: Select a point above the selected line.

- Enter the **Distance 0.6** and enable **Selected side** as shown.



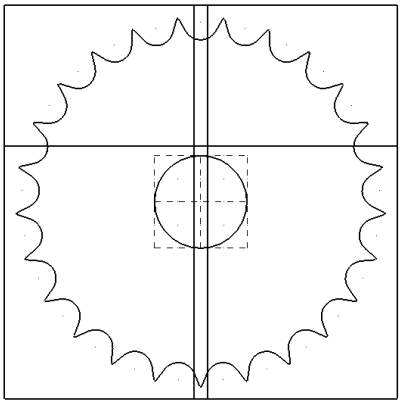
- Select the **OK** button to exit the command.

- The drawing should look as shown.



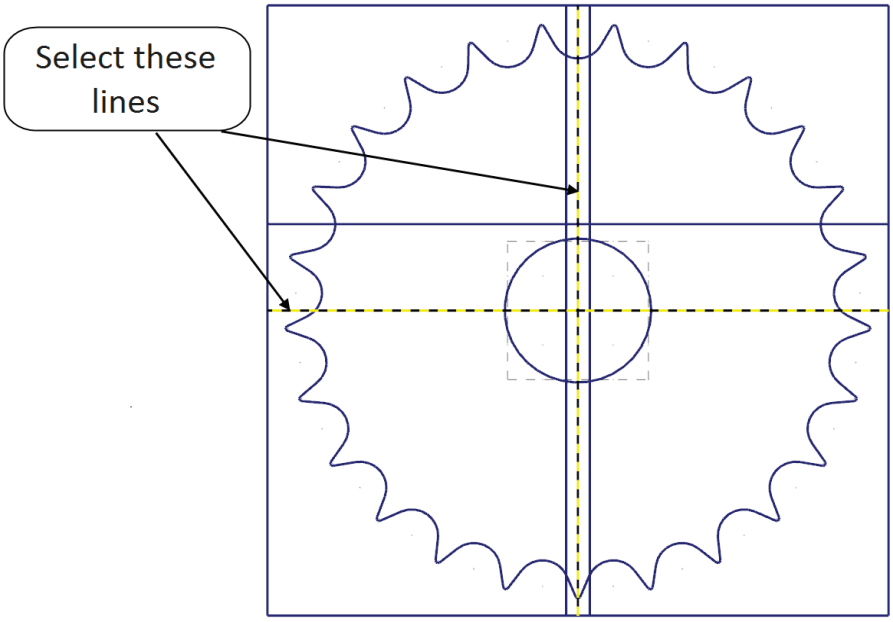
STEP 15: DELETE THE CONSTRUCTION LINES

Step Preview:



- Select the lines as shown in Figure: 15.0.1.

Figure: 15.0.1

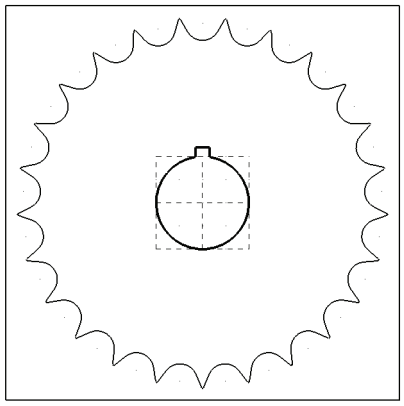


- Press the **Delete** button.

STEP 16: USE TRIMMING COMMANDS TO CLEAN THE KEYWAY

Clean the keyway using the trim three entities and divide commands. Divide command deletes entities based on the nearest intersection.

Step Preview:



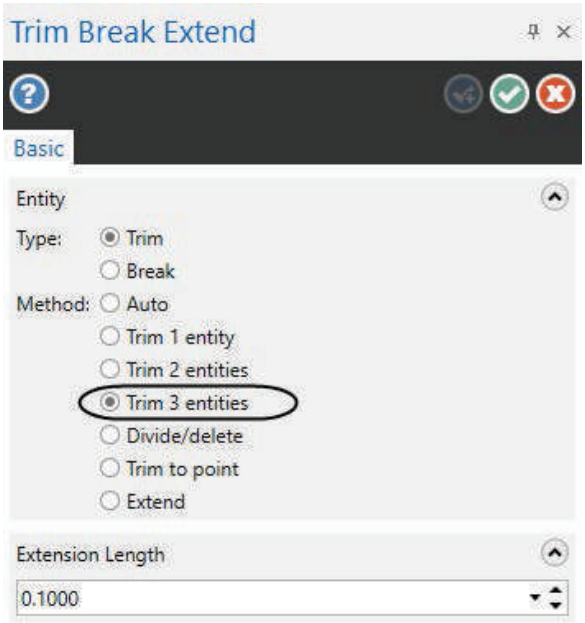
- Hover the cursor again above the key hole and scroll up the mouse wheel to zoom in.

NOTE: During the trimming process it is very important to select the entities exactly at the locations that are shown in the graphics in the next few pages.

WIREFRAME

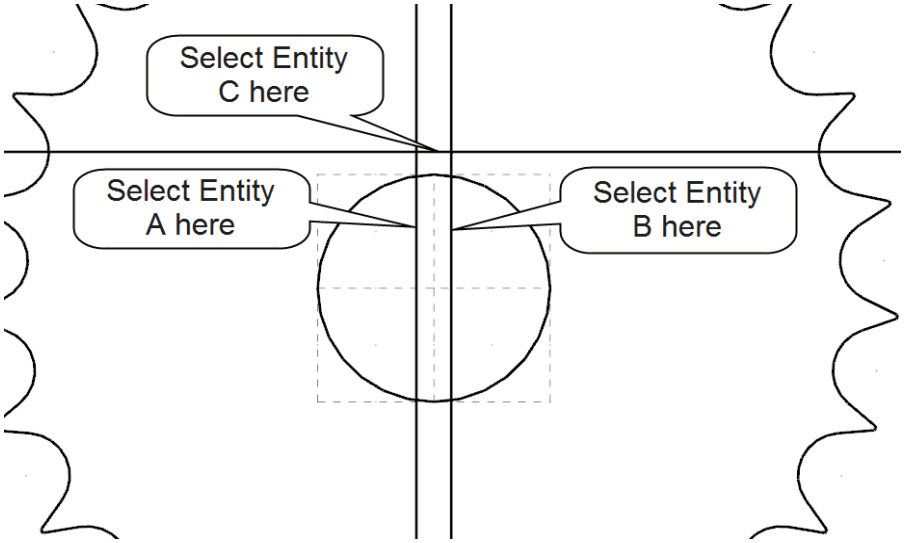
- From the **Modify** group, select the **Trim Break Extend** icon.

- Select the **Trim 3 entities** as shown.

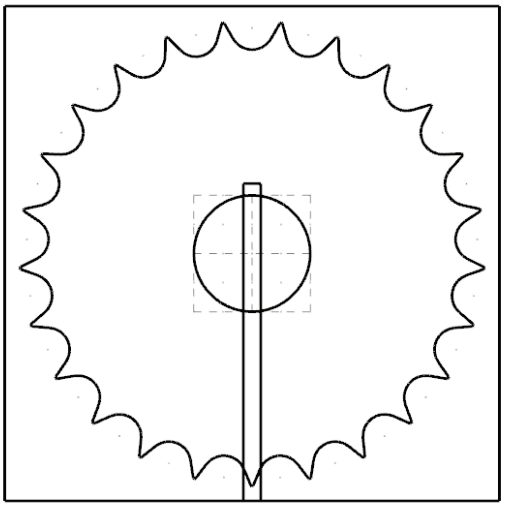


- [Select the first entity to trim/extend]: Click on Entity A as shown in Figure: 16.0.1.
- [Select the second entity to trim/extend]: Click on Entity B as shown in Figure: 16.0.1.
- [Select the entity to trim/extend to]: Click on Entity C as shown in Figure: 16.0.1.

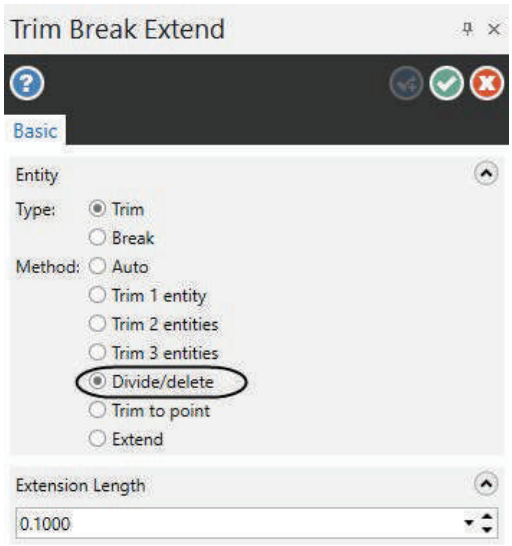
Figure: 16.0.1



- The geometry should look as shown.

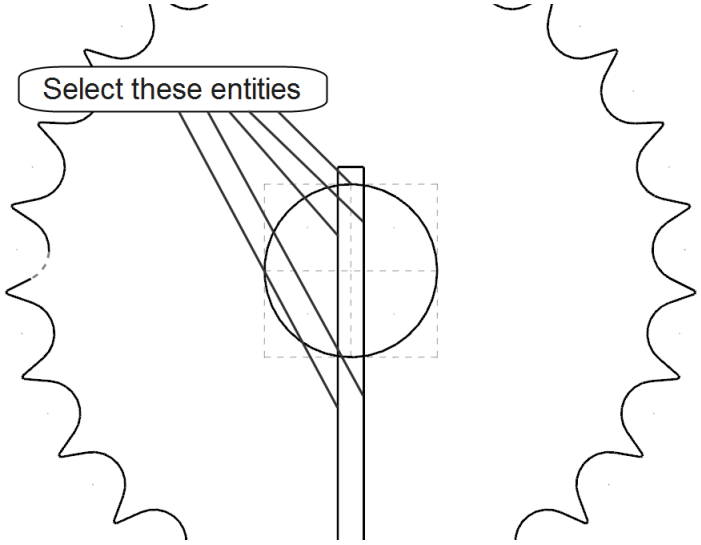


- Select **Divide/Delete** as shown.



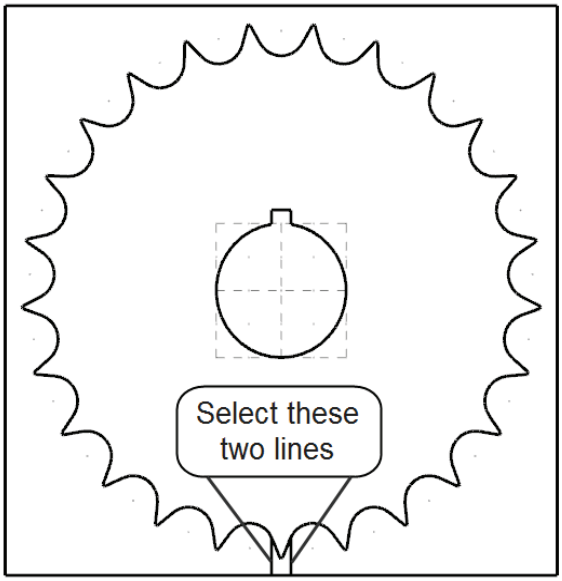
- Select the entities at the same locations as shown in Figure: 16.0.2.

Figure: 16.0.2



- Press **Alt+F1** to fit the drawing to the screen.
- Select the other two remaining lines as shown in Figure: 16.0.3.

Figure: 16.0.3

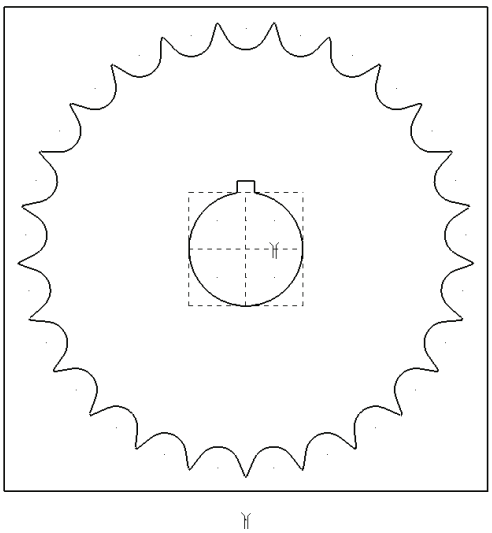


- Select the **OK** button to exit the command.

STEP 17: CREATE THE THREAD POINTS

Thread Point is the point where the machine threads the wire, usually a pre-drilled hole in the material. The wire cuts from the thread position to the start of the chain based on the settings for lead ins and lead outs. See **Getting Started** to learn how to create the Wire utilities toolbar.

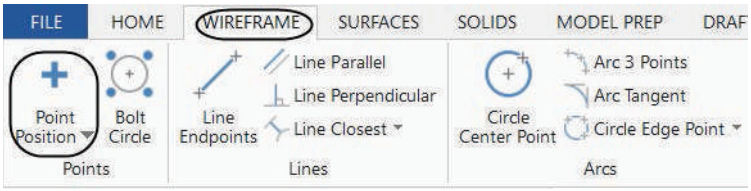
Step Preview:



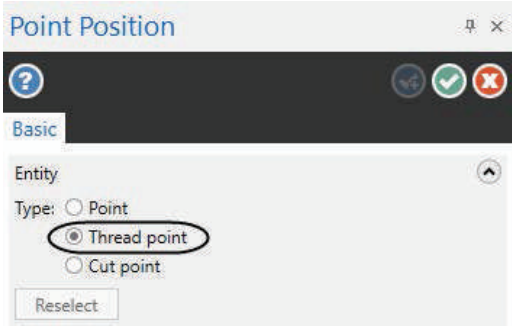
17.1 Create the Thread Point for the outside profile

WIREFRAME

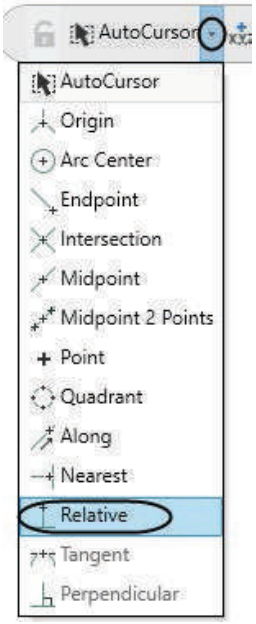
- From the **Points** group, select the **Point Position** icon.



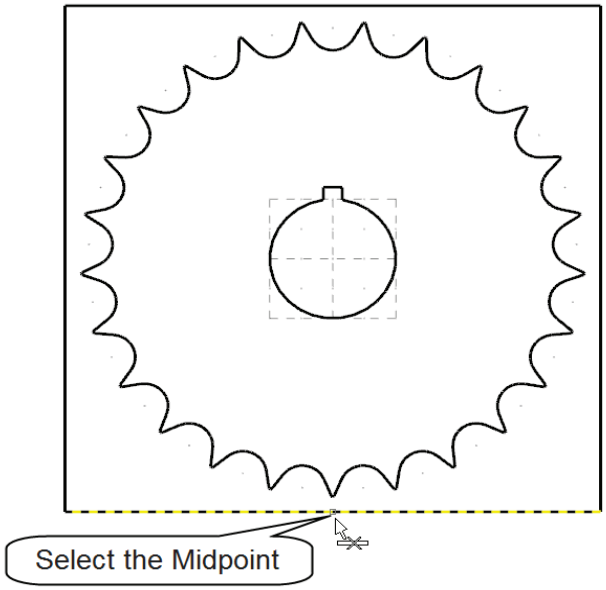
- Select **Thread Point** as shown.



- Click on the drop down arrow next to the **AutoCursor** and select **Relative** as shown.

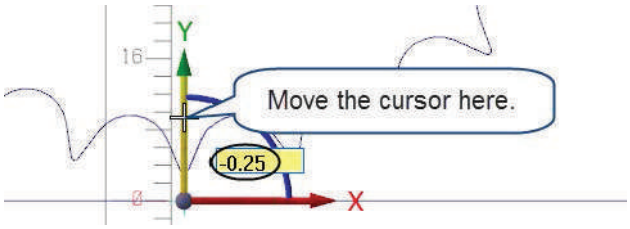


- [Pick a known point to use as a reference]: Select the **Midpoint** of the line as shown.



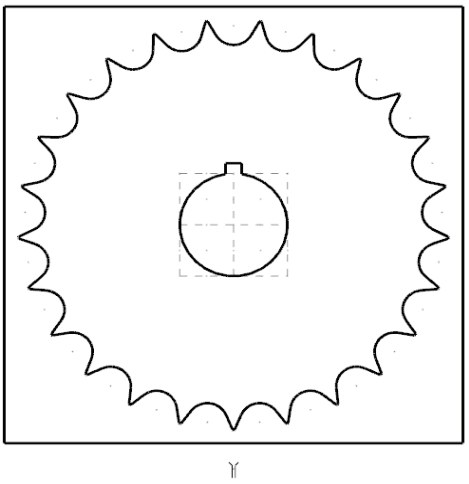
- Make sure that when selecting the **Midpoint**, the visual cue of the cursor changes as shown.

- Hover the cursor above-0.25 the **Y-axis** as shown, then enter **-0.25** to create the point below the line.



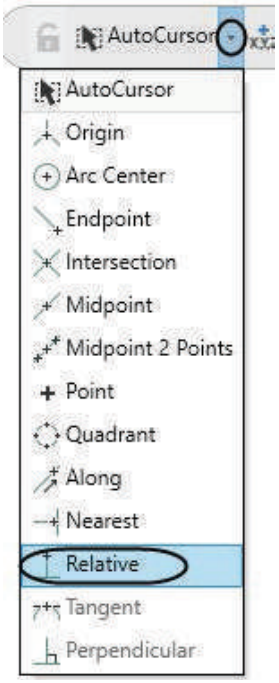
- Press **Enter** three times to create the point.

- Select **OK and Create New Operation** to continue with the same command.
- Press **Alt+F1** to fit the drawing to the screen.
- The geometry should look as shown.



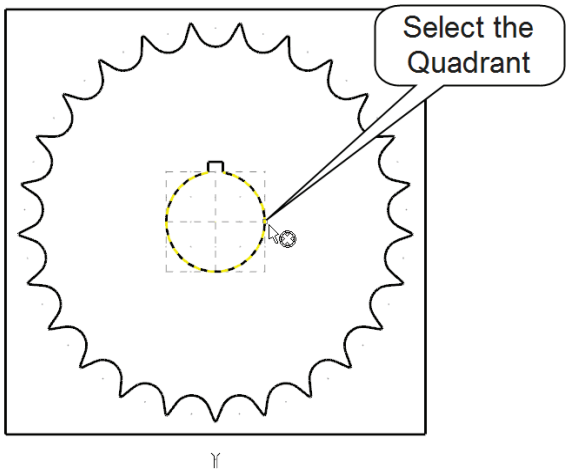
17.2 Create the Thread Point for inside profile

- From the **AutoCursor** drop down list, select **Relative**.

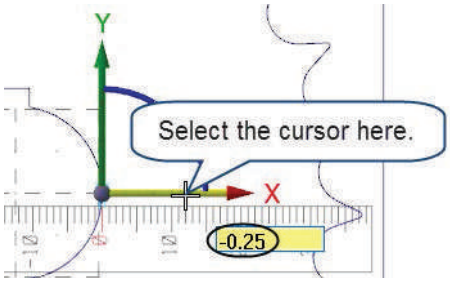


- [Pick a known point to use as a reference]: Select the **Quadrant** of the circle as shown in Figure: 17.2.1.

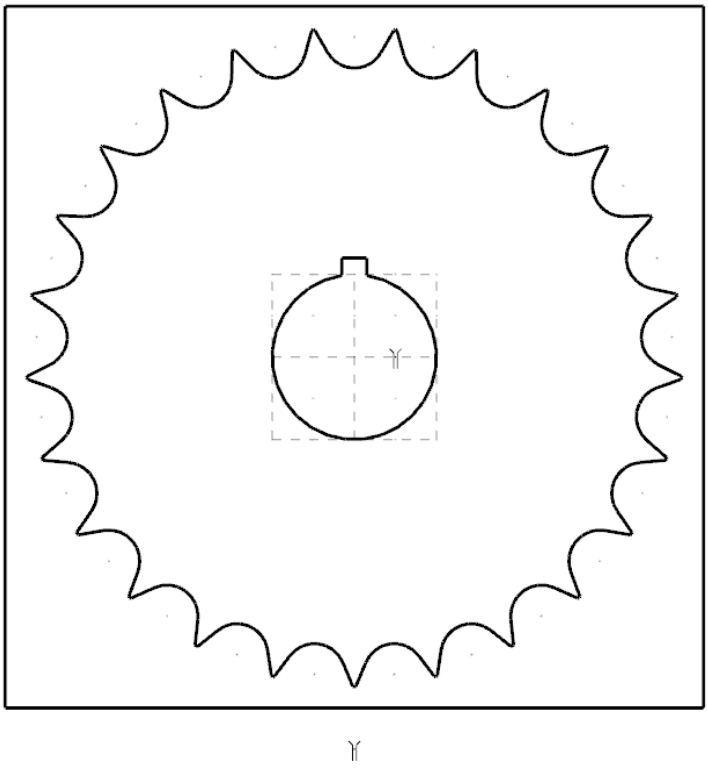
Figure: 17.2.1



- Hover the cursor above the **X-axis** as shown and enter **-0.25** to create the point to the left of the selected point and along the bottom line which is parallel to the X-Axis.



- Press **Enter** three times to create the point.
- Select the **OK** button to complete the command.
- The final geometry should look as shown.



STEP 18: SAVE THE FILE

- From the **Quick Access Toolbar**, select the **Save As** icon.
- Find a location on the computer to save your file. File name: "Tutorial 3_Die Punch.MCAM".

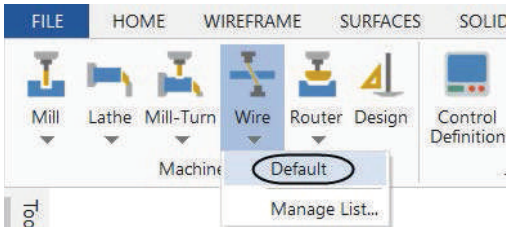
STEP 19: SELECT THE MACHINE AND SET UP THE STOCK

In Mastercam, you select a **Machine Definition** before creating any toolpaths. The **Machine Definition** is a model of your machine tool's capabilities and 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.MCAM-WMD. Once finished, select OK to exit the dialog box. Otherwise, please follow next step.

MACHINE

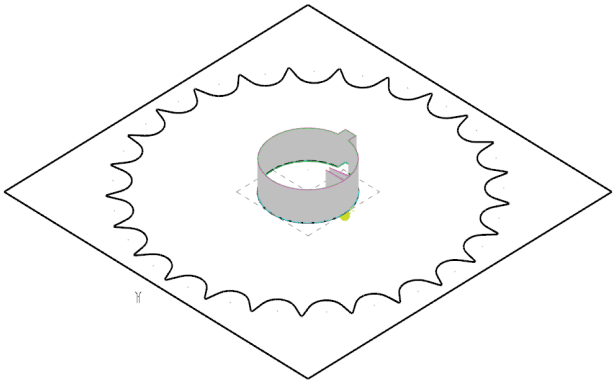
- From the **Machine Type** group, click the pull down arrow under **Wire**.
- Select the **Default**.



STEP 20: MACHINE THE INNER PROFILE USING CONTOUR WIREPATH

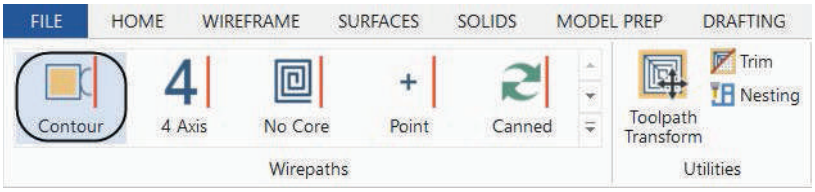
Contour wirepaths have the same general shape in both the XY plane (the lower contour) and the UV plane (the upper contour). Contour wirepaths can taper inward or outward, and you can specify the location of the land, the point at which the taper begins. You can further modify the shape of the contour 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:

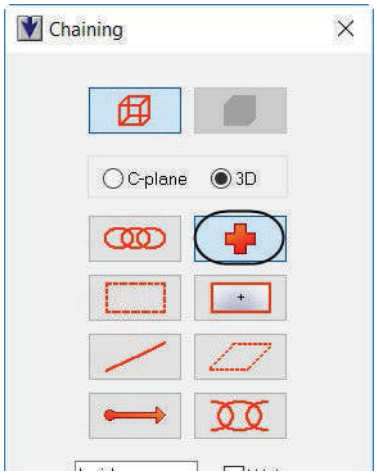


WIREPATHS

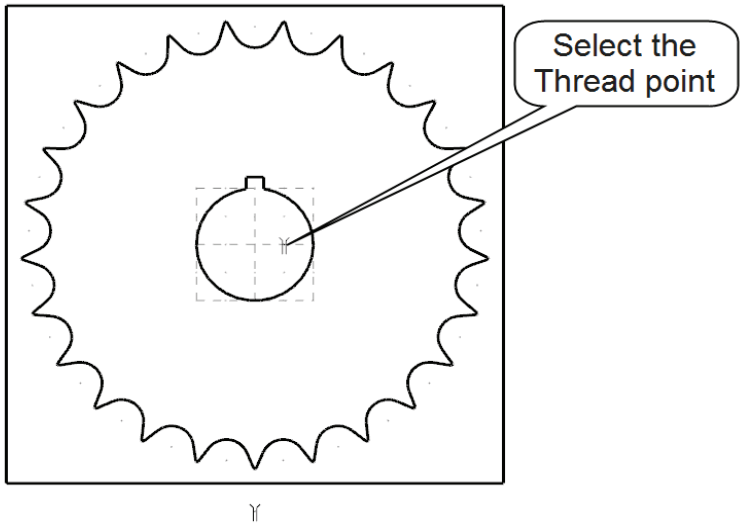
- From the **Wirepaths** group, select **Contour**.



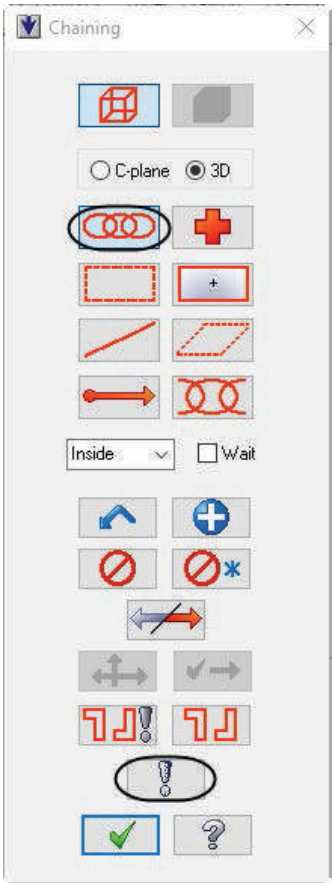
- From the **Chaining** dialog box, select the **Point** button.



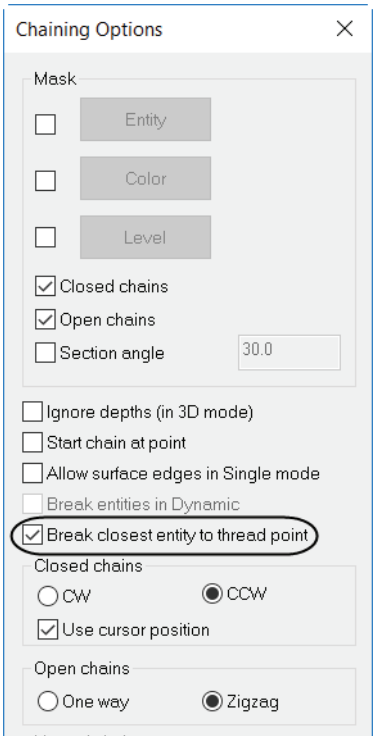
- [Contour: select chain 1]: Select the thread point as shown.



- From the **Chaining** dialog box, select the **Chain** button. Then select the **Option** button as shown.



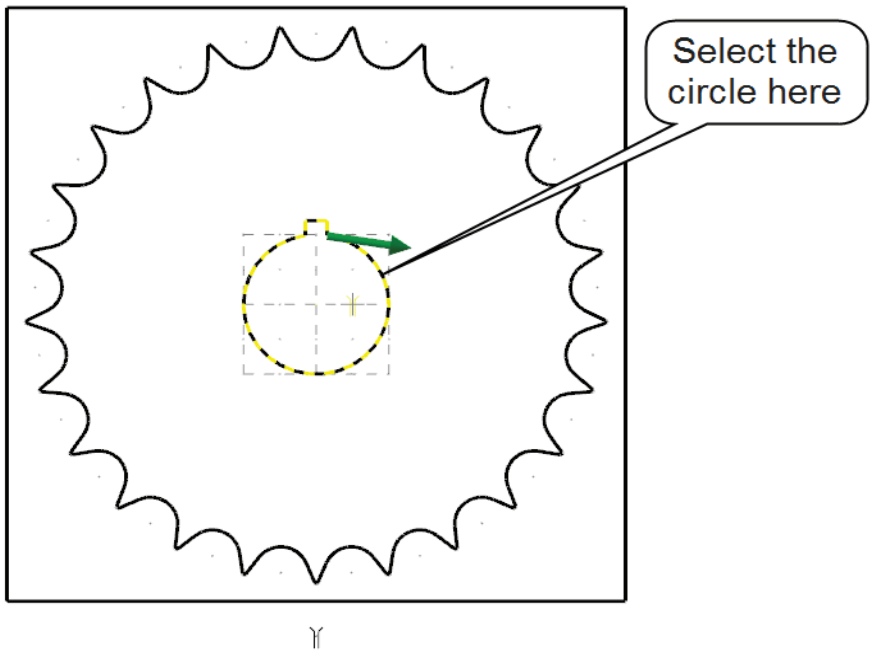
- Enable **Break closest entity to thread point**.



NOTE: Break closest entity to thread point establishes a wirepath start point by breaking the entity closest to the thread point at a position perpendicular to it. If a perpendicular position cannot be established, this setting is ignored.

- Select the **OK** button to exit **Chaining Options**.

- [Contour: select chain 2]: Select the circle as shown.

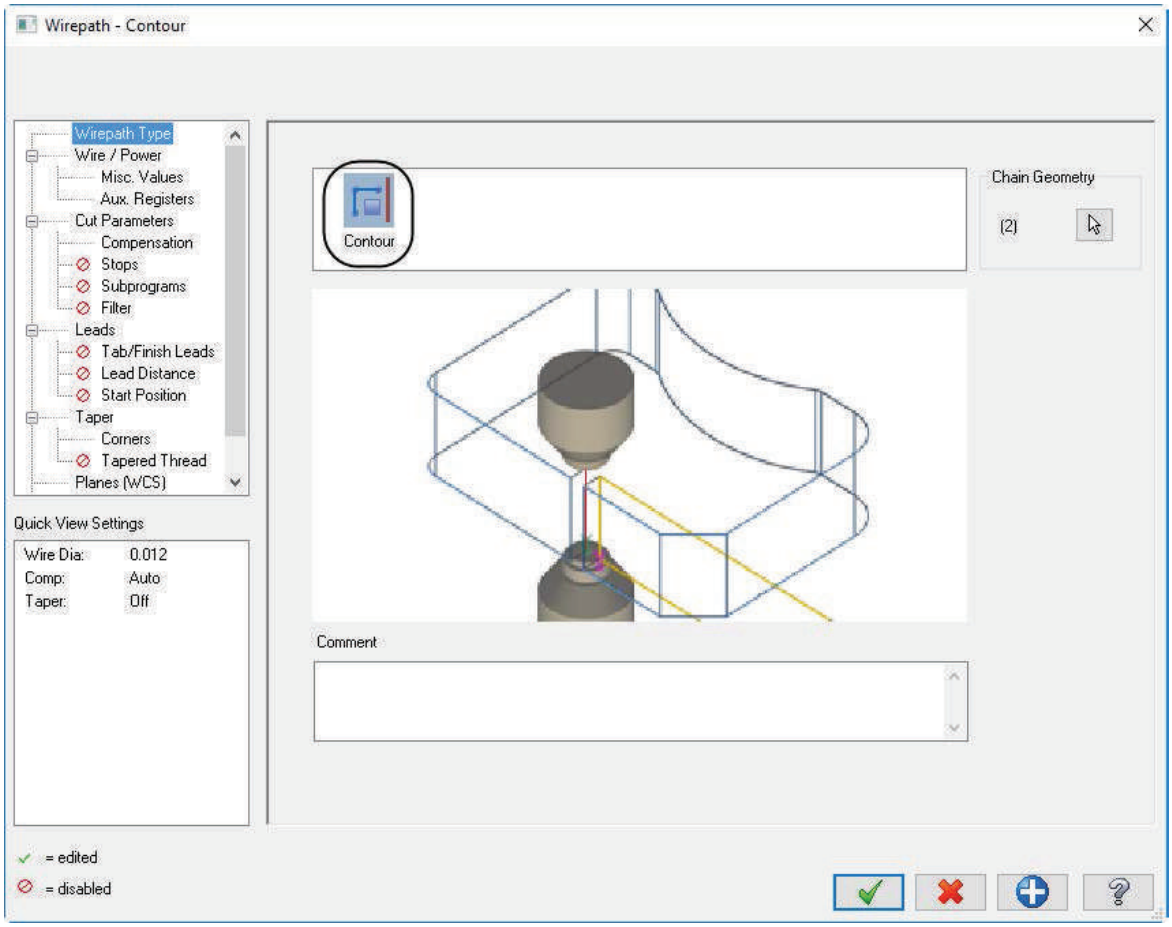


NOTE: The direction of the chain should be Clockwise; otherwise, use the **Reverse** button.

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

20.1 Wirepath Type

- In the **Wirepath type** page, the **Contour** toolpath should already be selected.



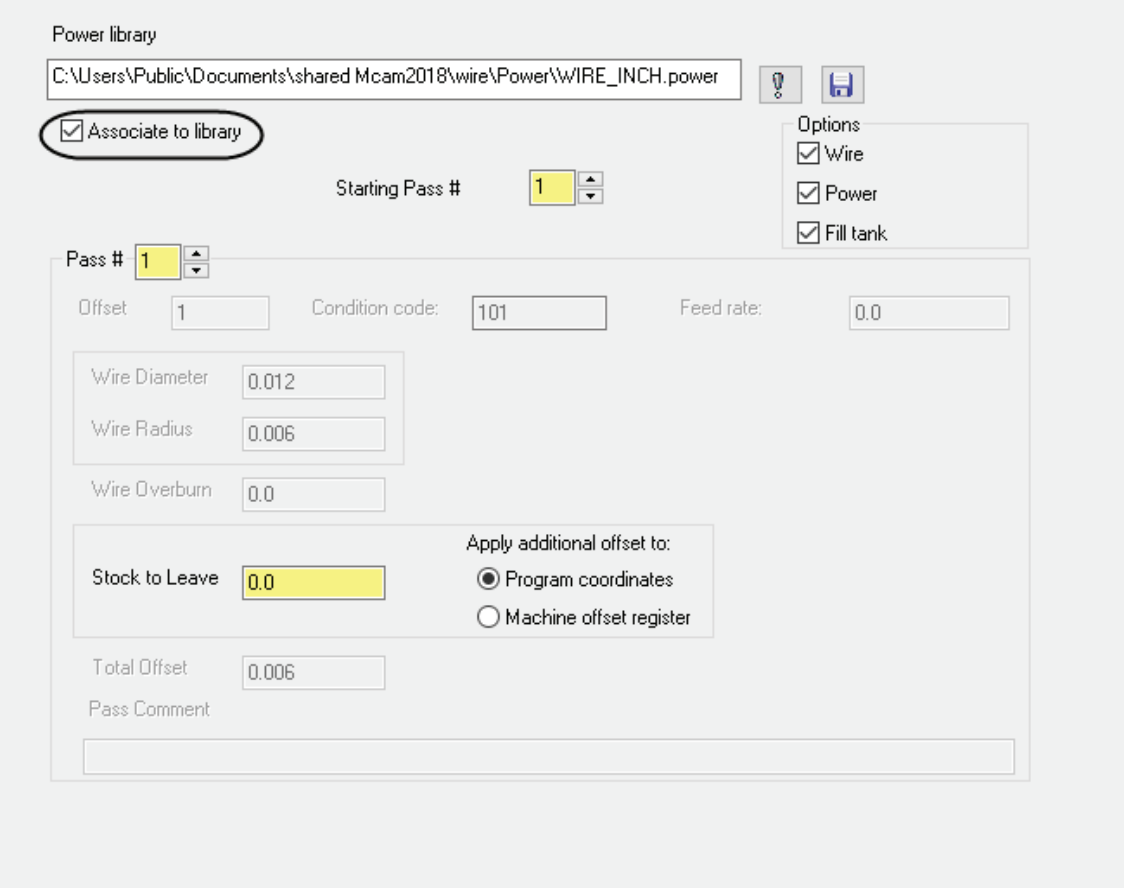
20.2 Set the Wire/Power parameters

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 deselecting the **Associate to library** check box.

- From the **Tree View** area, select **Wire/Power**.

NOTE: Refer to **Tutorial #1 Step 10.2** if you want to create your own wire power library. To open the library saved in Tutorial #1, click on the **Open** icon and select the library from the Power library list. Otherwise, use the default WIRE_INCH.power library as shown in this tutorial.

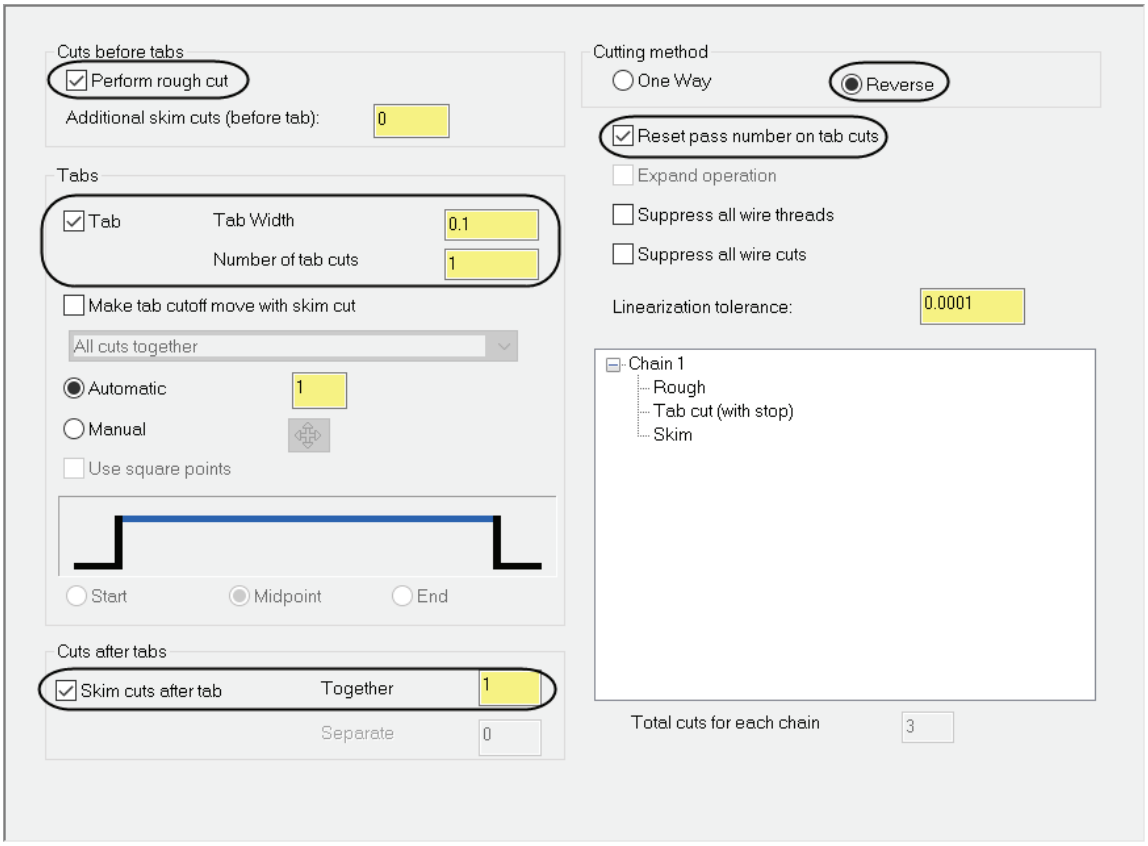
- Enable **Associate to library** to use the default **WIRE_INCH.power** library as shown.



20.3 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, one tab cut, and one finish pass.



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

Tab is the uncut material that secures the part to the material; can be cutoff with the skim cut pass or in a separate pass.

Skim cuts after tab allows you to enter the number of finish passes after cutting the tab.

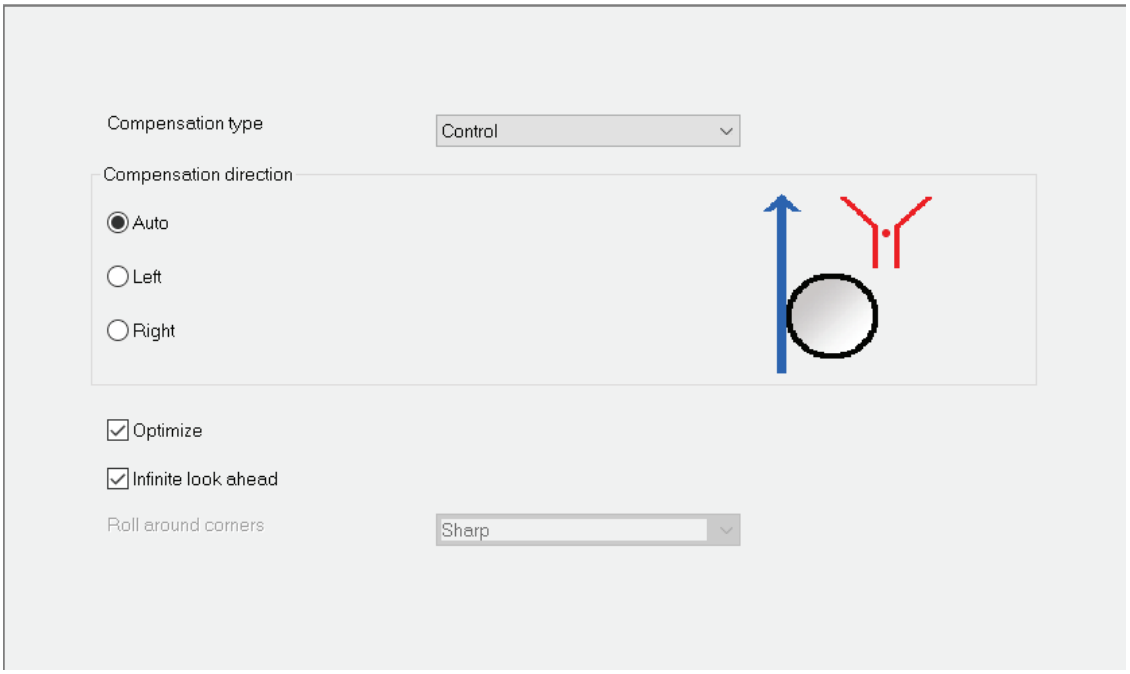
Cutting method set to **Reverse** is used when multiple passes are set for the toolpath. Reverse enabled reverses the direction of the previous pass at the end of the contour or at the tab.

Reset Pass Number on Tab Cuts is selected to use the power settings stored as Pass 1 in the wire power settings library for the first tab cut. Mastercam Wire uses the Pass 1 settings for the rough cut.

20.4 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**.
- Leave the **Compensation direction** set to **Auto**.



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.

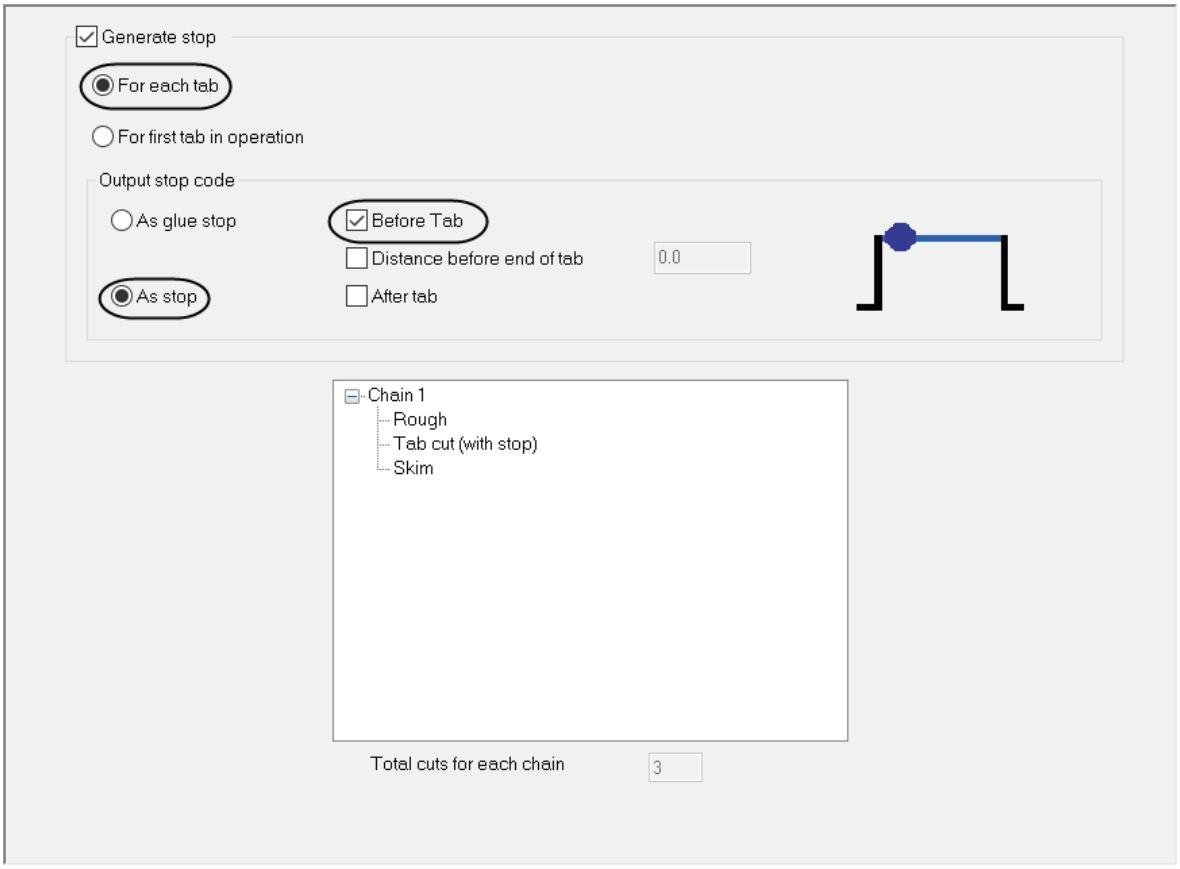
Infinite look ahead checks for wirepath self-intersections along the entire contour before creating the wirepath. If Mastercam Wire finds a wirepath intersection, it modifies the wirepath so that it does not cut the portion of the part that comes after the intersection.

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**.

20.5 Stops

Stops page allows you to create optional stops (M01) or program stops (M00) in a wirepath. For contour wirepaths, you have the options to apply the stops before and/or after tabs.

- From the **Tree View** area, select **Stops**.
- Make sure that the parameters are set to generate the stop (**M00**) before the **Tab**.



For first tab in operation outputs a stop code before the tab cut for the first tab cut on Chain 1 of the operation. Subsequent tab cuts throughout the operation do not include stop codes.

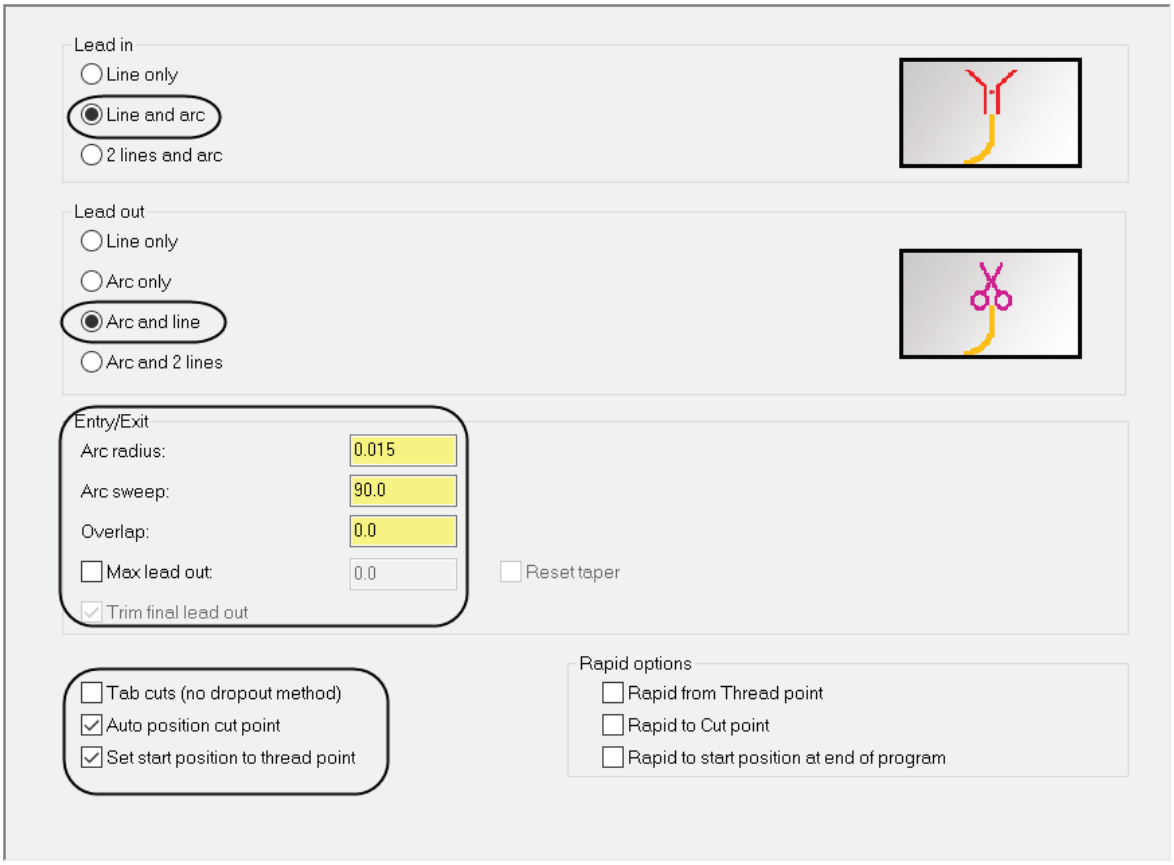
As stop outputs a program stop code (M00). Pauses the wire machine until the operator restarts it.

Before Tab and **After Tab** places the glue stop or the program stop (whichever stop code you select) either before or after the selected tab.

20.6 Leads

Leads page allows you to set the wire motion when entering or exiting a part.

- From the **Tree View** area, select **Leads**. Set the parameters as shown in the figure below.
- Make sure that the **Lead in** is set to **Line and arc**, and **Lead out** is set to **Arc and line**.



Lead in sets the shape of the part entry motion. The lead in extends from the thread point to the start of contour.

Line only creates a line between the thread point and start of contour.

Line and arc adds a line and arc between the thread point and start of contour. Enter values for the **Arc radius** and **Arc sweep**.

Lead out sets the shape of the part exit motion. Lead out extends from the end of contour to the cut point. You have the same options as in the Lead in plus the following extra options.

Arc only in which the wire arcs off the part. Enter values for the **Arc radius** and **Arc sweep**. A lead out radius uses the same **Arc radius** and **Arc sweep** values as the lead in.

Arc and 2 lines the wire arcs off the part, then creates two line moves to the cut point. Enter values for the **Arc radius** and **Arc sweep**. A lead out radius uses the same **Arc radius** and **Arc sweep** values as the lead in.

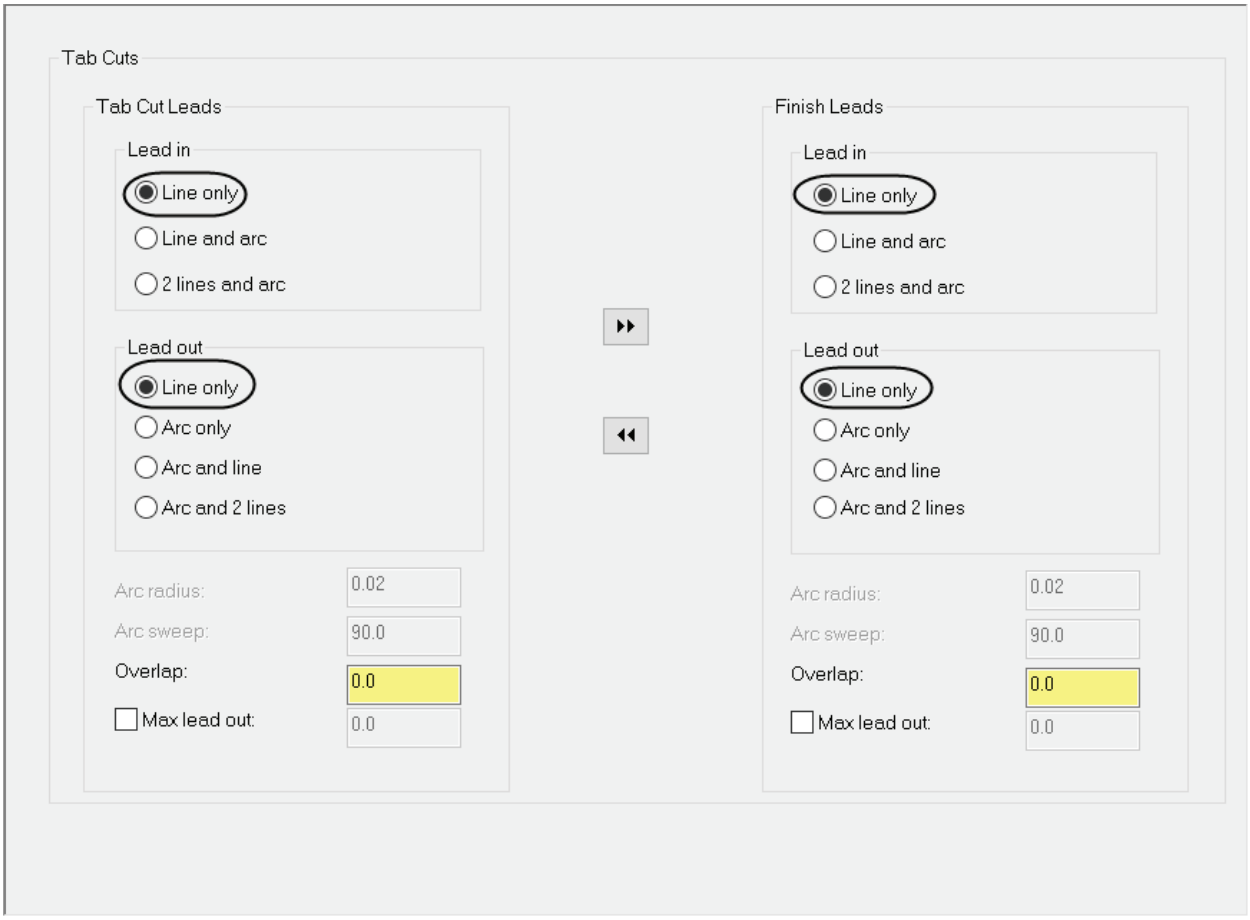
Overlap eliminates potential burrs by overlapping the start and end of the contour by the amount entered.

Tab Cuts (No dropout method) eliminates slugs or slivers of material that result from the tab cut.

20.7 Tab/Finish Leads

Tab/Finish Leads page allows you to set the wire motion when entering or exiting a part when making tab cuts (**Tab Cut Leads** section) and skim cuts after a tab cut (**Finish Leads** section).

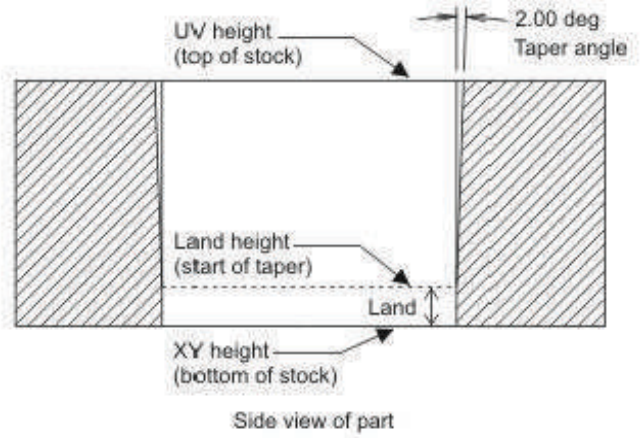
- From the **Tree View** area, select **Tab/Finish Leads**.
- Make sure that the **Lead in** and **Lead out** are set to **Line only** as shown.



20.8 Taper

Taper page allows you to set the parameters to taper the contour wirepath inward or outward and to specify the location of the land (the point at which the taper begins). You can set the Z height of the upper wire guide for rapid moves, the location of the upper wire guide and lower wire guide, the height of the part and the location of the XY plane as shown in Figure: 20.8.1.

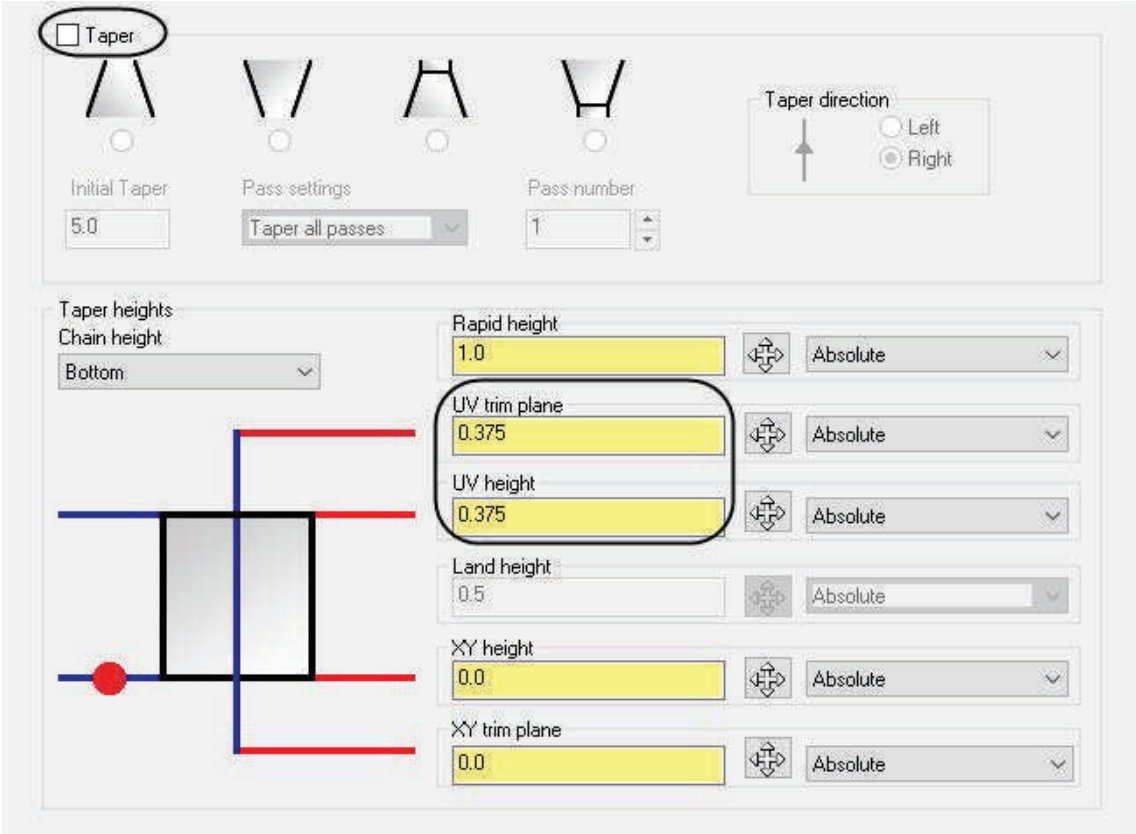
Figure: 20.8.1



- From the **Tree View** area, select **Taper**.

- Make sure that the **Taper** is disabled and change the parameters as shown in Figure: 20.8.2.

Figure: 20.8.2



Chain height options allows you to indicate where the geometry was chained (XY-**Bottom** or UV-**Top** height or, if there is a land, at the land height-**Middle**).

Rapid height sets the Z depth of a rapid move. Can be higher than the **UV height**, **UV trim**, or **UV extension planes** in order to clear any clamps.

UV trim plane can be used to set the location of the upper guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to G code 1015 in the NC file. It is usually set directly at the controller.

UV height sets the location of the XY plane that contains the geometry for the upper contour of a 4-Axis part.

Land height is available only when **Land up** or **Land down** is selected for **Taper** style, and sets the plane at which the wire pivots to the taper angle.

XY height sets the location of the XY plane that contains the geometry for the lower contour of a 4-Axis part.

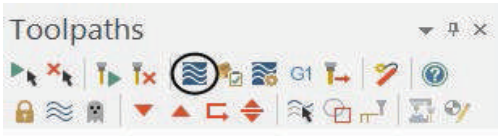
XY trim plane can be used to set the location of the lower guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to Gcode 1015 in the NC file. It can be set directly at the controller. When not using the XY trim plane, enter the same value as the XY height.

- Select the **OK** button to exit the **Wirepath - Contour** parameters.
- Select the **OK** button to exit the **Chain Manager** dialog box.

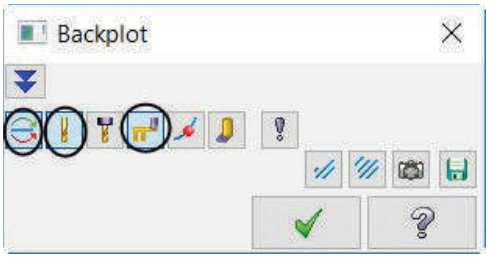
STEP 21: BACKPLOT THE WIRE CONTOUR WIREPATH

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 wirepaths, Mastercam displays the current X, Y, and Z coordinates in the lower left corner of the screen.

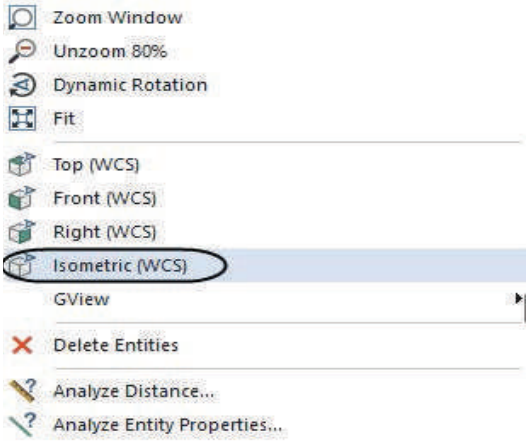
- Make sure that the toolpath is selected (signified by the green check mark on the folder icon).
- Select the **Backplot selected operations** button.



- Make sure that you have the following buttons turned on to see the tool and the rapid moves.



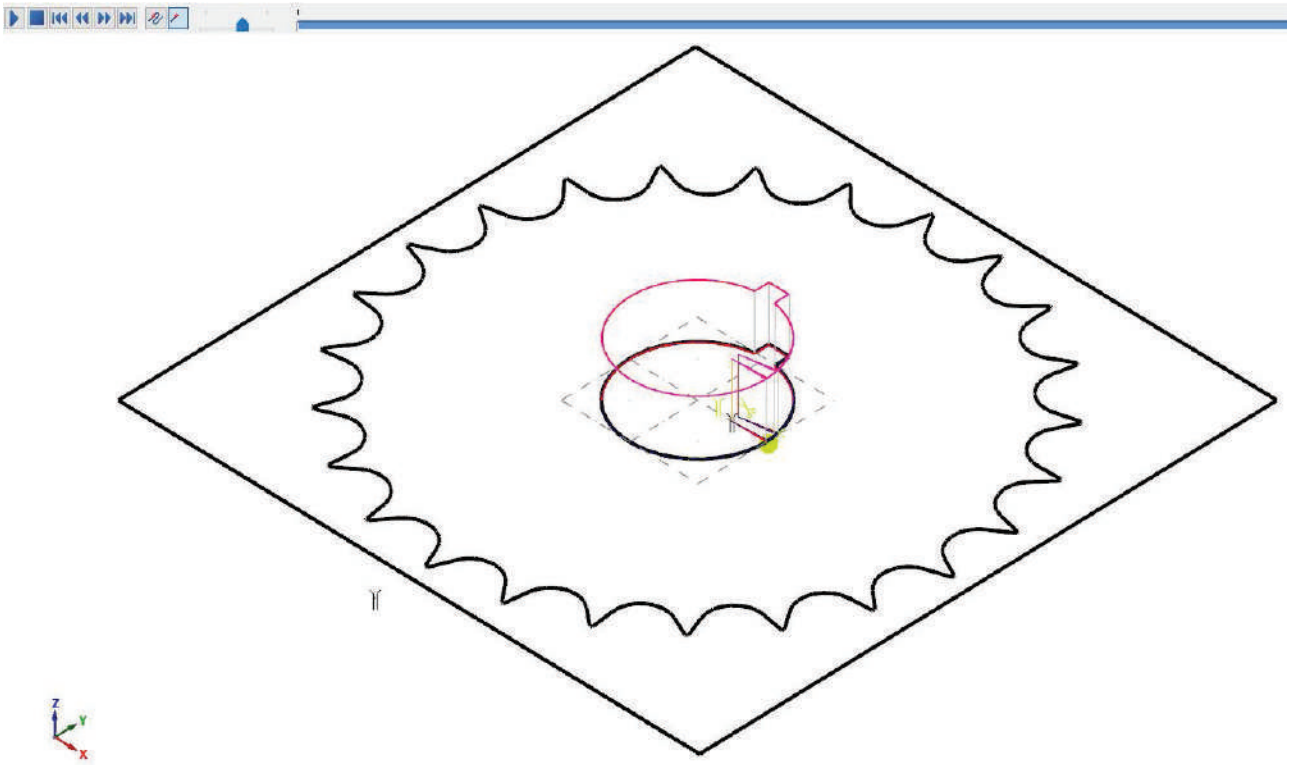
- Right mouse click in the graphics window and select **Isometric** as shown.



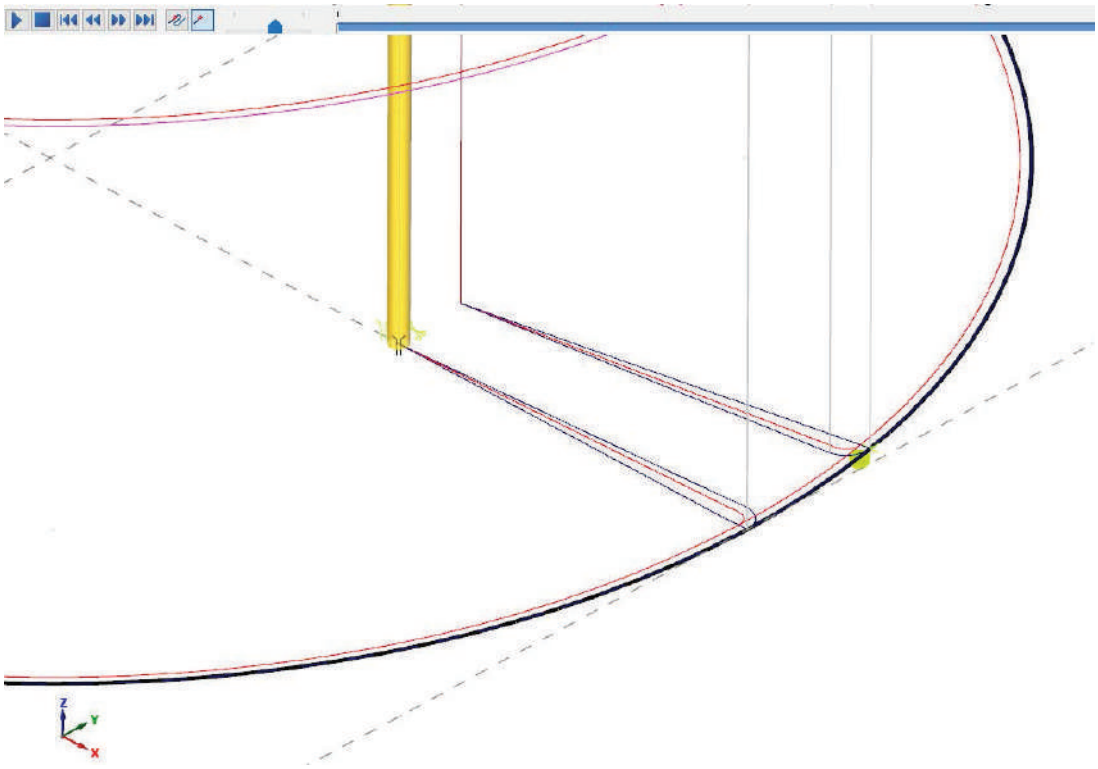
- Press **Alt+F1** to fit the drawing to the screen.
- You can adjust the speed of the backplot.

- You can step through the **Backplot** by using the **Step forward** or **Step back** buttons.
- Select the **Play** button to run **Backplot**.

- The toolpath should look as shown.

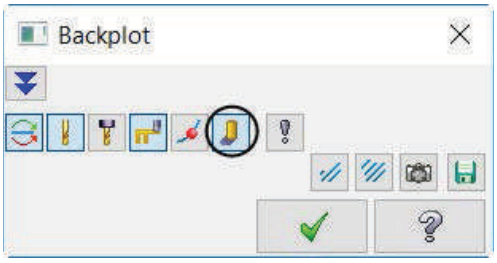


- Hover the cursor above the entry/exit area and using the mouse wheel to zoom in more.

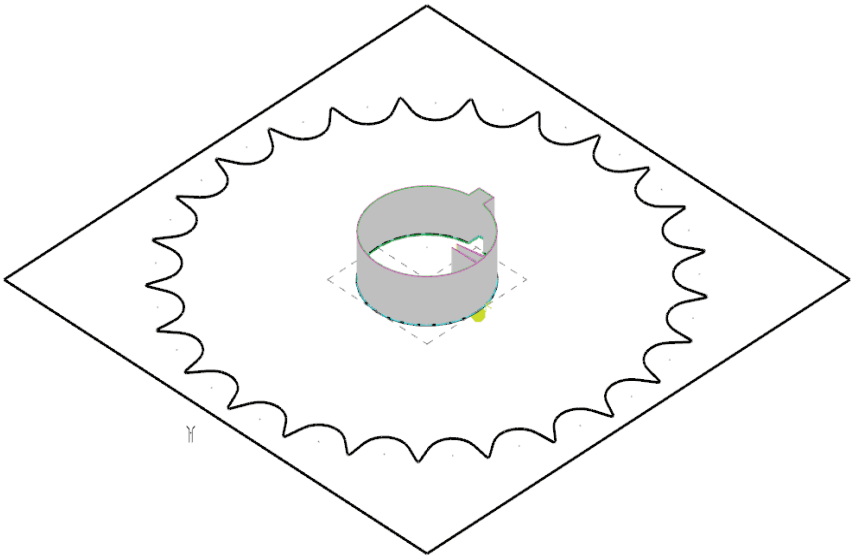


NOTE: You can see in a light green color the symbols for the thread and the cut (scissors) and also the symbol of the stop (filled circle).
No slivers or slug material are remaining after the tab cut due the small radius that we used to define the arc in the Lead in/out page.

- Press **Alt+F1** to fit the drawing to the screen.
- In the **Backplot** dialog box enable **Quick Verify** as shown.



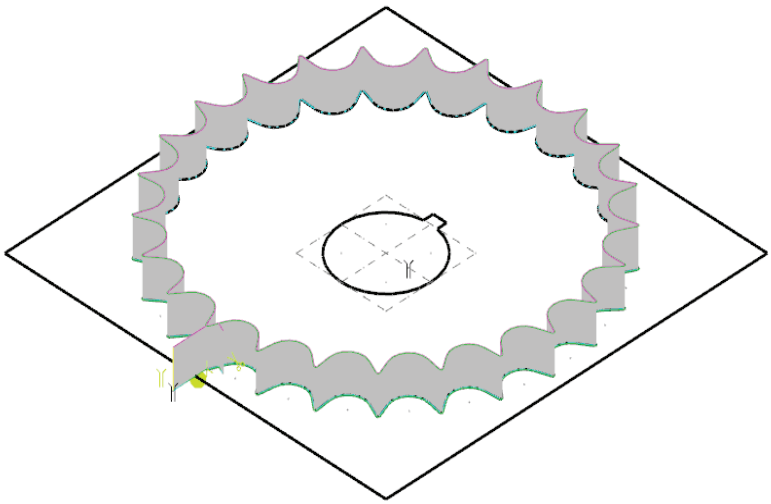
- The toolpath should look as shown.



- Select the **OK** button to exit **Backplot**.
- Press **Alt + T** to remove the toolpath display.

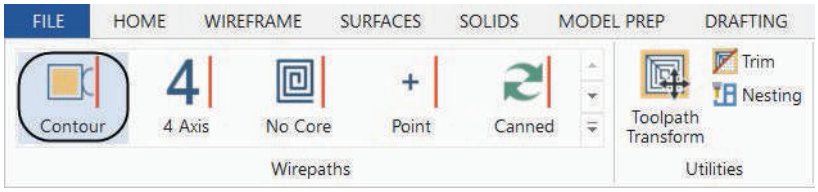
STEP 22: MACHINE THE OUTER PROFILE USING CONTOUR WIREPATH

Toolpath Preview:

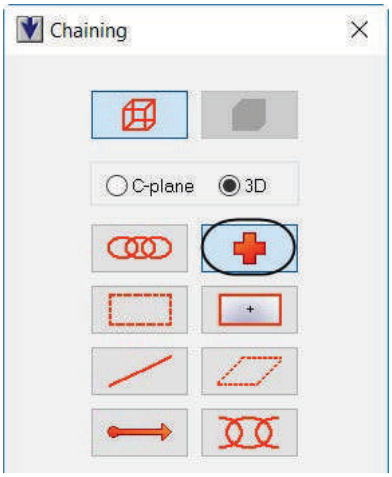


WIREPATHS

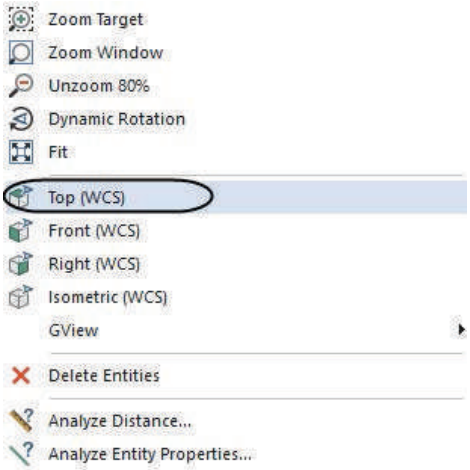
- From the **Wirepaths** group, select **Contour**.



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

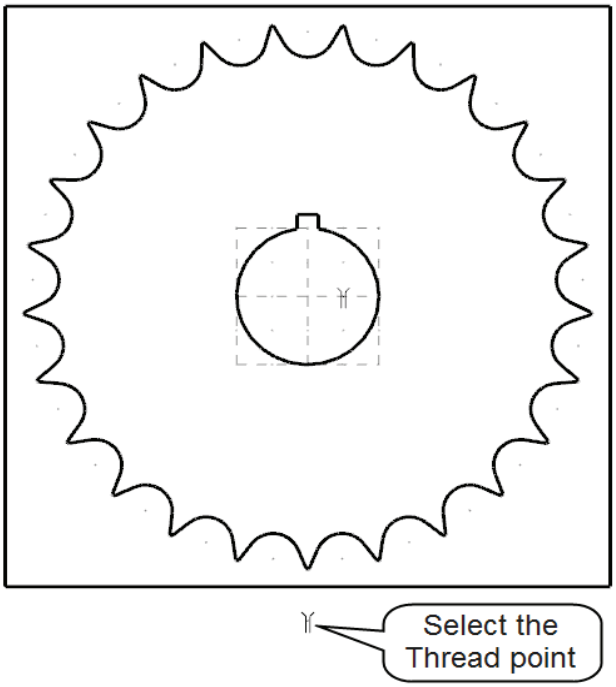


- Right mouse click in the graphics window and change the **Graphic view** to **Top** as shown.

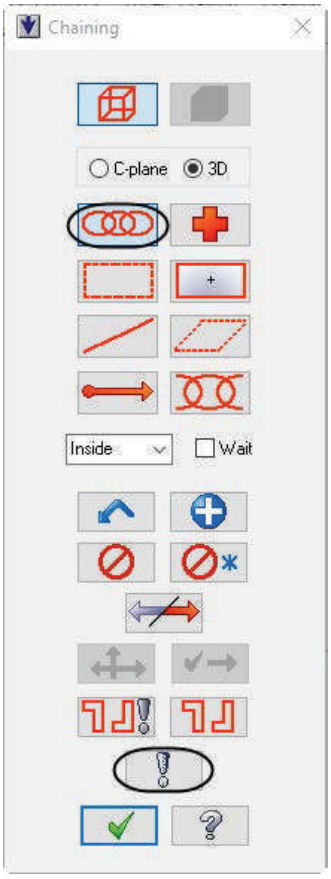


- Press **Alt+F1** to fit the drawing to the screen.

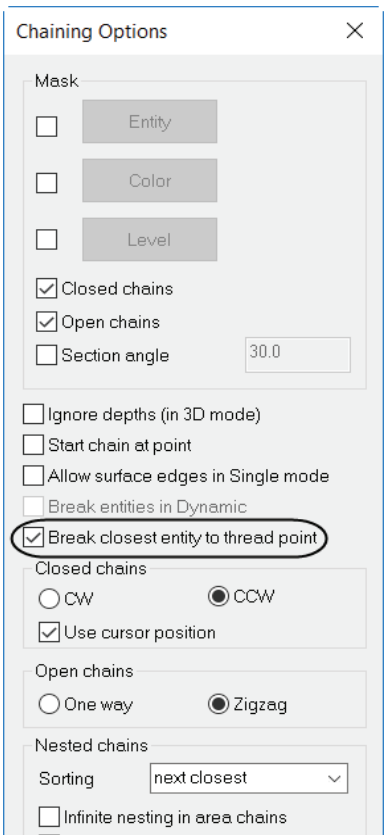
- [Contour: select chain 1]: Select the thread point as shown.



- From the **Chaining** dialog box, select the **Chain** button. Then select the **Options** button as shown.

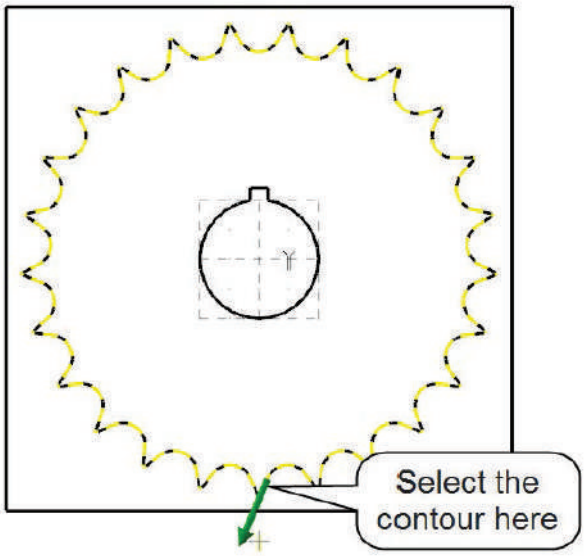


- Once the **Chaining Options** dialog box appears, please make sure that **Break closest entity to thread point** is enabled.



- Select the **OK** button to exit **Chaining Options**.

- [Contour: select chain 2]: Select the profile as shown.

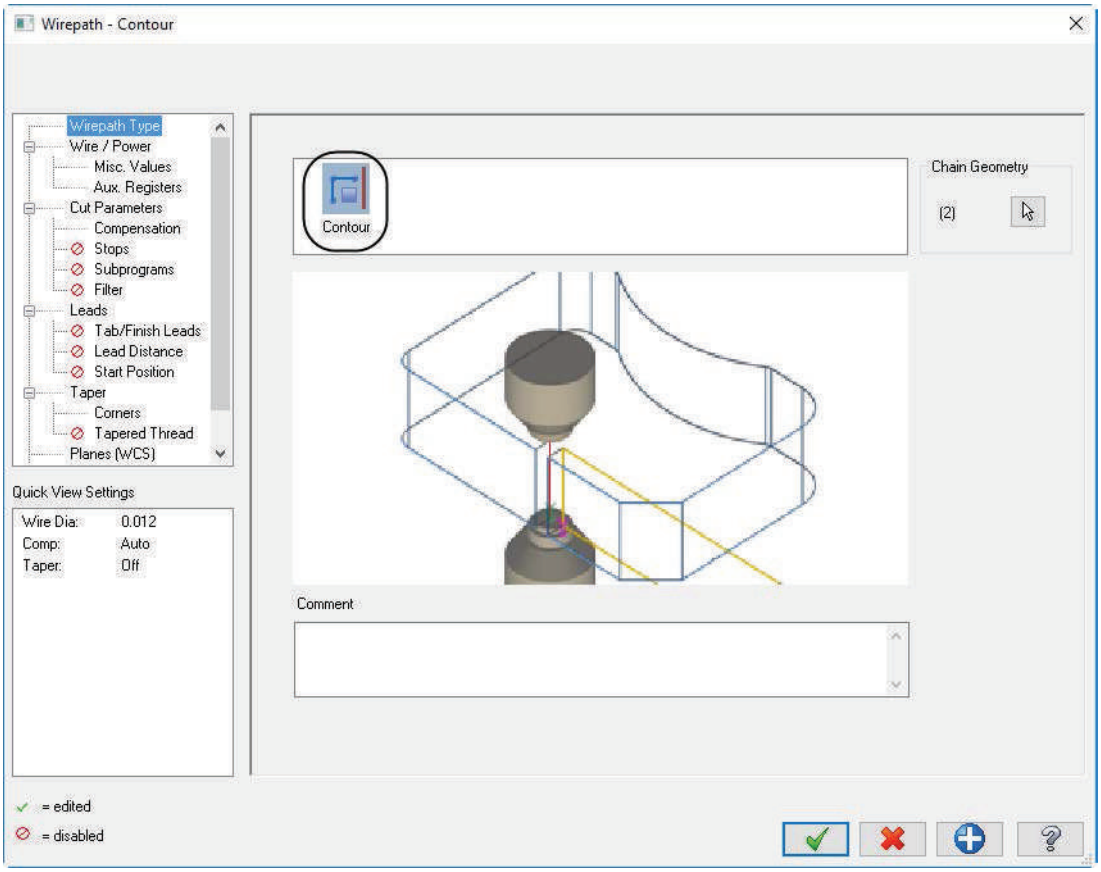


NOTE: The chain direction should be Clockwise. If not, use the **Reverse** button to flip the direction.

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

22.1 Wirepath Type

- In the **Wirepath type** page, the **Contour** toolpath should already be selected.



22.2 Set the Wire/Power parameters

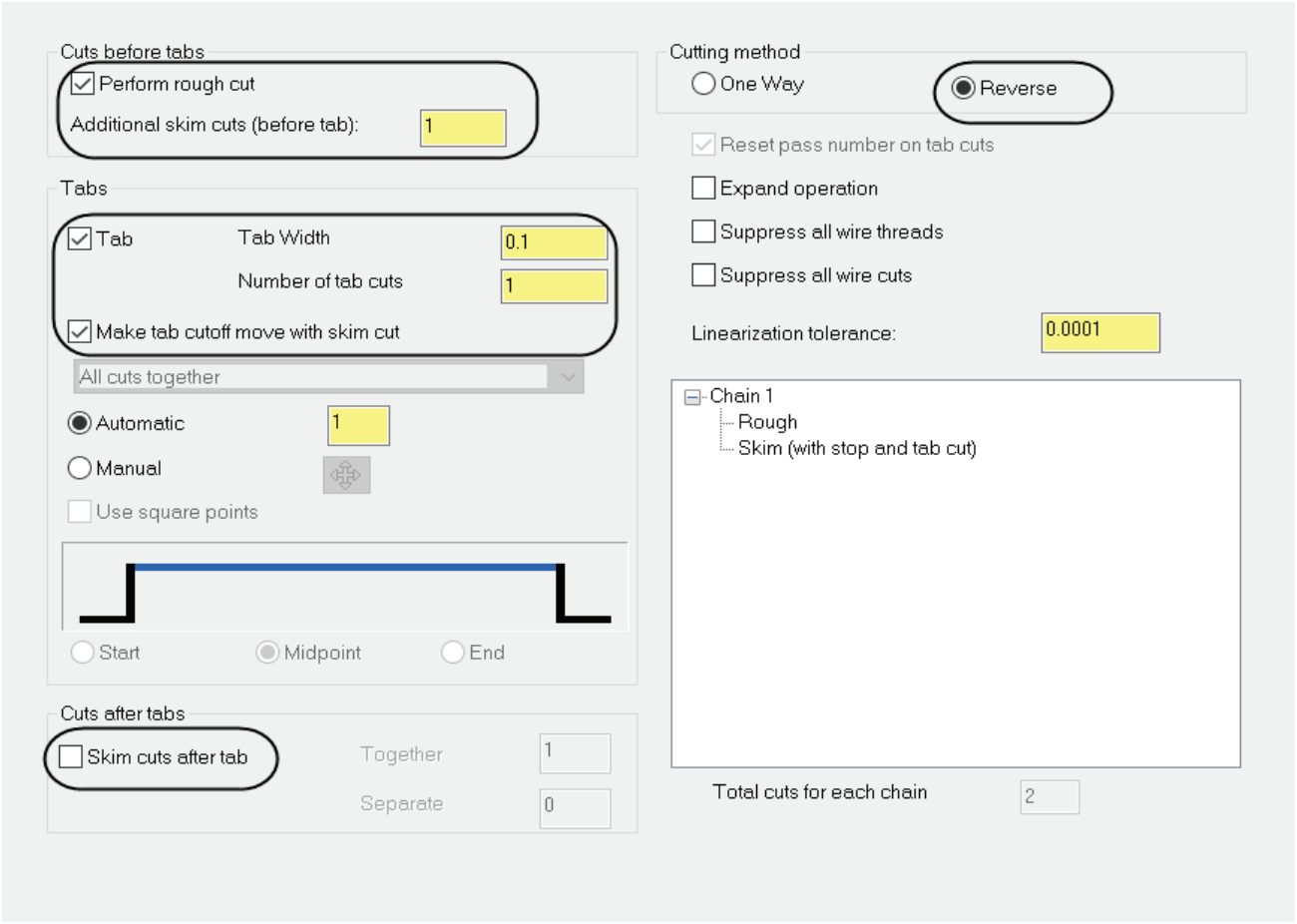
- From the **Tree View** area, select **Wire/Power**.

> **NOTE:** We are using the same default WIRE_INCH.power library. No changes are needed.

22.3 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 skim cut with a stop and tab cut as shown.



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

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

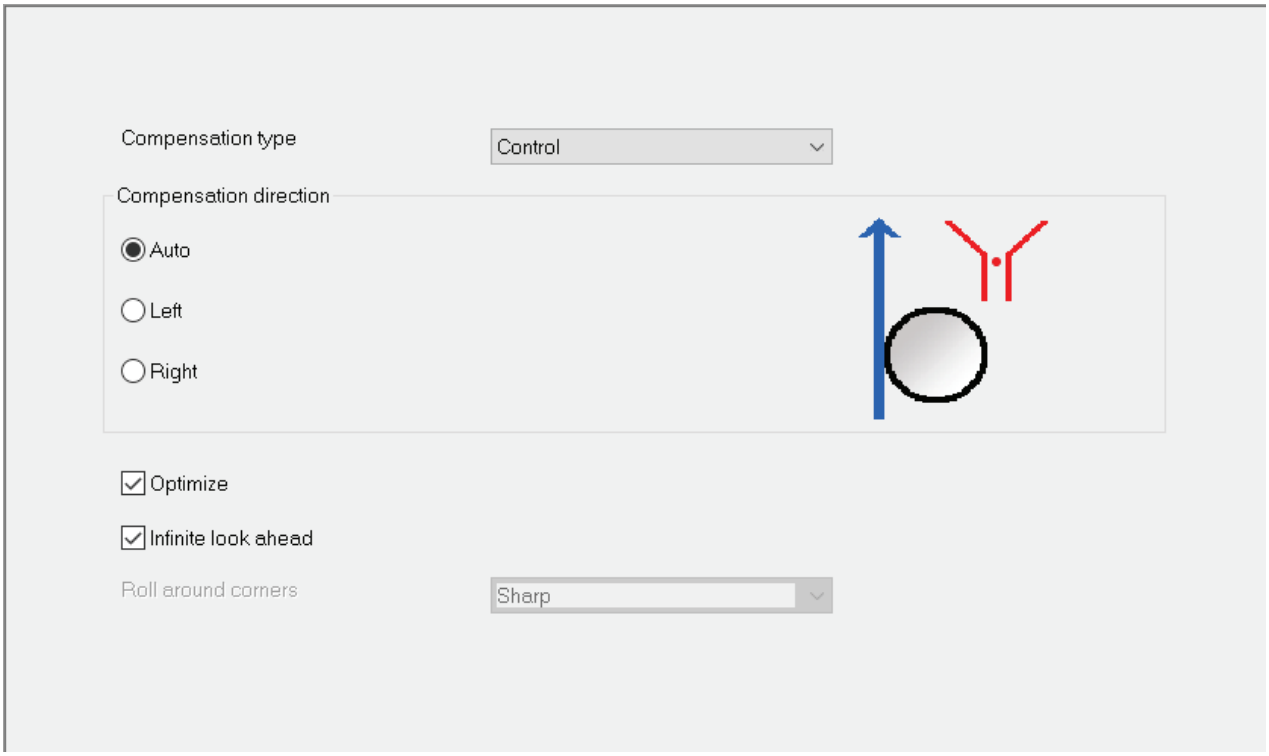
Tab is the uncut material that secures the part to the material; can be cutoff with the skim cut pass or in a separate pass.

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

Make tab cutoff move with skim cut allows you to combine the tab cut with the last skim cut. If you have no skim cut before the tab, the tab cut combines with the rough cut.

22.4 Set the Compensation parameters

- From the **Tree View** area, select **Compensation**.
- Leave the **Compensation direction** to **Auto**.



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.

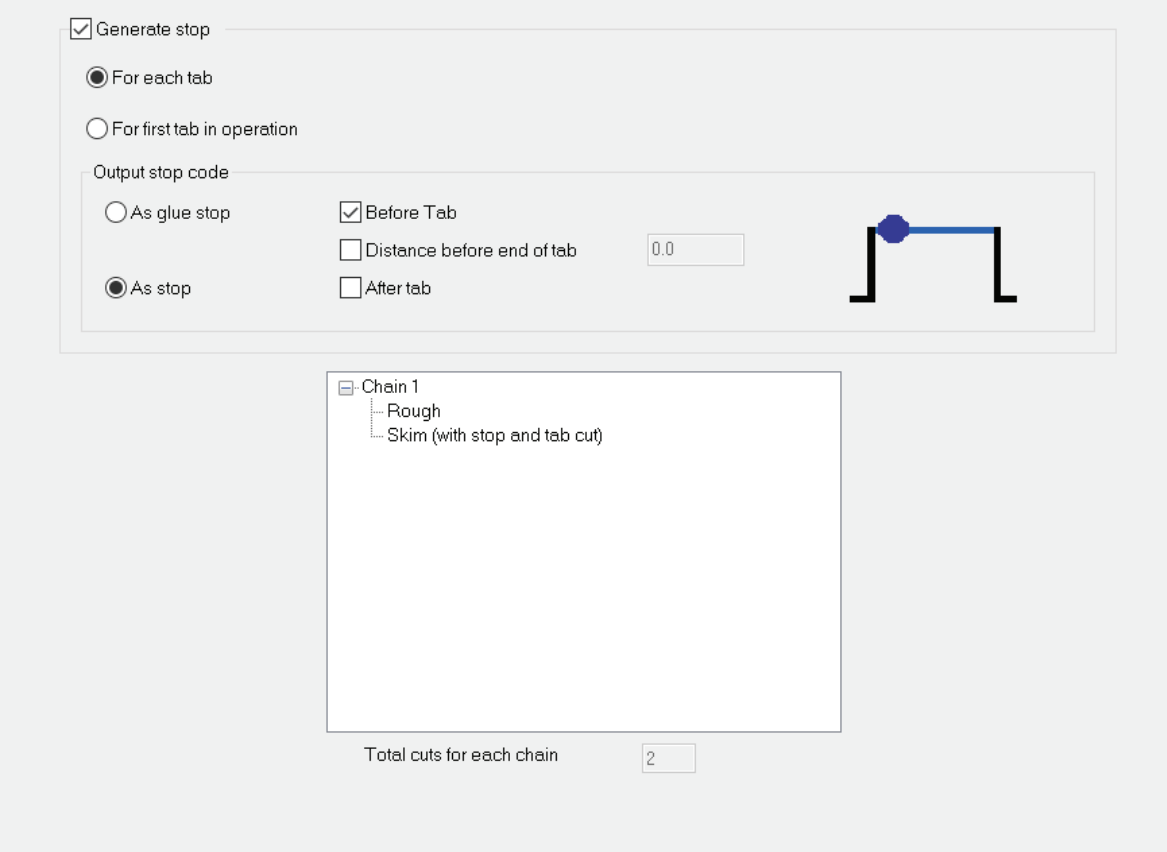
Infinite look ahead checks for wirepath self-intersections along the entire contour before creating the wirepath. If Mastercam Wire finds a wirepath intersection, it modifies the wirepath so that it does not cut the portion of the part that comes after the intersection.

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**.

22.5 Stops

Stops page allows you to create optional stops (M01) or program stops (M00) in a wirepath. For contour wirepaths, you have the options to apply the stops before and/or after tabs.

- From the **Tree View** area, select **Stops**.
- Make sure that the parameters are set to generate the stop (**M00**) before the **Tab**.



For first tab in operation outputs a stop code before the tab cut for the first tab cut on Chain 1 of the operation. Subsequent tab cuts throughout the operation do not include stop codes.

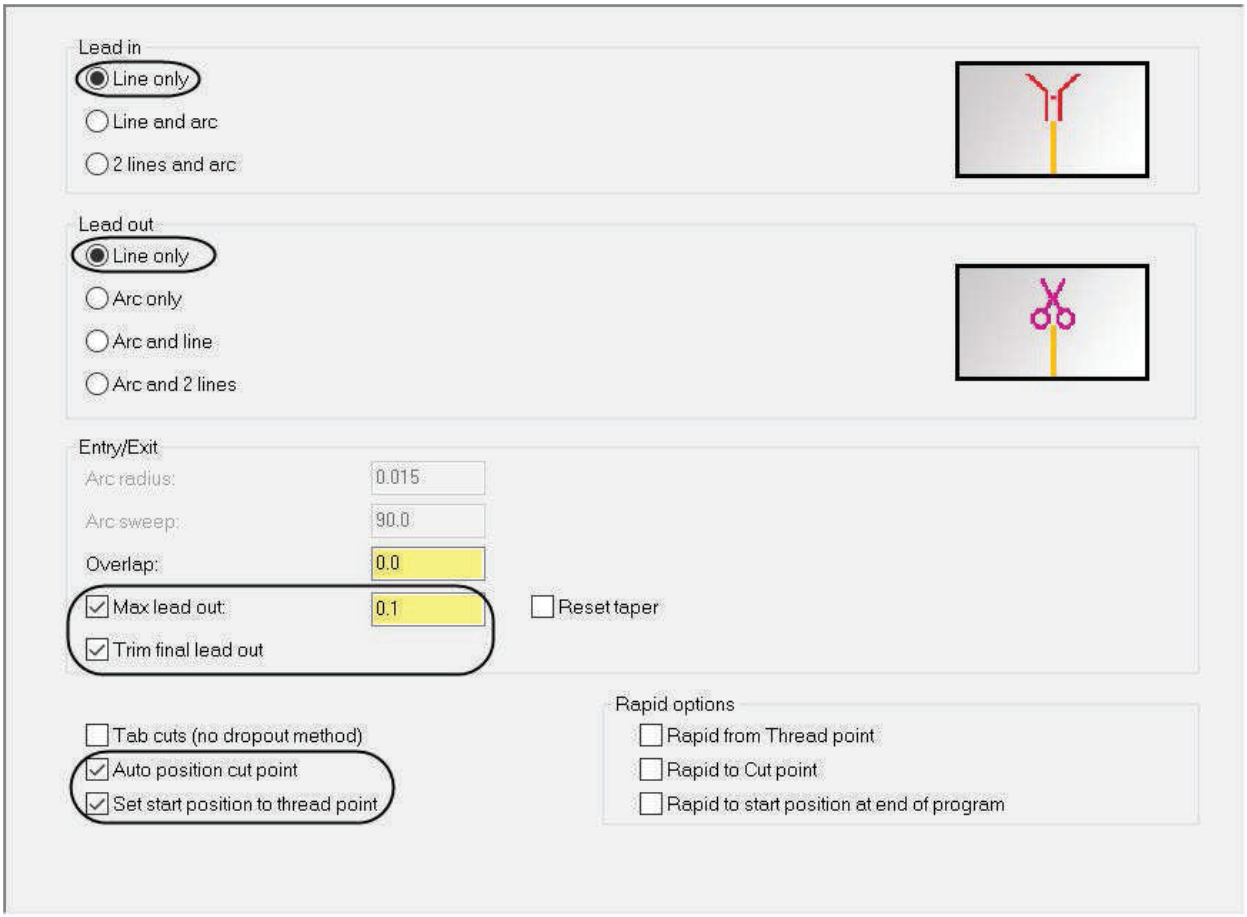
As stop outputs a program stop code (M00). Pauses the wire machine until the operator restarts it.

Before Tab and **After Tab** places the glue stop or the program stop (whichever stop code you select) either before or after the selected tab.

22.6 Leads

Leads page allows you to set the wire motion when entering or exiting a part.

- From the **Tree View** area, select **Leads**. Set the parameters as shown in the figure below.
- Make sure that the **Lead in** and **Lead out** are set to **Line only**.



Lead in sets the shape of the part entry motion. The lead in extends from the thread point to the start of contour.

Line only creates a line between the thread point and start of contour.

Lead out sets the shape of the part exit motion. Lead out extends from the end of contour to the cut point. You have the same options as in the Lead in plus the following extra options.

Max lead out allows you to shorten the lead out by entering a new value for the lead out. When disabled, the lead out equals the distance from the end of contour to the cut point for each pass.

Trim final lead out allows you to constrain the final lead out within the Max lead out length you defined.

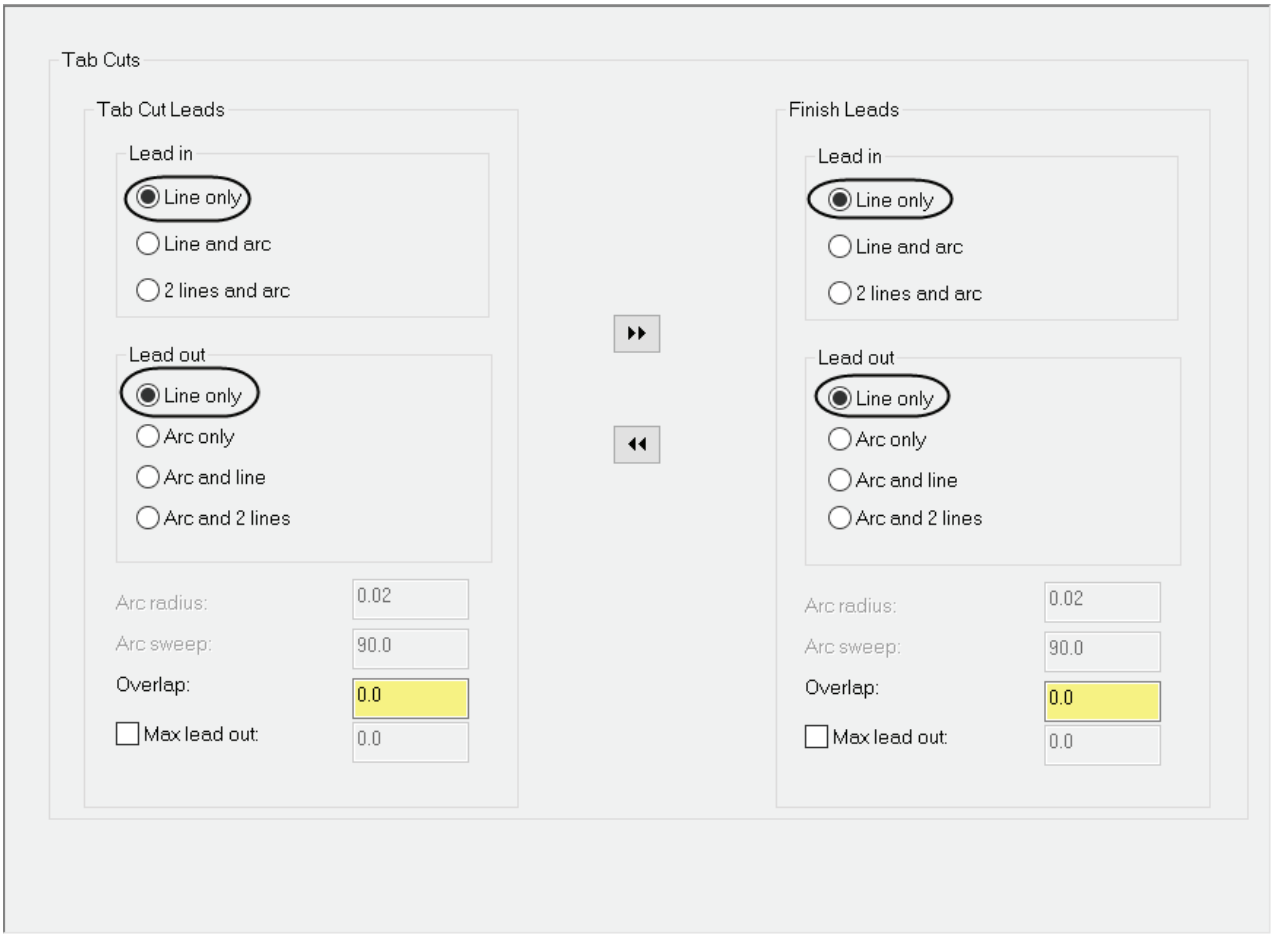
Auto position cut point allows Mastercam Wire to determine the most efficient cut point for a contour. Usually remains enabled.

Set start position to thread point automatically sets the start position to thread position. Enable to move the job start point to the thread point of the first contour in the operation.

22.7 Tab/Finish Leads

Tab/Finish Leads page allows you to set the wire motion when entering or exiting a part when making tab cuts (**Tab Cut Leads** section) and skim cuts after a tab cut (**Finish Leads** section).

- From the **Tree View** area, select **Tab/Finish Leads**.
- Make sure that the **Lead in** and **Lead out** are set to **Line only** as shown.

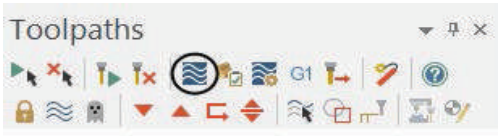


NOTE: The rest of the parameters can be left as they were set for the previous wire contour.

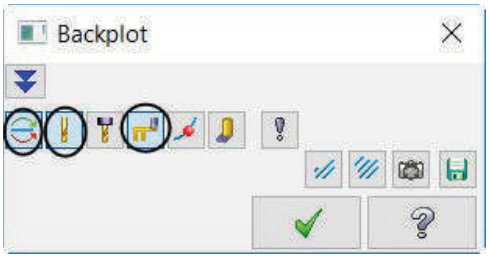
- Select the **OK** button to exit the **Wirepath - Contour** parameters.
- Select the **OK** button to exit the **Chain Manager** dialog box.

22.8 Backplot the last Wire Contour wirepath

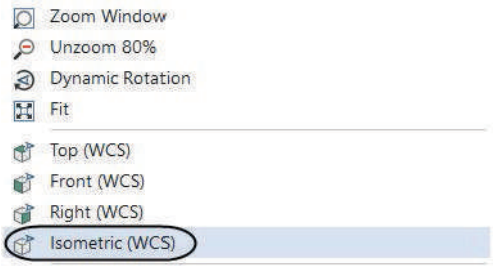
- Select the last Wire Contour wirepath. Then click on the **Backplot selected operations** button.



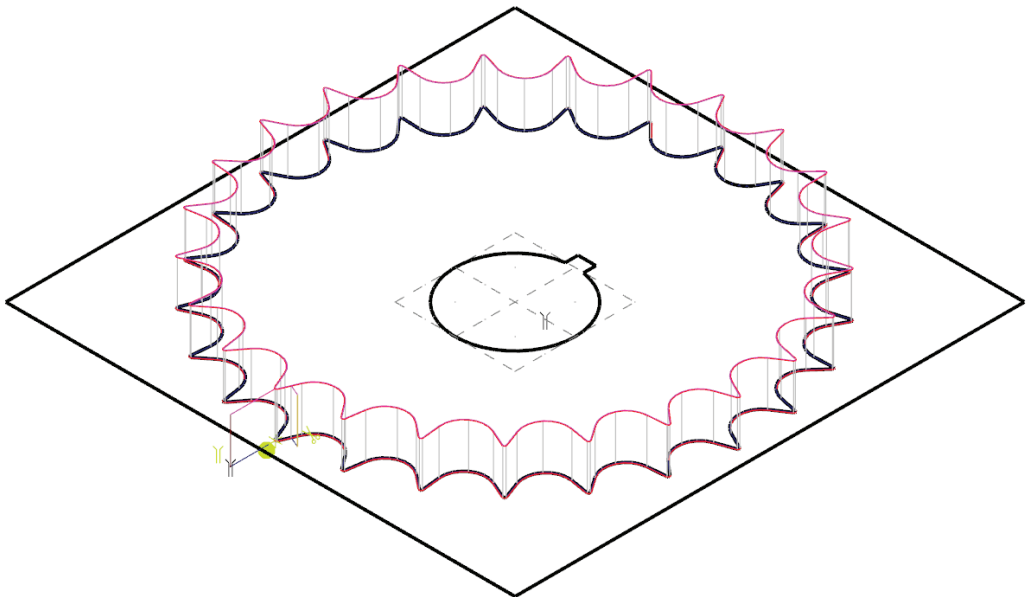
- Make sure that you have the following buttons turned on to see the tool and the rapid moves.



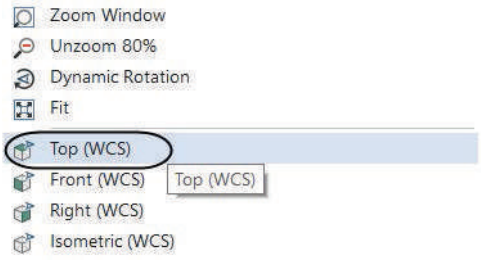
- Right mouse click on the graphics window and select the **Isometric** view as shown.



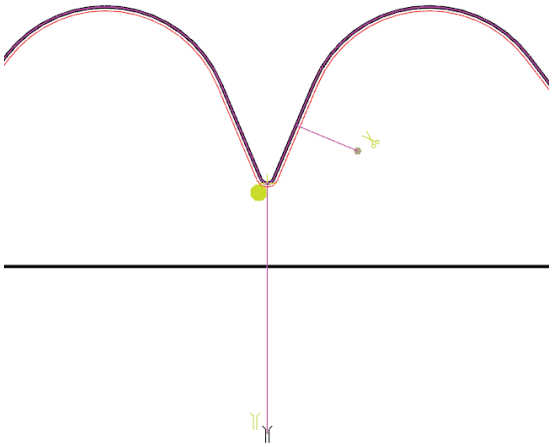
- Press **Alt+F1** to fit the drawing to the screen.
- Select the **Play** button to run **Backplot**.
- The toolpath should look as shown.



- Right mouse click in the graphics window and select the **Top** view as shown.

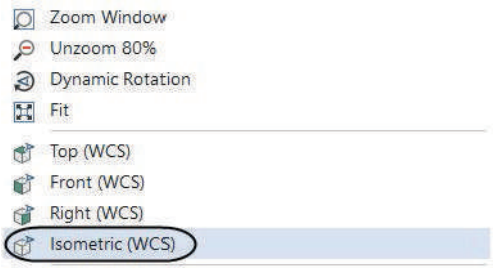


- Hover the cursor above the thread point and scroll up the mouse wheel to zoom in.



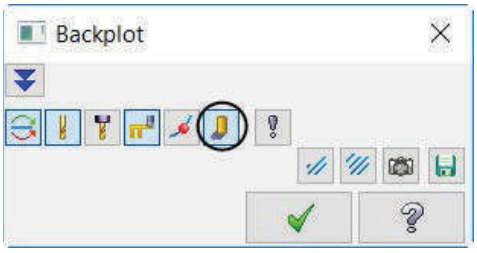
NOTE: The wire starts at the thread point and leads in along a line and makes the roughing pass in the CW direction. It leads out, before the tab (0.1" from the starting location) along the shorter line (max lead out). After that it returns along the same line and continues in the CCW direction with the skim cut. It stops (M00) at the same location where it started the roughing cut on the profile, allowing the operator to hold the part with a magnet. Once the operator starts the program again, the wire cuts the tab and retracts along the shorter line and cuts the wire.

- Change the **Graphics view** to **Isometric**.

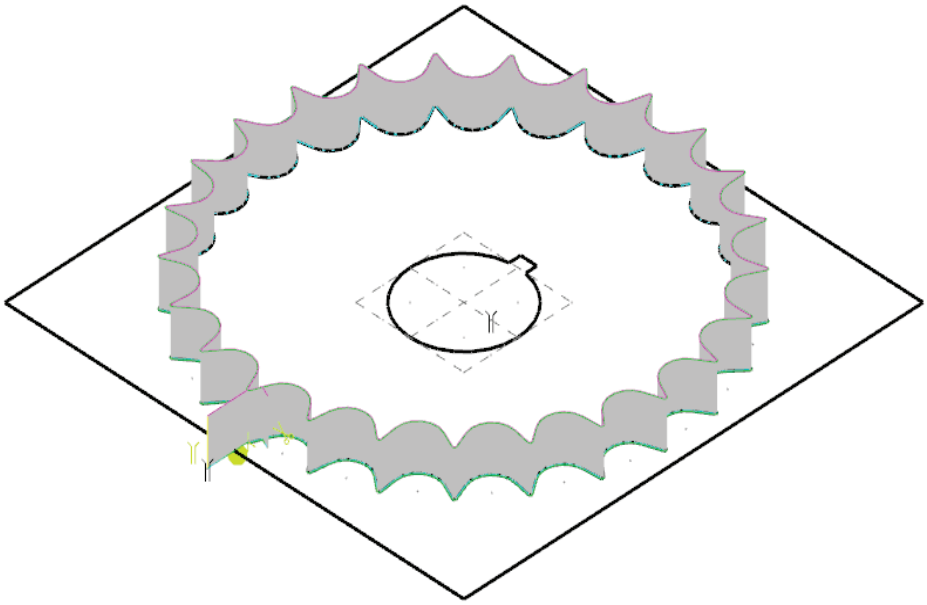


- Press **Alt+F1** to fit the drawing to the screen.

- In the **Backplot** dialog box enable **Quick Verify** as shown.



- The toolpath should look as shown.



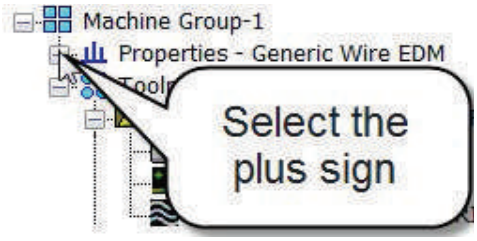
- Select the **OK** button to exit **Backplot**.
- Press **Alt + T** to remove the toolpath display.

STEP 23: VERIFY THE TOOLPATH

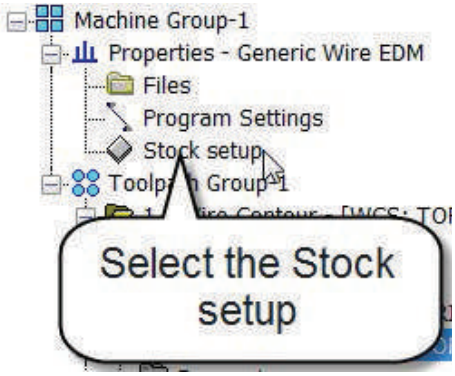
Simulates the machining of a part from a solid stock model display. The stock dimensions are based on the values that you can specified in the **Stock Setup**.

23.1 Set up the stock

- From the **Toolpaths Manager**, select the plus in front of the **Properties** to expand the **Toolpaths Group Properties**.

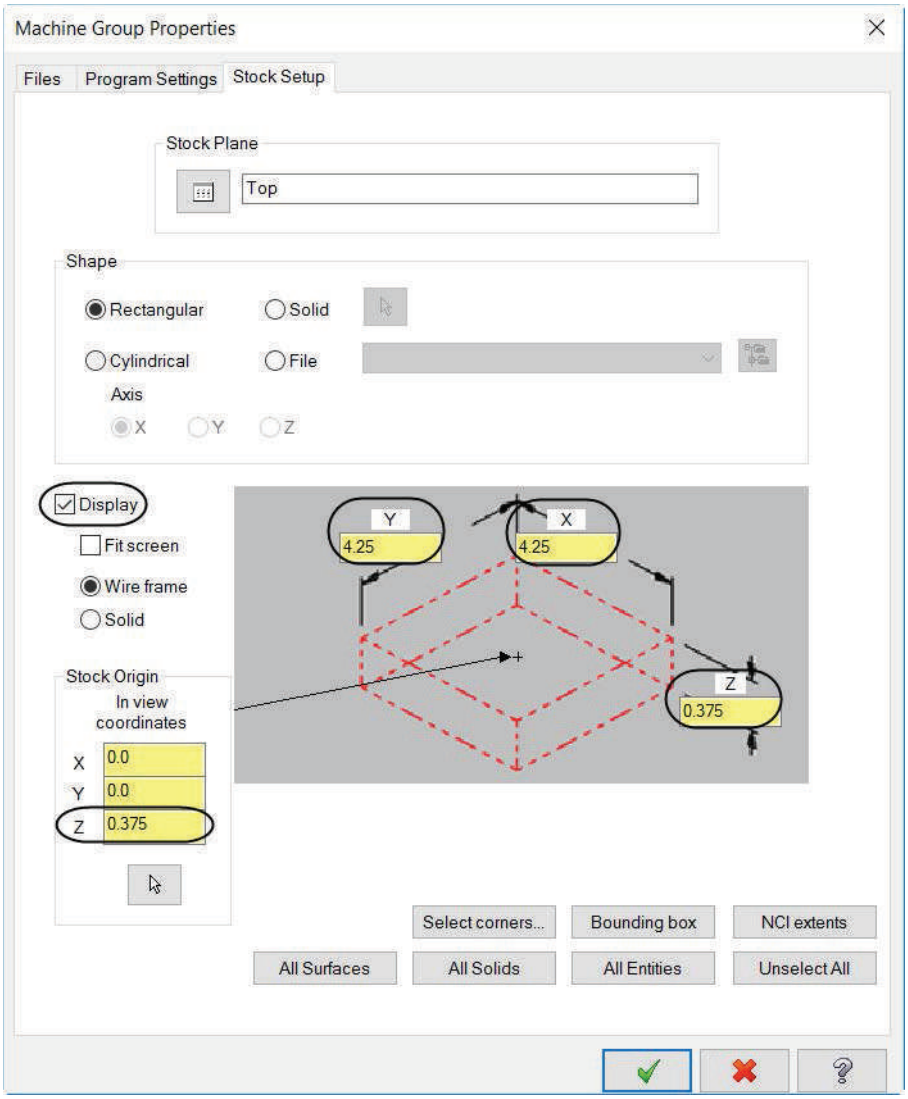


- Select the **Stock setup** to set the stock size.



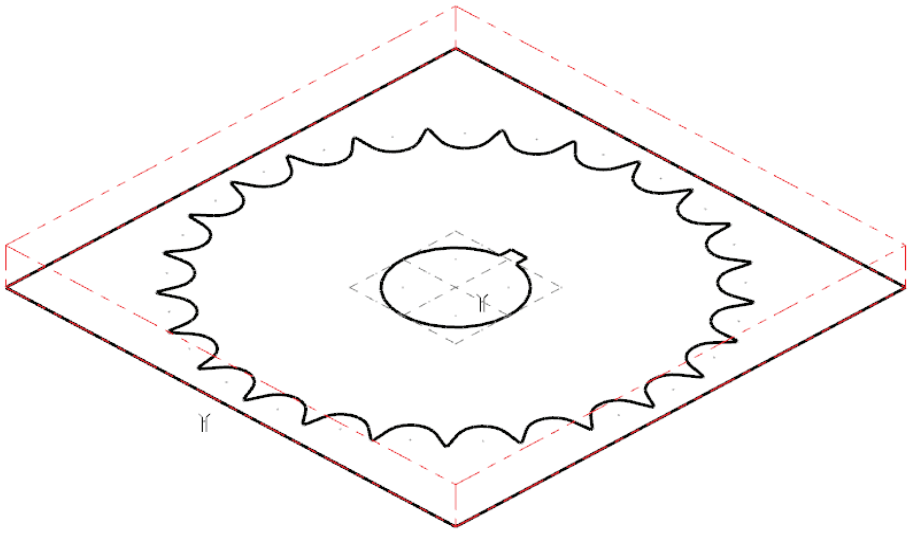
- Change the parameters to match Figure: 23.1.1.

Figure: 23.1.1



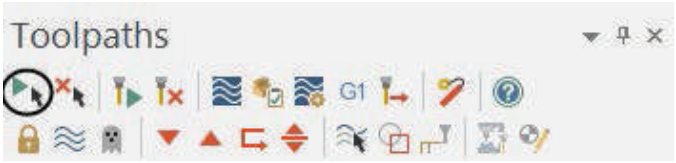
- Select the **OK** button to exit **Machine Group Properties**.

- The stock is displayed with phantom red lines and it should look as shown.

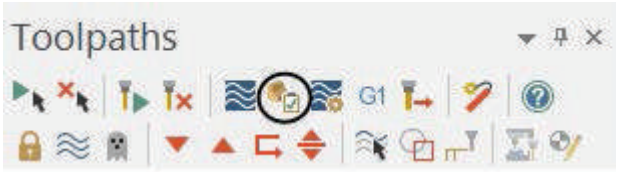


23.2 Verify the wirepaths

- From the **Toolpaths Manager**, click on the **Select all operation** icon to select both wirepaths.

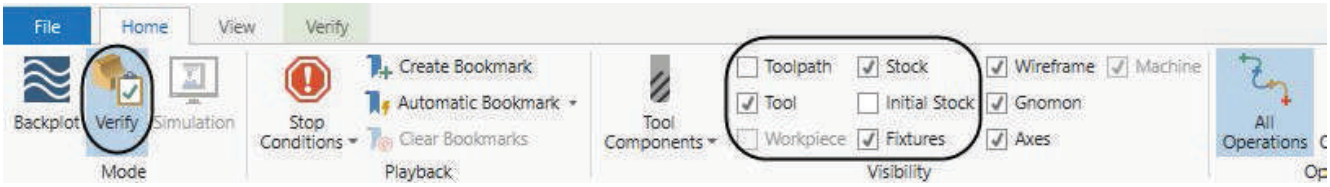


- Select the **Verify selected operations** button.

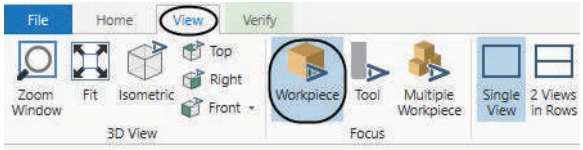


- From **Verify** change the settings for the **Visibility** as shown in Figure: 23.2.1.

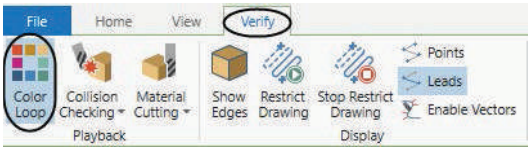
Figure: 23.2.1



- Select **View** tab and enable **Workpiece** as shown.



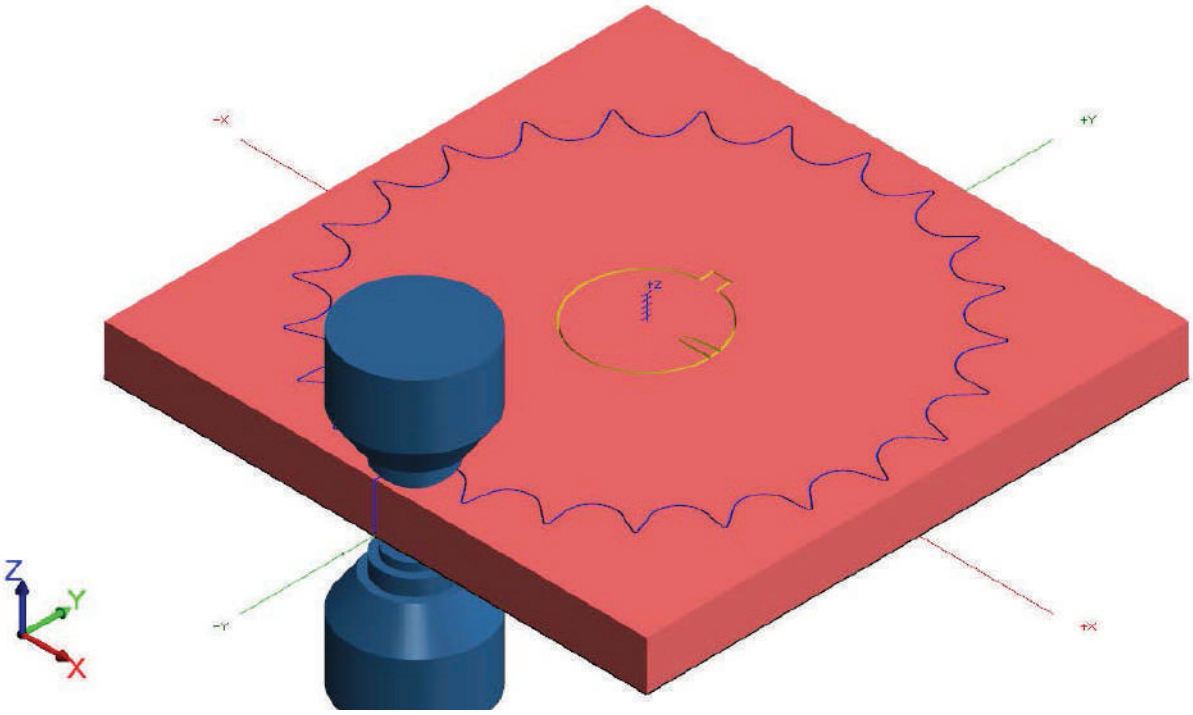
- Select **Verify** tab and enable **Color Loop** as shown.



- Right click and select the **Isometric** option.
- Right click and select the **Fit** option.
- Select the **Play** button to run **Verify**.

- The part should look as shown in Figure: 23.2.2.

Figure: 23.2.2

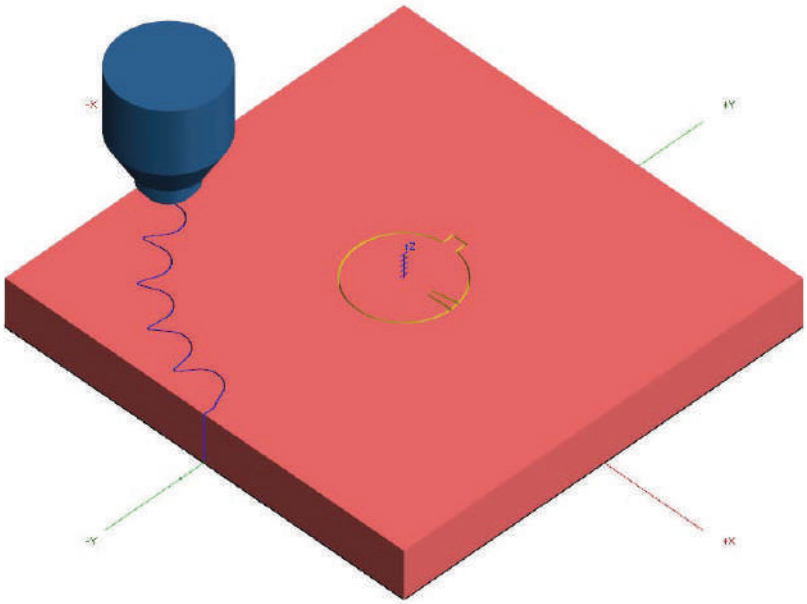


NOTE: To rotate the part, move the cursor to the center of the part and click and hold the mouse wheel and slowly move it in one direction.
To **Zoom In** or **Out** hold down the mouse wheel and scroll up or down as needed.

- Under the **View** tab, click on the **Isometric** icon

and then the **Fit** icon

to see the part in the original position if needed.
- To check the part step by step, click first on the **Start (Home)** icon.
- Click on the **Step Forward** icon 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.

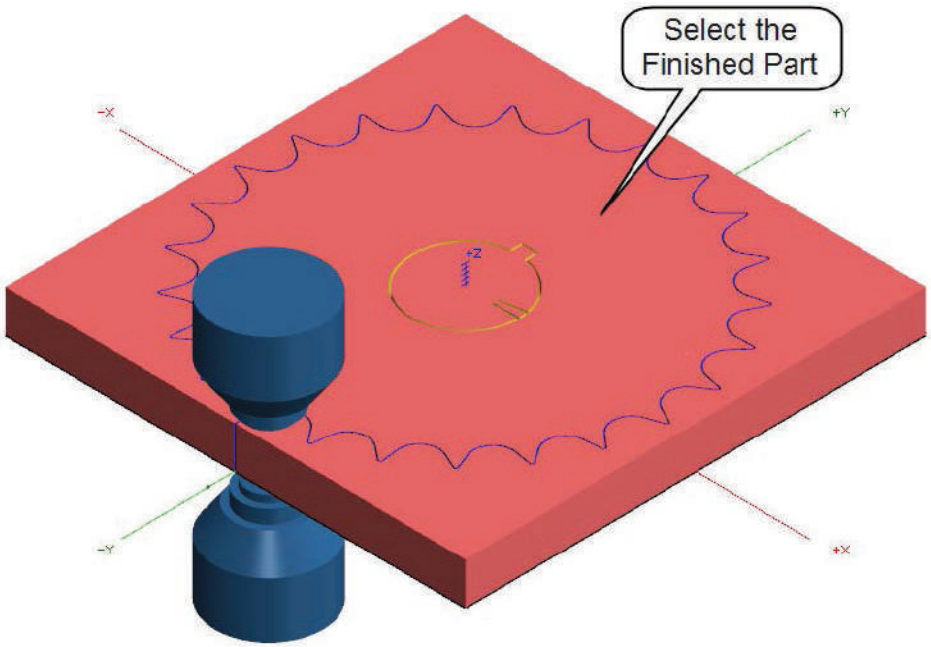
23.3 Chip removal

- Select the **Verify Tab** and select the **Keep Chips** icon as shown in Figure: 23.3.1.

Figure: 23.3.1

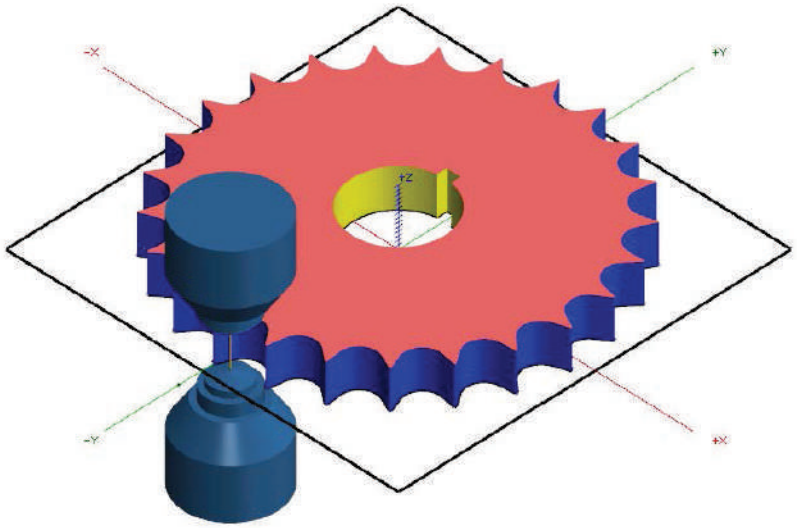
- Click on the part as shown in Figure: 23.3.2.

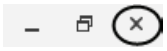
Figure: 23.3.2



- The part will appear as shown in Figure: 23.3.3.

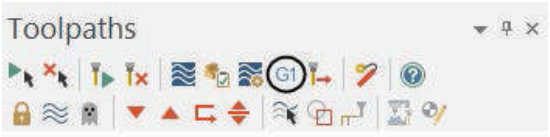
Figure: 23.3.3



- To exit **Mastercam Simulator** click on the **Close** icon. 

STEP 24: POST THE FILE TO GENERATE THE NC CODE

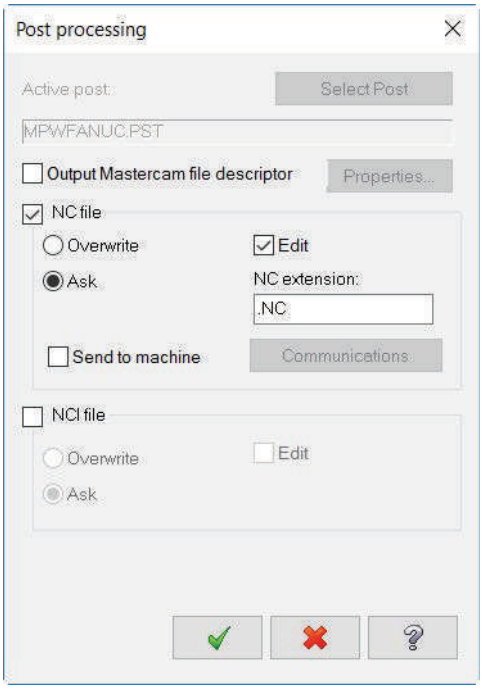
- Select **Post selected operations** button from **Toolpaths Manager**.



NOTE: The active **Post Processor** is a generic FANUC post processor.

- In the **Post processing** window, as shown in Figure: 24.0.1, if needed, make any necessary changes.

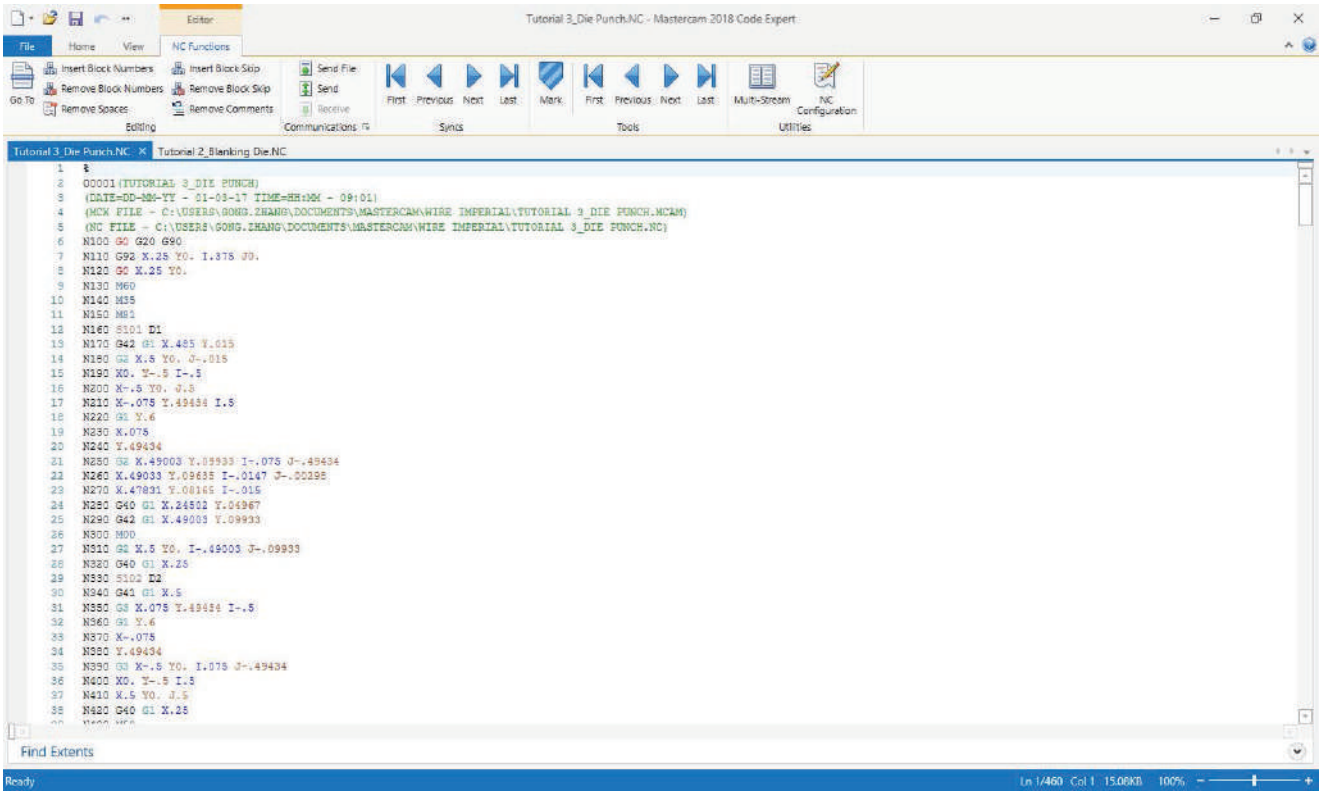
Figure: 24.0.1



- Select the **OK** button to continue.
- Save the NC file.

- The NC file will appear on the screen as shown in Figure: 24.0.2.

Figure: 24.0.2

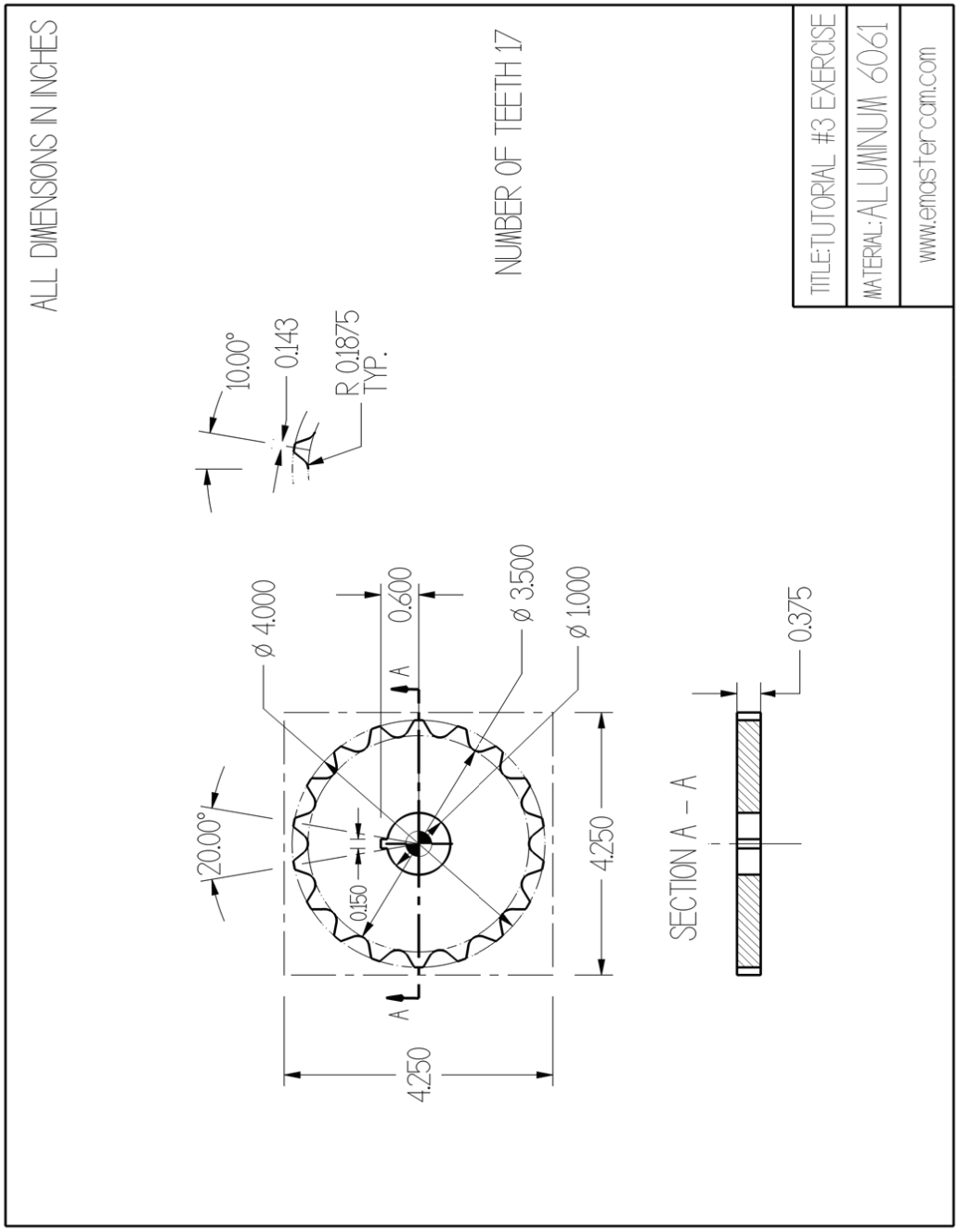


NOTE: The G code that you have created will appear on the screen.If the code looks okay, you can shut down the window without saving it. If you need to change the code, make sure you save it before you close the window.
How the program is sent to the machine depends on the shop setup.

- Select the "**X**" box at the upper right corner to exit the **Mastercam Code Expert.**

STEP 25: SAVE THE UPDATED MCAM FILE

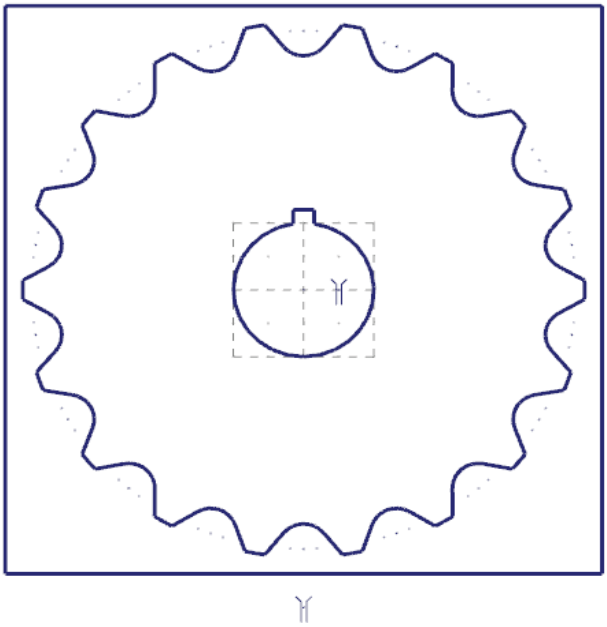
REVIEW EXERCISE -STUDENT PRACTICE



CREATE THE GEOMETRY FOR TUTORIAL #3 EXERCISE

Use these commands to create the geometry.

- Rectangle.
- Circle Center Point.
- Line Endpoint.
- Arc Tangent.
- Line Tangent.
- Line Parallel.
- Trim Entities.
- Mirror.
- Rotate
- Point Position - Thread Point.

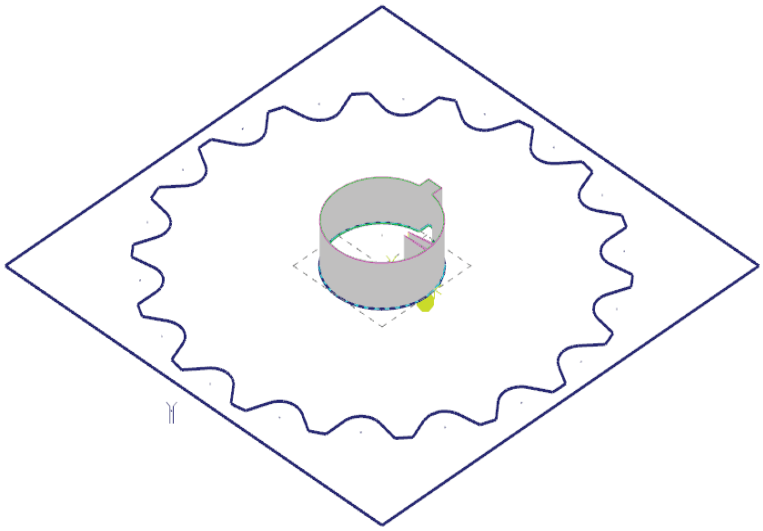


CREATE THE TOOLPATHS FOR TUTORIAL #3 EXERCISE

Create the Toolpaths for Tutorial #3 Exercise as per the instructions below.

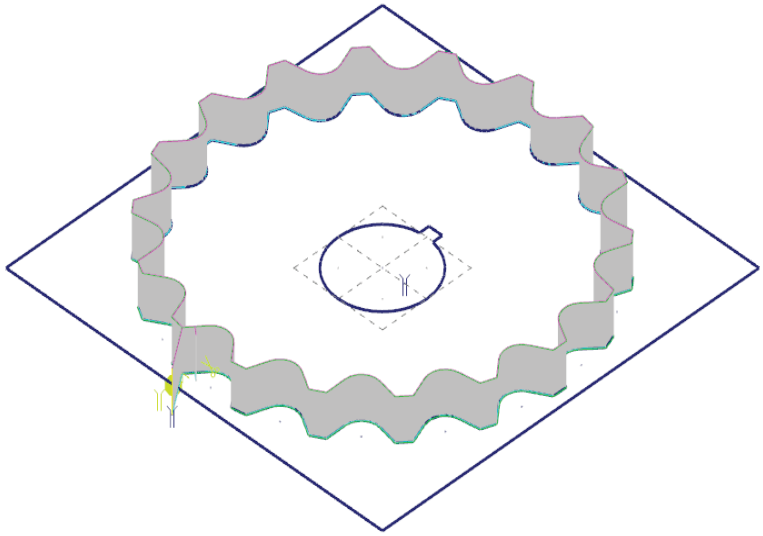
Machine The Inner Profile using Contour Wirepath.

- In order to complete this operation follow **page 171** to **page 184.**



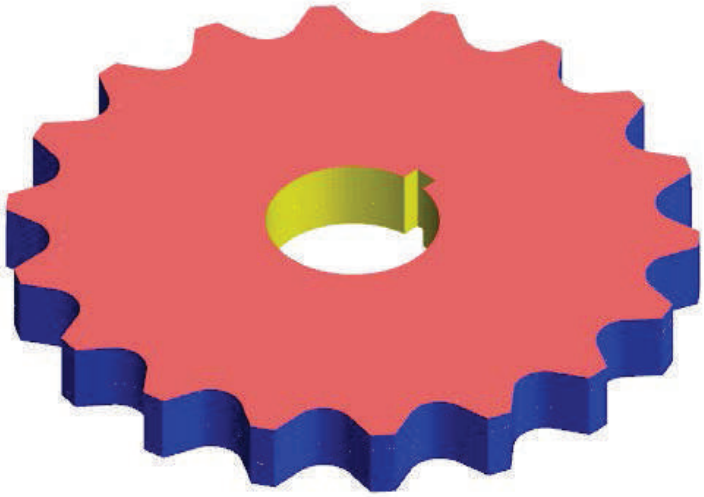
Machine The Outer Profile using Contour Wirepath.

- In order to complete this operation follow **page 187** to **page 198.**



Verify The Wirepath.

- Set the **Stock** according to the part.
- In order to complete this operation **Verify** the part and follow **page 201** to **page 206**.



NOTES:

TUTORIAL #3 QUIZ

- What command allows you to create a line tangent at an angle?

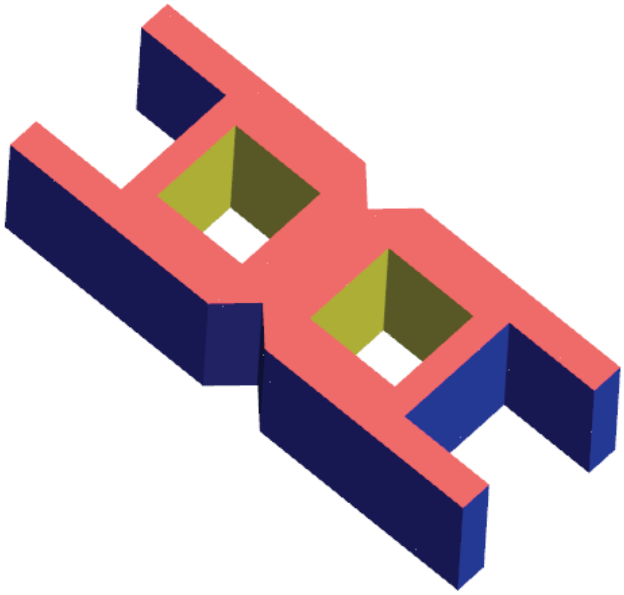
- What does Mirror command allow you to create?

- What does Divide command do?

- What is a Tab?

- What does Stop parameters allow you to set?

TUTORIAL #4



OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

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, Circle Center Point, Line Polar, Arc Tangent, Line Tangent, Trim, Mirror, Rotate, and Create Thread Point will be used.

Create the necessary Wirepaths to machine the part:

- Contour wirepath to cut the inner profile.
- Contour wirepath to cut the outer profile.

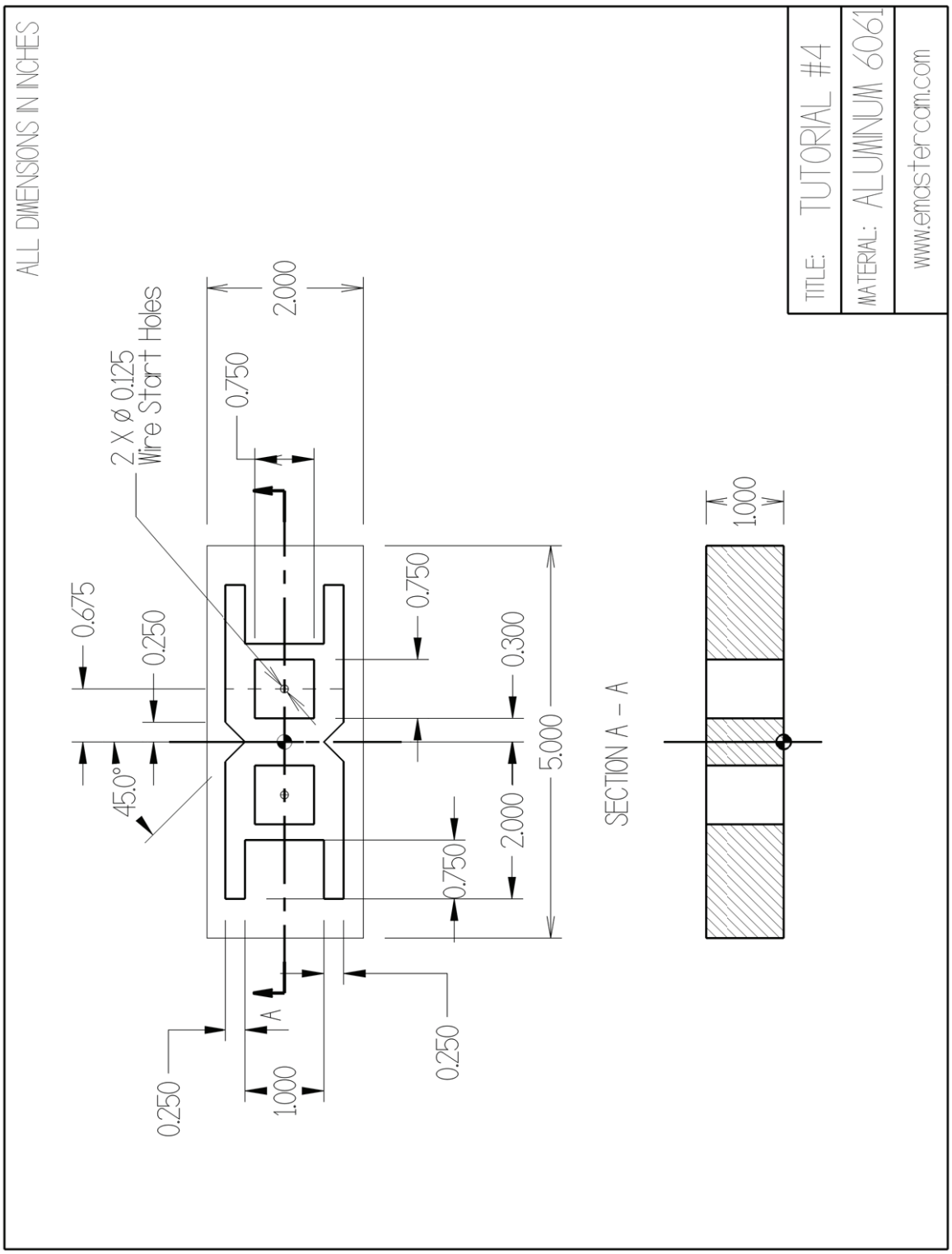
Backplot and the file:

- 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 then post process the file to obtain an NC file containing the necessary code for the machine.

This tutorial takes approximately one hour and 30 minutes to complete.


ALL DIMENSIONS IN INCHES
TITLE: TUTORIAL #4
MATERIAL: ALUMINUM 6061
www.emastercam.com
2 X ø 0.125
Wire Start Holes
2.000
0.750
0.675
0.250
0.750
0.300
5.000
45.0°
2.000
0.750
0.250
0.250
1.000
A
SECTION A - A
1.000

GEOMETRY CREATION

STEP 1: SETTING UP THE GRAPHIC USER INTERFACE

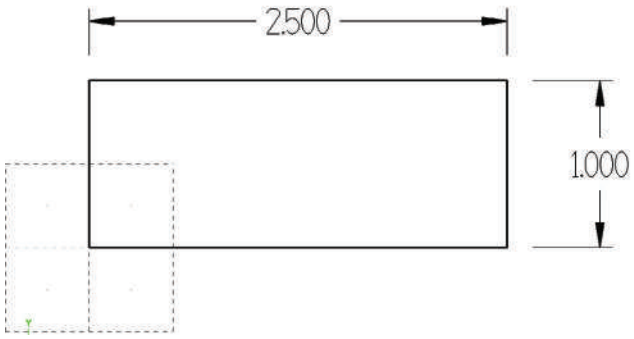
Please refer to the **Getting Started** section to set up the graphical user interface.

NOTE: The part is symmetric about both X and Y-Axis. You need to create just a quarter of the geometry and then mirror it about X and Y-Axis to create the entire geometry.

STEP 2: CREATE A RECTANGULAR SHAPE

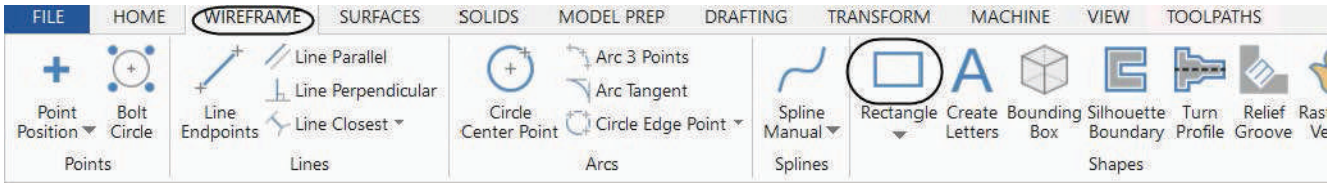
In this step you will create a rectangular shape given the width, the height, and the anchor point.

Step Preview:

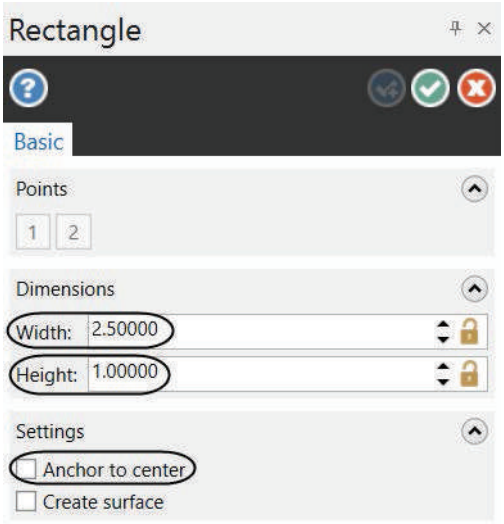


WIREFRAME

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

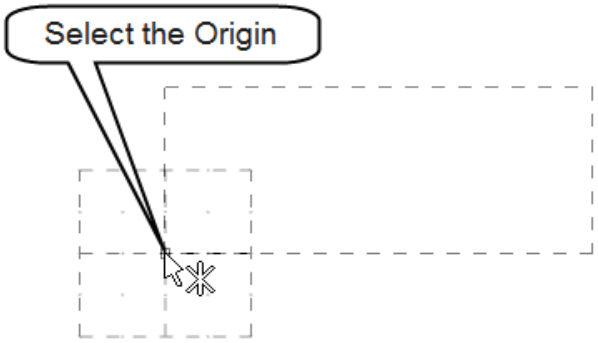


- Enter the **Width** and **Height** and, if needed, disable **Anchor to center** as shown.



- Press **Enter** after typing the values to see a preview of the rectangle.
- [Select position of base point]: Select the origin as shown in Figure: 2.0.1.

Figure: 2.0.1



- Make sure that when selecting the origin, the visual cue of the cursor changes as shown.

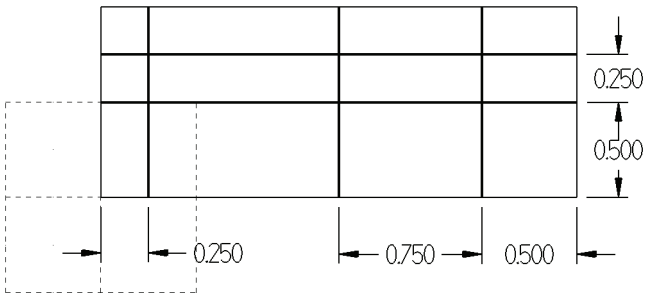
- Press **Alt+F1** to fit the drawing to the screen.
- Select the **OK** button to exit the command.

NOTE: While creating the 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 it first and then press **Delete** from the keyboard.

STEP 3: CREATE PARALLEL LINES FOR THE OUTSIDE PROFILE

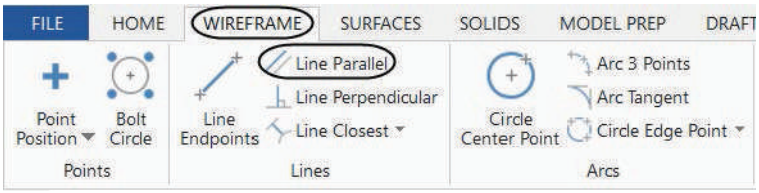
In this step you will create parallel lines that will be used to create the outside profile of the part.

Step Preview:

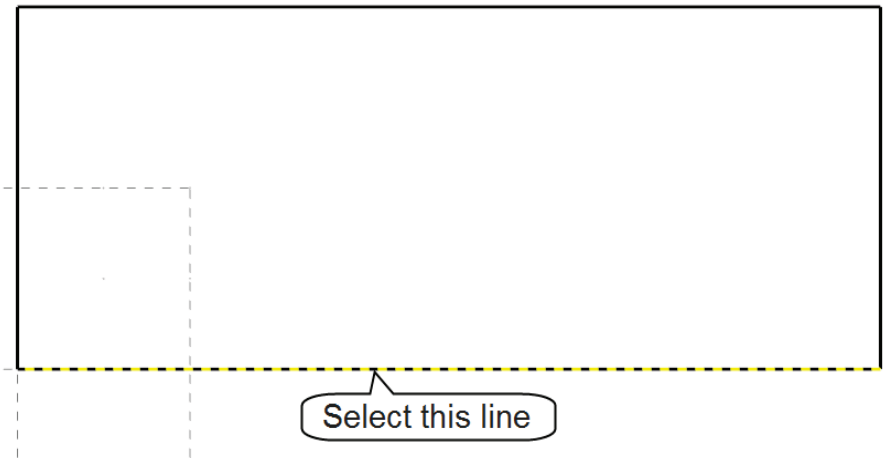


WIREFRAME

- From the **Lines** group, select the **Line Parallel** icon.

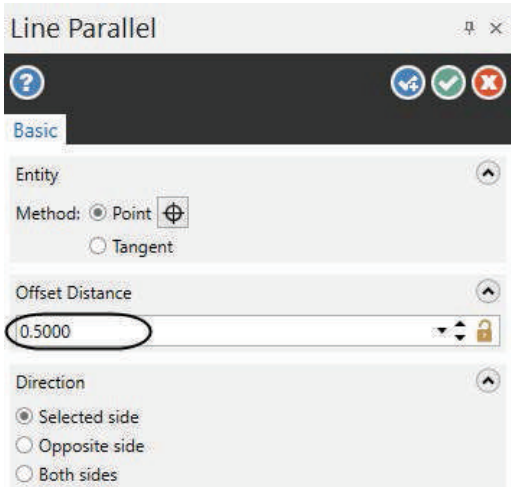


- [Select a line]: Select the lower horizontal line of the rectangle as shown.



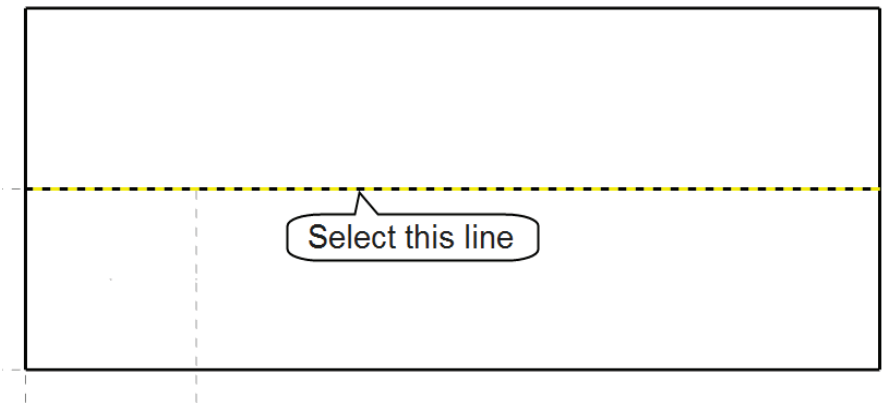
- [Select the point to place a parallel line through]: Click somewhere above the selected line.

- Enter the **Offset Distance 0.5** as shown. Press **Enter**.



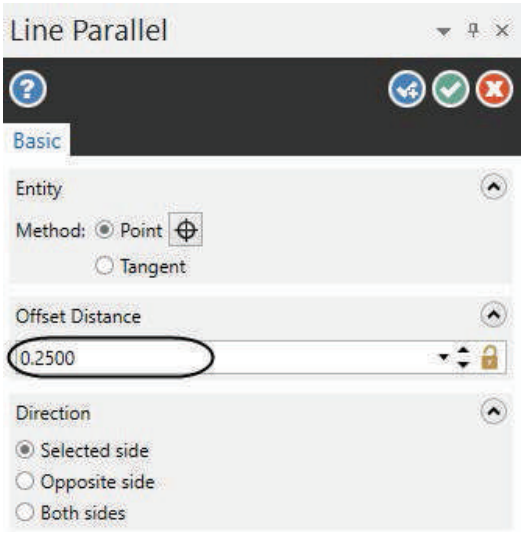
- Select **OK and Create New Operation** to continue with the same command.
- [Select a line]: Select the last line that you created as shown in Figure: 3.0.1.

Figure: 3.0.1



- [Select the point to place a parallel line through]: Click somewhere above the selected line.

- Enter the **Offset Distance 0.25** as shown. Press **Enter**.



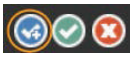
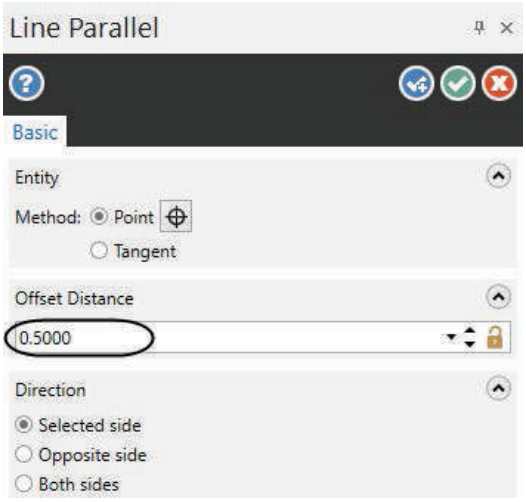
- Select **OK and Create New Operation** to continue with the same command. 
- [Select a line]: Select the right side vertical line of the rectangle as shown in Figure: 3.0.2.

Figure: 3.0.2

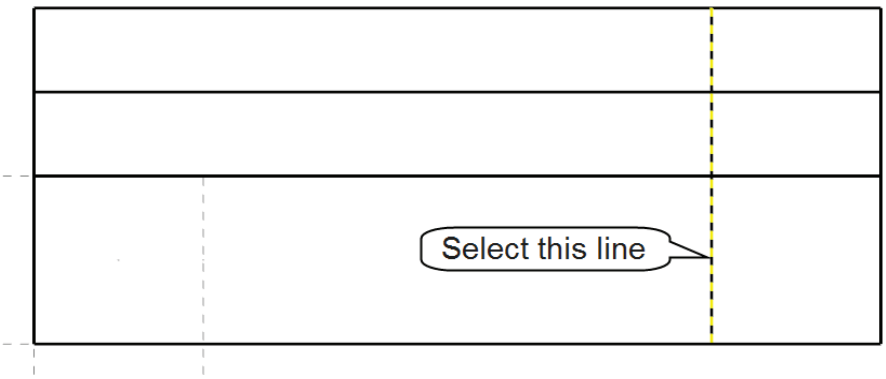
- [Select the point to place a parallel line through]: Click somewhere to the left of the selected line.

- Enter the **Offset Distance 0.5** as shown. Press **Enter**.



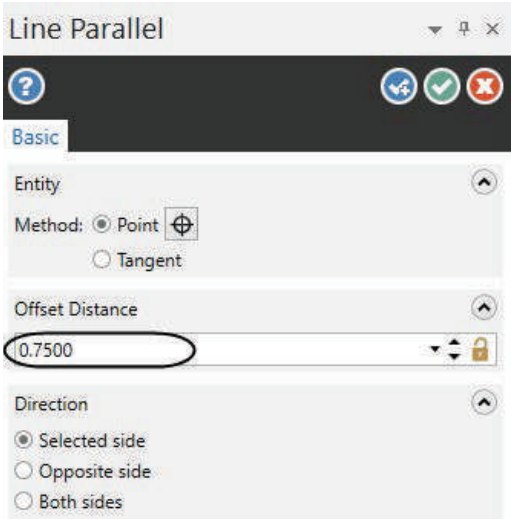
- Select **OK and Create New Operation** to continue with the same command.
- [Select a line]: Select the new vertical line as shown in Figure: 3.0.3.

Figure: 3.0.3



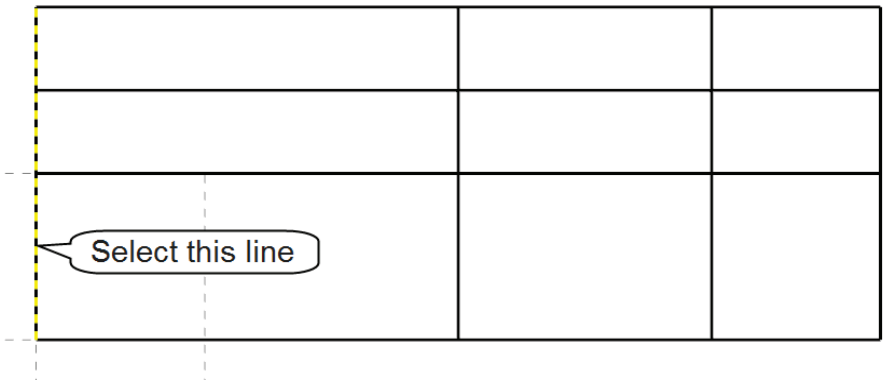
- [Select the point to place a parallel line through]: Click somewhere to the left of the selected line.

- Enter the **Offset Distance 0.75** as shown. Press **Enter**.



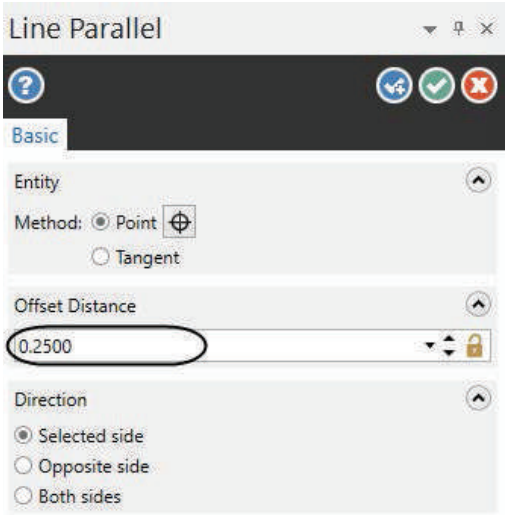
- Select **OK and Create New Operation** to continue with the same command.
- [Select a line]: Select the left side vertical line of the rectangle as shown in Figure: 3.0.4.

Figure: 3.0.4



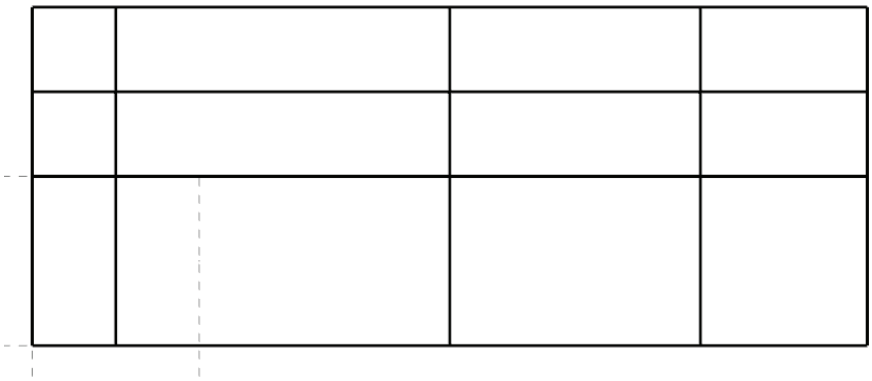
- [Select the point to place a parallel line through]: Click somewhere to the right of the selected line.

- Enter the **Offset Distance 0.25** as shown.



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

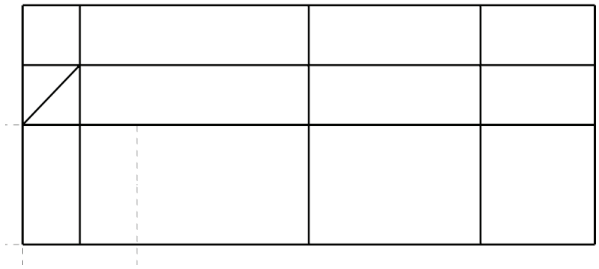
- The geometry should look as shown.



STEP 4: CREATE A LINE ENDPOINTS

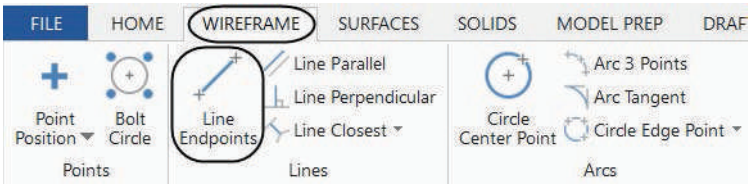
In this step you will create a line given the endpoints of the line.

Step Preview:



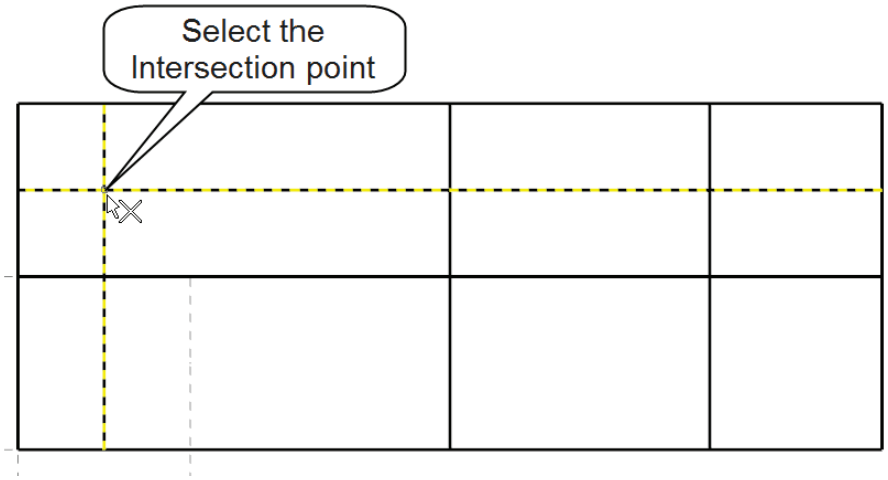
WIREFRAME

- From the **Lines** group, select the **Line Endpoints** icon.

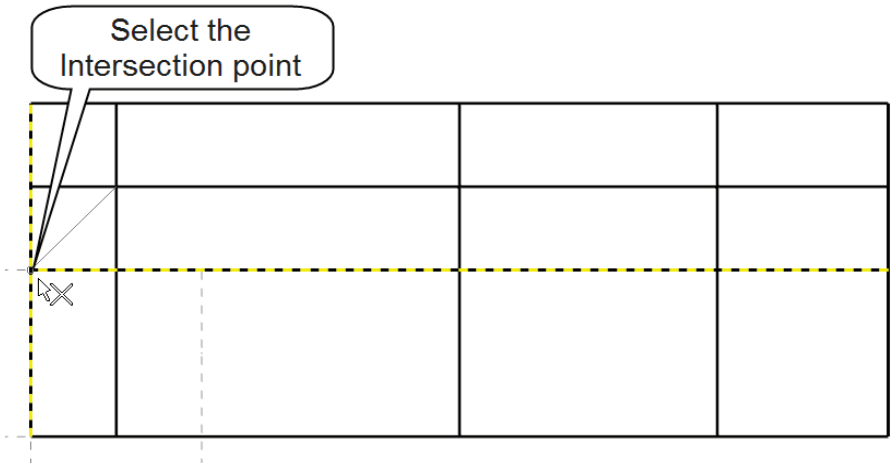


- [Specify the first endpoint]: Select the intersection point as shown in Figure: 4.0.1.

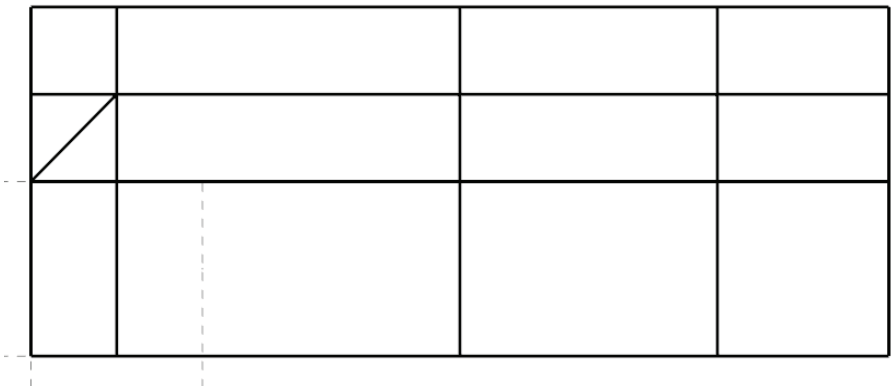
Figure: 4.0.1



- [Specify the second endpoint]: Select the intersection point (or endpoint) as shown.



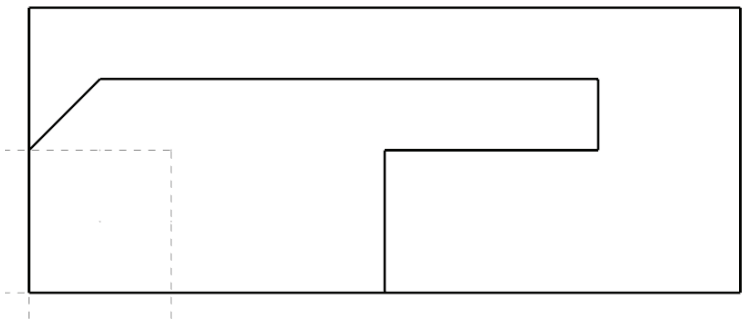
- Select the **OK** button to exit the command.
- The geometry should look as shown.



STEP 5: CLEAN UP THE OUTSIDE PROFILE USING TRIM TWO ENTITIES

Clean up the geometry by trimming the entities to their intersection point.

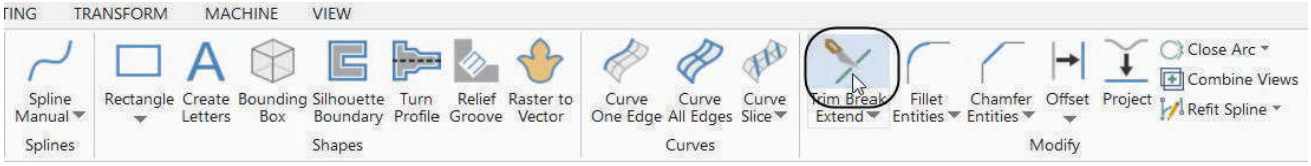
Step Preview:



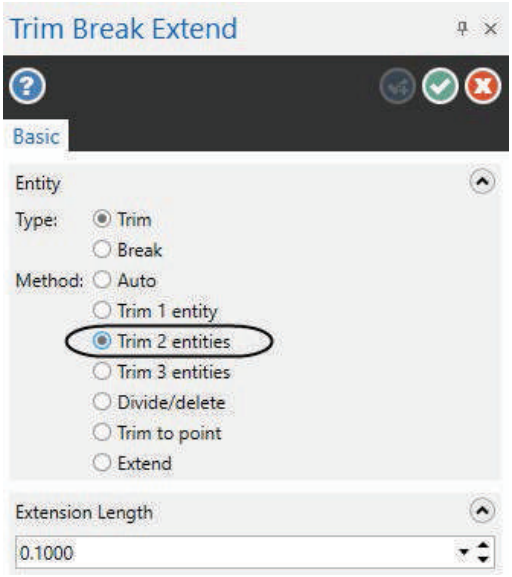
NOTE: During the trimming process it is very important to select the entities exactly at the locations that are shown in the graphics on the following pages.
Mastercam's auto-preview feature displays the result of the selected function as you move the mouse over the final entity selection. A solid line represents what will be created, while a dashed line represents what will be removed.

WIREFRAME

- From the **Modify** group, select the **Trim Break Extend** icon as shown.

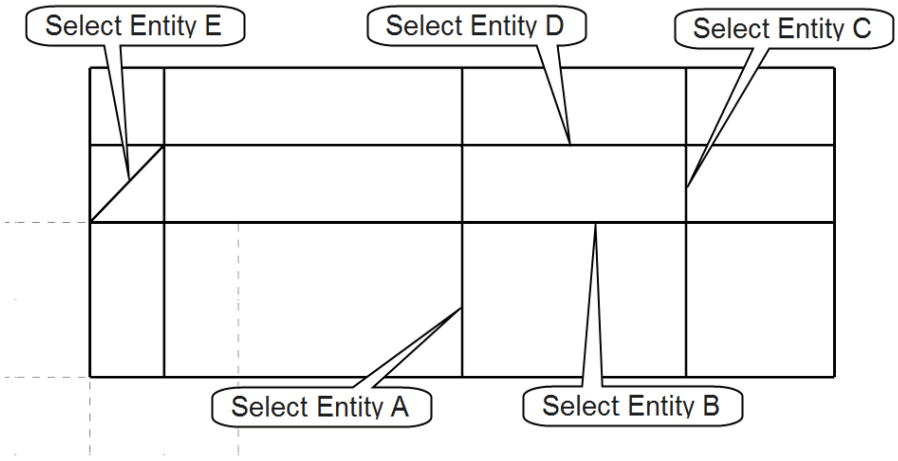


- Enable **Trim 2 entities** as shown.



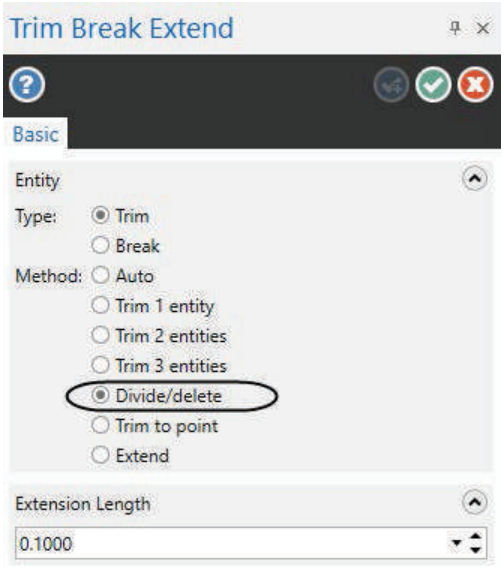
- [Select the entity to trim/extend]: Click on Entity A as shown in Figure: 5.0.1.
- [Select the entity to trim/extend to]: Click on Entity B as shown in Figure: 5.0.1.

Figure: 5.0.1



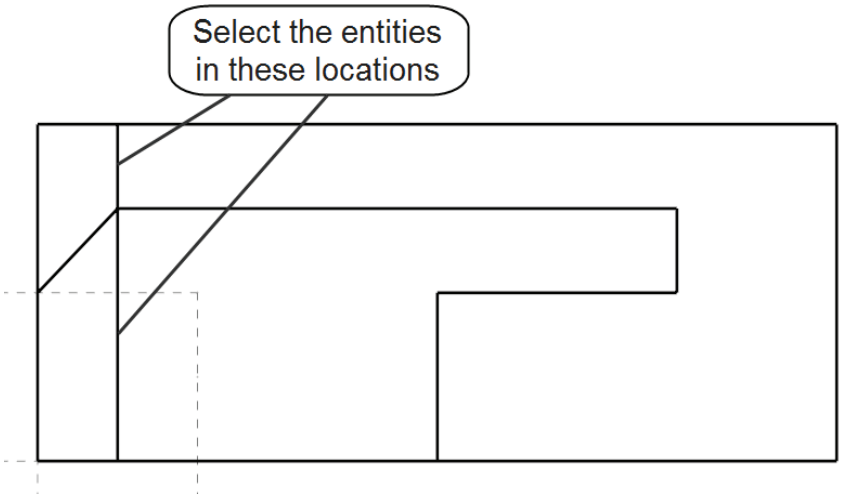
- [Select the entity to trim/extend]: Click on Entity B again as shown in Figure: 5.0.1.
- [Select the entity to trim/extend to]: Click on Entity C as shown in Figure: 5.0.1.
- [Select the entity to trim/extend]: Click on Entity C again.
- [Select the entity to trim/extend to]: Click on Entity D as shown in Figure: 5.0.1.
- [Select the entity to trim/extend]: Click on Entity D again.
- [Select the entity to trim/extend to]: Click on Entity E as shown in Figure: 5.0.1.

- Enable **Divide/delete** as shown.



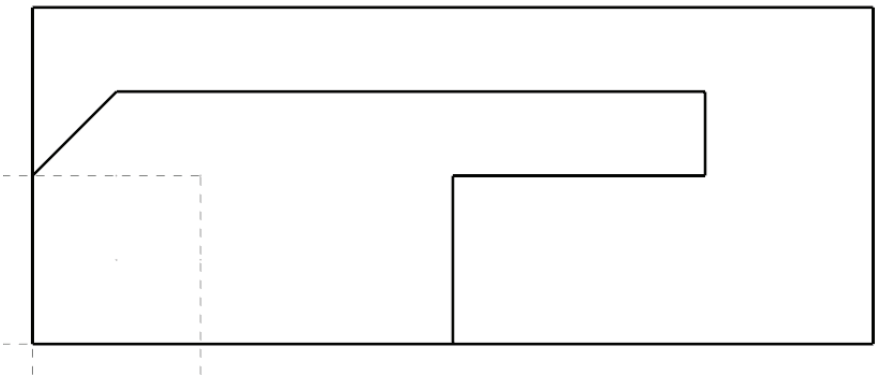
- Select the entities to delete them, at the same locations as shown in Figure: 5.0.2.

Figure: 5.0.2



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

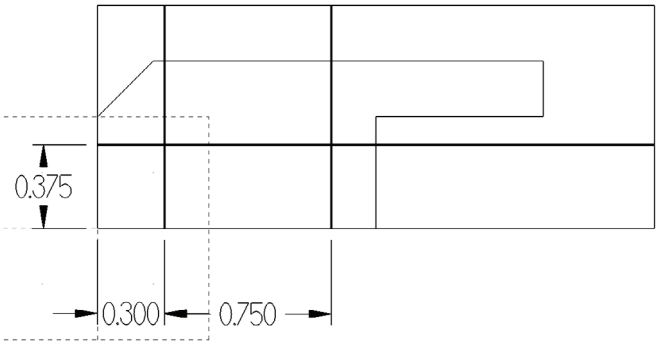
- The geometry should look as shown.



STEP 6: CREATE PARALLEL LINES FOR THE INSIDE PROFILE

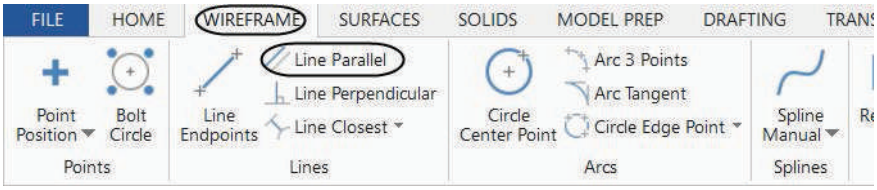
In this step you will create parallel lines that will be used to create the inside profile.

Step Preview:



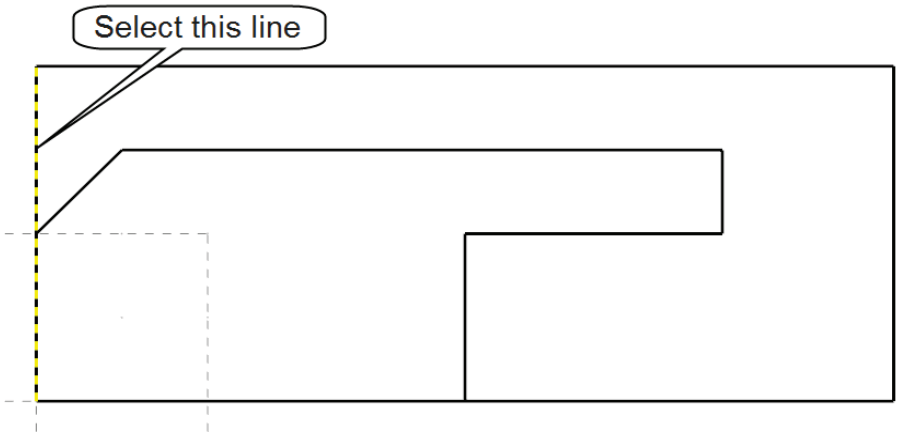
WIREFRAME

- From the **Lines** group, select the **Line Parallel** icon.

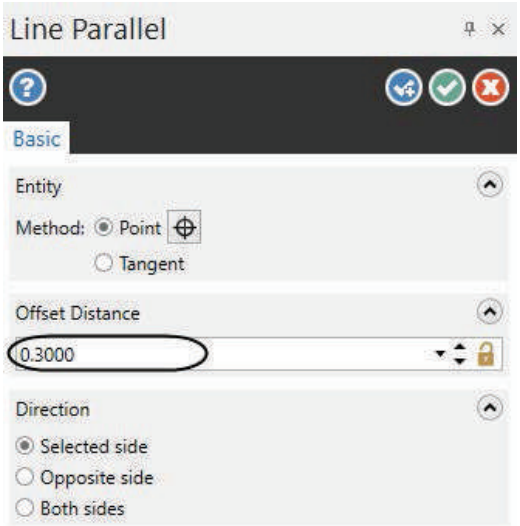


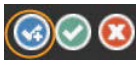
- [Select a line]: Select the left side vertical line of the rectangle as shown in Figure: 6.0.1.

Figure: 6.0.1



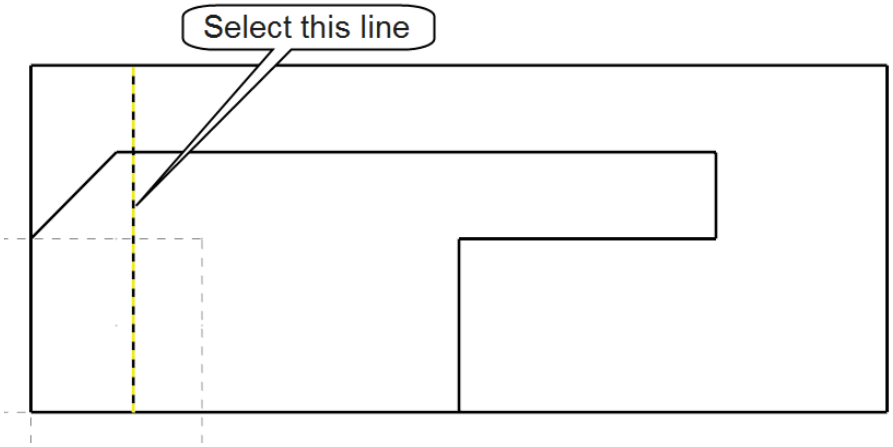
- [Select the point to place a parallel line through]: Click somewhere to the right of the selected line.
- Enter **Offset Distance 0.3** as shown. Press **Enter**.



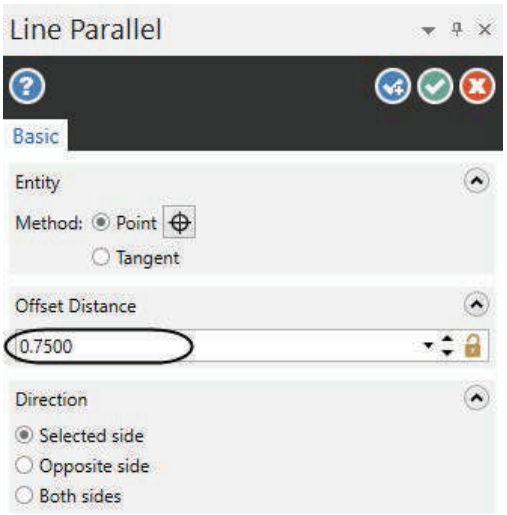
- Select **OK and Create New Operation** to stay in the command. 

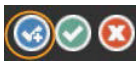
- [Select a line]: Select the last vertical line that you created as shown in Figure: 6.0.2.

Figure: 6.0.2



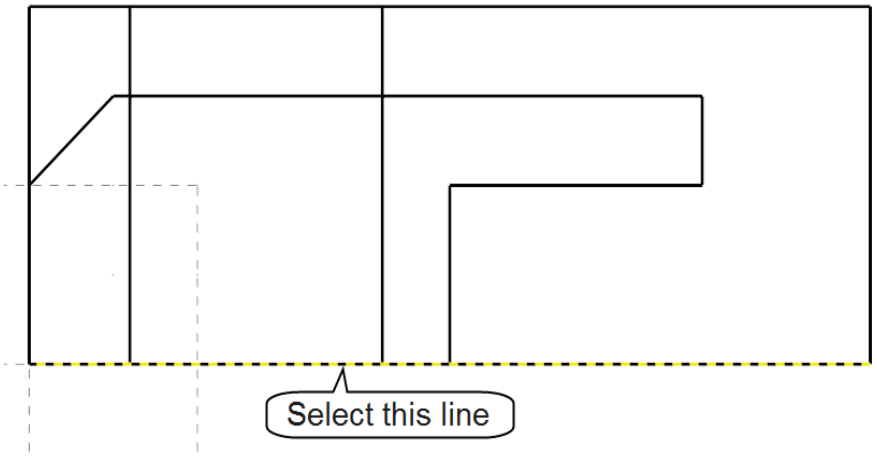
- [Select the point to place a parallel line through]: Click somewhere to the right of the selected line.
- Enter **Offset Distance 0.75** as shown. Press **Enter**.



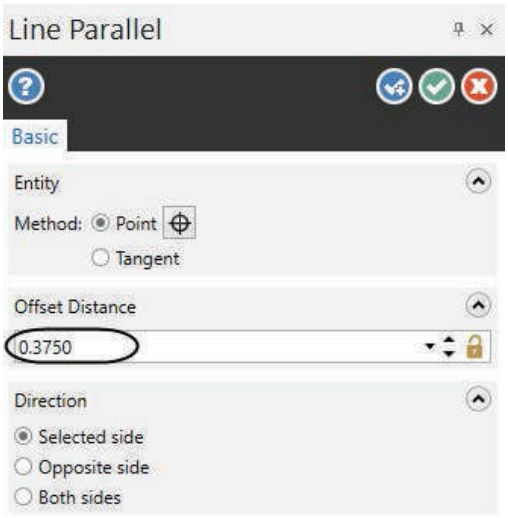
- Select **OK and Create New Operation** to stay in the command. 

- [Select a line]: Select the lower horizontal line of the rectangle as shown in Figure: 6.0.3.

Figure: 6.0.3

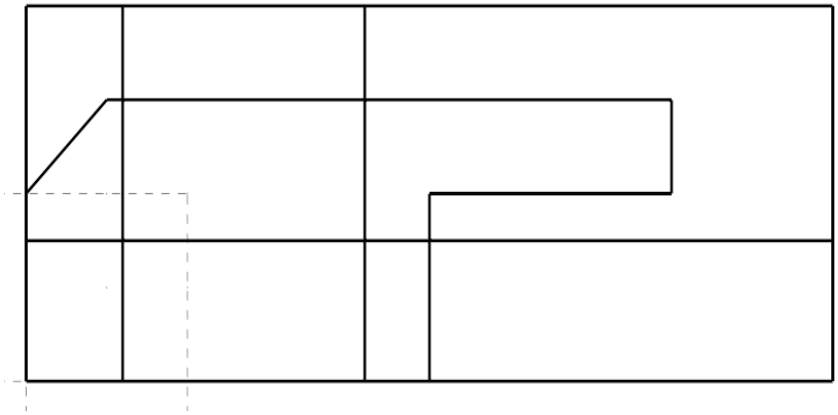


- [Select the point to place a parallel line through]: Click somewhere above the selected line.
- Enter **Offset Distance 0.375** as shown. Press **Enter**.



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

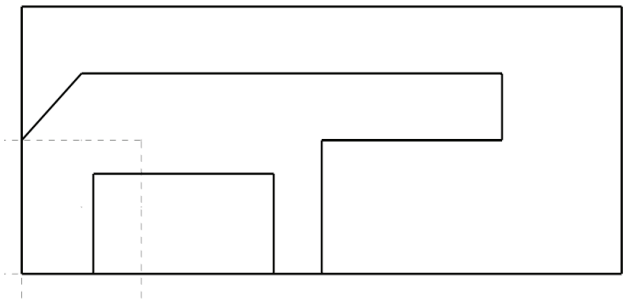
- The geometry should look as shown.



STEP 7: CLEAN THE INSIDE PROFILE USING TRIM THREE ENTITIES

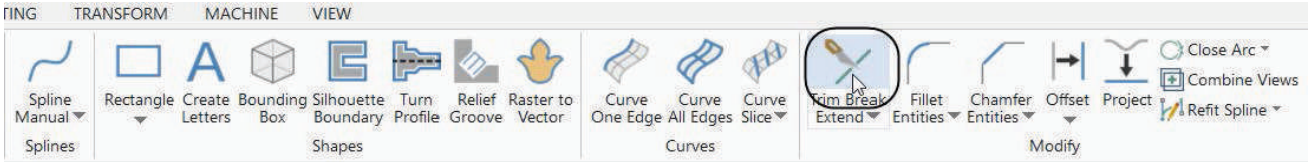
Clean up the geometry by trimming the entities to their intersection point.

Step Preview:

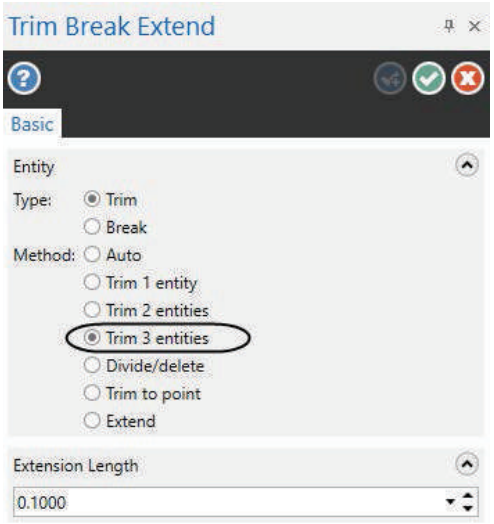


WIREFRAME

- From the **Modify** group, select **Trim Break Extend** as shown.

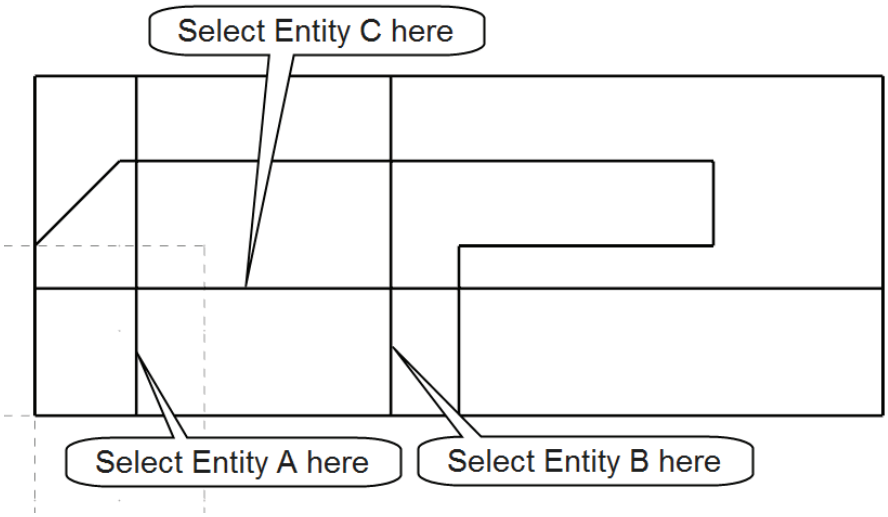


- Enable **Trim 3 entities** as shown.

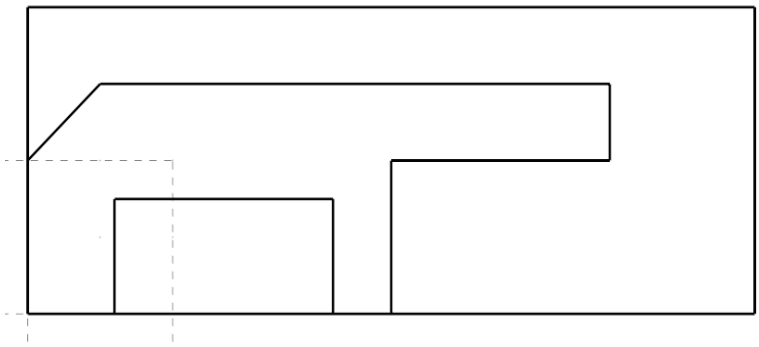


- [Select the first entity to trim/extend]: Click on Entity A as shown in Figure: 7.0.1.
- [Select the second entity to trim/extend]: Click on Entity B as shown in Figure: 7.0.1.
- [Select the second entity to trim/extend to]: Click on Entity C as shown in Figure: 7.0.1.

Figure: 7.0.1



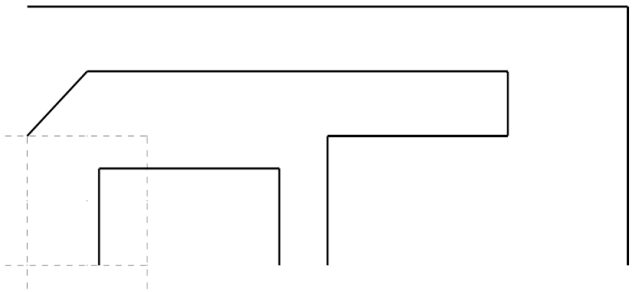
- The geometry should look as shown.



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

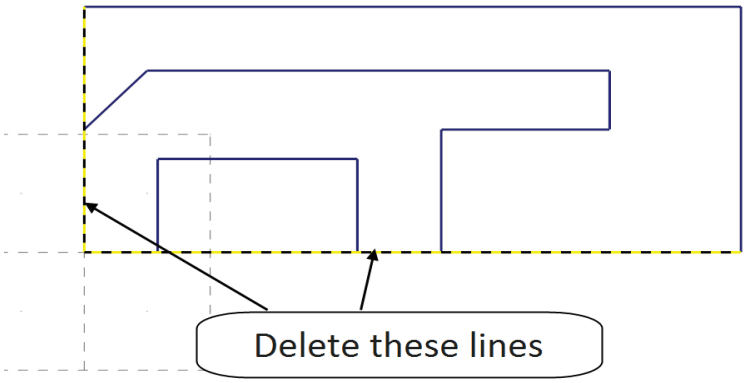
STEP 8: DELETE THE CONSTRUCTION LINES

Step Preview:



- Select the two lines as shown in Figure: 8.0.1.

Figure: 8.0.1

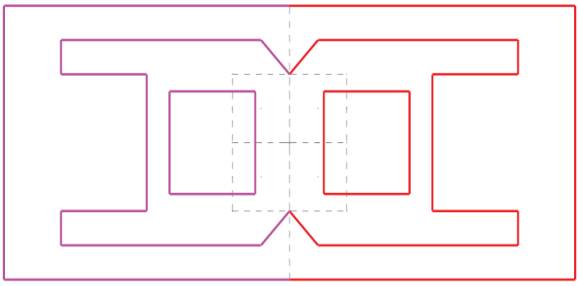


- Press **Delete** from the keyboard to delete these lines.

STEP 9: MIRROR THE GEOMETRY ABOUT X-AXIS AND Y-AXIS

The **Mirror** command allows you to create mirror images of geometric and drafting entities by reflecting them symmetrically with respect to a defined axis or point, including selected line or theoretical line defined between two points.

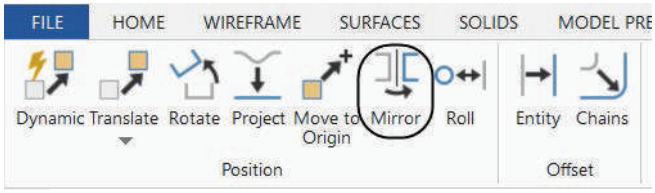
Step Preview:



NOTE: To select all entities for mirror, we are using the window selection. You select one corner of the window and then the opposite corner. Make sure that all entities are completely inside of the window before clicking in the opposite corner.

TRANSFORM

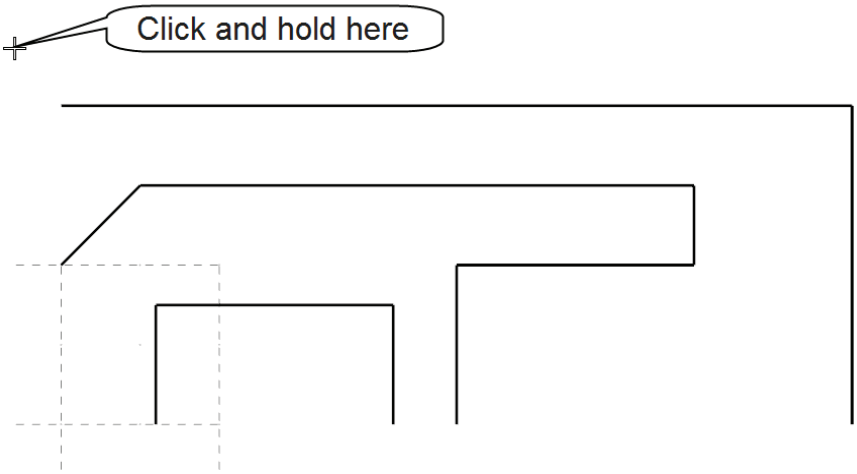
- From the **Position** group, select the **Mirror** icon.



- Hover the cursor in the center of the geometry and scroll down the mouse wheel to unzoom.

- [Mirror: select entities to mirror]: Left mouse click and hold in the upper left area of the window outside of the geometry as shown in Figure: 9.0.1.

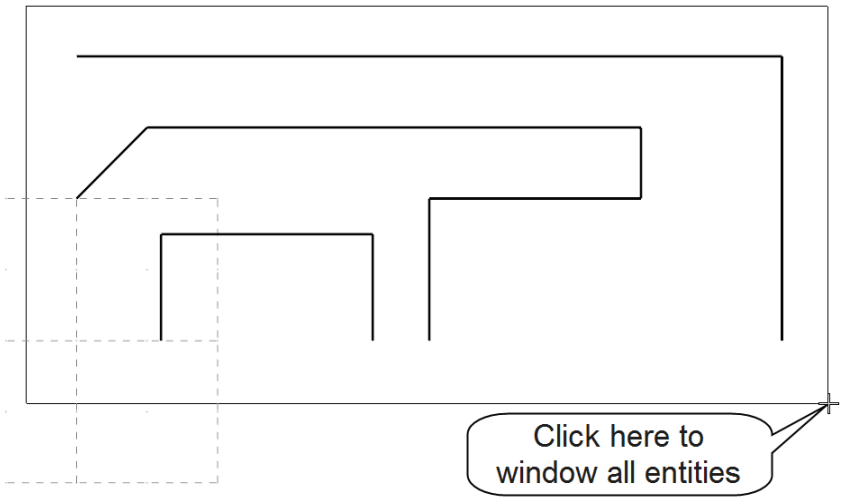
Figure: 9.0.1

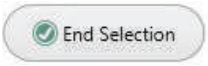


NOTE: The upper point has to be completely outside the geometry.

- Make a window that incorporates the entire geometry by dragging the cursor and clicking in the opposite corner as shown in Figure: 9.0.2.

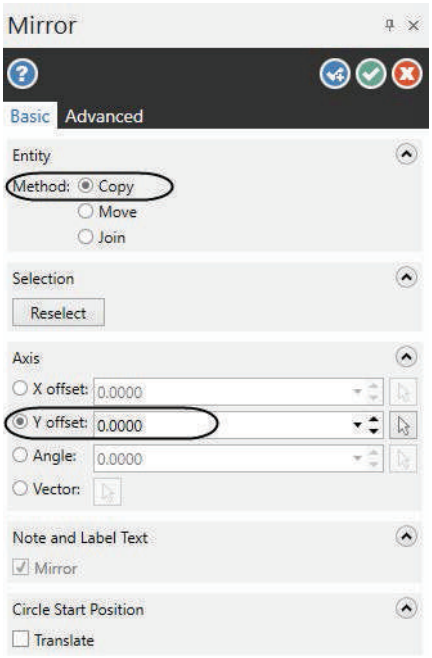
Figure: 9.0.2



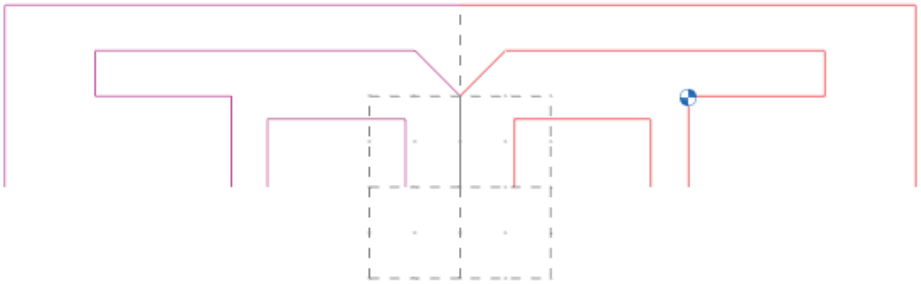
- Select the **End Selection** button (or press **Enter**) to finish the selection. 

- In the **Mirror** panel, make sure that **Copy** is selected and then enable **Y offset** as shown in Figure: 9.0.3.

Figure: 9.0.3



- Press **Alt+F1** to fit the drawing to the screen.
- The preview should look as shown.

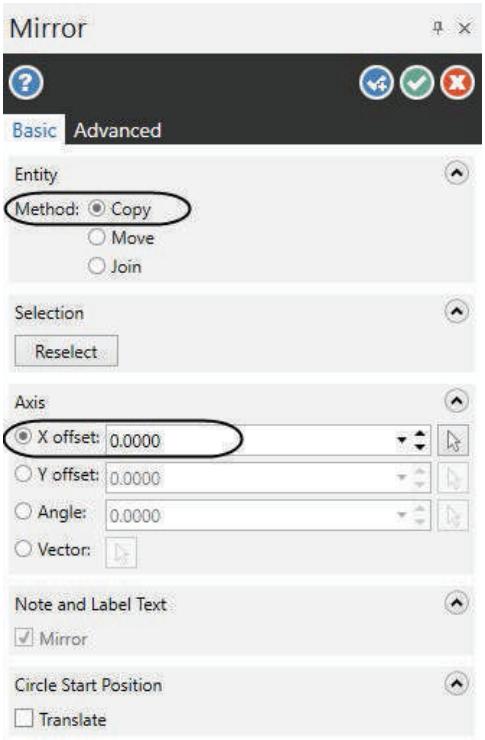


NOTE: When performing a transform function, Mastercam creates a temporary group from the original, highlighted in red, and a result from the transformed entities, highlighted in purple. These two colors stay in effect until you use the **Screen**, **Clear Colors** function or perform another transform function.

- Select the **OK and Create New Operation** button to continue with the **Mirror** command.
- [Mirror: select entities to mirror]: Unzoom the geometry and make a window to select the geometry in the same way as shown previously.
- Press **Enter** to continue.

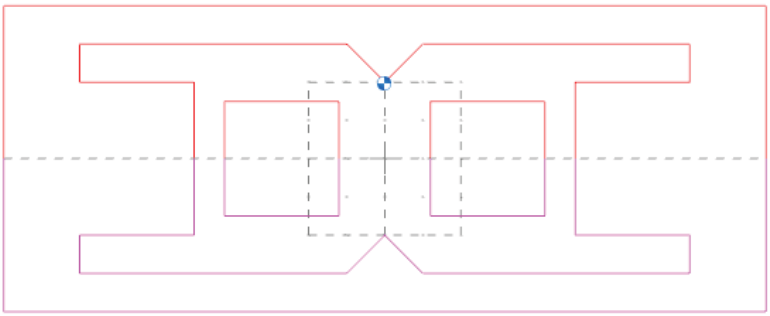
- In the **Mirror** panel, make sure that **Copy** is selected and then enable **X offset** as shown in Figure: 9.0.4.

Figure: 9.0.4



- Press **Alt+F1** to fit the drawing to the screen.

- The preview should look as shown.

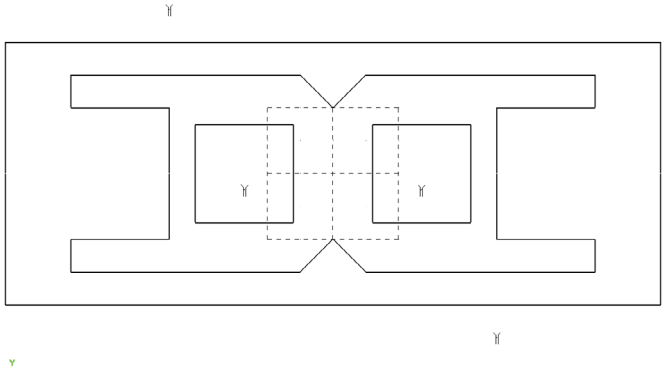


- Select the **OK** button to exit the **Mirror** panel.
- Right mouse click in the graphics window and select **Clear Colors** as shown.

STEP 10: CREATE THE THREAD POINTS

Thread Point is the point where the machine threads the wire, usually a pre-drilled hole in the material. The wire cuts from the thread position to the start of the chain based on the settings for lead ins and lead outs. See **Getting Started** to learn how to create the Wire utilities toolbar.

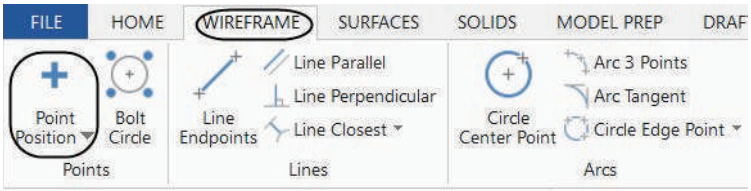
Step Preview:



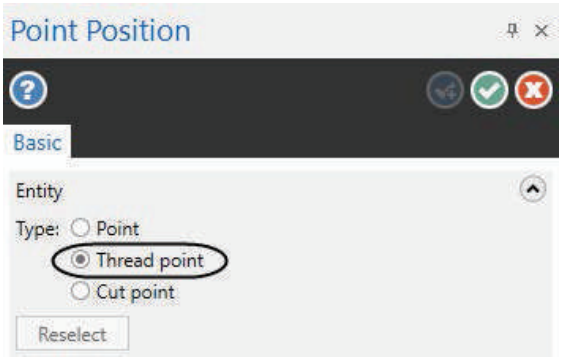
10.1 Create the Thread Point for the outside profile

WIREFRAME

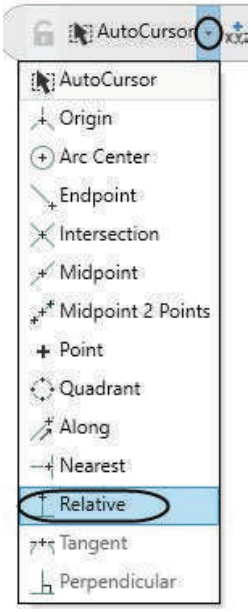
- From the **Points** group, select the **Point Position** icon.



- Select **Thread Point** as shown.

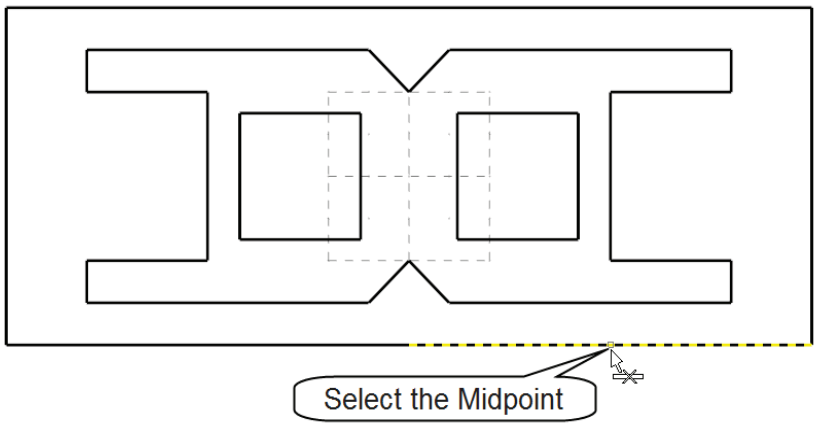


- From the **AutoCursor** drop down list, select **Relative**.

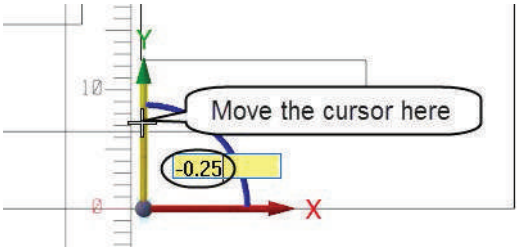


- [Enter a known point or change to Along mode]: Select the midpoint of the line as shown in Figure: 10.1.1.

Figure: 10.1.1

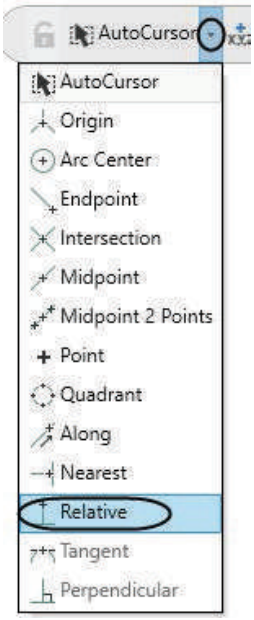


- Make sure that when selecting the midpoint, the visual cue of the cursor changes as shown.
- Hover the cursor above the **Y axis** as shown, then enter **-0.25** to create the point below the line.

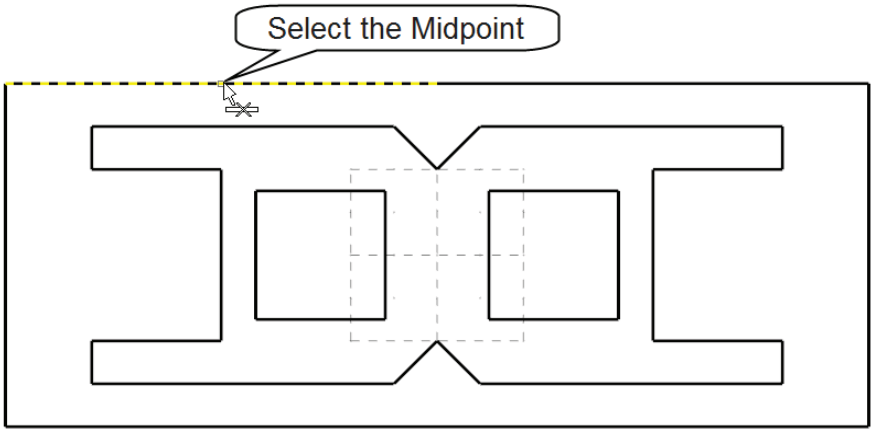


- Press **Enter** three times to create the point.

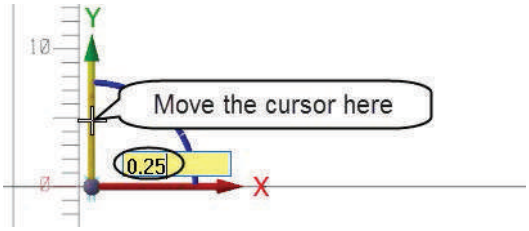
- From the **AutoCursor** drop down list, select **Relative**.

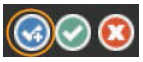


- [Pick a known point to use as a reference]: Select the Midpoint of the line as shown.

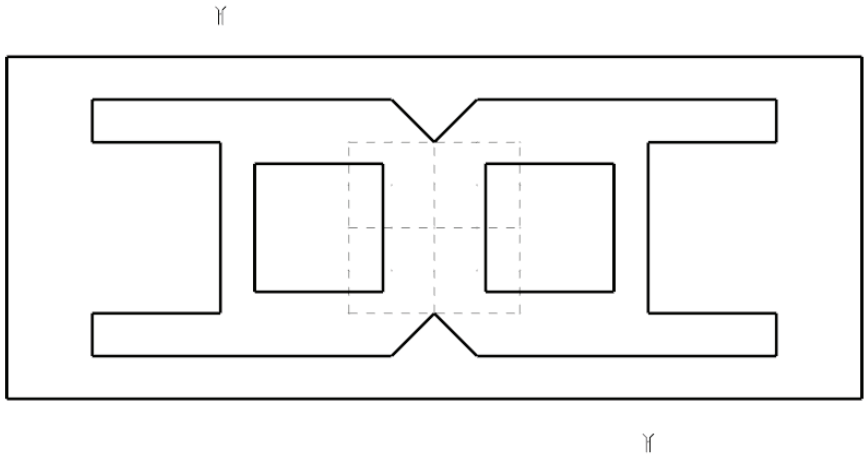


- Move the cursor to the **Y axis** as shown, enter **0.25** to create the point below the line.



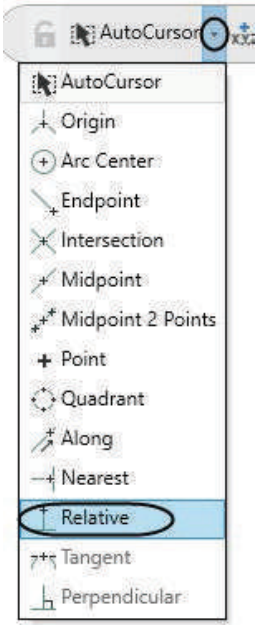
- Press **Enter** three times to create the point.
- Select the **OK and Create New Operation** to complete the command. 

- The geometry should look as shown.



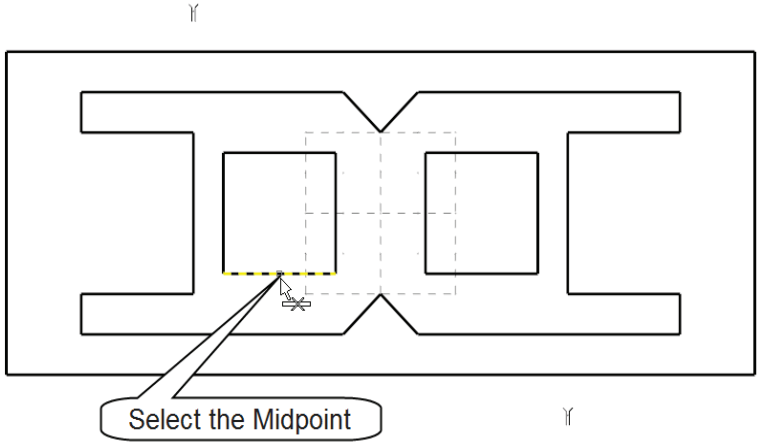
10.2 Create the Thread Point for inside profile

- From the **AutoCursor** drop down list, select **Relative**.

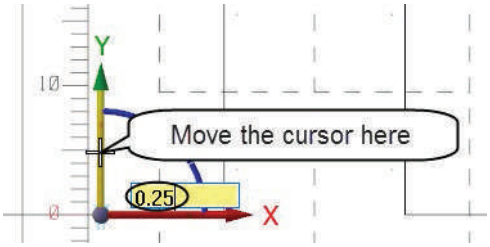


- [Enter a known point or change to Along mode]: Select the **midpoint** of the line as shown in Figure: 10.2.1.

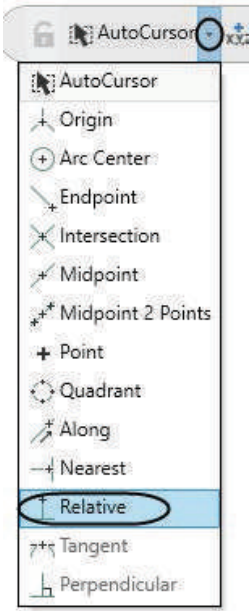
Figure: 10.2.1



- Move the cursor to the **Y axis** as shown, enter **0.25** to create the point above the line.

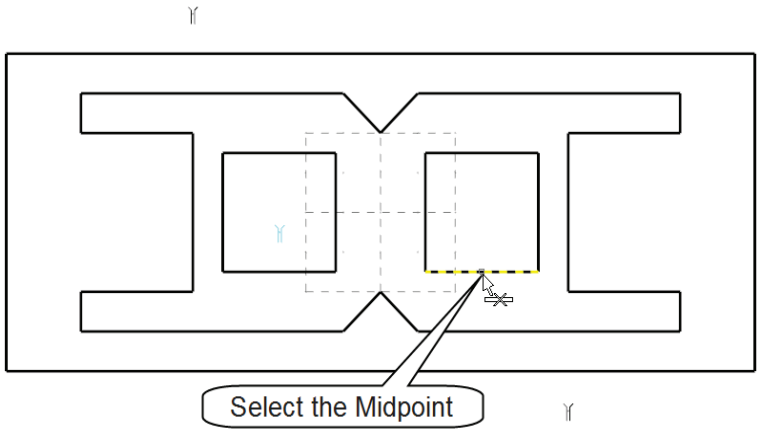


- Press **Enter** three times to create the point.
- From the **AutoCursor** drop down list, select **Relative**.

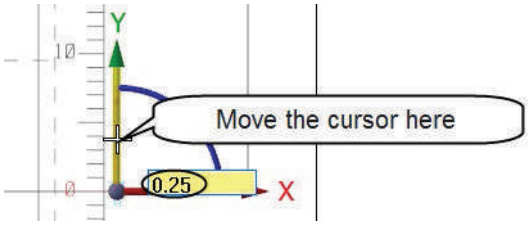


- [Enter a known point or change to Along mode]: Select the **midpoint** of the line as shown in Figure: 10.2.2.

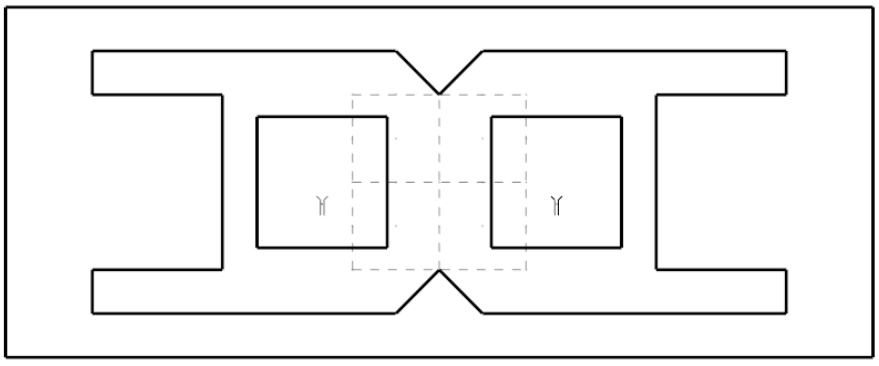
Figure: 10.2.2



- Move the cursor to the **Y axis** as shown, enter **0.25** to create the point above the line.



- Press **Enter** three times to create the point.
- Select the **OK** button to complete the command.
- The final geometry should look as shown.



STEP 11: SAVE THE FILE

FILE

- **Save As.**

- Click on the **Browse** icon as shown.
- Find a location on the computer to save your file.
- File name:"Tutorial 4_Multiple Contours and Tabs".

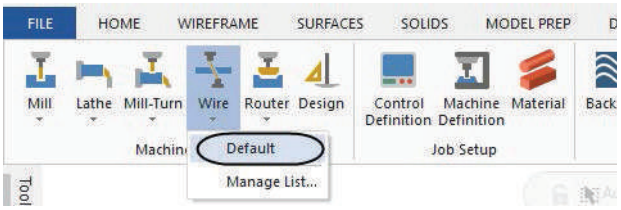
STEP 12: SELECT THE MACHINE AND SET UP THE STOCK

In Mastercam, you select a **Machine Definition** before creating any toolpaths. The **Machine Definition** is a model of your machine tool's capabilities and 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.MCAM-WMD. Once finished, select OK to exit the dialog box. Otherwise, please follow next step.

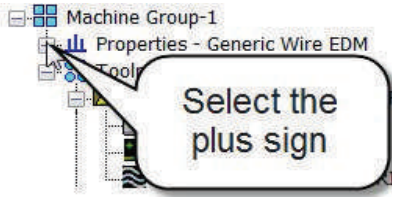
MACHINE

- Click on the pull down arrow under **Wire** and select **Default**.

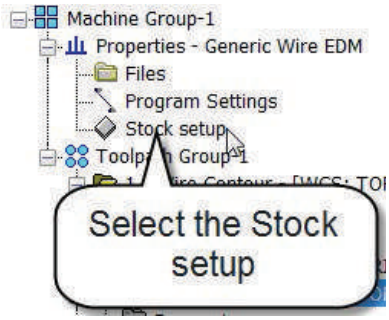


12.1 Set up the stock

- From the **Toolpaths Manager**, select the plus sign (+) in front of the **Properties** to expand the **Toolpaths Group Properties**.

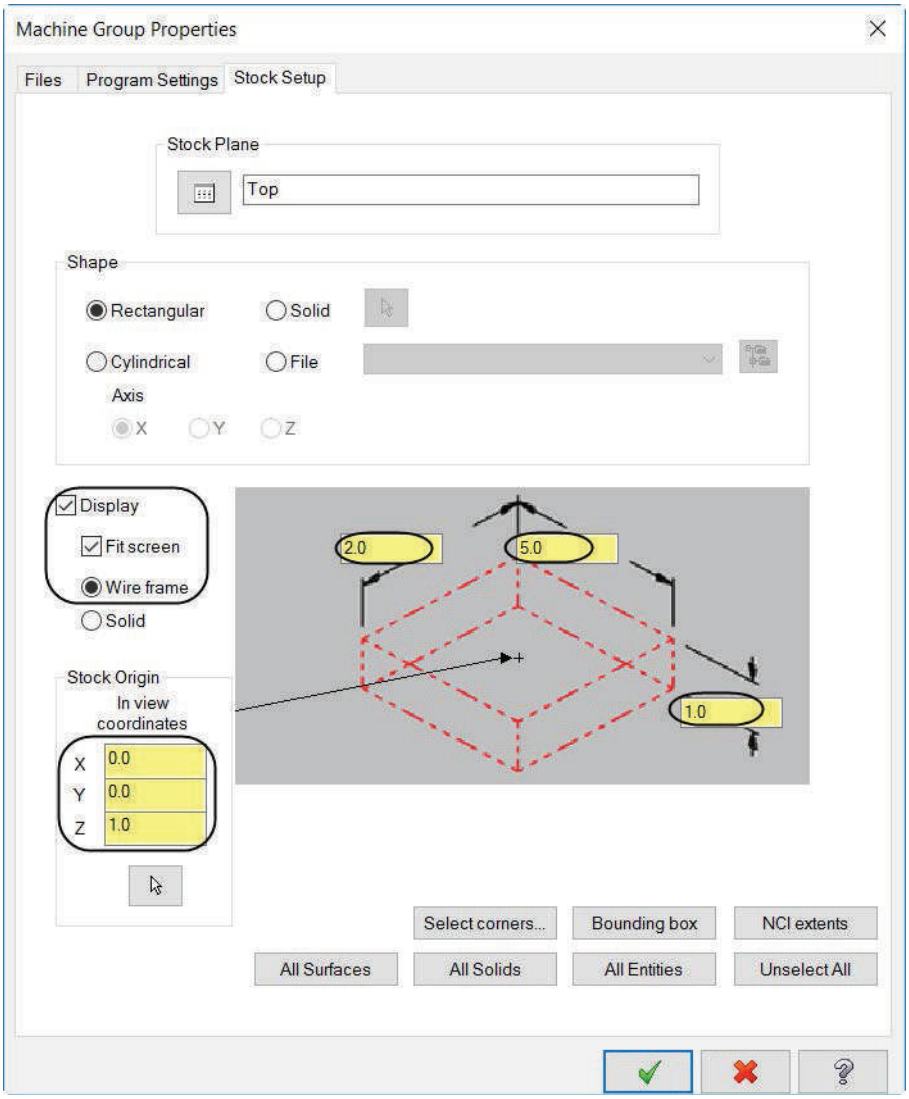


- Select the **Stock setup** to set the stock size.

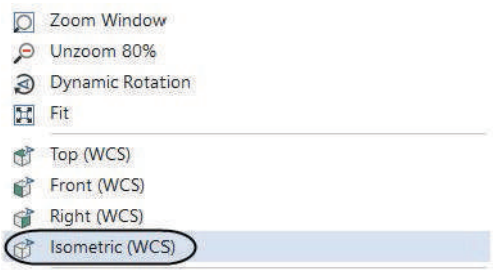


- Change the parameters to match Figure: 12.1.1.

Figure: 12.1.1

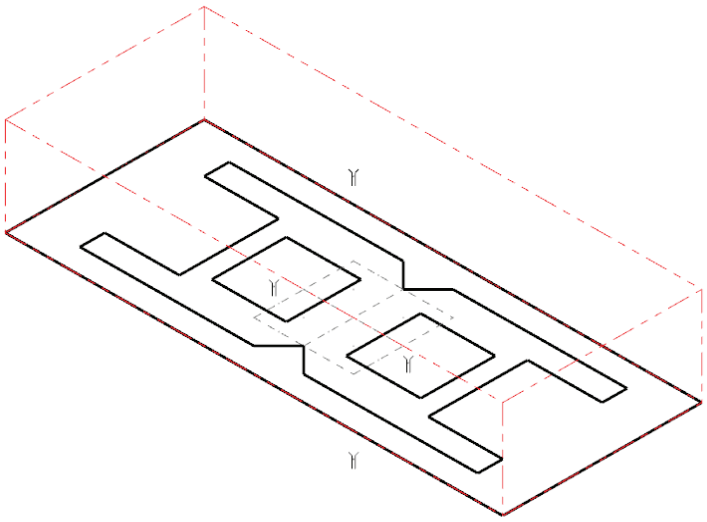


- Select the **OK** button to exit **Machine Group Properties**.
- Right mouse click on the graphics window and select the **Isometric** view as shown.

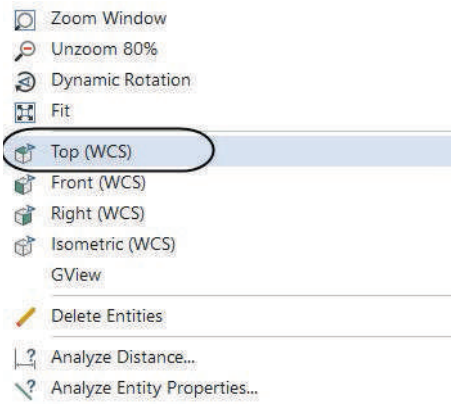


- Press **Alt+F1** to fit the drawing to the screen.

- The stock is displayed with phantom red lines and it should look as shown.



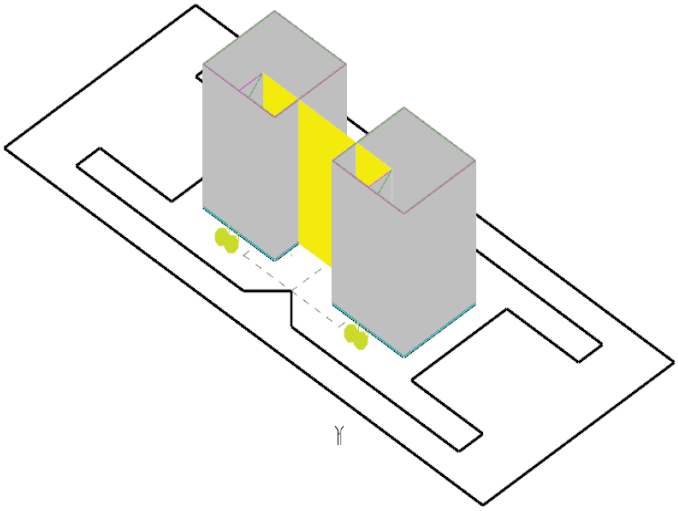
- Right mouse click in the graphics window and select **Top** view.



STEP 13: MACHINE MULTIPLE CONTOURS USING CONTOUR WIREPATH

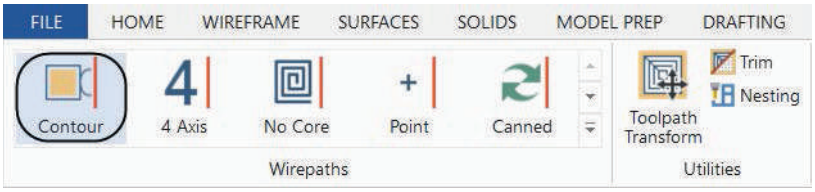
Contour wirepaths have the same general shape in both the XY plane (the lower contour) and the UV plane (the upper contour). Contour wirepaths can taper inward or outward, and you can specify the location of the land, the point at which the taper begins. You can further modify the shape of the contour 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. In this step you will learn how to select multiple contours using the window selection.

Toolpath Preview:

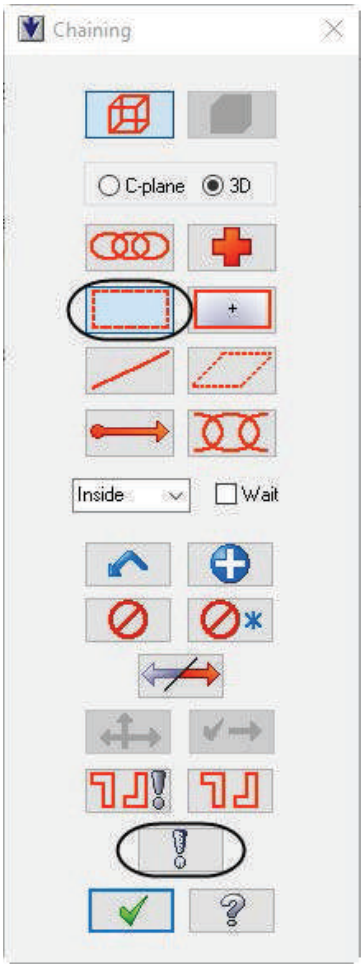


WIREPATHS

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

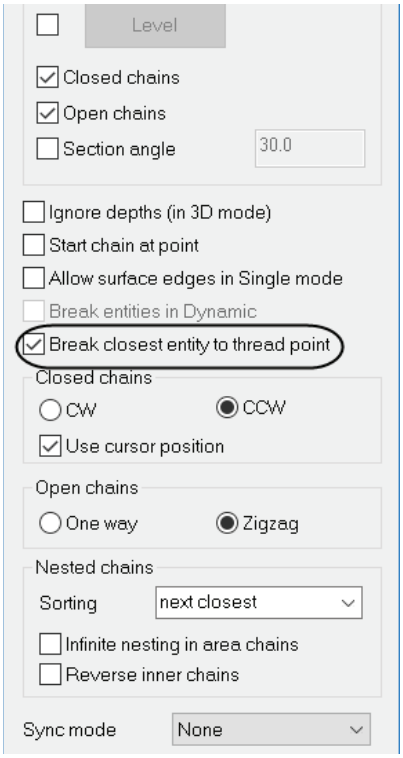


- From the **Chaining** dialog box, select the **Window** button. Then select the **Options** button.



- Enable **Break closest entity to thread point** as shown in Figure: 13.0.1.

Figure: 13.0.1



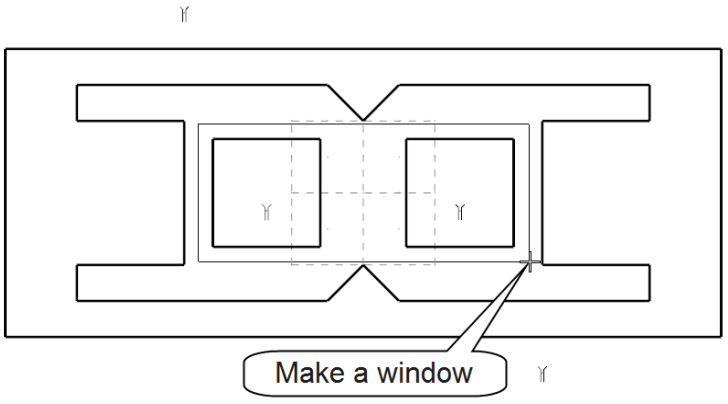
NOTE: Break closest entity to thread point establishes a wirepath start point by breaking the entity closest to the thread point at a position perpendicular to it. If a perpendicular position cannot be established, this setting is ignored.

- Select the **OK** button to exit **Chaining Options**.

- Make a window around the two rectangles as shown in Figure: 13.0.2.

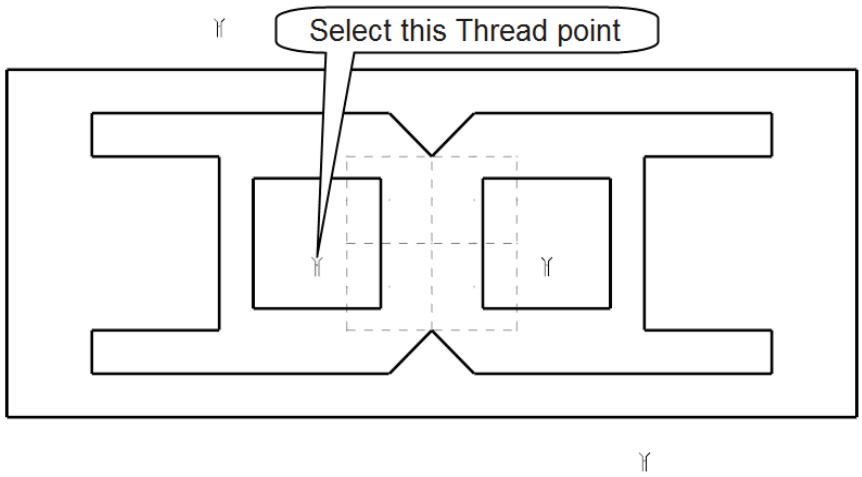
Figure: 13.0.2

NOTE: To make a window, left click and hold one corner of the window, and then move the cursor to the opposite corner and left click again. Make sure that the two rectangles are completely inside of the window.



- [Sketch approximate start point]: Select the thread point as shown in Figure: 13.0.3.

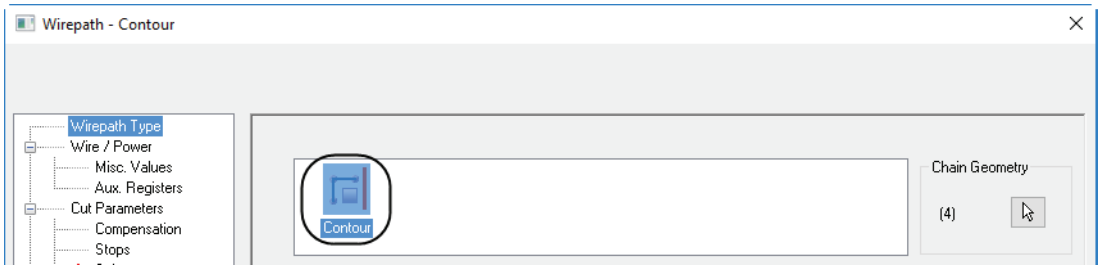
Figure: 13.0.3



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

13.1 Wirepath Type

- In the **Wirepath type** page, the **Contour** toolpath should already be selected.



13.2 Set the Wire/Power parameters

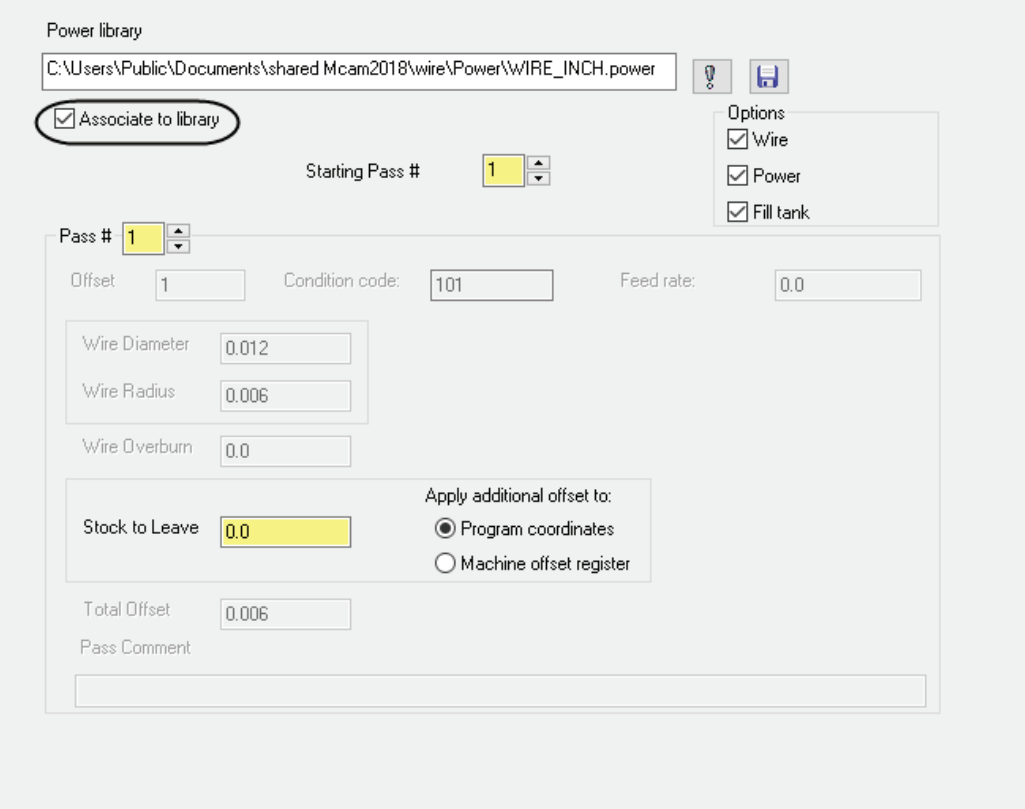
The **Wire/Power** page allows you to choose a wire power settings library file and use its values for the wirepath operation. You can also set the pass number to use for the first pass of the wirepath, and specify the settings for a single pass. To edit the power settings for each pass you have to disassociate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

- From the **Tree View** area, select **Wire/Power**.

NOTE: If you need to create your own wire power library, please see Tutorial #1 (**page 46**). To open the library saved in Tutorial #1, click on the **Open** icon and select the library from the Power library list. Otherwise, use the default WIRE_INCH.power library as shown in this tutorial.

- Enable **Associate to library** to use the default **WIRE_INCH.POWER** library as shown in Figure: 13.2.1.
- If an **Edit Power Settings** dialog box appears, select the option **No.**

Figure: 13.2.1

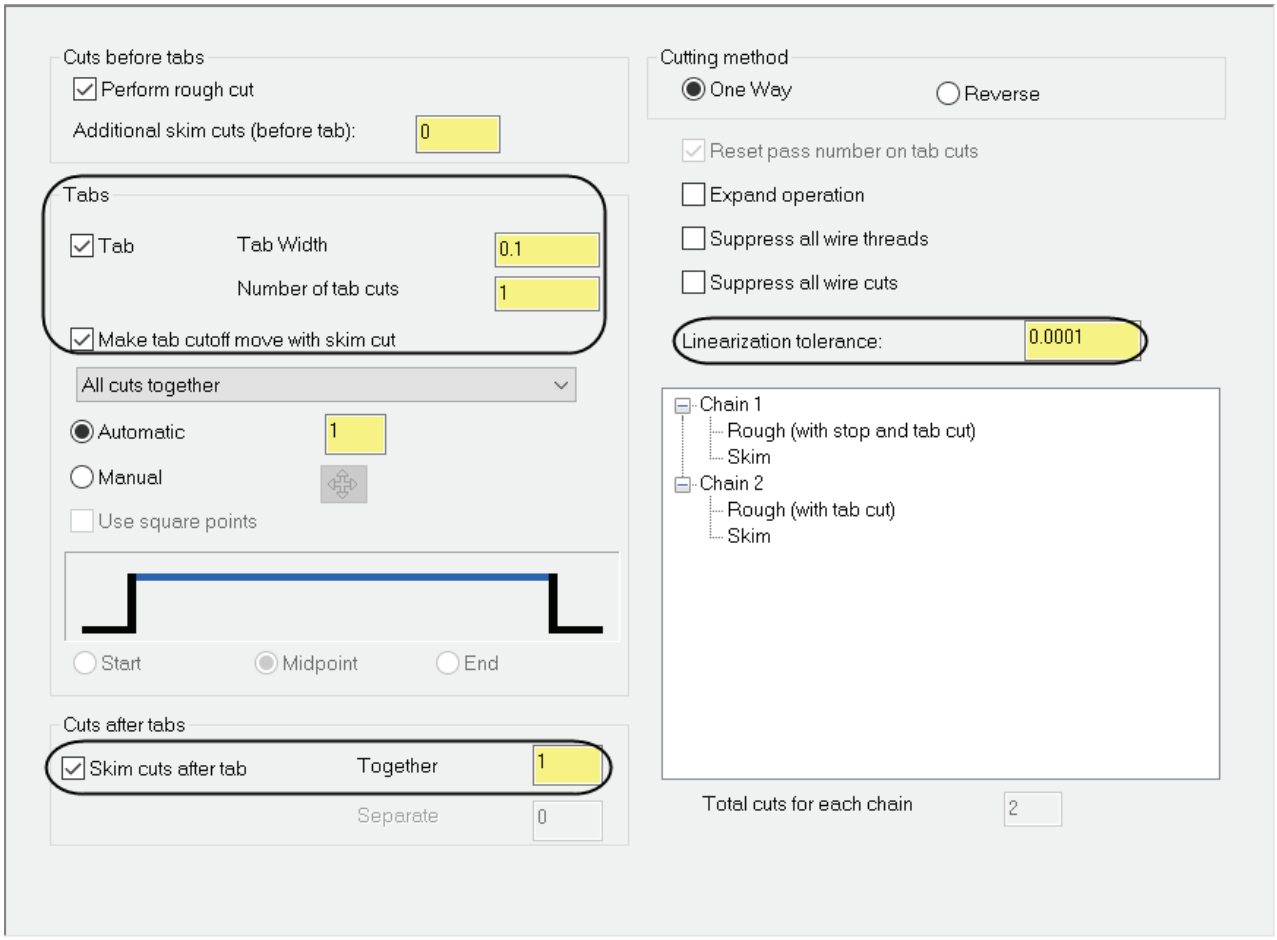


13.3 Set the Cut Parameters

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, one tab cut, and one finish pass as shown in Figure: 13.3.1.

Figure: 13.3.1



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

Tab is the uncut material that secures the part to the material; **Make tab cutoff with skim cut** pass cuts the tab with the **Rough**.

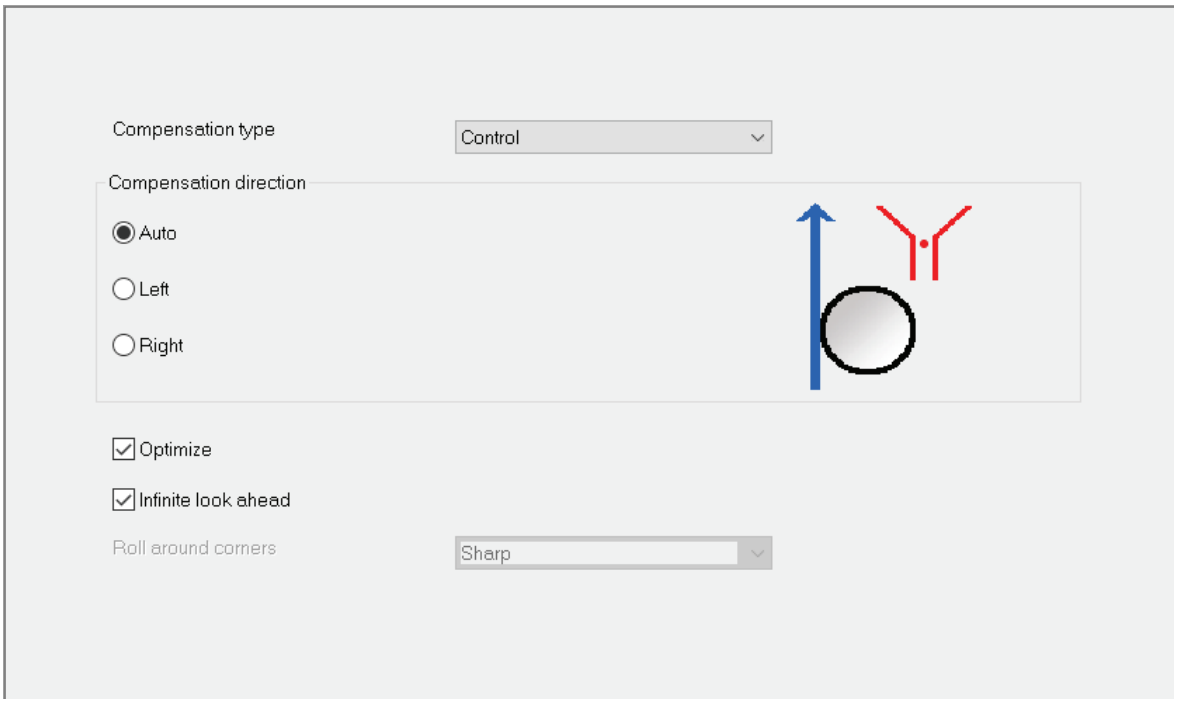
Skim cuts after tab allows you to enter the number of finish passes after cutting the tab.

13.4 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**.
- Leave the **Compensation direction** to **Auto** and disable **Infinite look ahead** as shown in Figure: 13.4.1.

Figure: 13.4.1



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.

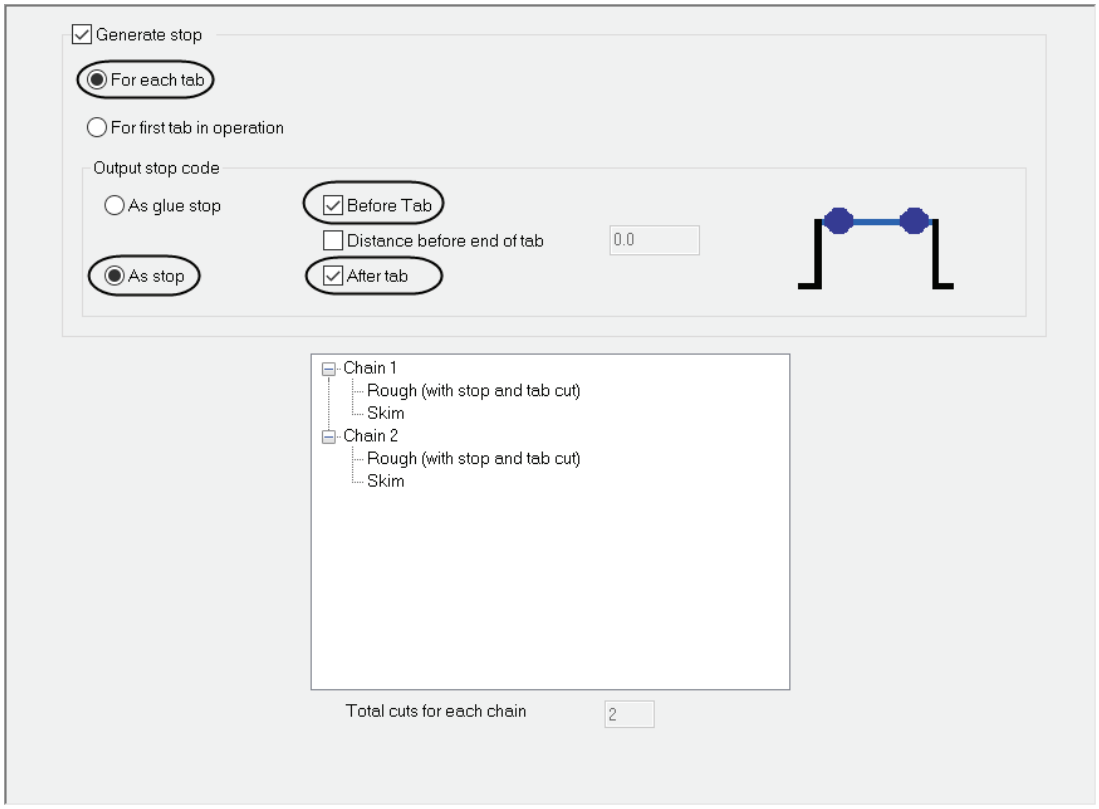
Roll Cutter Around Corners inserts arc moves around corners in the wirepath. The radius of the arc moves equals the radius of the wire. Not available with compensation type set to **Control**.

13.5 Stops

Stops page allows you to create optional stops (M01) or program stops (M00) in a wirepath. For contour wirepaths, you have the options to apply the stops before and/or after tabs.

- From the **Tree View** area, select **Stops** as shown in Figure: 13.5.1.
- Make sure that the parameters are set to generate the stop (**M00**) before the **Tab.**
- Notice that **Chain 2** now has a stop.

Figure: 13.5.1



For first tab in operation outputs a stop code before the tab cut for the first tab cut on Chain 1 of the operation. Subsequent tab cuts throughout the operation do not include stop codes.

As stop outputs a program stop code (M00). Pauses the wire machine until the operator restarts it.

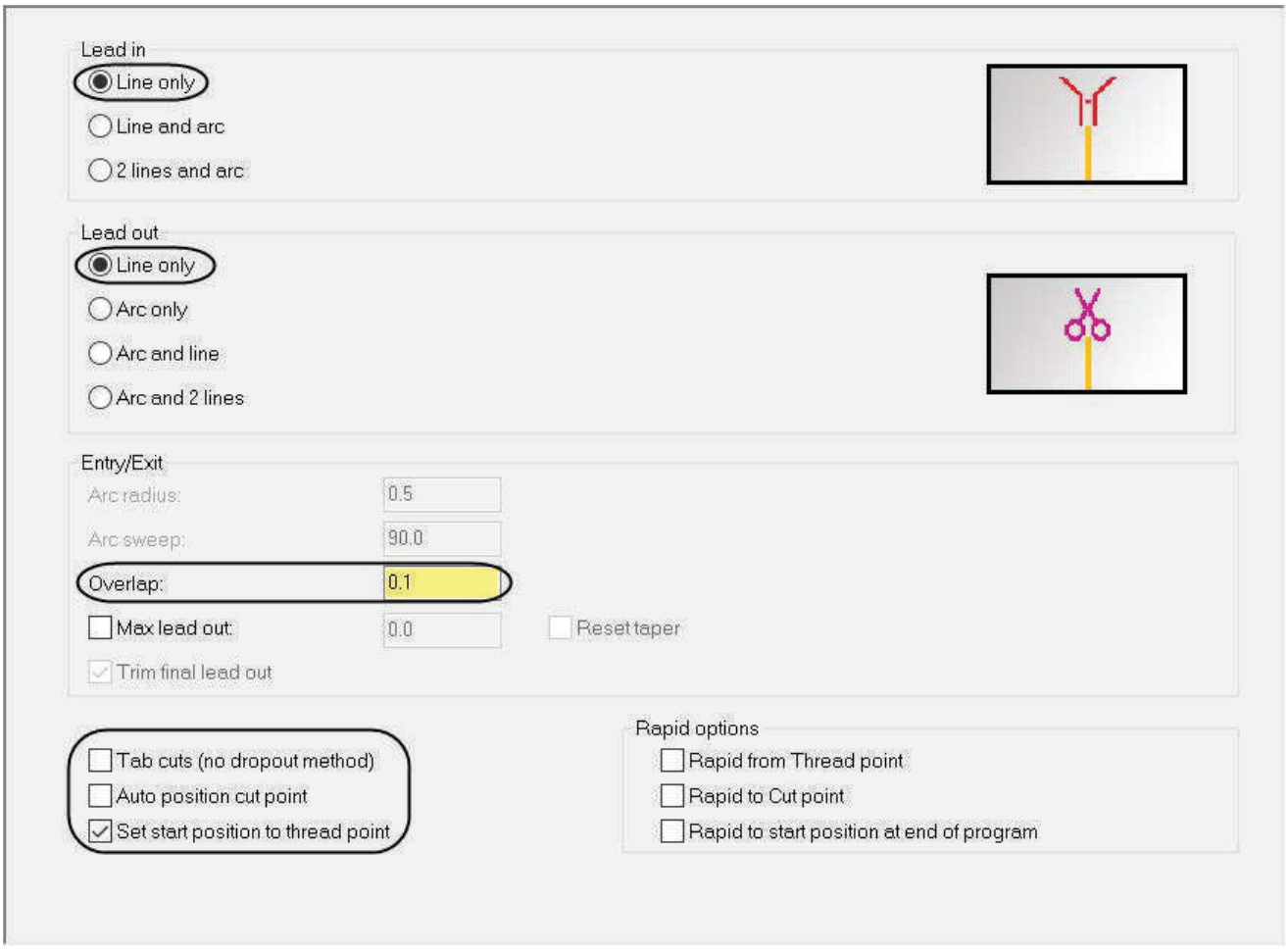
Before Tab and **After Tab** places the glue stop or the program stop (whichever stop code you select) either before or after the selected tab.

13.6 Leads

Leads page allows you to set the wire motion when entering or exiting a part.

- From the **Tree View** area, select **Leads**.
- Make sure that the **Lead in** and **Lead out** are set to **Line only**. Add an **Overlap** and set the rest of the parameters as shown in Figure: 13.6.1.

Figure: 13.6.1



Lead in sets the shape of the part entry motion. The lead in extends from the thread point to the start of contour.

Line only creates a line between the thread point and start of contour.

Lead out sets the shape of the part exit motion. Lead out extends from the end of contour to the cut point. You have the same options as in the Lead in plus the following extra options.

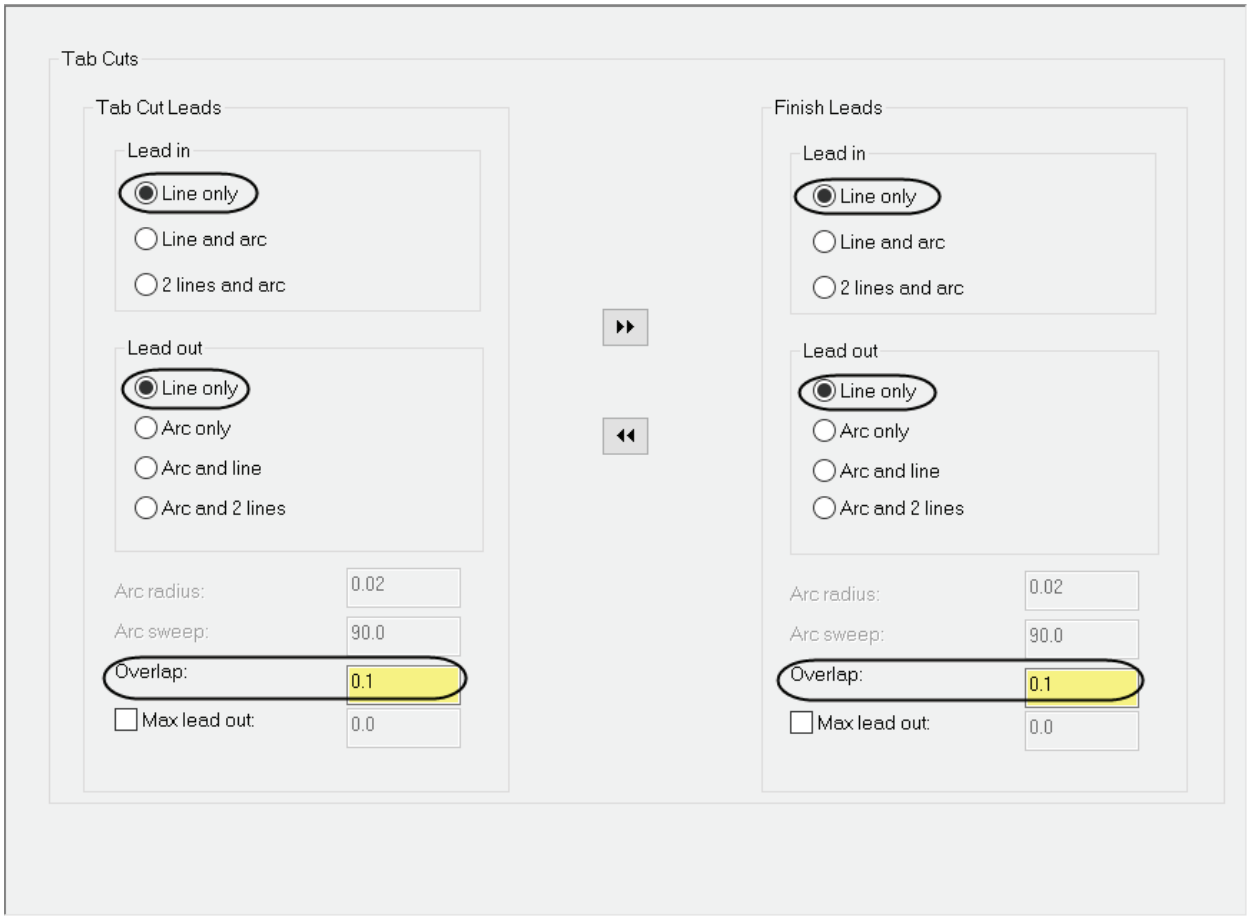
Overlap eliminates potential burrs by overlapping the start and end of the contour by the amount entered.

13.7 Set the Tab/Finish Leads

This page allows you to set the wire motion when entering or exiting a part when making tab cuts (Tab Cut Leads section) and skim cuts after a tab cut (Finish Leads section).

- From the **Tree View** area, select **Tab/Finish Leads**.
- Make sure that the **Lead in** and **Lead out** are set to **Line only,** add an **Overlap** as shown in Figure: 13.7.1.

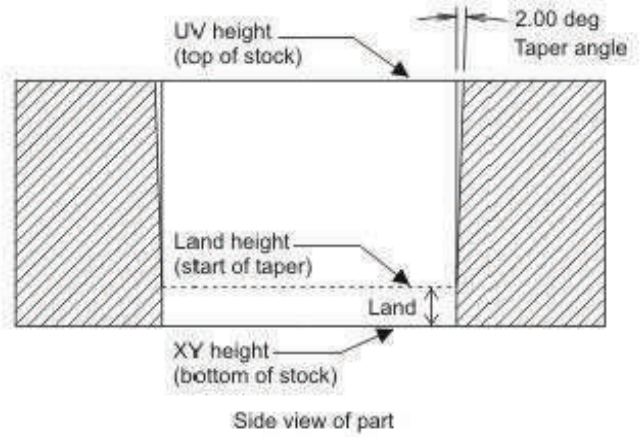
Figure: 13.7.1



13.8 Taper

Taper page allows you to set the parameters to taper the contour wirepath inward or outward and to specify the location of the land (the point at which the taper begins). You can set the Z height of the upper wire guide for rapid moves, the location of the upper wire guide and lower wire guide, the height of the part and the location of the XY plane as shown in Figure: 13.8.1.

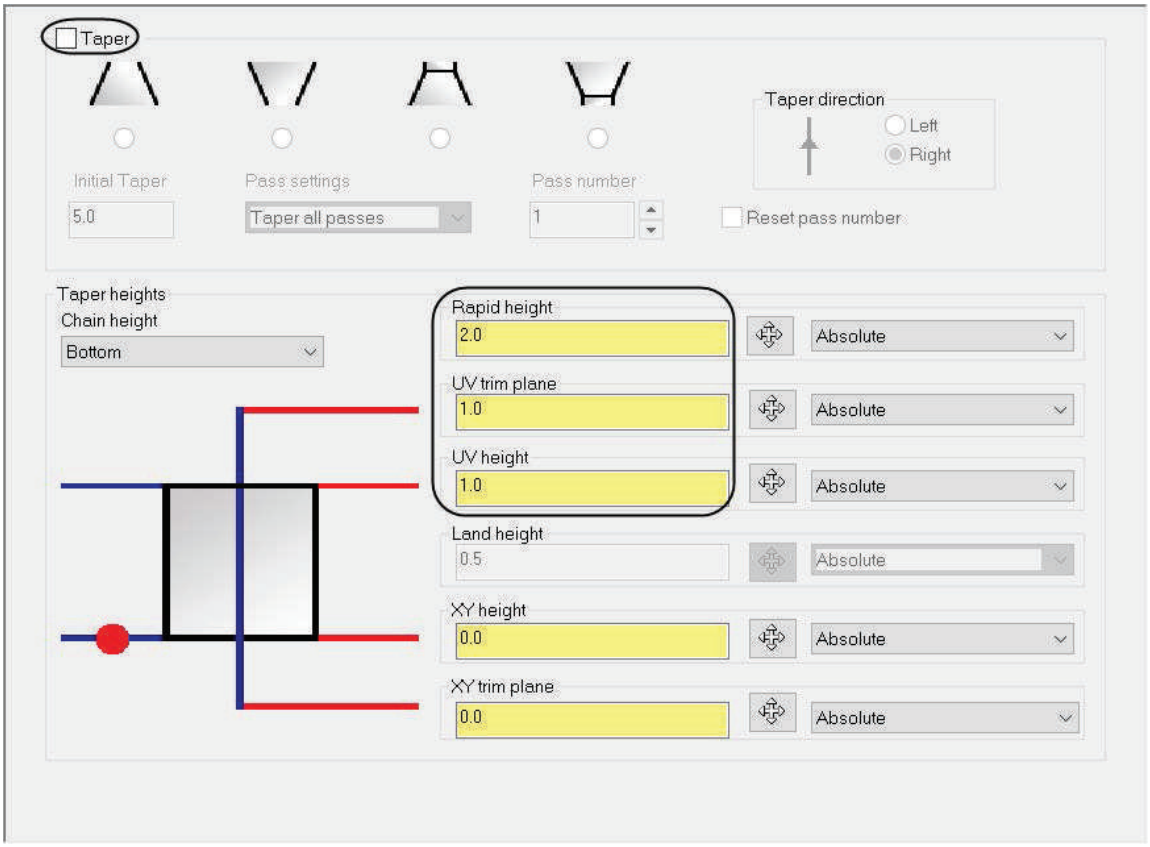
Figure: 13.8.1



- From the **Tree View** area, select **Taper**.

- Make sure that the **Taper** is disabled and change the heights as shown in Figure: 13.8.2.

Figure: 13.8.2



Chain height options allows you to indicate where the geometry was chained (XY-**Bottom** or UV-**Top** height or, if there is a land, at the land height-**Middle**).

Rapid height sets the Z depth of a rapid move. Can be higher than the **UV height**, **UV trim**, or **UV extension planes** in order to clear any clamps.

UV trim plane can be used to set the location of the upper guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to G code 1015 in the NC file. It is usually set directly at the controller.

UV height sets the location of the XY plane that contains the geometry for the upper contour of a 4-Axis part.

Land height is available only when **Land up** or **Land down** is selected for **Taper** style, and sets the plane at which the wire pivots to the taper angle.

XY height sets the location of the XY plane that contains the geometry for the lower contour of a 4-Axis part.

XY trim plane can be used to set the location of the lower guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to Gcode 1015 in the NC file. It can be set directly at the controller. When not using the XY trim plane, enter the same value as the XY height.

- Select the **OK** button to exit the **Wirepath - Contour** parameters.

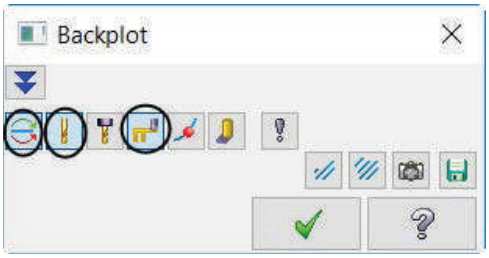
- Select the **OK** button to exit the **Chain Manager** dialog box.

STEP 14: BACKPLOT THE WIRE CONTOUR WIREPATH

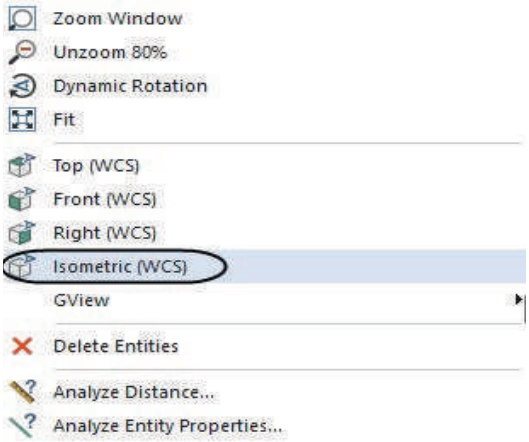
Backplotting shows the path the tools take to cut the part. This display lets you spot errors in the program before you machine the part. As you backplot wirepaths, Mastercam displays the current X, Y, and Z coordinates in the lower left corner of the screen.

- Make sure that the toolpath is selected (signified by the green check mark on the folder icon).
- Select the **Backplot selected operations** button.

- Make sure that you have the following buttons turned on to see the tool and therapid moves.

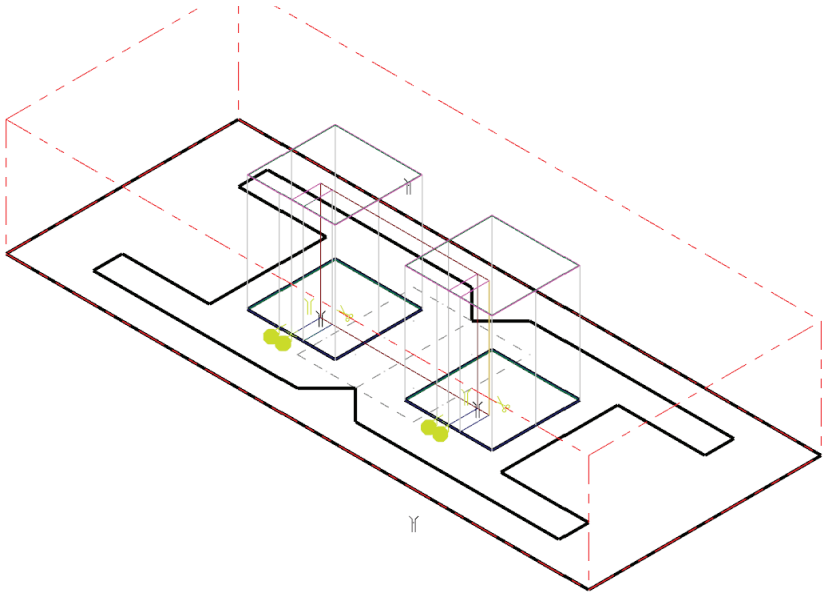


- Right mouse click in the graphics window and select **Isometric** as shown.

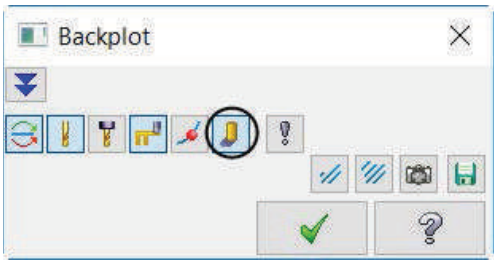


- Press **Alt+F1** to fit the drawing to the screen.
- Select the **Play** button to run **Backplot**.

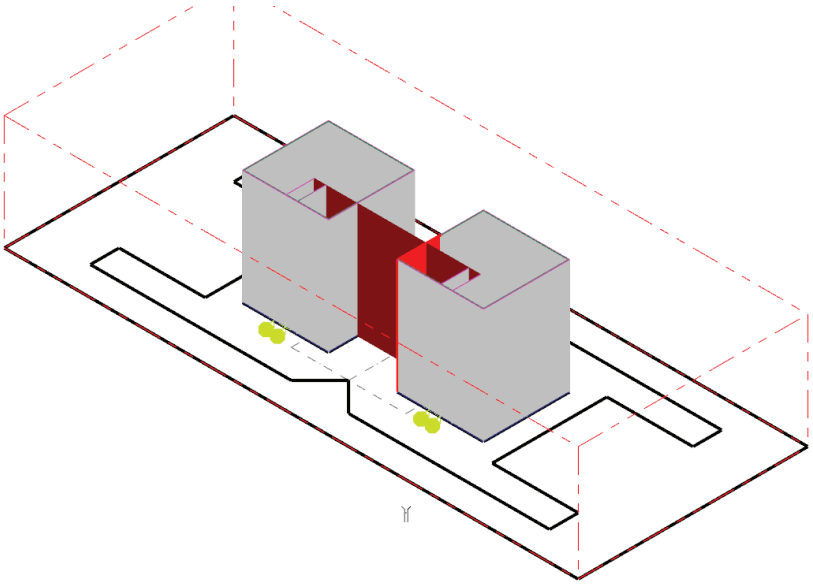
- The toolpath should look as shown.



- In the **Backplot** dialog box enable **Quick Verify** as shown.



- The toolpath should look as shown.

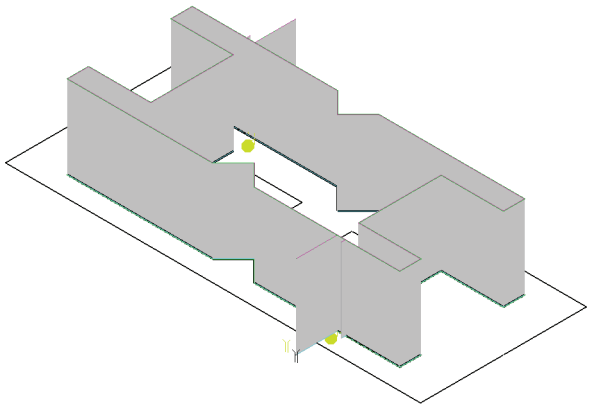


- Select the **OK** button to exit **Backplot**.

STEP 15: MACHINE THE OUTER PROFILE WITH MULTIPLE TABS

In this step you will learn how to machine the outer profile using multiple tabs. Multiple tabs are usually necessary to hold the part better while cutting the part. You will have to prepare the geometry and break the contour at the thread points before generating the wirepath.

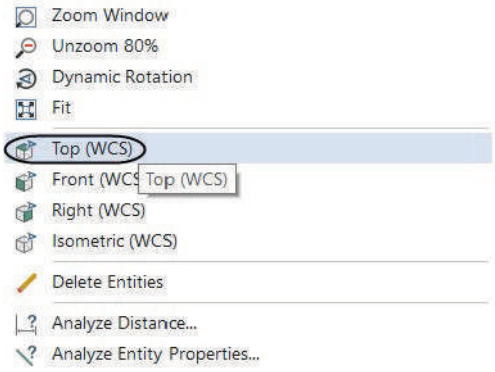
Toolpath Preview:



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

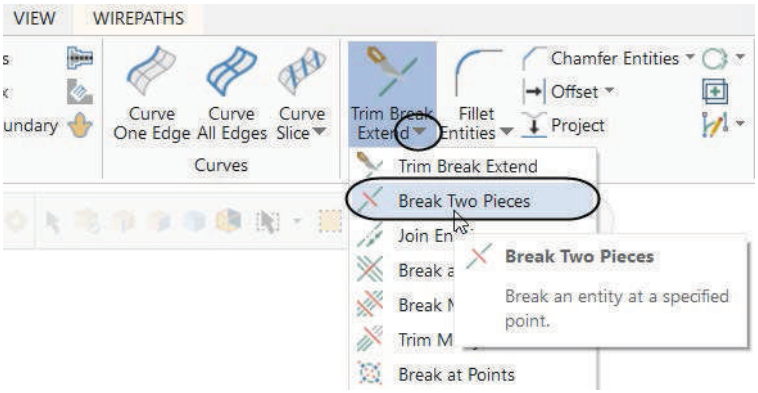
15.1 Break at geometry at the thread points

- Right mouse click in the graphics window and select **Top** as shown.



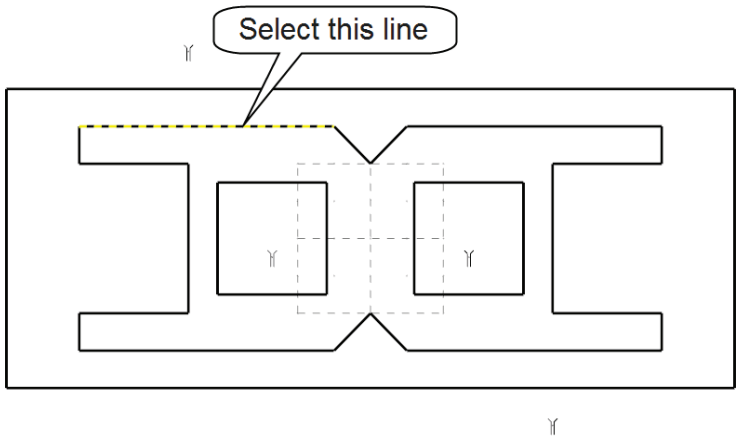
WIREFRAME

- From the **Modify** group, click on the pull down arrow under **Trim Break Extend** and select **Break Two Pieces** as shown.



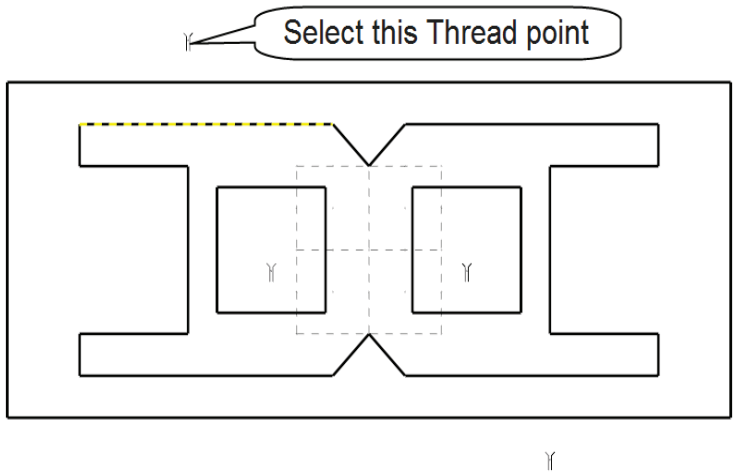
- [Select an entity to break]: Select the line as shown in Figure: 15.1.1.

Figure: 15.1.1



- [Indicate the break position]: Select the thread point as shown in Figure: 15.1.2.

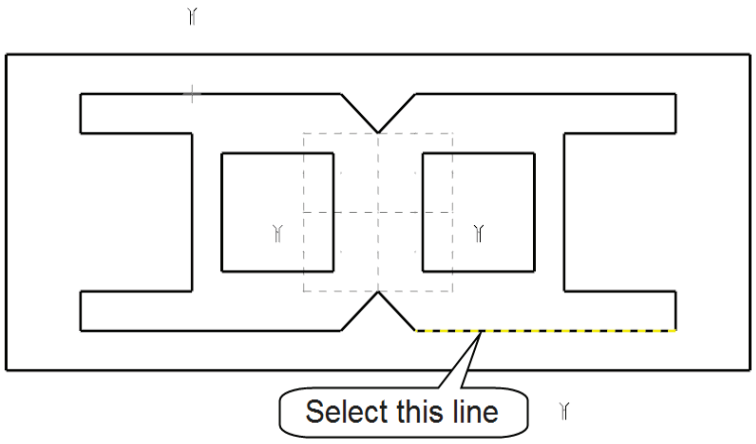
Figure: 15.1.2



NOTE: Although the point does not lie on the selected line, Mastercam calculates the closest position on the entity and breaks the entity to that point.

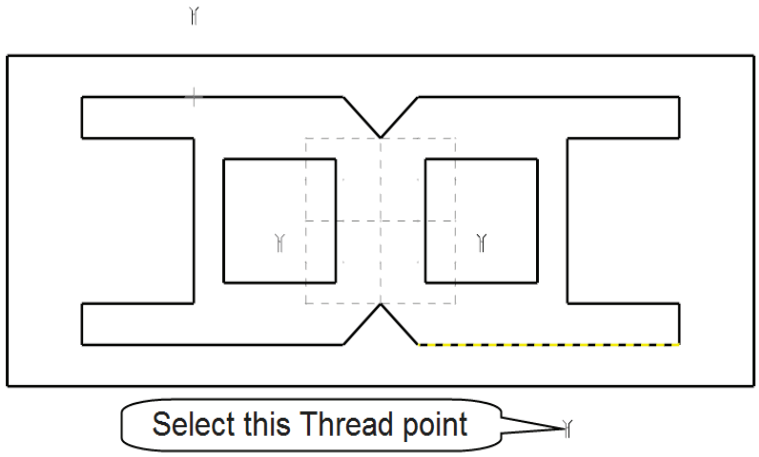
- [Select an entity to break]: Select the line as shown in Figure: 15.1.3.

Figure: 15.1.3



- [Indicate the break position]: Select the thread point as shown in Figure: 15.1.4.

Figure: 15.1.4

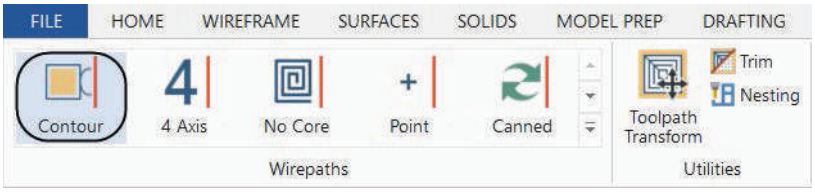


- Press **Esc** to exit the command.

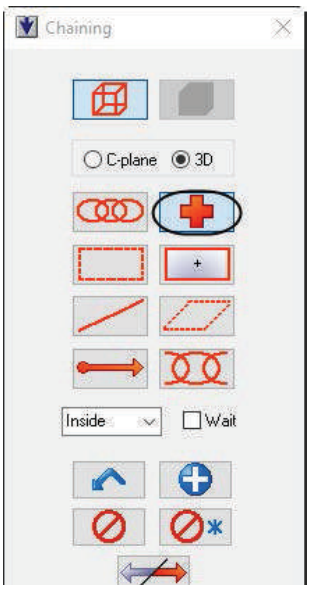
15.2 Create the contour wirepath with multiple chains and tabs

WIREPATHS

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

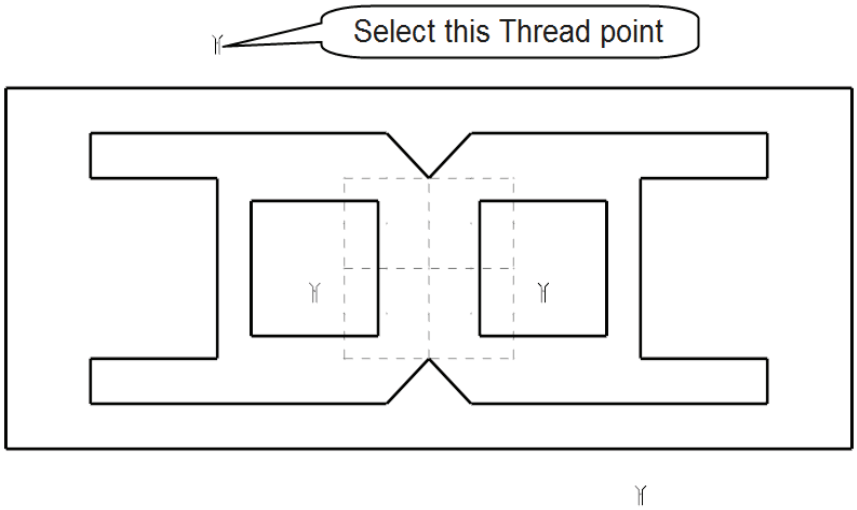


- From the **Chaining** dialog box, select the **Point** button.

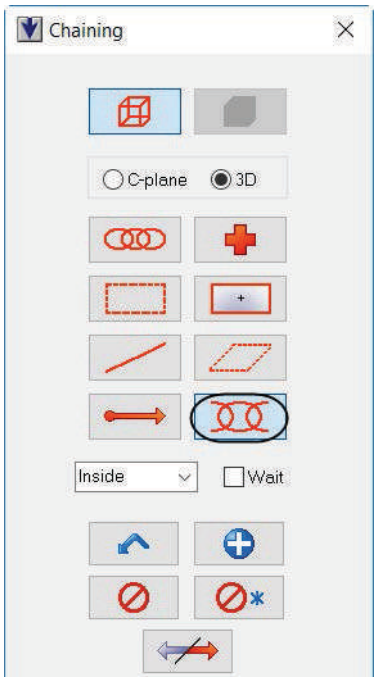


- [Contour: select chain 1]: Select the thread point as shown in Figure: 15.2.1.

Figure: 15.2.1

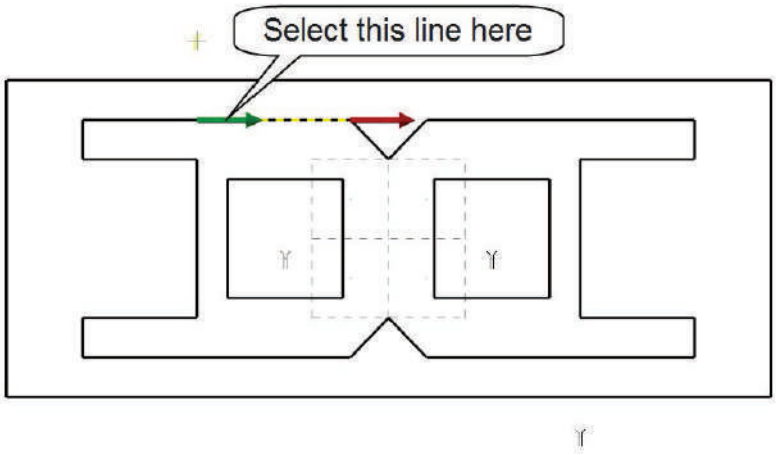


- From the **Chaining** dialog box, select the **Partial** button.



- [Select the first entity]: Select the line in the CW direction as shown in Figure: 15.2.2.

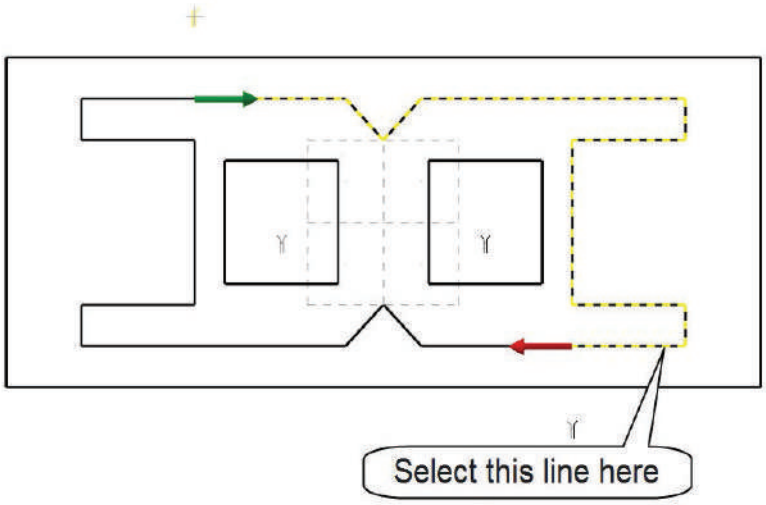
Figure: 15.2.2



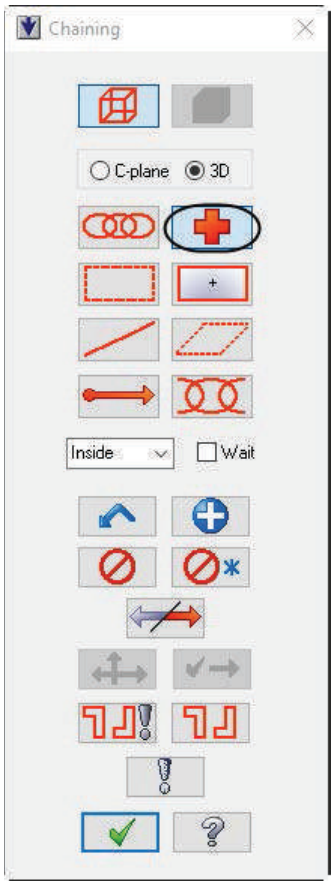
NOTE: If the chain is not selected in the proper direction, from the **Chaining** dialog box, click on the **Reverse** button to flip the direction.

- [Select the last entity]: Select the line as shown in Figure: 15.2.3.

Figure: 15.2.3

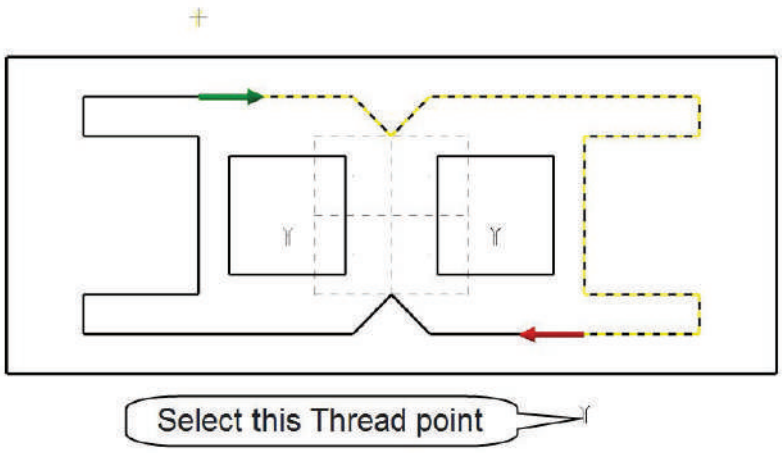


- From the **Chaining** dialog box, select the **Point** button.

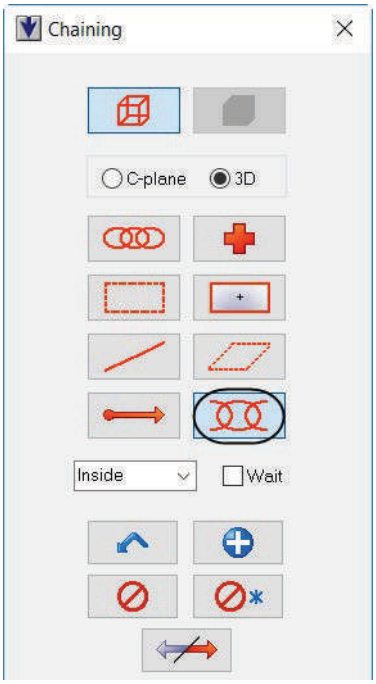


- [Contour: select chain 3]: Select the Thread point as shown in Figure: 15.2.4.

Figure: 15.2.4

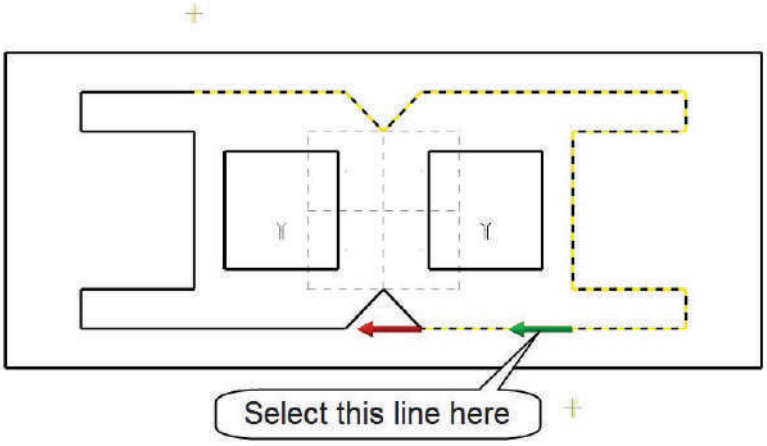


- From the **Chaining** dialog box, select the **Partial** button.



- [Select the first entity]: Select the line in the CW direction as shown in Figure: 15.2.5.

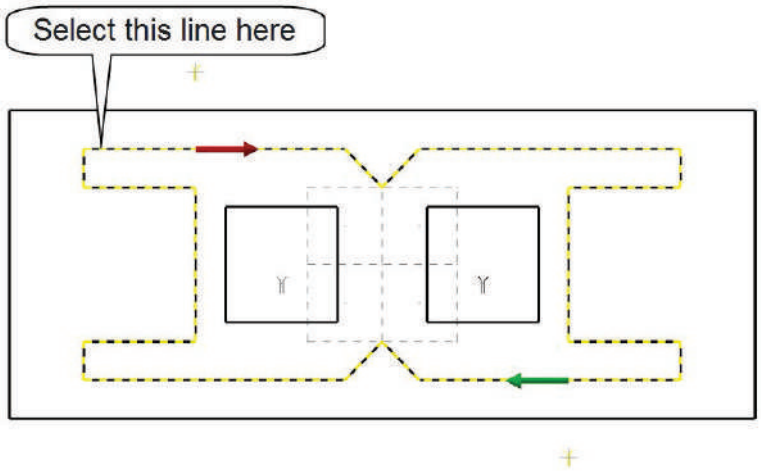
Figure: 15.2.5



NOTE: If the chain is not selected in the proper direction, from the **Chaining** dialog box, click on the **Reverse** button to flip the direction.

- [Select the last entity]: Select the line as shown in Figure: 15.2.6.

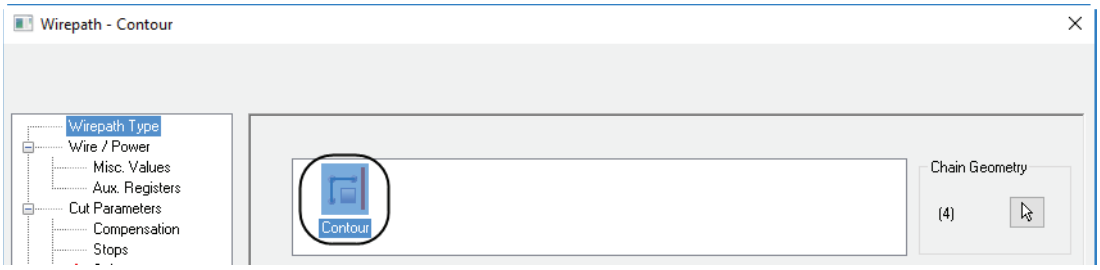
Figure: 15.2.6



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

15.3 Wirepath Type

- In the **Wirepath type** page, the **Contour** toolpath should be already selected.



15.4 Set the Wire/Power parameters

- From the **Tree View** area, select **Wire/Power**.

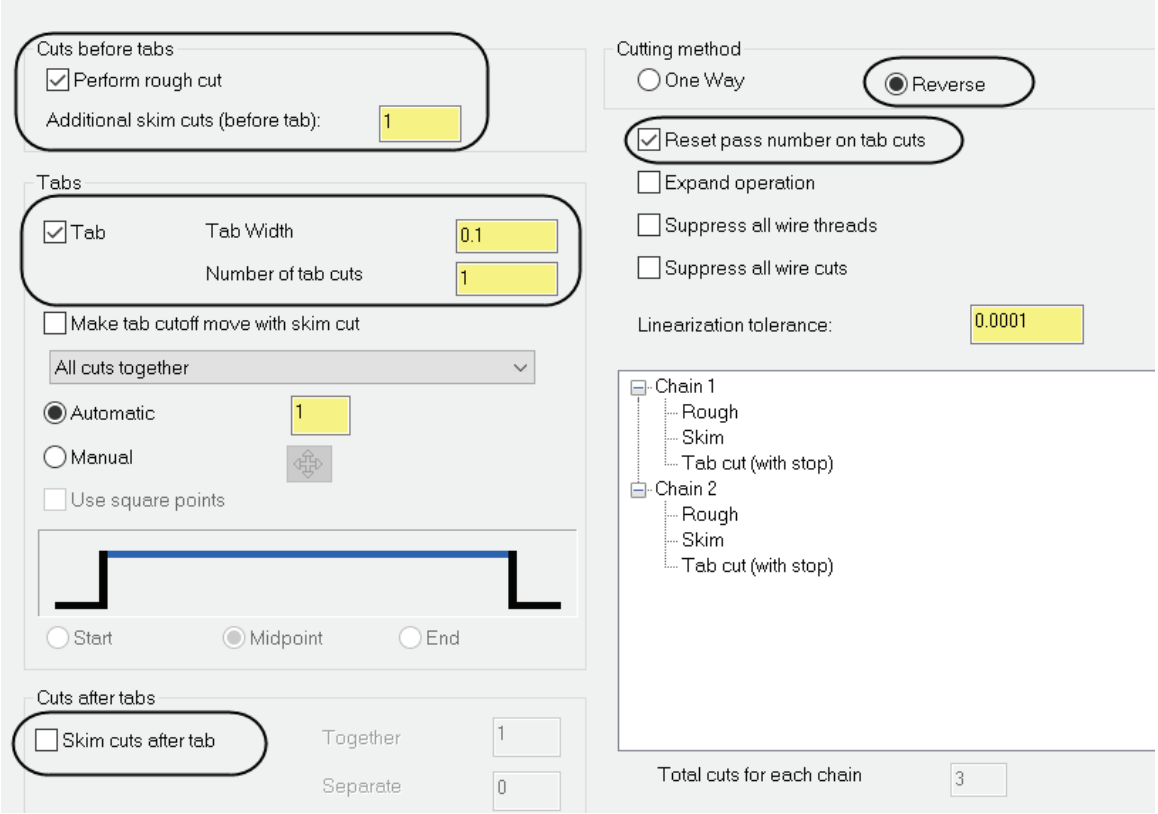
NOTE: We are using the same default WIRE_INCH.power library. No changes are needed.

15.5 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. 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 separate rough passes, skim cuts and tab cuts as shown in Figure: 15.5.1.
- Disable **Make tab cutoff move with skim cuts** and from the drop down list select **Rough, tab and finish separately**.

Figure: 15.5.1



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

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

Tab is the uncut material that secures the part to the material; can be cutoff with the skim cut pass or in a separate pass.

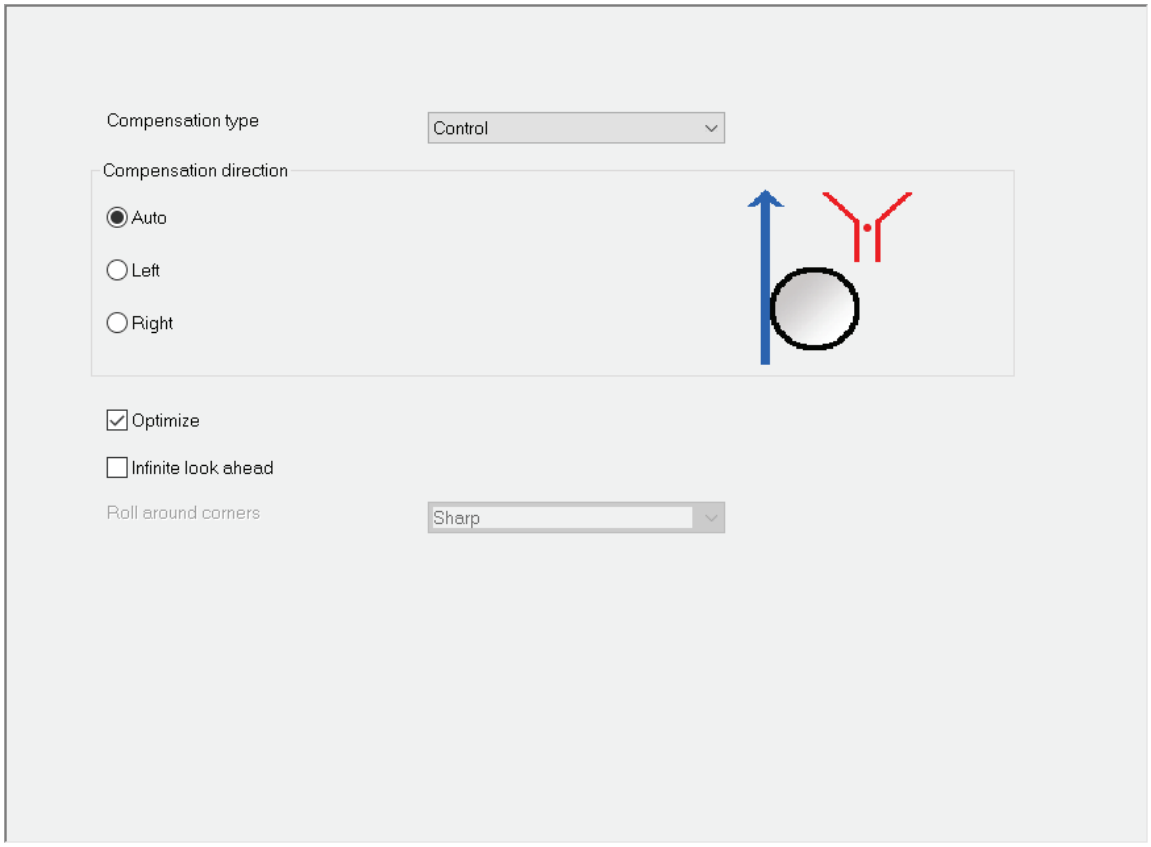
Cutting method set to **Reverse** is used when multiple passes are set for the toolpath. Reverse enabled reverses the direction of the previous pass at the end of the contour or at the tab.

Make tab cutoff move with skim cut allows you to combine the tab cut with the last skim cut. If you have no skim cut before the tab, the tab cut combines with the rough cut.

15.6 Set the Compensation parameters

- From the **Tree View** area, select **Compensation**.
- Leave the **Compensation direction** to **Auto** and disable **Infinite look ahead** as shown in Figure: 15.6.1.

Figure: 15.6.1



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.

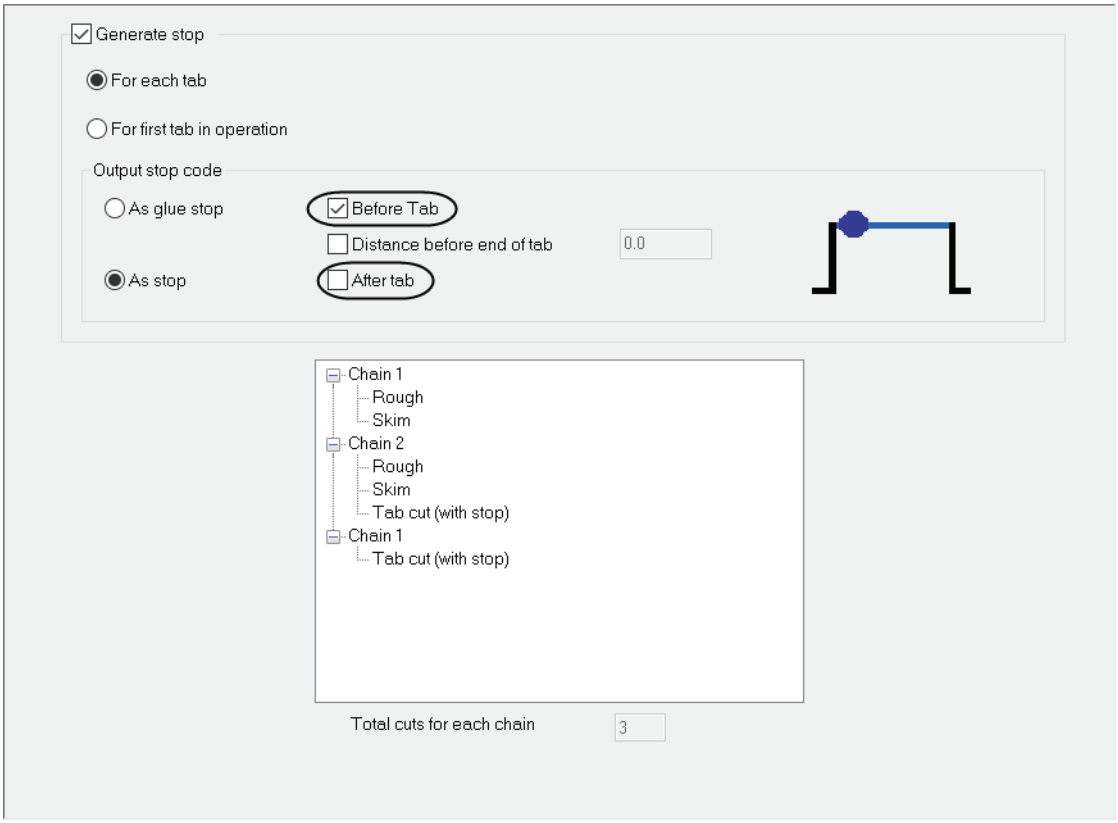
Infinite Look Ahead checks for wirepath self-intersections along the entire contour before creating the wirepath. If Mastercam Wire finds a wirepath intersection, it modifies the wirepath so that it does not cut the portion of the part that comes after the intersection.

Roll Cutter 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**.

15.7 Stops

Stops page allows you to create optional stops (M01) or program stops (M00) in a wirepath. For contour wirepaths, you have the options to apply the stops before and/or after tabs.

- From the **Tree View** area, select **Stops**.
- Make sure that the parameters are set to generate the stop (**M00**) before the **Tab** only.



For first tab in operation outputs a stop code before the tab cut for the first tab cut on Chain 1 of the operation. Subsequent tab cuts throughout the operation do not include stop codes.

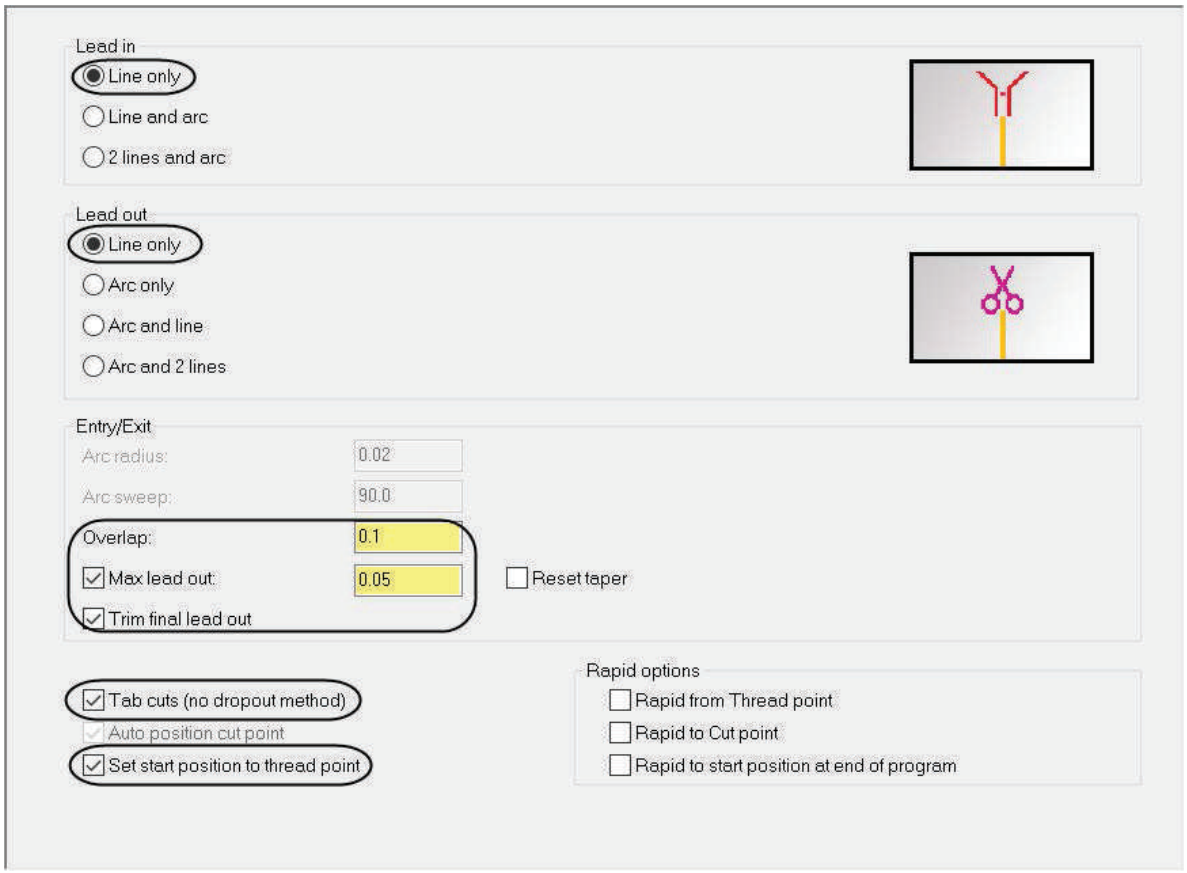
As stop outputs a program stop code (M00). Pauses the wire machine until the operator restarts it.

Before Tab and **After Tab** places the glue stop or the program stop (whichever stop code you select) either before or after the selected tab.

15.8 Leads

Leads page allows you to set the wire motion when entering or exiting a part.

- From the **Tree View** area, select **Leads**.
- Make sure that the **Lead in** and **Lead out** are set to **Line only** as shown.



Lead in sets the shape of the part entry motion. The lead in extends from the thread point to the start of contour.

Line only creates a line between the thread point and start of contour.

Lead out sets the shape of the part exit motion. Lead out extends from the end of contour to the cut point. You have the same options as in the Lead in plus the following extra options.

Overlap eliminates potential burrs by overlapping the start and end of the contour by the amount entered.

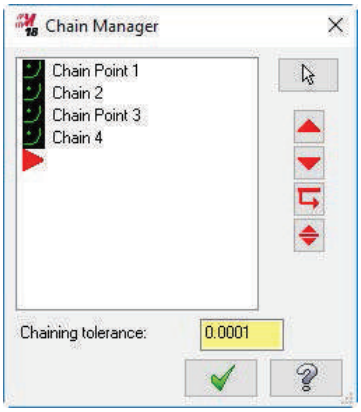
Max lead out allows you to shorten the lead out by entering a new value for the lead out. When disabled, the lead out equals the distance from the end of contour to the cut point for each pass.

Trim Final Lead Out allows you to constrain the final lead out within the Max lead out length you defined.

Set start position to thread point automatically sets the start position to thread position. Enable to move the job start point to the thread point of the first contour in the operation.

NOTE: The Taper parameters can be left as they were set for the previous wire contour.

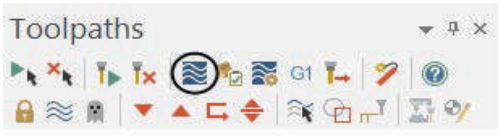
- Select the **OK** button to exit the **Wirepath - Contour** parameters.
- The **Chain Manager** should look as shown.



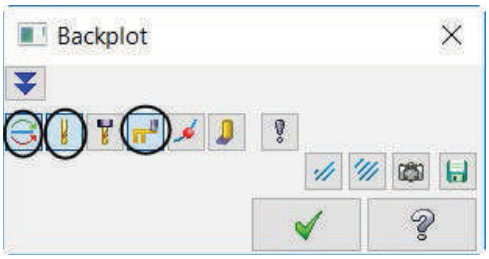
- Select the **OK** button to exit the **Chain Manager** dialog box.

15.9 Backplot the last Wire Contour wirepath

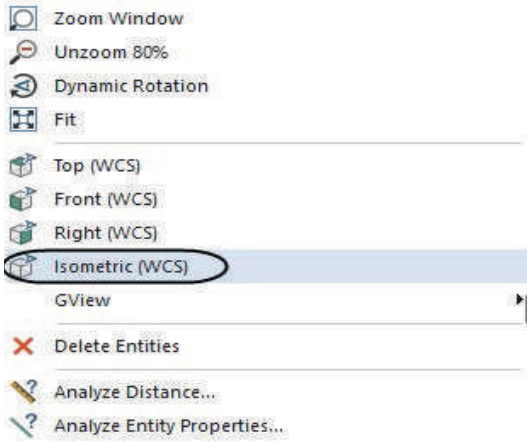
- Select the **Backplot selected operations** button.



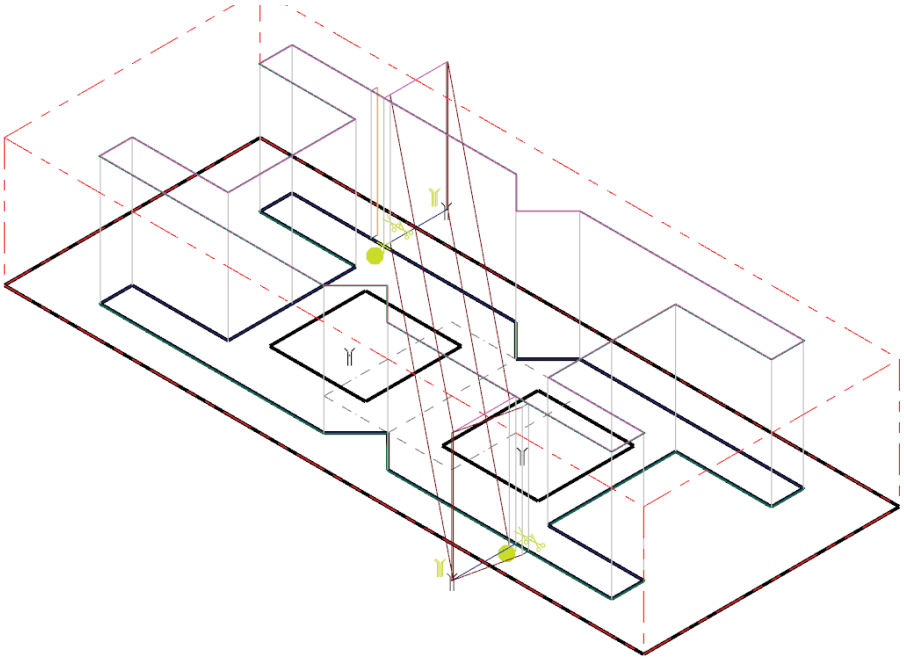
- Make sure that you have the following buttons turned on to see the tool and the rapid moves.



- Right mouse click in the graphics window and select **Isometric** as shown.

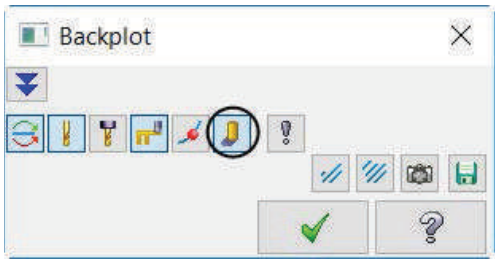


- Press **Alt+F1** to fit the drawing to the screen.
- Select the **Play** button to run **Backplot**.
- The toolpath should look as shown.

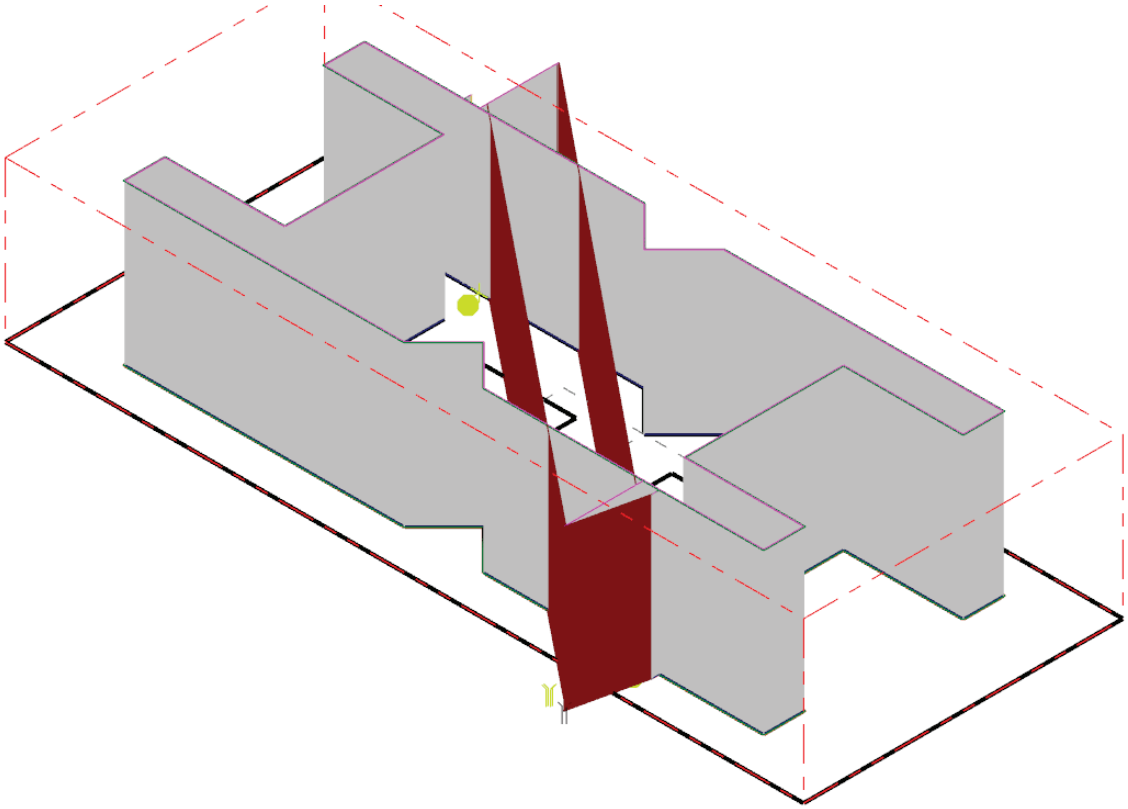


NOTE: The cuts are perform as follows:
First chain: Rough cut starts at the thread point and retracts before the tab on a distance of 0.05" (Max lead out). From here it is changing the cut direction and makes the skim cut still leaving the tab not cut. The wire is cut and then is repositioned above the second thread point. The second chain is rough and finish in the same way as the first chain. The tabs are cut after that.

- In the **Backplot** dialog box enable **Quick Verify** as shown.



- The toolpath should look as shown.



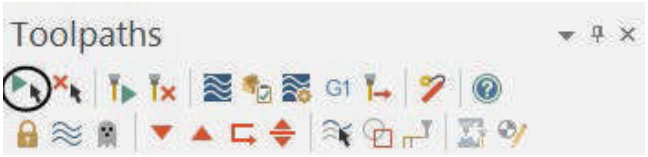
- Select the **OK** button to exit **Backplot**.

STEP 16: VERIFY THE WIREPATH

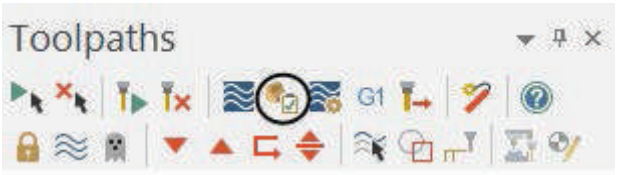
Verify simulates the machining of a part from a solid stock model display. The stock dimensions are based on the values that you can specified in the **Stock Setup**.

16.1 Verify the wirepaths

- From the **Toolpaths Manager**, click on the **Select all operation** icon to select both wirepaths.

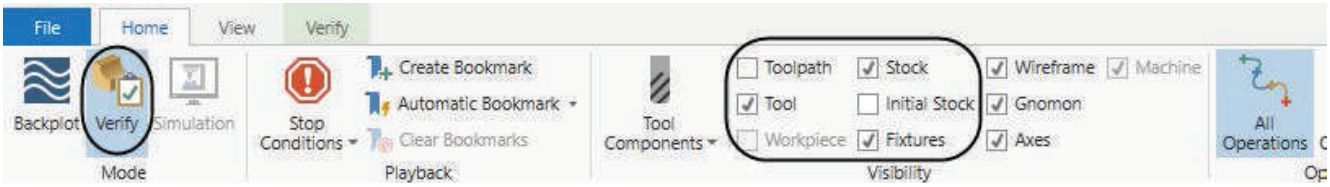


- Select the **Verify selected operations** button.

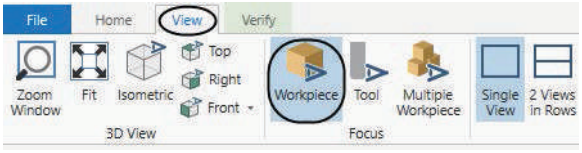


- From **Verify** change the settings for the **Visibility** as shown in Figure: 16.1.1.

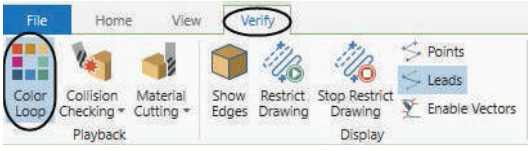
Figure: 16.1.1



- Select **View** tab and enable **Workpiece** as shown.

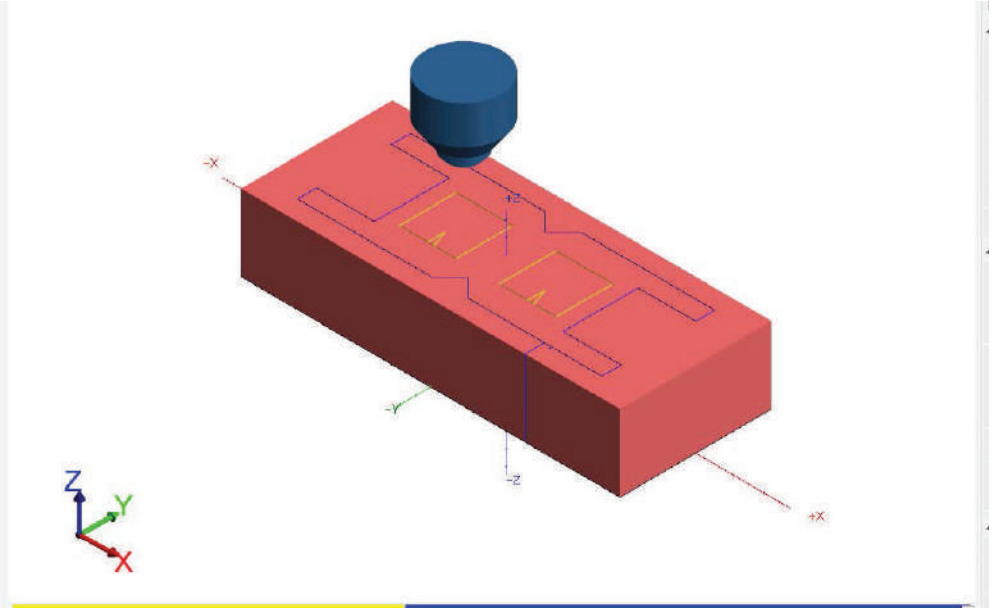


- Select **Verify** tab and enable **Color Loop** as shown.



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

- The part should look as shown.

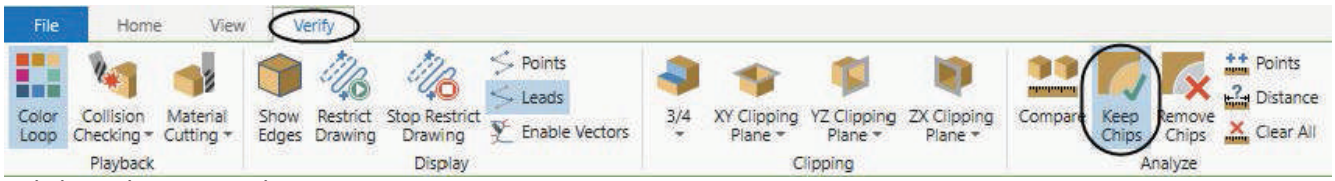


NOTE: To rotate the part, move the cursor to the center of the part and click and hold the mouse wheel and slowly move it in one direction.
To **Zoom In** or **Out** hold down the mouse wheel and scroll up or down as needed.

16.2 Chip removal

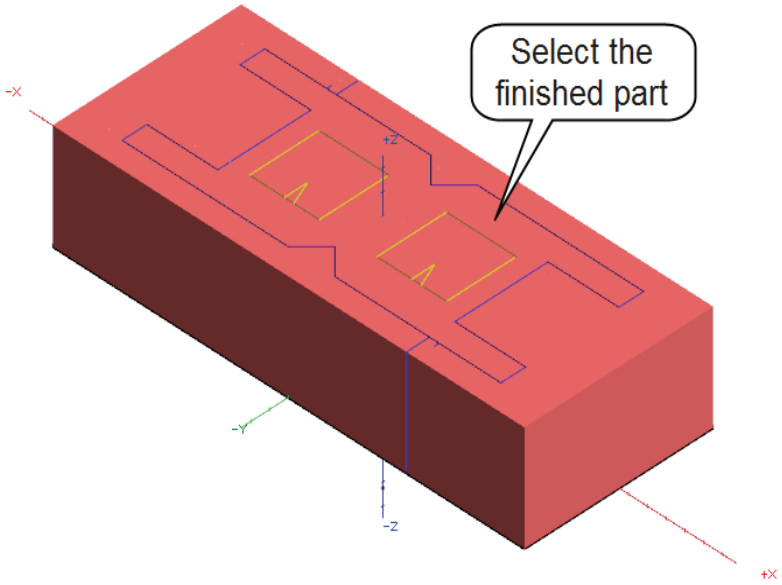
- Select the **Verify Tab** and select the **Keep Chips** icon as shown in Figure: 16.2.1.

Figure: 16.2.1



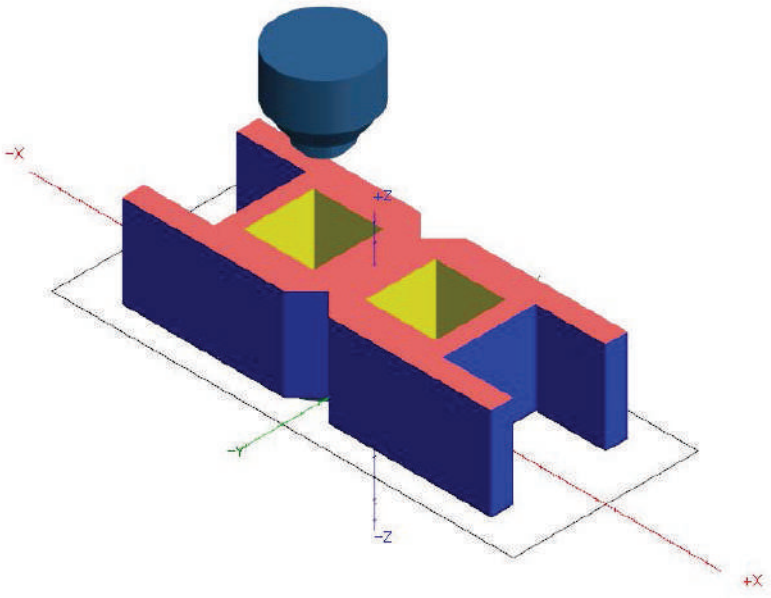
- Click on the part as shown in Figure: 16.2.2.

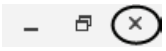
Figure: 16.2.2



- The part will appear as shown in Figure: 16.2.3.

Figure: 16.2.3



- To exit **Mastercam Simulator** click on the **Close** icon. 

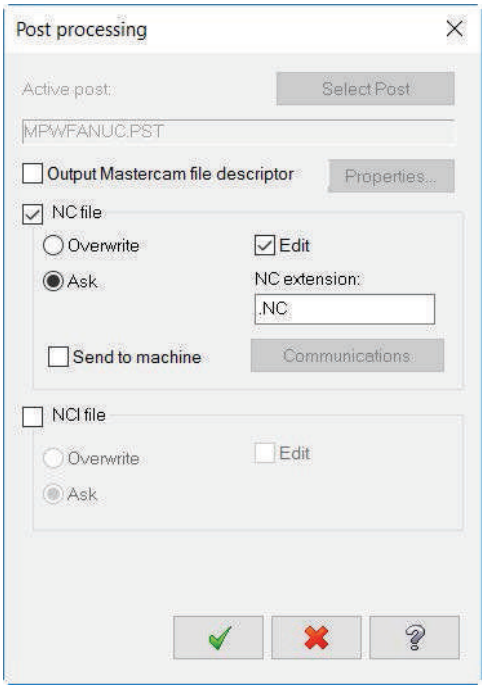
STEP 17: POST THE FILE TO GENERATE THE NC CODE

- Select **Post selected operations** button from **Toolpaths Manager**.

NOTE: The active **Post Processor** is a generic FANUC post processor.

- In the **Post processing** window, as shown in Figure: 17.0.1, if needed, make any necessary changes.

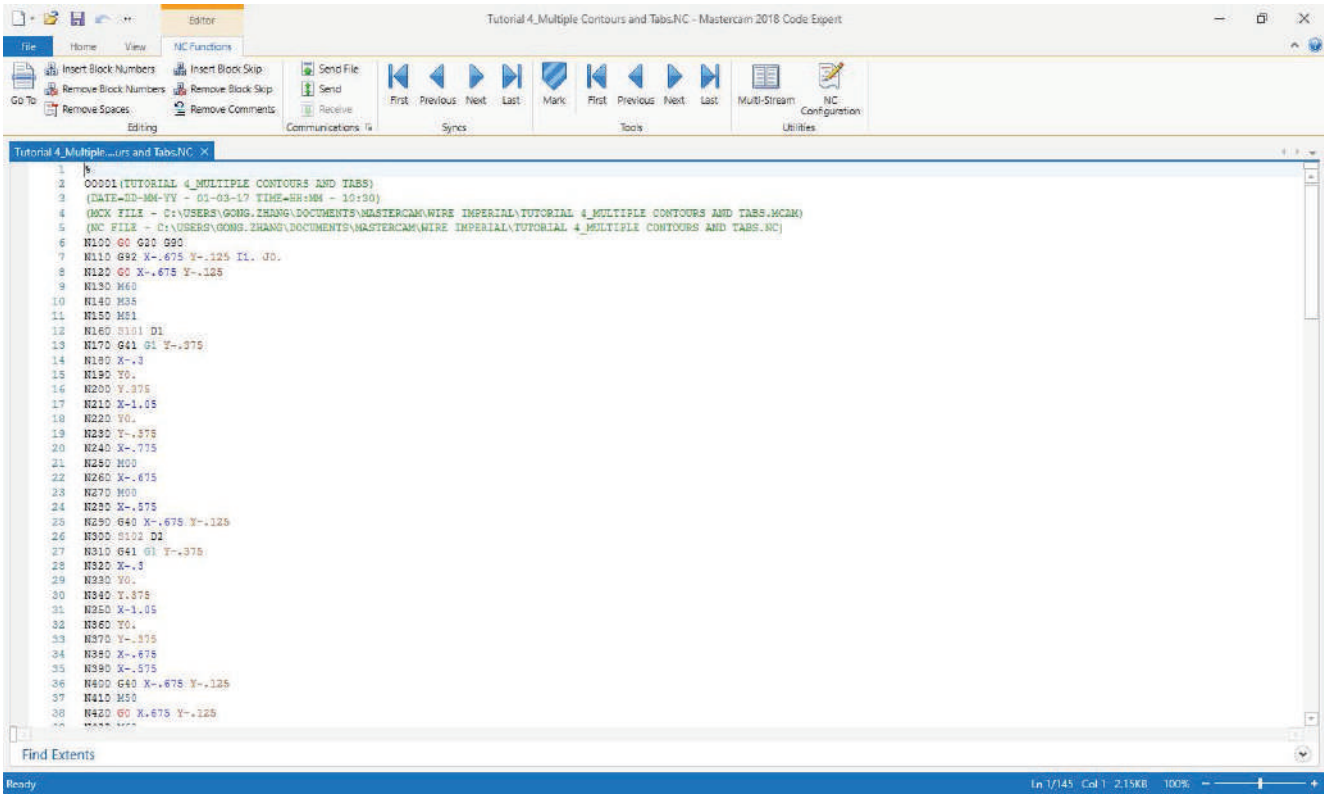
Figure: 17.0.1



- Select the **OK** button to continue.
- Save the NC file.

- The NC file will appear on the screen as shown in Figure: 17.0.2.

Figure: 17.0.2

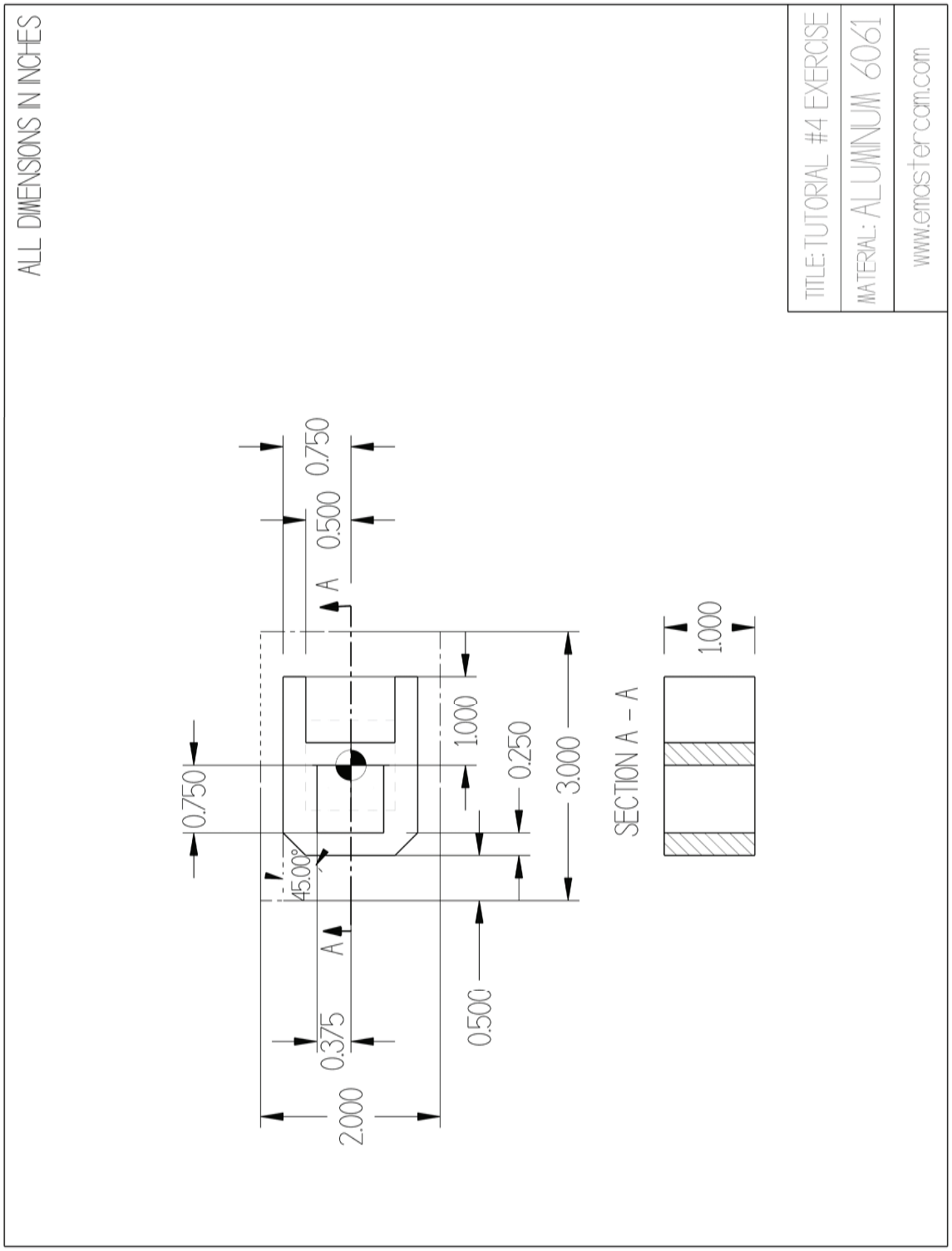


NOTE: The G-code that you have created will appear on the screen. If the code looks okay, you can shut down the window without saving it. If you need to change the code, make sure you save it before you close the window.
How the program is sent to the machine depends on the shop setup.

- Select the "**X**" box at the upper right corner to exit the **Mastercam Code Expert.**

STEP 18: SAVE THE UPDATED MCAM FILE

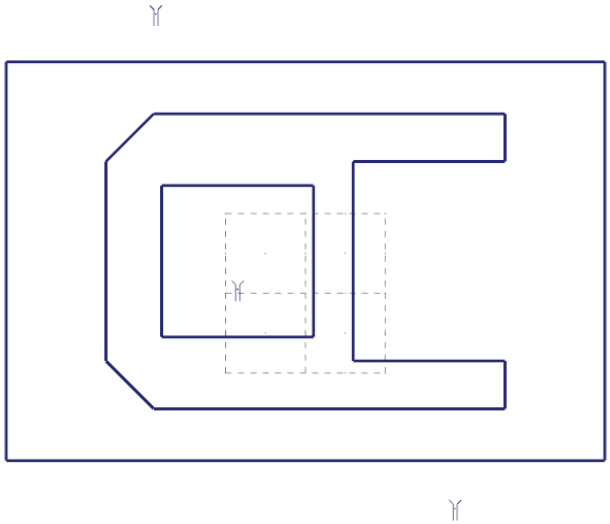
REVIEW EXERCISE -STUDENT PRACTICE



CREATE THE GEOMETRY FOR TUTORIAL #4 EXERCISE

Use these commands to create the geometry.

- Rectangle.
- Circle Center Point.
- Line Polar.
- Arc Tangent.
- Line Tangent.
- Line Parallel.
- Trim Entities.
- Mirror.
- Point Position - Thread Point.

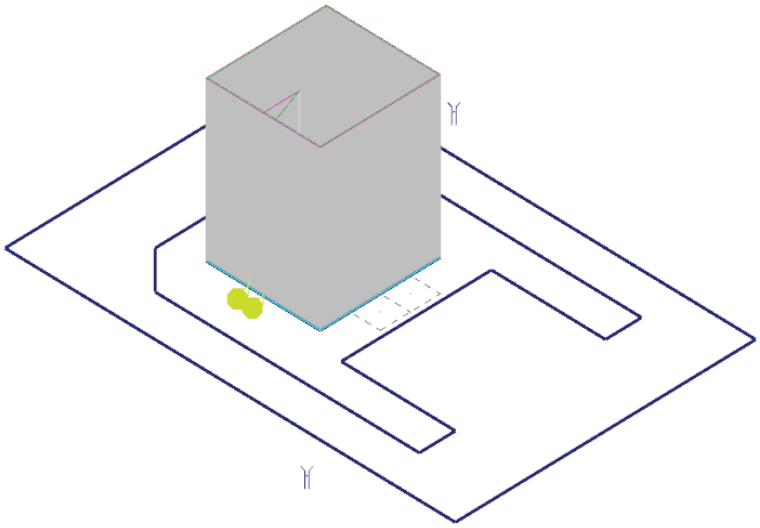


CREATE THE TOOLPATHS FOR TUTORIAL #4 EXERCISE

Create the Toolpaths for Tutorial #4 Exercise as per the instructions below.

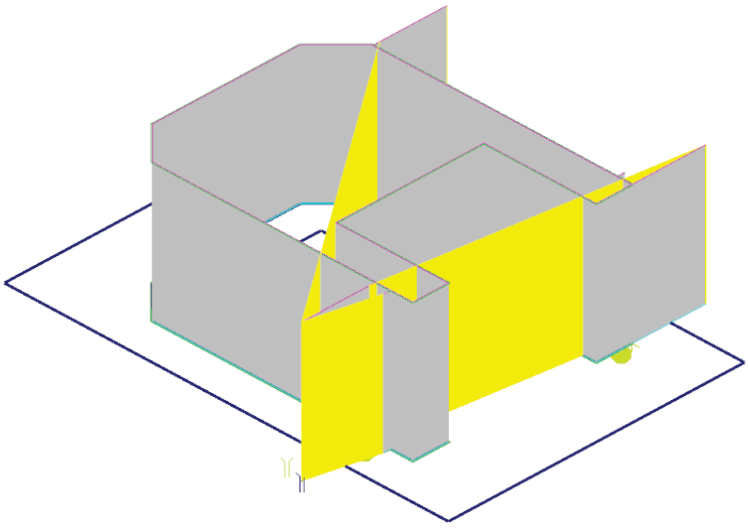
Machine The Inner Profile using Contour Wirepath.

- In order to complete this operation follow **page 252** to **page 264**.



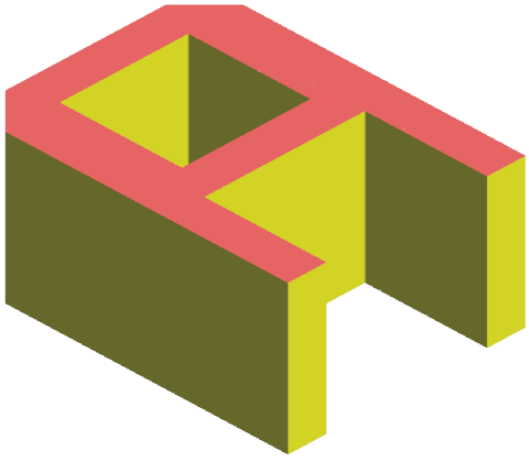
Machine The Outer Profile using Contour Wirepath.

- In order to complete this operation follow **page 266** to **page 279.**



Verify The Wirepath.

- Set the **Stock** according to the part.
- In order to complete this operation **Verify** the part and follow **page 282** to **page 285**.



NOTES:

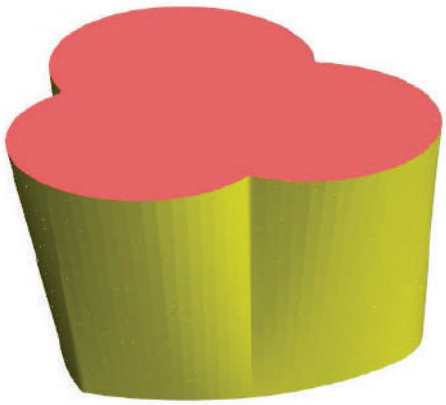
QUIZ #4

- Where do you have to select the entities when using Trim 2 entities?

- What does Compensation type set to Control output?

- What does Leads page allow you to set?

TUTORIAL #5



OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

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 3D CAD Model used to generate Wirepaths from:

- The student will create the upper and the lower 2D profiles that are needed to create the wirepaths.
- The student will also create the branches used to synchronize the upper and lower profiles.
- Geometry creation commands such as Create Rectangle, Circle Center Point, Arc Tangent, Fillet, Trim, Mirror, and Create Thread Point will be used. You will also learn how to use different construction depths.

Create the necessary Wirepaths to machine the part:

- 4 Axis wirepath to cut the shape.

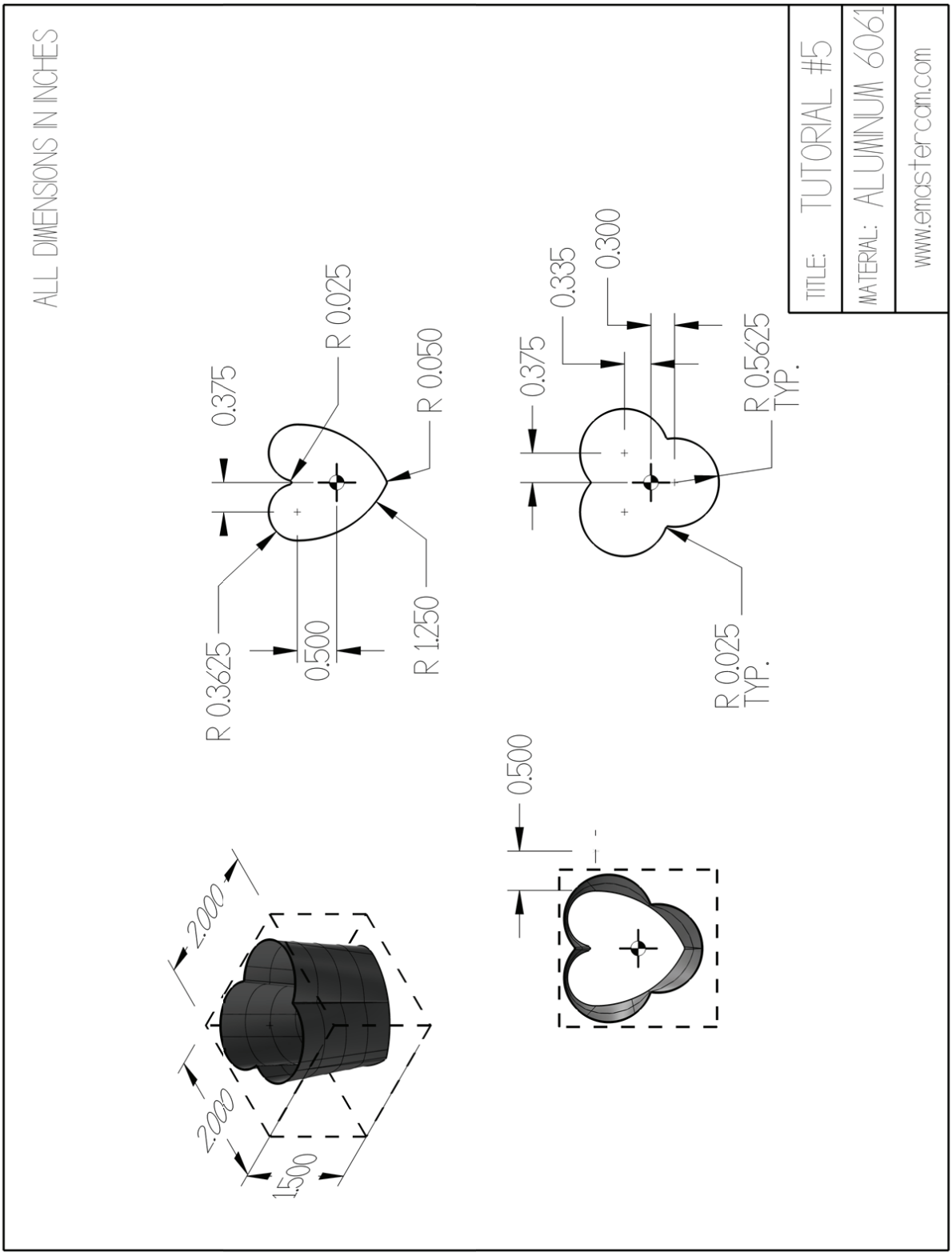
Backplot and the file:

- 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 then post process the file to obtain an NC file containing the necessary code for the machine.

This tutorial takes approximately one hour to complete.


ALL DIMENSIONS IN INCHES
TITLE: TUTORIAL #5
MATERIAL: ALUMINUM 6061
www.emastercam.com
R 0.025
R 0.050
0.375
R 0.3625
0.500
R 1.250
0.335
0.300
0.375
R 0.5625 TYP.
R 0.025 TYP.
0.500
2.000
2.000
1.500

GEOMETRY CREATION

STEP 1: SETTING UP THE GRAPHIC USER INTERFACE

Please refer to the **Getting Started** section to set up the graphical user interface.

STEP 2: CHANGE THE Z DEPTH OF THE CPLANE AND SWITCH TO 2D MODE

In this step you will learn how to change the Z depth of the Construction Plane (in our case, Top plane) and how to switch to 2D construction mode. In 2D mode, all the geometry is created parallel to the current construction plane and at the current Z depth setting. The top geometry of the part will be created in the 2D mode at Z depth 1.5.

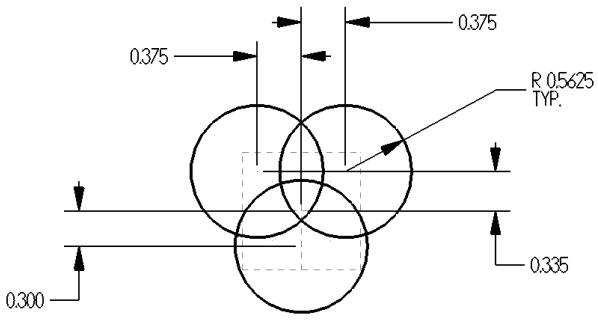
- Right mouse click in the graphic window and make sure that the **Construction mode** is set to **2D**.

- Enter **1.5** to change the **Z depth** of the construction plane.

STEP 3: CREATE THE 0.5625" RADIUS CIRCLES

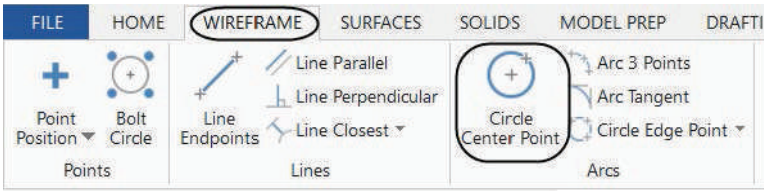
In this step you will create the three circles given the radius and the center locations.

Step Preview:

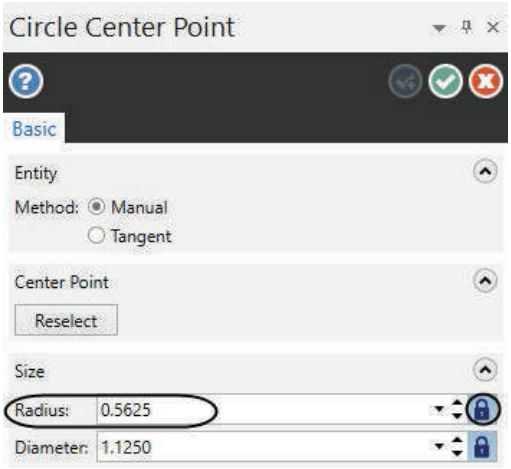


WIREFRAME

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



- Enter the Radius **0.5625** and click on the locker icon to lock the value as shown.



- [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.

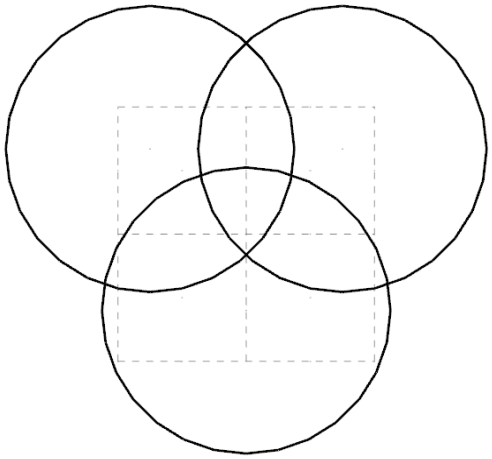
- Enter the coordinate values **-0.375 , 0.335** as shown.

- Press **Enter** twice to create the circle.
- [Enter the center point]: Select the **AutoCursor Fast Point** icon again and enter **0.375, 0.335**.

- Press **Enter** twice to create the circle.
- [Enter the center point]: Select the **AutoCursor Fast Point** icon again and enter **0, -0.3**

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

- The geometry should look as shown.

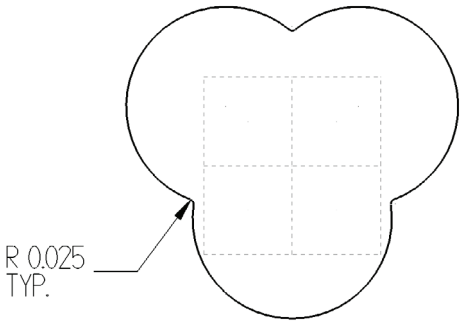


NOTE: While creating the geometry for this tutorial, if you make a mistake, you can undo the last step using the **Undo** icon. You can undo as many steps as needed. If you delete or undo a step by mistake, just use the **Redo** icon. To delete unwanted geometry, select the geometry first and then press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

STEP 4: CREATE FILLETS WITH THE RADIUS 0.025"

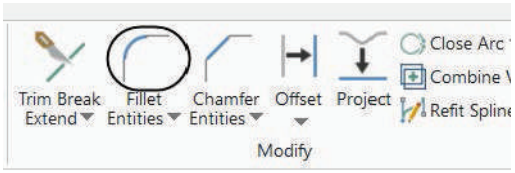
In this step you will clean the geometry by creating the 0.025" fillets.

Step Preview:

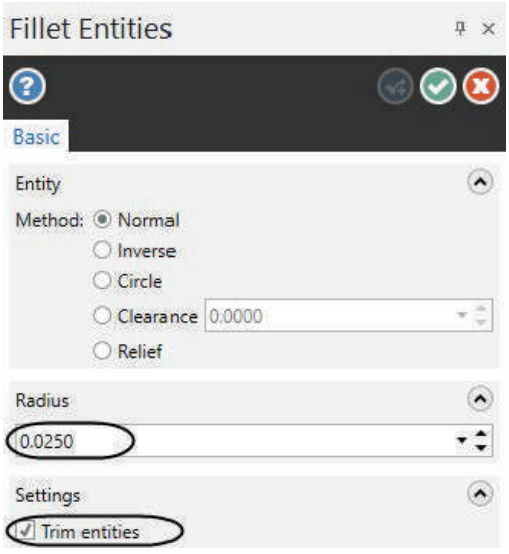


WIREFRAME

- From the **Modify** area, select **Fillet Entities.**



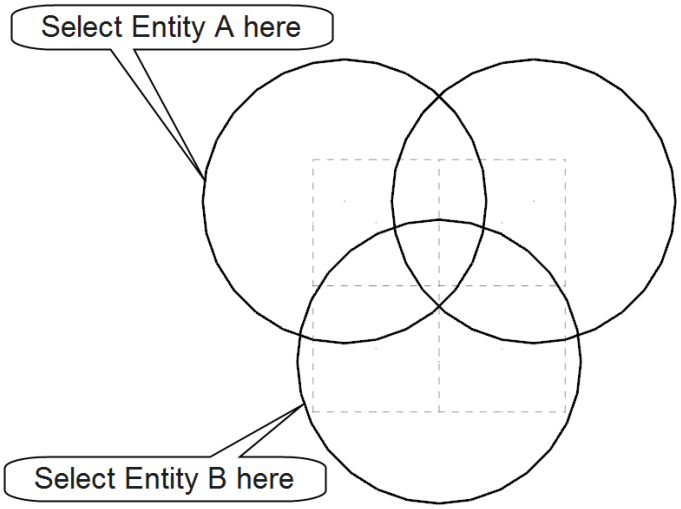
- Enter the **Radius 0.025** and ensure **Trim entities** is enabled as shown.



- [Fillet: Select an entity]: Select Entity A as shown in Figure: 4.0.1.
- [Fillet: Select another entity]: Select Entity B as shown in Figure: 4.0.1.

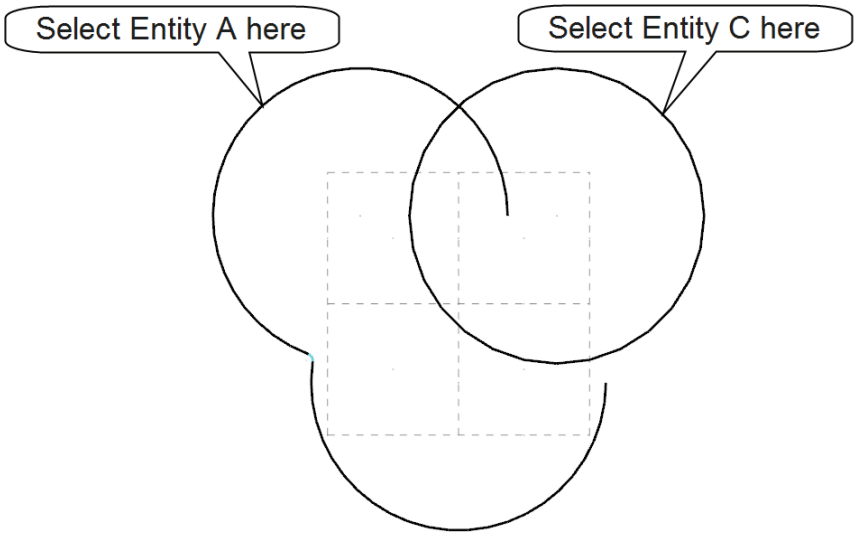
Figure: 4.0.1

NOTE: Make sure that you select the entities close to the locations shown in Figure: 4.0.1.



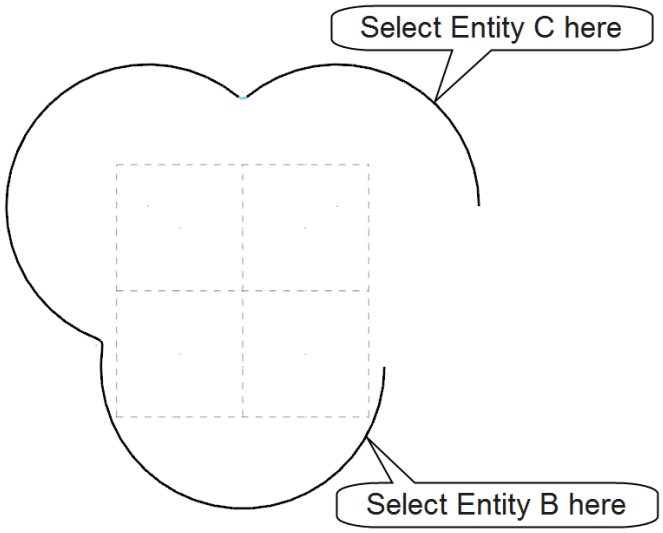
- [Fillet: Select an entity]: Select Entity A again as shown in Figure: 4.0.2.
- [Fillet: Select another entity]: Select Entity C as shown in Figure: 4.0.2.

Figure: 4.0.2

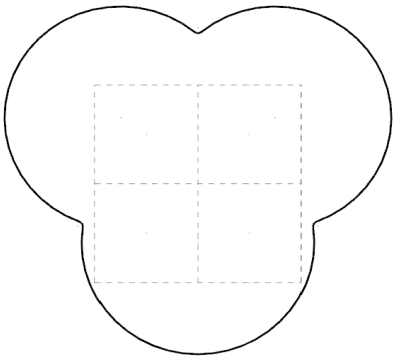


- [Fillet: Select an entity]: Select Entity C again as shown in Figure: 4.0.3.
- [Fillet: Select another entity]: Select Entity B as shown in Figure: 4.0.3.

Figure: 4.0.3



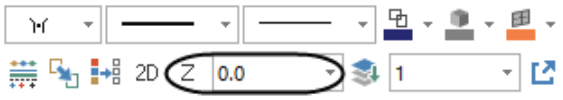
- Select the **OK** button to exit the command.
- The top geometry should look as shown.



STEP 5: CHANGE THE Z DEPTH OF THE CPLANE TO Z 0.0

In this step you will change the Z depth of the Construction Plane back to 0.0. The bottom geometry of the part should be created at Z 0.0.

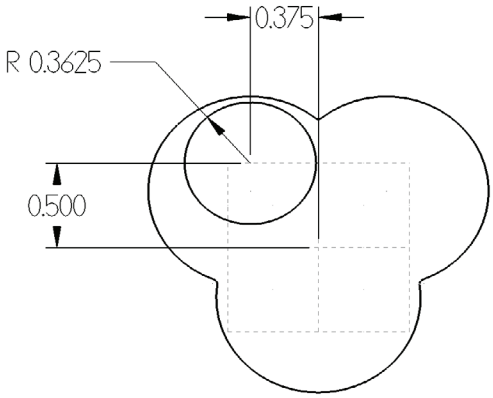
- Right mouse click in the graphics window and from the **Mini Toolbar**, click in the **Z** field and enter **0.0** to change the **Z depth** of the construction plane.



STEP 6: CREATE THE 0.3625" RADIUS CIRCLE

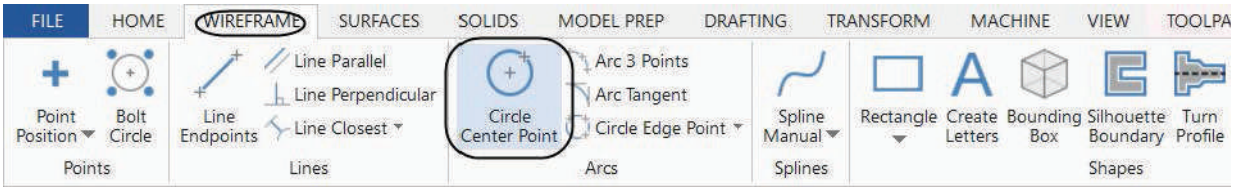
In this step you will create the circle given the radius and the center locations.

Step Preview:

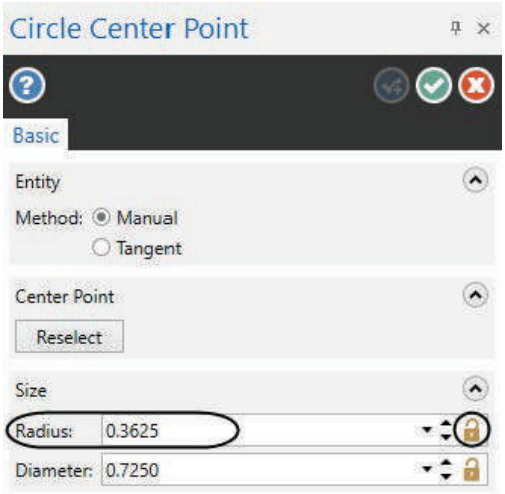


WIREFRAME

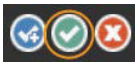
- From **Arcs** group, select **Center Point.**

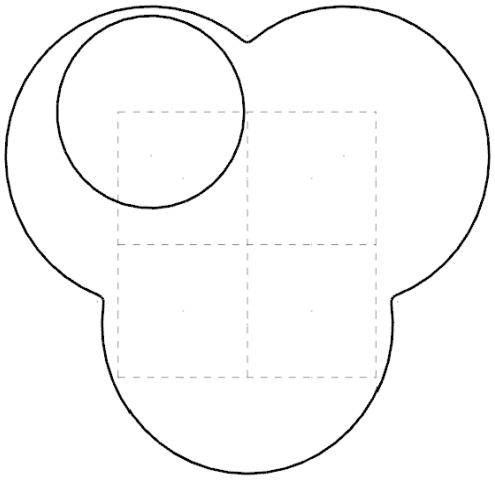


- Click on the locker icon to unlock the radius and enter the **Radius 0.3625**.



- [Enter the center point]: Select the **AutoCursor Fast Point** icon and enter **-0.375, 0.5**

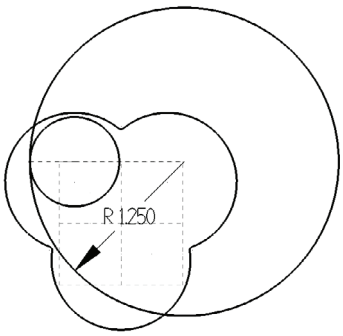
- Select the **OK** button to exit the command. 
- Press **Alt+F1** to fit the drawing to the screen.
- The geometry should look as shown.



STEP 7: CREATE THE 1.25" RADIUS TANGENT THROUGH A POINT ARC

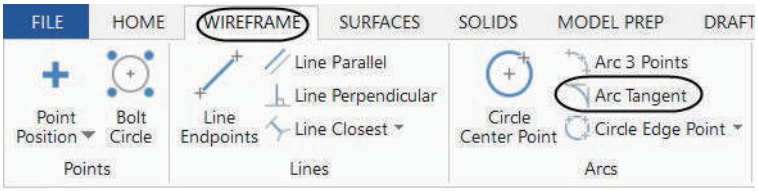
In this step you will create an arc tangent given the radius of the arc and the tangent point.

Step Preview:

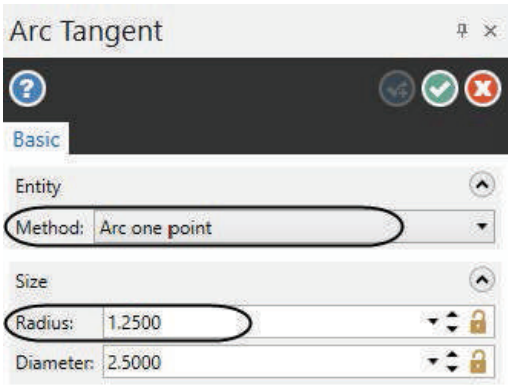


WIREFRAME

- From the **Arcs** group, select **Arc Tangent**.

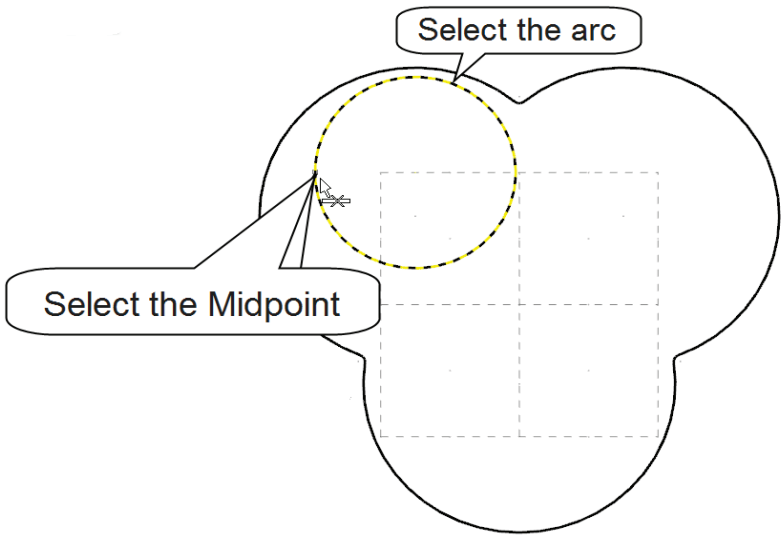


- Make sure that the **Method** is set to **Arc one point** as shown. Then enter the **Radius 1.25**..



- [Select the entity that the arc is to be tangent to]: Select the arc as shown in Figure: 7.0.1.

Figure: 7.0.1

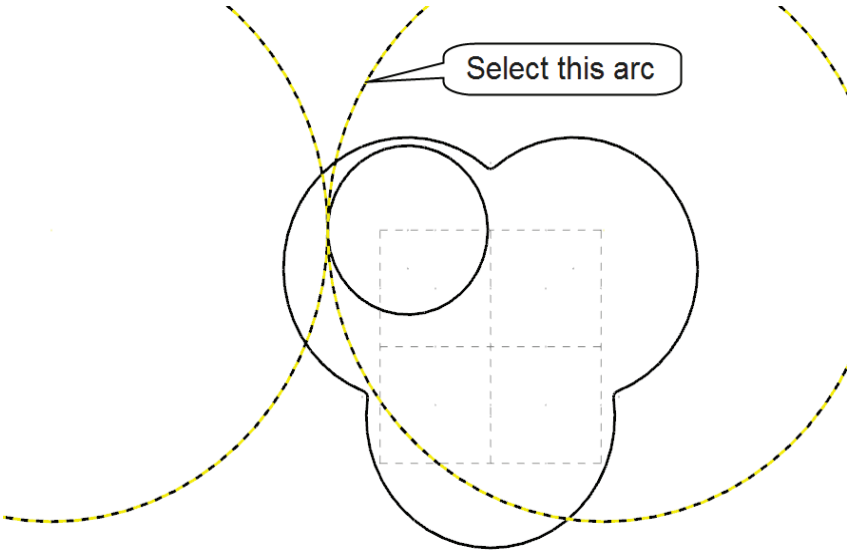


- [Specify the thru point]: Select the **Midpoint** of the arc as shown in Figure: 7.0.1.

NOTE: While creating an arc tangent through a point, Mastercam generates all possible solutions and prompts you to select the arc that you want to keep.

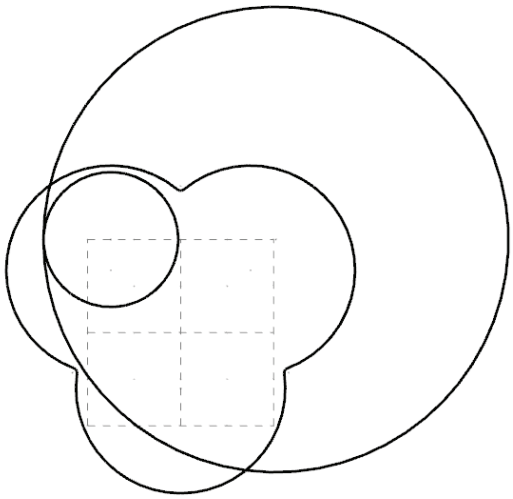
- [Select an arc]: Select the arc as shown in Figure: 7.0.2.

Figure: 7.0.2



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

- The geometry should look as shown.

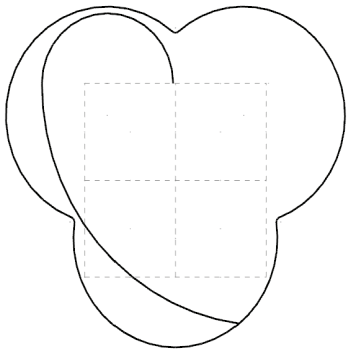


STEP 8: CLEAN THE GEOMETRY USING TRIM DIVIDE/DELETE

Divide command divides or deletes entities based on the nearest intersection.

NOTE: During the trimming process it is very important to select the entities exactly at the locations that are shown in the graphics in the next few pages. Mastercam's auto-preview feature displays the results of the selected function as you move the mouse over the final entity selection. A solid line represents what will be created, while a dashed line represents what will be removed.

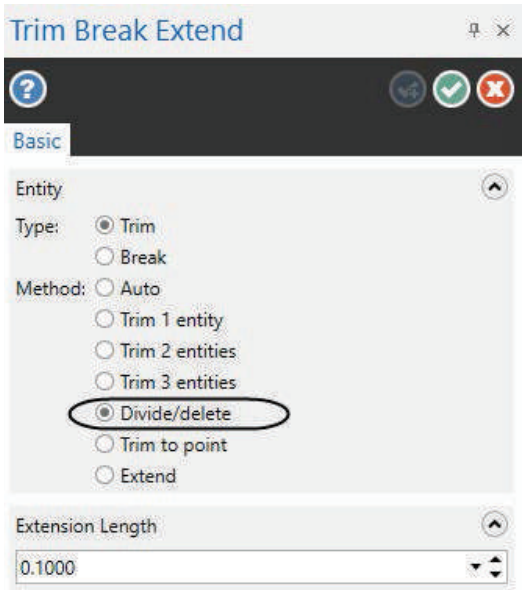
Step Preview:



WIREFRAME

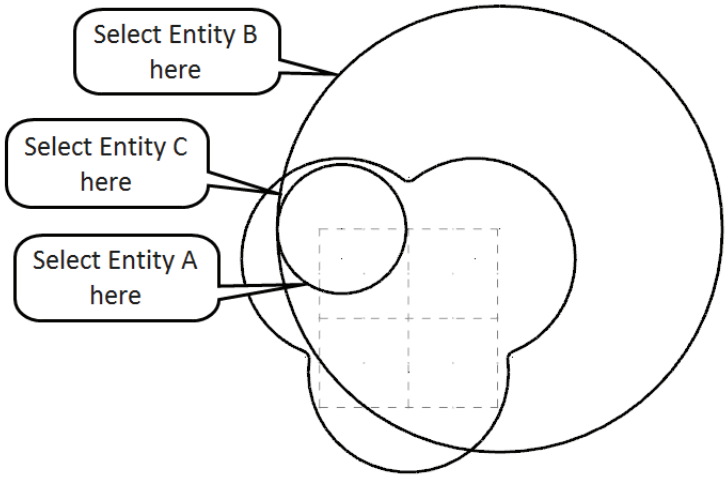
- From the **Modify** group, select **Trim Break Extend**.

- Enable **Divide/delete** as shown.



- [Select the curve to divide/delete]: Select Entity A as shown in Figure: 8.0.1.
- [Select the curve to divide/delete]: Select Entity B as shown in Figure: 8.0.1.
- [Select the curve to divide/delete]: Select Entity C as shown in Figure: 8.0.1

Figure: 8.0.1

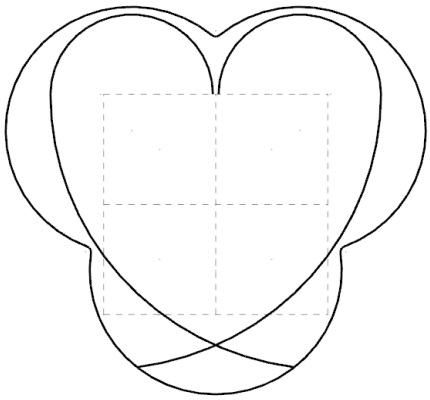


- Select the **OK** button to exit the command.

STEP 9: MIRROR THE GEOMETRY ABOUT Y-AXIS

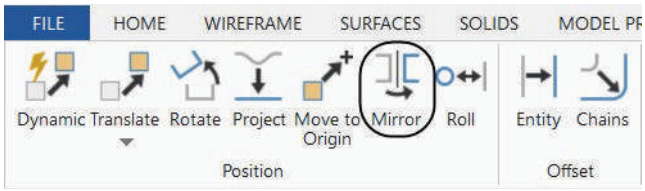
Mirror command allows you to create mirror images of geometric and drafting entities by reflecting them symmetrically with respect to a defined axis or point, selected line or theoretical line defined between two points.

Step Preview:



TRANSFORM

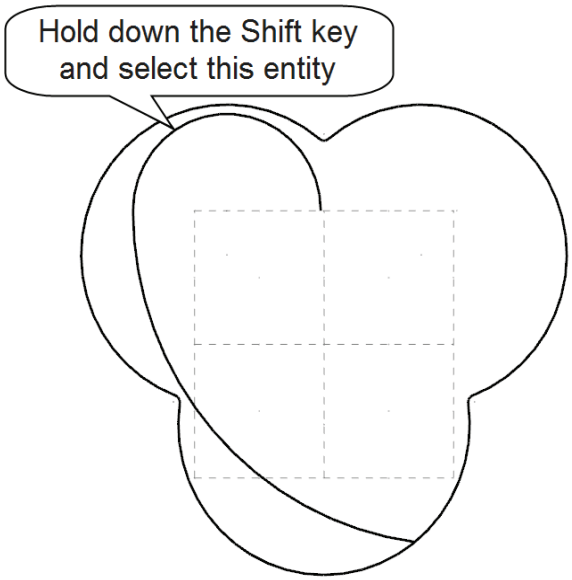
- From the **Position** group, select the **Mirror icon**.



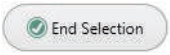
NOTE: To select all entities for mirror we are using the chain selection. Holding down the shift key and select one entity of the chain. Mastercam selects the entire chain.

- [Mirror: select entities to mirror]: Hold down the **Shift** key and select the entity as shown in Figure: 9.0.1.

Figure: 9.0.1

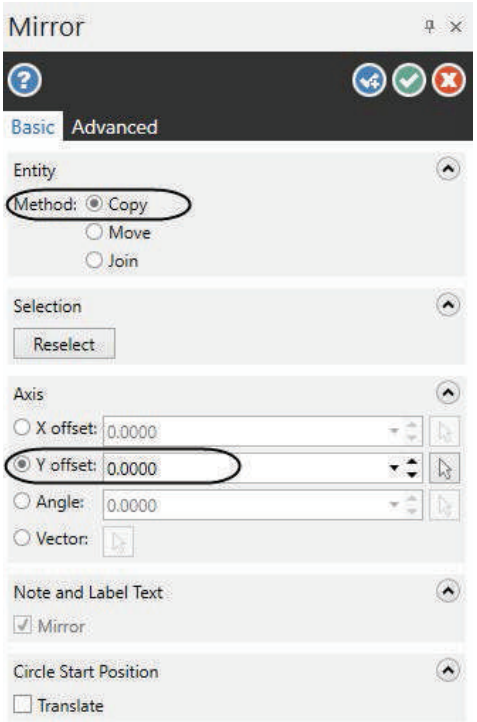


- Select the **End Selection** button (or press **Enter**) to finish the selection.

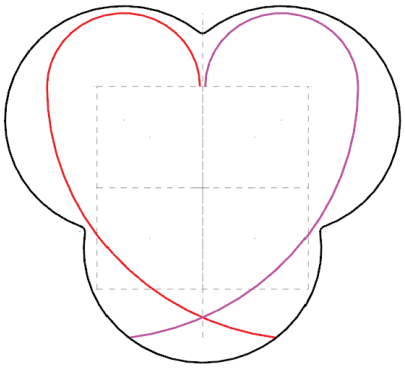


- In the **Mirror** panel make sure that **Copy** is selected and then enable **Y offset** as shown in Figure: 9.0.2.

Figure: 9.0.2



- The preview should look as shown.



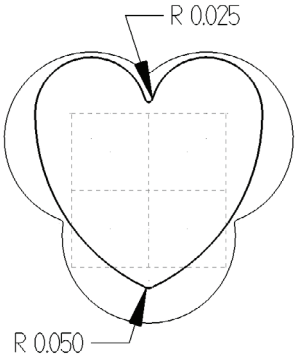
NOTE: When performing a transform function, Mastercam creates a temporary group from the originals (red) and a result (purple) from the transformed entities. However, they stay in effect only until you use the **Screen**, **Clear Colors** function or perform another transform function.

- Select the **OK** button to exit the **Mirror** panel.
- Right mouse click in the graphics window and select **Clear Colors** as shown.

STEP 10: CREATE FILLETS WITH THE RADIUS 0.025" AND 0.05"

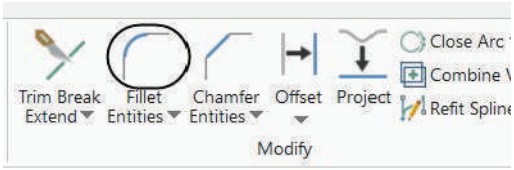
In this step you will clean the geometry by creating the 0.025" and the 0.05" fillets.

Step Preview:

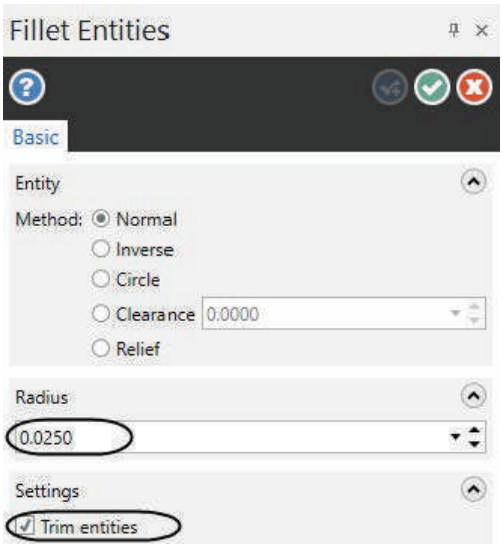


WIREFRAME

- From the **Modify** area, select **Fillet Entities.**

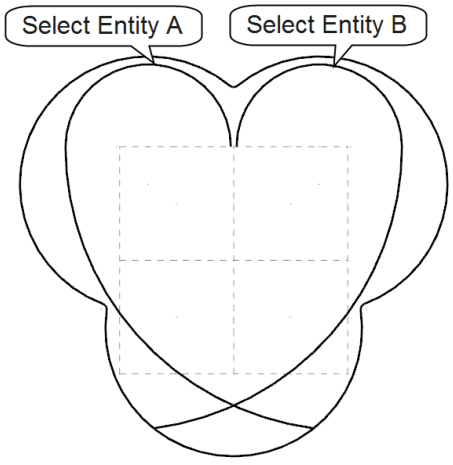


- Enter the **Radius 0.025** and ensure **Trim entities** is enabled as shown.



- [Fillet: Select an entity]: Select Entity A as shown in Figure: 10.0.1.
- [Fillet: Select another entity]: Select Entity B as shown in Figure: 10.0.1.

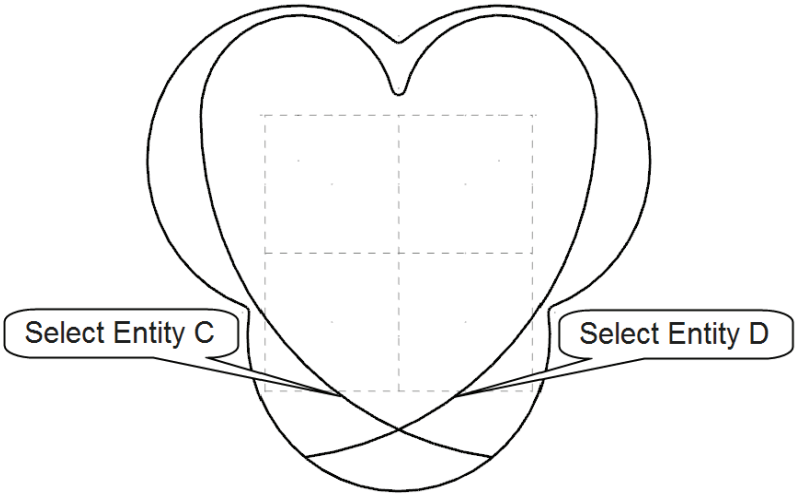
Figure: 10.0.1



NOTE: Make sure that you select the entities close to the locations shown in Figure: 10.0.1.

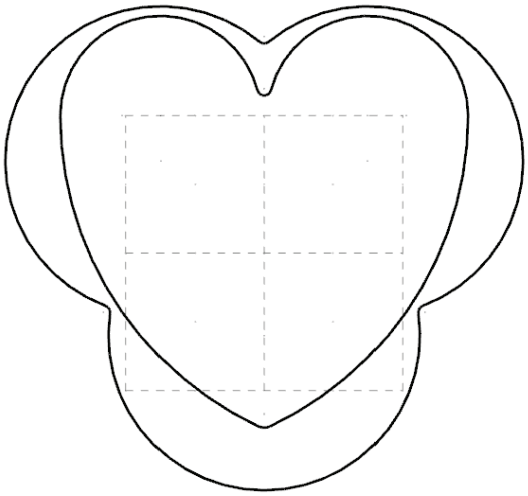
- Select **OK and Create New Operation** to finish the fillet.
- Enter the **Radius 0.05.**
- [Fillet: Select an entity]: Select Entity C as shown in Figure: 10.0.2.
- [Fillet: Select another entity]: Select Entity D as shown in Figure: 10.0.2.

Figure: 10.0.2

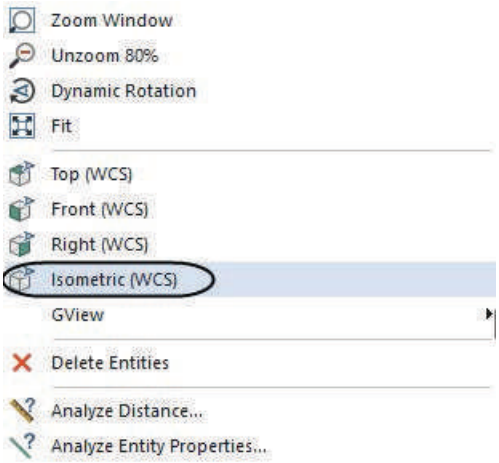


- Select the **OK** button to exit the command.

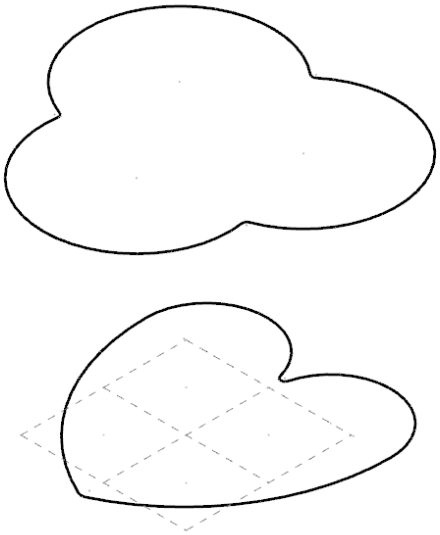
- The bottom geometry should look as shown.



- Right mouse click in the graphics window and select **Isometric** as shown.



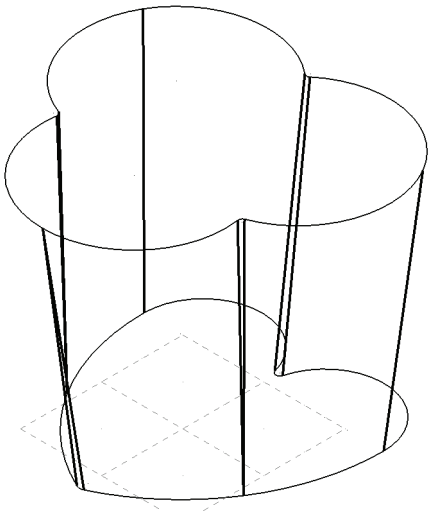
- Press **Alt+F1** to fit the drawing to the screen.
- The geometry should look as shown.



STEP 11: CREATE BRANCH LINES

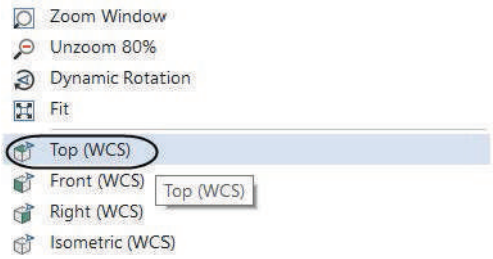
To create a proper 4-Axis wirepath, Mastercam needs to synch the top and the bottom chain to align them. There are different methods that allows you to synchronize the geometry; the preferred ones for a wirepath are **by Entity** and **by Branch**.
In our case we will use **by Branch**.
By Branch matches the chains at branch points. You will need to break the geometry where you want to create the branches and then create the lines that connect the top with the bottom geometry.

Step Preview:



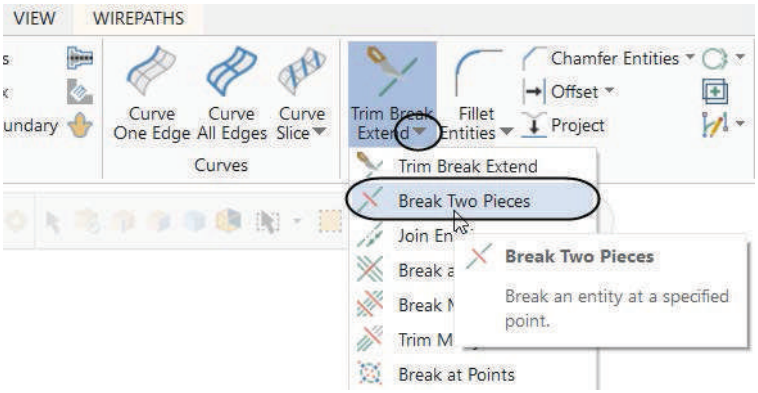
11.1 Break the top geometry

- Right mouse click in the graphics window and select **Top** as shown.



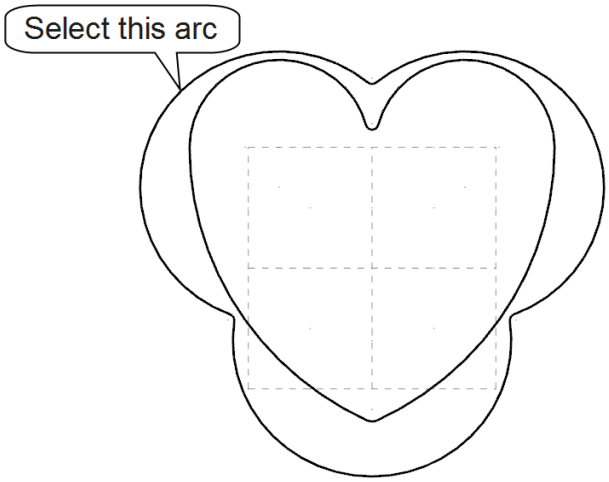
WIREFRAME

- From the **Modify** group, click on the down arrow under **Trim Break Extend** and select **Break Two Pieces** as shown.



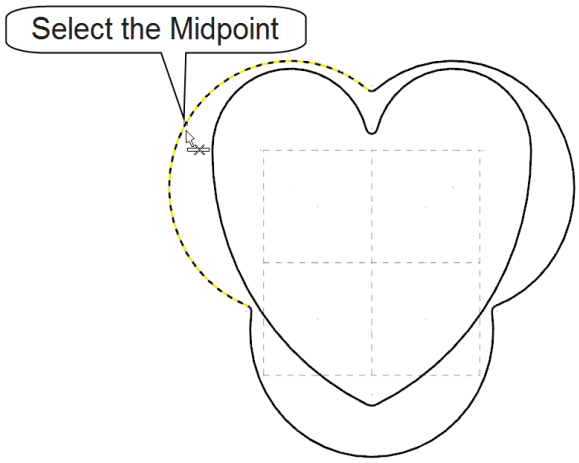
- [Select an entity to break]: Select the arc as shown in Figure: 11.1.1.

Figure: 11.1.1



- [Indicate the break position]: Select the **Midpoint** of as shown in Figure: 11.1.2.

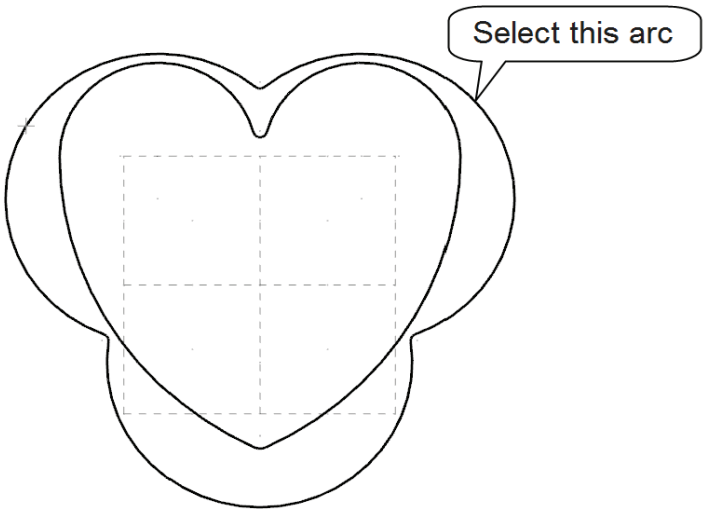
Figure: 11.1.2



NOTE: Mastercam creates a temporary point at the break point position. The point will disappear once you select the next entity that has to be broken.

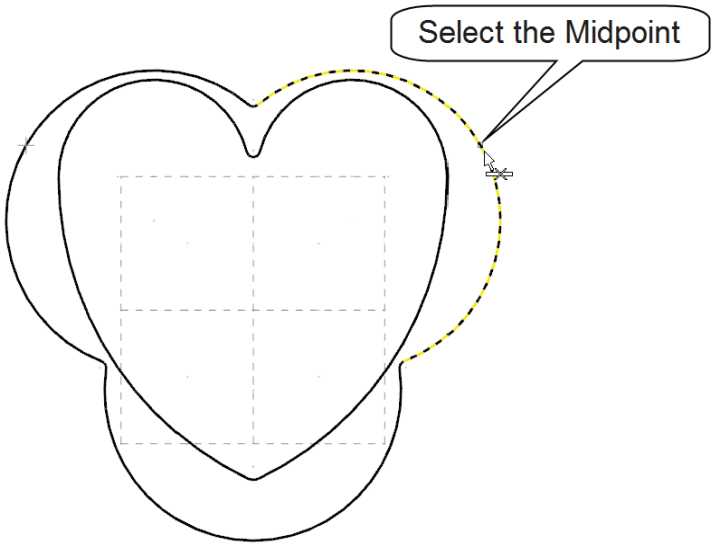
- [Select an entity to break]: Select the arc as shown in Figure: 11.1.3.

Figure: 11.1.3



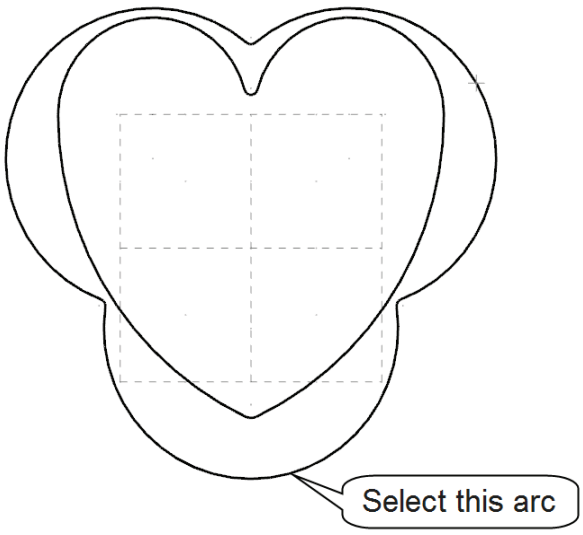
- [Indicate the break position]: Select the **Midpoint** of the arc as shown in Figure: 11.1.4.

Figure: 11.1.4



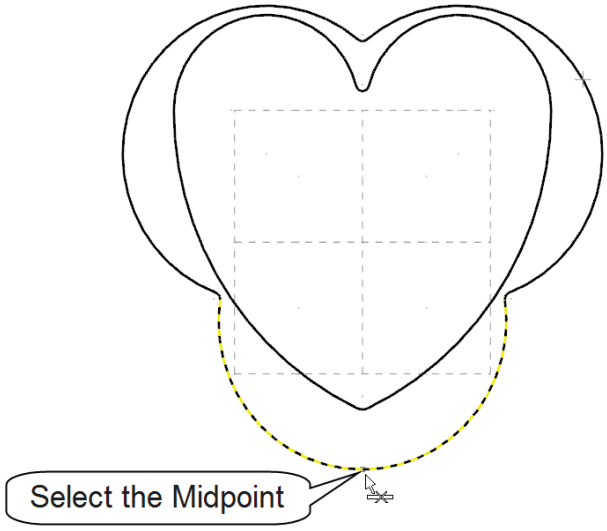
- [Select an entity to break]: Select the arc as shown in Figure: 11.1.5.

Figure: 11.1.5



- [Indicate the break position]: Select the **Midpoint** of the arc as shown in Figure: 11.1.6.

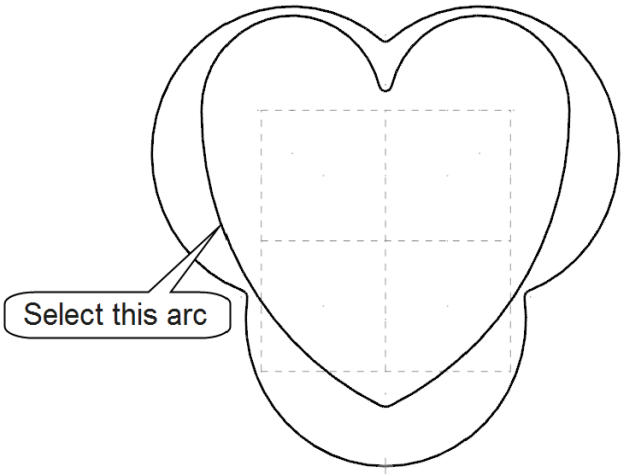
Figure: 11.1.6



11.2 Break the bottom geometry

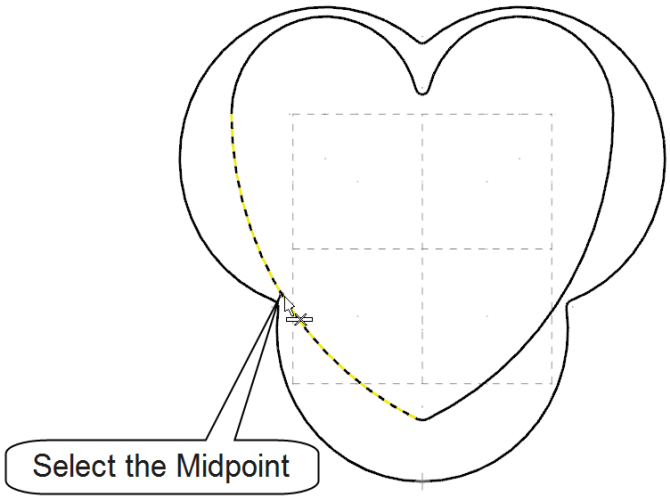
- [Select an entity to break]: Select the arc as shown in Figure: 11.2.1.

Figure: 11.2.1



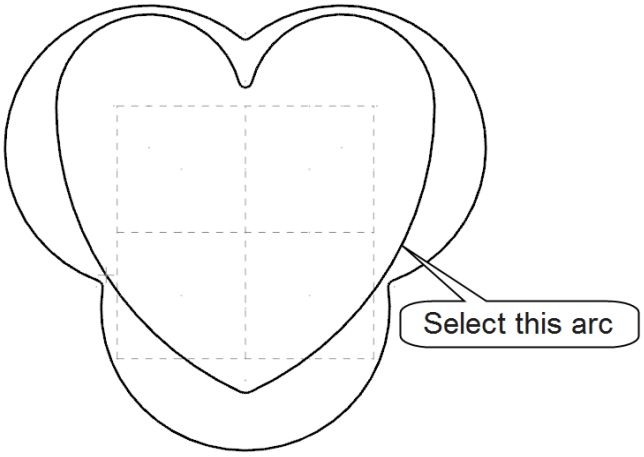
- [Indicate the break position]: Select the **Midpoint** of as shown in Figure: 11.2.2.

Figure: 11.2.2



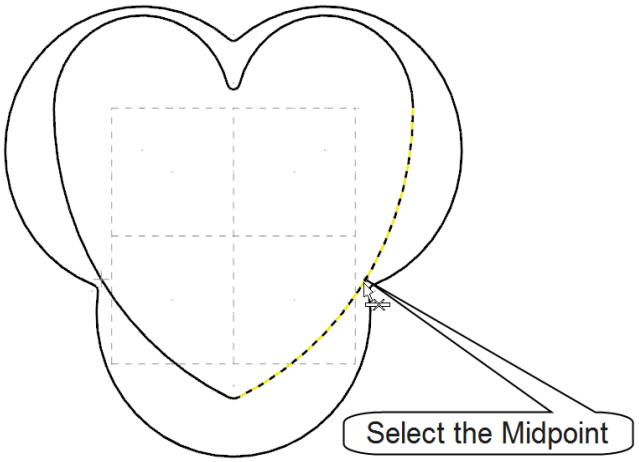
- [Select an entity to break]: Select the arc as shown in Figure: 11.2.3.

Figure: 11.2.3



- [Indicate the break position]: Select the **Midpoint** of as shown in Figure: 11.2.4.

Figure: 11.2.4

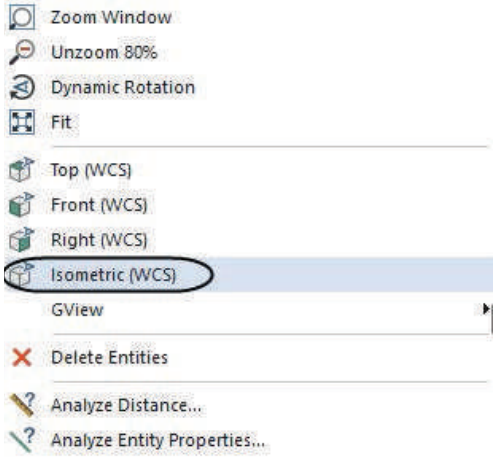


- Press **Esc** to exit the command.

11.3 Create points at the endpoints of all the entities

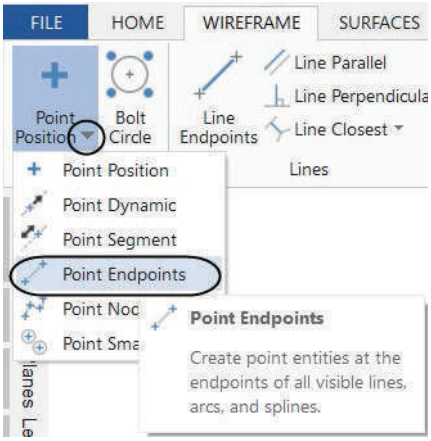
- Right mouse click in the graphics window and change the **Construction mode** to **3D**.

- Right mouse click in the graphics window and select **Isometric** as shown.



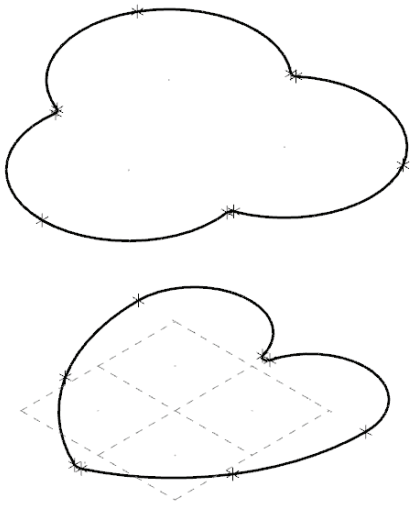
WIREFRAME

- From the **Points** group, click on the pull down arrow under **Point Position** and select **Point Endpoints** as shown.



NOTE: Mastercam automatically creates points at the endpoints of all the entities. You may need to change **Type** to **Point** in **Point Position** panel.

- The geometry should look as shown.

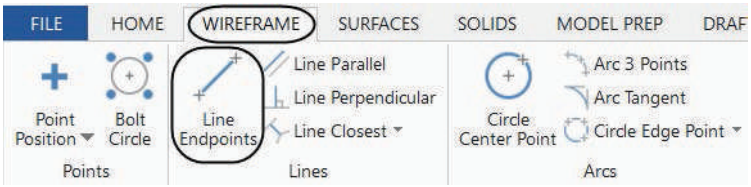


11.4 Create the lines using Line Endpoint command

- Right mouse click on the graphics window and select **Top** as shown.

WIREFRAME

- From the **Lines** group, select the **Line Endpoints** icon.



NOTE: Make sure that you are selecting the Points that were generated previously. The cursor icon for the Point should appear each time you select a point.

- [Specify the first endpoint]: Select P1 as shown in Figure: 11.4.1.
- [Specify the second endpoint]: Select P10 as shown in Figure: 11.4.1.

Figure: 11.4.1

- [Specify the first endpoint]: Select P1 again.
- [Specify the second endpoint]: Select P11.

- [Specify the first endpoint]: Select P2.
- [Specify the second endpoint]: Select P12.

- [Specify the first endpoint]: Select P3.
- [Specify the second endpoint]: Select P12 again.

- [Specify the first endpoint]: Select P4.
- [Specify the second endpoint]: Select P13.

- [Specify the first endpoint]: Select P5.
- [Specify the second endpoint]: Select P14.

- [Specify the first endpoint]: Select P6.
- [Specify the second endpoint]: Select P15.

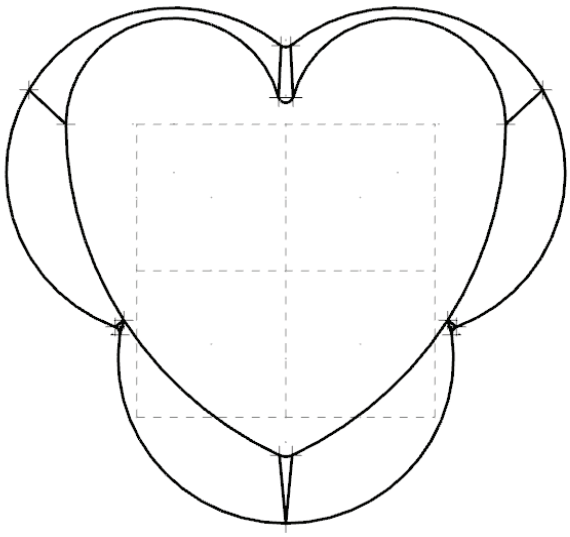
- [Specify the first endpoint]: Select P7.
- [Specify the second endpoint]: Select P16.

- [Specify the first endpoint]: Select P8.
- [Specify the second endpoint]: Select P17.

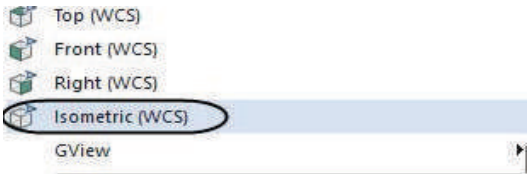
- [Specify the first endpoint]: Select P9.
- [Specify the second endpoint]: Select P17 again.

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

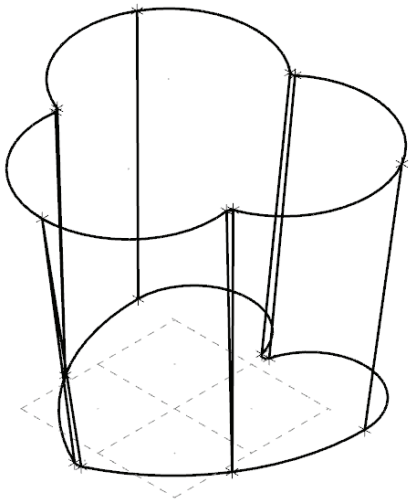
- The geometry should look as shown.



- Right mouse click in the graphics window and select **Isometric** as shown.

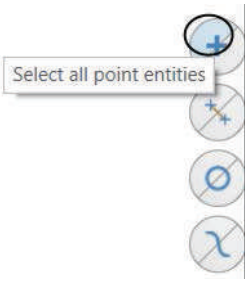


- The geometry should look as shown.

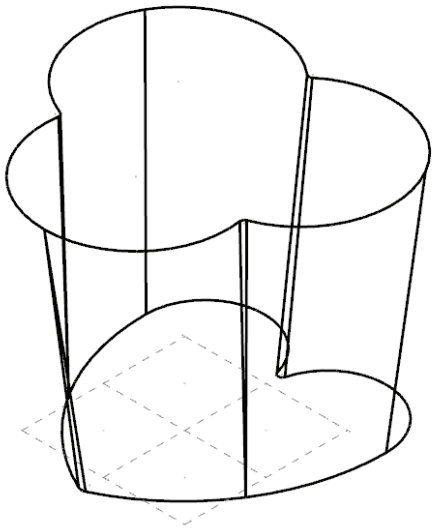


11.5 Delete all the points

- From the **Quick Mask** toolbar on the right side of the screen, click on the left side of the button to **Select all point entities** shown.



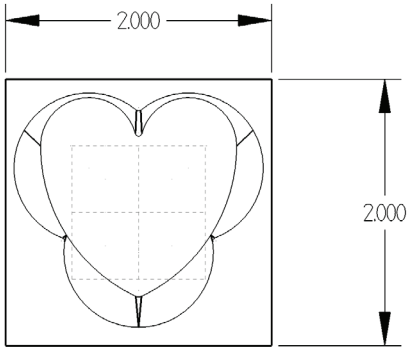
- Press **Delete** from the key board to delete all points.
- The geometry should look as shown.



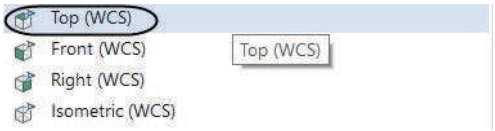
STEP 12: CREATE THE STOCK RECTANGLE

In this step you will learn how to create a rectangle given the width, the height, and the anchor location. This rectangle will be used to establish the stock.

Step Preview:

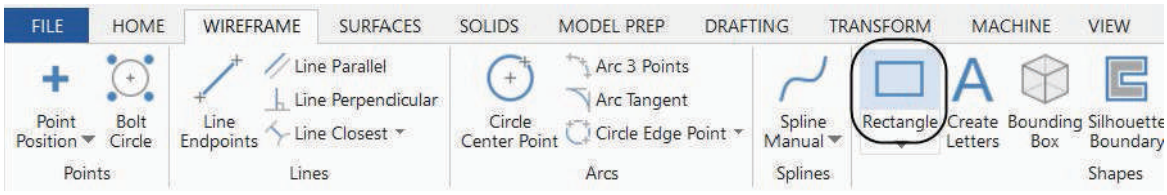


- Right mouse click on the graphics window and select **Top** as shown.

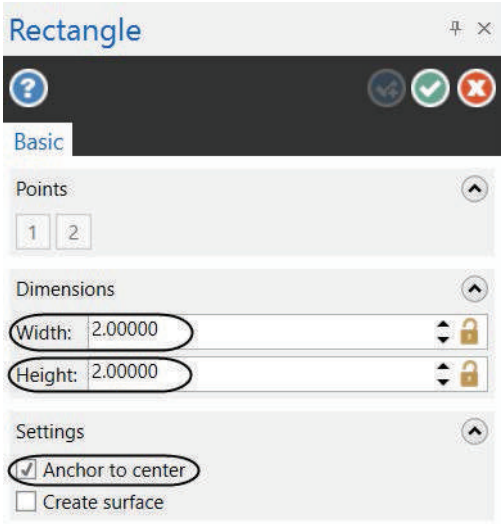


WIREFRAME

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

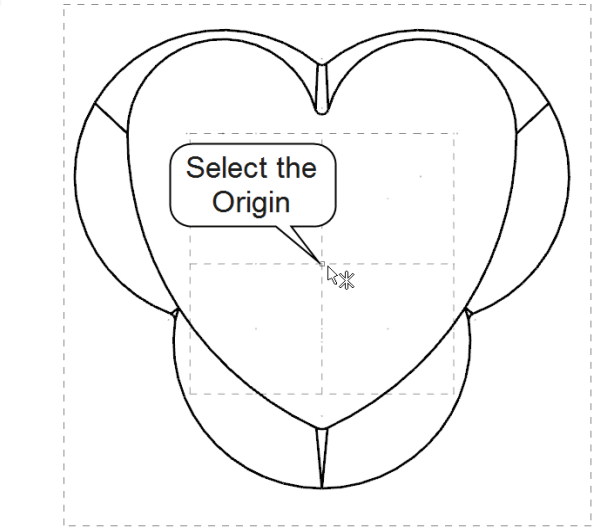


- Enter the **Width** and the **Height**, and enbale **Anchor to center** as shown.



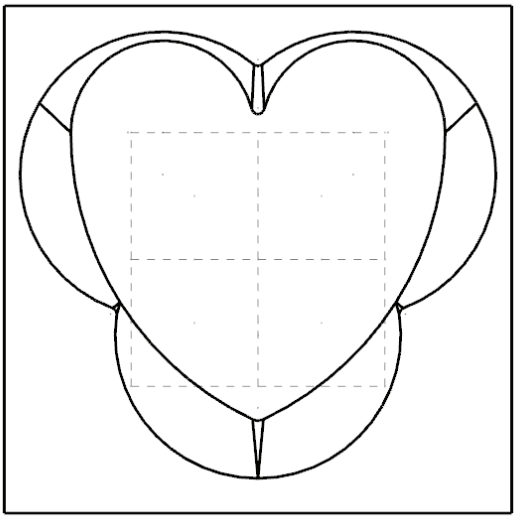
- Press **Enter** after typing the values to see a preview of the rectangle.
- [Select position of base point]: Select the Origin as shown in Figure: 12.0.1.

Figure: 12.0.1



- Make sure that when selecting the origin, the visual cue of the cursor changes as shown.
- Press **Alt+F1** to fit the drawing to the screen.
- Select the **OK** button to exit the **Rectangle** command.

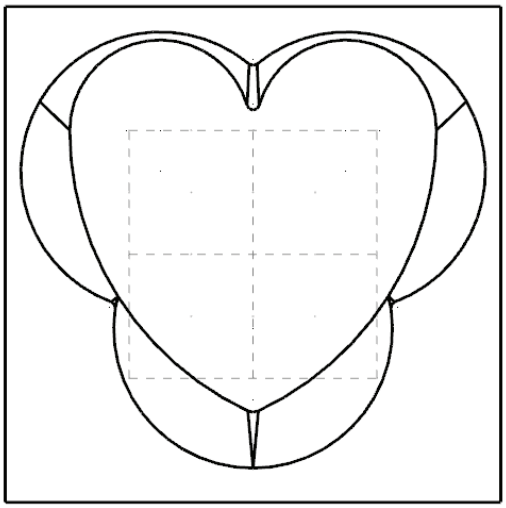
- The geometry should look as shown.



STEP 13: CREATE THE THREAD POINT

Thread Point is the point where the machine threads the wire, usually a pre-drilled hole in the material. The wire cuts from the thread position to the start of the chain based on the settings for lead ins and lead outs. See **Getting Started** to learn how to create the Wire utilities toolbar.

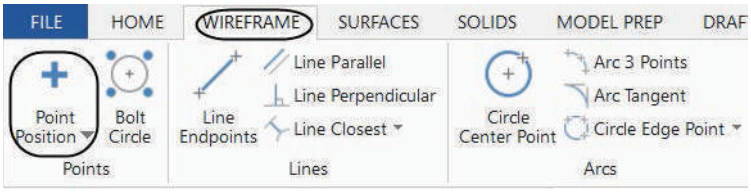
Step Preview:



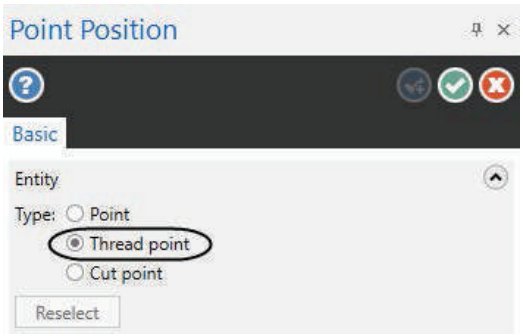
13.1 Create the Thread Point

WIREFRAME

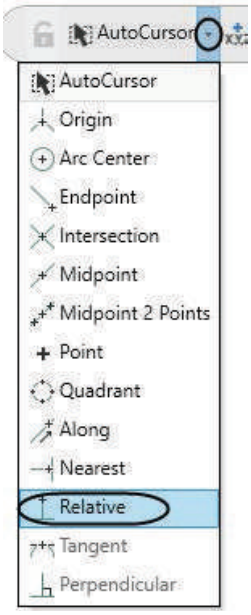
- From the **Points** group, select the **Point Position** icon.



- Select **Thread Point** as shown.

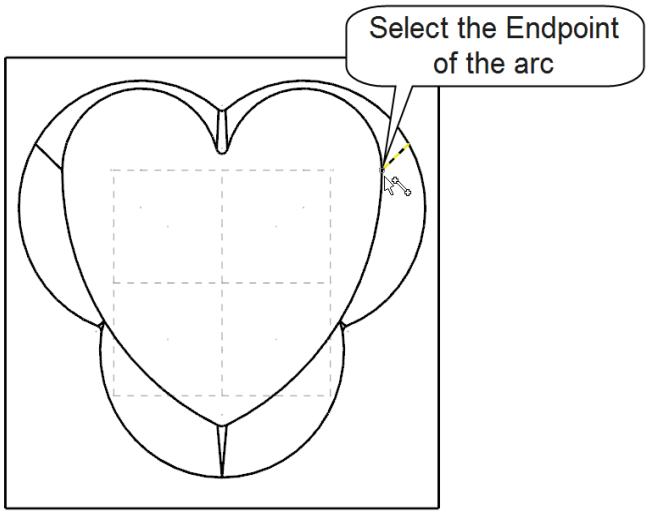


- Click on the drop down arrow next to the **AutoCursor** and select **Relative** as shown.

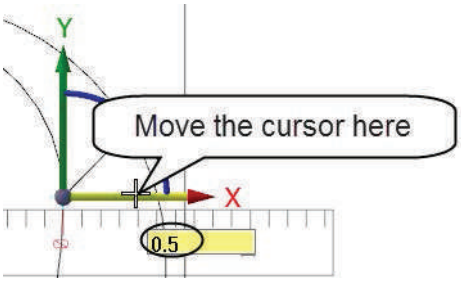


- [Pick a known point to use as a reference]: Select the **Endpoint** of the branch as shown in Figure: 13.1.1.

Figure: 13.1.1



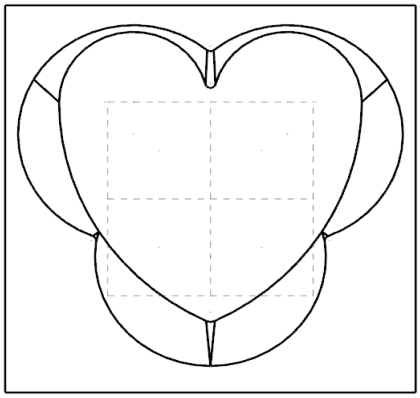
- Make sure that when selecting the **Endpoint**, the visual cue of the cursor changes as shown.
- Hover the cursor above the **X axis** as shown, then enter **0.5** to create the point below the line.



- Press **Enter** three times to create the point.

- Select the **OK** button to complete the command.
- Press **Alt+F1** to fit the drawing to the screen.

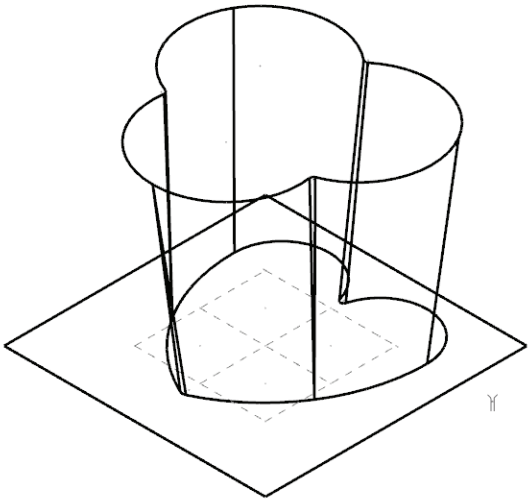
- The geometry should look as shown.



- Right mouse click in the graphics window and select **Isometric** as shown.

- Press **Alt+F1** to fit the drawing to the screen.

- The final geometry should look as shown.



STEP 14: SAVE THE FILE

FILE

- **Save As.**

- Click on the **Browse** icon as shown.
- Find a location on the computer to save your file. File name: "Tutorial 5_ 4 Axis".

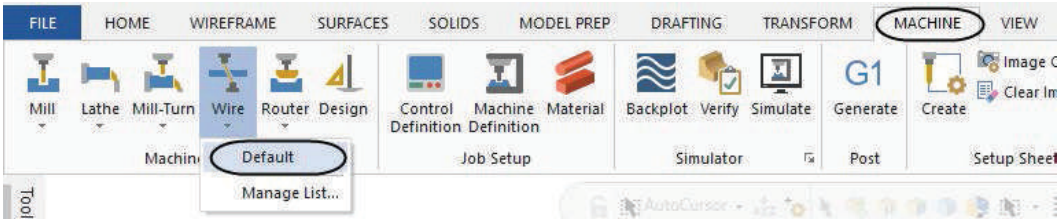
STEP 15: SELECT THE MACHINE AND SET UP THE STOCK

In Mastercam, you select a **Machine Definition** before creating any toolpaths. The **Machine Definition** is a model of your machine tool's capabilities and features and 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.WMD-9. Once finished, select OK to exit the dialog box. Otherwise, please follow next step.

MACHINE

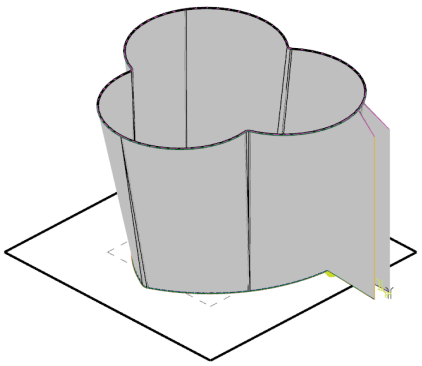
- From the **Machine Type** group, click on the pull down arrow under **Wire** and select **Default** from the list as shown.



STEP 16: MACHINE PART USING 4-AXIS WIREPATH

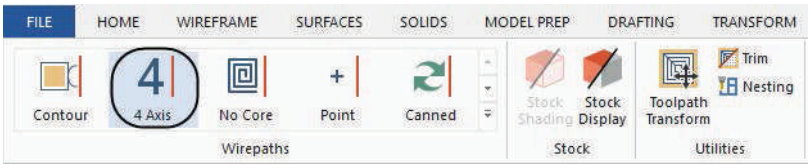
4-Axis wirepaths have geometry in both the XY plane (the lower contour) and UV plane (the upper contour). Where a contour wirepath derives a second contour from the geometry, a 4-Axis wirepath can have completely different geometry in the two planes, resulting in different wire movement in the XY and UV planes.

Toolpath Preview:

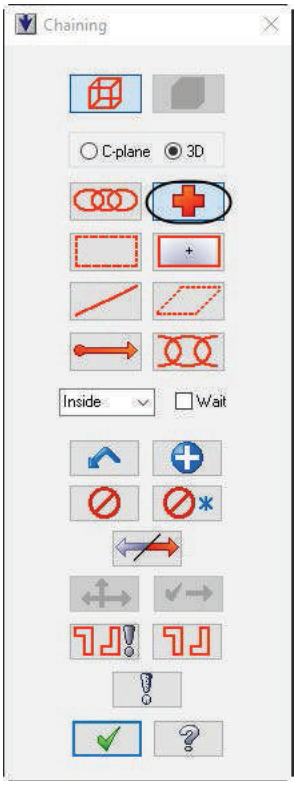


WIREPATHS

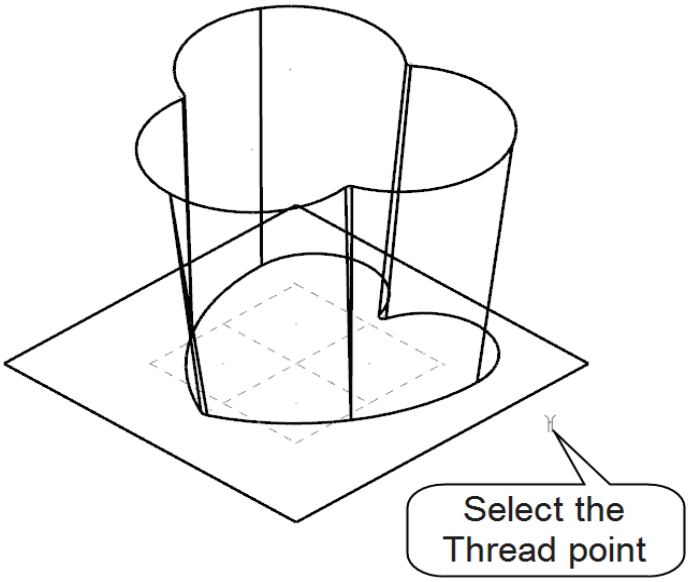
- From the **Wirepaths** group, select **4 Axis** as shown.



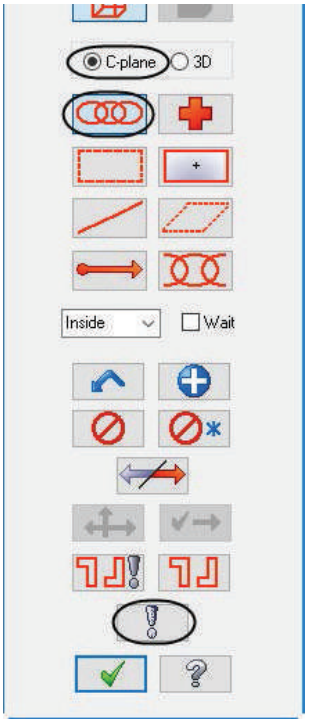
- From the **Chaining** dialog box, select the **Point** button.



- [Ruled: define chain 1]: Select the **Thread Point** as shown.



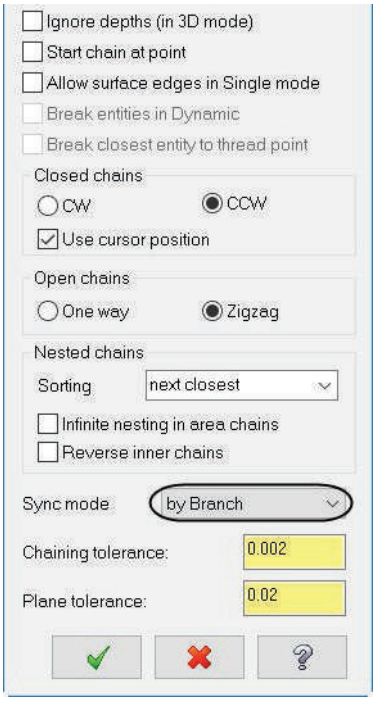
- From the **Chaining** dialog box, select the **Chain** button and enable **C-plane.**



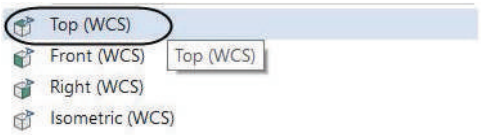
NOTE: C-plane allows you to chain only entities that are parallel to the current construction plane and at the same Z depth as the first entity you select.

- Select the **Options** button.

- Enable **Sync mode by Branch**.

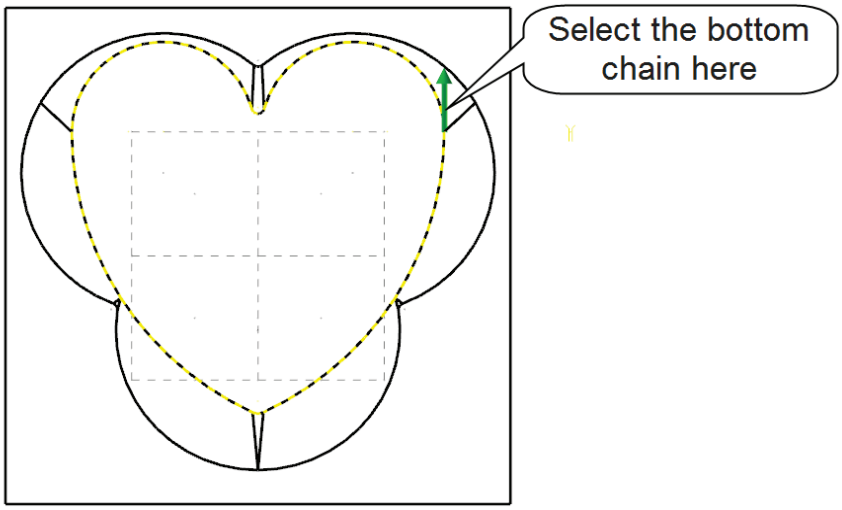


- Select the **OK** button to exit **Chaining Options**.
- Right mouse click in the graphics window and select the **Top** view as shown.



- [Ruled: define chain 2]: Select the bottom chain in the CCW direction as shown in Figure: 16.0.1.

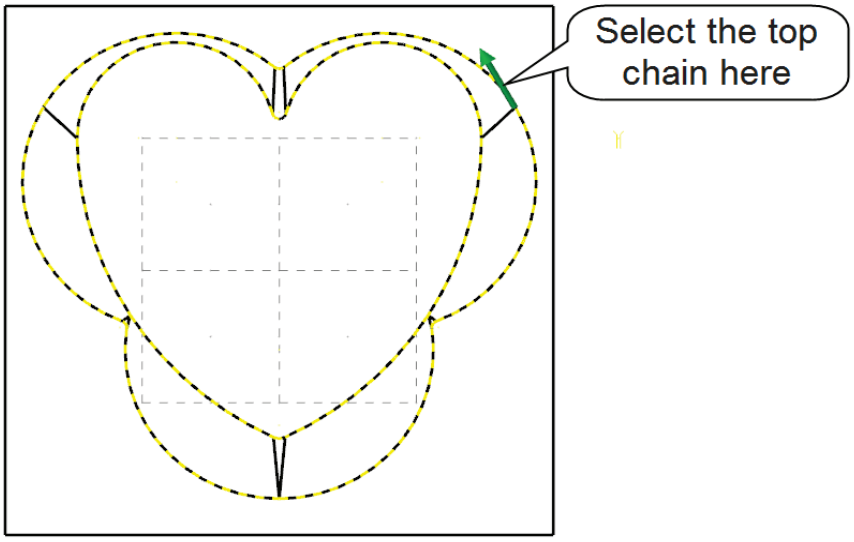
Figure: 16.0.1



NOTE: If you need to change the direction of the chaining, click on the **Reverse** button from the **Chaining** dialog box to flip the direction.

- [Ruled: define chain 3]: Select the top chain in the CCW direction as shown in Figure: 16.0.2.

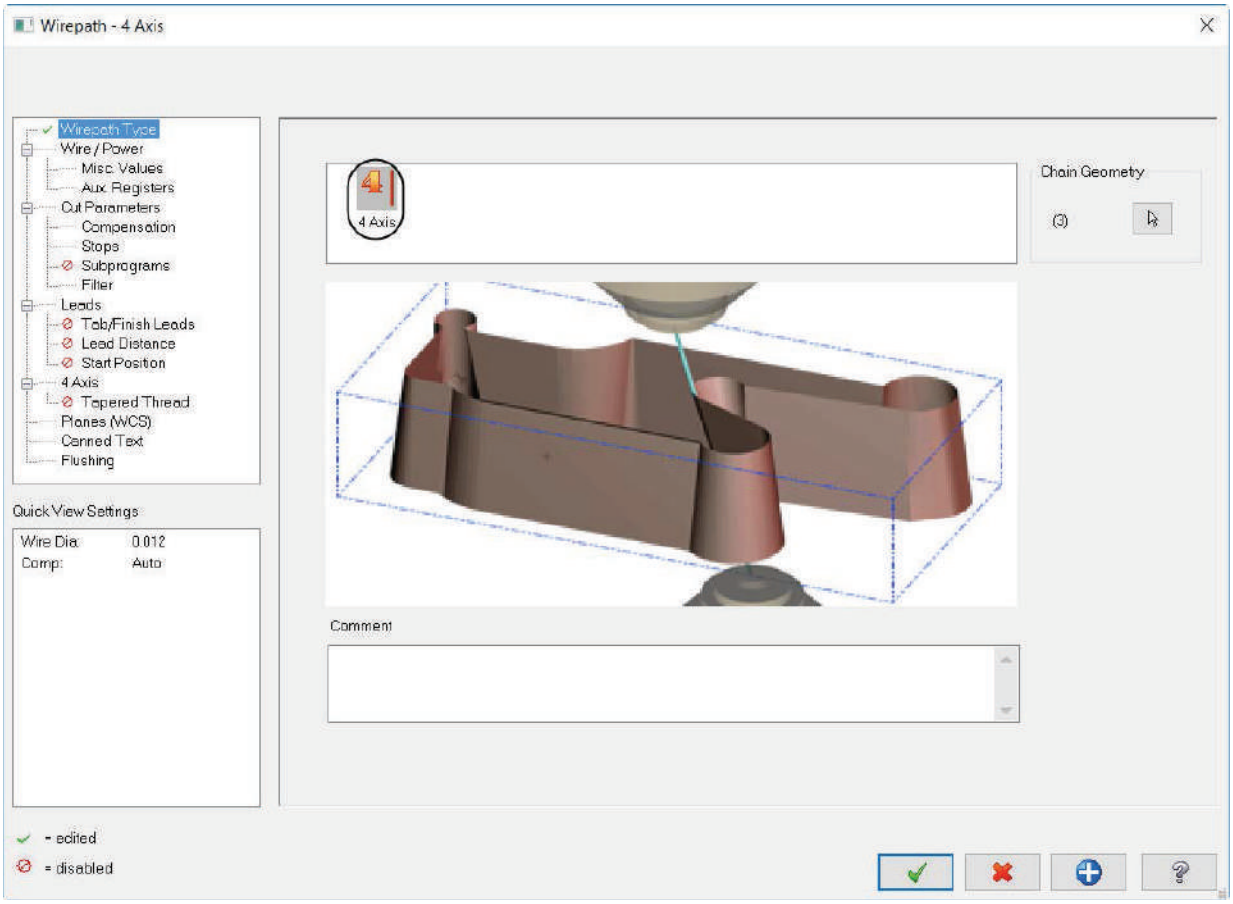
Figure: 16.0.2



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

16.1 Wirepath Type

- In the **Wirepath type** page, the **4-Axis** toolpath should be already selected.



16.2 Set the Wire/Power parameters

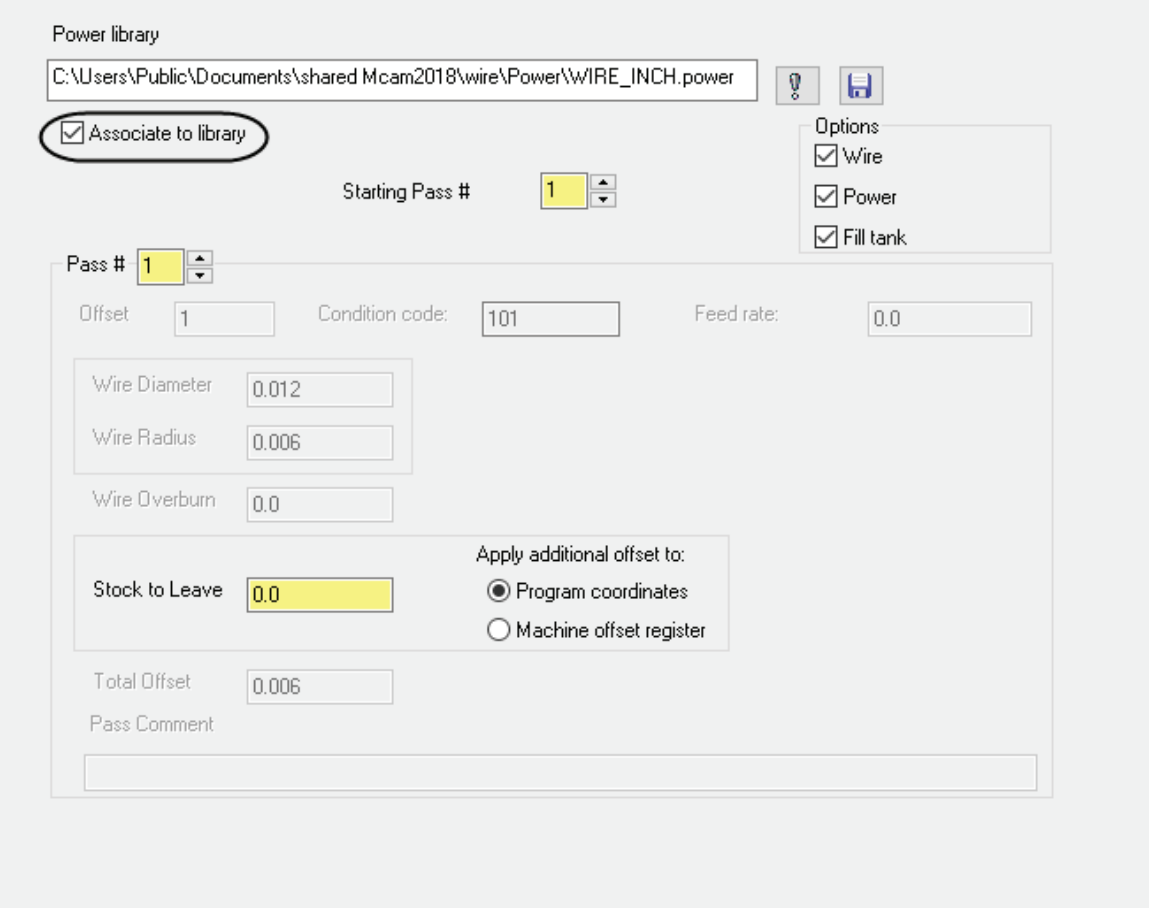
The **Wire/Power** page allows you to choose a wire power settings library file and use its values for the wirepath operation. You can also set the pass number to use for the first pass of the wirepath, and specify the settings for a single pass. To edit the power settings for each pass you have to disassociate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

- From the **Tree View** area, select **Wire/Power**.

NOTE: If you need to create your own wire power library, see Tutorial #1 (**page 46**). To open the library save in Tutorial #1, click on the **Open** icon and select the library from the Power library list. Otherwise, use the default WIRE_INCH.Power library as shown in this tutorial.

- Enable **Associate to library** to use the default **WIRE_INCH.power** library as shown in Figure: 16.2.1.
- If an **Edit Power Settings** dialog box appears, select the option **No.**

Figure: 16.2.1

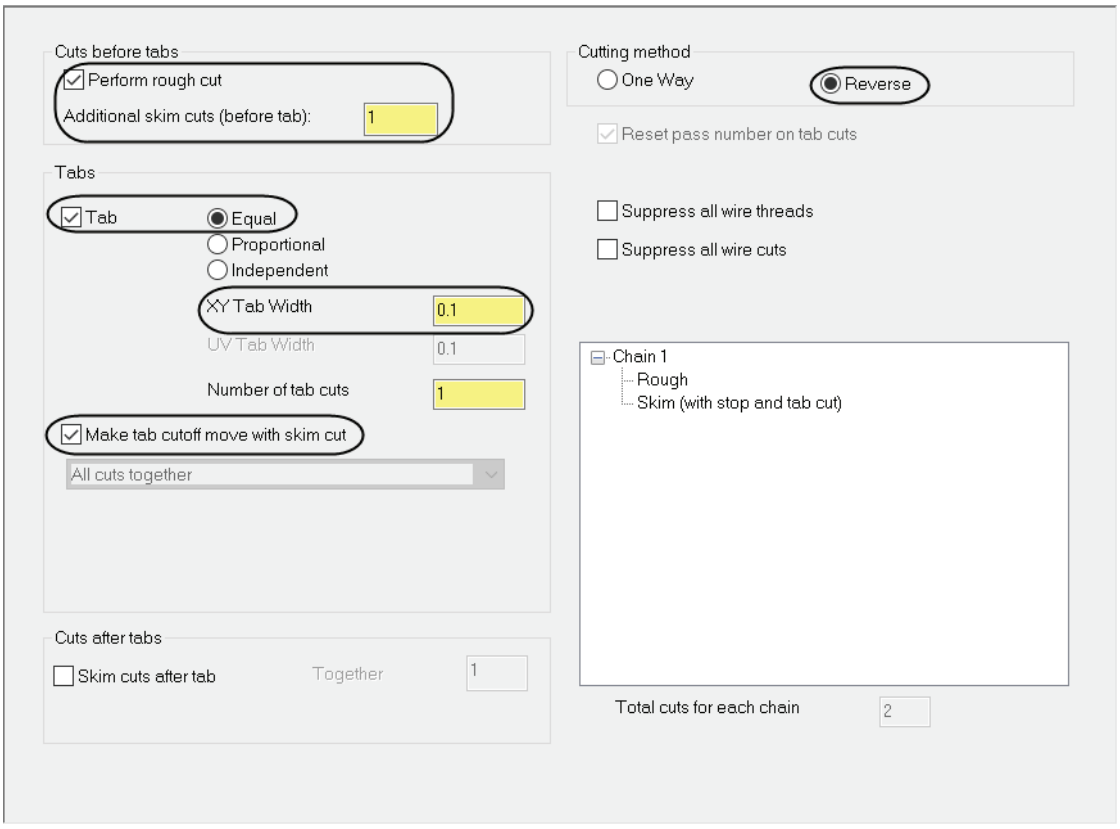


*16.3 Set the **Cut Parameters***

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 with stop and tab cut as shown in Figure: 16.3.1.

Figure: 16.3.1



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

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

Tab is the uncut material that secures the part to the material; can be cutoff with the skim cut pass or in a separate pass. Tab set to Equal sets the UV and XY tab widths equally.

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

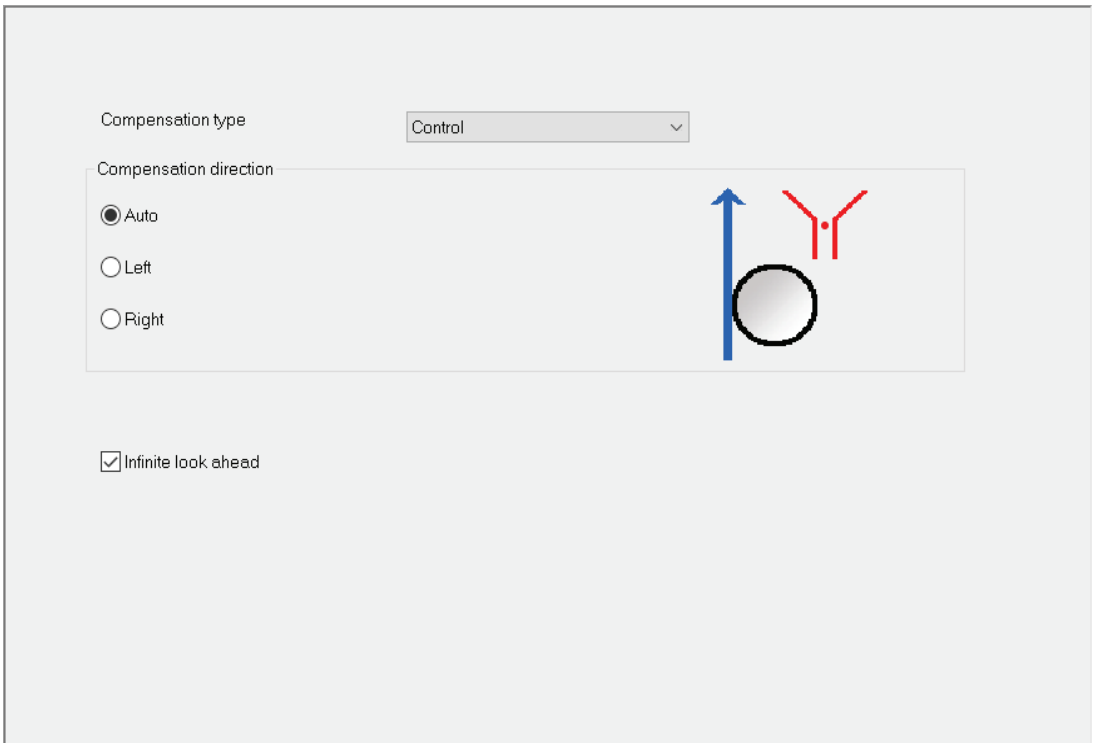
Make tab cutoff move with skim cut allows you to combine the tab cut with the last skim cut. If you have no skim cut before the tab, the tab cut combines with the rough cut.

16.4 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**.
- Leave the **Compensation direction** to **Auto** and leave the rest of the parameters as shown in Figure: 16.4.1.

Figure: 16.4.1



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

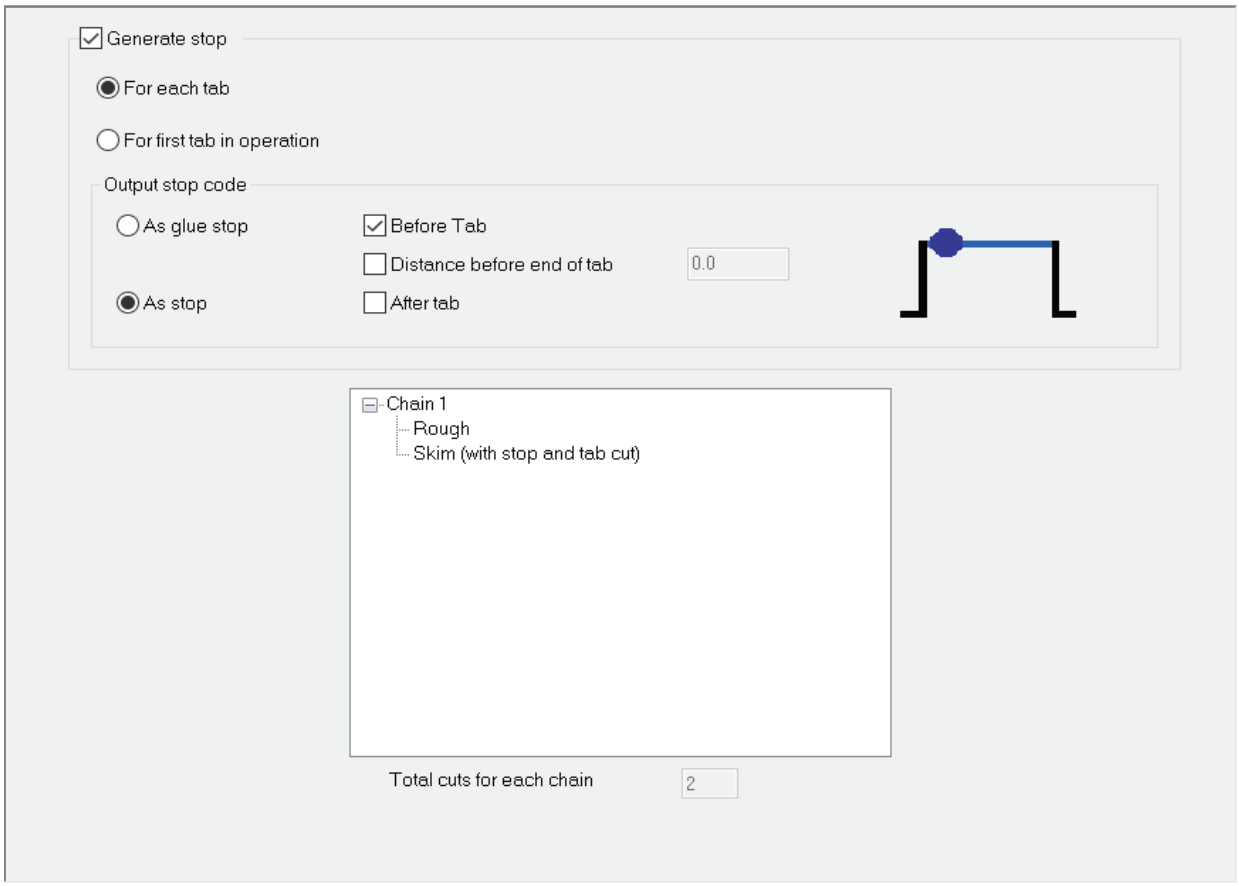
Compensation Direction allows you to offset the wire to the right or left of the wirepath. **Auto** sets the compensation direction depending on the location of the thread point inside or outside of the contour.

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.

16.5 Stops

Stops page allows you to create optional stops (M01) or program stops (M00) in a wirepath. For contour wirepaths, you have the options to apply the stops before and/or after tabs.

- From the **Tree View** area, select **Stops**.
- Make sure that the parameters are set to generate the stop (**M00**) before the **Tab**.



For each tab in operation outputs a stop code before the tab cut for all tab cuts on the selected chains.

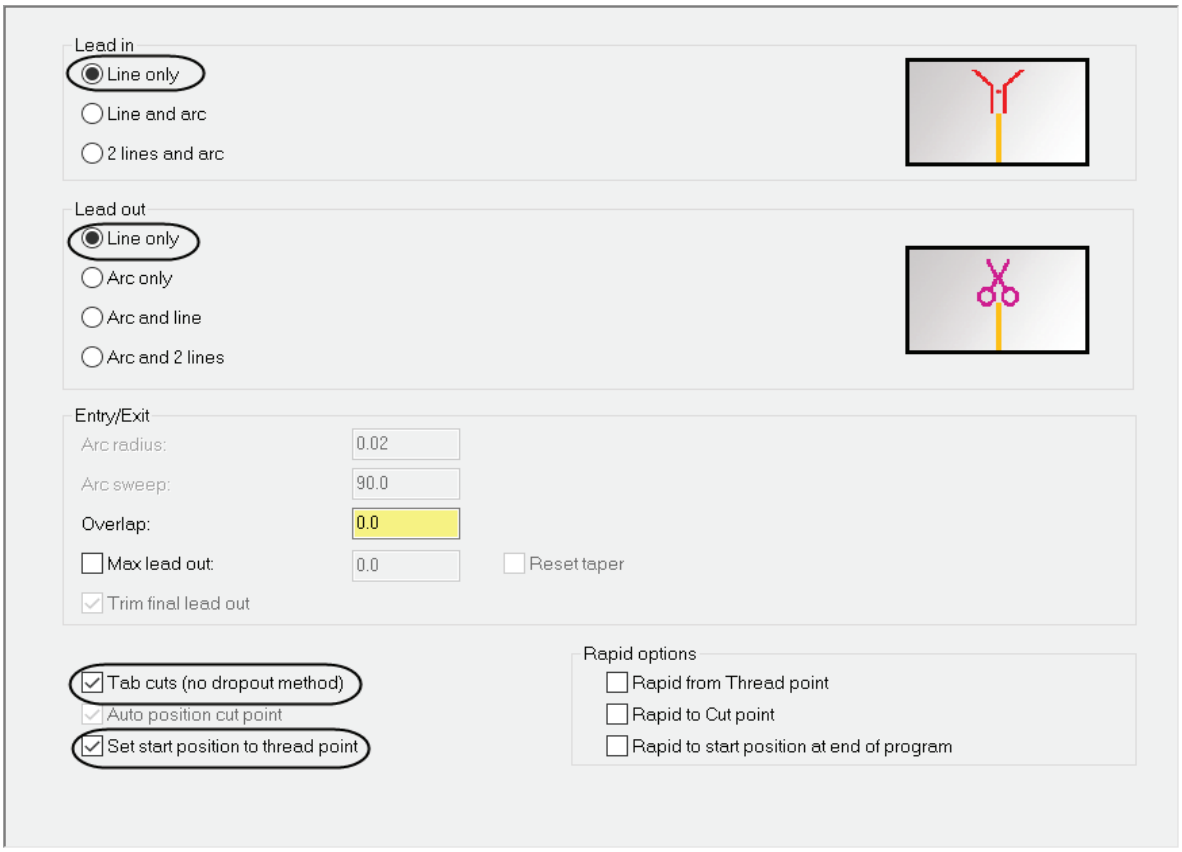
As stop outputs a program stop code (M00). Pauses the wire machine until the operator restarts it.

16.6 Leads

Leads page allows you to set the wire motion when entering or exiting a part.

- From the **Tree View** area, select **Leads**.
- Make sure that the **Lead in** and **Lead out** are set to **Line only,** and set the rest of the parameters as shown in Figure: 16.6.1.

Figure: 16.6.1



Lead in sets the shape of the part entry motion. The lead in extends from the thread point to the start of contour.

Line only creates a line between the thread point and start of contour.

Lead out sets the shape of the part exit motion. Lead out extends from the end of contour to the cut point.

Tab cuts (no dropout method) eliminates slugs or slivers of material that result from the tab cut. Enabling this check box disables the tab cut leads and finish cut leads.

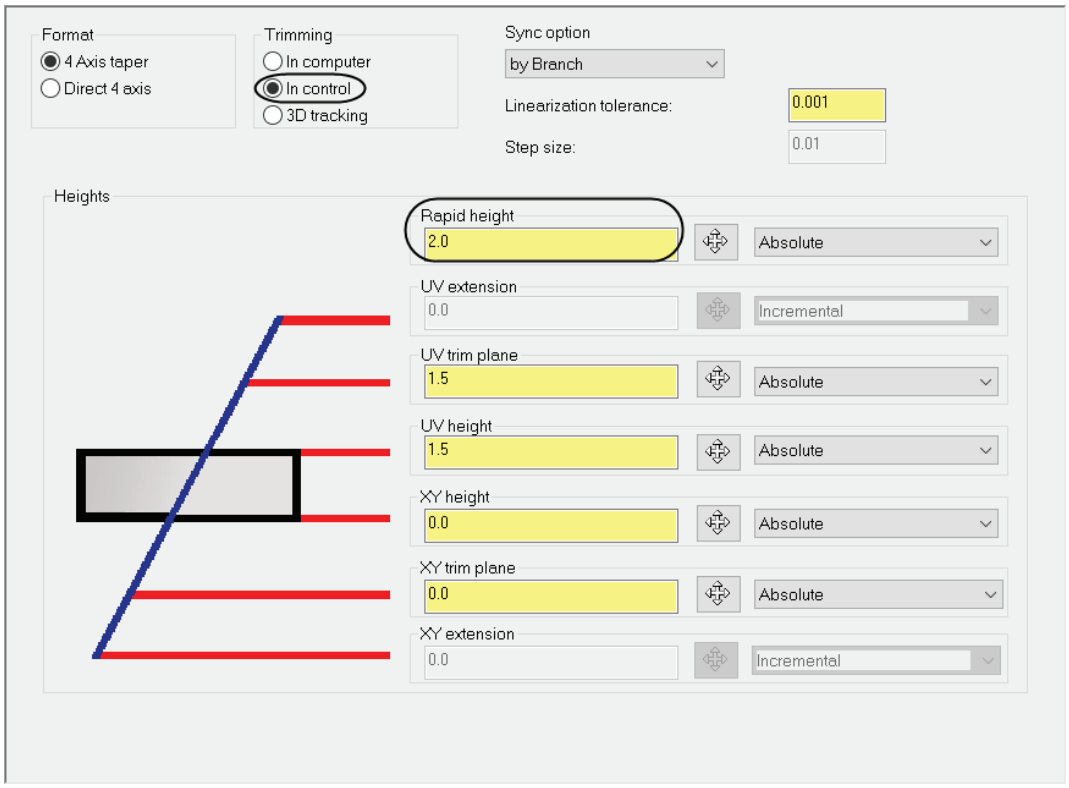
Set start position to thread point enabled moves the job start point to the thread point of the first contour in the operation.

16.7 4-Axis page

4-Axis page allows you to establish 4-Axis wirepath settings such as the linear arc move format (direct or taper), trimming options, and synchronization of the lower (XY) and upper (UV) contours.

- From the **Tree View** area, select **4-Axis** and make sure that the parameters are set as shown in Figure: 16.7.1.

Figure: 16.7.1



4-Axis taper outputs only UV/XY linear moves.

Rapid height sets the Z depth of a rapid move. Can be higher than the **UV height** or **UV trim planes** in order to clear any clamps.

UV trim plane can be used to set the location of the upper guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to G code 1015 in the NC file. It is usually set directly at the controller.

UV height sets the location of the XY plane that contains the geometry for the upper contour of a 4-Axis part.

XY height sets the location of the XY plane that contains the geometry for the lower contour of a 4-Axis part.

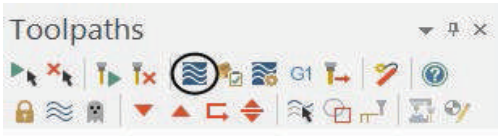
XY trim plane can be used to set the location of the lower guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to Gcode 1015 in the NC file. It can be set directly at the controller. When not using the XY trim plane, enter the same value as the XY height.

- Select the **OK** button to exit the **Wirepath - 4 Axis** parameters.

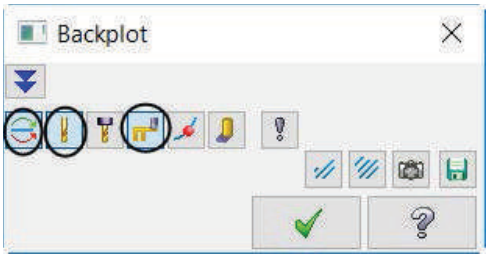
STEP 17: BACKPLOT THE WIRE CONTOUR WIREPATH

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 wirepaths, Mastercam displays the current X, Y, and Z coordinates in the lower left corner of the screen.

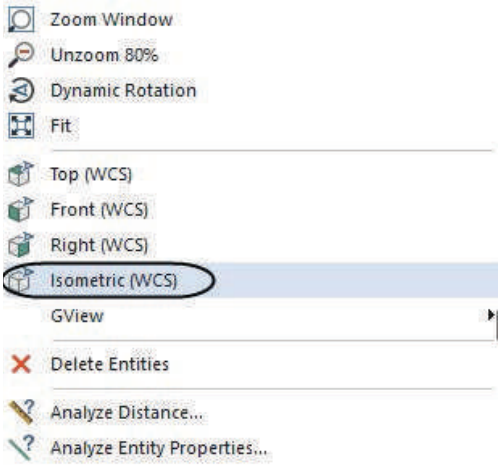
- Make sure that the toolpath is selected (signified by the green check mark on the folder icon).
- Select the **Backplot selected operations** button.



- Make sure that you have the following buttons turned on to see the tool and the rapid moves.



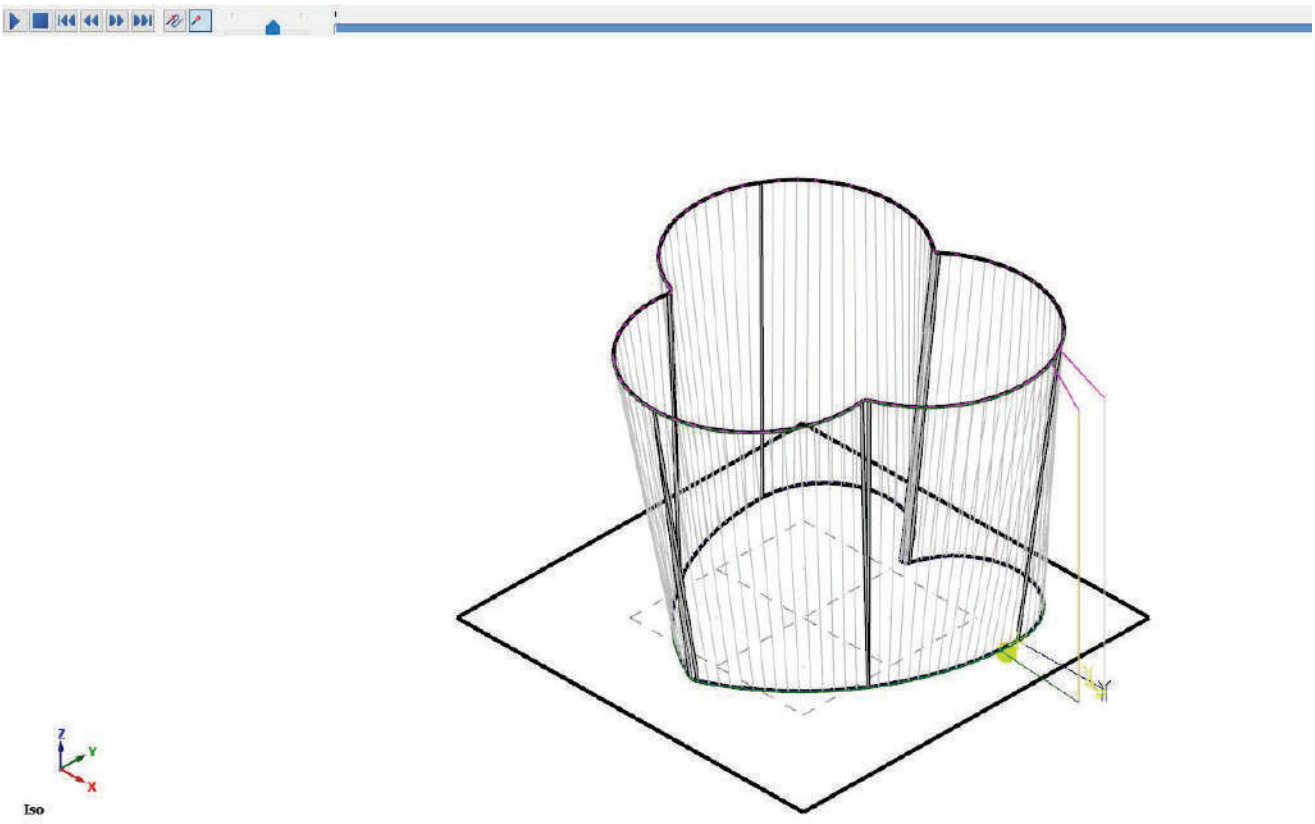
- Right mouse click in the graphics window and select **Isometric** as shown.



- Press **Alt+F1** to fit the drawing to the screen.
- Select the **Play** button to run **Backplot**.

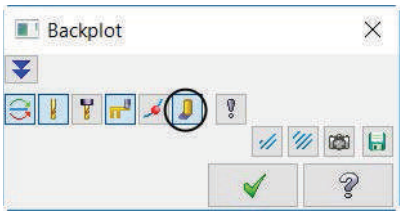
- The toolpath should look as shown Figure: 17.0.1.

Figure: 17.0.1

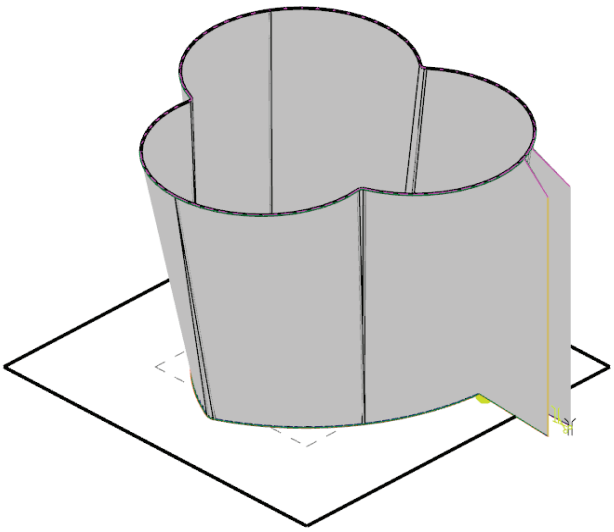


NOTE: The cuts are performed as follows:
The rough cut starts at the thread point and retracts before the tab. The skim cut is then performed in the reverse direction and the wire stops before the tab allowing the operator to hold the part. The tab is cut after that.

- In the **Backplot** dialog box enable **Quick Verify** as shown.



- The toolpath should look as shown.



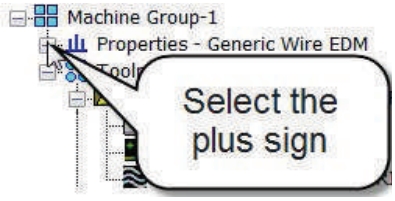
- Select the **OK** button to exit **Backplot**.

STEP 18: VERIFY THE WIREPATH

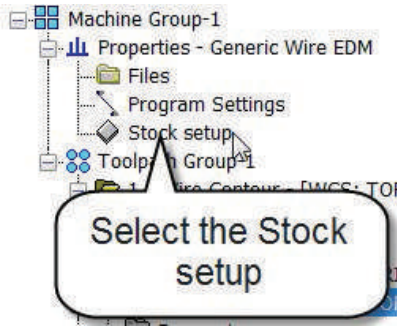
Simulates the machining of a part from a solid stock model display. The stock dimensions are based on the values that you can specified in the **Stock Setup**.

18.1 Set up the stock

- From the **Toolpaths Manager**, select the plus in front of the **Properties** to expand the **Toolpaths Group Properties**.

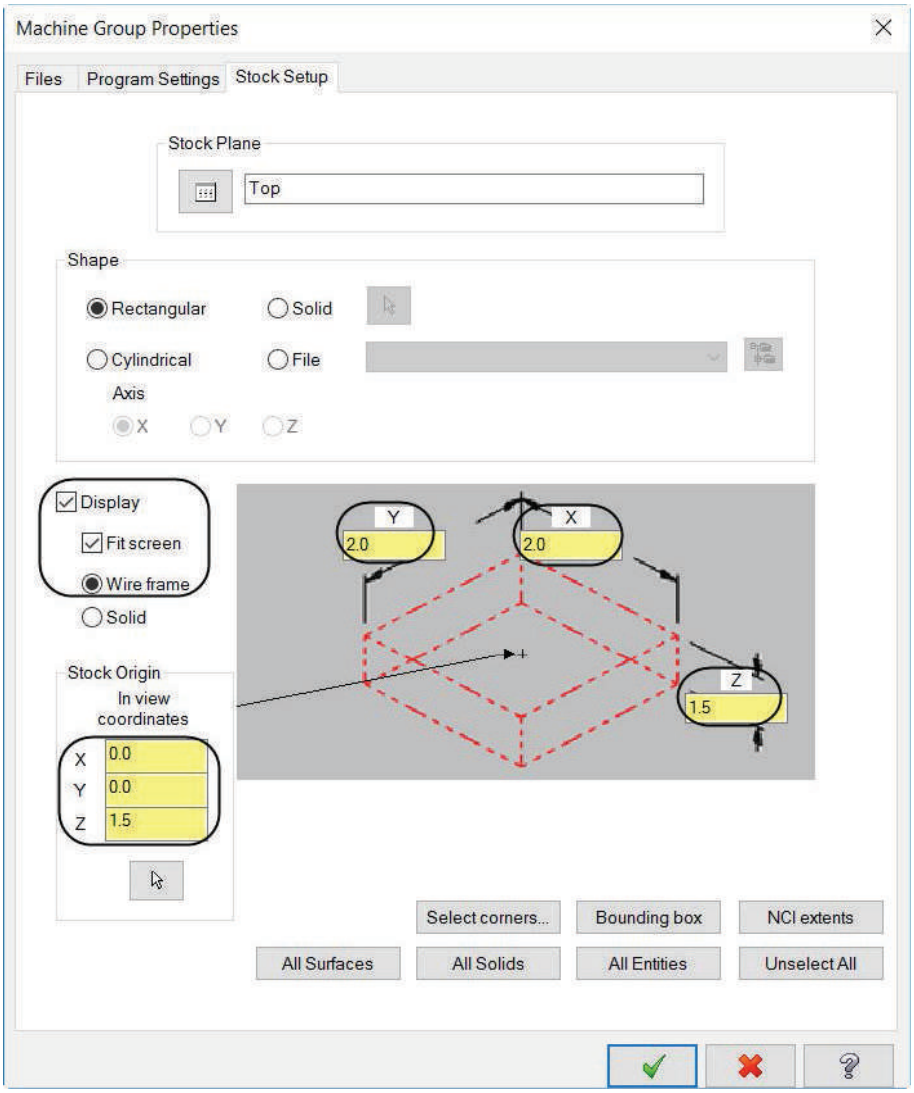


- Select the **Stock setup** to set the stock size.



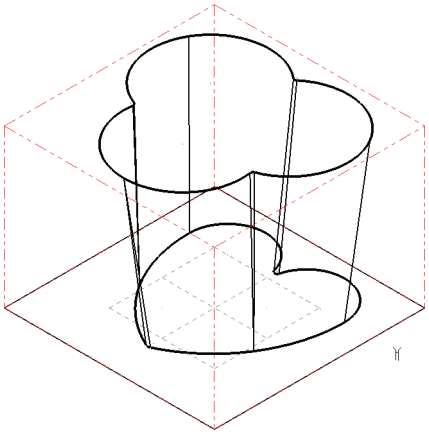
- Change the parameters to match the Figure: 18.1.1.

Figure: 18.1.1



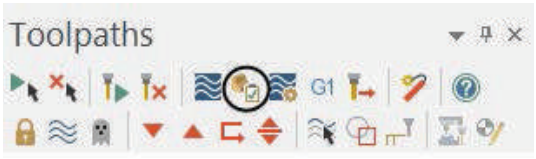
- Select the **OK** button to exit **Machine Group Properties**.

- If necessary, press **Alt+T** on your keyboard to disable the toolpath display.
- The stock is displayed with phantom red lines and it should look as shown.



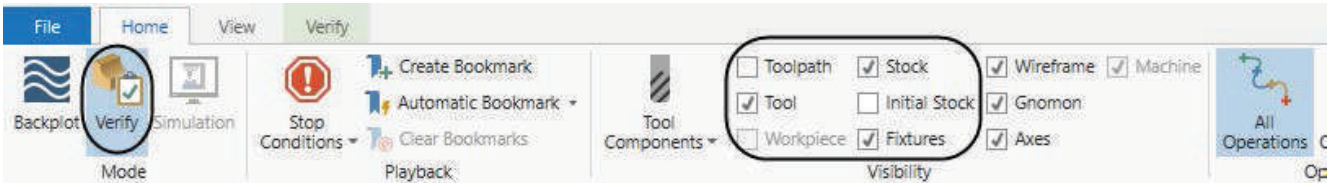
18.2 Verify the Wirepaths

- Select the **Verify selected operations** button.

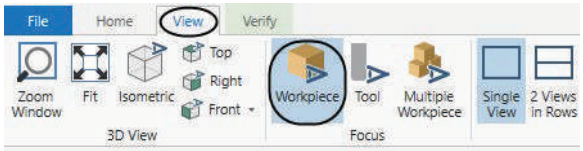


- From **Verify** change the settings for the **Visibility** as shown in Figure: 18.2.1.

Figure: 18.2.1



- Select **View** tab and enable **Workpiece** as shown.



- Select **Verify** tab and enable **Color Loop** as shown.

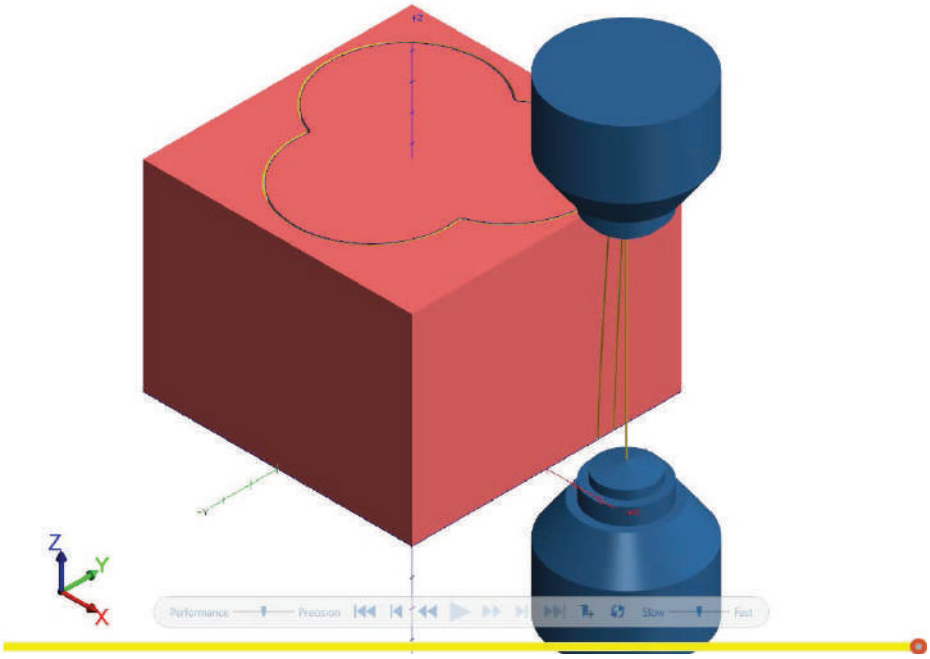
- Right click and select the **Isometric** option.

- Right click and select the **Fit** option.

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

- The part should look as shown in Figure: 18.2.2.

Figure: 18.2.2



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.

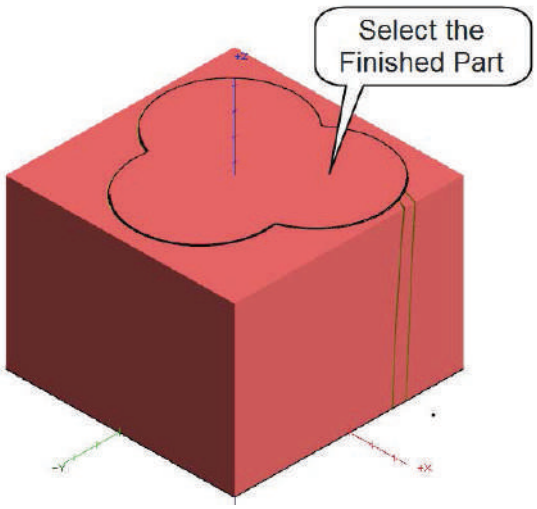
18.3 Chip removal

- Select the **Verify Tab** and select the **Keep Chips** icon as shown in Figure: 18.3.1.

Figure: 18.3.1

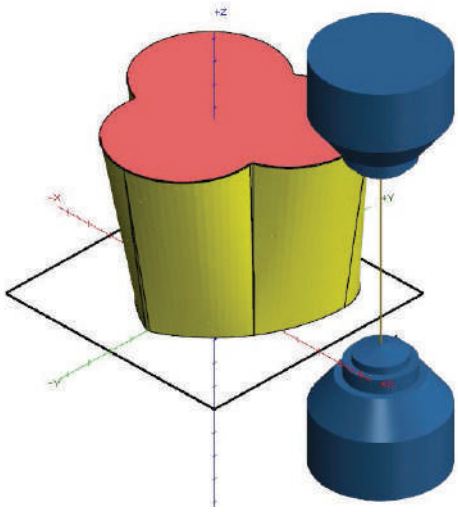
- Click on the part as shown in Figure: 18.3.2.

Figure: 18.3.2

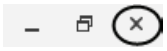


- The part will appear as shown in Figure: 18.3.3.

Figure: 18.3.3



- To exit **Mastercam Simulator** click on the **Close** icon.



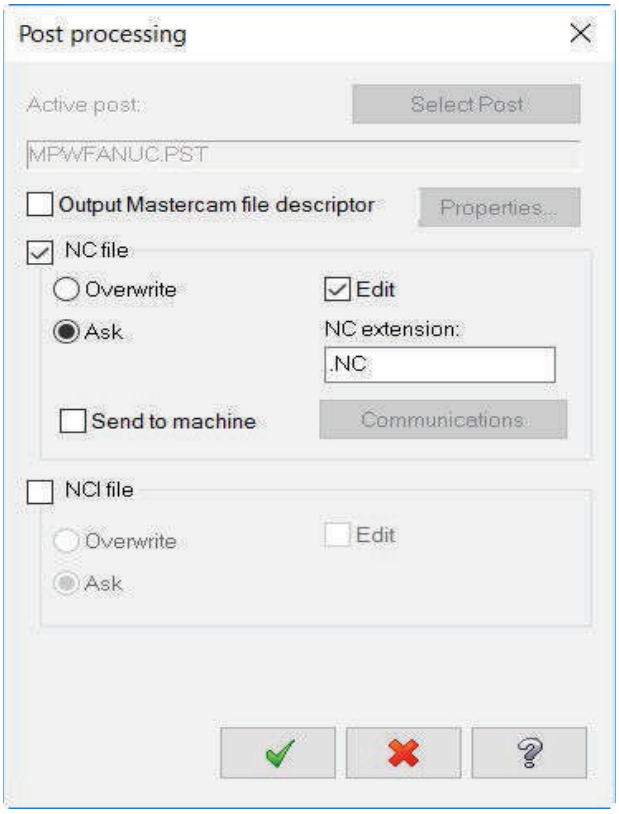
STEP 19: POST THE FILE TO GENERATE THE NC CODE

- Select **Post selected operations** button from **Toolpaths Manager**.

NOTE: The active **Post Processor** is a generic FANUC post processor.

- In the **Post processing** window, if needed, make any necessary changes as shown in Figure: 19.0.1.

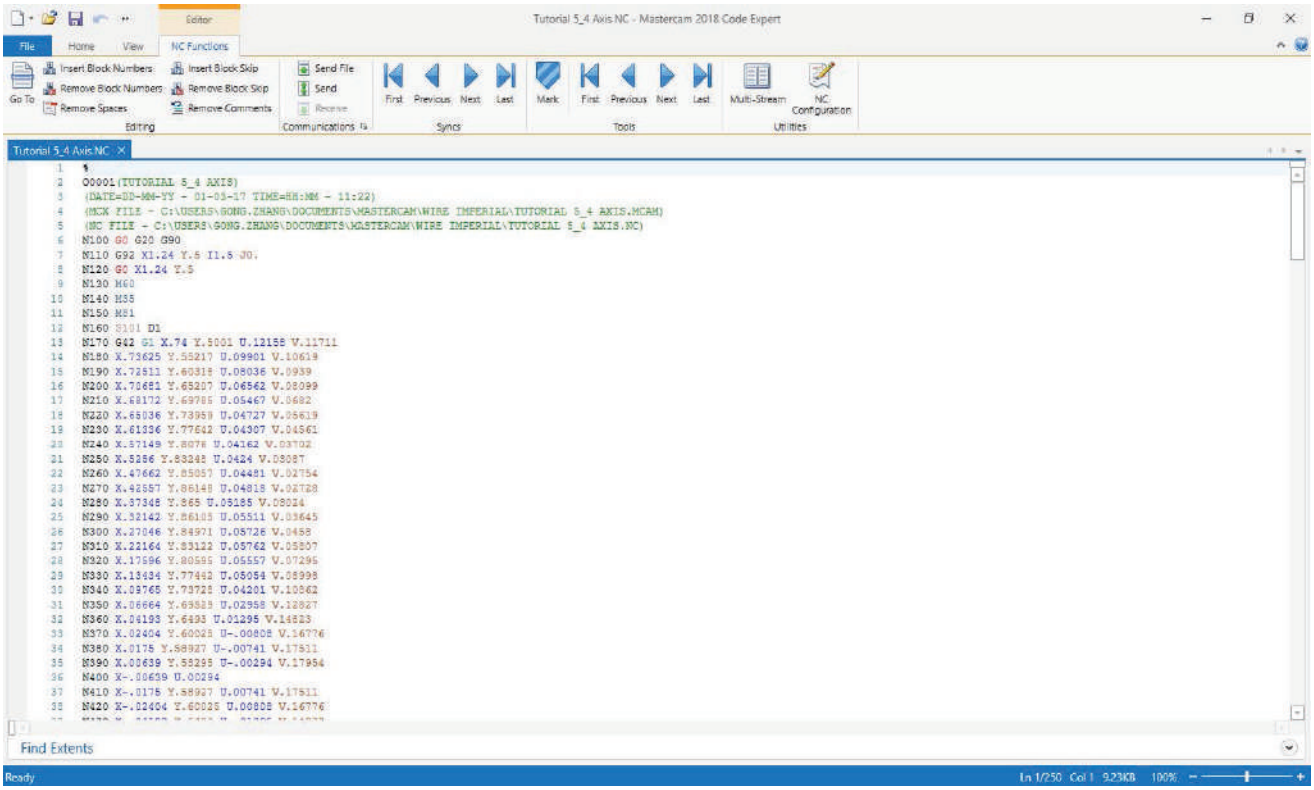
Figure: 19.0.1



- Select the **OK** button to continue.
- Save the NC file.

- The NC file will appear on the screen as shown in Figure: 19.0.2.

Figure: 19.0.2

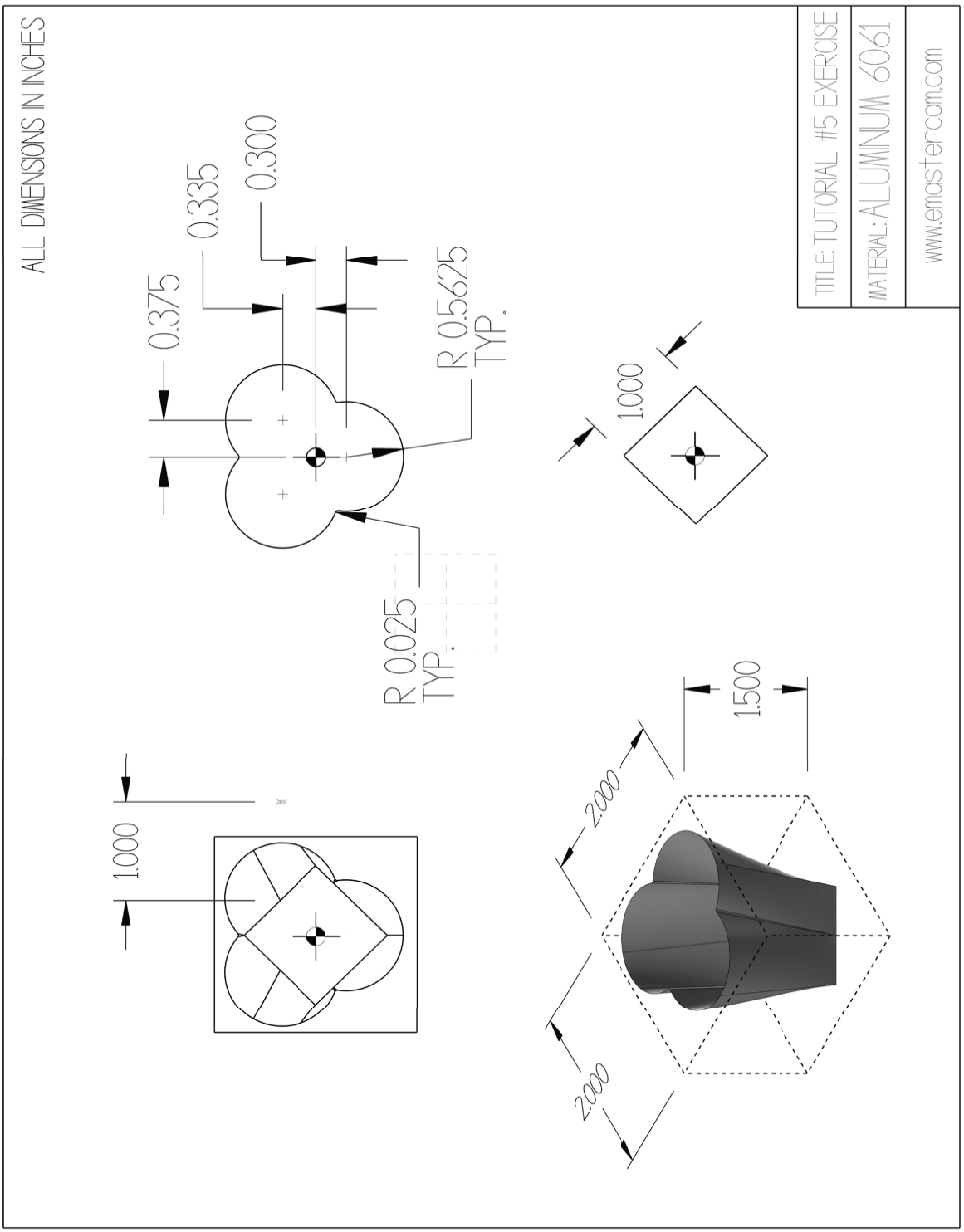


NOTE: The G-code that you have created will appear on the screen. If the code looks okay, you can shut down the window without saving it. If you need to change the code, make sure you save it before you close the window.
How the program is sent to the machine depends on the shop setup.

- Select the "**X**" box at the upper right corner to exit the **Mastercam Code Expert.**

STEP 20: SAVE THE UPDATED MCAM FILE

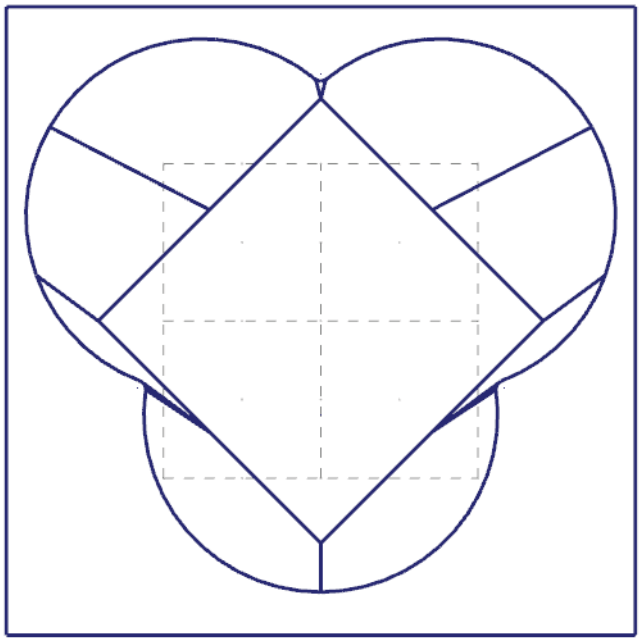
REVIEW EXERCISE -STUDENT PRACTICE



CREATE THE GEOMETRY FOR TUTORIAL #5 EXERCISE

Use these commands to create the geometry.

- Rectangle.
- Circle Center Point.
- Fillet.
- Rectangular Shapes.
- Break Two Pieces.
- Line Endpoints.
- Point Position - Thread Point.

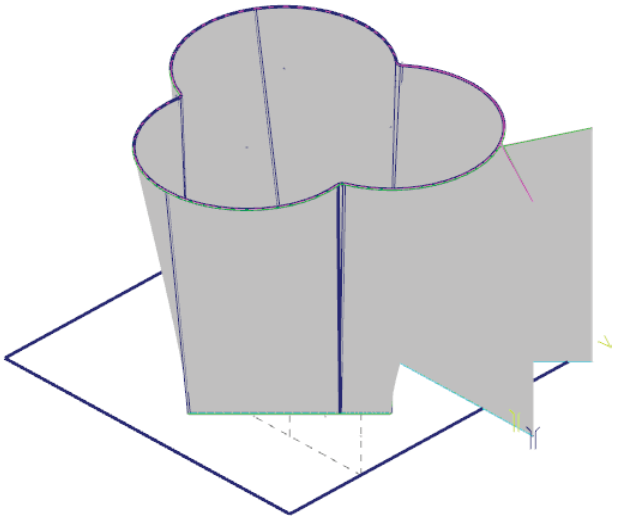


CREATE THE TOOLPATHS FOR TUTORIAL #5 EXERCISE

Create the Toolpaths for Tutorial #5 Exercise as per the instructions below.

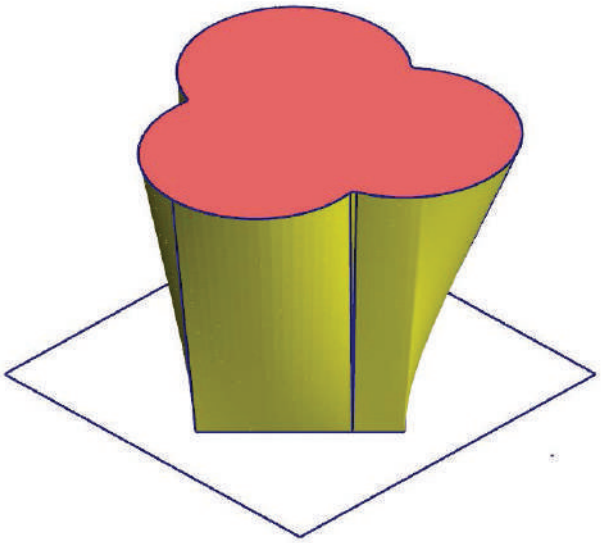
Machine The Part using 4 - Axis Wirepath.

- In order to complete this operation follow **page 334** to **page 345.**



Verify The Wirepath.

- Set the **Stock** according to the part.
- In order to complete this operation **Verify** the part and follow **page 348** to **page 352**.



NOTES:

TUTORIAL #5 QUIZ

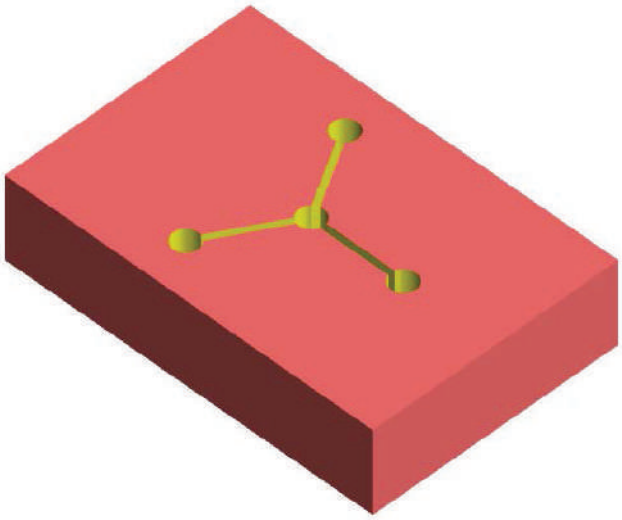
- How is the geometry created in 2D construction mode?

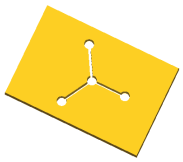
- How do you create the branches used to synchronize By Branch the top and the bottom profiles?

- When should you use a 4 Axis Wirepath?

- What does 4 Axis page allow you to set?

TUTORIAL #6





OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

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, Circle Center Point, Line Polar, Line Parallel, Trim, and Bolt Circle will be used.

Create the necessary Wirepaths to machine the part:

- Wire No Core wirepath to cut the inner profile.

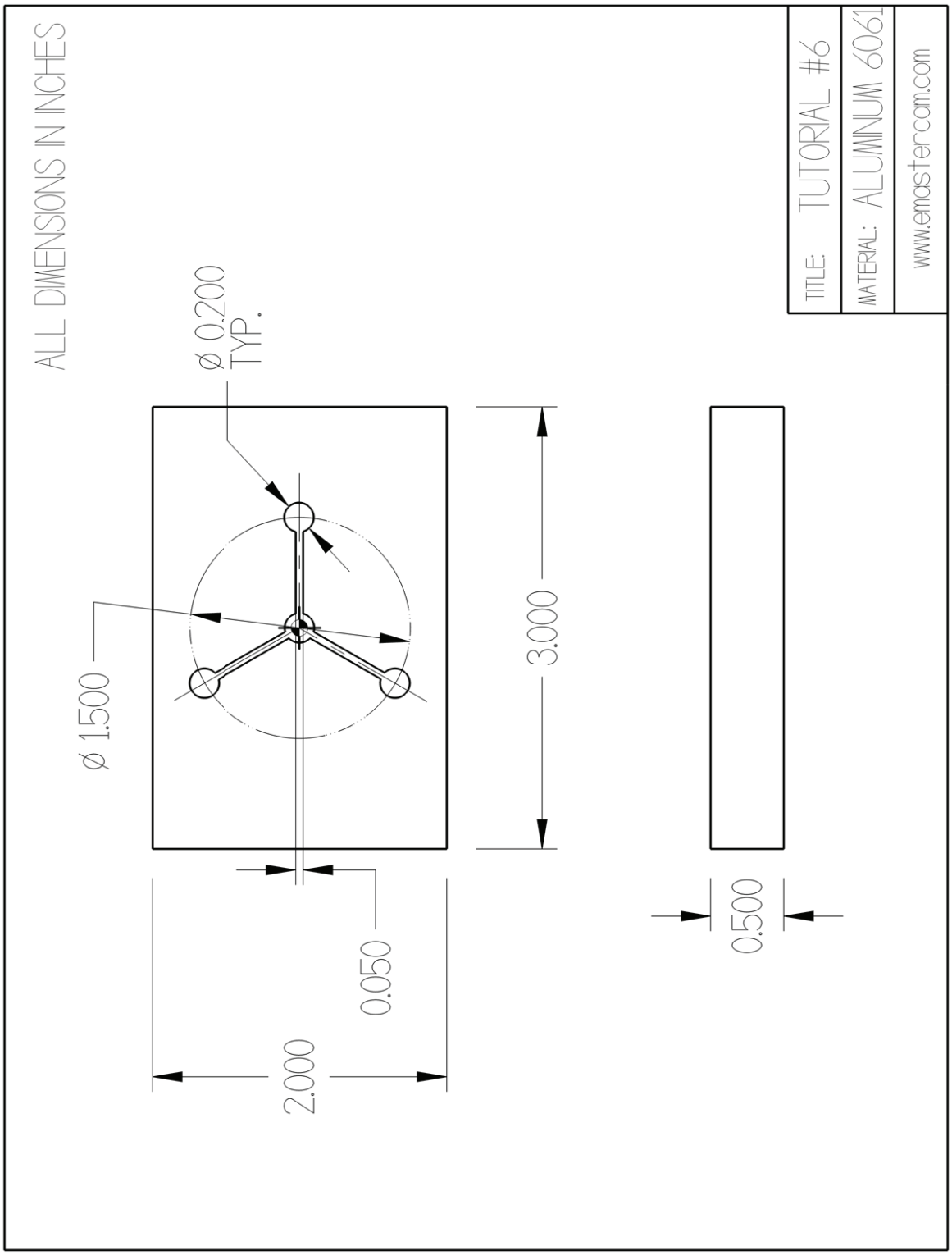
Backplot and the file:

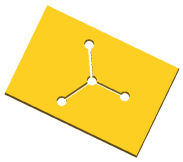
- The Backplot will be used to simulate a step by step process of the wire's movements.
- The Verify will be used to watch the wire machine the part out of a solid model.

Post Process the file to generate the G-code:

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

This tutorial takes approximately 45 minutes to complete.


ALL DIMENSIONS IN INCHES
TITLE: TUTORIAL #6
MATERIAL: ALUMINUM 6061
www.emastercam.com
ø 0.200
TYP.
ø 1.500
3.000
0.050
2.000
0.500



GEOMETRY CREATION

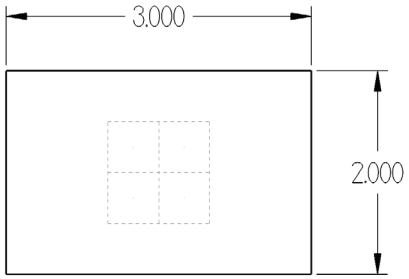
STEP 1: SETTING UP THE GRAPHIC USER INTERFACE

Please refer to the **Getting Started** section to set up the graphical user interface.

STEP 2: CREATE THE 3.0" BY 2.0" RECTANGLE

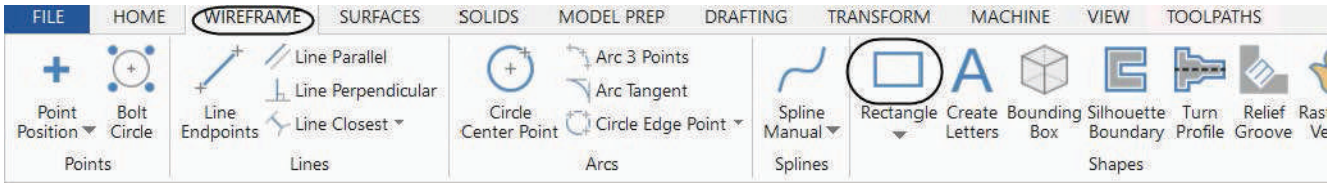
In this step you will learn how to create a rectangle given the width, the height, and the anchor location.

Step Preview:

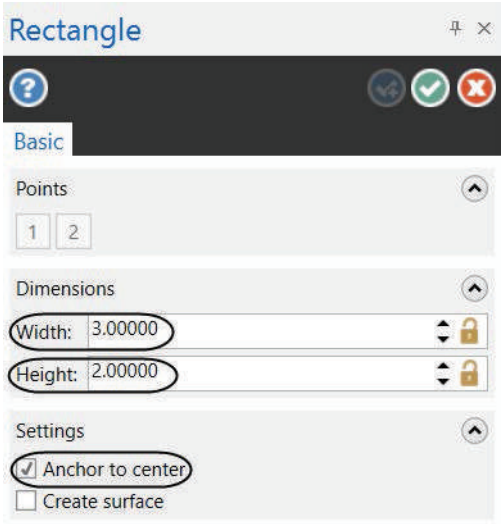


WIREFRAME

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

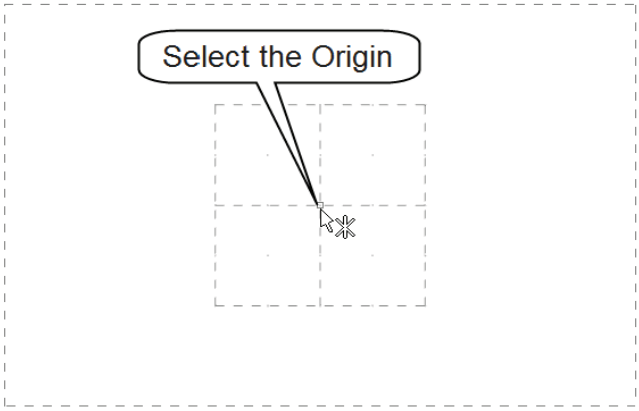


- Enter the **Width** and the **Height**, and enable **Anchor to center** as shown.



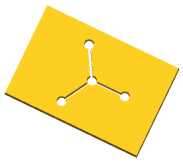
- Press **Enter** after typing the values to see a preview of the rectangle.
- [Select position of base point]: Select the **Origin** as shown in Figure: 2.0.1.

Figure: 2.0.1



- Make sure that when selecting the **Origin**, the visual cue of the cursor changes as shown.

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



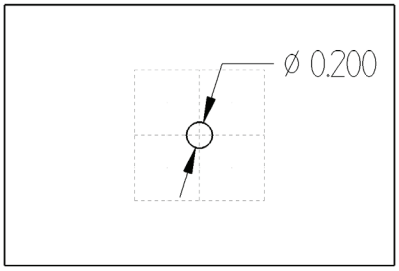
- Press **Alt+F1** to fit the drawing to the screen.

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.

STEP 3: CREATE THE 0.2" DIAMETER CIRCLE

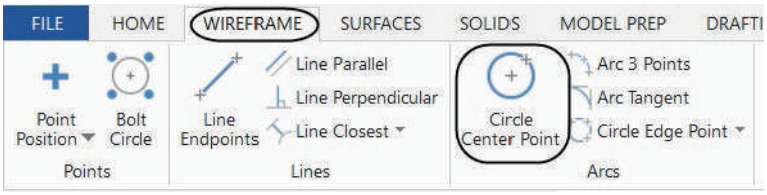
In this step you will create one circle given the diameter and the center locations.

Step Preview:

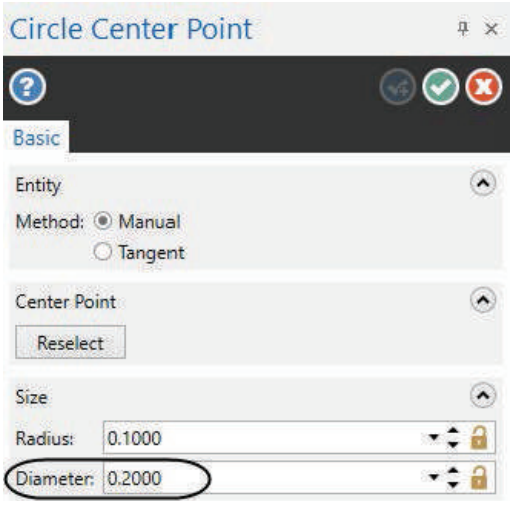


WIREFRAME

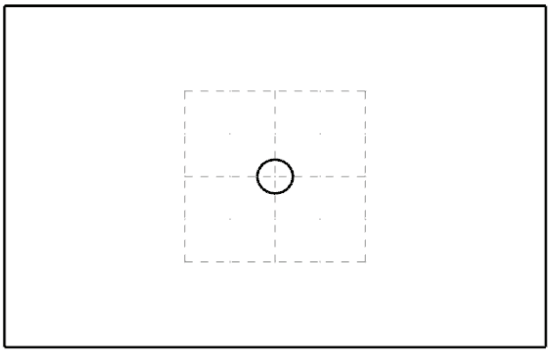
- From the **Arcs** group, select the **Circle Center Point** icon.



- Enter the **Diameter 0.2**



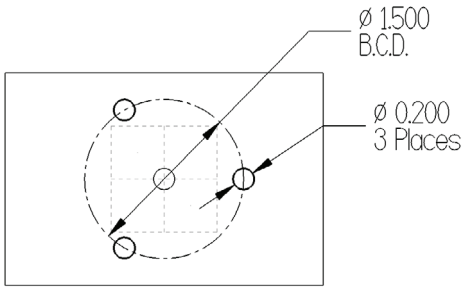
- [Enter the center point]: Select the **Origin** as shown in Figure: 2.0.1.
- Select the **OK** button to exit the command.
- Press **Alt+F1** to fit the drawing to the screen.
- The geometry should look as shown.



STEP 4: CREATE THE THREE 0.2" DIAMETER CIRCLES ON A BOLT CIRCLE

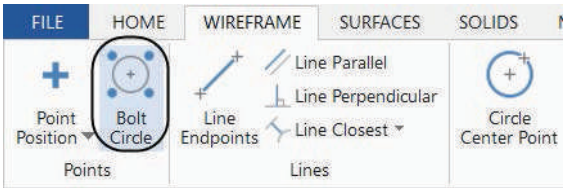
In this step you will create the three circles on a bolt circle given the base point and the radius of the bolt circle, as well as the number and the diameter of the three circles.

Step Preview:



WIREFRAME

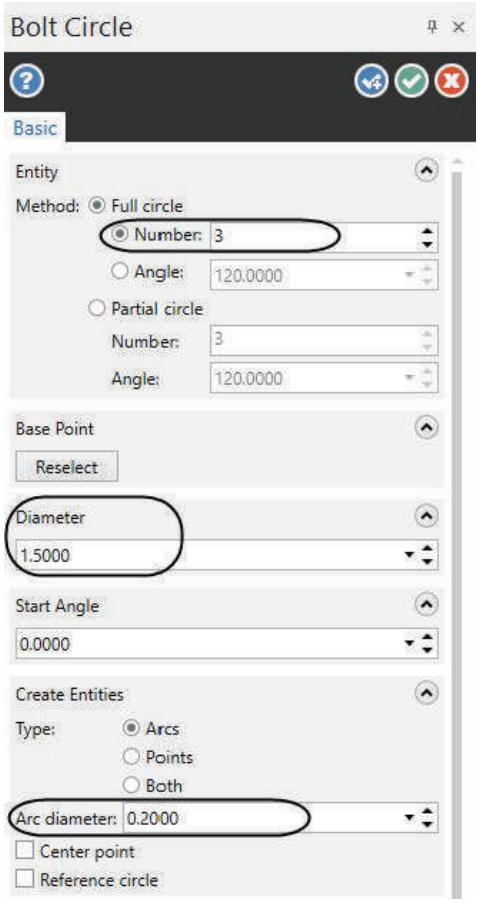
- From the **Points** group, select the **Bolt Circle** icon.



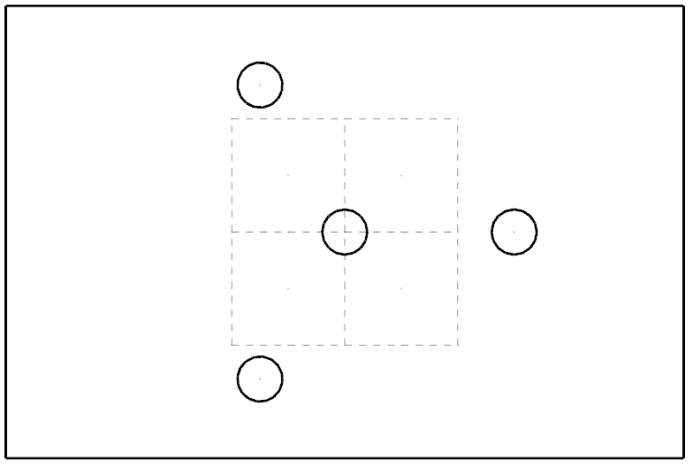
- [Select position of base point]: Select the **Origin** as shown before.

- Change the parameters in the **Bolt Circle** panel as shown in Figure: 4.0.1.

Figure: 4.0.1



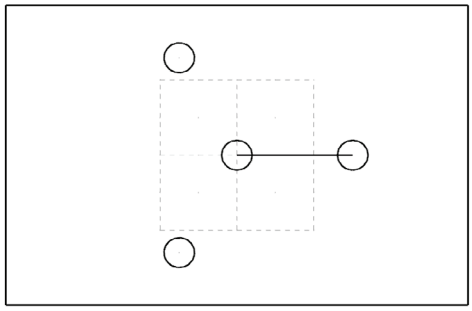
- Select the **OK** button to exit **Bolt Circle** panel.
- The geometry should look as shown.



STEP 5: CREATE A LINE GIVEN THE ENDPOINT

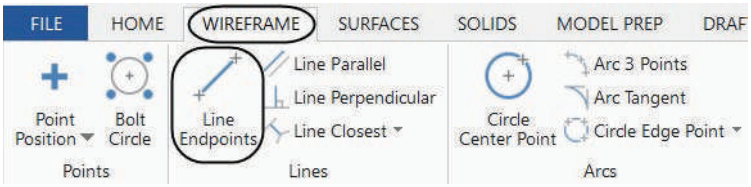
Using Create Line Endpoint command, you will create a line given the endpoints.

Step Preview:



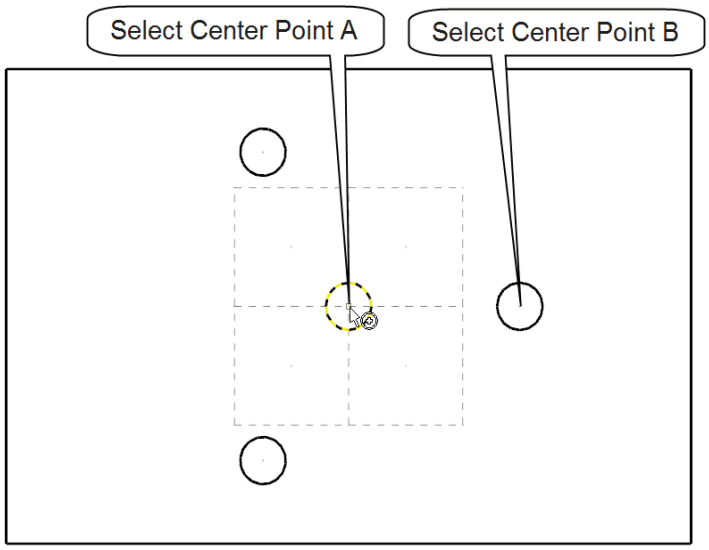
WIREFRAME

- From the **Lines** group, select the **Line Endpoints** icon.



- [Specify the first endpoint]: Select Center Point A as shown in Figure: 5.0.1.

Figure: 5.0.1

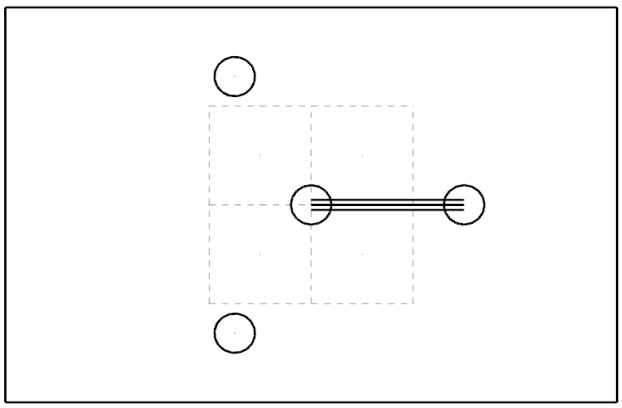


- [Specify the second endpoint]: Select the Center Point B as shown in Figure: 5.0.1.
- Select the **OK** button to exit the command.

STEP 6: CREATE PARALLEL LINES

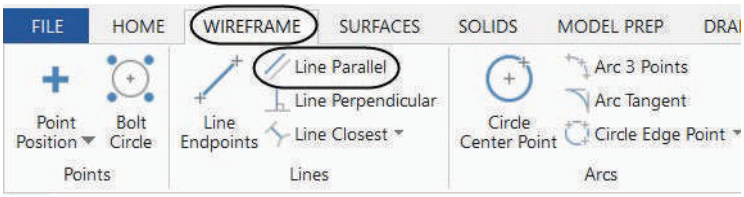
In this step you will create lines parallel to an existing line at a given distance.

Step Preview:



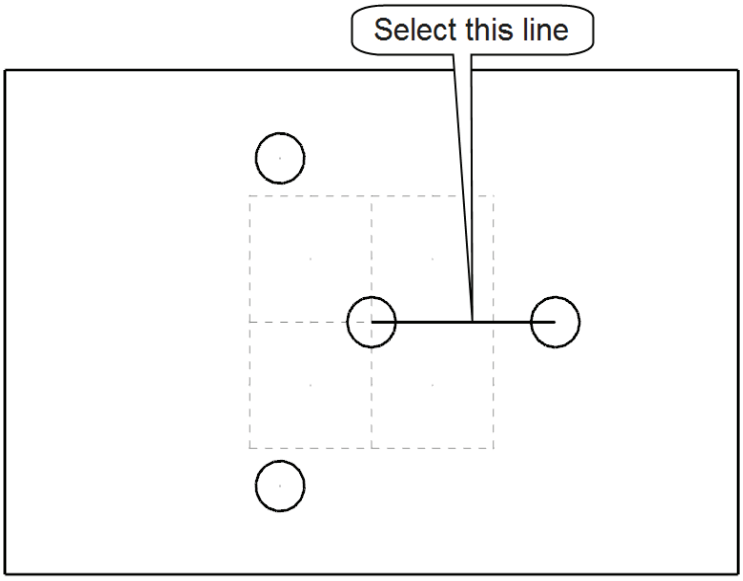
WIREFRAME

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

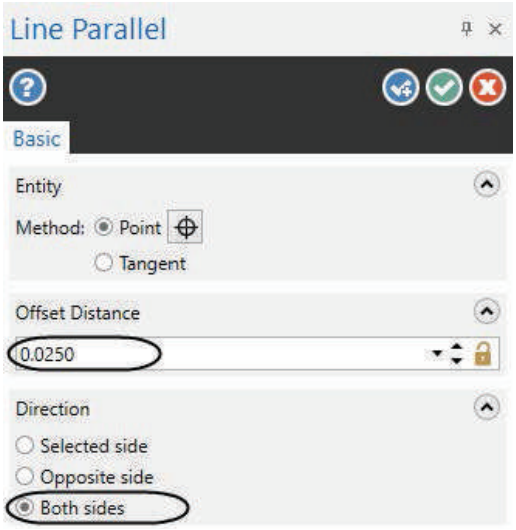


- [Select a line]: Select the horizontal line as shown in Figure: 6.0.1.

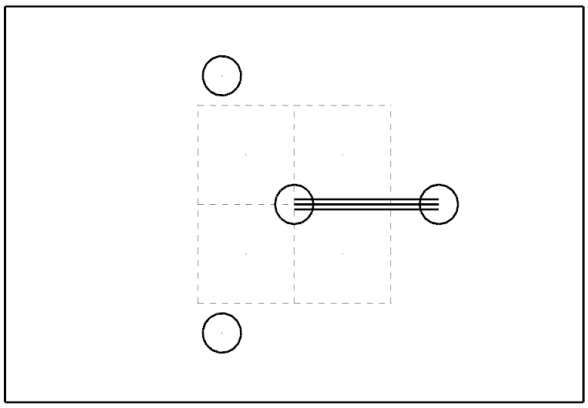
Figure: 6.0.1



- [Select the point to place a parallel line through]: Click somewhere above the selected line.
- Enter the **Offset Distance 0.025**.

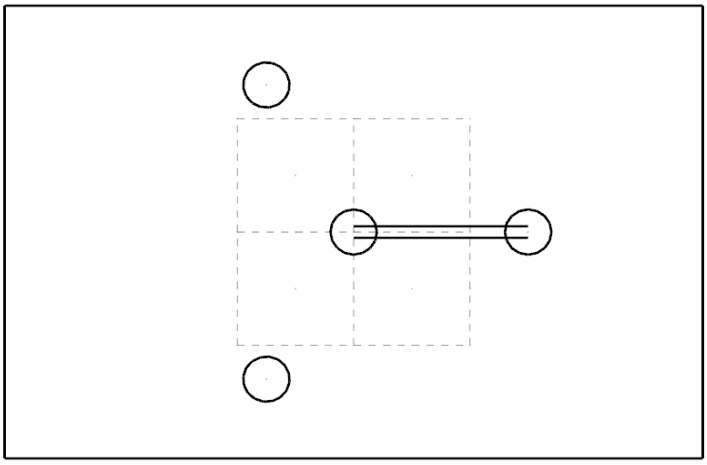


- Select the **OK** button to exit the same command.
- The geometry should look as shown.



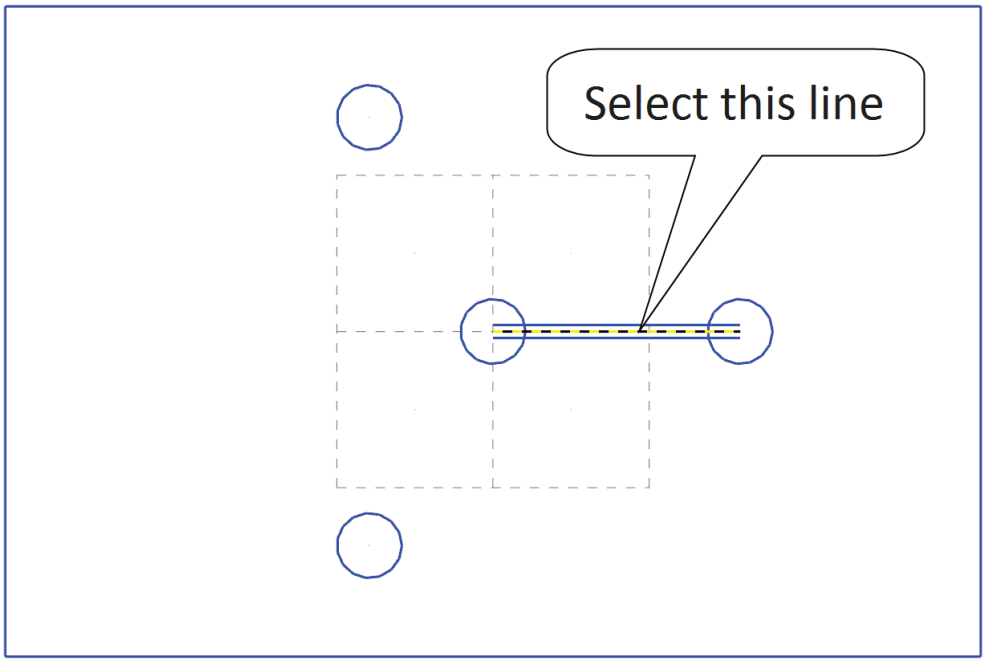
STEP 7: DELETE THE CONSTRUCTION LINE

Step Preview:



- Select the line as shown in Figure: 7.0.1.

Figure: 7.0.1

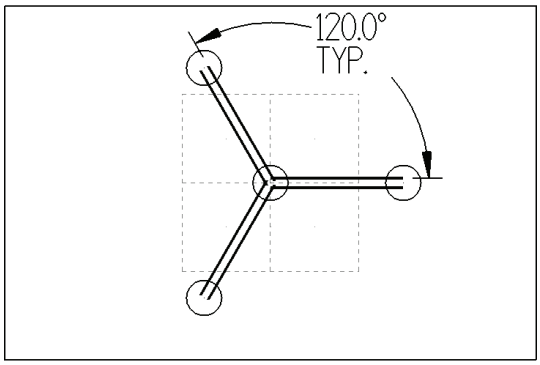


- Press **Delete** to delete the selected line.

STEP 8: ROTATE/COPY

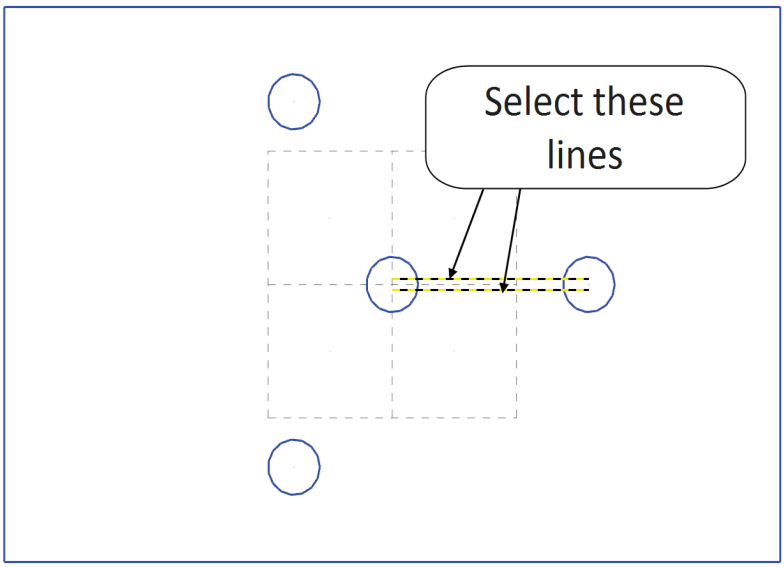
In this step you will learn how to create the rest of the channels using **Transform Rotate** command with the copy option enabled. The parallel lines will be copied 2 times and rotated around the origin.

Step Preview:



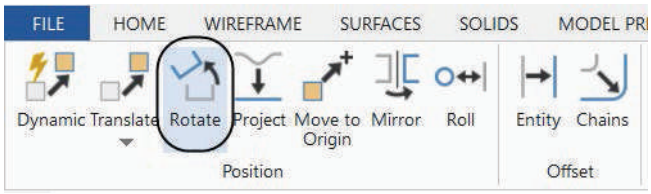
- Select the two parallel lines as shown in Figure: 8.0.1.

Figure: 8.0.1



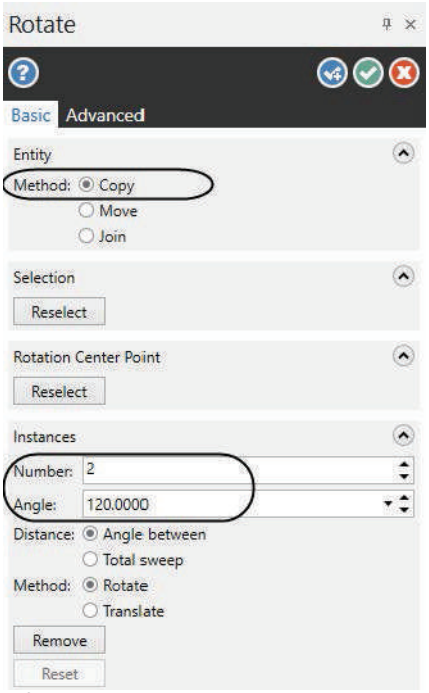
TRANSFORM

- From the **Position** group, select **Rotate** as shown.

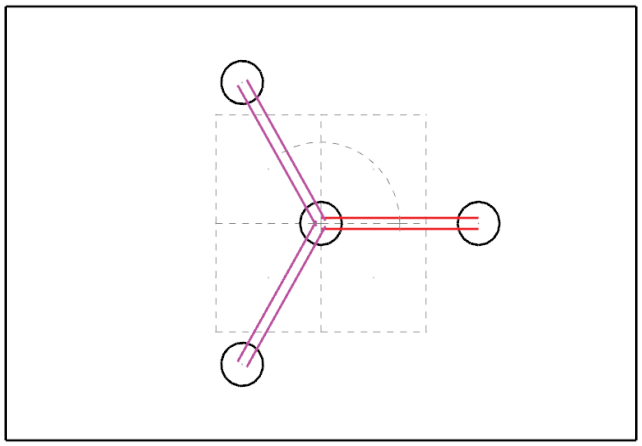


- Make sure that **Copy** is enabled in the **Rotate** panel and set the number of copies (**#**) to **2** and the **Angle** to **120.0** as shown in Figure: 8.0.2.

Figure: 8.0.2



- Press **Enter** and the preview of the generated geometry should look as shown.



- Select the **OK** button to exit the **Rotate** panel.

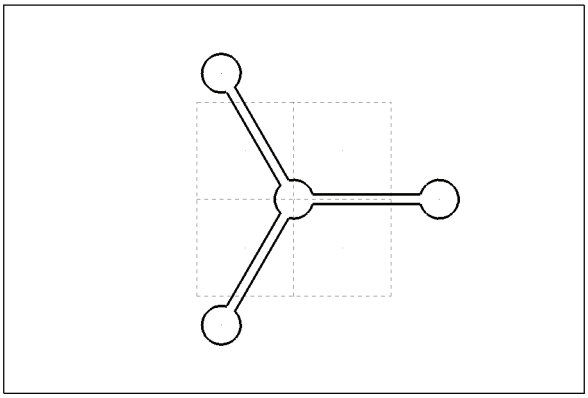
NOTE: When performing a transform function, Mastercam creates a temporary group from the original, highlighted in red, and a result from the transformed entities, highlighted in magenta. These two colors will remain until you use the **Screen**, **Clear Colors** function or perform another transform function.

- Right mouse click in the graphics window and select **Clear Colors** as shown.

STEP 9: CLEAN THE GEOMETRY USING TRIM DIVIDE

Divide command divides or deletes entities based on the nearest intersection.

Step Preview:



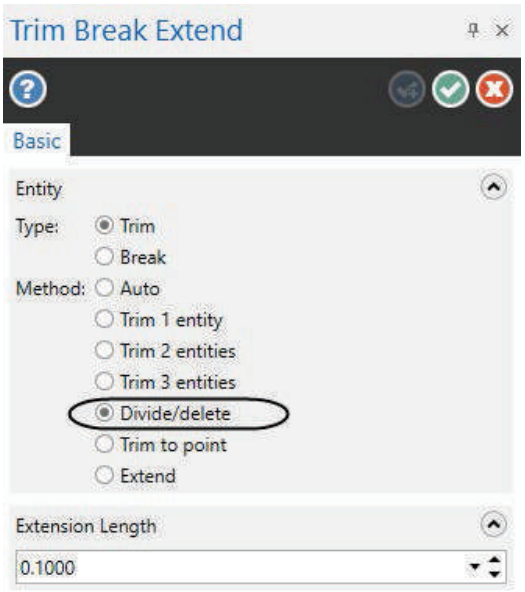
NOTE: During the trimming process it is very important to select the entities exactly at the locations that are shown in the graphics. Mastercam's auto-preview feature displays the results of the selected function as you move the mouse over the final entity selection. A solid line represents what will be created. A dashed line represents what will be removed.

- Hover the cursor above the **Origin** (center of the part) and scroll up the mouse wheel to zoom in as shown above.

WIREFRAME

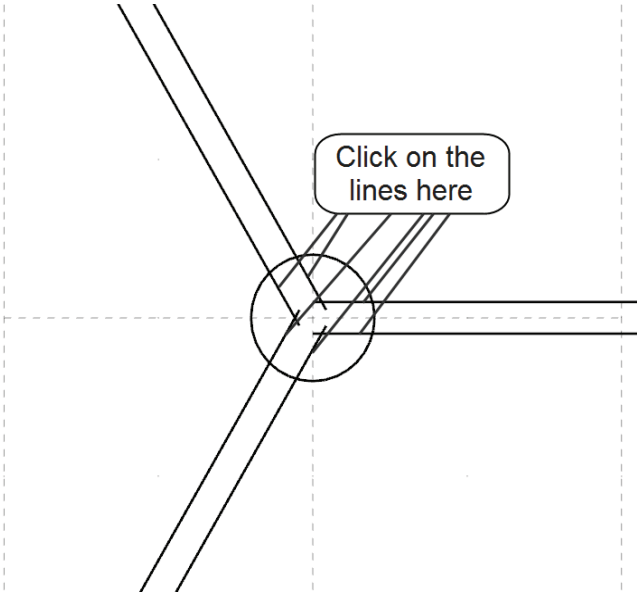
- From the **Modify** group, select **Trim Break Extend**.

- Enable **Divide/delete** as shown.

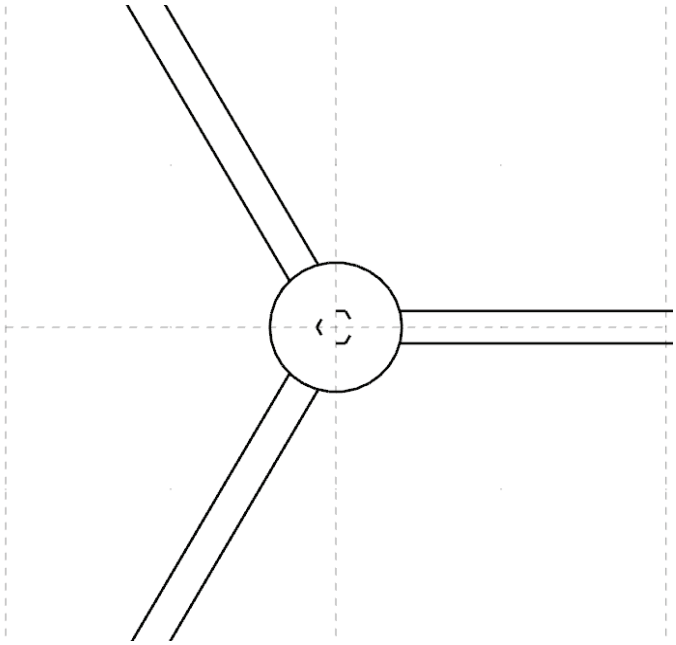


- [Select the curve to divide/delete]: Click on the lines inside of the circle to divide them as shown in Figure: 9.0.1.

Figure: 9.0.1

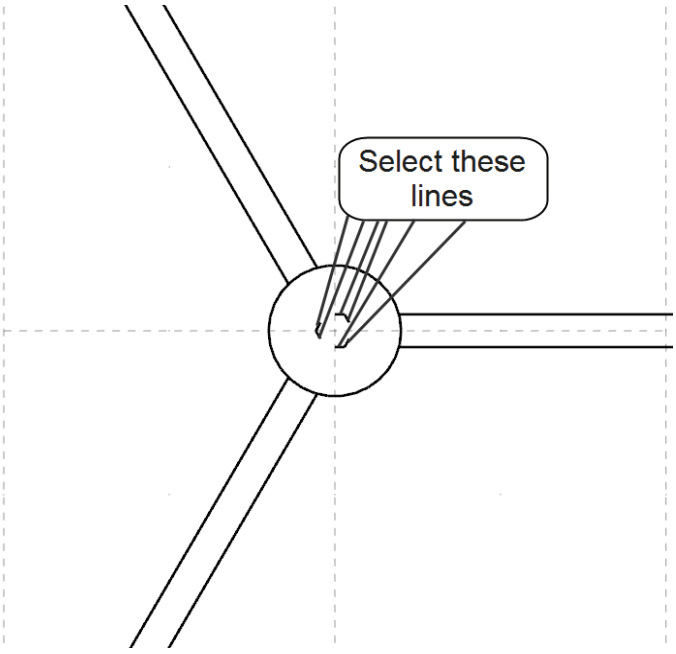


- The geometry should look as shown.



- Click on the remaining lines inside of the circle to delete them as shown in Figure: 9.0.2.

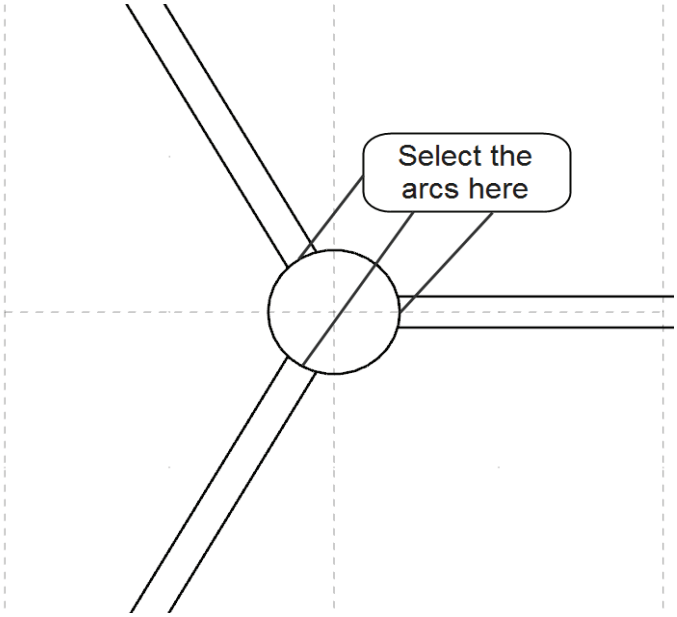
Figure: 9.0.2



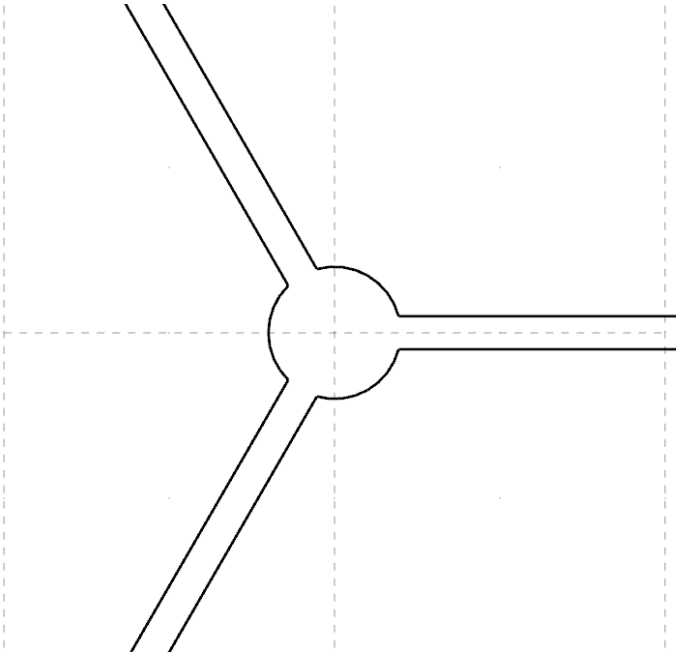
NOTE: Divide can be used for both trimming or deleting.

- Divide the circle by selecting the parts of the arcs that need to be removed as shown in Figure: 9.0.3.

Figure: 9.0.3



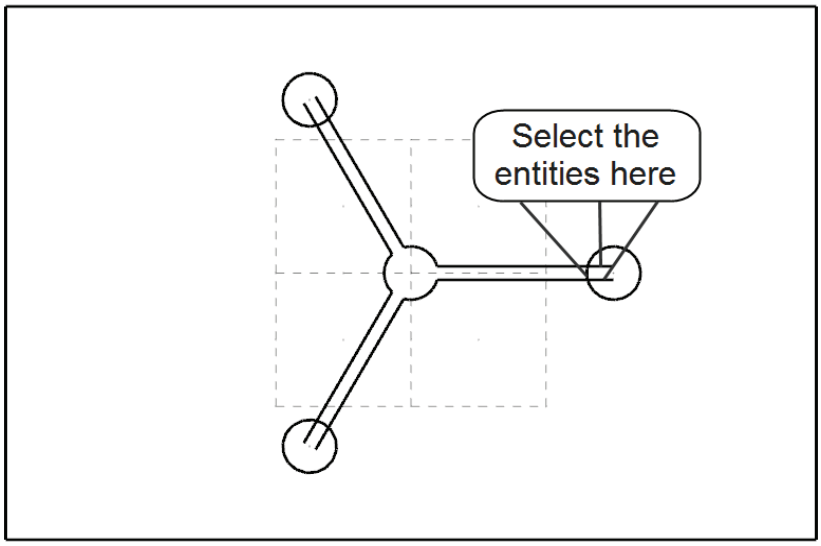
- The center of the shape should look as shown.



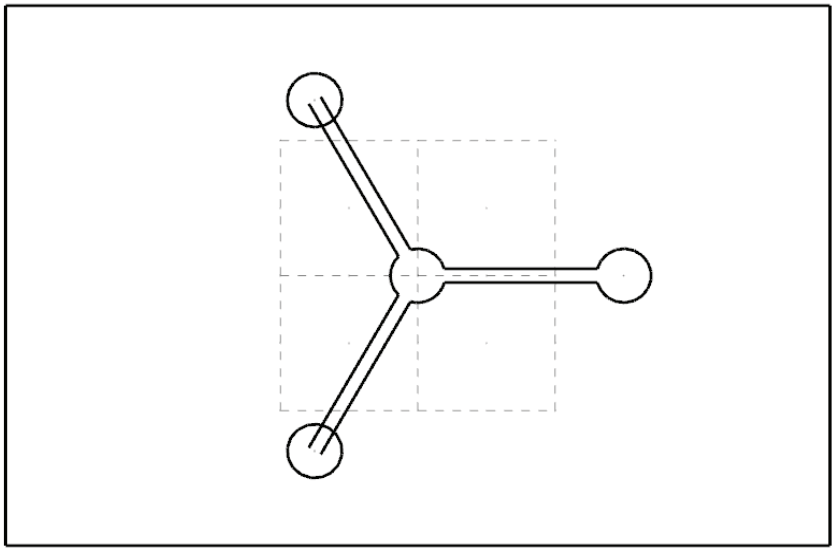
- Press **Alt+F1** to fit the drawing to the screen.

- Clean up the right side circle by selecting the entities to be divided as shown in Figure: 9.0.4.

Figure: 9.0.4

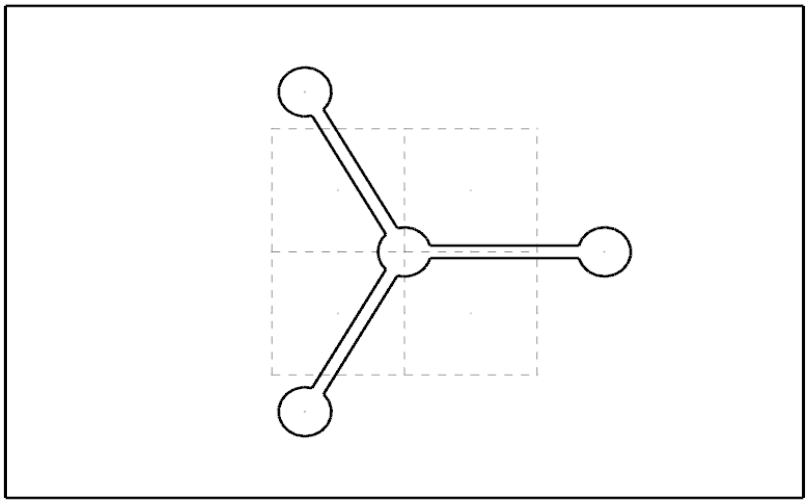


- The geometry should look as shown.



- Clean the other two circles in the same way.

- The final geometry should look as shown.



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

STEP 10: SAVE THE FILE

FILE

- **Save As.**

- Click on the **Browse** icon as shown.

- Find a location on the computer to save your file. File name: "Tutorial 6_ No Core Wirepath".

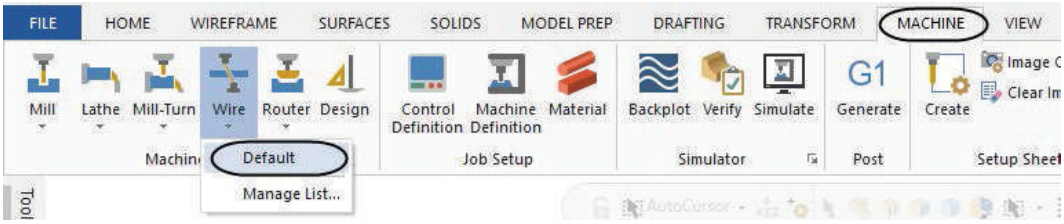
STEP 11: SELECT THE MACHINE AND SET UP THE STOCK

In Mastercam, you select a **Machine Definition** before creating any toolpaths. The **Machine Definition** is a model of your machine tool's capabilities and features and 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.WMD-9. 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** from the list as shown.



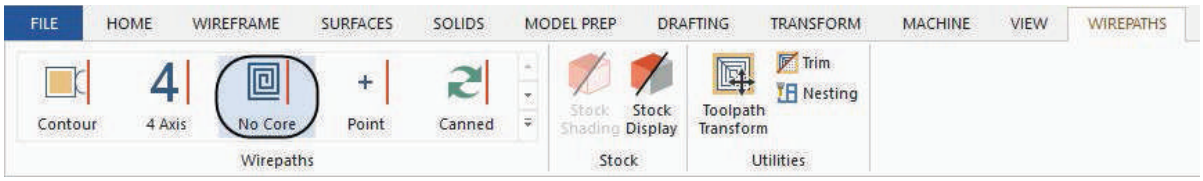
STEP 12: MACHINE PART USING NO CORE WIREPATH

No core wirepaths pocket out areas of material without producing a slug. The wirepath typically starts at a pre-drilled hole in the material, and zigzags or spirals outward until all material within the chained geometry is removed.

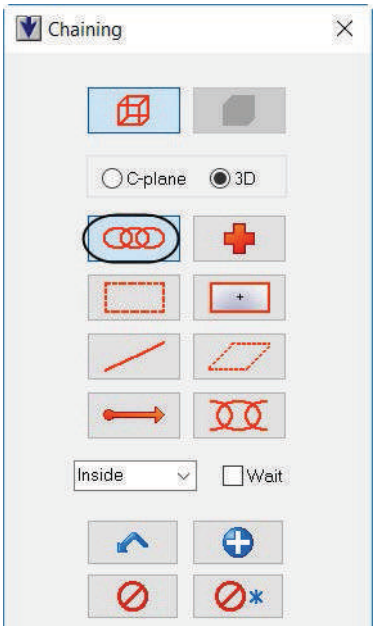
Toolpath Preview:

WIREPATHS

- From the **Wirepaths** group, select **No Core**.

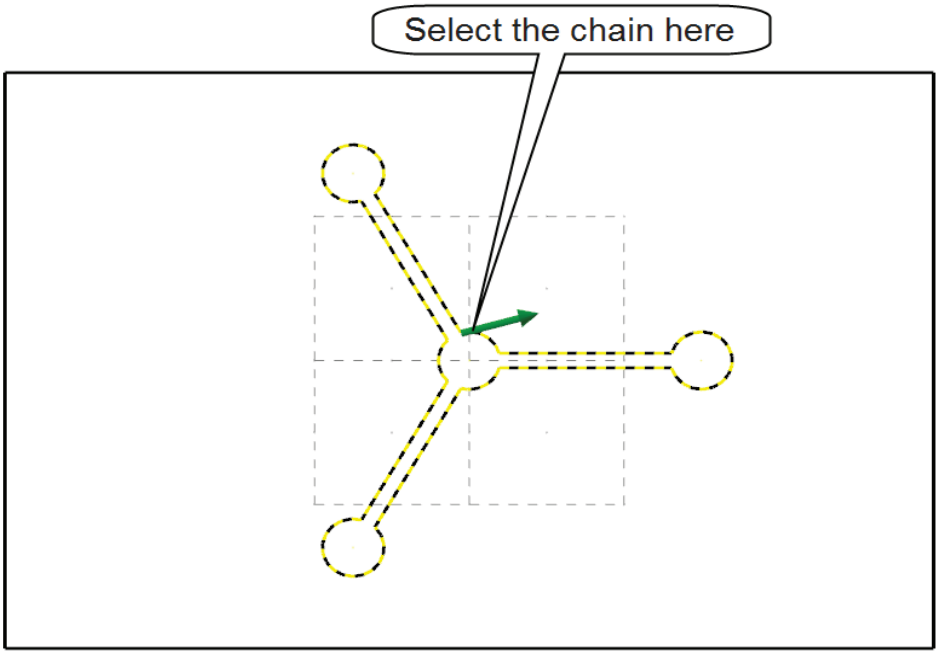


- From the **Chaining** dialog box, select the **Chain** button if needed.



- [Select Pocket chain 1]: Select the chain in CW direction as shown in Figure: 12.0.1.

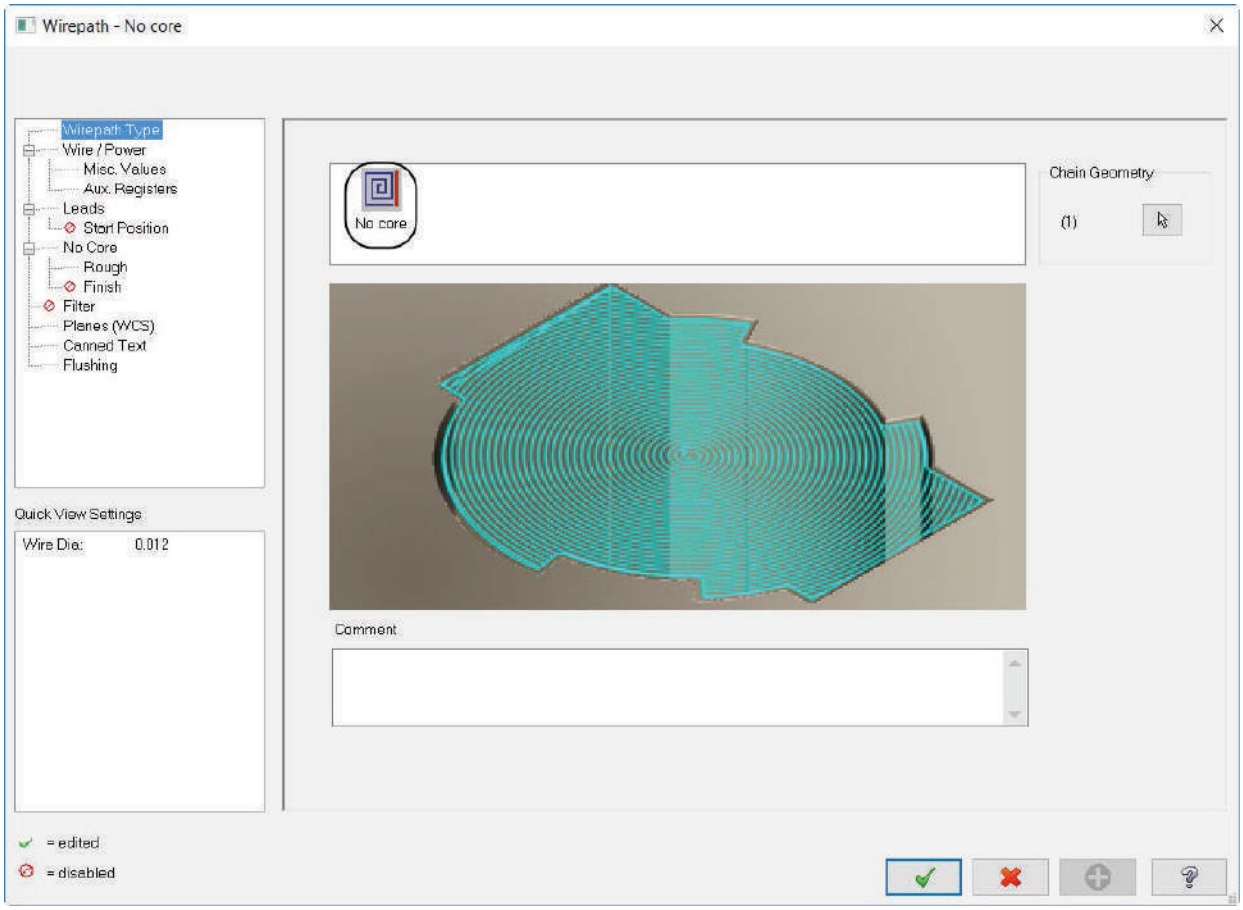
Figure: 12.0.1



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

12.1 Wirepath Type

- In the **Wirepath type** page, the **No core** toolpath should be already selected.



12.2 Set the Wire/Power parameters

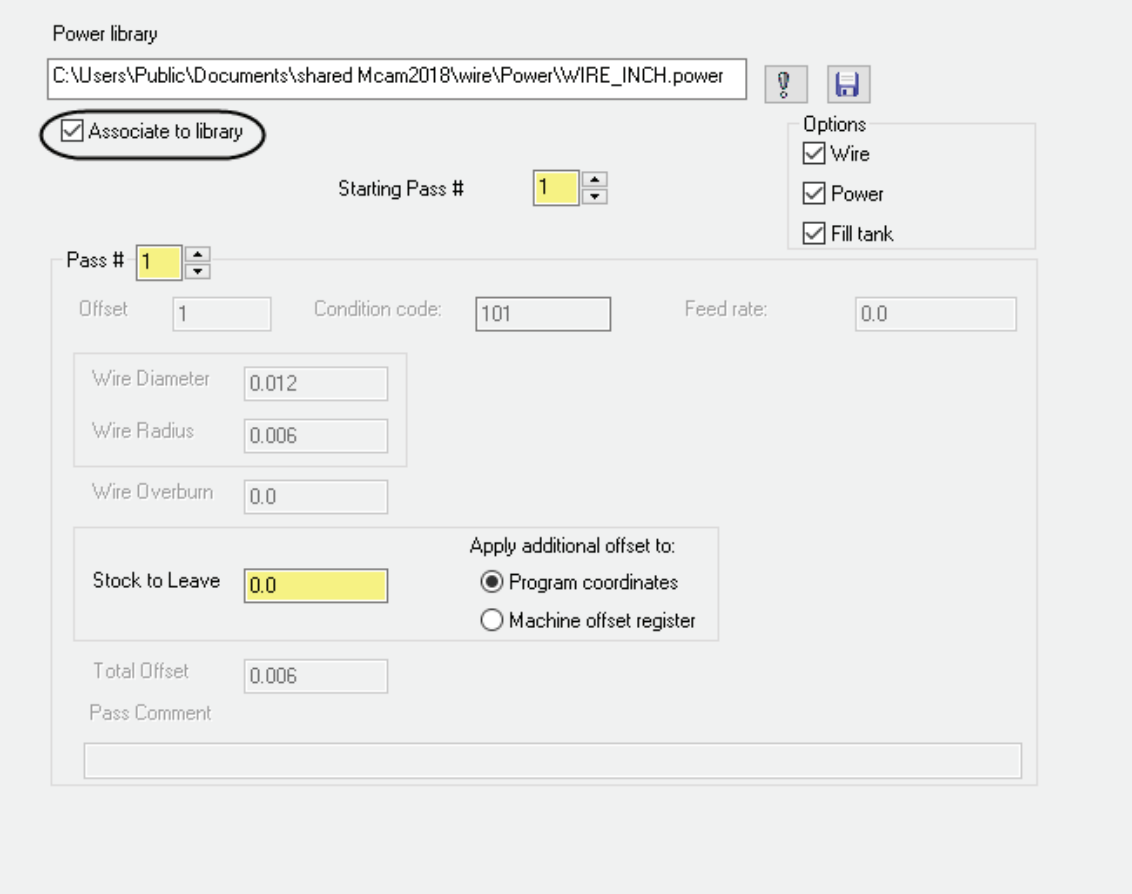
The **Wire/Power** page allows you to choose a wire power settings library file and use its values for the wirepath operation. You can also set the pass number to use for the first pass of the wirepath, and specify the settings for a single pass. To edit the power settings for each pass you have to disassociate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

- From the **Tree View** area, select **Wire/Power**.

NOTE: If you need to create your own wire power library, see Tutorial #1 (**page 46**). To open the library saved in Tutorial #1, click on the **Open** icon and select the library from the Power library list. Otherwise, use the default WIRE_ INCH.power library as shown in this tutorial.

- Enable **Associate to library** to use the default **WIRE_INCH.power** library as shown in Figure: 12.2.1.
- If an **Edit Power Settings** dialog box appears, select the option **No**.

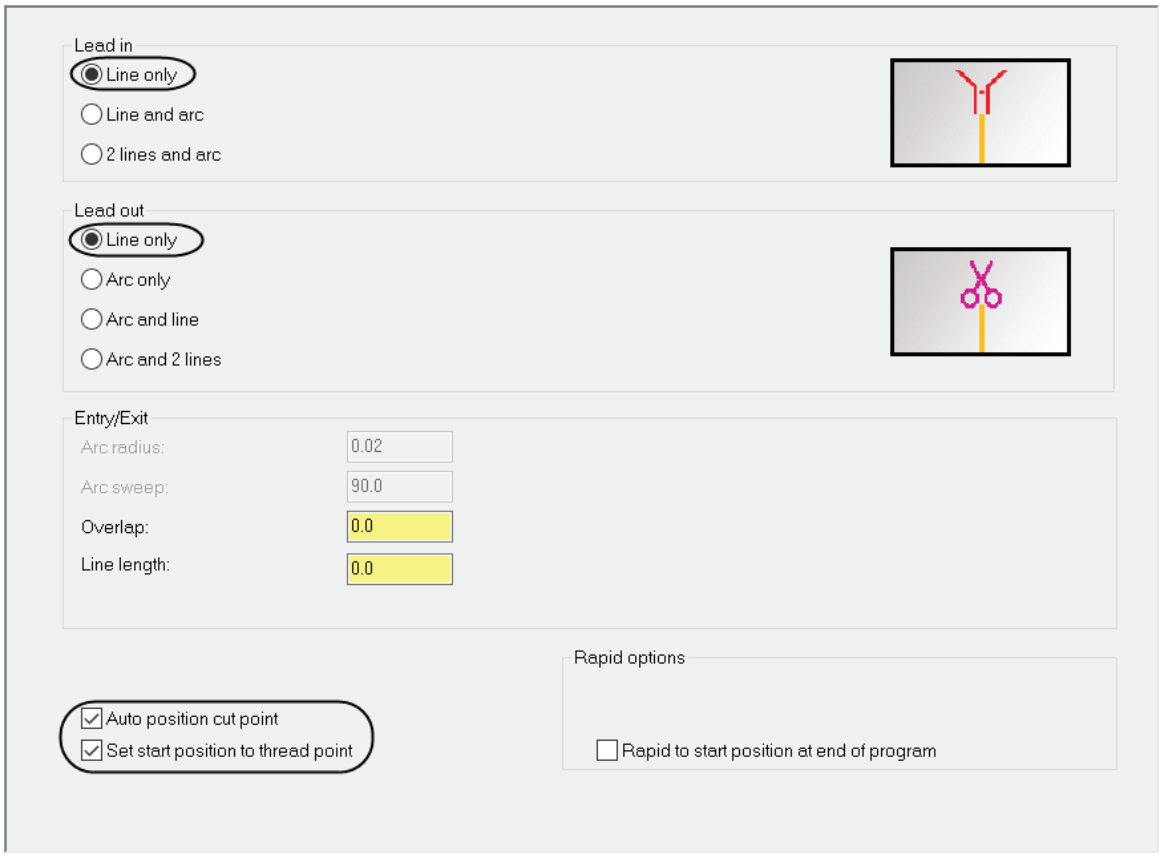
Figure: 12.2.1



12.3 Leads

Leads page allows you to set the wire motion when entering or exiting a part.

- From the **Tree View** area, select **Leads**.
- Make sure that the **Lead in** and **Lead out** are set to **Line only,** and set the rest of the parameters as shown.



Lead in sets the shape of the part entry motion. The lead in extends from the thread point to the start of contour.

Line only creates a line between the thread point and start of contour.

Lead out sets the shape of the part exit motion. Lead out extends from the end of contour to the cut point.

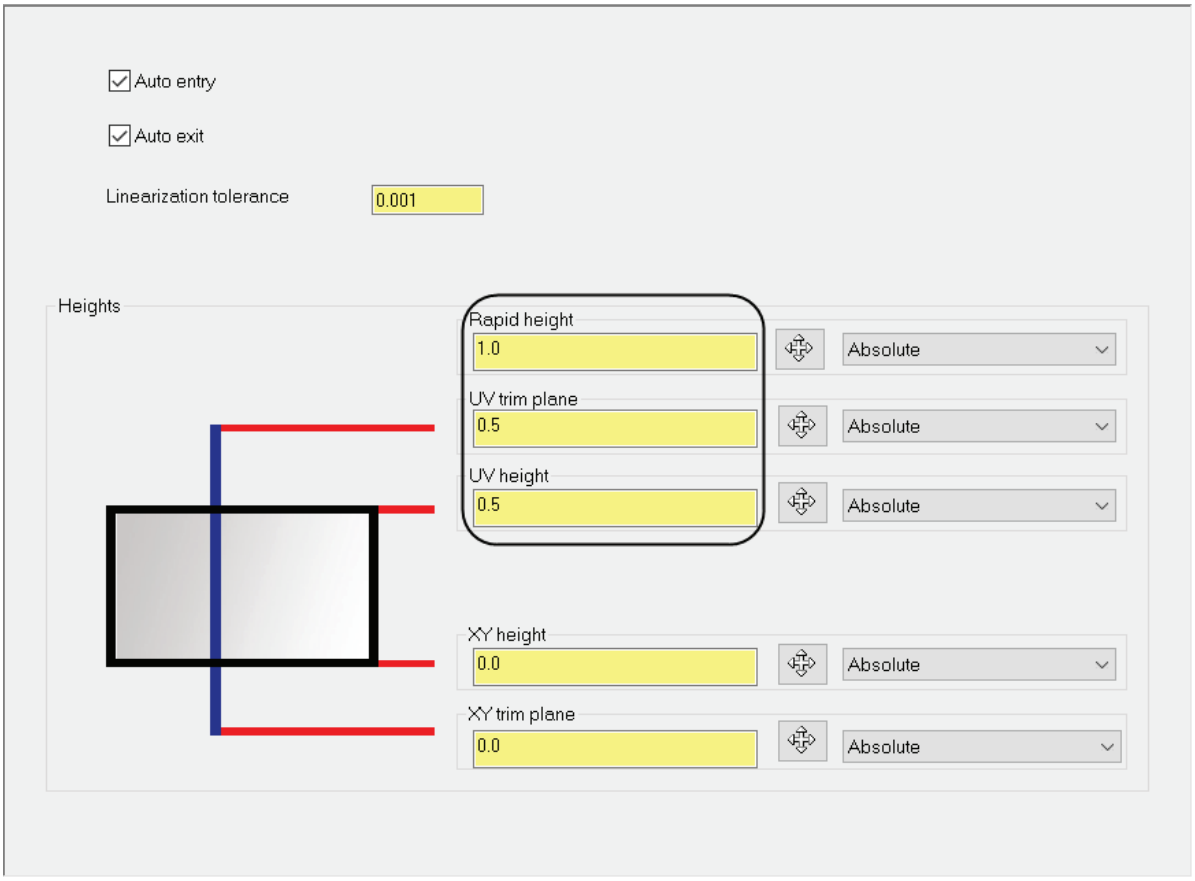
Auto Position Cut Point allows Mastercam to determine the most efficient cut point for a contour.

Set start position to thread point enabled moves the job start point to the thread point of the first contour in the operation.

12.4 No Core

No core page allows you to establish No core wirepath settings such as the Z depth of a rapid move, the positions of the upper and lower wire guides, and the UV and XY plane heights (usually the upper and lower contour of the geometry).

- From the **Tree View** area, select **No core** and make sure that the parameters are set as shown.



Auto entry/exit starts/ends the No core wirepath at the thread position.

Rapid height sets the Z depth of a rapid move. Can be higher than the **UV height** or **UV trim planes** in order to clear any clamps.

UV trim plane can be used to set the location of the upper guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to G code 1015 in the NC file. It is usually set directly at the controller.

UV height sets the height of the part, the upper contour of the wirepath.

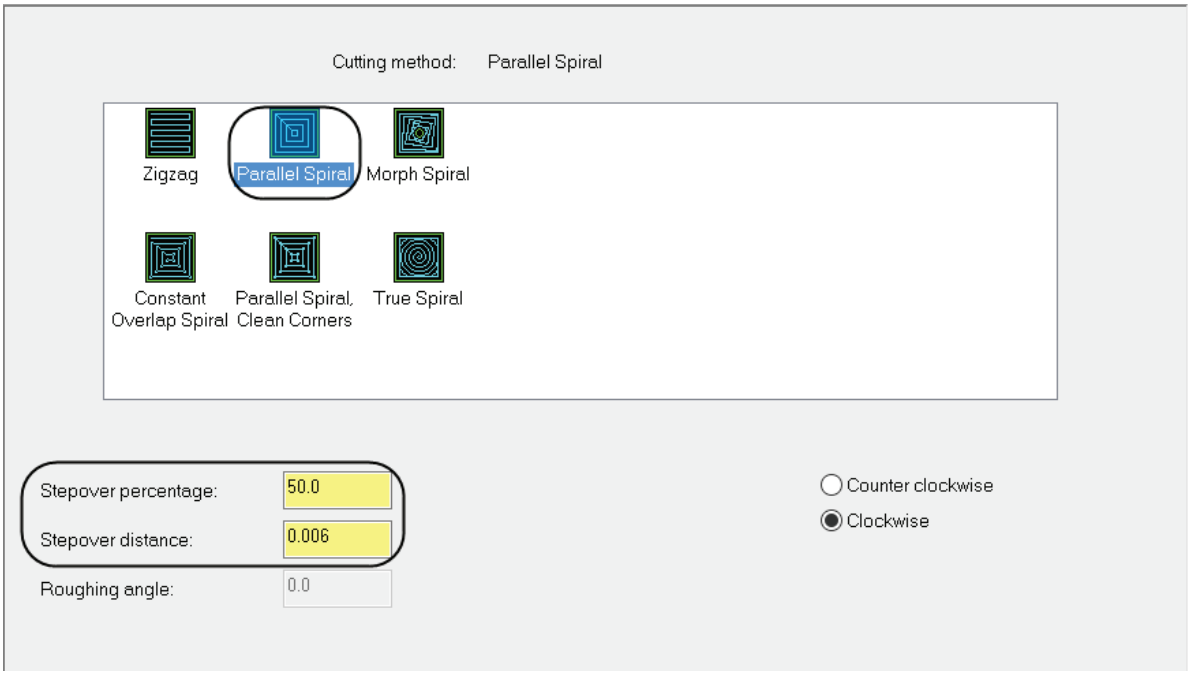
XY height sets the location of the XY plane that contains the geometry for the lower contour of the wirepath.

XY trim plane can be used to set the location of the lower guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to Gcode 1015 in the NC file. It can be set directly at the controller. When not using the XY trim plane, enter the same value as the XY height.

12.5 Rough

Rough page allows you to set up rough passes on no core wirepaths. You can define the type of no core cutting method (Zigzag, Constant Overlap Spiral, Parallel Spiral, Parallel Spiral with Clean Corners, Morph Spiral, or True Spiral). It also allows you to set the stepover percentage and distance.

- From the **Tree View** area, select **Rough** and make sure that the parameters are set as shown.



Cutting method allows you to set the cutting pattern that Mastercam Wire will use to clean out the No core part.

Stepover percentage sets the distance the wire shifts over between XY moves, as a percentage of wire diameter. Changing the stepover percentage automatically adjusts the stepover distance.

Clockwise sets the direction for the spiral no core cutting methods.

- Select the **OK** button to exit the **Wirepath - No core** parameters.

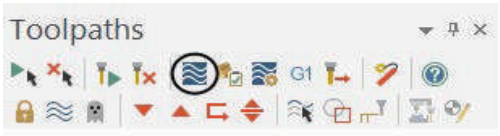
NOTE: A thread warning appears on the screen. "You have checked auto entry but have not chained a thread point. Press OK to thread at 0,0." Mastercam will automatically thread the wire at the origin.

- If a warning appears, select the **OK** button to continue.

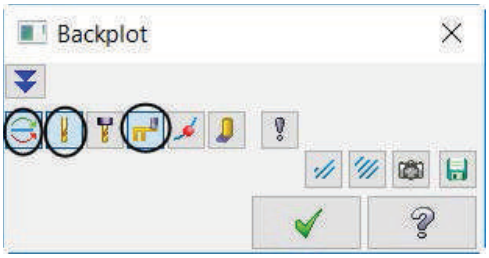
STEP 13: BACKPLOT THE WIRE CONTOUR WIREPATH

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 wirepaths, Mastercam displays the current X, Y, and Z coordinates in the lower left corner of the screen.

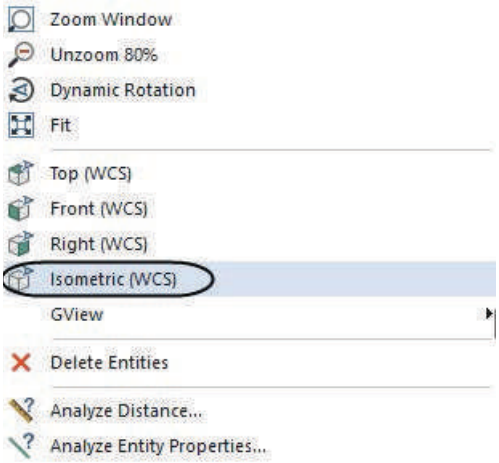
- Make sure that the toolpath is selected (signified by the green check mark on the folder icon).
- Select the **Backplot selected operations** button.



- Make sure that you have the following buttons turned on to see the tool and the rapid moves.



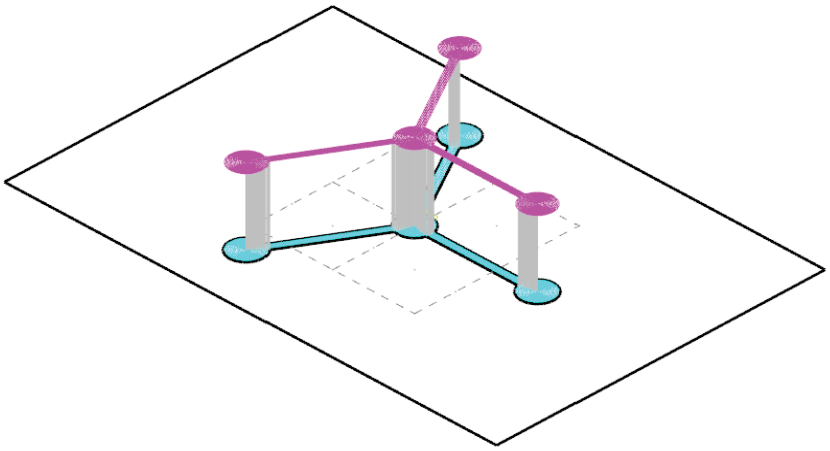
- Right mouse click in the graphics window and select **Isometric** as shown.



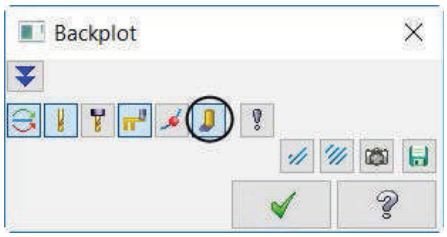
- Press **Alt+F1** to fit the drawing to the screen.
- Select the **Play** button to run **Backplot**.

- The toolpath should look as shown in Figure: 13.0.1.

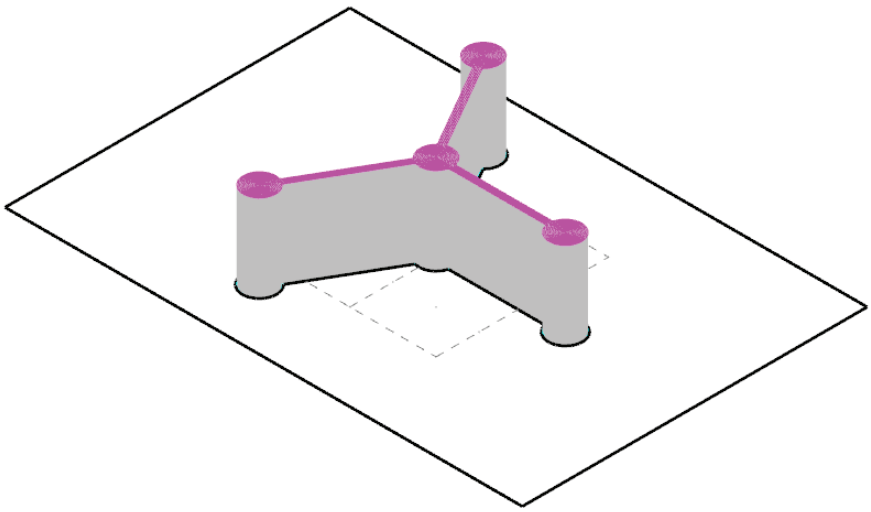
Figure: 13.0.1



- In the **Backplot** dialog box enable **Quick Verify** as shown.



- The toolpath should look as shown.



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

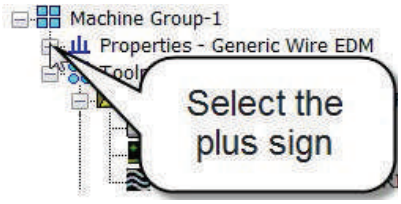
NOTE: The material inside of the shape is completely removed.

STEP 14: VERIFY THE WIREPATH

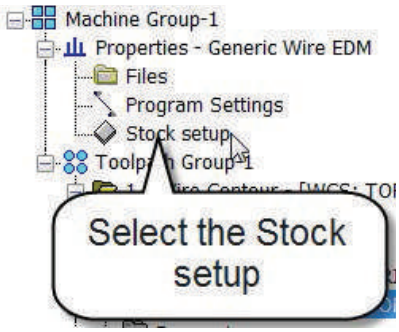
Verify simulates the machining of a part from a solid stock model display. The stock dimensions are based on the values that you can specified in the **Stock Setup**.

14.1 Set up the stock

- From the **Toolpaths Manager**, select the plus sign (+) in front of the **Properties** to expand the **Toolpaths Group Properties**.

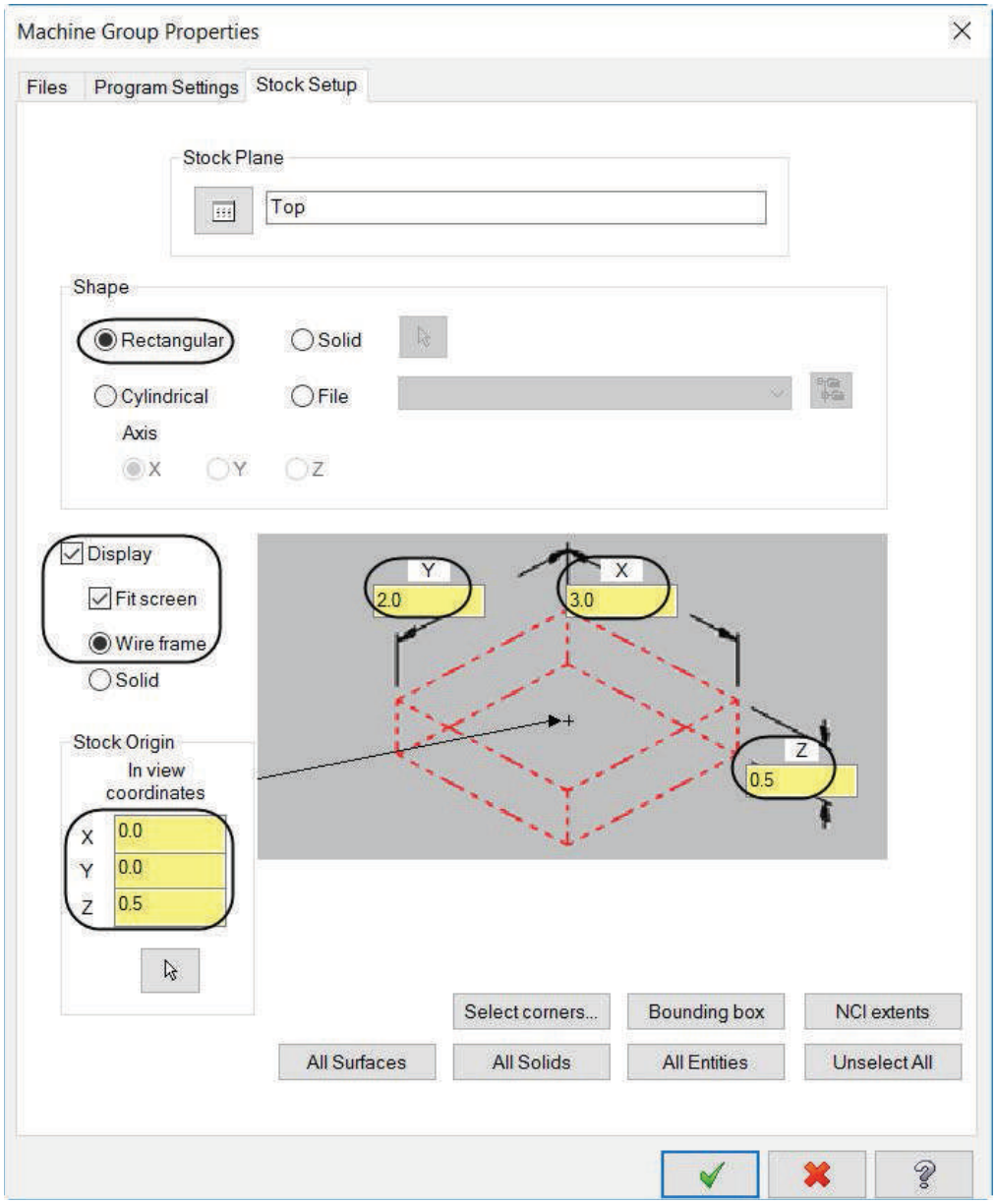


- Select the **Stock setup** to set the stock size.



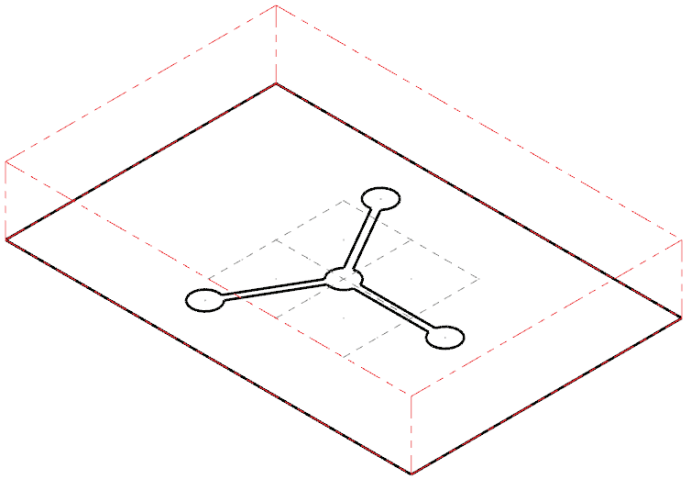
- Change the parameters to match Figure: 14.1.1.

Figure: 14.1.1



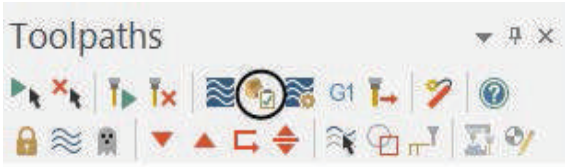
- Select the **OK** button to exit **Machine Group Properties**.

- Press **Alt+T** on your keyboard to remove the toolpath display from your screen.
- The stock is displayed with phantom red lines and it should look as shown.



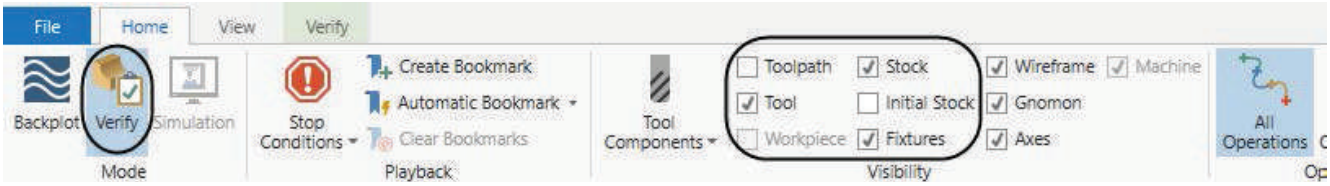
14.2 Verify the Wirepaths

- Select the **Verify selected operations** button.

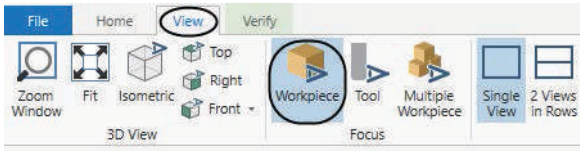


- From **Verify** change the settings for the **Visibility** as shown in Figure: 14.2.1.

Figure: 14.2.1



- Select **View** tab and enable **Workpiece** as shown.

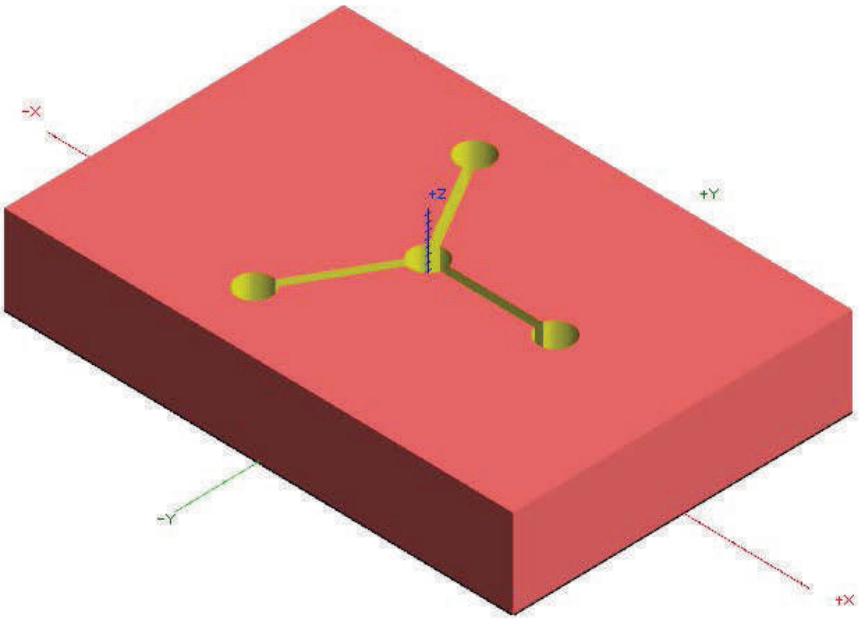


- Select **Verify** tab and enable **Color Loop** as shown.

- Right click and select the **Isometric** option.
- Right click and select the **Fit** option.
- Select the **Play Simulation** button to run **Verify**.

- The part should look as shown in Figure: 14.2.2.

Figure: 14.2.2



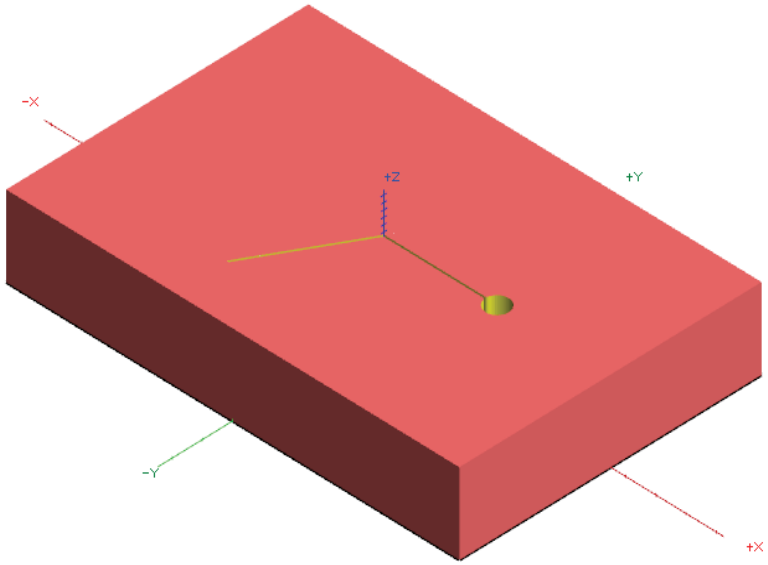
NOTE: To rotate the part, move the cursor to the center of the part and click and hold the mouse wheel and slowly move it in one direction.
To **Zoom In** or **Out** hold down the mouse wheel and scroll up or down as needed.

- Under the **View** tab, click on the **Isometric** icon

and then the **Fit** icon

to see the part in the original position if needed.
- To check the part step by step, click first on the **Start (Home)** icon.
- Click on the **Step Forward** icon to see the tool moving one step at a time.

- The part should look as shown after several steps.

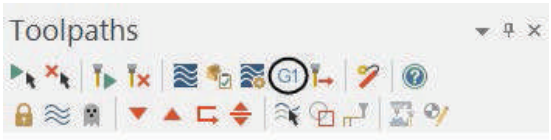


- Click on the **Step Forward** until the toolpath is completed.

- To exit **Mastercam Simulator** click on the **Close** icon.

STEP 15: POST THE FILE TO GENERATE THE NC CODE

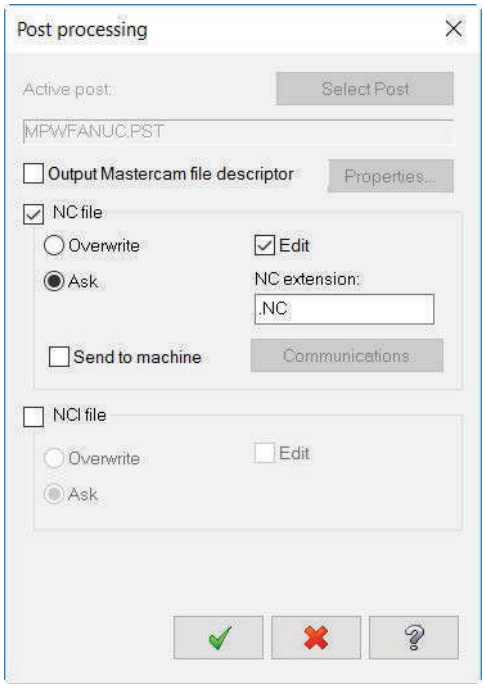
- Select **Post selected operations** button from **Toolpaths Manager**.



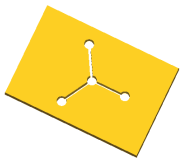
NOTE: The active **Post Processor** is a generic FANUC post processor.

- In the **Post processing** window, if needed, make any necessary changes as shown in Figure: 15.0.1.

Figure: 15.0.1

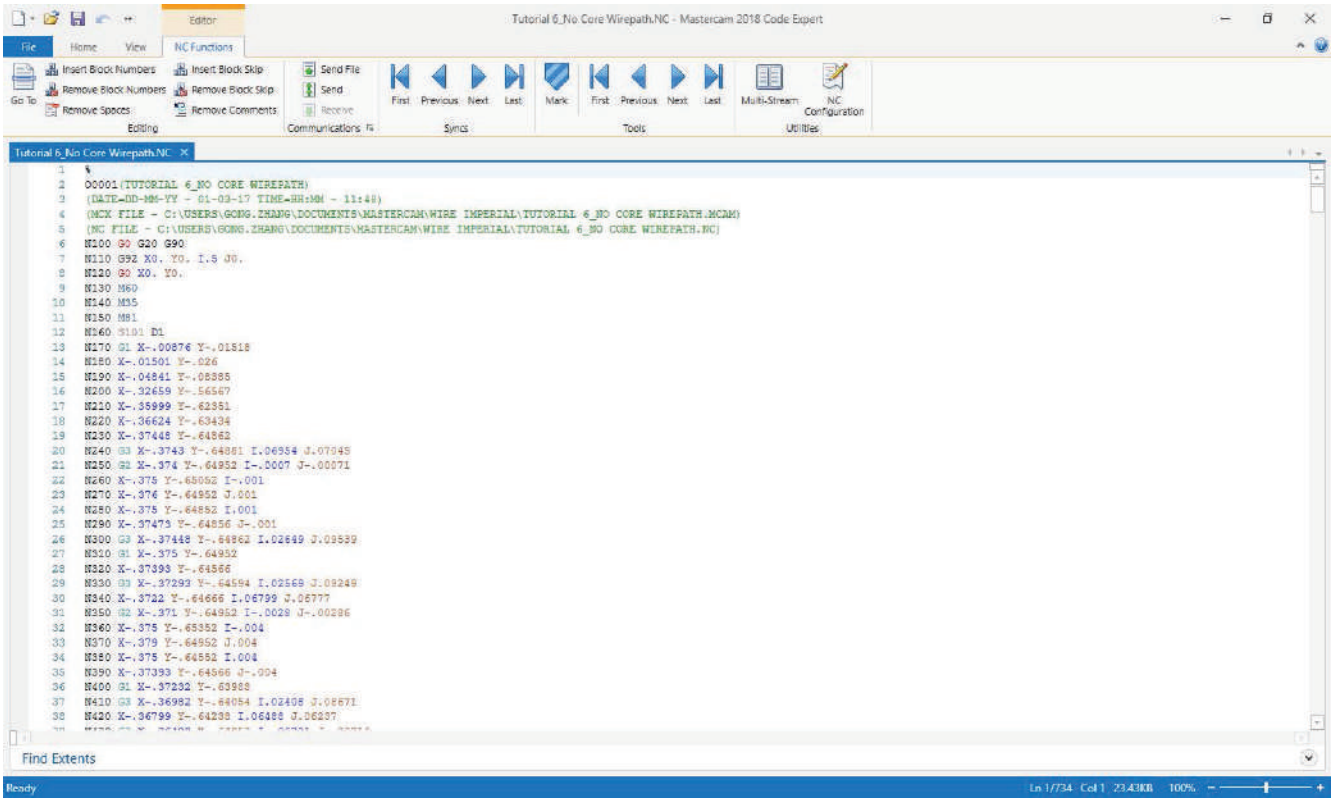


- Select the **OK** button to continue.
- Save the NC file.



- The NC file will appear on the screen as shown in Figure: 15.0.2.

Figure: 15.0.2

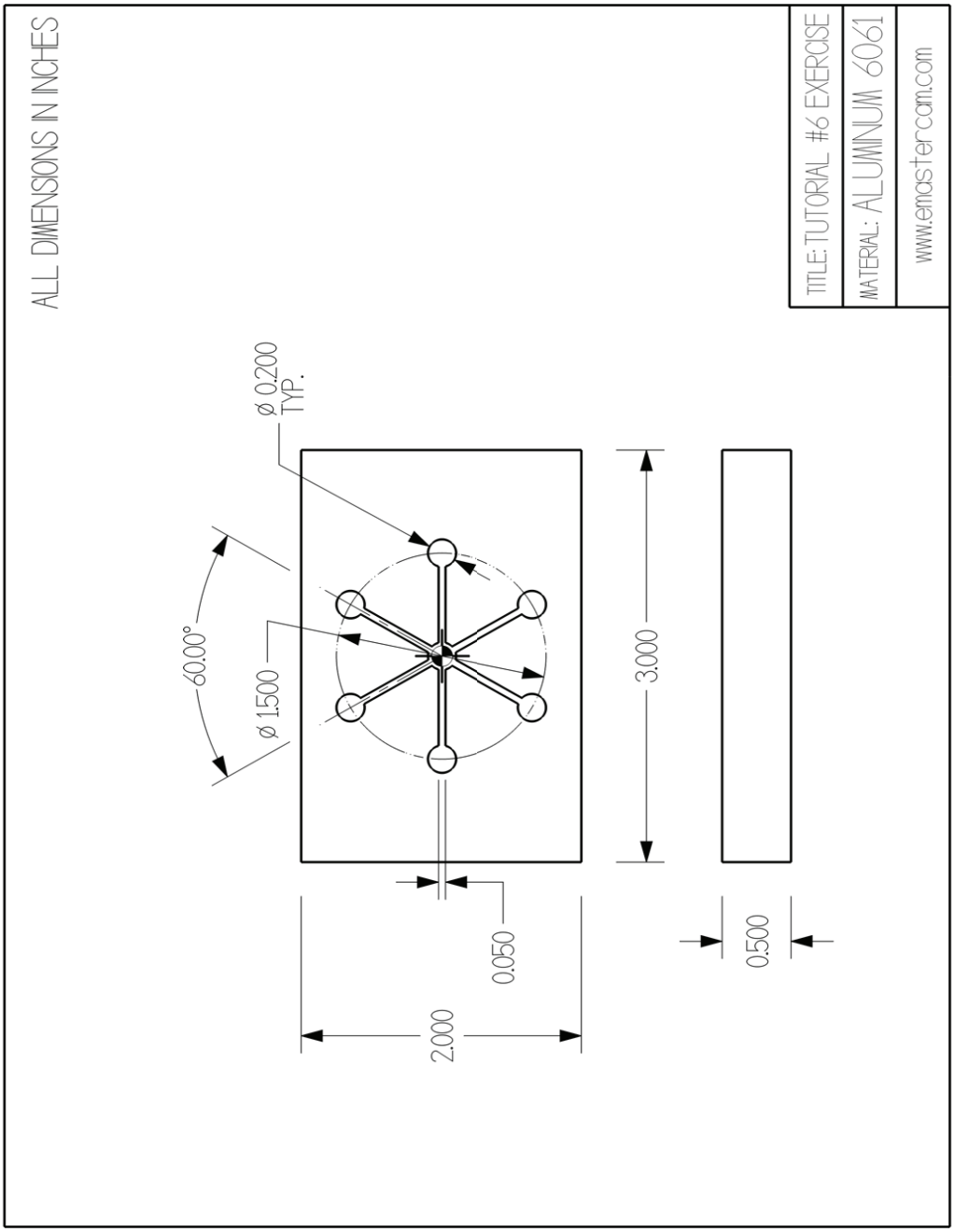


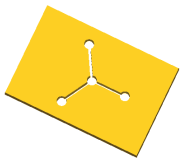
NOTE: The G-code that you have created will appear on the screen. If the code looks okay, you can shut down the window without saving it. If you need to change the code, make sure you save it before you close the window down.
How the program is sent to the machine depends on the shop setup.

- Select the "**X**" box at the upper right corner to exit the **Mastercam Code Expert.**

STEP 16: SAVE THE UPDATED MCAM FILE

REVIEW EXERCISE -STUDENT PRACTICE

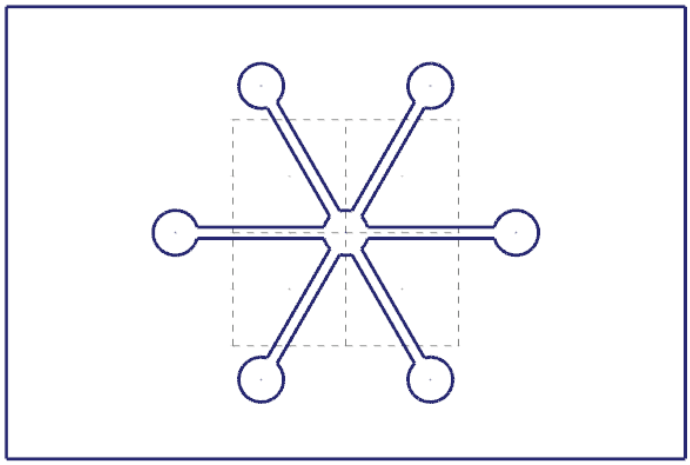




CREATE THE GEOMETRY FOR TUTORIAL #6 EXERCISE

Use these commands to create the geometry.

- Rectangle.
- Circle Center Point.
- Bolt Circle
- Line Endpoints.
- Line Parallel.
- Rotate/Copy.
- Trim Entities.

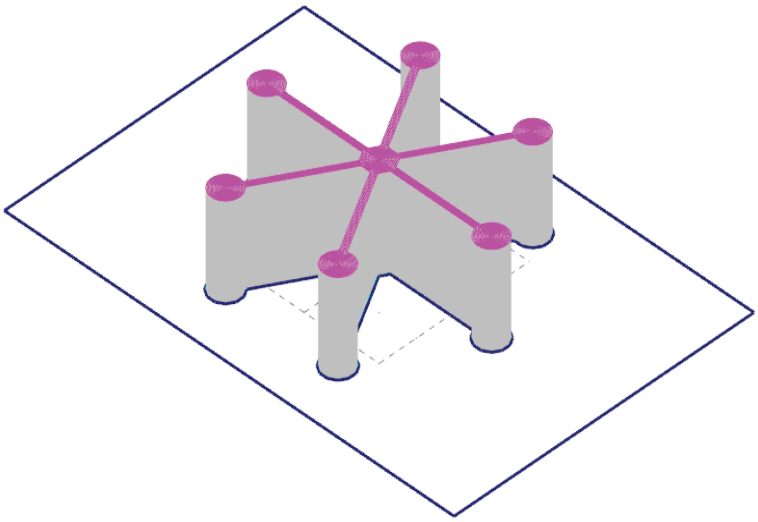


CREATE THE TOOLPATHS FOR TUTORIAL #6 EXERCISE

Create the Toolpaths for Tutorial #6 Exercise as per the instructions below.

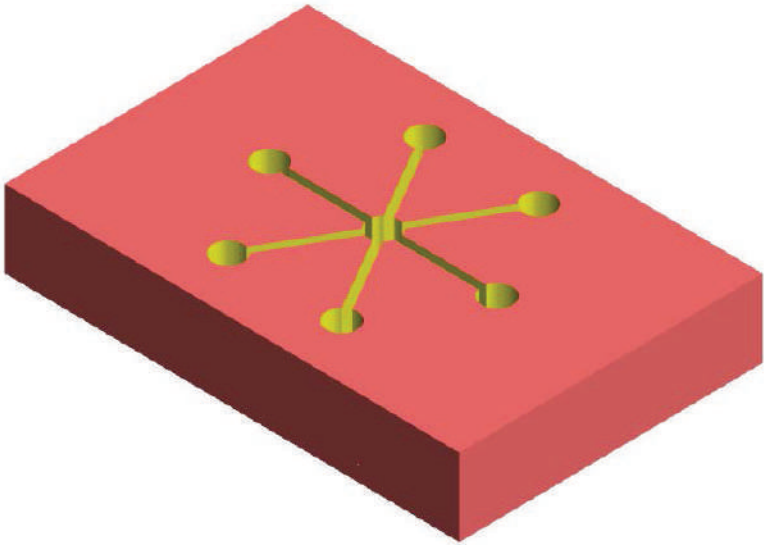
Machine The Part Using No Core Wirepath.

- Inorder to complete this operation follow **page 383** to **page 389**.



Verify The Wirepath.

- Set the **Stock** according to the part.
- Inorder to complete this operation Verify the part and follow **page 392** to **page 395**.



NOTES:

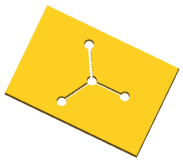
TUTORIAL #6 QUIZ

- What do you need to know to use Bolt Circle command to create circles equally spaced along a circle?

- How can you zoom in to better see a smaller area of your geometry?

- When should you use a No Core Wirepath?

- What does Rough page allow you to set?



QUIZ ANSWERS

Objectives:

✓ The answers to the 6 Tutorial quizzes.

WIRE TUTORIAL QUIZ ANSWERS

Tutorial 1 Answers

- What command allows you to copy a shape several times along the X-Axis?
- The **Translate** command with the **Copy** option enabled.

- How can you remove the temporary group/result colors?
- Use **Clear Colors** function or start another transform command.

- What is the Thread Point?
- The **Thread Point** is the point where the machine threads the wire.

- What is the Cut Point?
- The **Cut Point** is the point where the machine cuts the wire before moving to the next 4.

- What does Backplot do?
- **Backplotting** shows the path the wire takes to cut the part. This display lets you spot errors in the program before you machine the part. As you backplot toolpaths, Mastercam displays the current X, Y, and Z coordinates in the lower left corner of the screen.

Tutorial 2 Answers

- What command allows you to create a line given one endpoint, the length, and the angle of the line?
- **Line Endpoint** allows you to set in the Ribbon bar the line length and the angle.

- In which order you have to select the three entities when using Trim 3 entities command?
- 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.

- When should you use a Contour Wirepath?
- When the geometry has the same general shape in both the XY plane (the lower contour) and the UV plane (the upper contour).

- What does the Land Height set?
- **Land height** sets the plane at which the wire pivots to the taper angle.

- What does Verify do?
- **Verify** simulates the machining of a part from a solid stock model display. The stock dimensions are based on the values that you can specified in the **Stock Setup**.

Tutorial # 3 Answers

- What command allows you to create a line tangent at an angle?
- **Line Endpoint** command with the tangent option enabled in the Ribbon Bar.

- What does Mirror command allow you to create?
- The **Mirror** command allows you to create mirror images of geometric and drafting entities by reflecting them symmetrically with respect to a defined axis or point.

- What does Divide command do?
- **Divide** command deletes entities based in the nearest intersection.

- What is a Tab?
- A **Tab** is the uncut material that secures the part to the material; can be cutoff with the skim cut pass or in a separate pass.

- What does Stop parameters allow you to set?
- **Stops** page allows you to create optional stops (M01) or program stops (M00) in a wirepath. For contour wirepaths, you have the options to apply the stops before and/or after tabs.

Tutorial # 4 Answers

- Where do you have to select the entities when using Trim 2 entities?
- You have to select the two entities on the side that you are keeping after trimming.

- What shortcut key allows you to force a window selection?
- **Ctrl** key forces a window selection.

- What does Compensation type set to Control output?
- **Compensation type** sets to **Control** outputs control codes for compensation and does not compute the compensated wirepath.

- What does Leads page allow you to set?
- **Leads** page allows you to set the wire motion when entering or exiting a part.

Tutorial # 5 Answers

- How is the geometry created in 2D construction mode?
- In 2D mode, all the geometry is created parallel to the current construction plane and at the current Z depth setting.

- How do you create the branches used to synchronize By Branch the top and the bottom profiles?
- You will need to break the geometry where you want to create the branches and then create the lines that connect the top with the bottom geometry.

- When should you use a 4 Axis Wirepath?
- **4-Axis wirepaths** have geometry in both the XY plane (the lower contour) and UV plane (the upper contour). Where a contour wirepath derives a second contour from the geometry, a 4-Axis wirepath can have completely different geometry in the two planes, resulting in different wire movement in the XY and UV planes.

- What does 4 Axis page allow you to set?
- **4-Axis** page allows you to establish 4-Axis wirepath settings such as the linear arc move format (direct or taper), trimming options, and synchronization of the lower (XY) and upper (UV) contours.

Tutorial # 6 Answers

- What do you need to know to use Bolt Circle command to create circles equaly spaced along a circle?
- To use **Bolt Circle** you have to know the base point, the radius of the bolt circle, as well as the number and the diameter of the circles

- How can you zoom in to better see a smaller area of your geometry?
- Hover the mouse above the focus point and then scroll up the mouse wheel.

- When should you use a No Core Wirepath?
- **No Core** wirepaths pocket out areas of material without producing a slug.

- What does Rough page allow you to set?
- **Rough** page allows you to set up rough passes on no core wirepaths. You can define the type of no core cutting method and set the stepover distance.

GENERAL NOTES

Objectives:

✓ Learn how to customize Mastercam.
✓ Learn how to use and set the construction plane, the graphic views, and the depth of the construction plane.
✓ Learn about the chaining.
✓ Learn about the Toolpaths Manager.
✓ Learn about the Machine Group Properties.
✓ Learn about the Contour wirepath options.
✓ Learn about 4-Axis wirepath.
✓ Learn about No Core wirepath.

DEFAULT KEY ASSIGNMENTS

Alt + 1	Top View.
Alt + 2	Front View.
Alt + 5	Right Side View.
Alt + 7	Isometric View.
Alt + A	Auto Save.
Alt + C	To access C-Hooks.
Alt + D	To open Drafting Options.
Alt + E	To hide/show entities.
Alt + F1	To fit all geometry to the screen.
Alt + F2	To unzoom.
Alt + F4	To exit File.
Alt + G	To display screen grid.
Alt + H	To open online help.
Alt + P	To revert to previous view.
Alt + S	Shade On/Off.
Alt + T	To hide/show toolpaths.
Alt + X	To set attributes from a selected entity.
Ctrl + C	To copy.
Ctrl + Z	To undo.
Ctrl + X	To cut.
Ctrl + Y	To redo.
Ctrl + V	To paste.
F1	To zoom to window.
F2	To unzoom.
F4	To analyze selected entity.
F5	To delete selected entities.
F9	To display coordinate info.
Shift + Ctrl + R	To regenerate the screen.

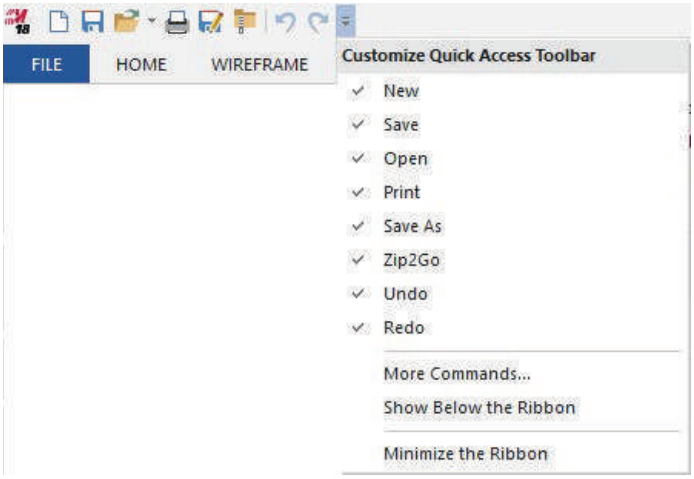
CUSTOMIZING THE INTERFACE

Mastercam makes it easy for you to customize the interface so that the functions that are most important to you are always easily available and functions that you do not need can be hidden.
You can customize the **Ribbon bars** and the graphics window right-click menu. Also, you can create new tabs as well.

CUSTOMIZING QUICK ACCESS TOOLBAR

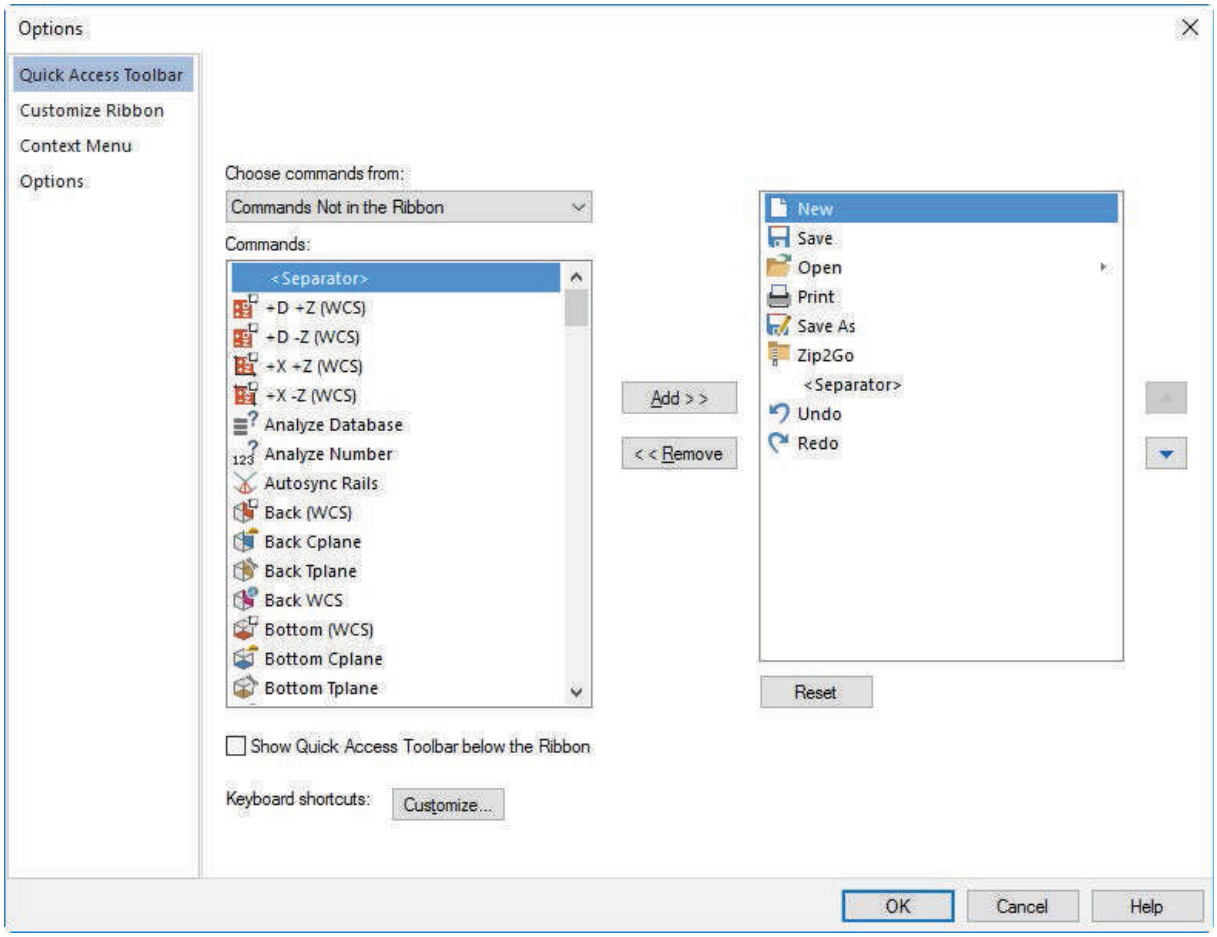
- By default, **Quick Access Toolbar** has the following functions as shown.

- You can move the **QAT** above or below the **Ribbon bar** and you can customize it as needed.



- From the **FILE** tab, select **Options**.

- The **Options** dialog box with the **Quick Access Toolbar** should look as shown.



- The ability to customize Mastercam is a powerful tool for improving productivity. For example, you can customize the workspace for different machine tools so only options relevant to that machine are displayed.

NOTE: Changing the drop-down and flyout menus now would make it difficult to follow instructions in this book. Leave the menus as they are until you've completed this book and feel confident in using Mastercam.

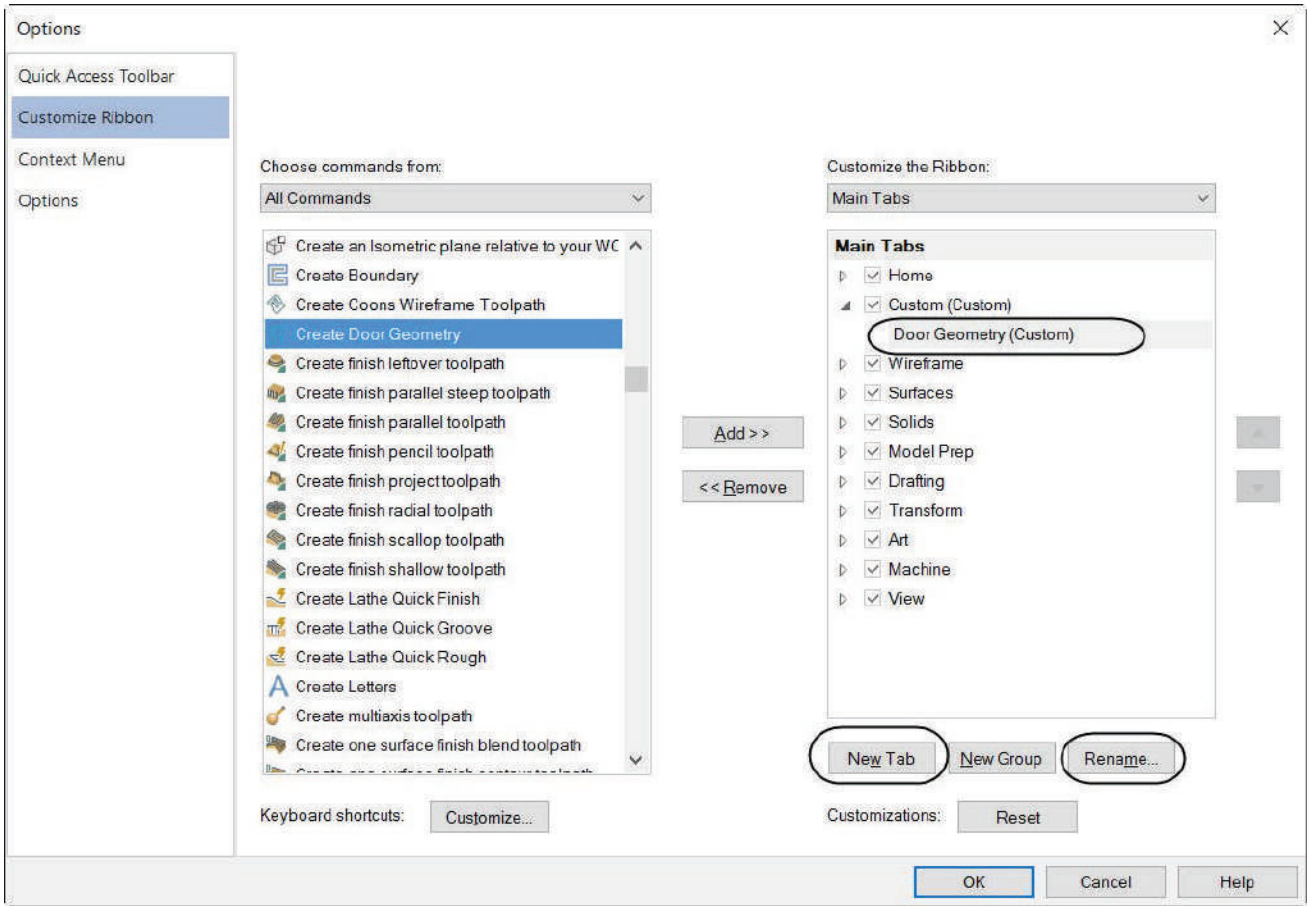
CREATE A NEW TAB

Creating a new tab allows you to add new commands to the **Ribbon bars**.

FILE

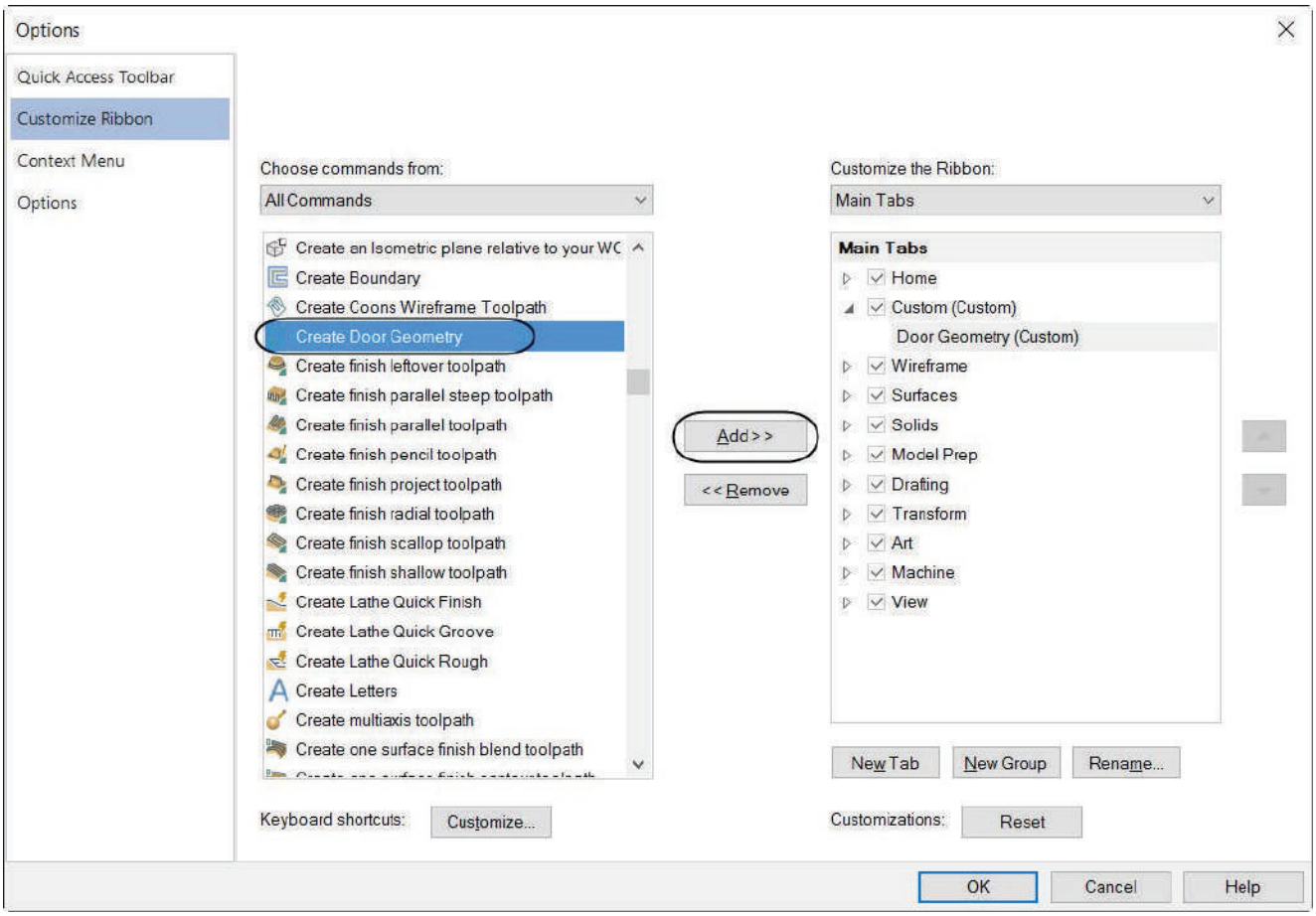
- **Options**.
- From the **Options** dialog box, select **Customize Ribbon**.
- Select **New Tab** button to create a new tab and a new group as shown in Figure: 1.
- Rename both tab and group as shown in Figure: 1.

Figure: 1



- Click on the **Choose command from** drop down arrow to display a list with all Mastercam commands. In this case, we will add the **Create Door Geometry** command to the **Ribbon bar**.

- Select the **Create Door Geometry** command and click on the **Add** button as shown.



- Select **OK** to exit the **Options** dialog box.
- The tab should look as shown.

- If you want to create another group that contains, for example, the toolpaths that you are usually using, follow the same method and create a new group by selecting **New Group**.

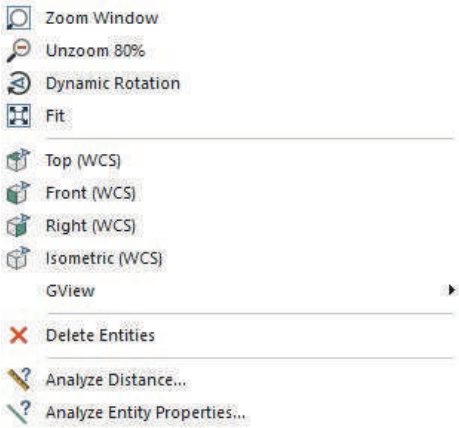
CUSTOMIZE THE RIGHT MOUSE BUTTON MENU

Customizing the right mouse button menu allows you to add, remove and rearrange commands.

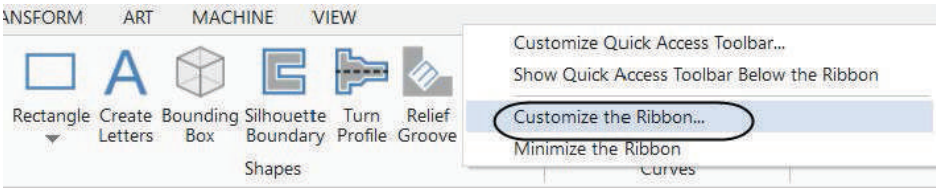
NOTE: Adding or removing drop down menu functions is associative. All references to the modified menus are updated (for example the modified drop down menu is used as a submenu in other drop down menus). When you add or remove functions from drop down menus and submenus, you immediately see the results in the current Mastercam session.

EXAMPLE: ADD THE ZOOM TARGET TO THE RIGHT MOUSE BUTTON MENU

- The default **Right mouse button menu** appears as shown.

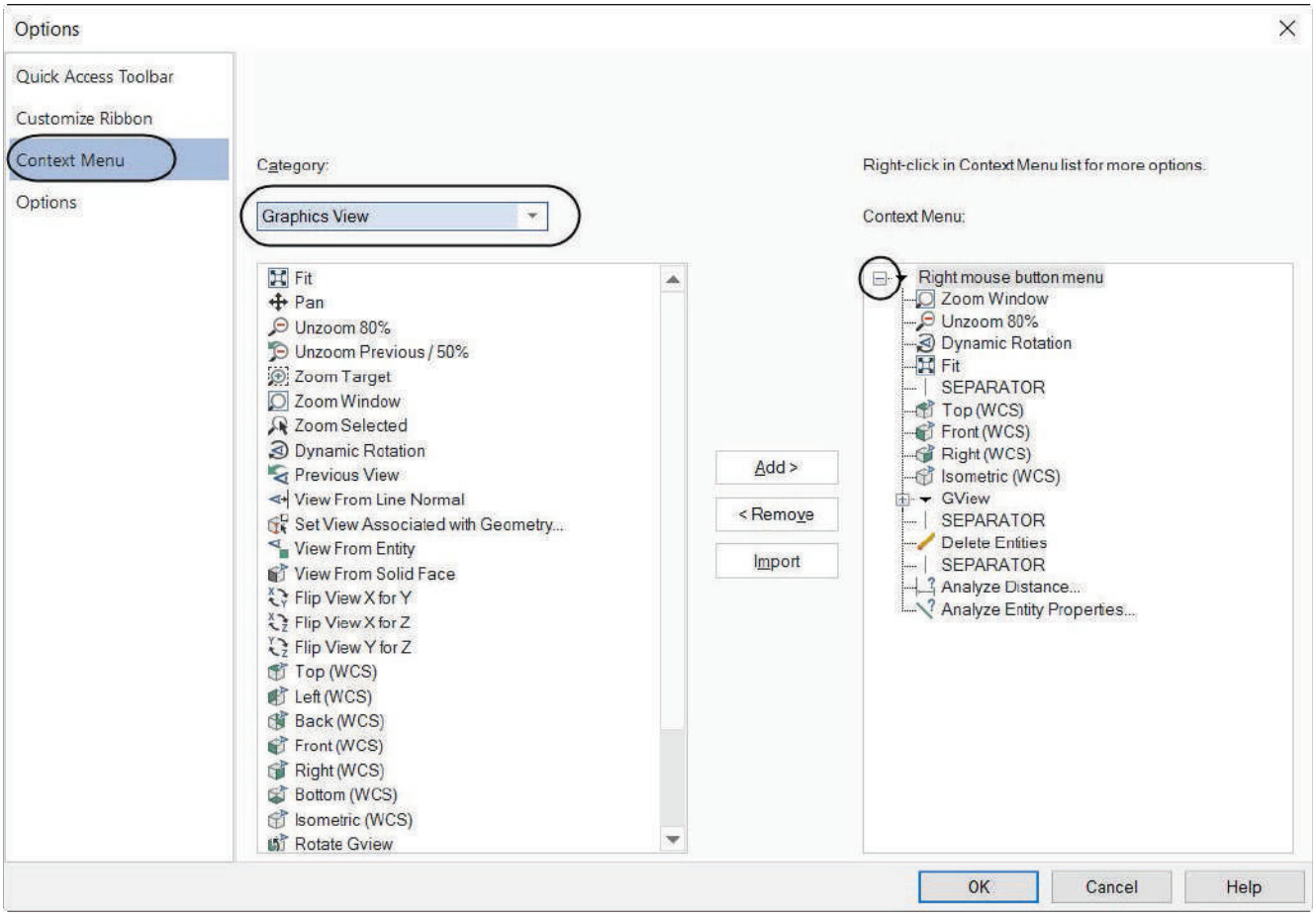


- Right mouse click above the **Ribbon bar** and select **Customize the Ribbon** as shown.

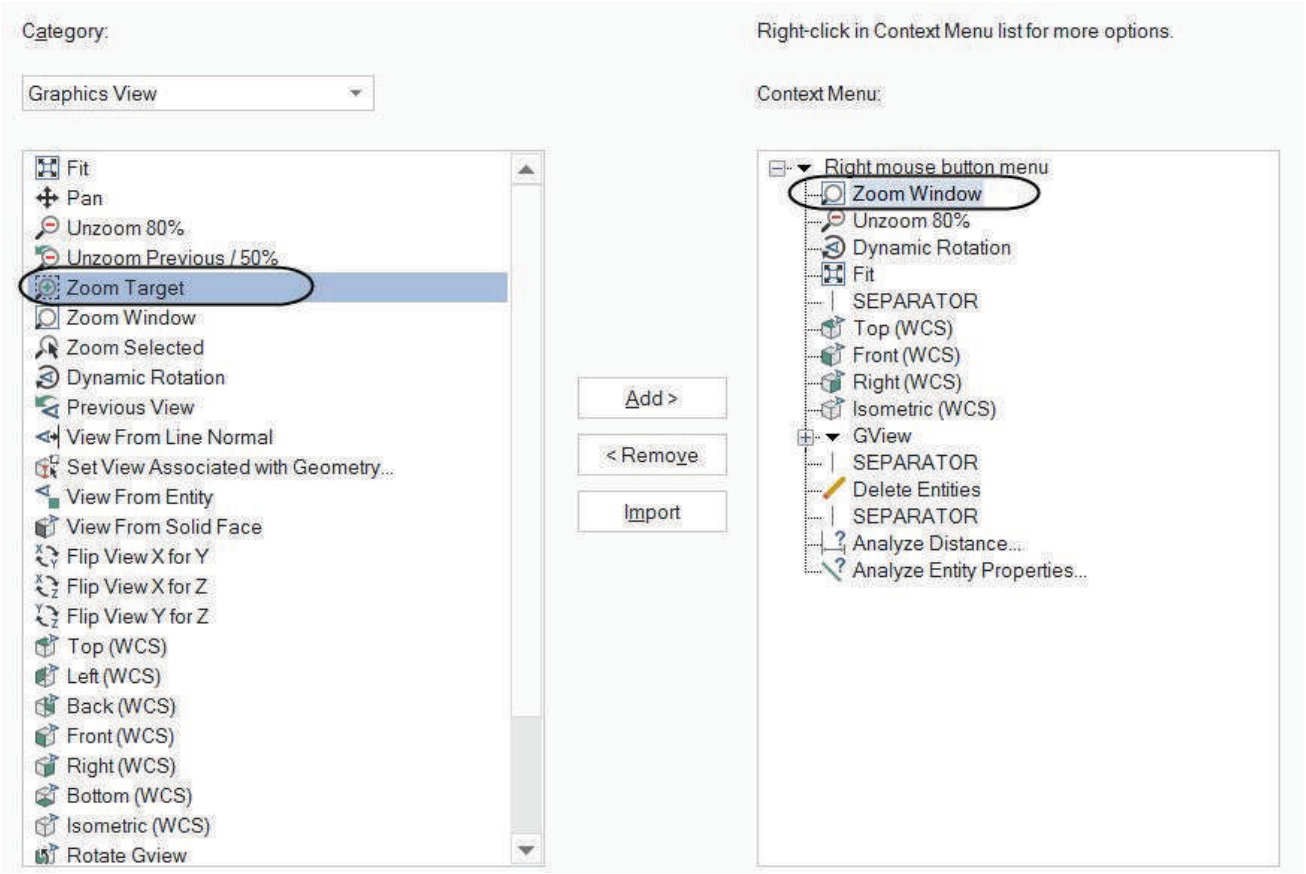


- Select **Context Menu** tab.
- Pick the plus (+) next to **Right mouse button menu**.

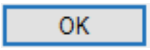
- Make sure that **Graphics View** is selected as shown.



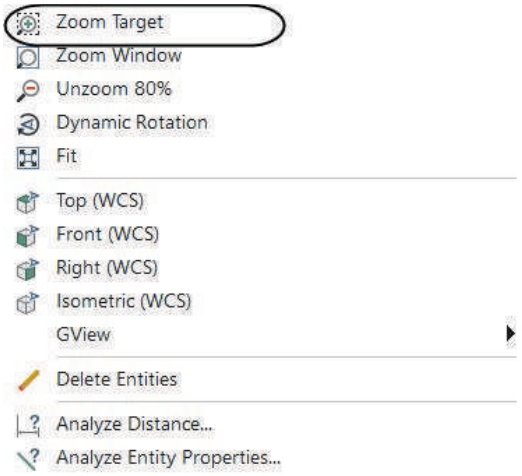
- Select the **Zoom Target** button in the **Commands** list.
- In the **Context Menu**, select **Zoom Window** to highlight it.



- Click on **Add**.

- Select the **OK** button to exit the **Options** dialog box. 

- **Right mouse click** in the graphics window to check the new menu**.**



KEY MAPPING

- **Key Mapping** allows you to map Mastercam functions to keyboard shortcuts, making your interactions with Mastercam more customized and efficient.
- The combination of key sequences you can use to create keyboard shortcuts are listed below (where "key" is an alphanumeric character or keyboard function key F1-F12).

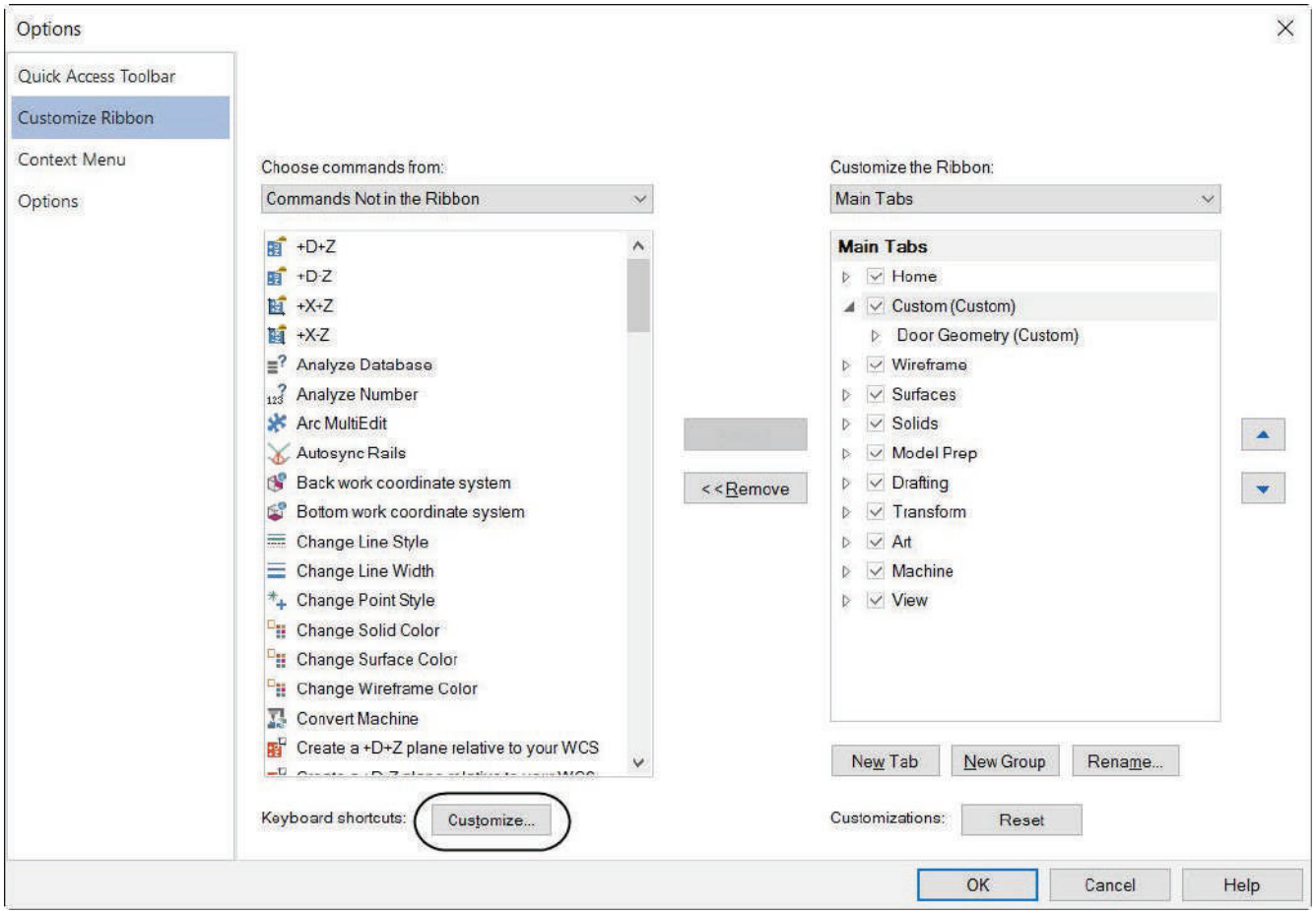
- **Alt** + **Key**.
- **Ctrl** + **Key**.
- **Ctrl** + **Alt** + **Key**.
- **Shift** + **Alt** + **Key**.
- **Shift** + **Ctrl** + **Key**.
- **Shift** +**Ctrl** + **Alt** + **Key**.
- **F1**-**F12**.

NOTE: You cannot map functions to key sequences which use a single alphanumeric key or one in combination with a shift key. These keys are reserved for use by Mastercam default keyboard shortcuts. For example, many of the single alphanumeric keys or **Shift** + **Key** sequences are used with data entry, **Ribbon bar**, and dialog box shortcut keys.

EXAMPLE: ALLOCATE CTRL + L FOR CREATE LINE ENDPOINT

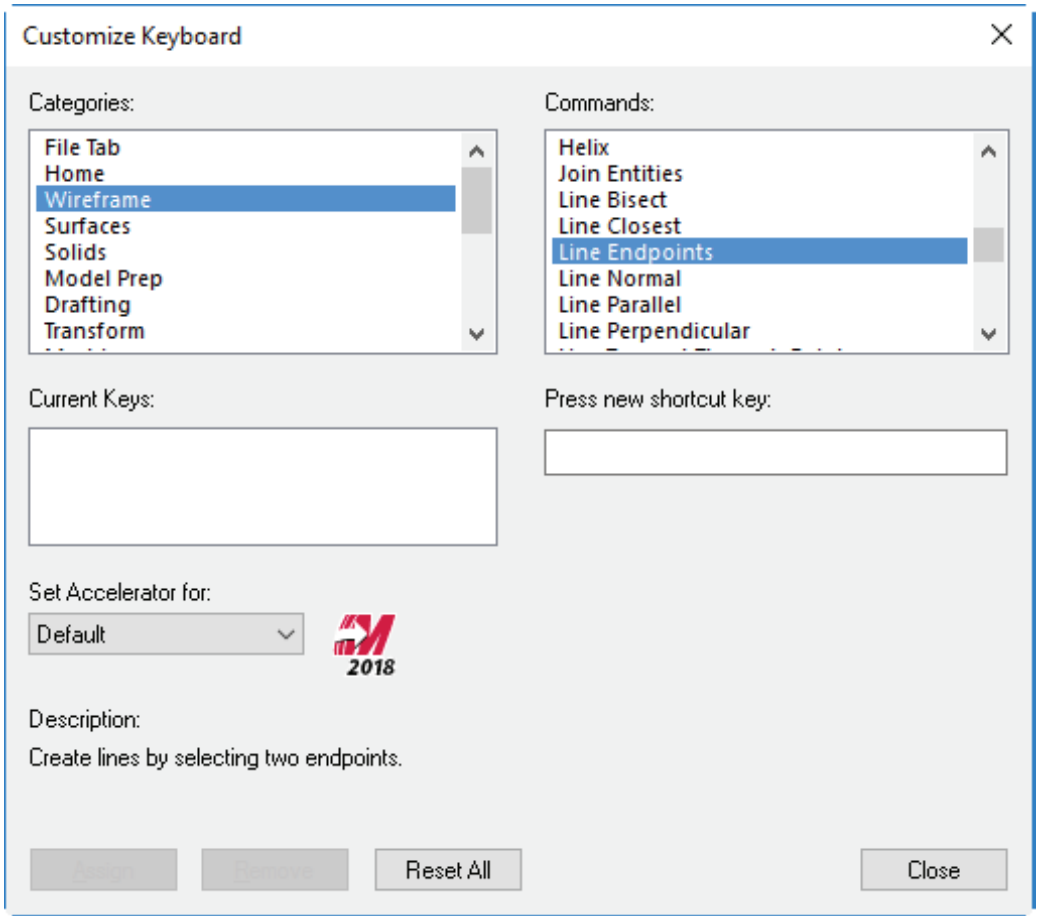
FILE

- **Options.**
- Select **Customize** as shown.

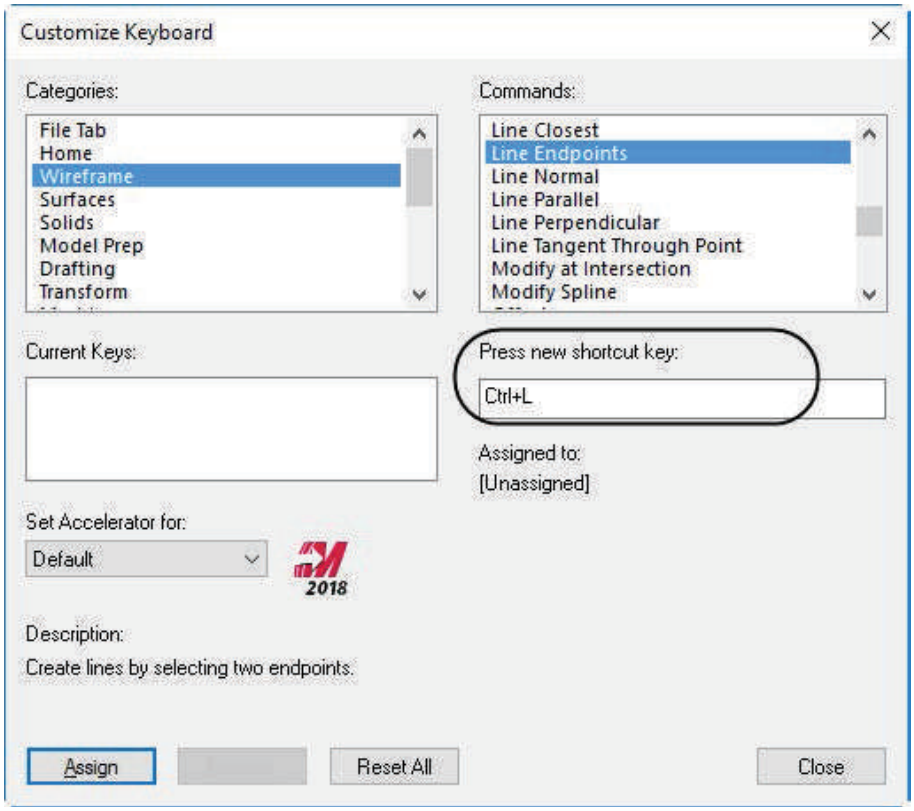


- Select **Wireframe** as the **Category**.

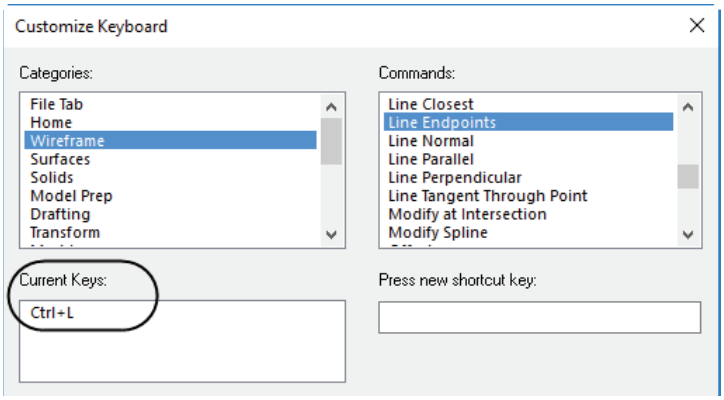
- Select the command **Line Endpoints.**



- Press **Ctrl** + **L** (in this sequence) in the **Press new shortcut key** area as shown.



- Click on the **Assign** button.
- The new key will be displayed in the **Current Keys** field.



- Select the **OK** button to exit the **Customize Keyboard** dialog box.
- Select the **OK** button to exit the **Options** dialog box.
- Press **Ctrl** + **L** on your keyboard and the **Line Endpoints** panel will be automatically opened.
- To remove a shortcut assignment, select the command associated with the shortcut name you want to delete from the **Customize Keyboard** dialog box.
- In the **Current Keys** field, select the shortcut and select **Remove**.

The shortcut is removed from its assignment to the selected command and can now be assigned to another command.

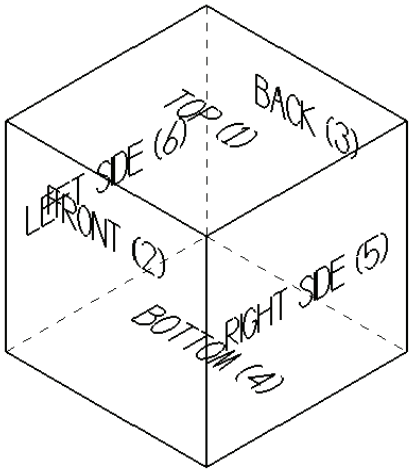
CREATING 3D WIREFRAME

ABOUT VIEWS

The main tool for applying coordinate systems in Mastercam is the view. A view consists of two main parts:
A plane or slice through the coordinate system.
An origin, or zero point.

Graphics View
The **Gview** is the plane from which you view your work in the graphics window.
There are 7 standard graphics views in Mastercam as shown.

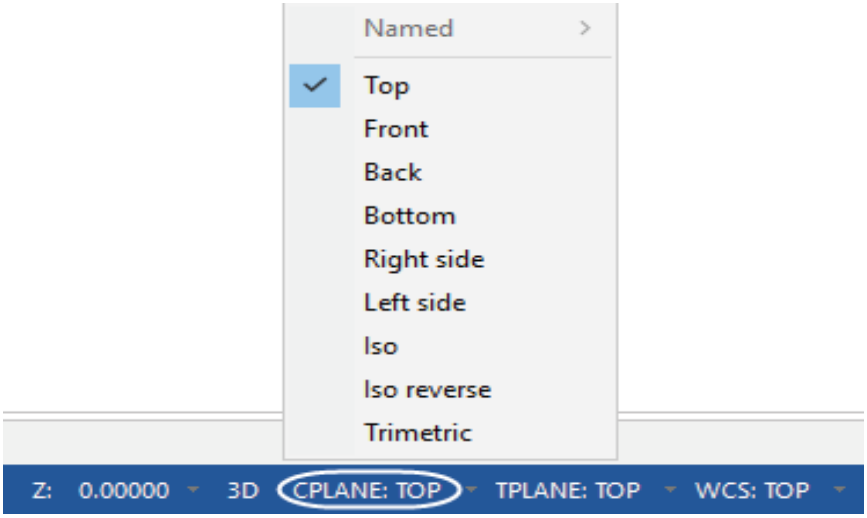
View 1 = Top.
View 2 = Front.
View 3 = Back.
View 4 = Bottom.
View 5 = Right Side.
View 6 = Left Side.
View 7 = Isometric.
View 8 = Isometric Reverse.
View 9 = Trimetric.



Construction Plane (Cplane)

The **Cplane** is a flat, two dimensional space on which you create geometry and that can be defined anywhere in 3D space. There are 7 predefined construction planes in Mastercam, the same as for **Gview**.

- To set the desired **Cplane**, choose **CPLANE** from the **Status Bar** and select the desired **Construction plane** from the list.



NOTE: Mastercam 2018 default construction mode is **3D**. This plane allows you to draw in unrestricted 3D space. You can use the 3D plane while creating geometry that can be related with existing geometry.

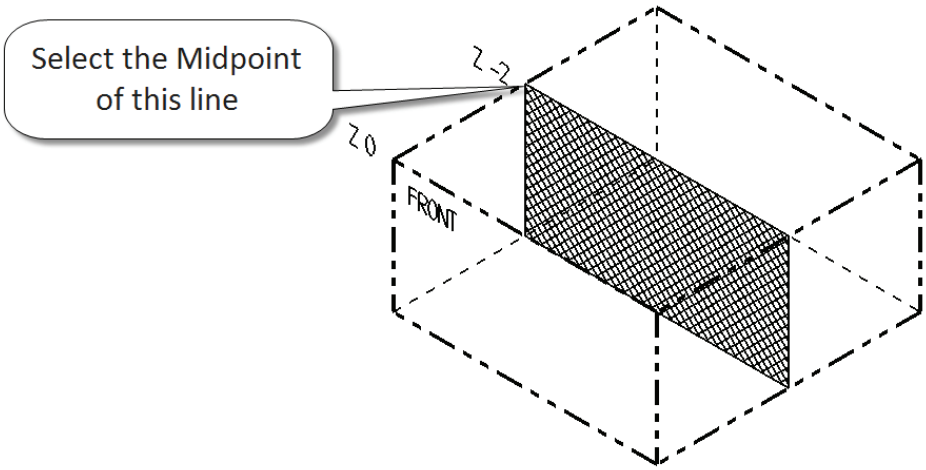
ABOUT CONSTRUCTION DEPTH (Z DEPTH)

The Z depth corresponds to a point on the Z axis of the correct construction plane (Cplane) with respect to the system origin (X0, Y0, Z0).

How to set the Z depth

- First, set up the current **plane** and then set the **Z** depth at which you want to create or modify the geometry.
- To set the Z depth, select **Z** from the **Status bar**.

- Select a point from the existing geometry as shown.



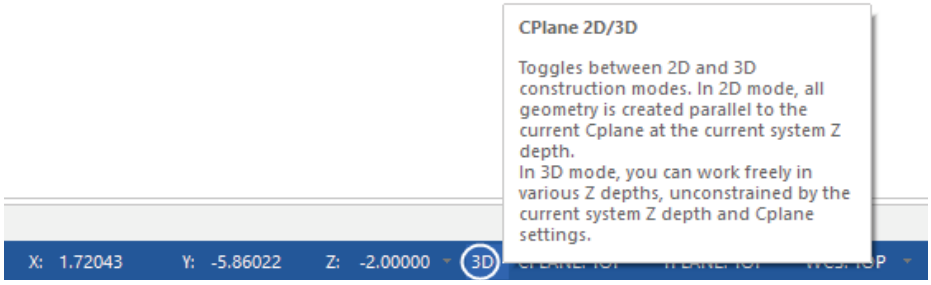
OR

- Enter the value in the **Z** field as shown.

- To set the **3D** construction plane, choose the **2D/3D** toggle button from the **Status bar**.

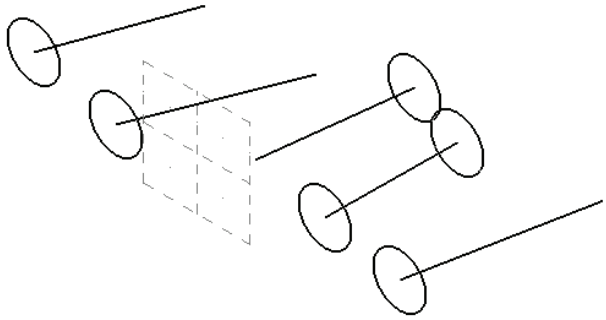
*Example using the **2D** construction plane to create the geometry shown below*

- Set the **2D** construction plane by choosing the **2D/3D** toggle button as shown.



- Select the **Front Construction plane**.

- Create the circles by selecting the endpoints of the lines as the center points as shown.



NOTE: Using the **2D** construction plane as shown in this case **does not require multiple changes of the construction depth.**

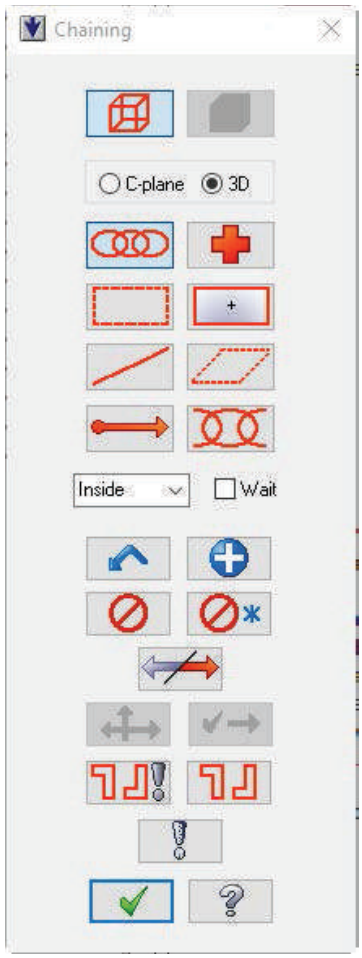
CHAINING

Chaining is a process used to select a number of entities (lines, arcs, points, splines, etc.) when modifying or transforming existing entities or when creating a toolpath. The entities form a chain when the distance between two consecutive entity endpoints is less than the chaining tolerance (0.002 in the system configuration).

When using chain for deleting entities you have to preselect the geometry. Hold down the Shift Key and select one entity of the chain. Mastercam automatically selects all the entities that could form the chain.

*In **Wireframe** mode, the **Chaining** dialog box provides chain selection methods including:*

- **C-Plane** - allowing you to select geometry in the current construction plane.
- **3D** - allows you to select 3D geometry.



Chain - allows to select entities that form a chain.

Point - allows to select a point that can be used as starting point.

Window - allows to select entities by drawing a rectangle around them.

Area - allows to selects closed boundaries around a point where you click the mouse.

Single - allows you to select one entity.

Polygon - allows you to select entities inside of a multi sided irregular window that you draw around the entities.

Vector - creates chains from entities that intersect a vector line.

Partial - creates an open chain by selecting the first entity and the last entity of the chain.

Other controls let you manage the selection process.

Last - reselects entities chained during the previous operation.

End Chain - forces an end of the chain.

Unselect Chain.

Unselect All Chains.

Reverse chaining direction - swaps the chain direction.

Adjust - guide the chaining when a branch point has been reached and allows you to select a different direction or entity.

Next - guide the chaining when a branch point has been reached and allows you to continue in the direction of the red arrow.

Chain Feature Option - allows you to set the way Mastercam can recognize related chains.

Chain Features - instructs Mastercam to add chains based on criteria set in the Options.

Options - lets you refine the way Mastercam chains entities.

HOW TO CHAIN

When **Chaining Geometry** to create a toolpath, it is important to chain the geometry in the proper direction as it will determine the direction the cutter travels when machining the part. Below is an example on how to select geometry using the chaining method.

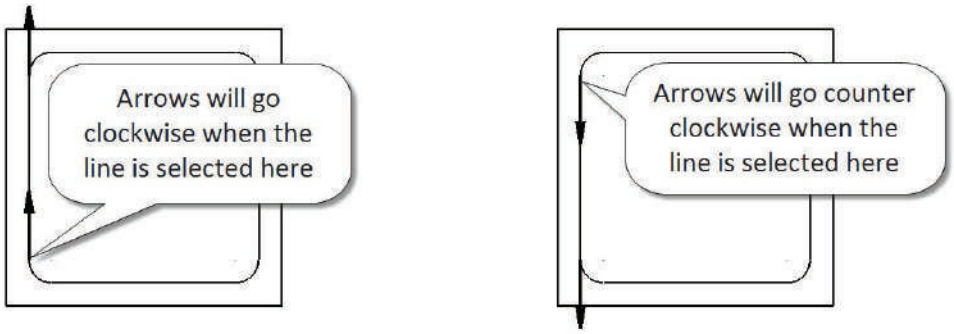
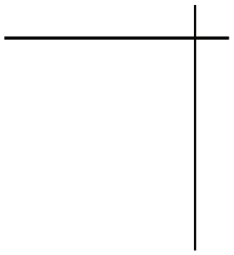
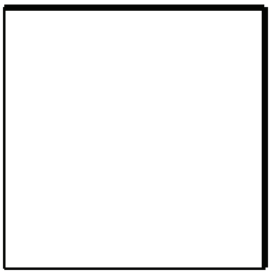
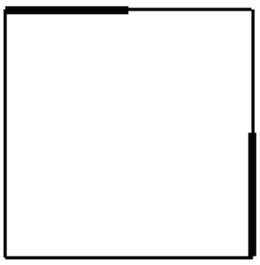


Table 1: Chaining Problem Diagnostic

Description	Check	Fix
Chain stops: Lines do not touch.	Zoom into inspect.	Trim lines together using Trim 1 entity.

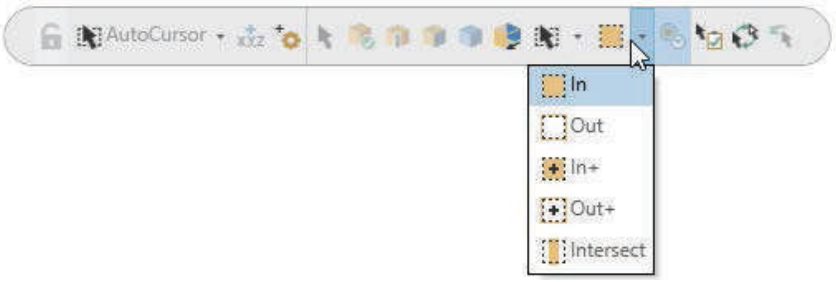
Table 1: Chaining Problem Diagnostic

Description	Check	Fix
Chain stops: Lines cross. 	Zoom into inspect.	Trim lines using Trim 2 entities.
Lines do not touch.	Look at geometry in ISO Gview.	Use TRANSFORM/ Project.
Duplicate entities: Identical entities on top of each other. 	Zoom into the area and you will notice a thicker line.	Delete Duplicate entities.
Overlapping entities: Entities of different lengths on top of each other. 	Zoom the area, use Analyze menu and move the cursor on the top of the entities.	Use Findoverlap.dll C-hook.

WINDOW SELECTION

Window is a process used to select a number of entities (lines, arcs, points, splines etc.). Window selection is used when modifying or transforming existing entities or creating a toolpath.

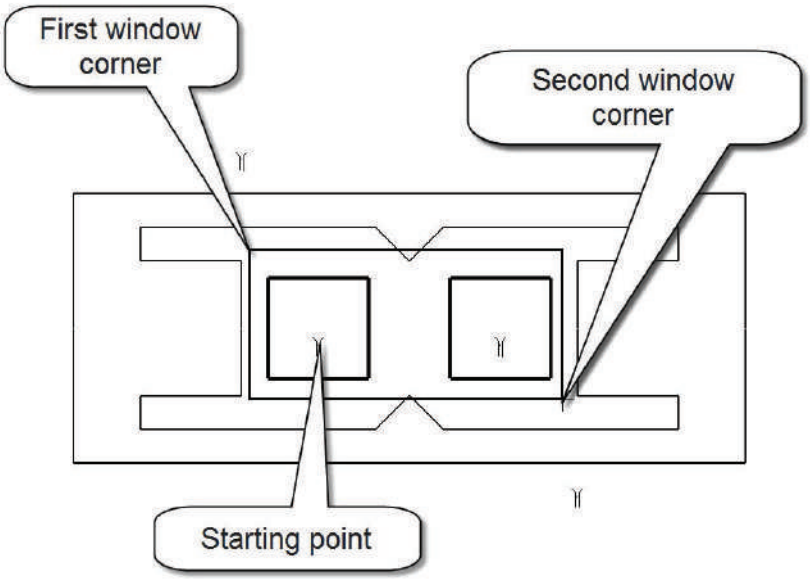
From the **General Selection** toolbar, select the drop down arrow to choose how the entities will be selected.



- **In** - allows you to select the entities that are completely inside the window.
- **Out** - allows you to select the entities that are completed outside the window.
- **In +** - allows you to select the entities that are completed inside and intersect the window.
- **Out +** - allows you to select the entities that are completed outside and intersecting the window.
- **Intersect** - allows you to select only the entities that intersect the window.

HOW TO SELECT GEOMETRY INSIDE OF A WINDOW

When using windows to select geometry in order to create a toolpath it is important to establish the proper direction the cutter travels when machining the part. This can be set in the chaining Options.



CHAINING AND WINDOW OPTIONS

Both **Chaining** and **Window** selections have an Options dialog box used to set default values for a number of choices that would affect how you select the geometry.

- To change options for the selection process you have to choose the **Options** button which is found in the **Chaining** dialog box.

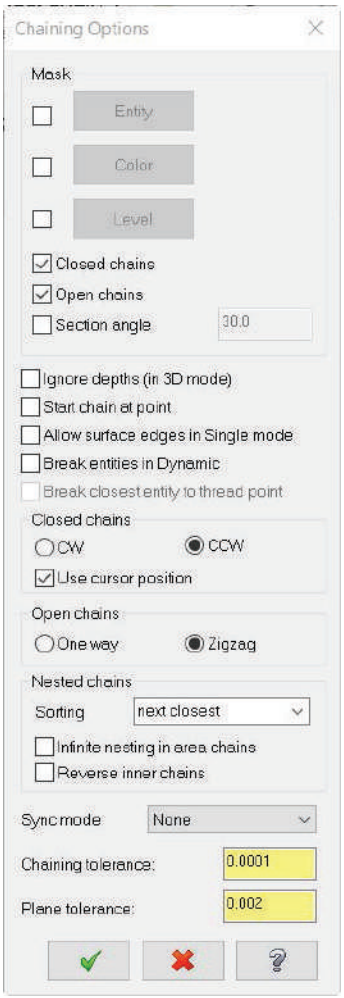
CHAINING OPTIONS

The **Chain Mask** options speed up chaining for a large number of entities or when branch points are involved. They let you "pre-select" categories of entities to build your chain from. Mastercam provides Entity, Colour and Level. All the mask does is block all the entities that do not match the mask criteria.

- **Closed chains** lets you select all entities adjacent to the selected entity in the forward and reverse directions until it reaches the end of a chain or a branch point.
- **Open chains** allows you to select a section of the first entity and the last entity of a chain. Mastercam creates an open chain which includes all adjacent entities between these two points.
- **Section angle** applies only to **Section** chaining. In section chaining Mastercam chains along a set of smooth curves. Curves that are less then the section stop angle are considered tangent for this chaining method. When Mastercam encounters an angle that exceeds the section stop angle or a branch point it ends the chain.
- **Ignore depths** (in 3D mode) lets you chain entities that could form a chain (the distance between two consecutive entity endpoints is smaller than the chain tolerance) but have different depths.
- **Start chain at point** will start the chaining from a point that you create on an endpoint of an entity in the chain.
- **Allow surface edges in Single mode** allows you to select, in single mode the edges of a surface.
- **Break entities in Dynamic** sets the system to automatically break the entities at intersection while in **Dynamic** mode.

Nested Chains

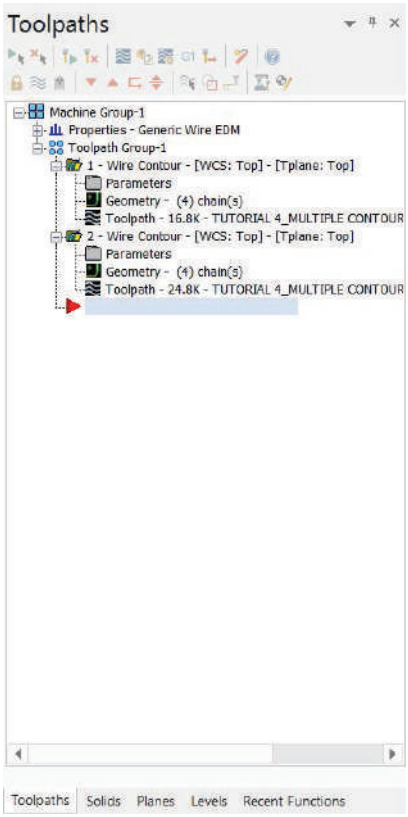
- **Sorting** lets you choose different sorting options while selecting geometry that is completely contained within a closed boundary. You can speed up chaining using **next closest**. Choosing **inside to outside** sorts and chains the curves from the innermost to the outermost. You can sort the chains starting with the outermost chains using **outside to inside**. It also allows you to sort and optimize the chaining using **optimize inside to outside** or **optimize outside to inside**.



- **Infinite nesting in area chains** one mouse click chains all of the curves and points within a closed boundary. When unchecked, Mastercam chains the outer and inner boundary closest to the point you entered to define the area chain and any curves or point between the two.
- **Reverse inner chains** changes the direction of chaining for the inner chains.
- **Sync mode** allows Mastercam to break a chain into a number of separate chains for the purpose of creating the surface. The Sync mode option of the Chaining Options dialog box determines how Mastercam should divide up the chains. You may need to choose a sync mode if you have attempted to create a surface and found that it twists.
- **Chaining tolerance** sets the default chaining tolerance value.
- **Plane tolerance** sets the default plane tolerance value.

TOOLPATHS MANAGER

The **Toolpaths Manager** lists all operations in the current job. Use this dialog box to sort, edit, regenerate, and post any operations. You can enter this dialog box by selecting the Toolpaths tab.



Select all operations

- This button selects all operations listed in the **Toolpath Manager**. Selected operations are represented by a check mark on the folder.

Select all dirty operations

- This button selects all dirty operations listed in the **Toolpaths Manager**. Dirty operations are operations in which changes were made (parameters or geometry changes). A red "X" mark will be displayed on the toolpath icon.

Regenerate all selected operations

This button will recreate all the operations after changes have been made.

Regenerate all dirty operations

This button will recreate all the dirty operations after changes have been made.

Backplot selected operations

Simulates tool movement for the selected operations.

Verify selected operations

Simulates machining for the selected operations.

Backplot/Verify Options

Sets parameters for backplot and/or Verify. You can set the Stock shape to a box, a cylinder, a solid, a stock model operation, or to a file.

Post selected operations

Runs the post processor for selected operations and creates the NC program.

High Feed

Optimizes feed rates for selected operations.

Delete all operations, groups and tools

Deletes all machine groups in the toolpath manager except for the last one in the list. This deletion includes operations, tools and machine definitions and you will not be able to undo this deletion.

Help

Opens Mastercam help.

Toggle locking on selected operations

Locks and unlocks selected operations. Locking the operations prevents you from regenerating the toolpath.

Toggle toolpath display on selected operations

Displays and hides selected operations in the graphics window.

Toggle posting on selected operations

Enables and disables posting on selected operations.

Move insert the arrow

Position the insert arrow where the new operation will be inserted.

Only display selected toolpaths

When enabled, displays toolpaths in the graphics window only for the operations you select in the Toolpath Manager list.

Only display associative geometry

When enabled, displays all the geometry associated with the selected operations, provided the geometry is not hidden or blanked and is on levels that are visible.

Hide rapid retract move

Hides and shows all rapid retract moves in the current file.

Tool setup

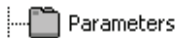
Opens the Tool Setup Manager dialog box.

Edit reference positions

- Opens the Machine Reference Position Manager dialog box. Use to edit the location of the reference positions.

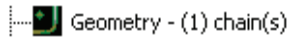
OPERATION LIST AREA

This area displays the order of operations in the current job. Select the + or - signs in front of the file folder icon for an operation to **display/collapse** the pieces of that operation, including **parameters, geometry** and the **NCI file.**



Parameters.

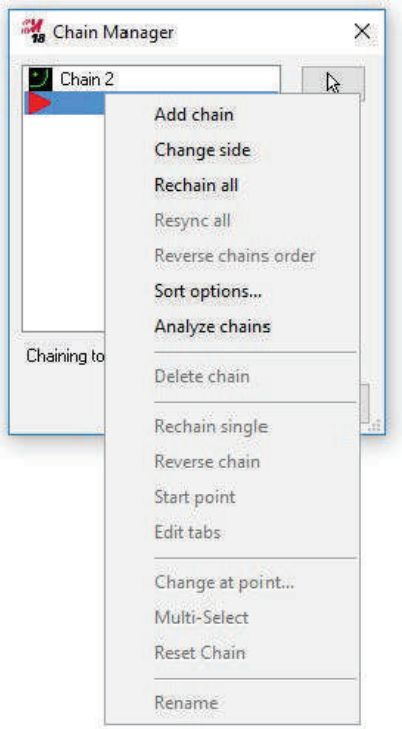
- Select the **Parameters** icon to edit the toolpath parameters.

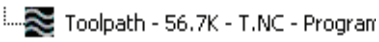


Geometry.

- Select the **Geometry** icon to edit the chains and points to be machined in that operation.
- Right mouse click in the list area and you can change the following.

- **Add chain.**
- **Change side.**
- **Rechain all.**
- **Resync all.**
- **Reverse chains order.**
- **Sort options.**
- **Analyze chain.**
- **Delete chain.**
- **Rechain single.**
- **Reverse chain.**
- **Start point.**
- **Edit tabs.**
- **Change at point.**
- **Multi-Select.**
- **Reset Chain.**
- **Rename.**





Toolpath.

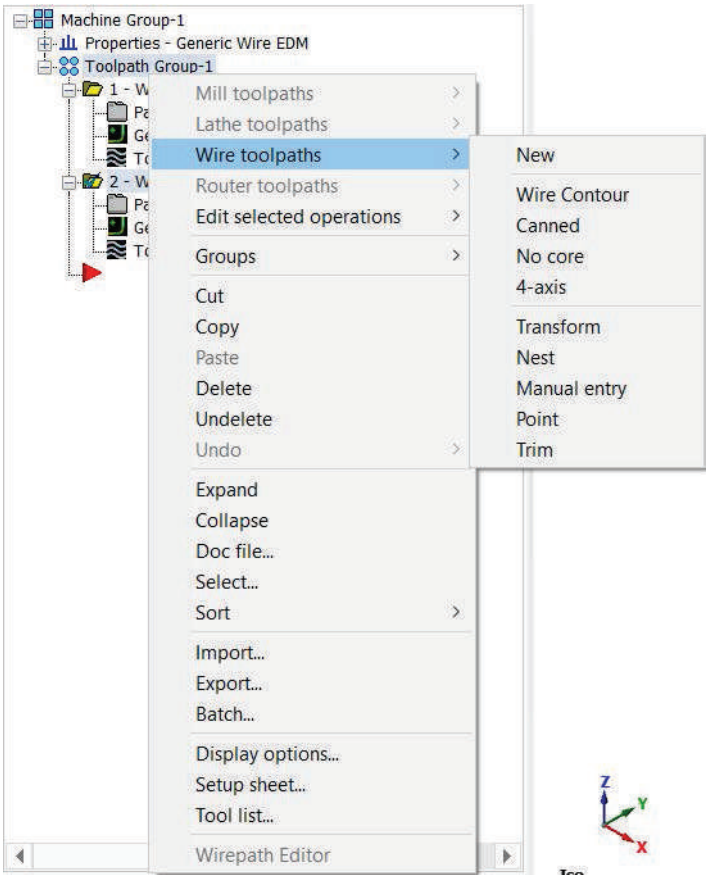
- **Toolpath** contains the **NCI file**, a separate intermediate file that contains all the information on the toolpath data. The post-processor uses the NCI file to create a NC file for your specific machine/control. Select the **toolpath** icon and the system will open the **Backplot** command.

ABOUT RIGHT MOUSE CLICK IN THE TOOLPATHS MANAGER LIST AREA

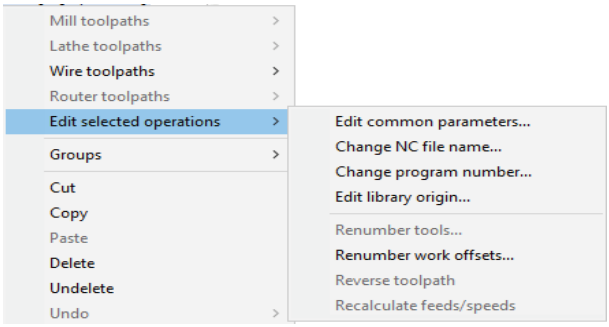
Right mouse click in the Toolpaths Manager list area opens an extensive menu of commands.

- **Wire toolpaths** displays a list with all operations available in Mastercam. You can start to create an operation from here. For example by selecting **Toolpaths**, **Contour** the system will prompt you, to chain the geometry to be machined.

- **Wire toolpaths, New** allows you to delete all operations and tools that were already created.



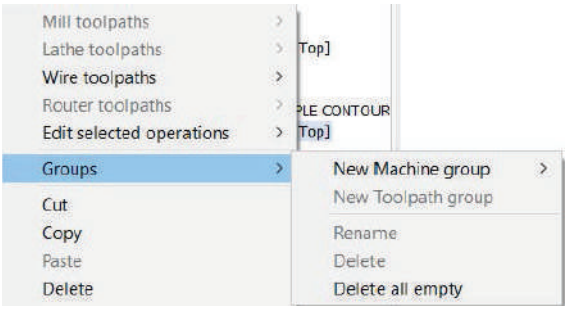
Edit Selected operations



- **Edit common parameters** - allows you to edit the common parameters of all the operations inside of a toolpath group. For example you can change the Feed plane value for all the operations inside the group. Make sure that you have previously selected all operations.
- **Change NC file name** - allows you to rename the selected operations with a different NCI file name.
- **Change program number** - allows you to renumber the NC program.
- **Renumber tools** - allows you to renumber the tools by giving the first tool number and the increment number.
- **Renumber work offsets** - allows you to renumber the work offsets when using more than one offset (tombstone application).
- **Reverse toolpath** - allows you to reverse the chaining direction to change the cutting direction.
- **Recalculate feeds/speeds** - allows you to adjust the feed rate, plunge rate, retract rate, and spindle speed for multiple toolpaths based on a new stock material. This procedure requires that the feed calculation in **Tool Settings** be set to **Material**.

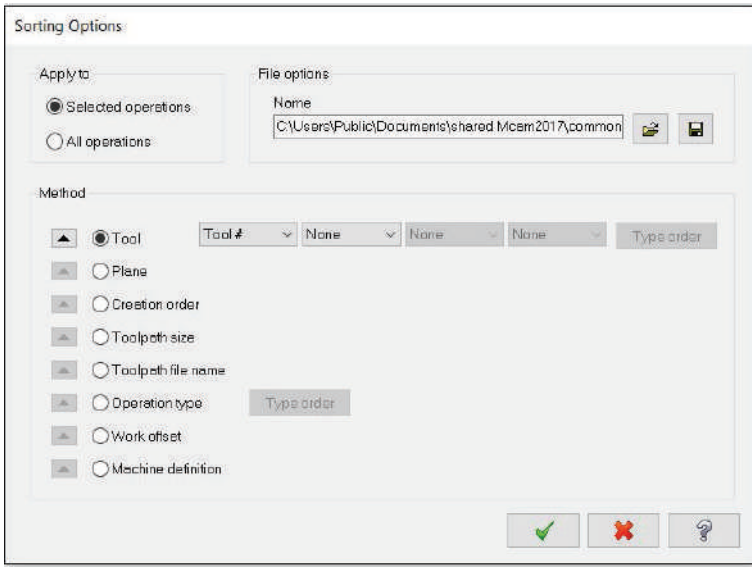
Groups

- **Groups** - allows you to organize operations. Operation groups are handy for quick selection of toolpaths. Selecting a group automatically selects all operations within the group.
- **New Machine group** - instructs the system to add a new machine group.
- **New Toolpath group** - instructs the system to add a new toolpath group.
- **Rename** - allows you to change the group name.
- **Delete** - instructs the system to delete a selected group.
- **Delete all empty** - instructs the system to delete all groups that have no operations.
- **Cut, Copy/Paste** - allows you to reorder the operations.
- **Delete/Undelete** - allows you to delete selected operations or recover them using undelete.



- **Expand** - expands all the operations and the groups inside of the **Toolpath Manager**.
- **Collapse** - collapses all the operations and groups inside of the **Toolpath Manager**.
- **Doc file** - creates a text file that list all of the groups, operations, tools, geometry and NCI destinations that are in the **Toolpath Manager**.

- **Select** - opens a dialog box that allows you to choose different options to select the operations.
- **Sort** - allows you to reorganize operations.
- **Import** - allows you to import saved operations from a library.
- **Export** - allows you to export operations to a library.
- **Batch** - lets you generate the toolpath at a later time and post to multiple NCI destinations, setting time limits on running the batch operations.
- **Display options** - sets the **Toolpath Manager** display options.
- **Setup Sheet** - provides information about the work piece, including the operations, tool reference, total machining time and comments added during programming in a printable format.



- **Collision check** - allows you to check for collision and gouges in the part. A collision occurs when the tool contacts material during a rapid move. A gouge occurs when the tool removes more material than desired.
- **Toolpath Editor** - allows you to modify an existing toolpath. You can:

  Edit tool motion at specific point positions in the toolpath.

  Add points to increase tool motion.

  Move points to adjust tool motion.

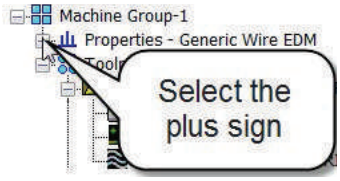
  Delete selected points, passes, cuts and sections of a toolpath.

MACHINE GROUP PROPERTIES

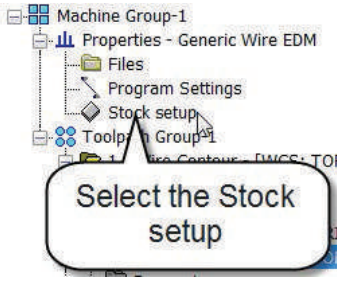
Machine Group Properties sets appropriate defaults for the jobs you program such as operations, tools, views, stock, toolpath configuration questions, feed calculations, and material settings. It is a good place to start before programming a job.

NOTE: The stock settings are used for the toolpath verification.

- Select the plus sign in front of **Properties** to expand the **Toolpath Group Properties**.

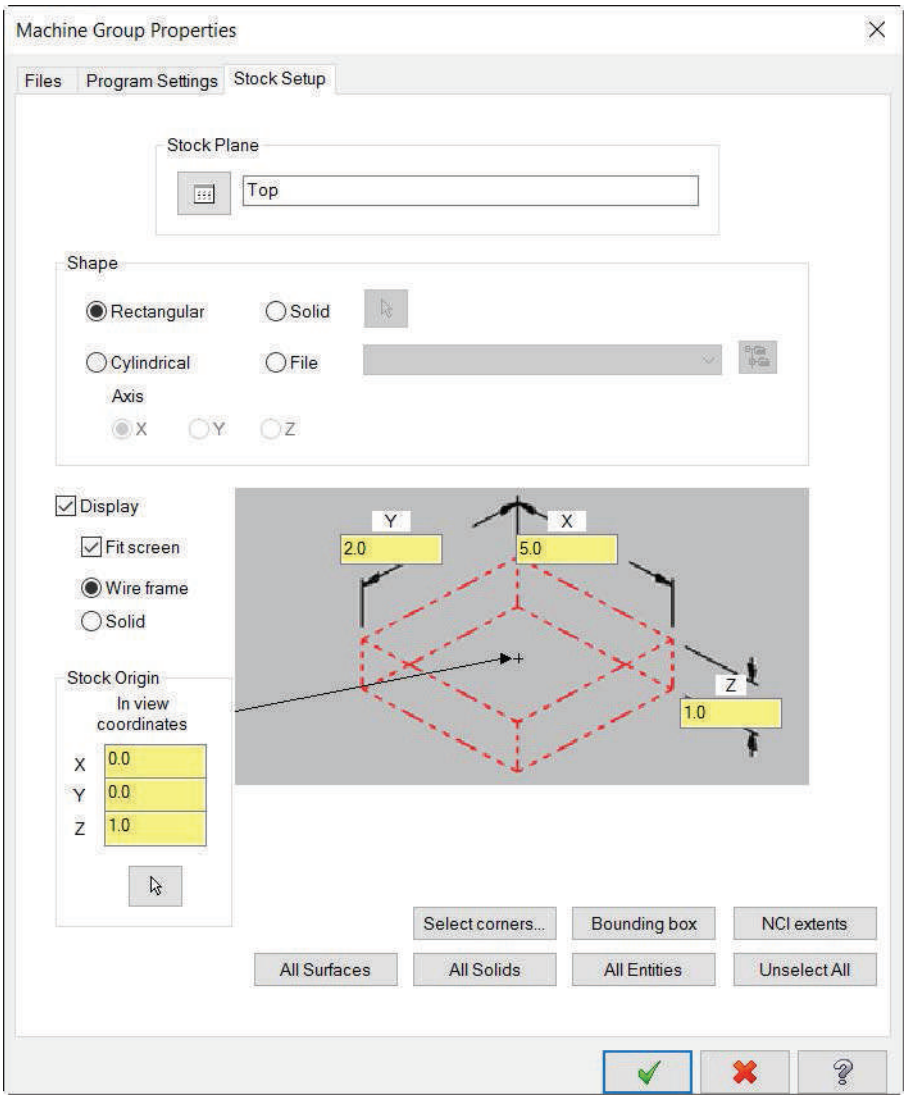


- Select **Stock setup**.



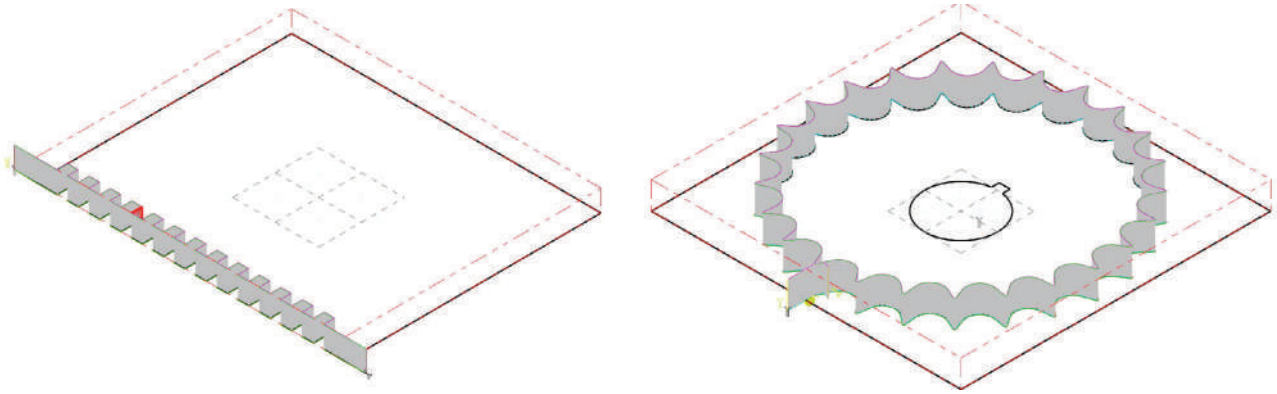
STOCK SETUP

- **Shape** - allows you to choose between:
- **Rectangular.**
- **Cylindrical**
- **Solid.**
- **File** - allows you to import an STL file as the stock.
- **Display** - allows you to set the stock as **wireframe** or **solid** and to fit the stock to the screen.
- **Stock Origin** - values adjust the position of the stock, ensuring that you have an amount of extra stock around the finished part.
- **Select Corners** - allows you to select opposite corners of the stock from existing geometry.
- **Bounding Box** - defines the stock limits by finding the extents of the selected geometry.
- **NCI extents** - establishes the stock size based on the tool movements in the NCI file.
- **All Surfaces/All solids/All entities** - defines the stock based on all visible surfaces or solids or all entities in the file.



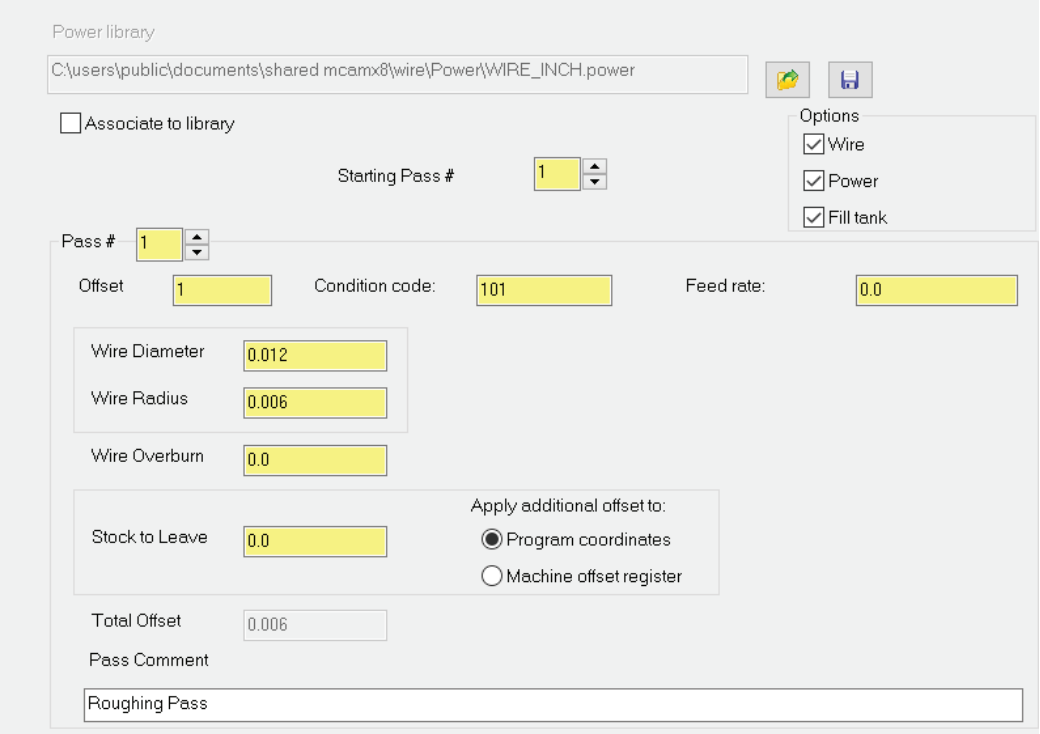
CONTOUR WIREPATH

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.



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 deselecting the **Associate to library** check box.



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 for condition 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.

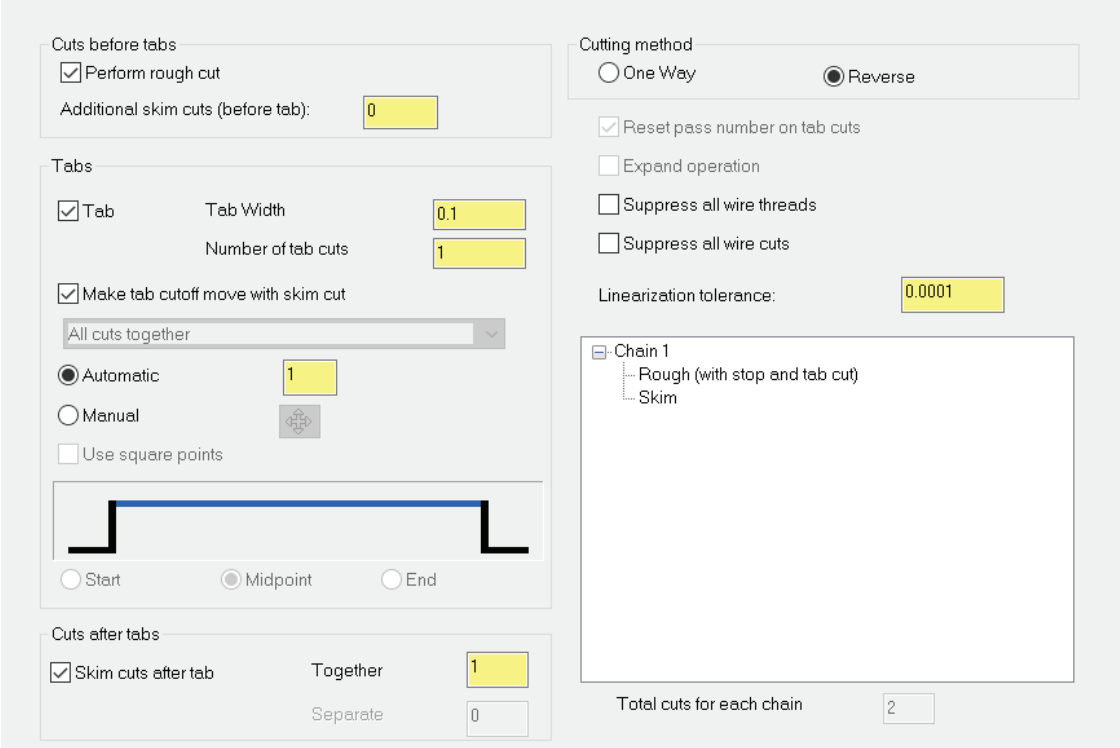
Wire diameter/radius sets the wire diameter/radius value and is automatically updated when one of the values is entered.

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.

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.



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

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

Tab is the uncut material that secures the part to the material; can be cutoff with the skim cut pass or in a separate pass.

Skim cuts after tab allows you to enter the number of finish passes after cutting the tab.

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

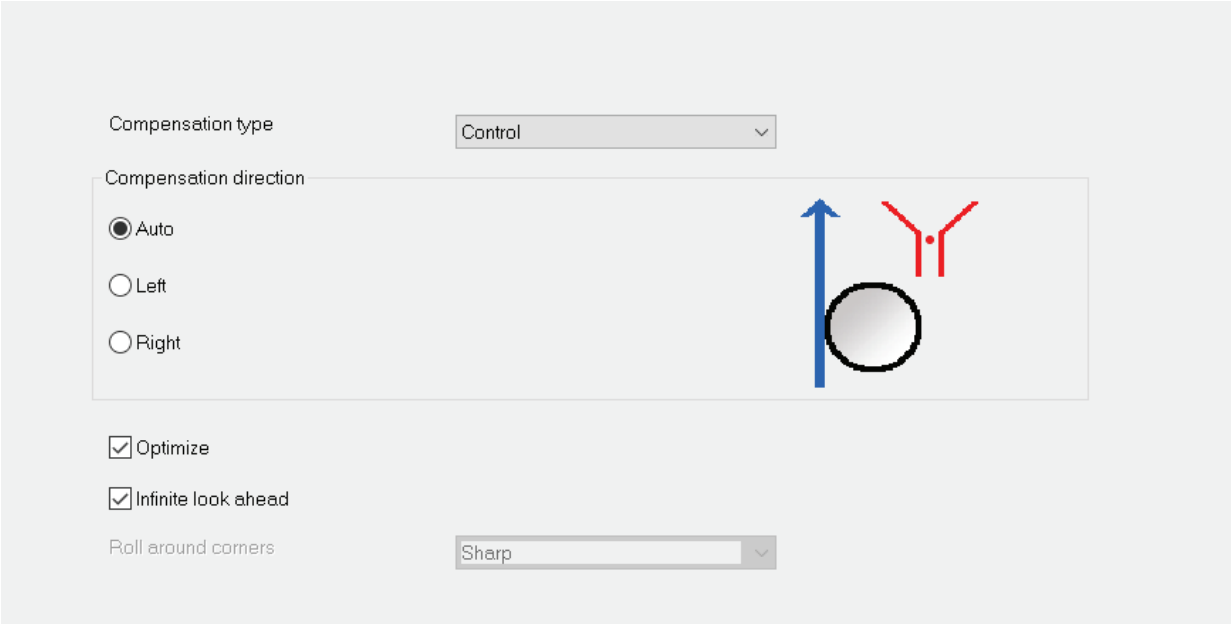
Reset Pass Number on Tab Cuts is selected to use the power settings stored as Pass 1 in the wire power settings library for the first tab cut. Mastercam Wire uses the Pass 1 settings for the rough cut.

Suppress all wire threads enabled will output no thread flags to the wirepath's NCI file.

Suppress all wire cuts enabled will output no cut flags will to the wirepath's NCI file.

Compensation parameters

The **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.



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.

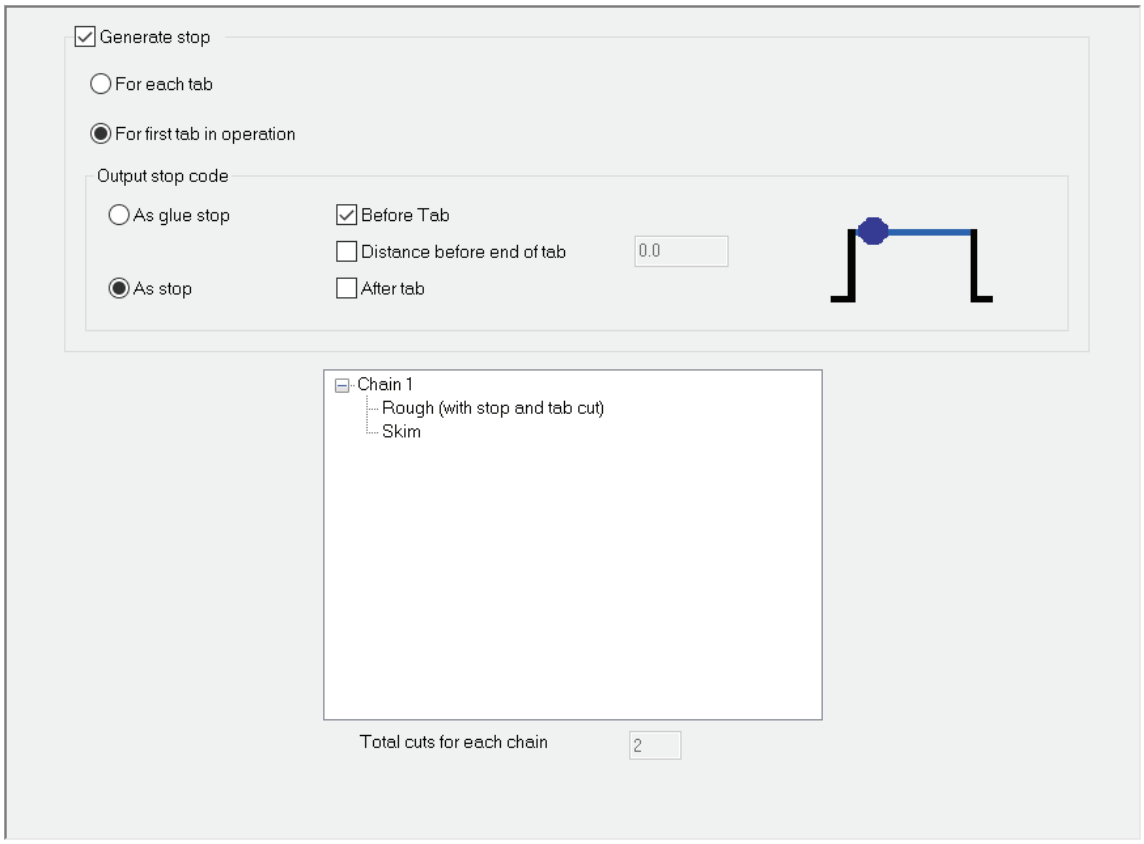
Optimize eliminates arcs in the wirepath that are less than or equal to the radius of the wire.

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 Cutter 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**.

Stops

The **Stops** page allows you to create optional stops (M01) or program stops (M00) in a wirepath. For contour wirepaths, you have the options to apply the stops before and/or after tabs.



For each tab outputs a stop code before the tab cut for the first tab cut on each chain of the operation.

For first tab in operation outputs a stop code before the tab cut for the first tab cut on Chain 1 of the operation. Subsequent tab cuts throughout the operation do not include stop codes.

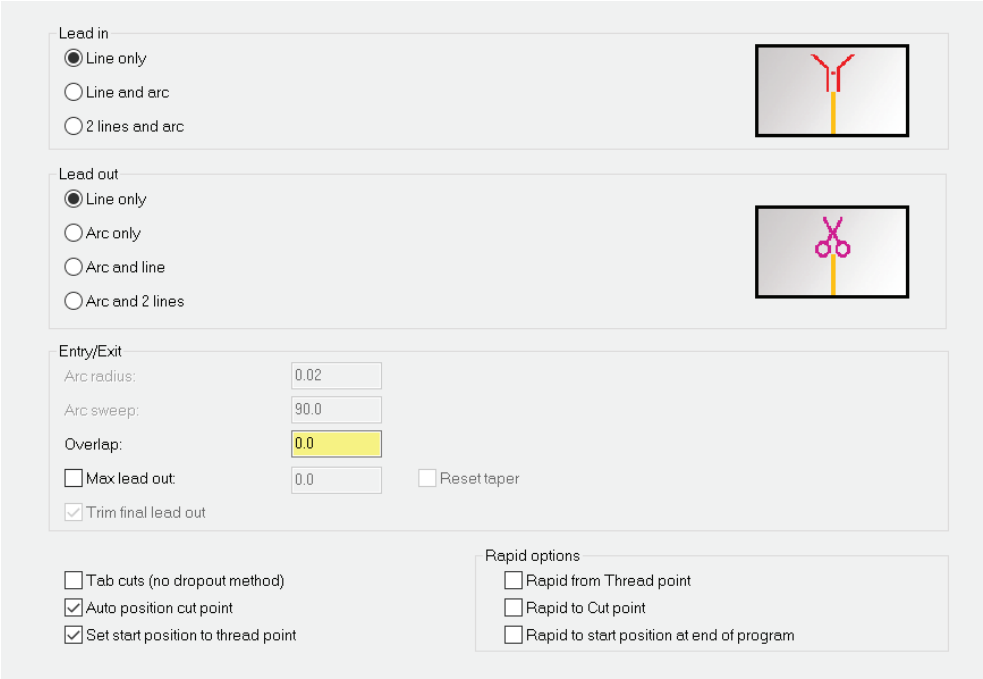
As stop outputs a program stop code (M00). Pauses the wire machine until the operator restarts it.

Before Tab and **After Tab** places the glue stop or the program stop (whichever stop code you select) either before or after the selected tab.

Distance before end of tab places the glue stop or the program stop (whichever stop code you select) at a distance from the end of the tab. Enter the distance in the field to the right of the option. This lets you cut partially through the tab, stop, secure the slug, and continue to complete the cutoff.

Leads

The **Leads** page allows you to set the wire motion when entering or exiting a part.



Lead in sets the shape of the part entry motion. The lead in extends from the thread point to the start of contour.

Line only creates a line between the thread point and start of contour.

Line and arc adds a line and arc between the thread point and start of contour. Enter values for the **Arc radius** and **Arc sweep**.

Lead out sets the shape of the part exit motion. Lead out extends from the end of contour to the cut point. You have the same options as in the Lead in plus the following extra options.

Arc only in which the wire arcs off the part. Enter values for the **Arc radius** and **Arc sweep**. A lead out radius uses the same **Arc radius** and **Arc sweep** values as the lead in.

Arc and two lines the wire arcs off the part, then creates two line moves to the cut point. Enter values for the **Arc radius** and **Arc sweep**. A lead out radius uses the same **Arc radius** and **Arc sweep** values as the lead in.

Overlap eliminates potential burrs by overlapping the start and end of the contour by the amount entered.

Max lead out allows you to shorten the lead out by entering a new value for the lead out.

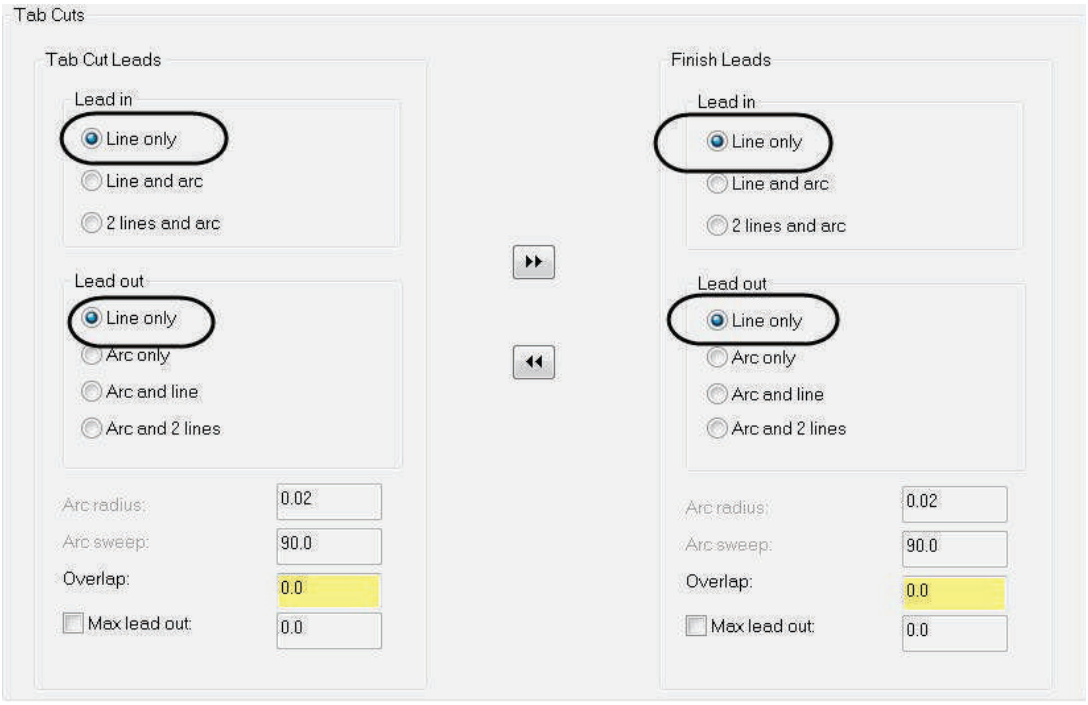
Tab Cuts (No dropout method) eliminates slugs or slivers of material that result from the tab cut.

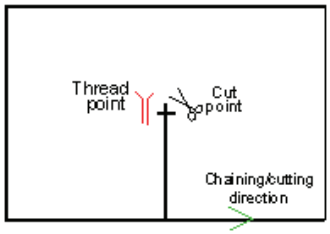
Auto positions cut point determines the most efficient cut point for a contour. Usually remains enabled.

Set start position the thread point moves the job start point to the thread point of the first contour in the operation. Disable to insert a rapid move from the start point to the thread point.

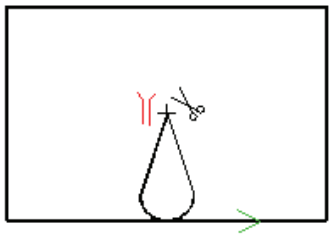
Tab/Finish Leads

The **Tab/Finish Leads** page allows you to set the wire motion when entering or exiting a part when making tab cuts (**Tab Cut Leads** section) and skim cuts after a tab cut (**Finish Leads** section).

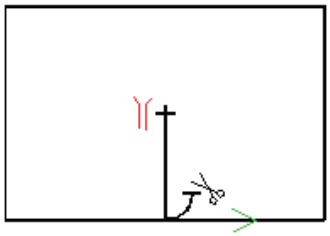




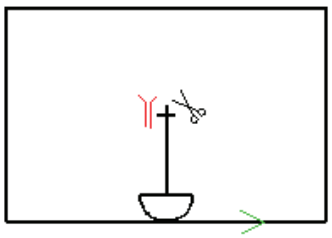
Line only lead in
and lead out



Line and radius lead in
Radius and line lead out



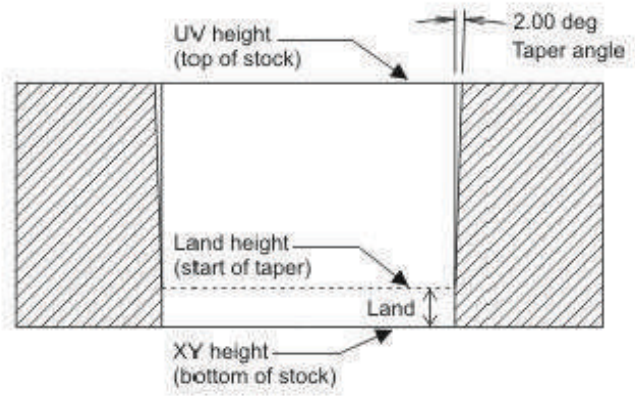
Line only lead in
Radius only lead out



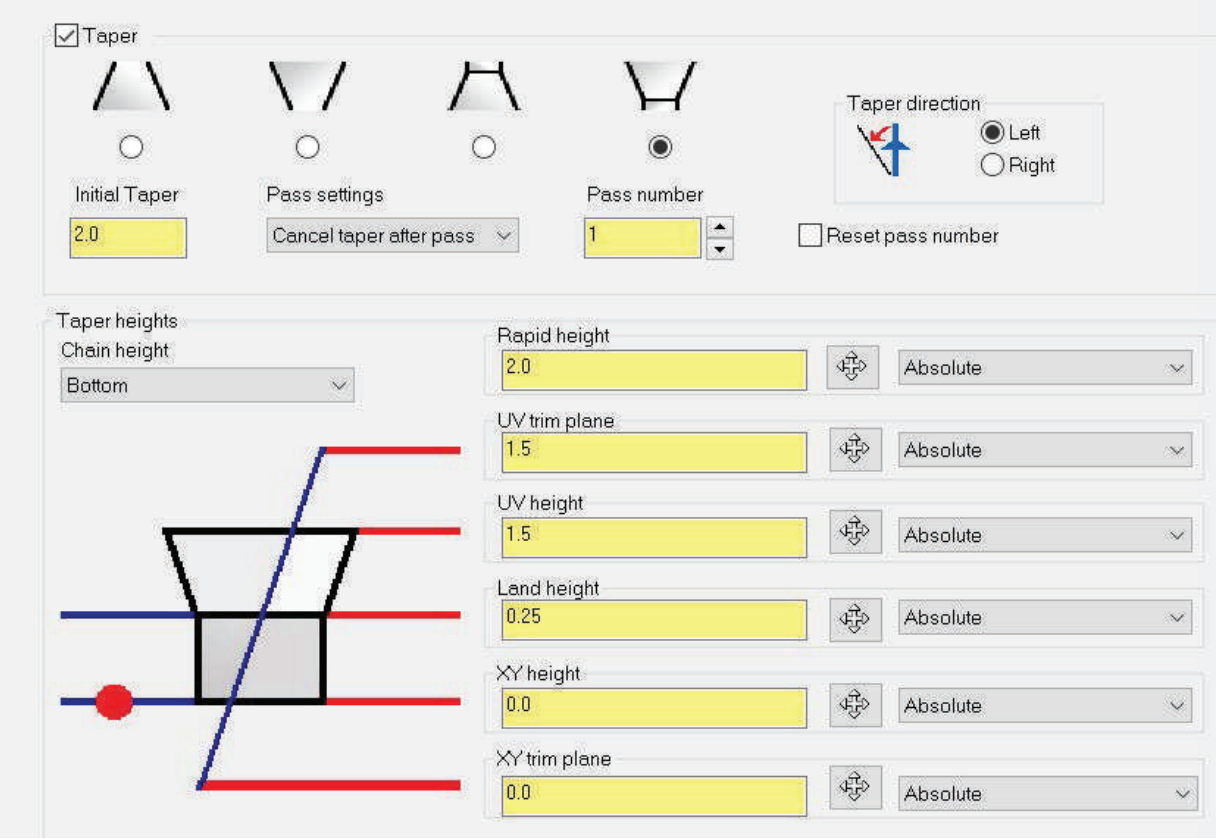
2 lines and radius lead in
Radius and 2 lines lead out

Taper

The **Taper** page allows you to set the parameters to taper the contour wirepath inward or outward and to specify the location of the land (the point at which the taper begins). You can set the Z height of the upper wire guide for rapid moves, the location of the upper wire guide and lower wire guide, the height of the part and the location of the XY plane as shown in the figure below.



Side view of part



Chain height options allows you to indicate where the geometry was chained (XY-**Bottom** or UV-**Top** height or, if there is a land, at the land height-**Middle**).

Rapid height sets the Z depth of a rapid move. Can be higher than the **UV height**, **UV trim**, or **UV extension planes** in order to clear any clamps.

UV trim plane can be used to set the location of the upper guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to G code 1015 in the NC file. It is usually set directly at the controller.

UV height sets the location of the XY plane that contains the geometry for the upper contour of a 4-axis part.

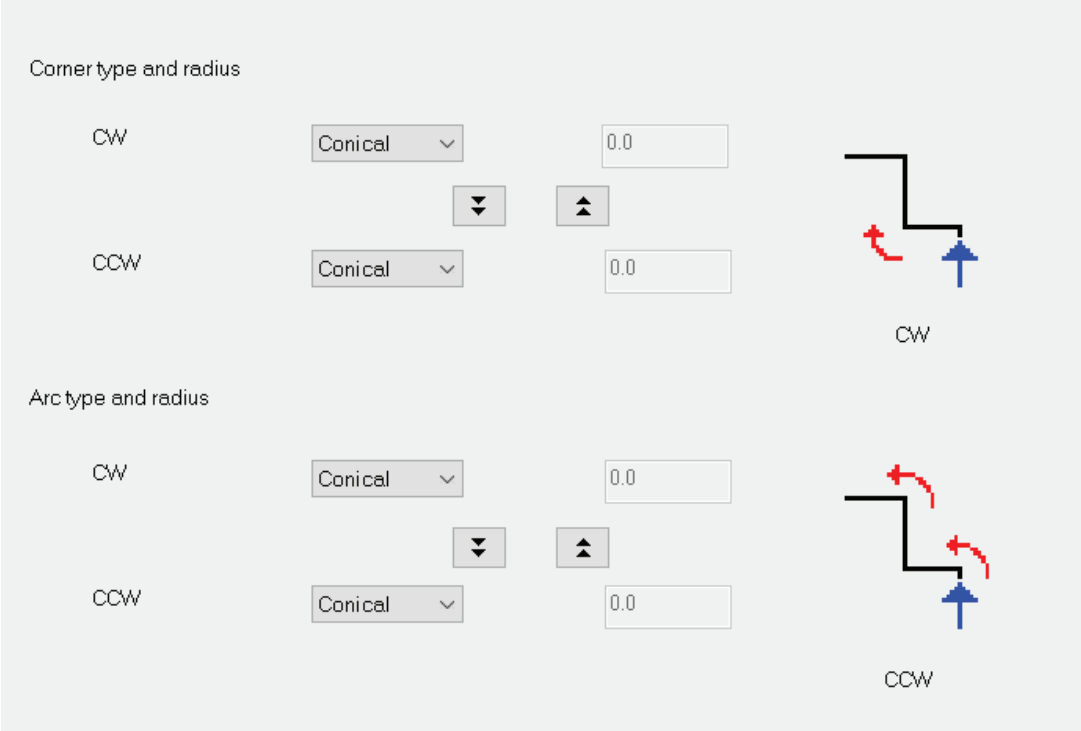
Land height is available only when **Land up** or **Land down** is selected for **Taper** style, and sets the plane at which the wire pivots to the taper angle.

XY height sets the location of the XY plane that contains the geometry for the lower contour of a 4-Axis part.

XY trim plane can be used to set the location of the lower guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to G code 1015 in the NC file. It can be set directly at the controller. When not using the XY trim plane, enter the same value as the XY height.

Corners

The **Corners** page allows you to control how the wire cuts the corners in the offset contour (the contour that creates the taper). You have a choice between the following combinations: Conical, Sharp, Constant, Other, Fixed and Fish Tail.



Corner type and radius set to **Sharp** creates an angled moved around the corner of the part.

Conical creates a cone-shaped move around the corner of the part. Conical results in a move that is bigger in the contour than in the geometry.

Constant keeps the corner or arc in the contour the same size as the corner or arc in the geometry.

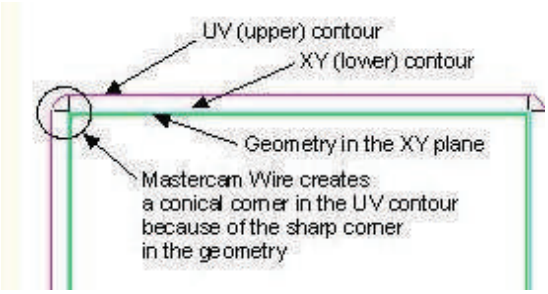
Other allows you to specify a radius for a custom corner defined in the post processor.

Fixed applies the same radius to all corners. Fixed corners must be supported by the post processor for this selection to appear.

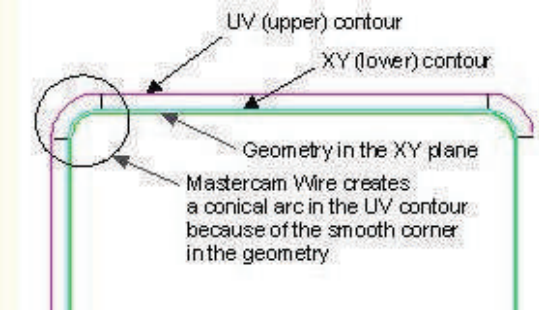
Fishtail causes the wire to move slightly off the part, swing around in a loop, and then reenter the part. Fishtail corners create very sharp corners.

Conical

- **Corner Type.**

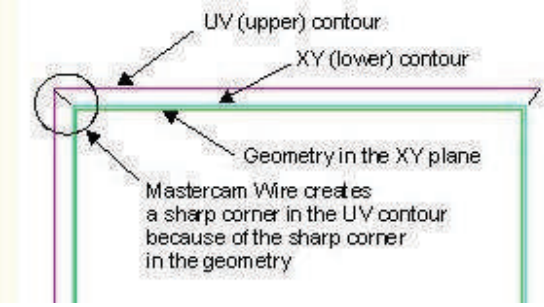


- **Arc Type.**

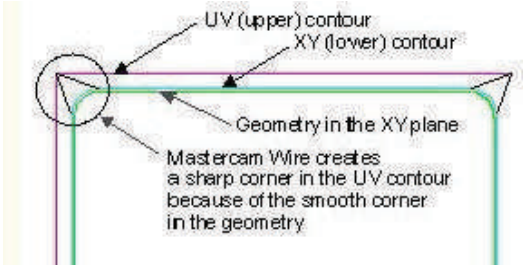


Sharp

- **Corner Type.**

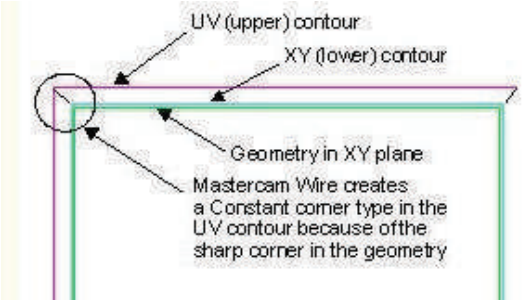


- **Arc Type.**

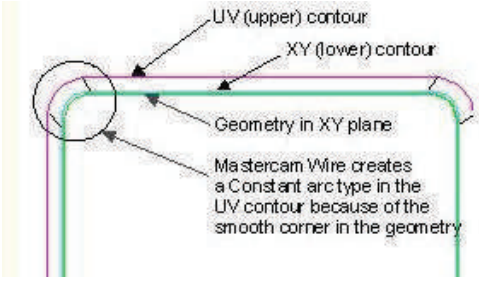


Constant

- **Corner Type.**

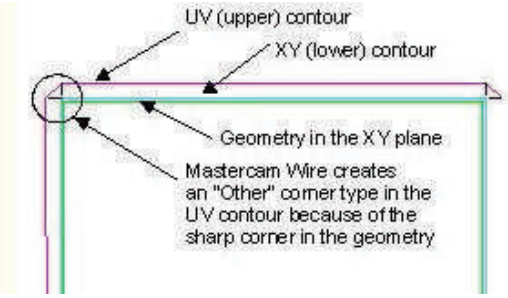


- **Arc Type.**

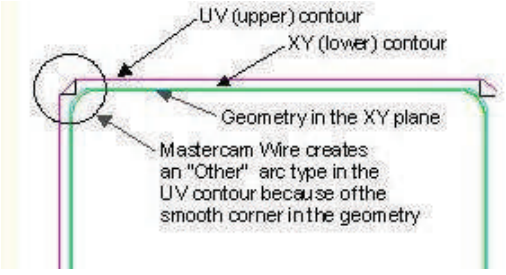


Other

- **Corner Type.**

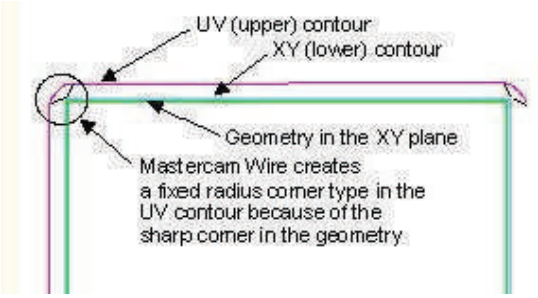


- **Arc Type.**

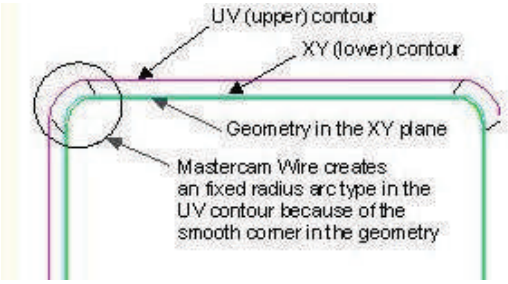


Fixed

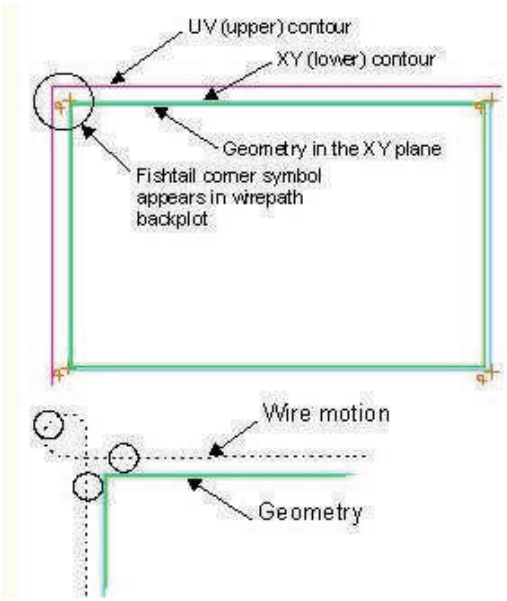
- **Corner Type.**



- **Arc Type.**

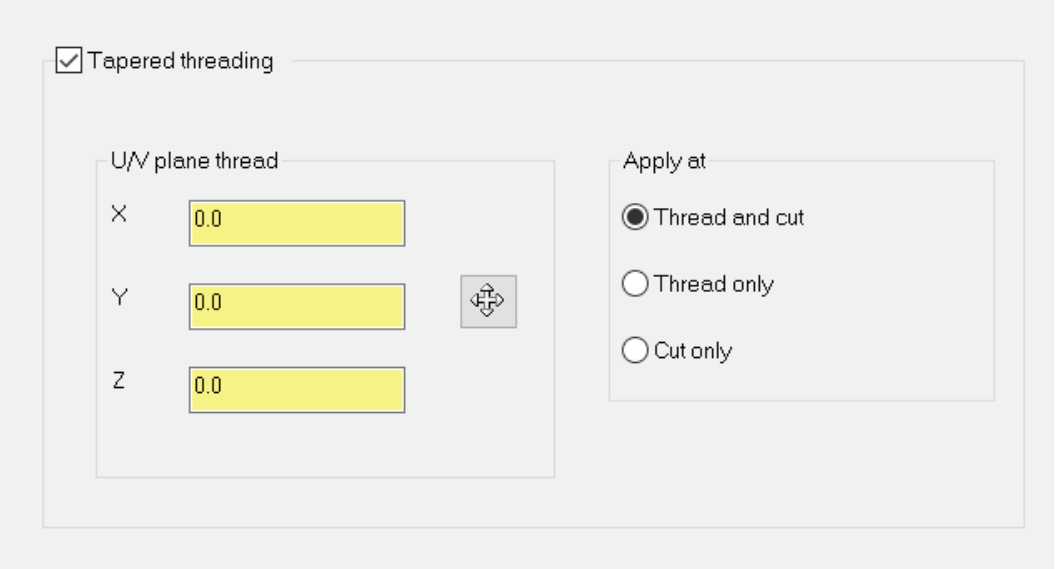


Fish tail



Tapered Threading

The **Tapered Threading** page allows you to specify an XY location in the UV plane for the thread or cut position, to create a tapered thread (non-vertical thread cut).

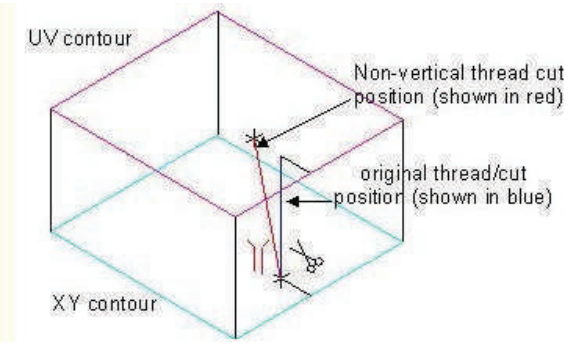


X, Y, Z set the axes coordinates for the UV thread/cut position.

Thread and cut sets the thread and cut position to the non-vertical coordinates.

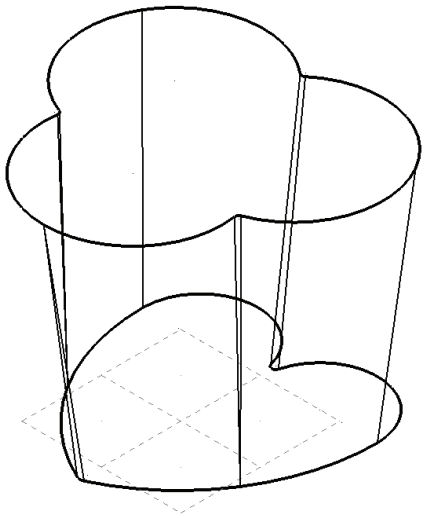
Thread only sets the thread position to the non-vertical coordinates independent of the cut position.

Cut only sets the cut position to the non-vertical coordinates independent of the thread position.



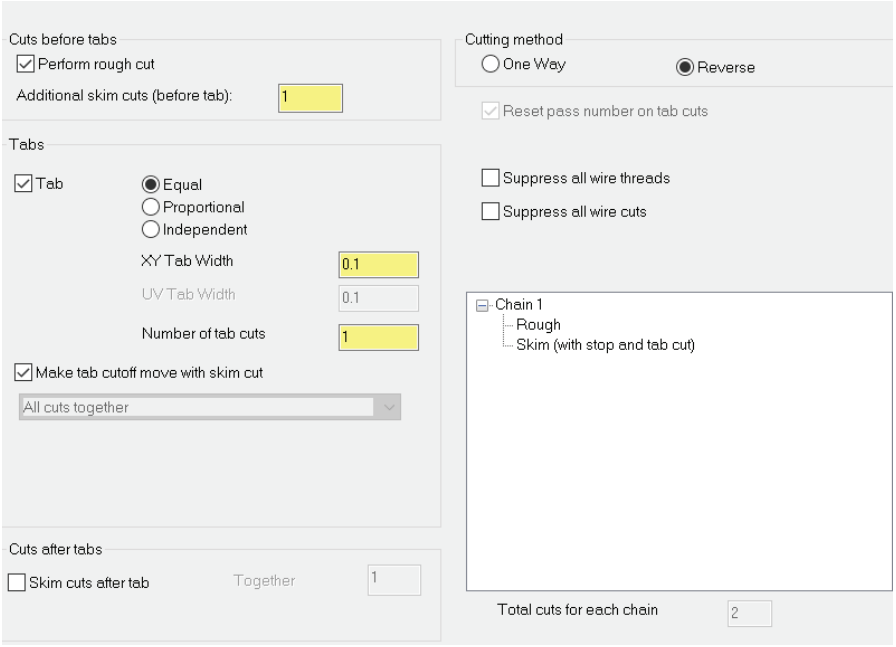
4-AXIS WIREPATH

4-Axis wirepaths have geometry in both the XY plane (the lower contour) and UV plane (the upper contour). Where a contour wirepath derives a second contour from the geometry, a 4-Axis wirepath can have completely different geometry in the two planes, resulting in different wire movement in the XY and UV planes.



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.



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

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

Tab - is the uncut material that secures the part to the material; can be cutoff with the skim cut pass or in a separate pass.

Equal - sets the UV and XY tab widths equally.

Proportional - calculates the UV Tab Width value by multiplying the length of the final entity on the UV plane by the ratio of the XY Tab Width value to the length of the final entity on the XY plane.

Independent - allows you to set the tab width on both the XY and UV planes.

Skim cuts after tab - allows you to enter the number of finish passes after cutting the tab.

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

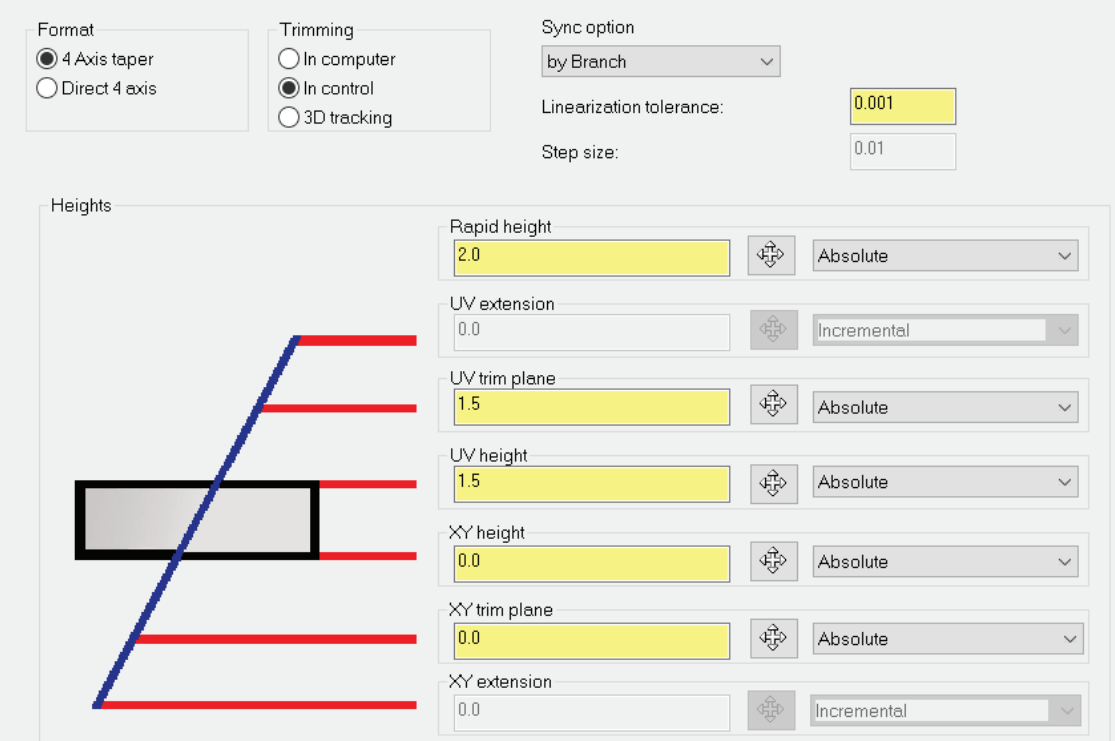
Reset Pass Number on Tab Cuts - is selected to use the power settings stored as Pass 1 in the wire power settings library for the first tab cut. Mastercam Wire uses the Pass 1 settings for the rough cut.

Suppress all wire threads - enabled will output no thread flags to the wirepath's NCI file.

Suppress all wire cuts - enabled will output no cut flags to the wirepath's NCI file.

4-Axis

The **4-Axis** page allows you to establish 4-Axis wirepath settings such as the linear arc move format (direct or taper), trimming options, and synchronization of the lower (XY) and upper (UV) contours.



4-Axis taper - outputs only UV/XY linear moves.

Direct 4-Axis - outputs both XY/UV linear and arc moves.

3D tracking - requires that your wire machine support 3D tracking to use this parameter. It includes variable Z depths with the XY and UV planes.

Rapid height - sets the Z depth of a rapid move. Can be higher than the **UV height** or **UV trim planes** in order to clear any clamps.

UV trim plane - can be used to set the location of the upper guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to G code 1015 in the NC file. It is usually set directly at the controller.

UV height - sets the location of the XY plane that contains the geometry for the upper contour of a 4-axis part.

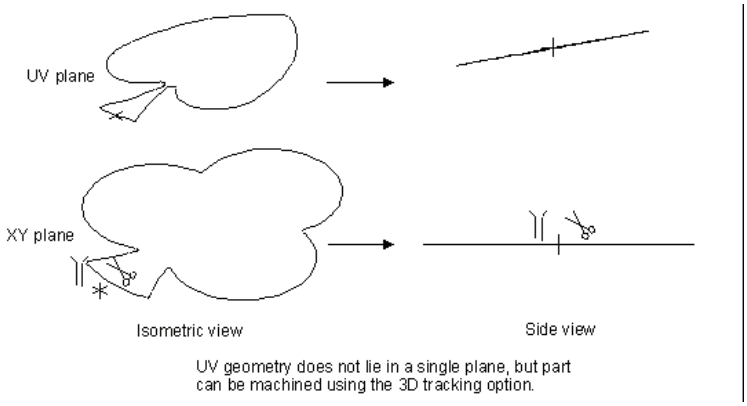
XY height - sets the location of the XY plane that contains the geometry for the lower contour of a 4-axis part.

XY trim plane - can be used to set the location of the lower guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to G code 1015 in the NC file. It can be set directly at the controller. When not using the XY trim plane, enter the same value as the XY height.

3D Tracking

3D tracking includes variable Z depths with the XY and UV planes. When supported by the wire EDM machine and control, 3D tracking allows machining of geometry that doesn't lie completely in a single construction plane. With 3D tracking, the wire guides on the machine follow the non planar geometry. When 3D tracking is selected, Mastercam Wire enables two additional planes: the XY and UV extensions.

UV Extension is used for geometry with variable depths that does not lie completely in a single plane when 3D tracking is supported by the wire EDM machine. Give a positive incremental value to lift the extension above the UV height and trim planes.



UV geometry does not lie in a single plane, but part can be machined using the 3D tracking option.

Sync Option

Sync option synchronizes the upper and lower contours using chain synchronization. When synchronizing the contours, Mastercam Wire breaks each chain into a number of separate sub chains then matches up the chains using the Sync mode selection. The Sync option provides a choice of methods that Mastercam Wire can use to place points along the chains, which it matches up when synchronizing the chains.
When Mastercam Wire synchronizes the chains, it aligns the top contour with the bottom contour by entity, point, branch point, node (if both contours are made up of splines) or where you tell it (manual, manual/ density).

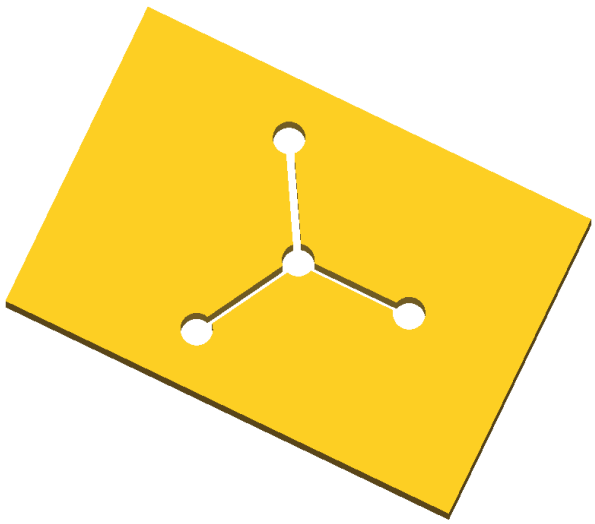
None synchronizes the chains by dividing them into an even number of segments using the Step Size option in the 4-axis parameters dialog box tab. The value you enter for Step Size determines how far apart these points are placed. We recommend that you keep this value small to retain accuracy in the corners. In many cases, the default value of .01 is sufficient.

By branch requires branch lines to be added to the geometry to create sync points as shown in the example above.
By entity matches the endpoints of each entity and requires both chains to have an equal number of entities.
By node applies only to parametric splines and synchronizes the two chains by nodes on the splines.

Manual and manual/density allow you to manually place the sync points.

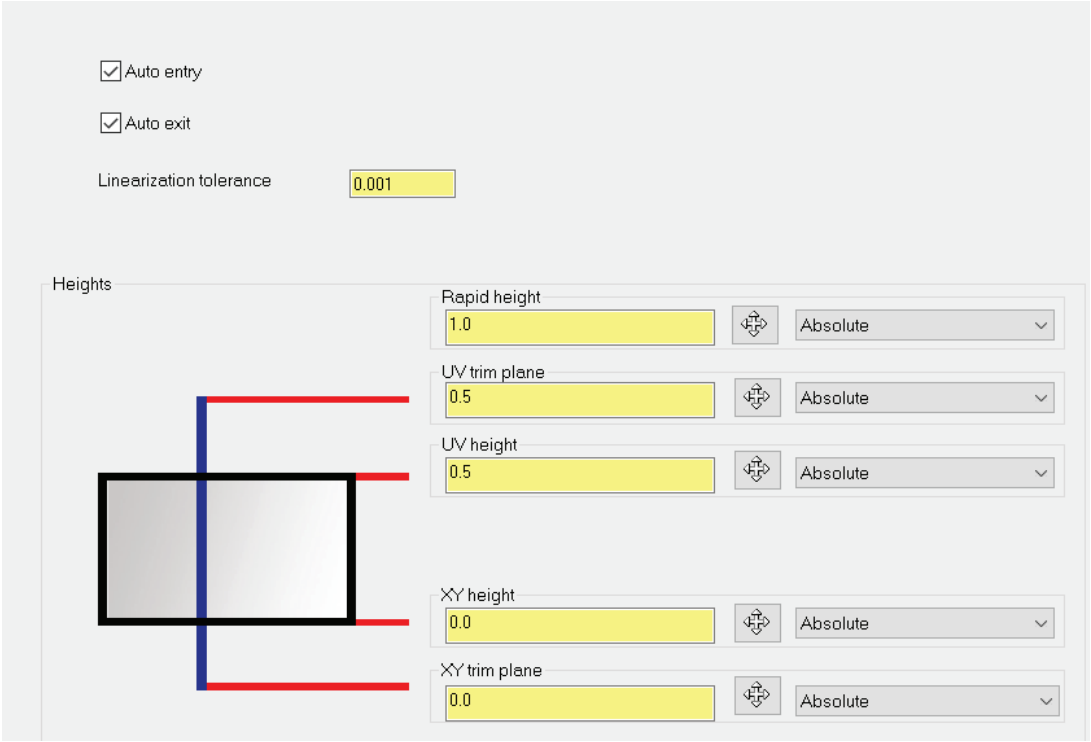
NO CORE

No core wirepaths pocket out areas of material without producing a slug. The wirepath typically starts at a pre-drilled hole in the material, and zigzags or spirals outward until all material within the chained geometry is removed.



No Core

No core page allows you to establish No core wirepath settings such as the Z depth of a rapid move, the positions of the upper and lower wire guides, and the UV and XY plane heights (usually the upper and lower contour of the geometry).



Auto entry/exit - starts/ends the No core wirepath at the thread position.

Rapid height - sets the Z depth of a rapid move. Can be higher than the **UV height** or **UV trim planes** in order to clear any clamps.

UV trim plane - can be used to set the location of the upper guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to G code 1015 in the NC file. It is usually set directly at the controller.

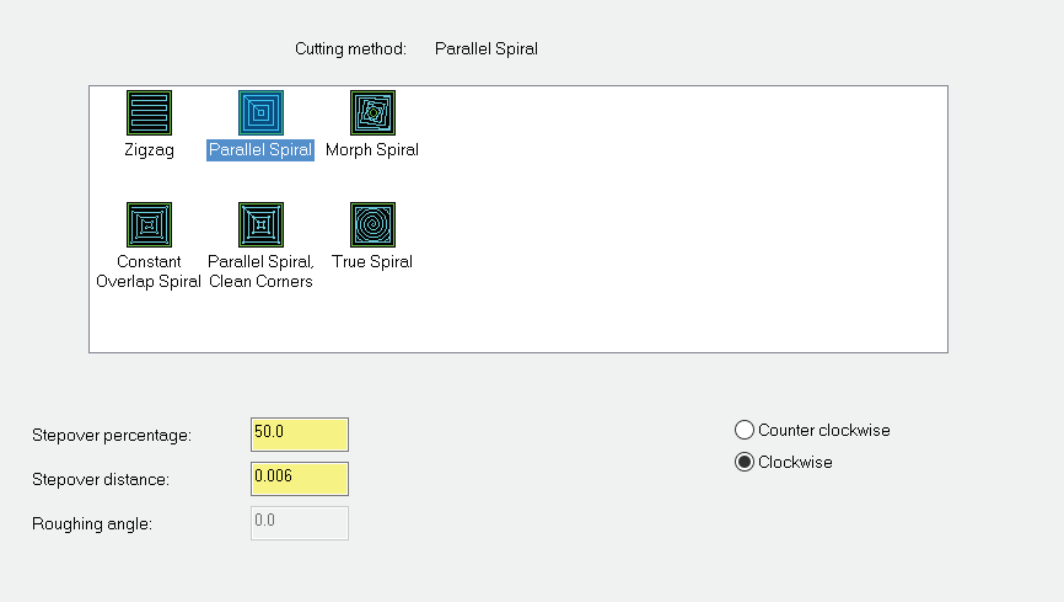
UV height - sets the height of the part, the upper contour of the wirepath.

XY height - sets the location of the XY plane that contains the geometry for the lower contour of the wirepath.

XY trim plane - can be used to set the location of the lower guide on the wire machine that may be required by the control to locate the guide in relation to the part. Trim plane values are written to G code 1015 in the NC file. It can be set directly at the controller. When not using the XY trim plane, enter the same value as the XY height.

Rough

The **Rough** page allows you to set up rough passes on no core wirepaths. You can define the type of no core cutting method (Zigzag, Constant Overlap Spiral, Parallel Spiral, Parallel Spiral with Clean Corners, Morph Spiral, or True Spiral). It also allows you to set the stepover percentage and distance.



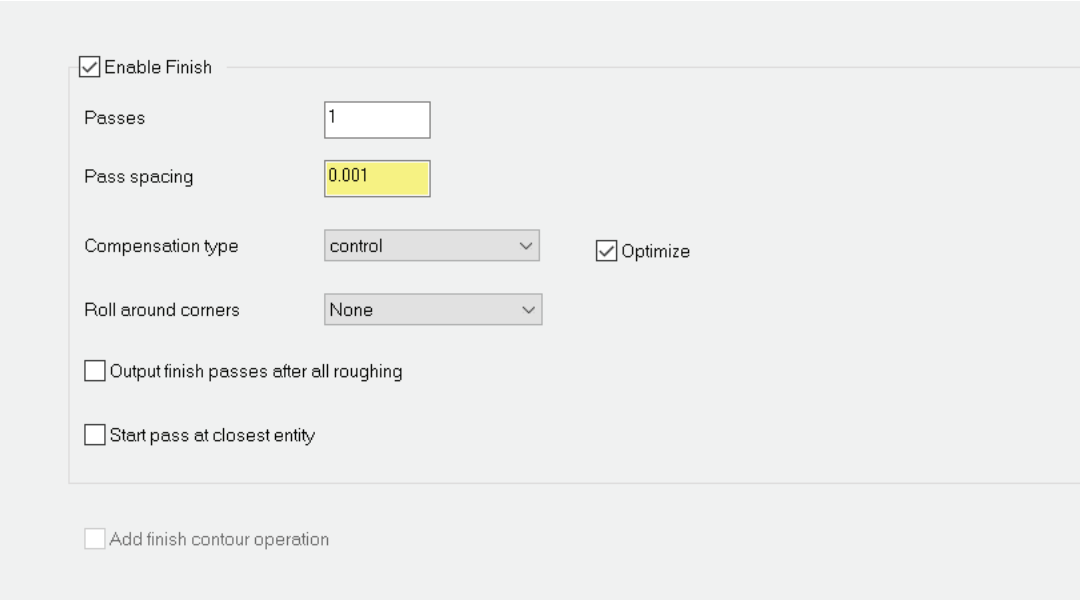
Cutting method allows you to set the cutting pattern that Mastercam Wire will use to clean out the No core part.

Stepover percentage sets the distance the wire shifts over between XY moves, as a percentage of wire diameter. Changing the stepover percentage automatically adjusts the stepover distance.

Clockwise sets the direction for the spiral no core cutting methods.

Finish

The **Finish** page allows you to create and set up finish passes on no core wirepaths.



Passes - allows you to set the the number of finish passes.

Pass spacing - sets the finish pass cut amount.

Compensation type - sets the Wire compensation for the finish pass to offset the wire as part of the wirepath (Computer), to have the control offset the wire (Control), or both to have an offset applied in the wirepath but also provide the ability for additional compensation to be set in the control (Both).

Optimize - eliminates arcs in the wirepath that are less than or equal to the radius of the wire and helps prevent gouging.

Roll around corners - allows you to insert arc moves around corners in the wirepath. The radius of the arc moves equals the radius of the wire.

Output finish passes after all roughing - option is used when multiple no core wirepaths exist in a single operation, Mastercam Wire can cut the roughing pass on each no core first. After completing all the roughing passes, Mastercam Wire then cuts the finish passes on each no core wirepath.

Start pass at closest entity - starts the finish pass at the closest endpoint of the closest entity at the end of the roughing wirepath.

Add finish contour operation - launches the Contour Wirepath dialog box so that additional finish passes can be specified with the full functionality of the contour wirepath. Generated as a separate operation in Toolpath Manager.

Numerics

3D Tracking 457
4-Axis Page 345, 456
4-Axis Wirepath 454
4-Axis Wirepaths 334

A

About Construction Depth (Z Depth) 424
Arc 136
Arc Tangent To One Entity 136
Attributes 7
  2D / 3D Construction 7
  Clear Color 7
  Color 7
  Level 7
  Line Style 7
  Line Width 7
  Point Style 7
  Z Depth 7

B

Backplotting 53
Bolt Circle 368

C

Chaining 426
Chaining And Window Options 430
Change the Wireframe Color 7
Compensation Page 49, 443
Construction Plane (Cplane) 423
Contour Wirepath 39, 440
Corners Page 106, 449
Create a Polar Line 84
Create a Rectangle 18
Create a Rectangular Shape 218
Create Arc Tangent To Three Entities 147
Create Line Tangent At An Angle 139
Create Parallel Lines 20, 77
Creating 3d Wireframe 422
Customizing Toolbars 411
Cut Parameters 48
Cut Parameters Page 442
Cut Point 36

D

Default Key Assignments 410
DOOR GEOMETRY 415

F

Finish Page 461

G

Graphics View 422
GUI - Graphical User Interface 3
  Backstage (FILE) 3
  Graphic Area 3
  QAT - Quick Access Toolbar 3
  Quick Mask Buttons 3
  Ribbon 3
  Right Click Menu 3
  Scale 3
  Selection Bar 3
  Tabs 3
  Toolpaths/Solids/Planes Manager 3
  WCS 3

K

Key Mapping 418

L

Leads Page 50, 445
Line Endpoints 4
  Endpoint Coordinates 5
  Endpoint, Length and Angle 5

M

Machine Definition 39
Machine Group Properties 438
Manager Panels 9
  Auto Hide 10
  Toolpaths Manager 9
Mastercam To Metric
  Current Session Only 11
  Default 12

N

No Core Page 459
No Core Wirepaths 383, 458

O

Operation List Area 434

P

Post processing 63
Post Processor 63

Q

Quiz Answers 405

R

Reorganizing The Toolbars 4
Right Mouse Click In The Toolpaths Manager 435
Rotate/Copy 152
Rough Page 389, 460

S

Set the Grid 13
Stock Setup 56, 439
Stops Page 444
Synch Option 457

T

Tab/Finish Leads Page 181, 446
Taper Page 51, 447
Tapered Threading Page 453
Thread Point 33
Toolpath Manager 431
Translate/Copy 29
Trim 2 Entities 31
Trim 3 Entities 25

V

Verify 56

W

Window Selection 429
Wire/Power Page 46, 441

Z

Z Depth 424

Did you like this book?

Discover other Mastercam Training Solutions at

www.**eMastercam**.com

eMastercam has the largest Mastercam community on the web with over 56,000 members and over a million forum posts. While there, be sure to join the conversation!

Training Solutions Available:

Training Tutorials
Handbooks
Instructor Kits
Professional Courseware
Multimedia
Online Video eCourses
CNC STEM Activities
Site Licenses
Sitewide Subscriptions!

Mastercam® @Courses

LOG IN & LEARN TODAY!

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

Highlights Include:

- Online previews with table of contents including the time it takes to complete each session.
- Mastercam 2018 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.
- **Lifetime access**

www.**eMastercam**.com/ecourses

Only $149 each!

NEW! eMastercam.com

Sitewide Subscription

ALL eBook* and eCourses

only $**39.99**/Month!

LEARN MORE: WWW.**eMastercam**.com

OCTOPUZ®

Robot Software

Program and simulate all your robots
in the same cell, for all your applications,
while your robots are still working.

Book a demo and see for yourself!

www.**OCTOPUZ**.com

Does your school have robots? You'll want to check out OCTOPUZ.

As easy to teach as it is to learn, OCTOPUZ is an offline robot programming and simulation software used world-wide in industry. This powerful solution prepares your students for the real world of robotics and provides them with the necessary skills to not only be hired, but to excel.

With OCTOPUZ you can:

1. Create your custom cell.
2. Program your application.
3. Verify using simulation.
4. Generate and post code to ALL your robot brands.

www.**OCTOPUZ**.com/for-education

OCTOPUZ®
Robot Programming & SimulationSoftware

Post processors that are requested worldwide.

We speak your machine's language.

Implementing successful post processors for decades, the post department at In-House Solutions has earned a reputation for quality, resulting in one of the largest post departments in the world. We have an extensive library that is continually developed and expanded to include current machines and controllers. If a post is not already available, we will develop a custom one for nearly any machine. We have great relationships with OEMs who provide technical information for both machines and controls, which in turn allows us to produce post processors that will generate edit-free code and run your machine efficiently.

www.**inhousesolutions**.com/posts

In-House Solutions