Mastercam®
MILL LEVEL 1
TRAINING TUTORIAL SERIES
X6

Mastercam X6

Mill Level 1 Training Tutorial

To order more books:

Call 1-800-529-5517 or

Visit www.inhousesolutions.com or

Contact your Mastercam dealer

Mastercam X^6 Mill Level 1 Training Tutorial

Copyright: 1998 - 2011 In-House Solutions Inc. All rights reserved

Software: Mastercam X^6

Author: Mariana Lendel

ISBN: 978-1-926955-26-1

Revision Date: November 18, 2011

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

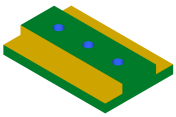
Mastercam is a registered trademark of CNC Software, Inc.

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

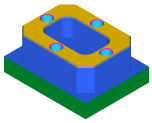
Mill Level 1 Training Tutorial Contents

Getting Started 1

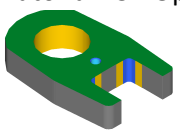
Tutorial #1 - Open Contour and Drill Toolpaths 11



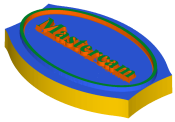
Tutorial #2 - Facing, Pocket, Contour, Drilling and Contour-Chamfer Toolpaths 59



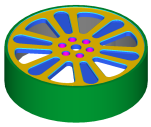
Tutorial #3 - Open Pocket, Contour Remachining, Circle Mill, Drill and Contour Toolpaths 133



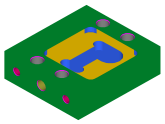
Tutorial #4 - 2D HS Core Mill, 2D HS Dynamic Core Mill, Pocket Island Facing and Pocket Remachining Toolpaths 231



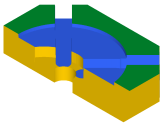
Tutorial #5 - 2D HS Dynamic Area Mill, Drill, Circle Mill and Contour Chamfer Toolpaths 309



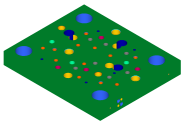
Tutorial #6 - Working with Tool Planes, 2D HS Area Mill, 2D HS Rest Mill, Drill and Slot Mill Toolpaths 417



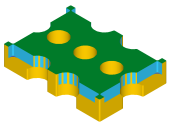
Tutorial #7 - 2D HS Dynamic Core Mill, 2D HS Core Mill, 2D HS Blend, 2D HS Peel Toolpaths 567



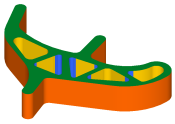
Tutorial #8 - Feature Based Drilling (FBD) Toolpath 639



Tutorial #9 - Feature Based Milling Toolpath 703



Tutorial #10 - 2D HS Dynamic Area Mill, 2D HS Dynamic Rest Mill, 2D HS Dynamic Core Mill, 2D HS Dynamic Contour and Machine Simulation 765



General Notes 831

Creating/Editing Tools 889

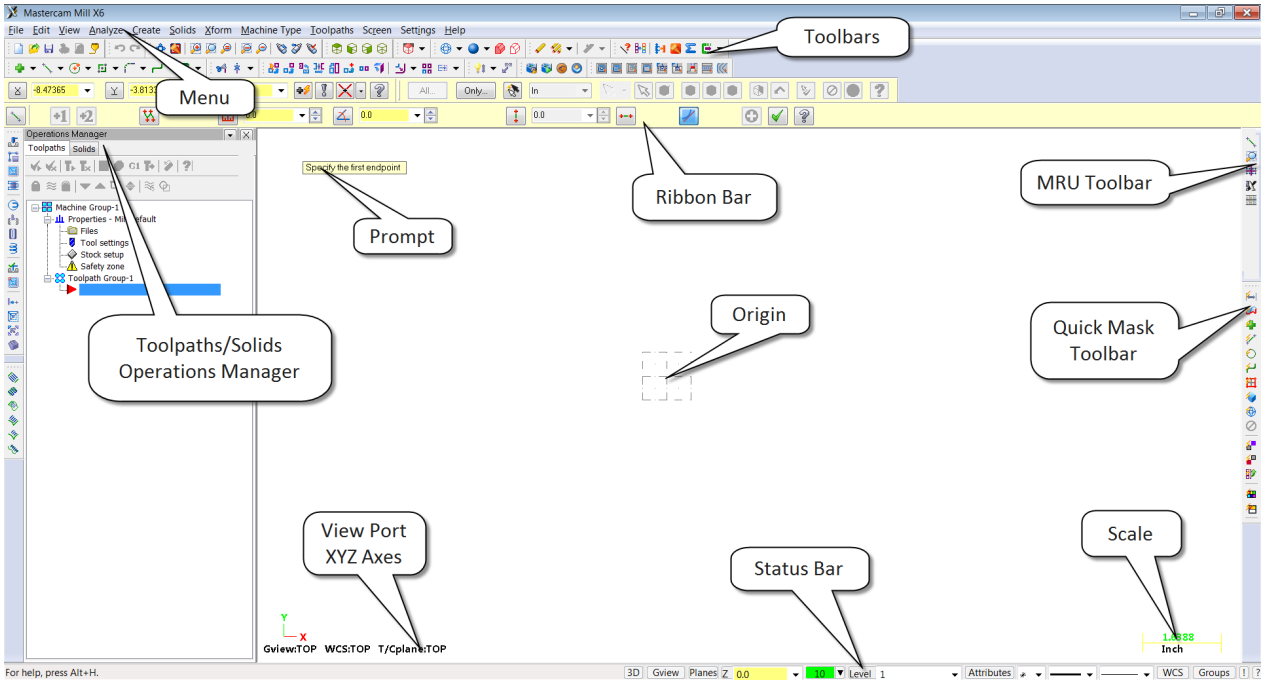
Quiz Answers 897

GETTING STARTED

Objectives:

✓ The Student will learn about the Graphical User Interface.
✓ The Student will learn how to navigate through Mastercam.
✓ How to use the Status Bar to set the attributes.
✓ Set the Toolbar States.
✓ Setting the Grid.

STEP 1: GUI - GRAPHICAL USER INTERFACE



Menu	Allows you to select all the functions in Mastercam to create geometry and toolpaths.
Toolbars	Can be used instead of the menu to create geometry and toolpaths.
Ribbon Bar	Allows you to enter the values and settings that define the entities that you are currently creating or modifying.
Function prompt	Prompts the user for info.
Status Bar	Allows you to set the attributes (color, level, style, and width) and the View/Plane and Z depth currently used.
Toolpaths/Solid Manager	Lists the history of the toolpath operations and solids.
Origin	Geometry origin from which the system measures the point coordinates in X,Y and Z axes in the current plane.
Graphic Area	Workspace area in Mastercam where the geometry displays.
MRU Toolbar	Lists the most recently used functions.
Quick Mask Toolbar	Lets you select all entities of a specific type.
Scale	Shows you a scale of the object on the screen.
View Port XYZ Axes	Inform you which Graphics view, WCS and Toolplane/Construction plane you are working in.

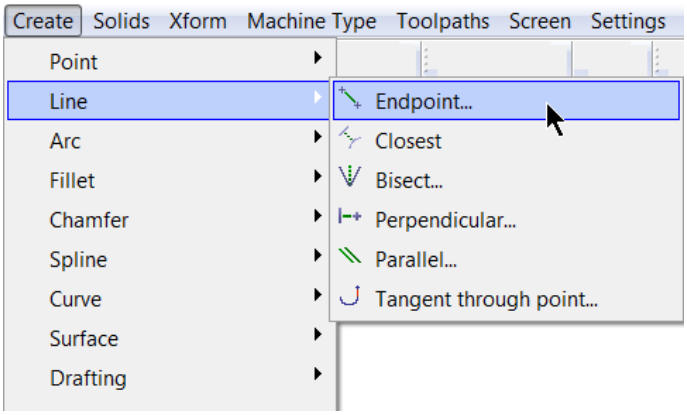
STEP 2: NAVIGATE THROUGH MASTERCAM

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

2.1 Using the Menu to select the command Create Line Endpoint

- Left click on **Create.**
- Move the cursor on the drop-down menu to the **Line** function. This will open a flyout menu with all the commands related to create lines.
- Left-click on the desired command Endpoint as shown in Figure: 2.1.1.

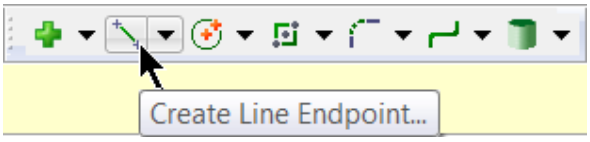
Figure: 2.1.1



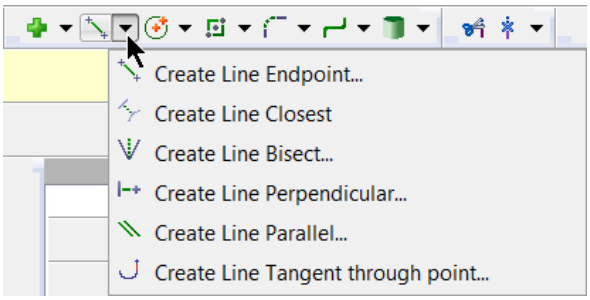
2.2 Using the Toolbars to select the command Create Line Endpoint

- Left Click on the **Create Line Endpoint** command icon as shown in Figure: 2.2.1.

Figure: 2.2.1



- To see the commands which are found under Create Line, in the menu bar choose the drop down arrow to the right of the icon as shown.

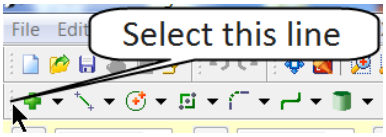


2.3 Reorganizing the Toolbars

NOTE: Mastercam is fully customizable meaning you can move the toolbars anywhere you desire allowing you to work more efficiently.

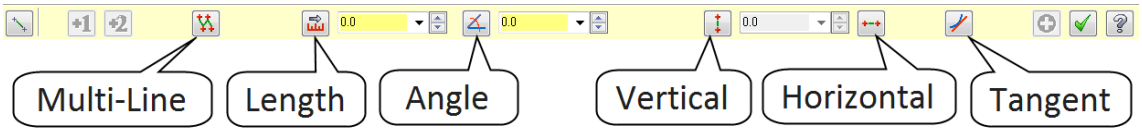
- To move a toolbar select the left vertical line in front of the toolbar and drag it to the desired location on the screen as shown in Figure: 2.3.1.

Figure: 2.3.1



NOTE: To customize the toolbars please see the General Notes.

2.4 Create Line Endpoint Ribbon Bar example



- To create a line left click on the screen. Then enter the distance and angle if necessary.
- To continue making lines choose the **Apply** button from the ribbon bar or press enter. To exit the current command select the **OK** button or press the **Esc** button.
- To undo the last command use 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.

2.5 Function Prompt

- Prompts the user to execute a command.
- For example this prompt is used in the **Create Line Endpoint** command as shown in Figure: 2.5.1.

Figure: 2.5.1

2.6 Status Bar

- The Status bar seen in Figure: 2.6.1 is used to set the drawing attributes, view, plane, Z depth and construction mode.

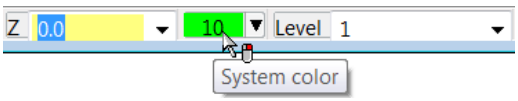
Figure: 2.6.1

2D / 3D Construction	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.
Gview	Sets the graphics view which you will view your geometry from. You can choose a different standard Gview, an existing named view, create a new named view or set the Gview to the current Cplane.
Planes	Sets the construction plane in which you create and manipulate your geometry. You can change the current plane by selecting a different standard plane, selecting an existing named view, creating a new named view and by setting the plane to equal the current Gview.
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.
Color	Assigns the current color. To change the current color click in the color field and select a colour from the colour pallet.
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 or select the level using one of the following methods.
Attributes	Lets you change one or more drawing attributes.
Point Style	Displays and sets the systems point style.
Line Style	Shows and sets the system line style.
Line Width	Displays and sets the current system line width.

Work Coordinate System (WCS)	Allows you to redefine the WCS. You can choose from a list of stand views, specify origin coordinates, access the view manager or use the named function to access the view selection dialog box.
Groups	Defines a collection of entities or operations that can be manipulated as a single entity.

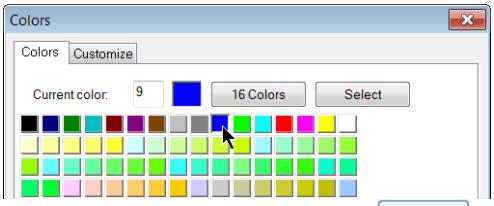
2.7 Change the Current System Colour

- Left Click on **System Color** as shown.



- Select the desired color from the dialog box as shown in Figure: 2.7.1.

Figure: 2.7.1



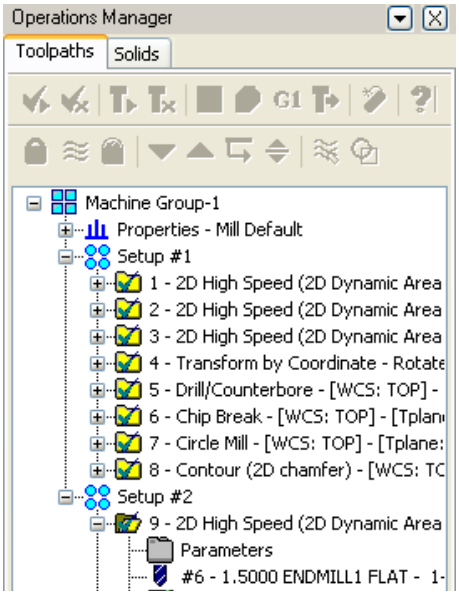
- Select the **OK** button to exit the command and begin creating geometry in the color of your choice.

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

2.8 The Toolpath Manager

- The Toolpath Manger displays all the operations for the current part.
- You can sort, edit, regenerate, verify and post any operations.
- For more information on the Toolpath Manager, please refer to **General Notes**.
- The Manger can be hidden to gain more space in the graphics area for creating geometry. Press **Alt + O** to hide the manager and **Alt + O** to reopen the manager as shown in Figure: 2.8.1.

Figure: 2.8.1

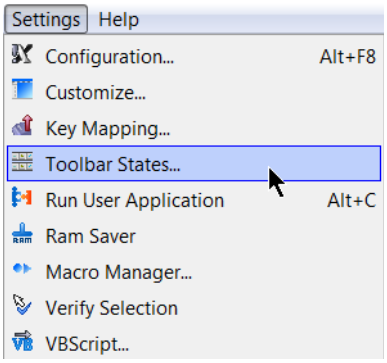


STEP 3: SETTING THE TOOLBAR STATES

- Before starting the geometry creation we should customize the toolbars to see the toolbars required to create the geometry and machine a 2D part.

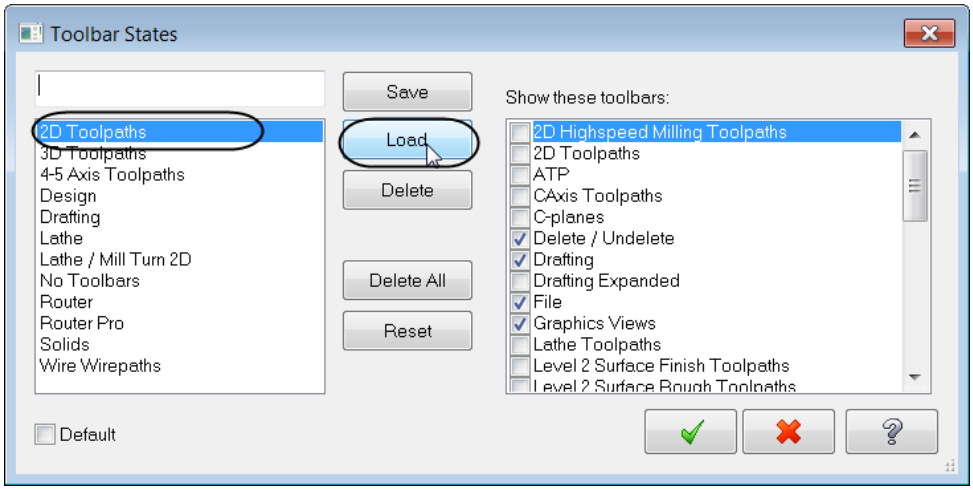
Settings

- **Toolbar States** as shown.



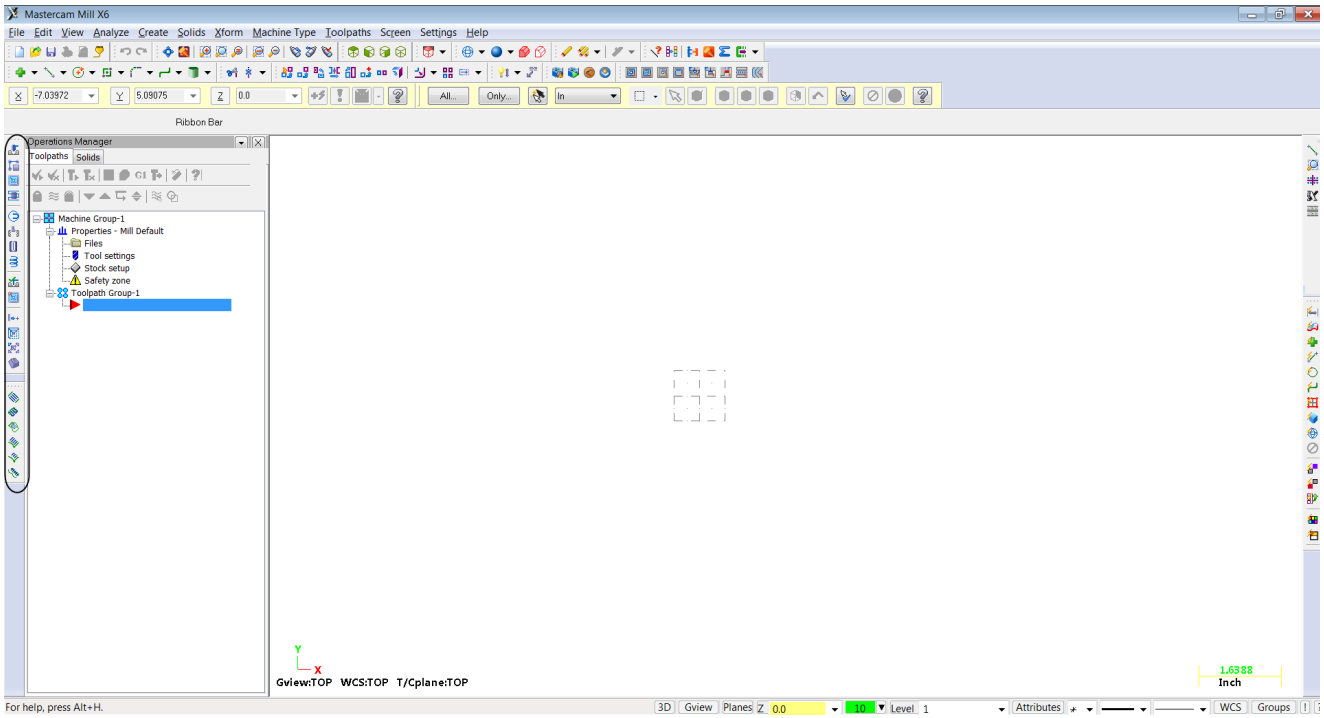
- From the left column select **2D Toolpaths**.
- Choose the **Load** button as shown in Figure: 3.0.1.

Figure: 3.0.1



- Select the OK button to accept the toolbar states.
- The 2D Toolpaths will be displayed to the left of the **Operations Manager** as shown in Figure: 3.0.2.

Figure: 3.0.2



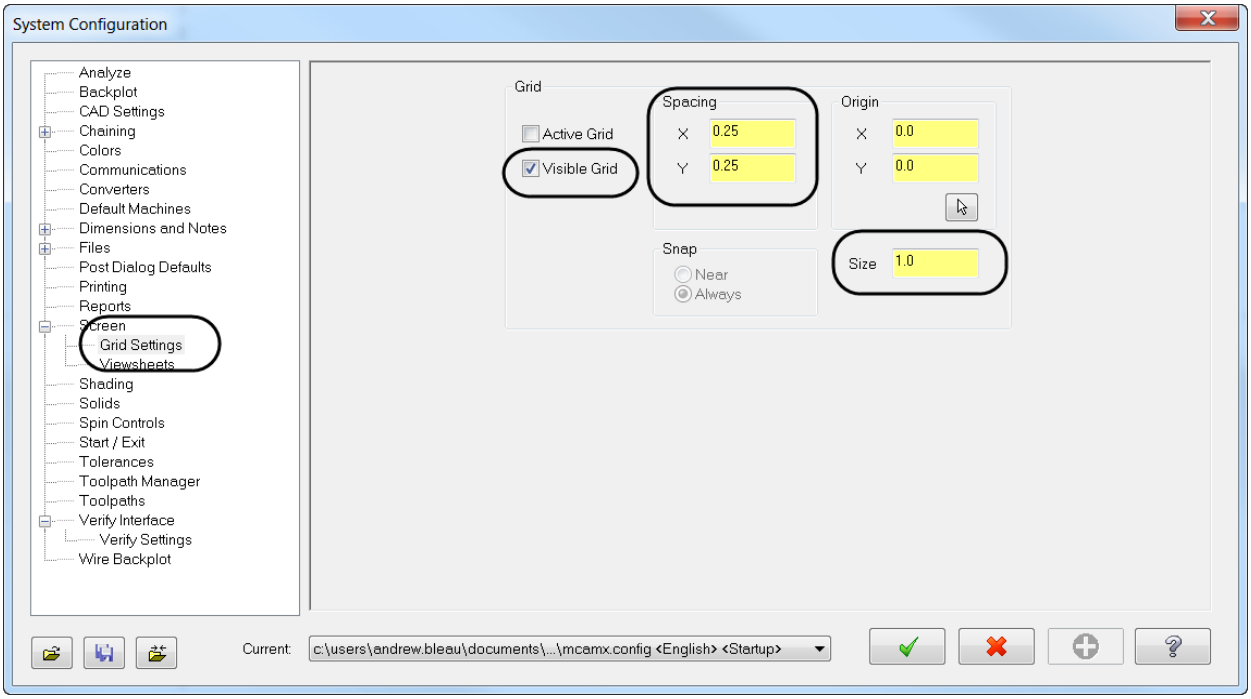
STEP 4: SETTING THE GRID

- Before beginning to create geometry you will enable the **Grid**. This will show you where the origin is.

Settings

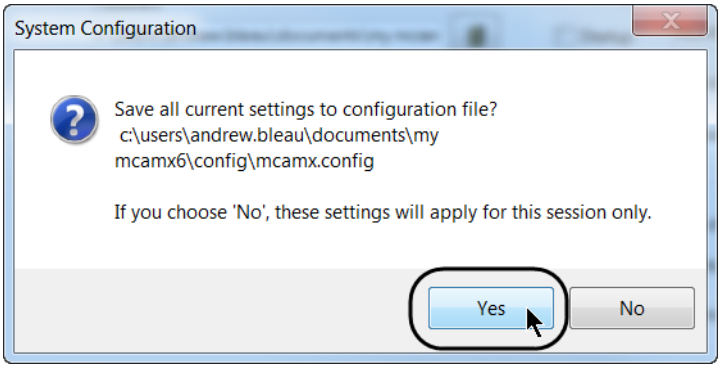
- **Configuration.**
- Select **Screen** from the configuration **Topics.**
- Select the plus sign (+) beside screen as shown in Figure: 4.0.1.

Figure: 4.0.1



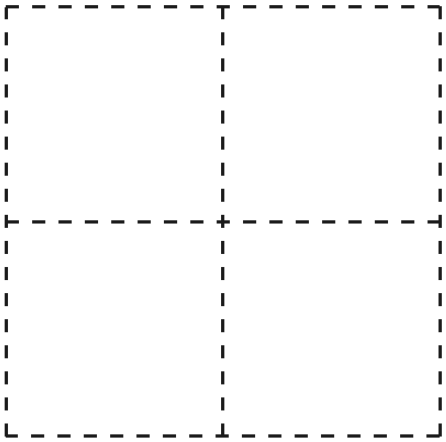
- In **Grid Settings** enable **Visible Grid** and change the **Spacing** to **X = 0.25 and Y = 0.25**.
- Set the **Size** to **1.0.**

- Choose the Ok button to exit.
- Select **Yes** to save the setting as shown.

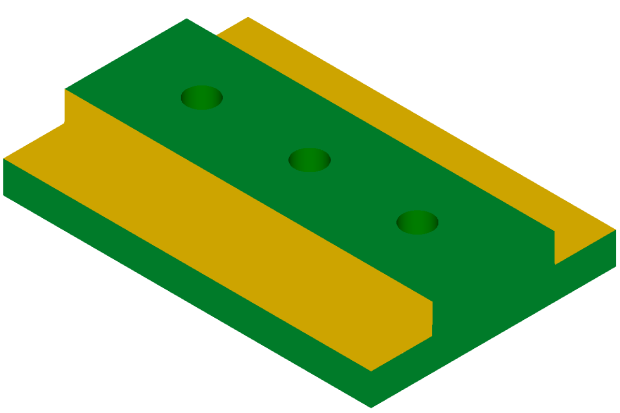


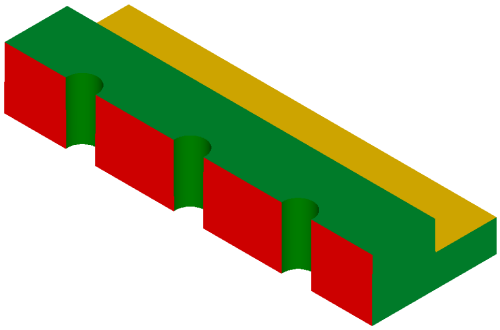
- The grid will appear on your screen as shown in Figure: 4.0.2.

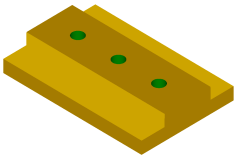
Figure: 4.0.2



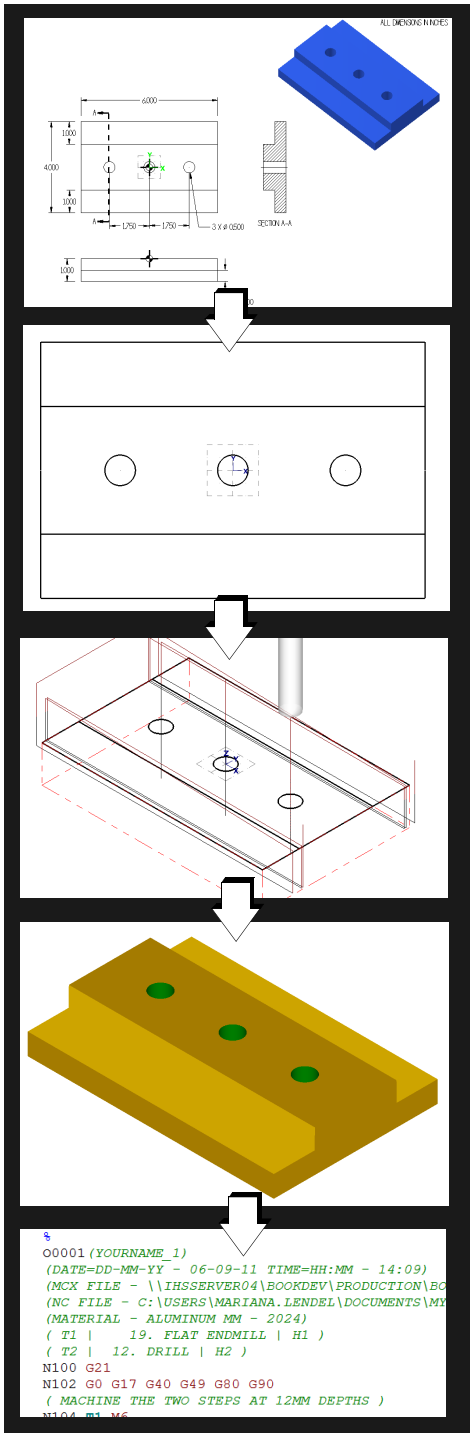
TUTORIAL #1







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 go about creating the geometry in Mastercam.

Create the 2D CAD Model used to generate Toolpaths from:

- The student will only create the Top 2D geometry needed to create the toolpaths.
- Geometry creation commands such as create rectangle, line parallel, mirror and circle center point will be used.

Create the necessary Toolpaths to machine the part:

- The student will set up the stock size to be used and the clamping method used.
- An Open Contour toolpath will be created to remove the material on the both steps.
- A Drilling toolpath will be created to machine the through holes.

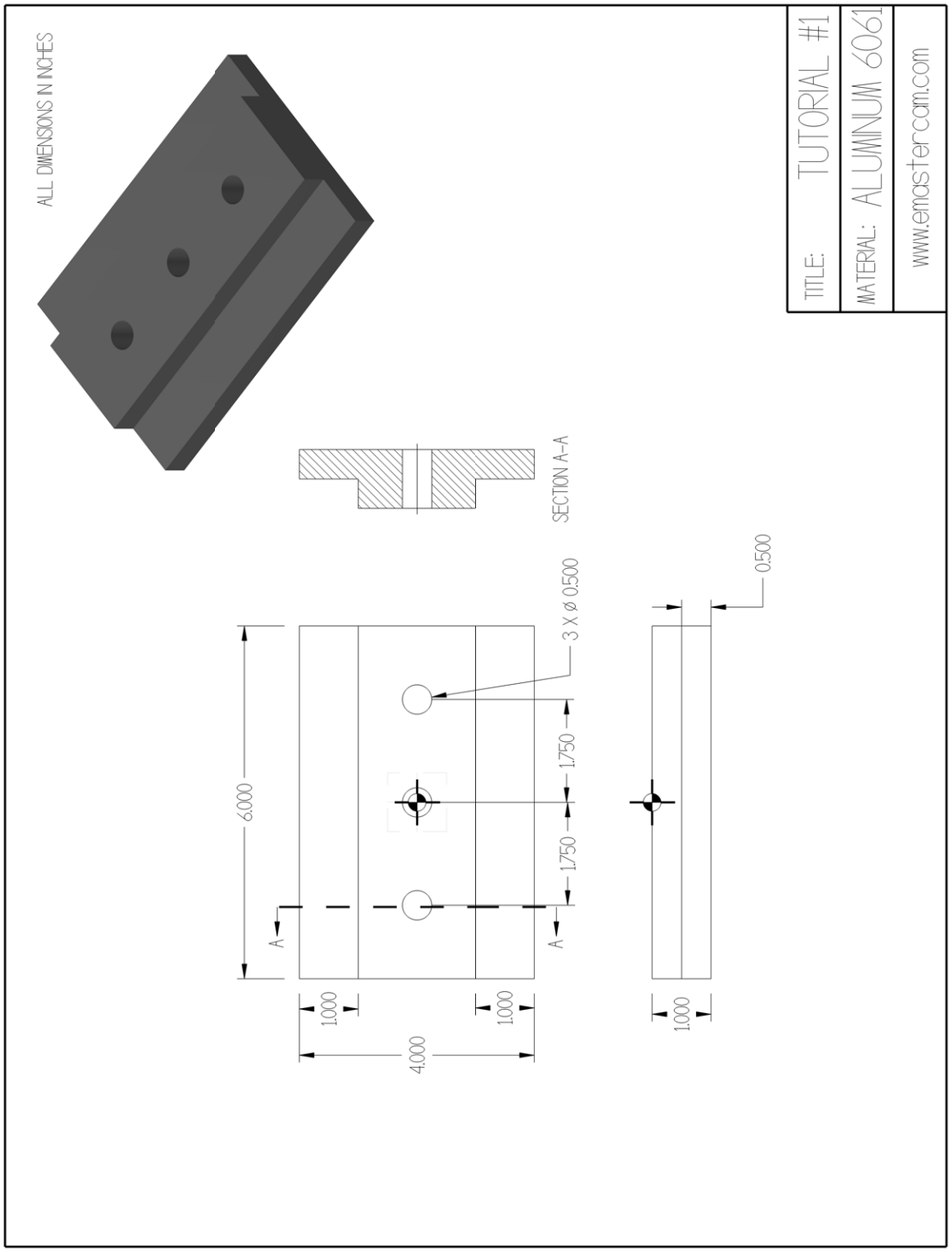
Backplot and Verify the file:

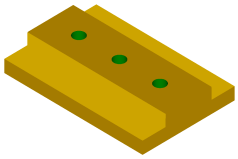
- The Backplot will be used to simulate a step by step process of the tool's movements.
- The Verify will be used to watch a tool machine the part out of a solid model.

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 #1
MATERIAL: ALUMINUM 6061
www.emastercam.com
SECTION A-A
3 X ø 0.500
0.500
6.000
1.750
1.750
A
A
1.000
1.000
4.000
1.000



GEOMETRY CREATION

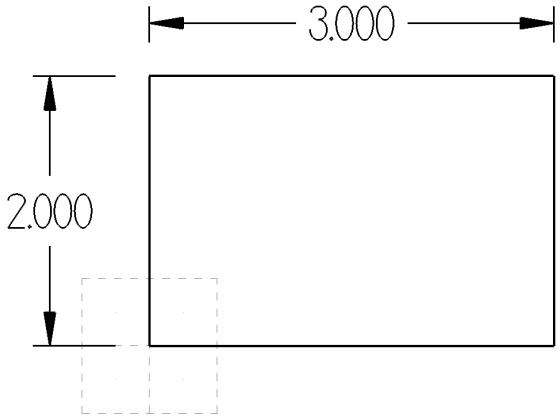
STEP 1: SETTING UP THE GRAPHIC USER INTERFACE

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

STEP 2: CREATE A RECTANGLE

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

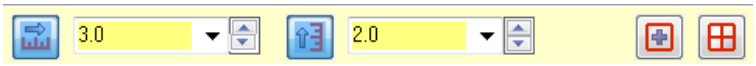
Step Preview:



2.1 Create a 3.0" by 2.0" Rectangle

Create

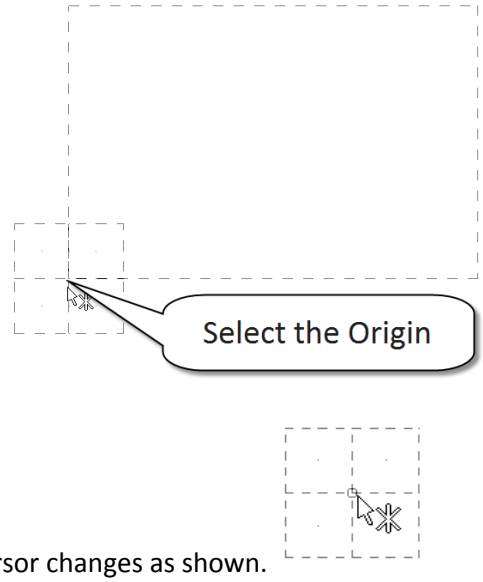
- **Rectangle.**
- Enter the **Width** and **Height** as shown.



- Press **Enter** after typing the values to see a preview of the rectangle.

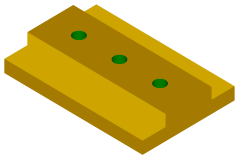
- [Select position for first corner]: Select the **Origin** as shown in Figure: 2.1.1.

Figure: 2.1.1



- 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.
- Use the **Fit** icon to fit the drawing to the screen.

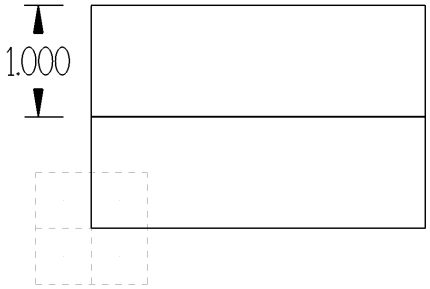
NOTE: During the geometry creation of this tutorial, if you make a mistake you can undo the last step using the **Undo** icon. You can undo as many steps as needed. If you delete or undo a step by mistake, just use the **Redo** icon. To delete unwanted geometry, select it first and then press **Delete**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



STEP 3: CREATE THE PARALLEL LINE

In this step you will learn how to create a parallel line knowing the distance.

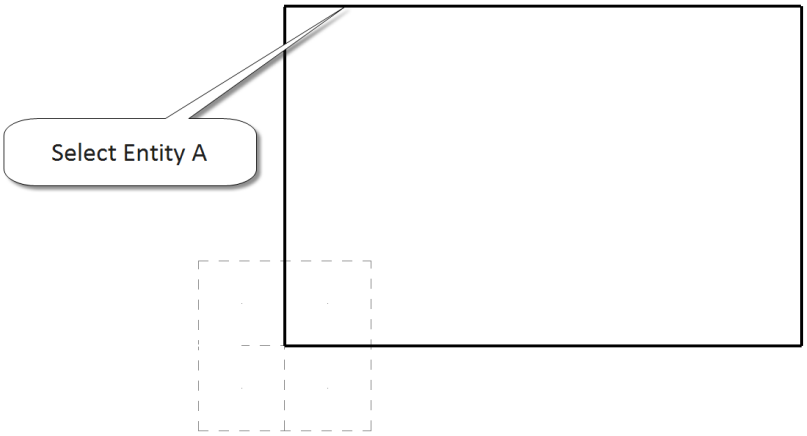
Step Preview:



Create

- **Line.**
- **Parallel.**
- [Select a line]: Select **Entity A** as shown in Figure: 3.0.1.

Figure: 3.0.1



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

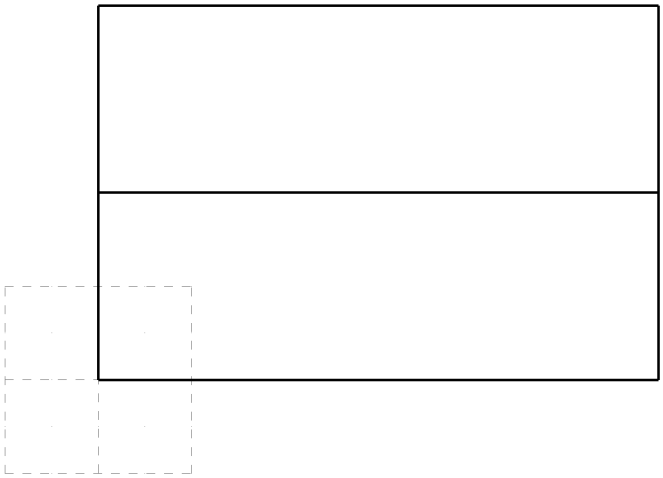
NOTE: The color of the geometry is cyan which means that the entity is "live" and you can still change the line parameters if needed.

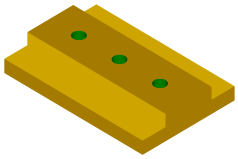
- Enter the **Distance** **1.0"** (press **Enter**).

NOTE: To continue using the same command you can either select the **Apply** button or press **Enter**. To exit the command you can either start a new command or select the **OK** button.

- Select the **OK** button to exit the command.
- The part will appear as shown in Figure: 3.0.2.

Figure: 3.0.2





STEP 4: DELETE THE CONSTRUCTION LINES

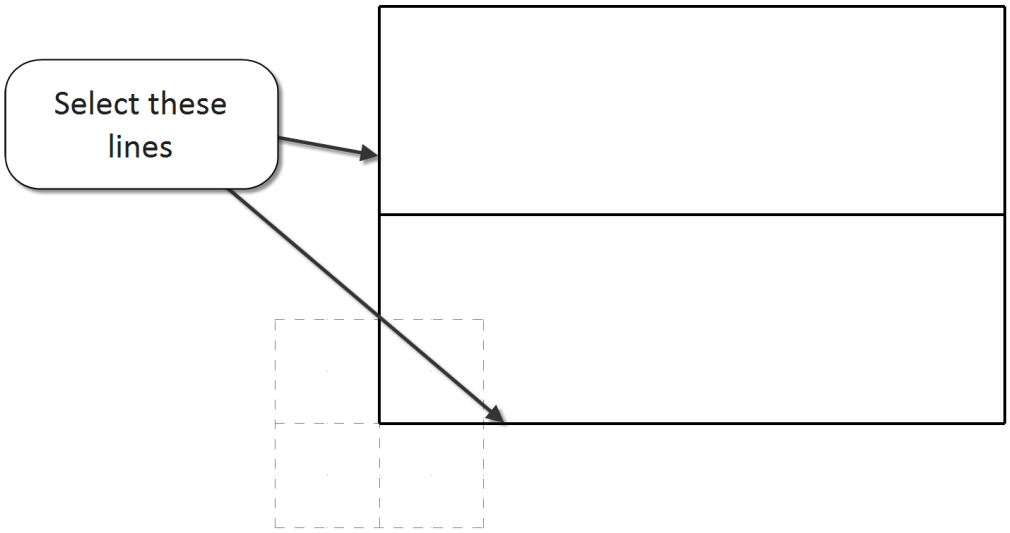
In this step you will delete two lines in preparation to mirror the geometry.

Step Preview:

Edit

- **Delete.**
- **Delete Entities.**
- Select the two lines as shown in Figure: 4.0.1.

Figure: 4.0.1

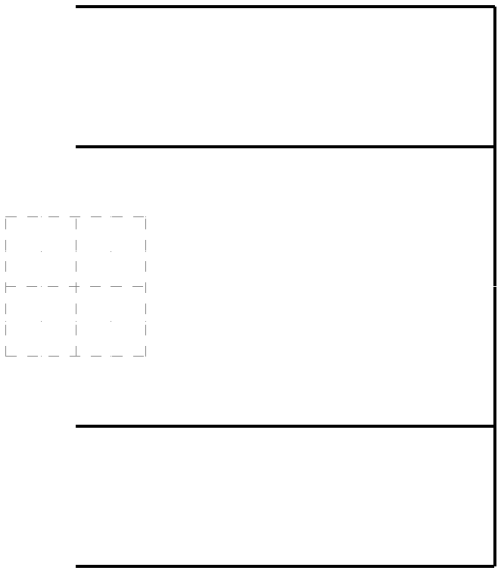


- Select the **Delete entity** icon.
- Select the **Repaint** icon to refresh the screen.

STEP 5: MIRROR THE GEOMETRY

In this step you will mirror the geometry to create the shape of the part.

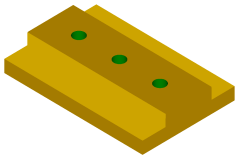
Step Preview:



5.1 Mirror about the X-Axis

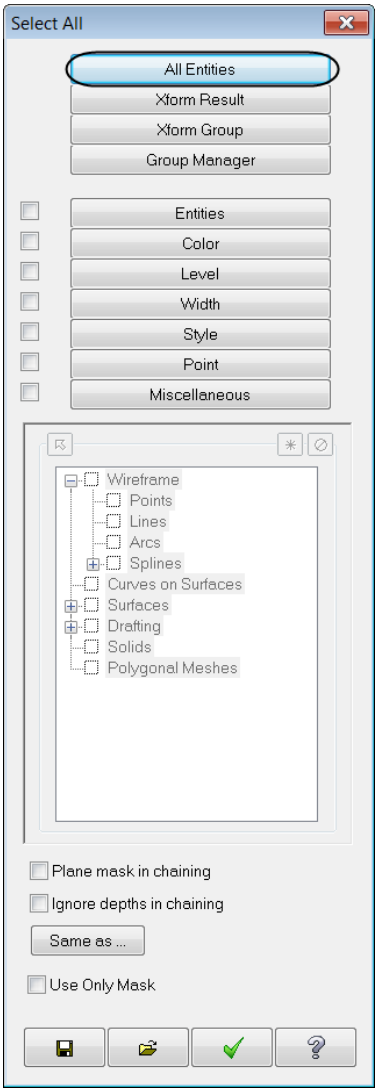
Xform

- **Mirror.**
- [Select the entities to mirror]: Click on the **All** button. All...

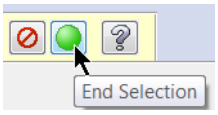


- Click on the **All Entities** button in the **Select All** dialog box as shown in Figure: 5.1.1.

Figure: 5.1.1



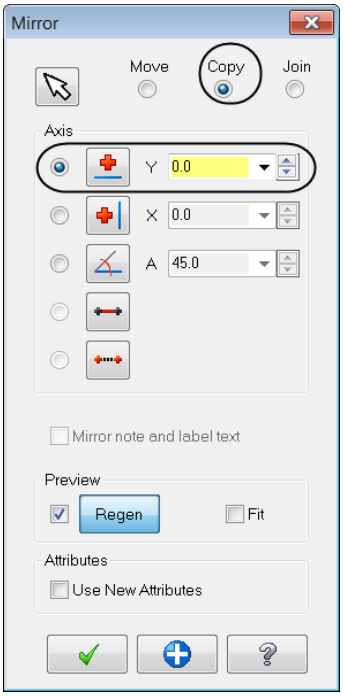
NOTE: The entire geometry is highlighted yellow (selection color).



- Click on the **End Selection** button or press Enter to finish the selection.
- Set the parameters to match the following dialog box (ensure **Copy** is picked to keep the original entities).

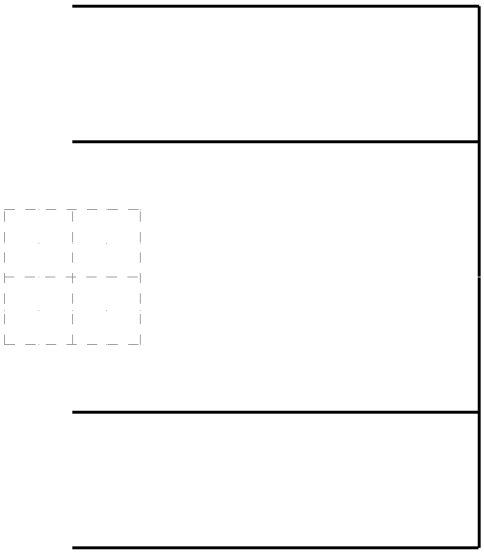
- Choose the option to mirror about the **X-axis** as shown in Figure: 5.1.2.

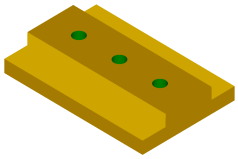
Figure: 5.1.2



- Select the **Apply** button to continue in the same command.
- Use the **Fit** icon to fit the drawing to the screen.
- The part will appear as shown in Figure: 5.1.3.

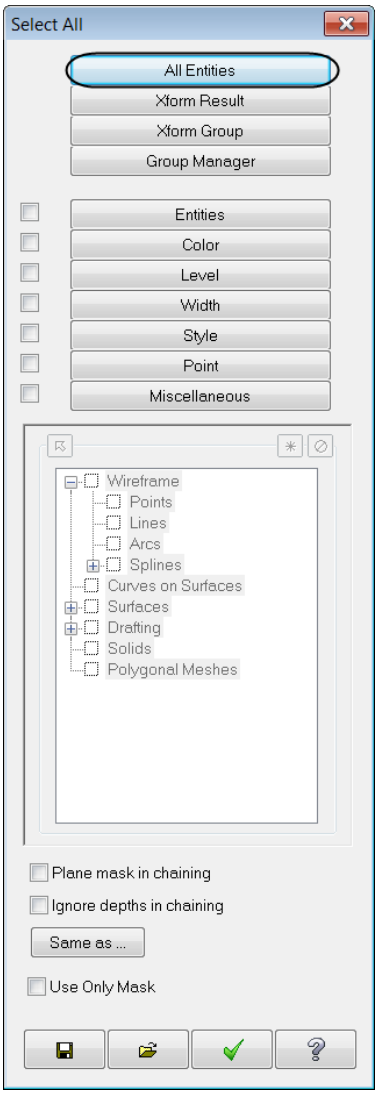
Figure: 5.1.3





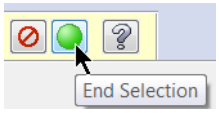
5.2 Mirror about the Y-Axis

- [Select the entities to mirror]: Click on the **All** button.

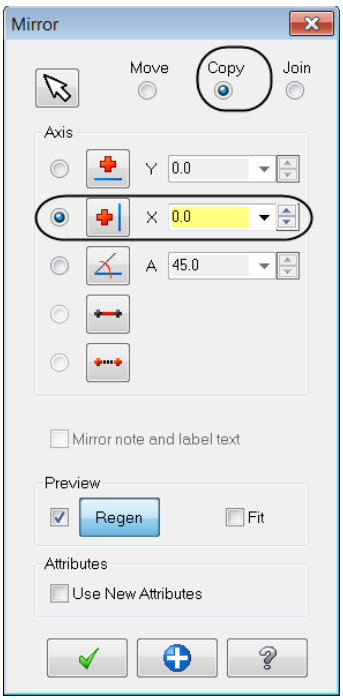


- Click on **All Entities** button in the **Select All** dialog box.

NOTE: The entire geometry is highlighted yellow (selection color).

- Click on the **End Selection** button. 

- Set the parameters to match the following dialog box (ensure **Copy** is picked to keep the original entities).
- Choose the option to mirror about the **Y-axis**.

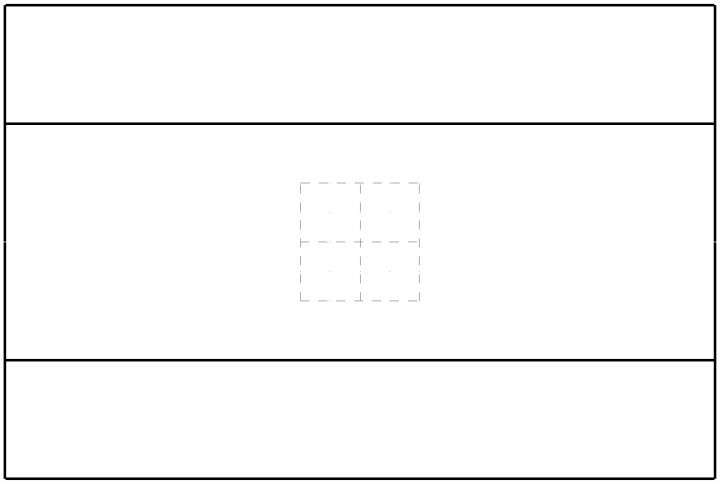


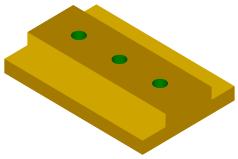
- Select the **OK** button to exit the mirror dialog box.
- Use the Fit icon to fit the drawing to the screen.

Screen

- **Clear Colors.**
- The part will appear as shown in Figure: 5.2.1.

Figure: 5.2.1

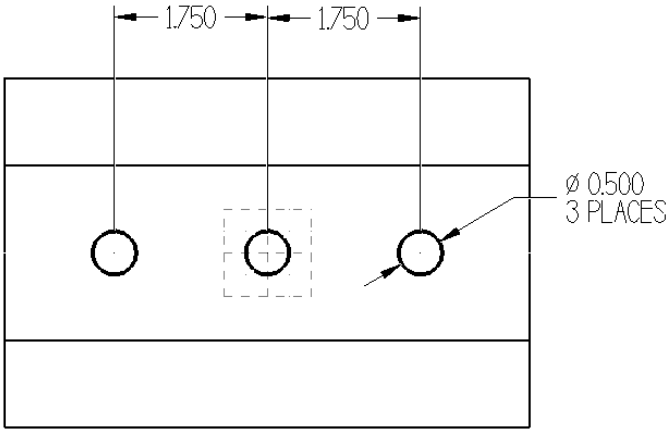




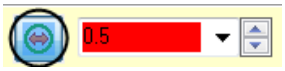
STEP 6: CREATE ARCS

In this step you will create points which will later be used to drill holes in the part.

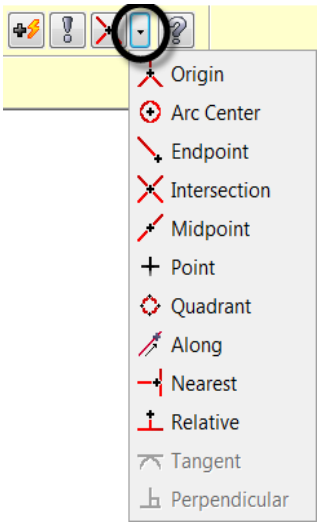
Step Preview:

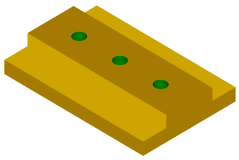


Create

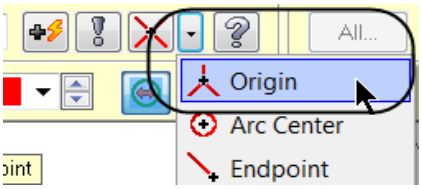
- **Arc.**
- **Circle Center Point.**
- Input a **Diameter** of **0.5"** in the ribbon bar and then select the icon to lock the value. 

- Select the **Fast Point** icon **from the AutoCursor Ribbon Bar.**
- Enter the dimensions **-1.75, 0** then hit **Enter** on your keyboard.
- Select the **Fast Point** again and enter the dimensions **1.75, 0** and hit **Enter** on your keyboard once again.
- From the **AutoCursor Ribbon Bar,** select the drop down arrow to view the **AutoCursor's Override** options**.**





- Select the option **Origin** from the list. This will place a circle with the center at the origin.

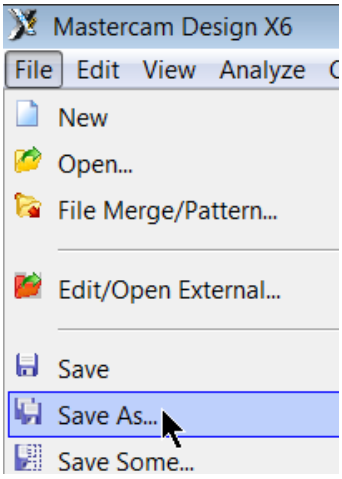


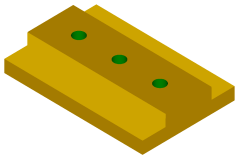
- Once complete choose the **OK** button to exit the command.

STEP 7: SAVE THE FILE

File

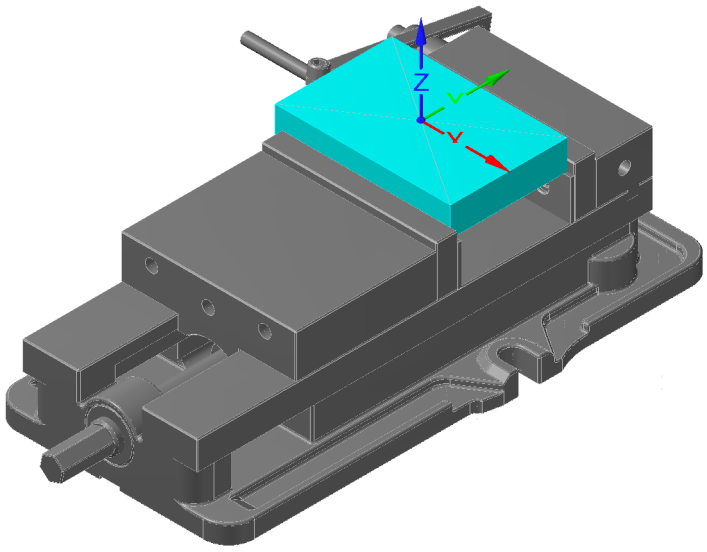
- **Save As.**
- File name: "Your Name_1".
- Select the **OK** button to save your file.





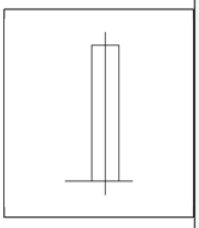
TOOLPATH CREATION

SUGGESTED FIXTURE:



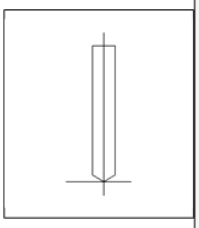
SETUP SHEET:

TOOL LIST



TYPE:	Endmill1 Flat	FLUTE LENGTH:	2.0
DIA OFFSET:	1	OVERALL LENGTH:	3.75
HOLDER:		CORNER RAD:	0.0
NUMBER:	1	# OF FLUTES:	4
LENGTH OFFSET:	1		
MFG TOOL CODE:	_		
CHUCK:	_		

#1 - 0.7500 ENDMILL1 FLAT - 3/4 FLAT ENDMILL



TYPE:	Drill	FLUTE LENGTH:	2.0
DIA OFFSET:	2	OVERALL LENGTH:	3.0
HOLDER:		CORNER RAD:	0.0
NUMBER:	2	# OF FLUTES:	2
LENGTH OFFSET:	2		
MFG TOOL CODE:	_		
CHUCK:	_		

#2 - 0.5000 DRILL - 1/2 DRILL

STEP 8: SELECT THE MACHINE AND SET UP THE STOCK

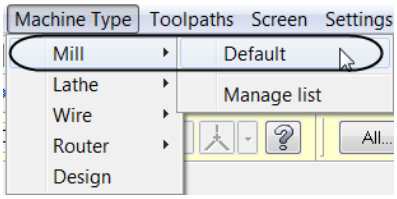
In Mastercam, you select a **Machine Definition** before creating any toolpaths. The **Machine Definition** is a model of your machines capabilities and features. It acts like a template for setting up your machine. The machine definition ties together three main components: The schematic model of your machines components, the control definition that models your control capabilities, and the post processor that will generate the required machine code (G-code). For a Mill Level 1 exercise (2D toolpaths) we need just a basic machine definition.

NOTE: For the purpose of this tutorial, we will be using the Default mill machine.

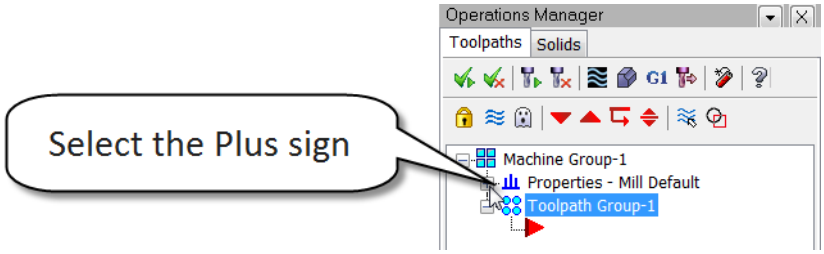
- To display the **Operations Manager** press **Alt** + **O**.
- Use the **Fit** icon to fit the drawing to the screen.

Machine type

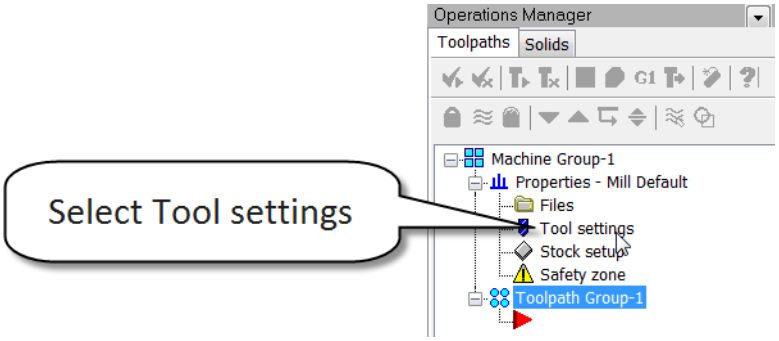
- **Mill.**
- **Default.**

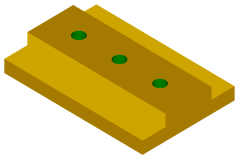


- Select the plus sign in front of **Properties** in the **Toolpaths Manager** to expand the **Toolpaths Group Properties.**



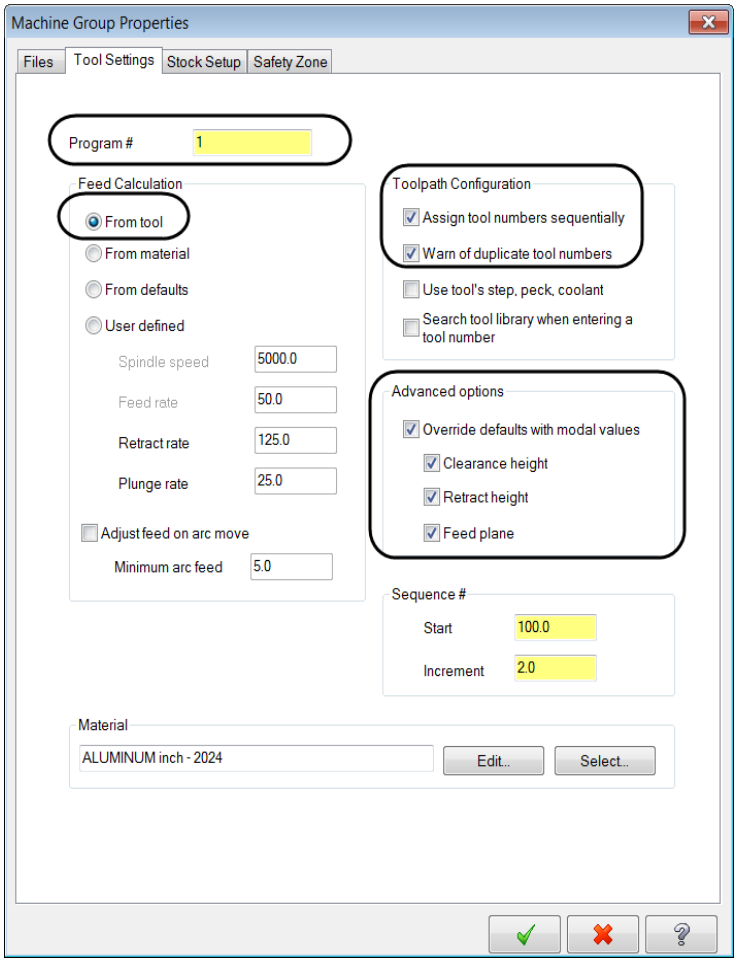
- Select **Tool Settings** to set the tool parameters.





- Change the parameters to match Figure: 8.0.1.

Figure: 8.0.1



Program # is used to enter a number if your machine tool requires a number for a program name.

Assign tool numbers sequentially allows you to overwrite the tool number from the library with the next available tool number. (First operation tool number 1; Second operation tool number 2, etc.)

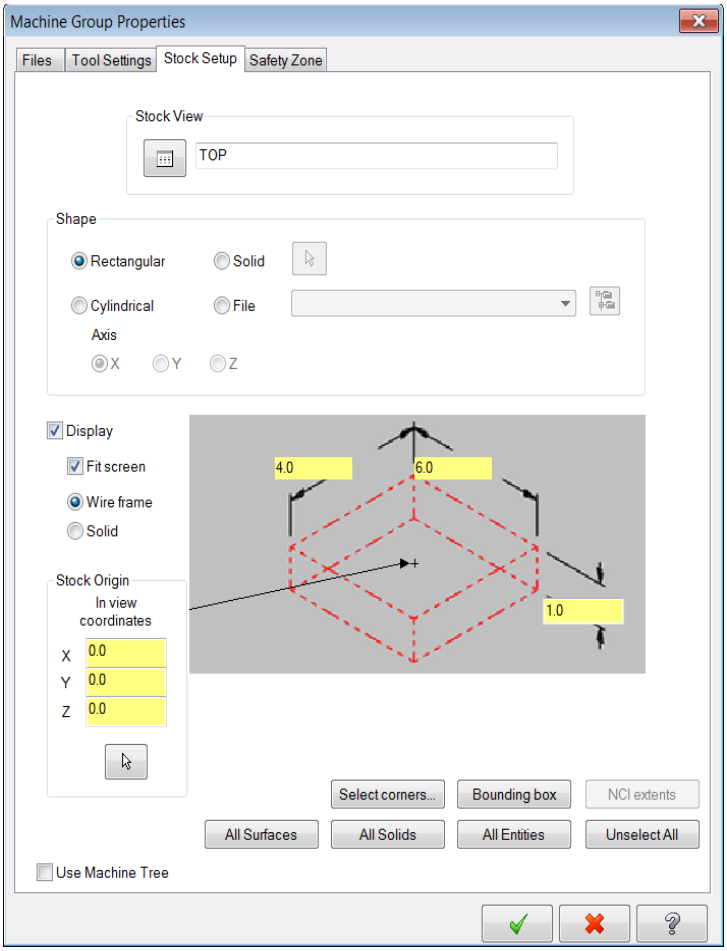
Warn of duplicate tool numbers allows you to get a warning if you enter two tools with the same number.

Override defaults with modal values enables the system to keep the values that you enter.

Feed Calculation set **From tool** uses feed rate, plunge rate, retract rate and spindle speed from the tool definition.

- Select the **Stock setup** tab to define the stock.
- Set the stock shape to **Rectangular** if needed and enter the stock dimensions as shown in Figure: 8.0.2.

Figure: 8.0.2

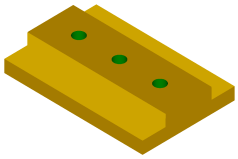


The **Stock Origin** values adjust the positioning of the stock, ensuring that you have equal amount of extra stock around the finished part.

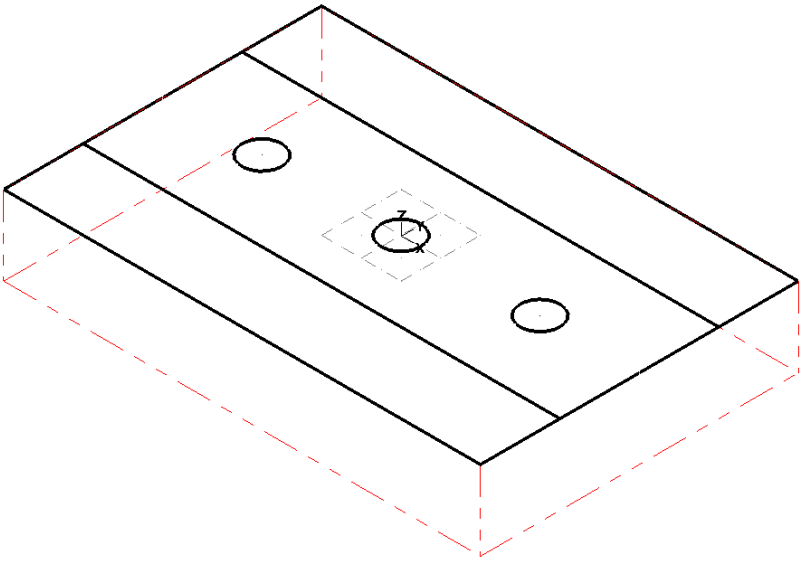
Display options allow you to set the stock as **Wireframe** and to fit the stock to the screen. (Fit Screen)

NOTE: The **stock** model that you create can be displayed with the part geometry when viewing the file or the toolpaths, during backplot, or while verifying toolpaths. In the graphics, the plus shows you where the stock origin is. The default position is the middle of the stock. Click on the corner of the part to set it as the stock origin.

- Select the **OK** button to exit **Machine Group Properties**.
- Select the **Isometric** view from the graphics view toolbar to see the stock.
- Use the **Fit** icon to fit the drawing to the screen.



- The stock model will appear as shown.



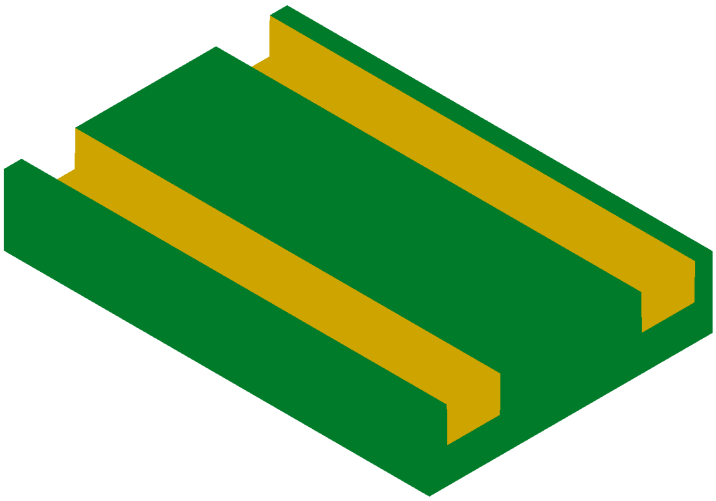
NOTE: The stock is not geometry and can not be selected.

- Select the **Top** view from the view toolbar to see the part from the top.

STEP 9: MACHINE THE TWO STEPS USING OPEN CONTOUR TOOLPATH

Contour toolpaths remove the material along a path defined by a chain of curves. Contour toolpaths only follows a chain; they do not clean out an enclosed area.

Toolpath Preview:

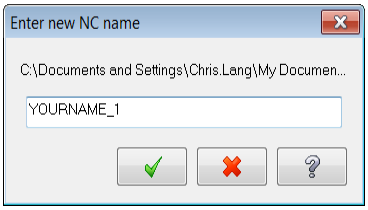


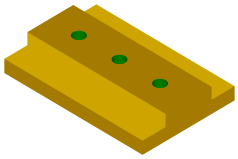
9.1 Chain Selection

A chain of entities consists of one or more entities linked together in order and direction. The distance between the endpoints of two consecutive entities of the chain has to be equal or less than the chaining tolerance (0.0001"). In an open chain, the start point is placed at the end of the chain closest to the selection point and the chain direction points to the opposite end of the chain. See the **User Notes** chapter for more information on chaining.

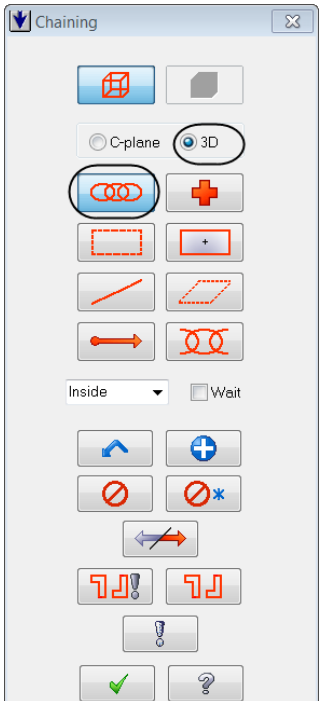
Toolpaths

- **Contour.**
- If a prompt appears to **Enter new NC name**, select the **OK** button to accept the default.



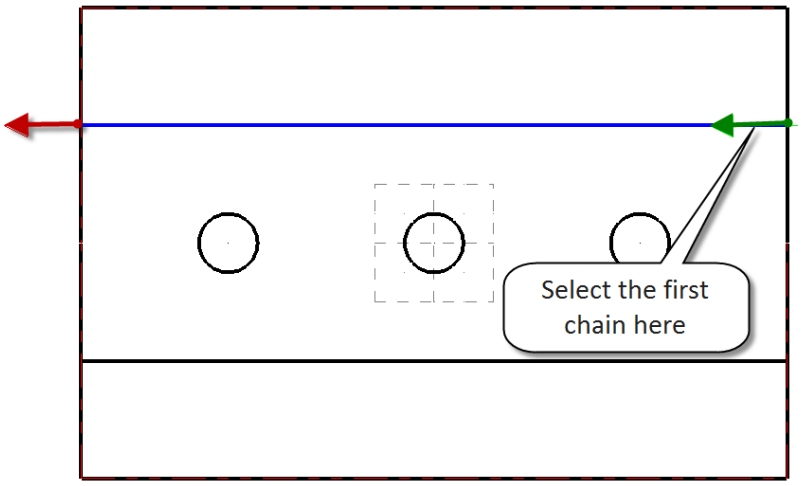


- Enable the **Chain** button in the chaining dialog box to be able to select just the outside contour.



- Select the first chain as shown in Figure: 9.1.1.

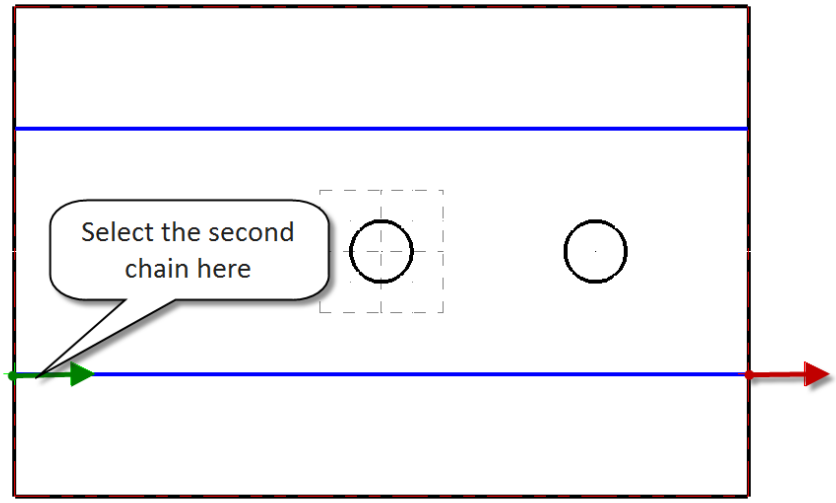
Figure: 9.1.1



NOTE: The direction of the chain determines the cutter compensation side (Left or Right) to orient the tool on the proper side of the geometry. For our example be sure to chain the contour in a **CCW** direction. Otherwise, select the **Reverse** button.

- To be able to machine both contours in one operation select the second chain as shown in Figure: 9.1.2.

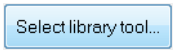
Figure: 9.1.2



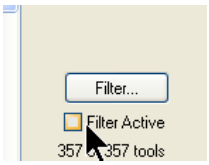
- Select the **OK** button to exit **Chaining**.
- In the **Toolpath Type** page, the **Contour** icon will be selected.

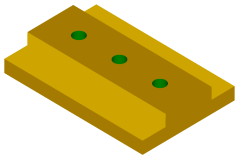
NOTE: Mastercam updates the pages as you modify them and then marks them, in the **Tree view list,** with a green check mark. Pages that are not changed are marked with a red circle and slash.

9.2 Select a 3/4" Flat Endmill from the library and set the Tool Parameters

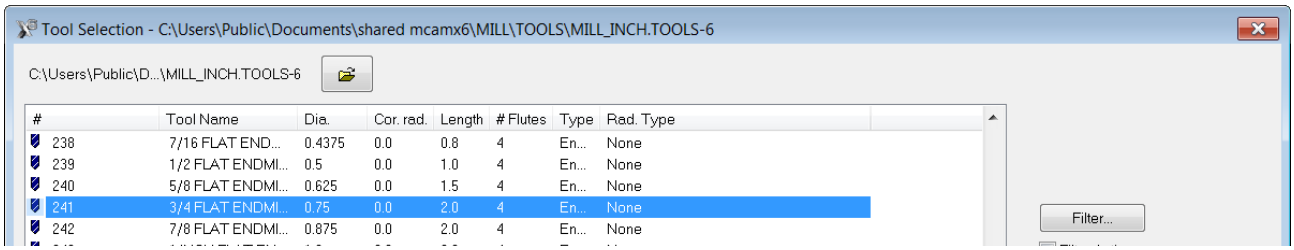
- Select **Tool** from the **Tree view** list.
- Click on the **Select library tool** button. 

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



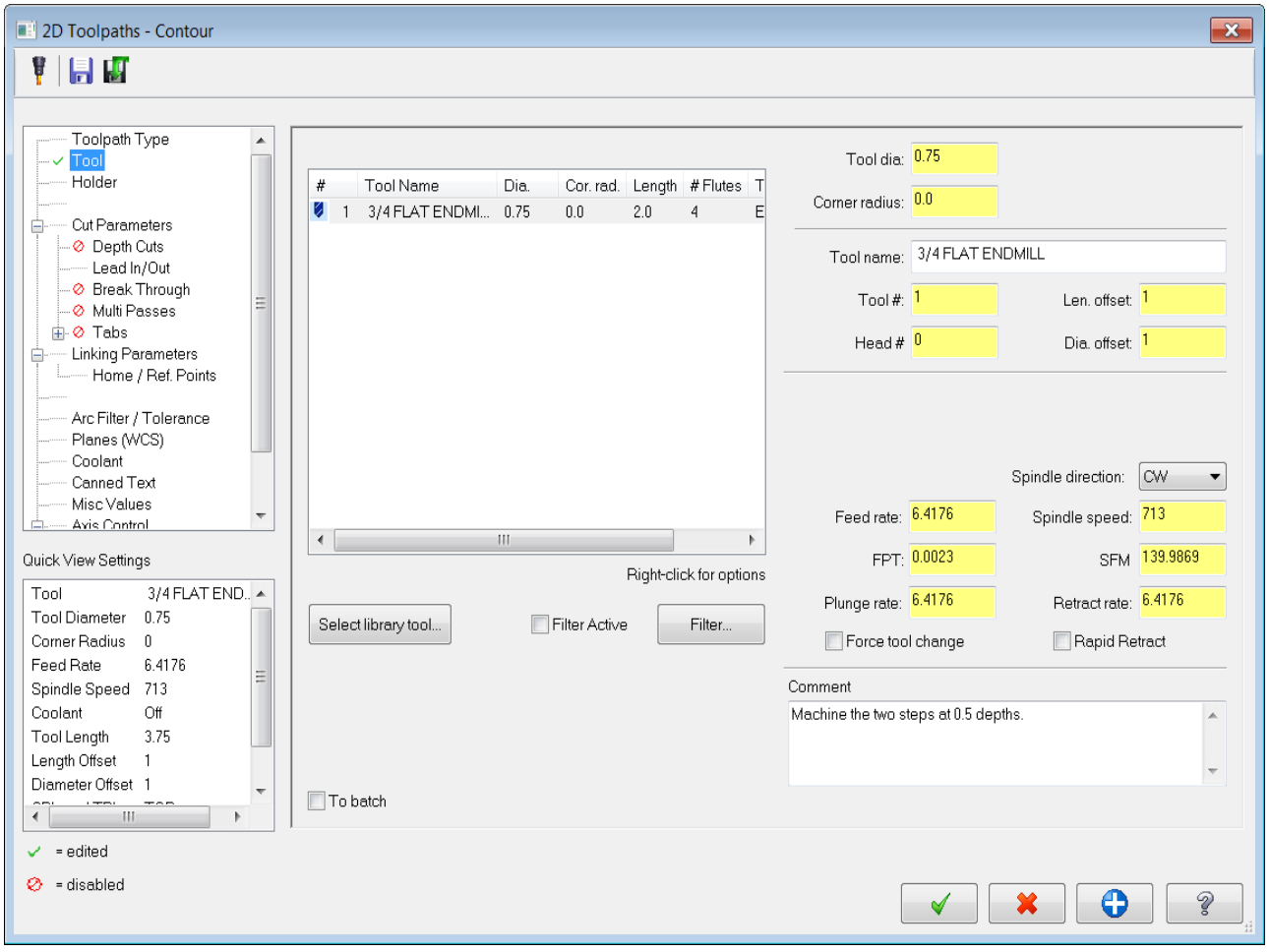


- Scroll down and select the **3/4" Flat Endmill** (# 241) as shown.



- Select the tool in the **Tool Selection** page and then select the **OK** button to exit.
- Make all the necessary changes as shown in Figure: 9.2.1.

Figure: 9.2.1



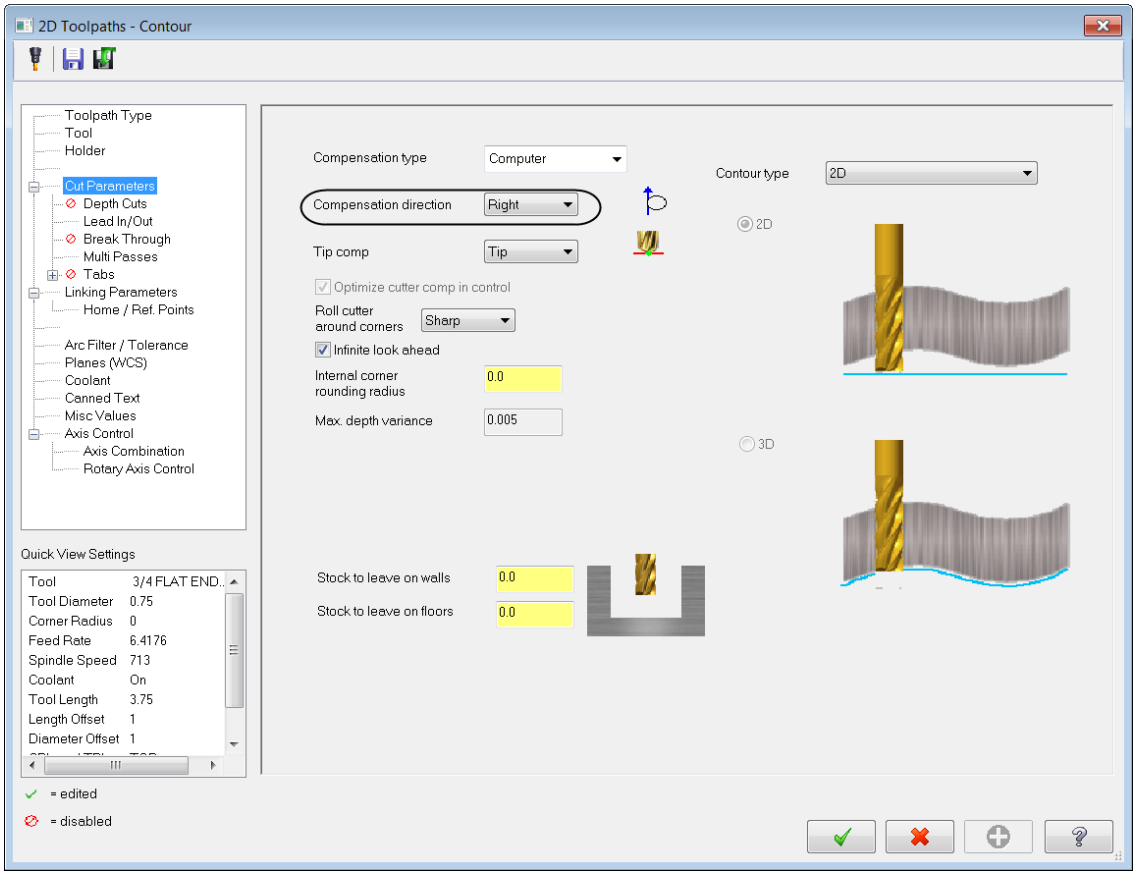
The **Feed rate**, **Plunge rate**, **Retract rate** and **Spindle speed** are selected from tool definition. You may change these values as per your part material and tools.

In the **Comment** field enter a comment to help identify the toolpath in the **Toolpaths/Operations Manager** such as the one shown above.

9.3 Set the Cut Parameters

- From the **Tree view list**, select **Cut Parameters** and make the changes as shown in Figure: 9.3.1.

Figure: 9.3.1



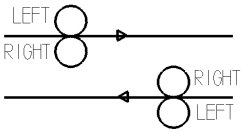
Compensation type set to **Computer** allows Mastercam to compensate the toolpath based on the tool diameter and does not output G41/G42 in the code.

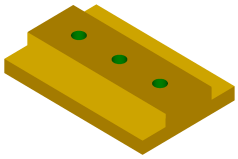
Compensation direction set to **Right** compensates the toolpath to the right of the chain based on the chaining direction. See the Figure: 9.3.2.

Roll cutter around corners set to **Sharp** inserts arc moves around corners in the toolpath. The radius of the arc moves is equal with the radius of the tool.

Infinite look ahead prevents the toolpath from crossing itself (fish tail).

Figure: 9.3.2

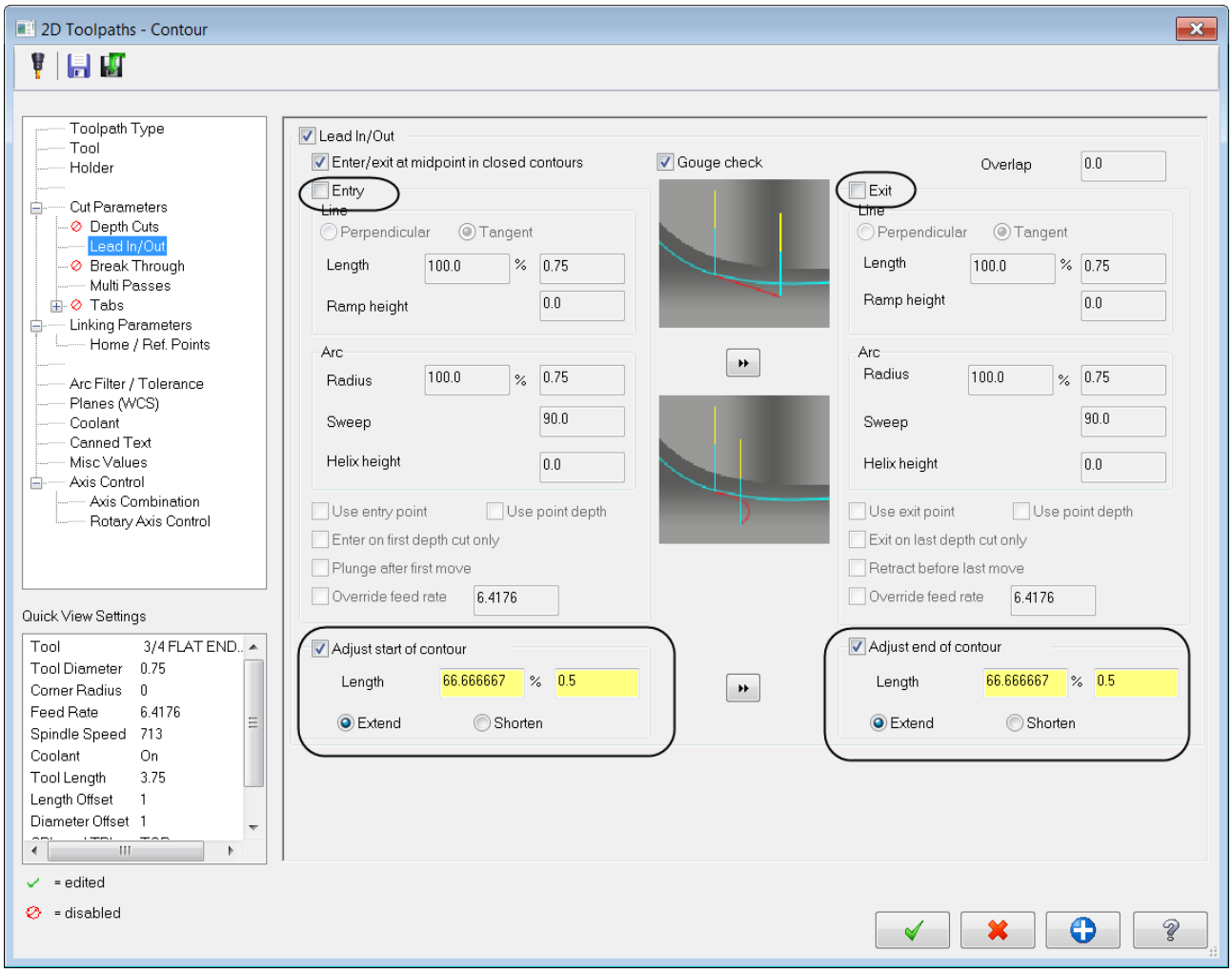




9.4 Set the Lead In/Out

- From the **Tree view list**, select **Lead in/out** and change the parameters as shown.
- Make sure that you disable the **Entry** and the **Exit** options and enable **Adjust start/end of contour** as shown in Figure: 9.4.1.

Figure: 9.4.1



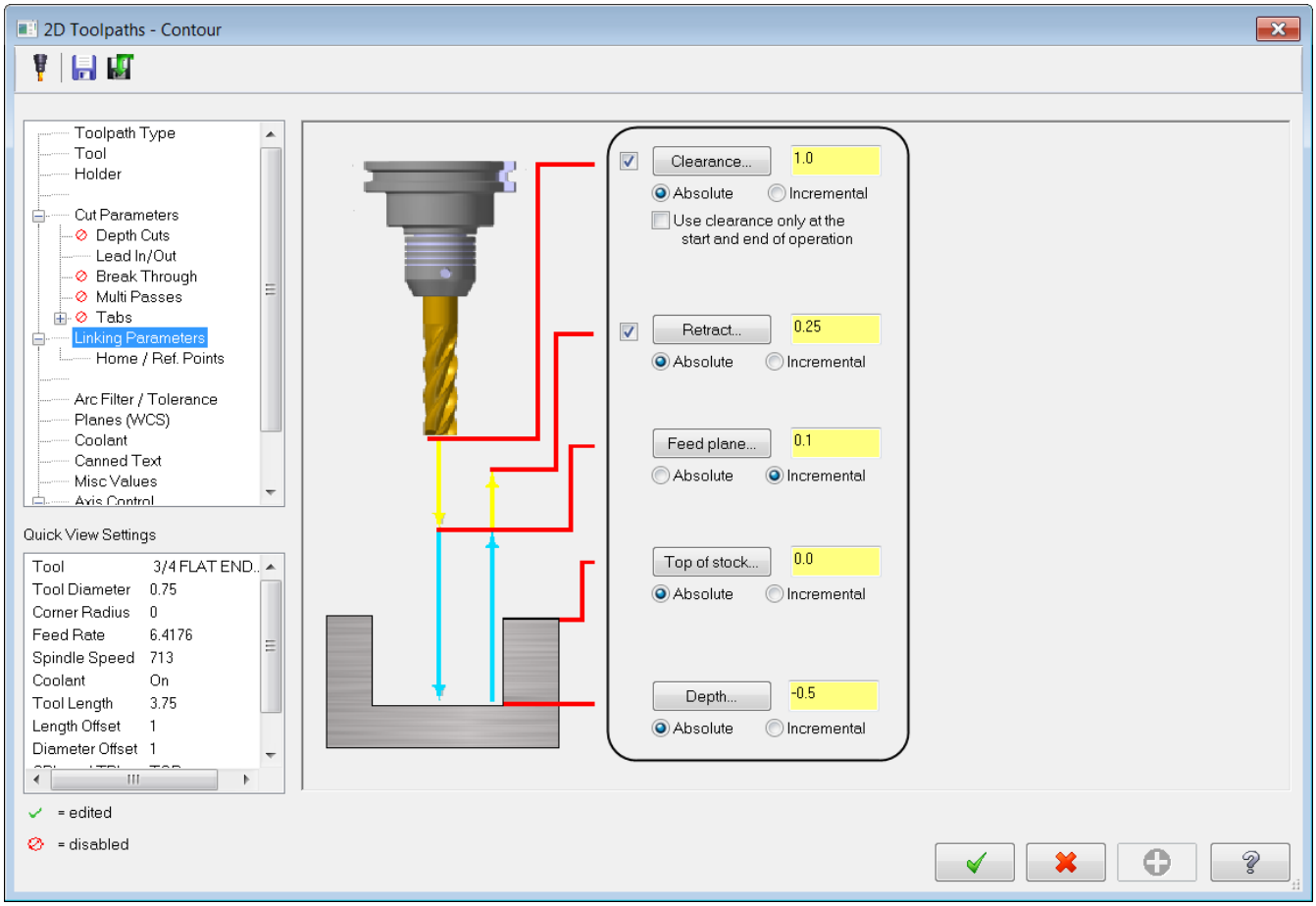
Lead In/Out allows you to select a combination of a **Line** and an **Arc** at the beginning and/or end of the contour toolpath for a smooth entry/exit while cutting the part.

Adjust start/end of contour moves the starting/ending position in open contours by adding (Extend) or removing (Shorten) the specified length.

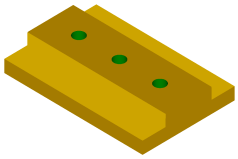
9.5 Set the Linking Parameters

- From the **Tree view list**, select **Linking Parameters** and change the parameters as shown Figure: 9.5.1.

Figure: 9.5.1



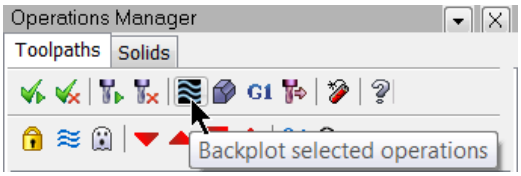
- Select the **OK** button to exit the contour parameters.



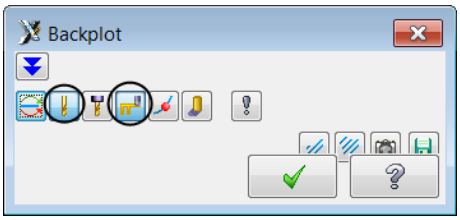
STEP 10: BACKPLOT THE TOOLPATHS

Backplotting shows the path the tools take to cut the part. This display lets you spot errors in the program before you machine the part. As you backplot toolpaths, Mastercam displays 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 (they will appear pushed down) to see the tool and the rapid moves.



- Select the **Isometric** view from the view toolbar to see the stock.

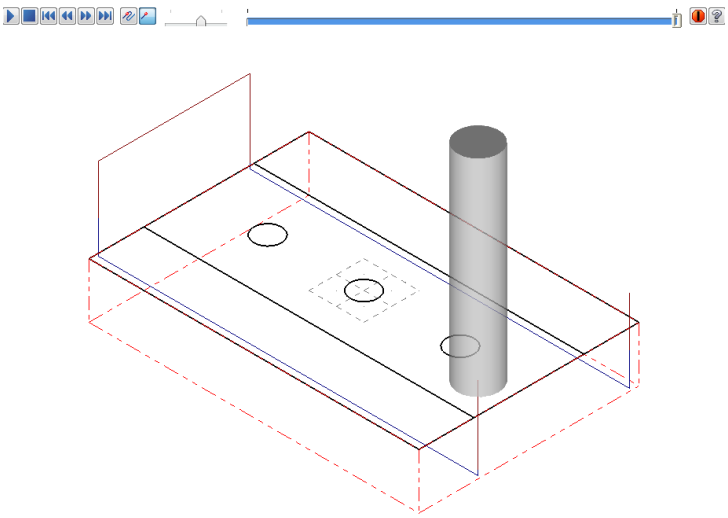
- Select the **Fit** button.

- 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 in the **VCR** bar.

- The part should appear as shown in Figure: 10.0.1.

Figure: 10.0.1

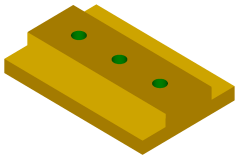


- Select the **OK** button to exit **Backplot**.

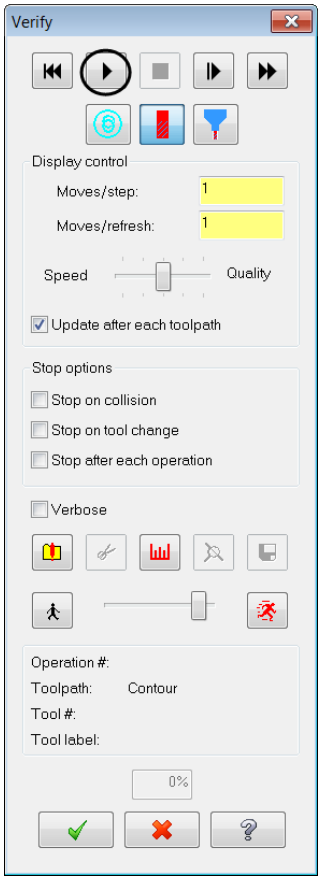
STEP 11: VERIFY THE TOOLPATH

Verify allows you to use a solid model to simulate the machining of a part. The model created by verification represents the surface finish, and shows collisions, if any exist.

- In the **Operations Manager** choose the **Contour operation.**
- Choose the **Verify selected operations** icon from the **Operations Manager**.
- Set the **Verify speed** by moving the slider bar in the speed control bar as shown.

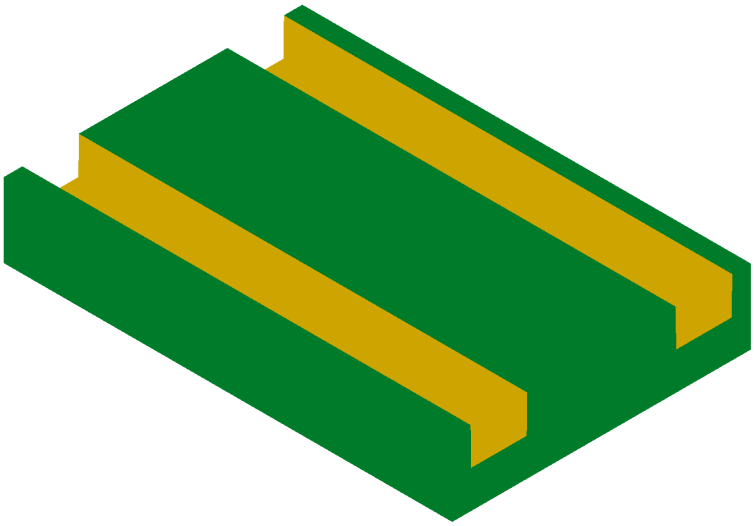


- Select the **Play** button to start simulation.

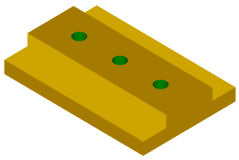


- The part will appear as shown in Figure: 11.0.1.

Figure: 11.0.1



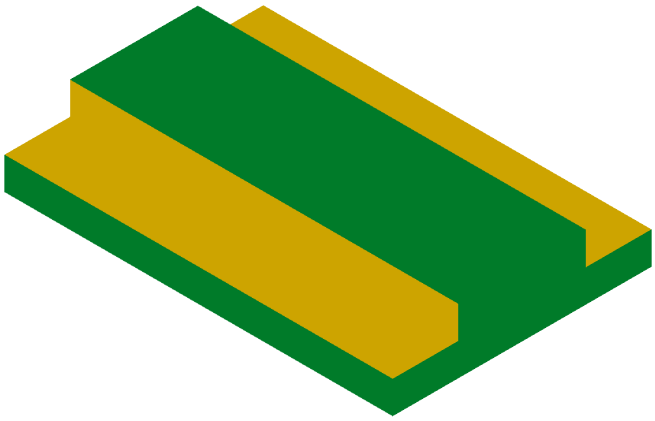
- Select the **OK** button to exit **Verify**.



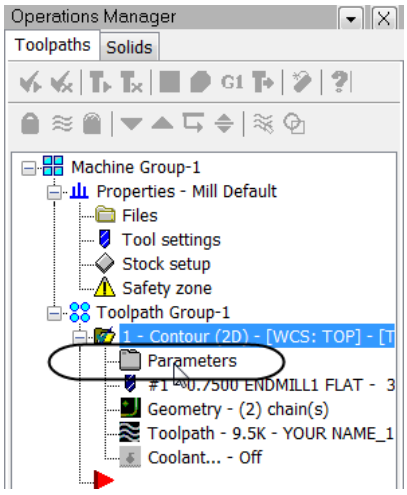
STEP 12: ADD MULTI PASSES

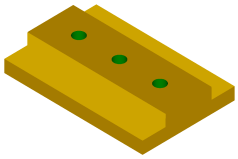
Multi Passes lets you define multiple cutting passes. The tool approaches the part geometry at the cutting depth in steps instead of cutting right to the part geometry.

Toolpath Preview:



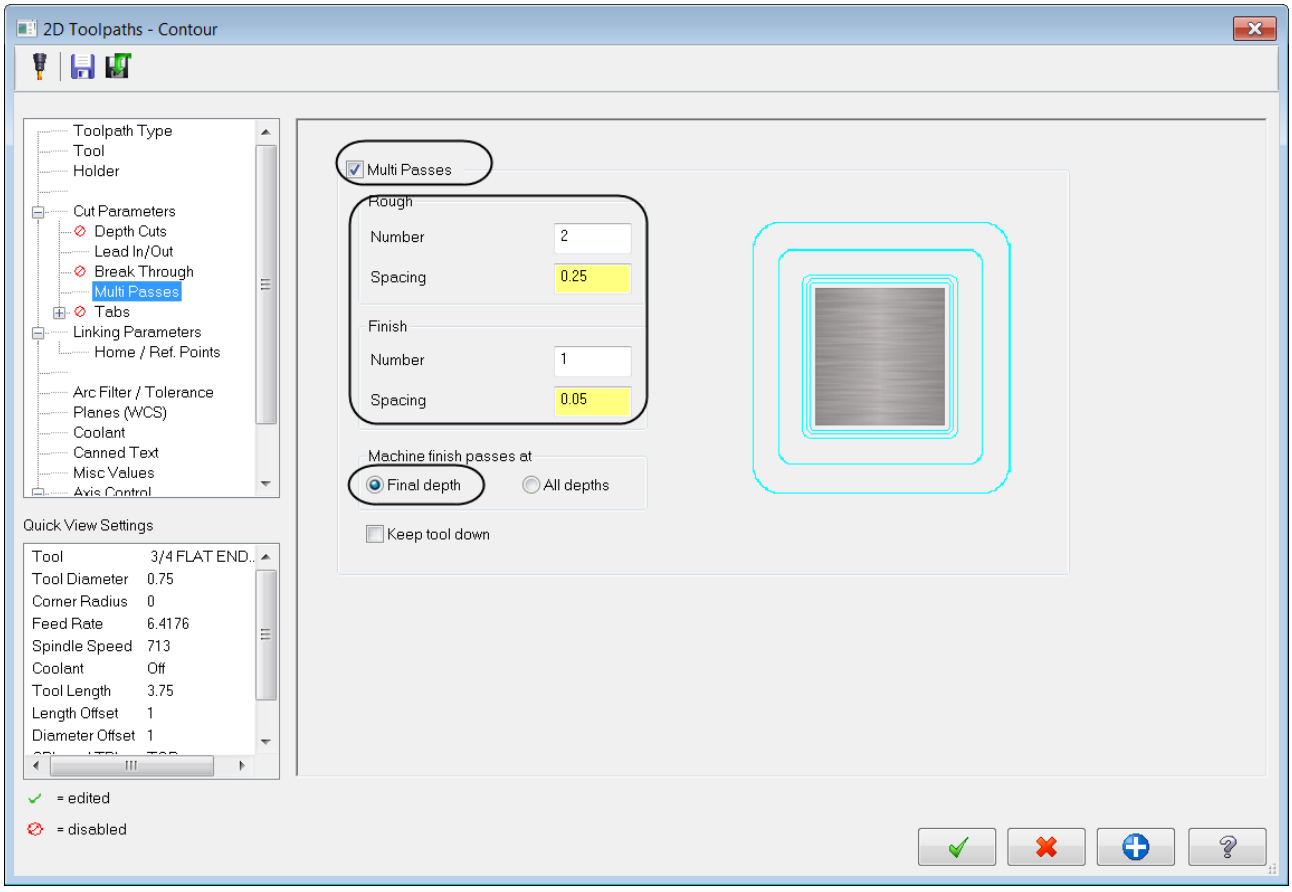
- Select Parameters for the contour toolpath.





- From the **Tree view list** select **Multi Passes** and make the necessary changes as shown in Figure: 12.0.1.

Figure: 12.0.1



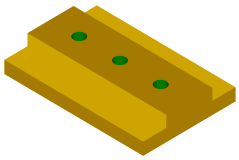
Rough Passes/Finish Passes allows you to enter the number of passes and amount of material you want to remove per pass.

Machine finish passes at: performs the finish pass at the final depth. All depths performs the finish passes at all depths. This option only applies if depth cuts are enabled.

- Once the changes have been made to the parameters select the **OK** button to exit the **2D Toolpaths- Contour** parameters.

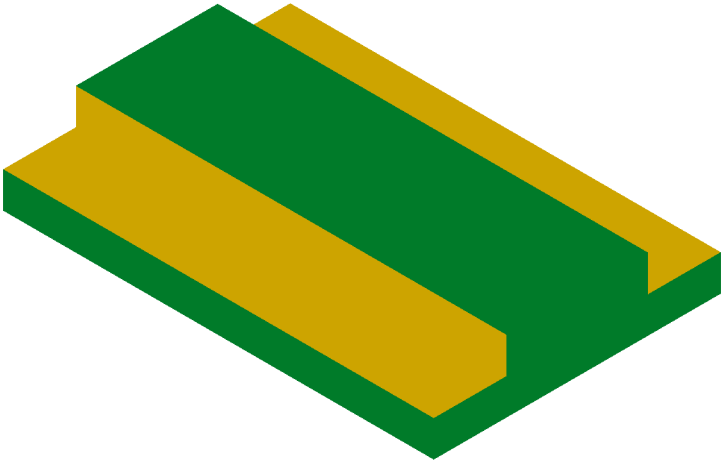
NOTE: We made a change to the Contour parameters, this toolpath is now marked as a dirty toolpath/ operation. A dirty toolpath or operation occurs anytime you make changes to the toolpath parameters.

- Select the **Regenerate all dirty operations** button to recalculate the changes made to the toolpath.
- Once the operation has been regenerated **Backplot** and **Verify** your toolpath page 38 to review these procedures.



- The part will appear as shown in Figure: 12.0.2.

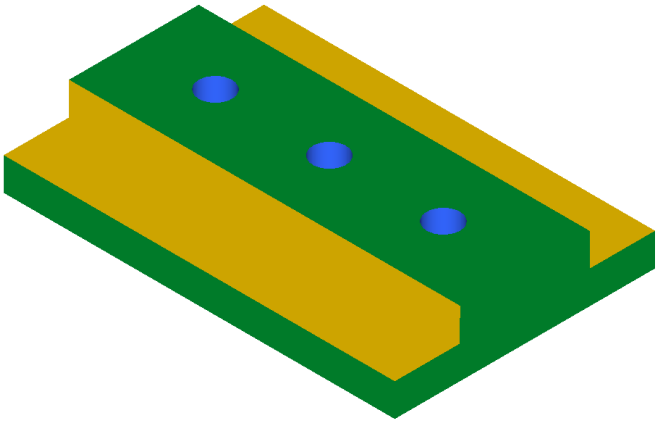
Figure: 12.0.2



STEP 13: DRILL 3 HOLES

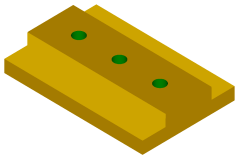
A drill toolpath is used for cutting holes in material.

Toolpath Preview:



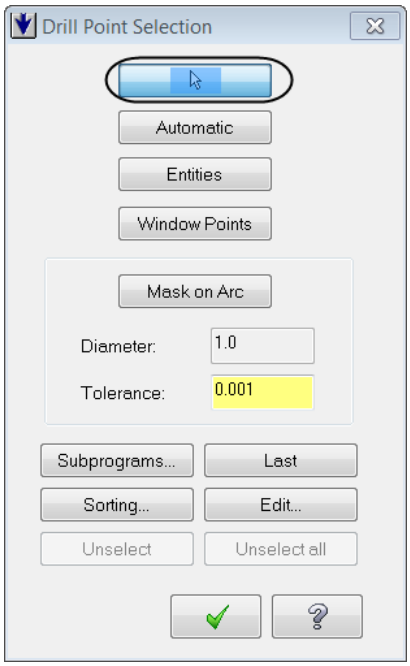
Toolpaths

- **Drill.**

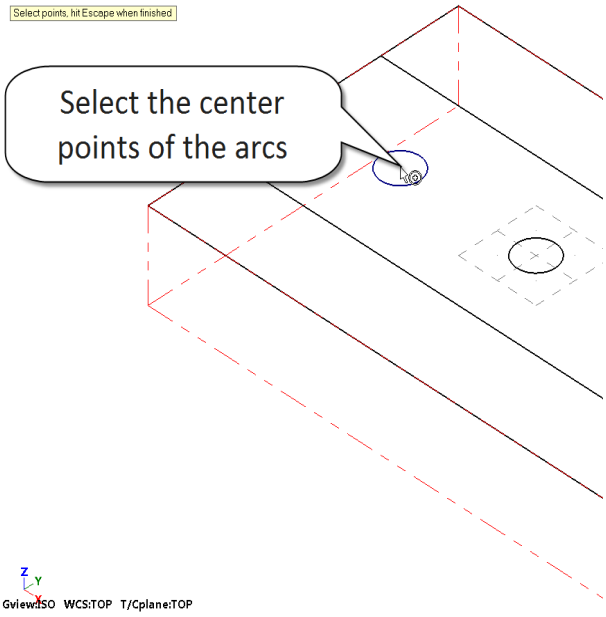


- The **Drill point Selection** dialog box will appear prompting you to select the arc center points as shown.

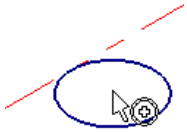
Figure: 13.0.1

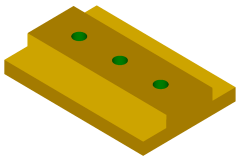


- Select the center of all three arcs on the part.



- Ensure the **Auto Cursor** icon appears as shown before selecting the points as shown.



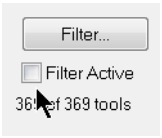


- Once the 3 arcs have been selected choose the **OK** button to exit the **Drill Point Selection** dialog box.
- In the **Toolpath Type** page, the **Drill** icon will be selected.

13.1 Select a 1/2" Drill from the library and set the Tool Parameters

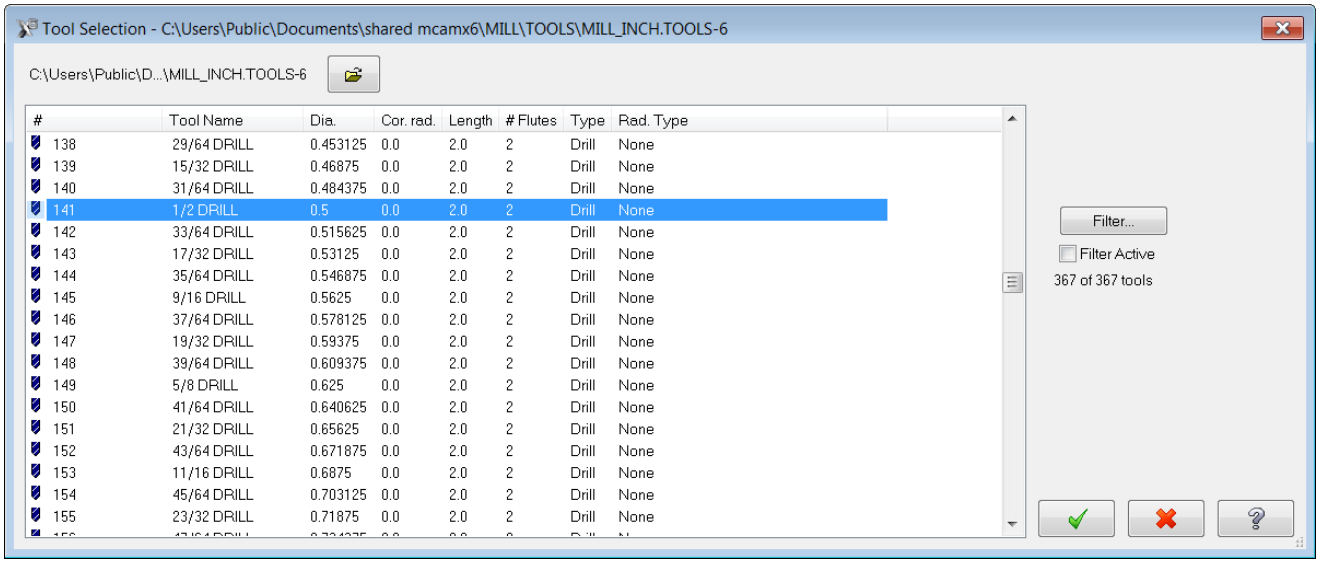
- Select **Tool** from the **Tree view list**.
- Click on the **Select library tool** button.

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

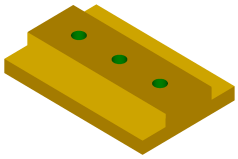


- Scroll down and select the **1/2" Drill** (#141) as shown in Figure: 13.1.1.

Figure: 13.1.1

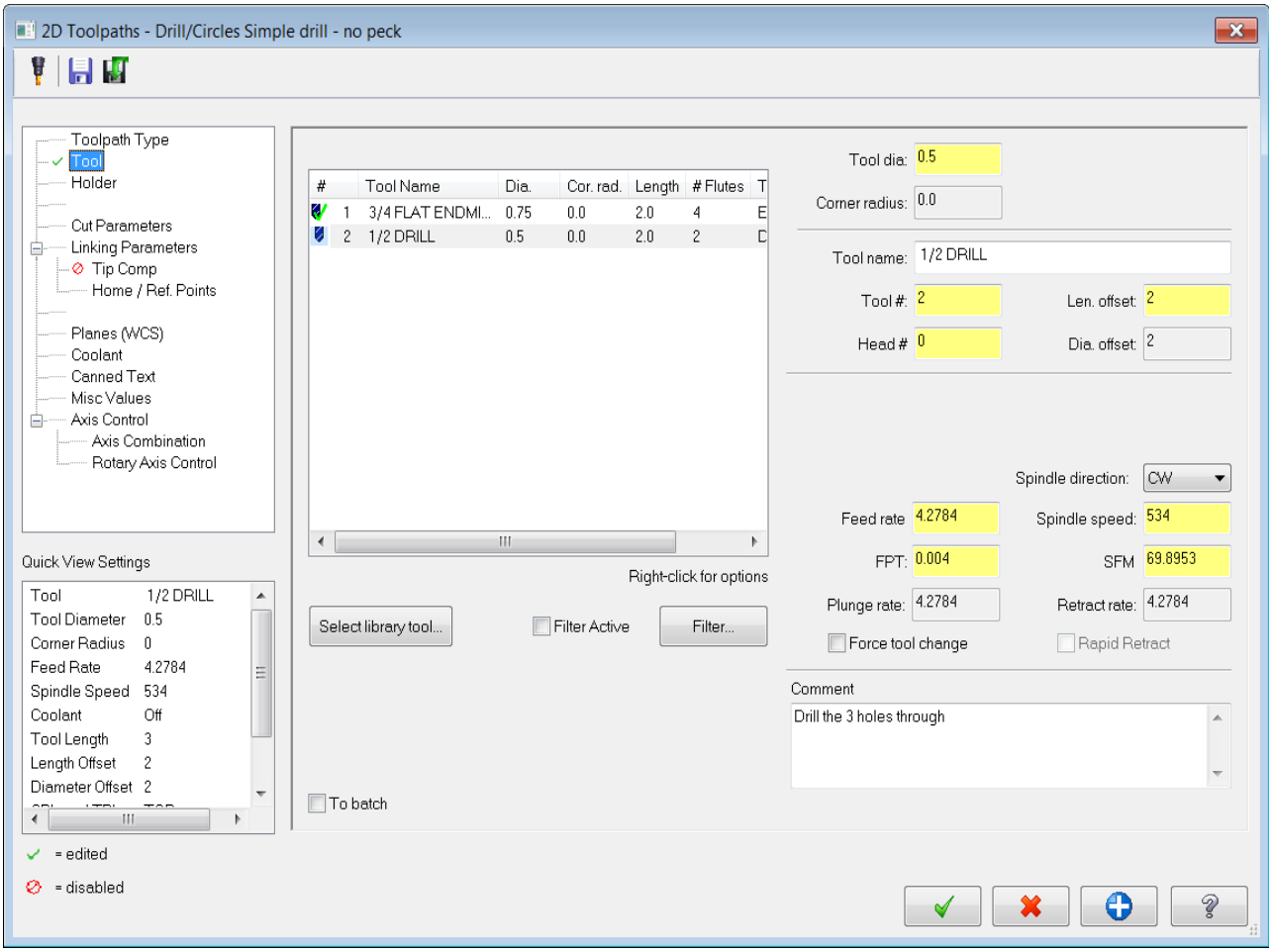


- Select the tool in the **Tool Selection** page and then select the **OK** button.



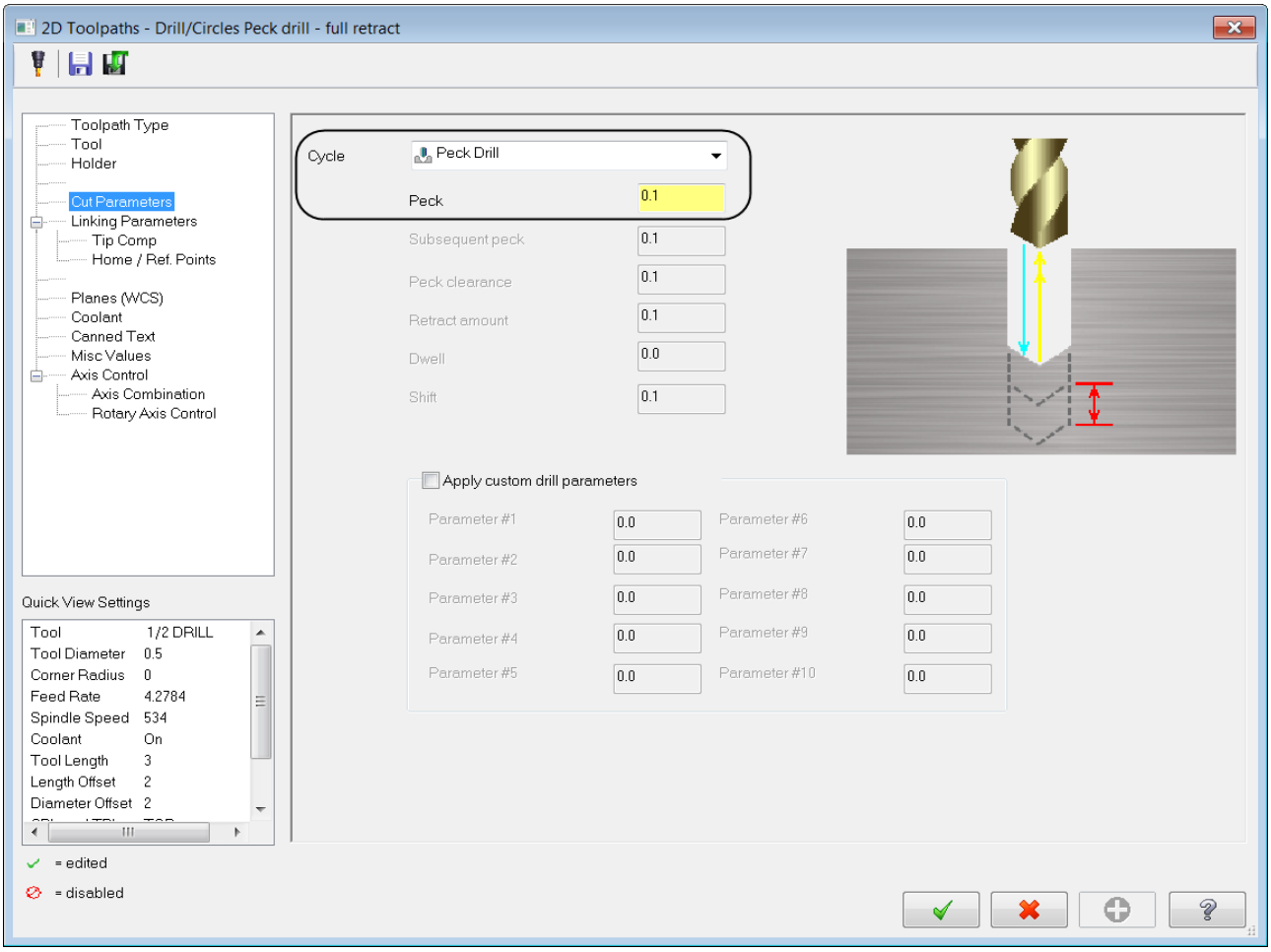
- Make the necessary changes as shown in Figure: 13.1.2.

Figure: 13.1.2

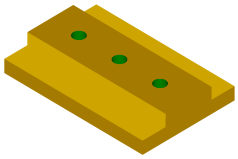


- Select the **Cut Parameters** page and change the **Cyclce** to **Peck Drill** as shown in Figure: 13.1.3.

Figure: 13.1.3

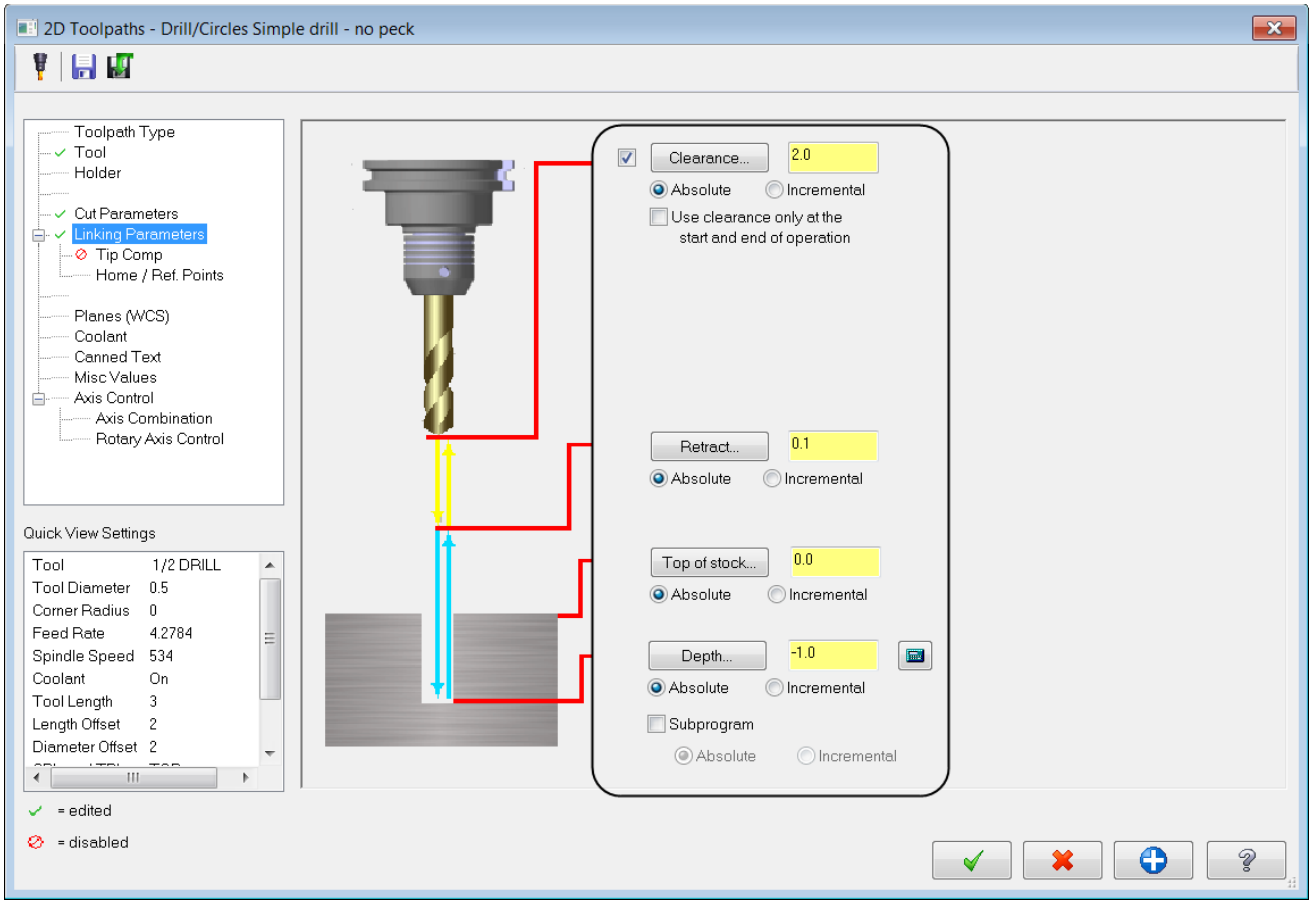


Peck Drill retracts fully out of the drill hole to remove material chips. Often used with chips that are hard to remove.



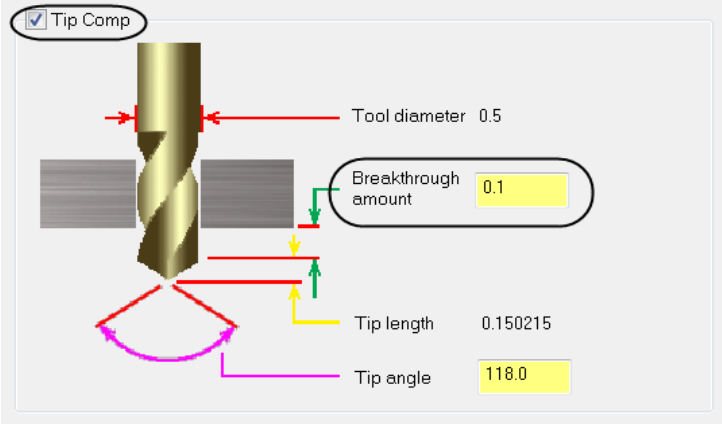
- Choose **Linking parameters**, enable **Clearance** and set the **Depth** to **-1.0"** as shown in Figure: 13.1.4.

Figure: 13.1.4

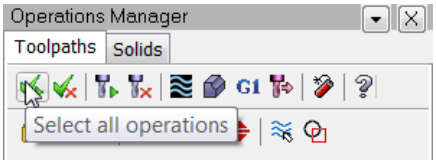


- Choose the option **Tip Comp,** enable this option and make the necessary changes as shown in Figure: 13.1.5.

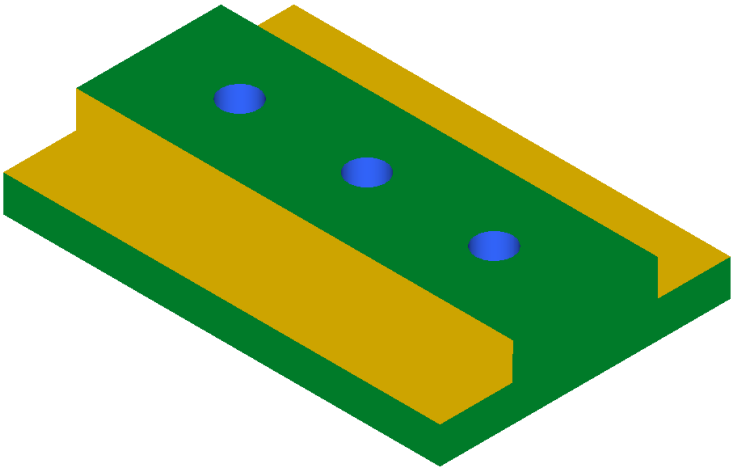
Figure: 13.1.5

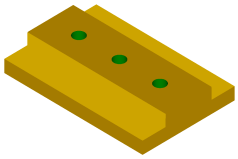


- Select the **OK** button to generate the toolpath.
- To **Backplot** and **Verify** your toolpath page 38 to review these procedures.
- To select both operations click on the **Select all operations** button.



- The finished part will appear as shown.



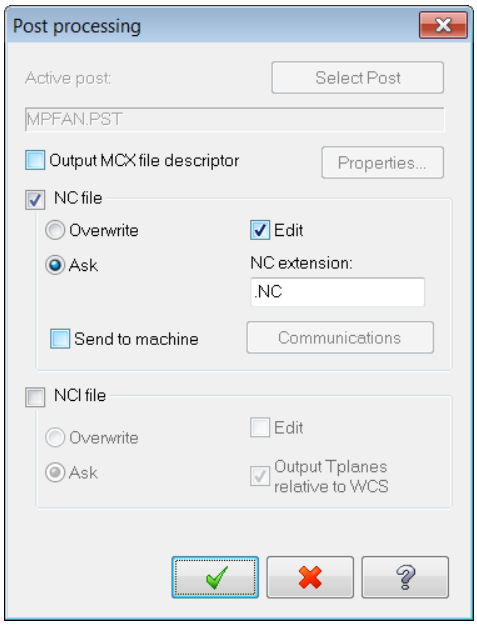


STEP 14: POST THE FILE

Posting refers to a process where the toolpaths in Mastercam are converted to a format that can be understood by the machines control, specifically the G-codes. In most cases every machine will require its own post processor; customized to produce code formatted to meet the machines exact requirements.

- Ensure all operations are selected, if they are not use the button **Select all operations** in the **Operations Manager.**
- Select the **Post selected operations** button from the **Operations Manager.**
- In the **Post processing** window make the necessary changes as shown in Figure: 14.0.1.

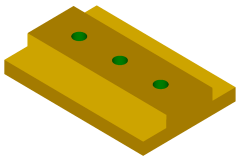
Figure: 14.0.1



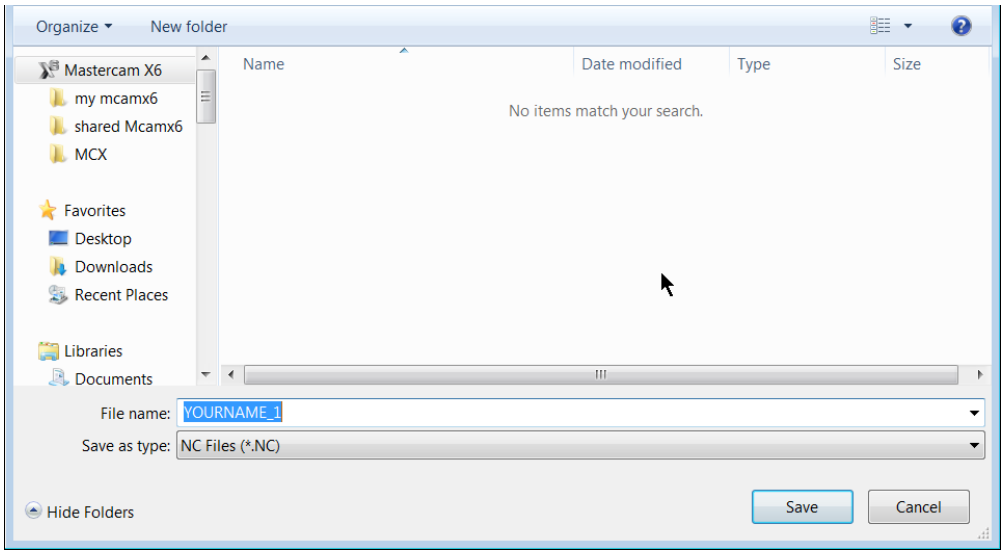
NC File enabled allows you to keep the NC file and to assign the same name as the MCX file.

Edit enabled allows you to automatically launch the default editor.

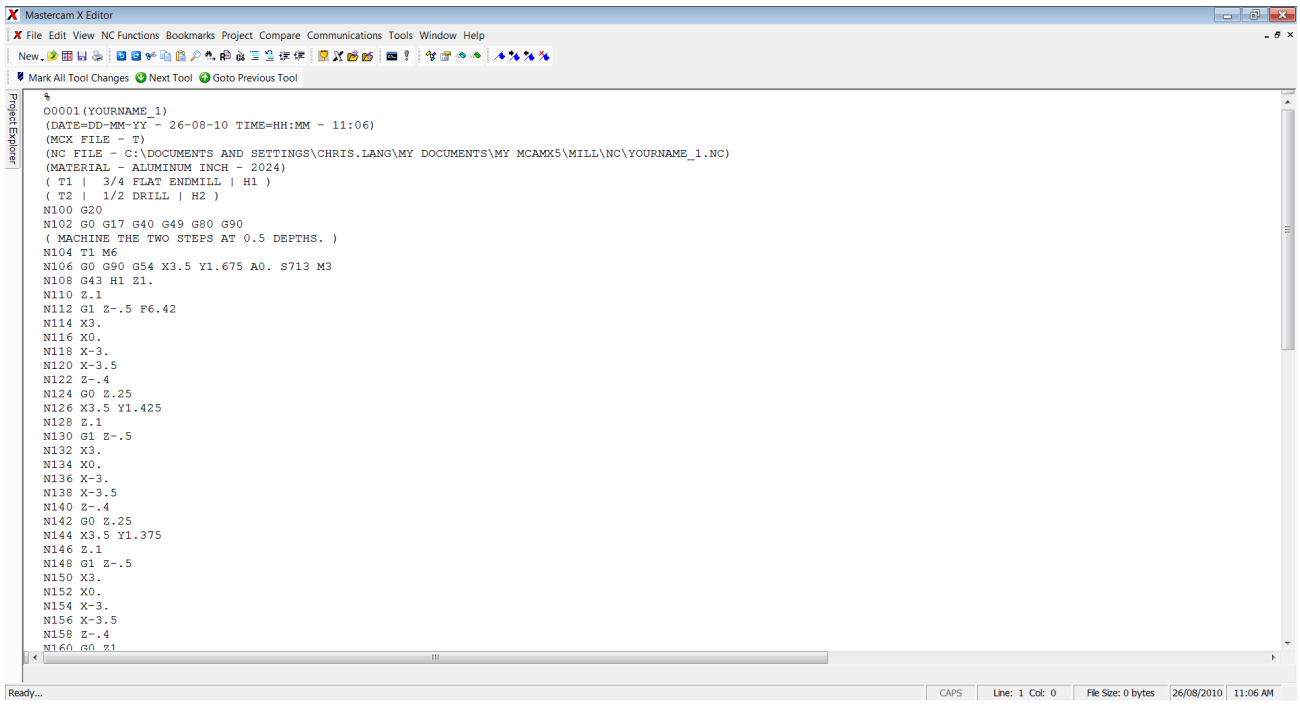
- Select the **OK** button to continue.



- Select the **OK** button to save the NC file.



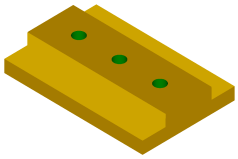
- A window with the NC program will appear.



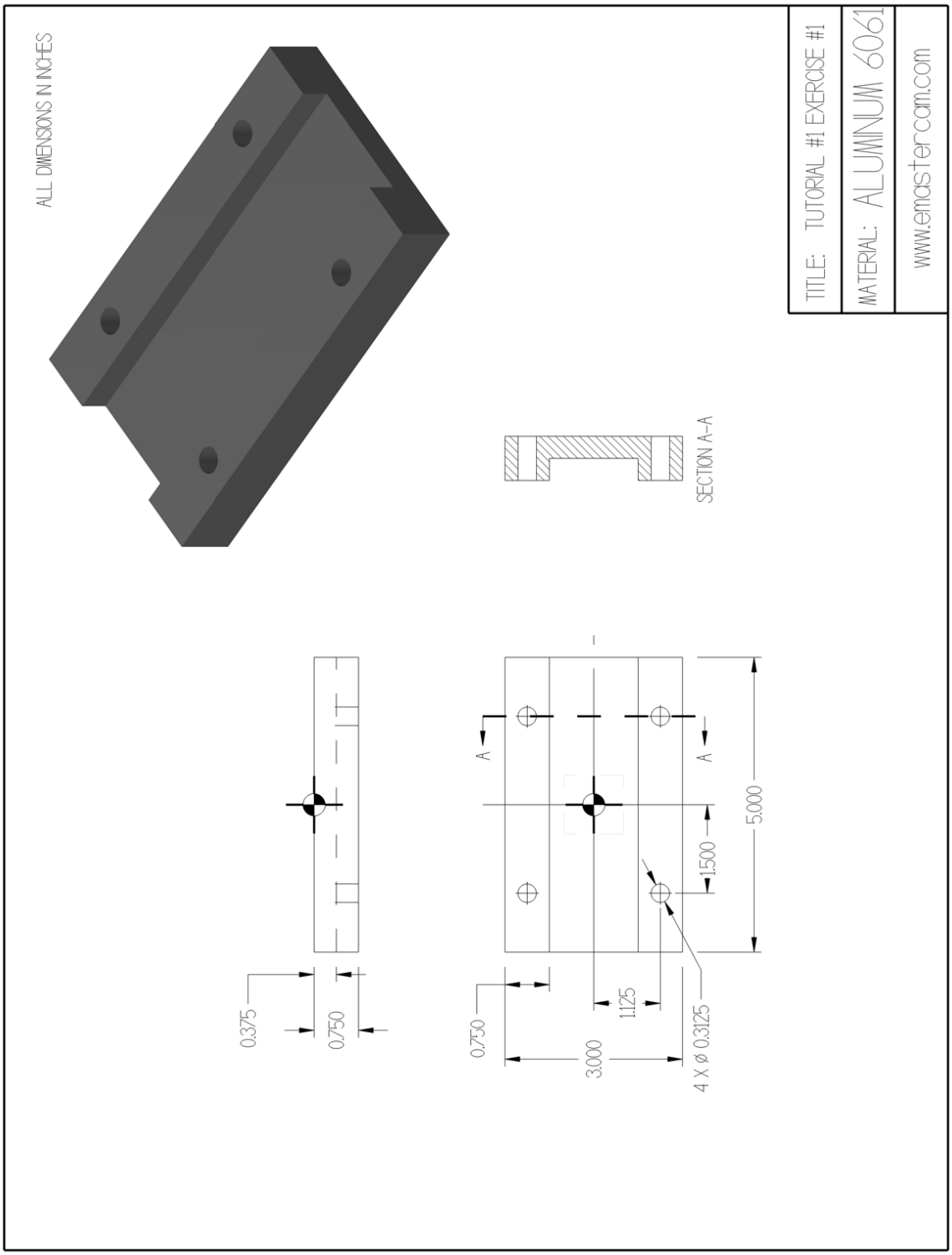
- Select the red **"X"** box at the upper right corner to exit the editor.

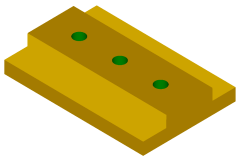
STEP 15: SAVE THE UPDATED MCX FILE

- Select the **Save** icon.



REVIEW EXERCISE -STUDENT PRACTICE

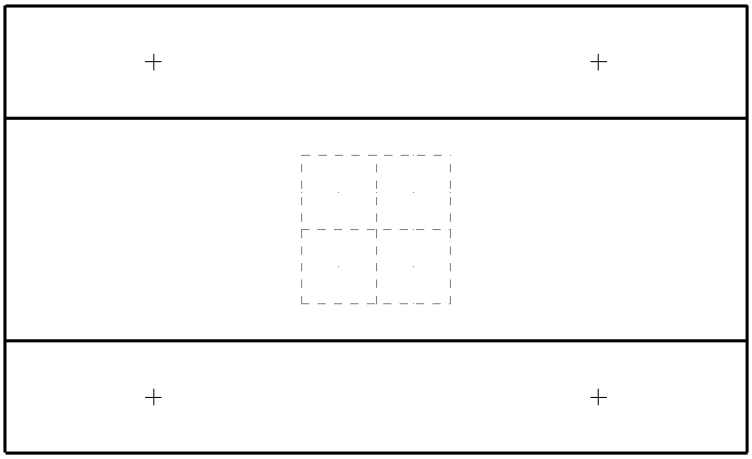


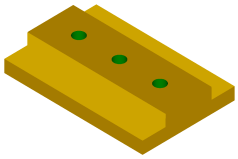


CREATE THE GEOMETRY FOR TUTORIAL #1 EXERCISE

Use these commands to create the geometry:

- Create 1/4 of the geometry.
- Create a rectangle.
- Create Line Parallel.
- Create Point Position (position using Fast Point).
- Delete Entities.
- Xform Mirror.
- Edit Join Entities.



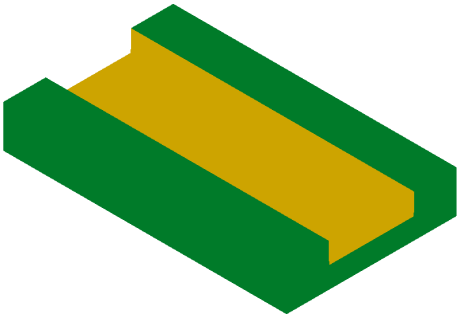


CREATE THE TOOLPATHS FOR TUTORIAL #1 EXERCISE

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

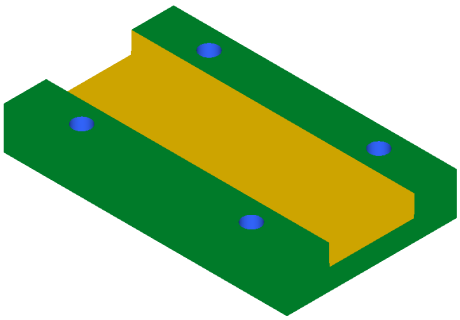
Set the machine properties including the stock.
Remove the material in the center of the part Contour (2D).

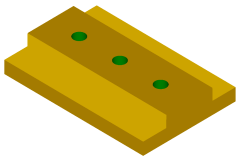
- Use a 7/8" Flat Endmill.
- Based on your chaining direction ensure the Compensation direction is set correct.
- No Depth Cuts.
- Lead In/Out ensure the arc radius is set to zero.
- No Break Through, Multi Passes.
- Set the depth according to the drawing.



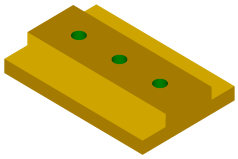
Drill the holes through the part.

- Use a 5/16" Drill.
- Choose the Drill/Counterbore cycle.
- Depth set according to the drawing.
- Enable Tip Comp and input a Breakthrough amount.





NOTES:



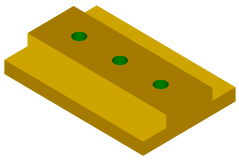
TUTORIAL #1 QUIZ

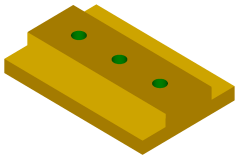
- What is a Contour Toolpath used for?

- What does Backplot do?

- When would you use Multi Passes?

- When would you use the drill cycle Peck Drill?





Other Available Mastercam Training Solutions:

Training Tutorial Series
Mill Level 1 (2D) | Mill Level 3 (3D) | Lathe | Solids | Beginner | Design | Router | Art | Intro to Multiaxis
Advanced Multiaxis | Wire | Indexing | Project Workbook

Instructor Kits for the Training Tutorial Series
Instructor Kit (for Mill Level 1 & Mill Level 3) | Instructor Kit for Mill & Lathe (Mill Level 1, Mill Level 3 & Lathe)

Metric Training Tutorial Series
Mill Level 1 | Mill Level 3 | Lathe | Solids | Beginner | Design | Router | Art | Intro to Multiaxis
Advanced Multiaxis | Wire | Indexing

Professional Courseware Series
Mill Level 1 | Mill Level 3 | Lathe | Multiaxis | Router

Handbook Series
Volume 1 | Volume 2 | Volume 3 | Volume 1 Learning Kit | Volume 1 Curriculum | Volume 2 Curriculum

CNC Courseware
Single Set | Site License | Premium Site License

Multimedia
Mill Level 1 | Mill Level 3 | Lathe | Intro to Multiaxis | Handbook Video Series

To order: contact your local reseller or visit www.emastercam.com

Mastercam X6
TRAINING TUTORIAL SERIES

About the Author:
Mariana Lendel has a Bachelor Degree in Mechanical Engineering and over ten years of experience as a CNC programmer. For the past decade, Mariana has been a Mastercam instructor at In-House Solutions where she teaches Mill Level 1, Level 3 and Multiaxis; Lathe, Wire EDM, Router, Solids and Art. Her trainees include members of the industrial workforce as well as instructors from High Schools and Colleges across Canada. Mariana and her team continue to develop the most extensive collection of Mastercam Training Solutions available.

In-House Solutions
240 Holiday Inn Drive, Unit A
Cambridge, ON, Canada N3C 3X4
T 800-529-5517 | F 519-658-1335
www.inhousesolutions.com

ISBN 926955-26-1-6
Mill Level 1 Tutorial
9 789269 552611