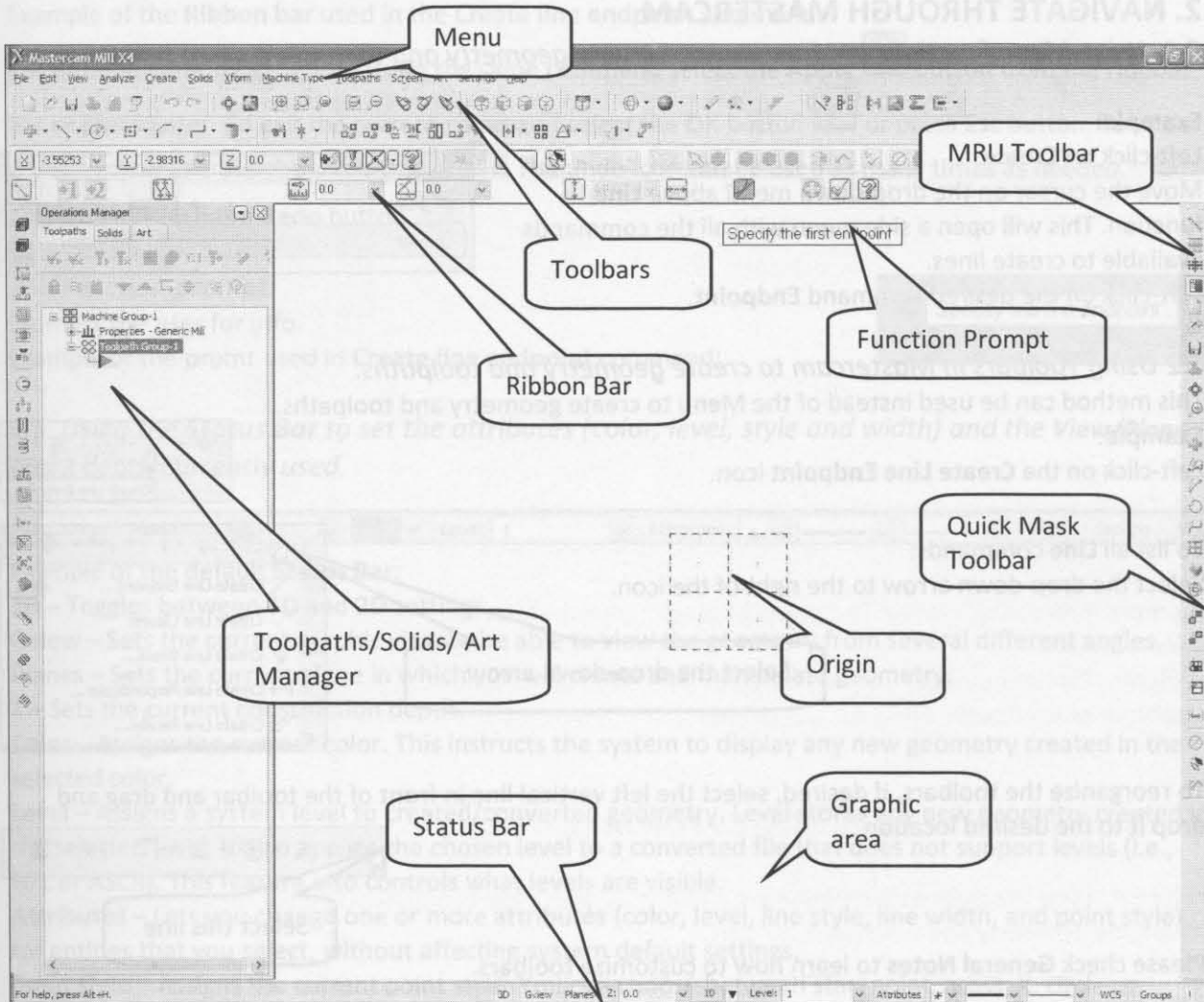


TUTORIAL SERIES FOR

Mastercam X⁴

GETTING STARTED

1. GUI-GRAPHIC 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 entity that you are currently creating or modifying.

Function prompt – Prompts the user for info.

Mill Level 1

Status Bar – Allows you to set the attributes (color, level, style and width) and the View/Plane and Z depth currently used.

Toolpaths/Solids 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 – List of the most recently used functions.

Quick Mask Toolbar – Lets you select all entities of the specified type.

2. NAVIGATE THROUGH MASTERCAM

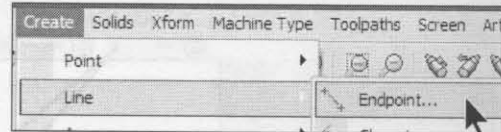
2.1. Using Menu functions in Mastercam to create geometry and toolpaths.

Example:

Left-click on **Create**.

Move the cursor on the drop-down menu above **Line** function. This will open a side menu with all the commands available to create lines.

Left-click on the desired command **Endpoint**.

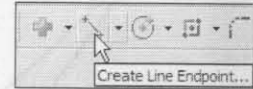


2.2 Using Toolbars in Mastercam to create geometry and toolpaths.

This method can be used instead of the Menu to create geometry and toolpaths.

Example:

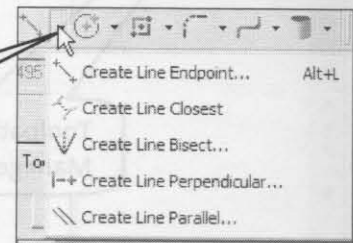
Left-click on the **Create Line Endpoint** icon.



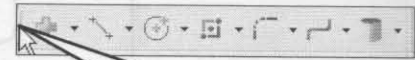
To list all **Line** commands:

Select the drop-down arrow to the right of the icon.

Select the drop-down arrow



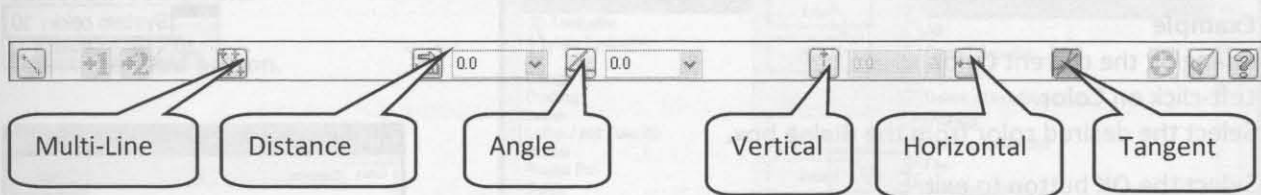
To reorganize the toolbars, if desired, select the left vertical line in front of the toolbar and drag and drop it to the desired location.






Select this line


Please check **General Notes** to learn how to customize toolbars.

2.3. Using the Ribbon Bar to enter the value and define the settings while creating or modifying geometry.



Example of the **Ribbon bar** used in the **Create line endpoint** command.

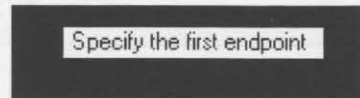
To continue creating geometry using the same command select the **Apply**  button from the ribbon bar or press Enter. To exit the current command select the **OK** button  or press Esc button. To undo the last command use the **Undo** button . The undo icon can be used as many times as needed.

Mastercam also has a **Redo** button. 

2.4. **Function prompt.**

Prompts the user for info.

Example of the prompt used in **Create line endpoint** command:



2.5. Using the **Status Bar** to set the attributes (color, level, style and width) and the **View/Plane** and **Z** depth currently used.



Example of the default **Status Bar**:

3D – Toggles between **3D** and **2D** settings.

Gview – Sets the current graphic view to be able to view the geometry from several different angles.

Planes – Sets the current plane in which you will create and manipulate geometry.

Z – Sets the current construction depth.

Color – Assigns the current color. This instructs the system to display any new geometry created in the selected color.

Level – Assigns a system level to created/converted geometry. Level stores any new geometry created in the selected level. It also applies the chosen level to a converted file that does not support levels (i.e., NFL or ASCII). This feature also controls what levels are visible.

Attributes – Lets you change one or more attributes (color, level, line style, line width, and point style) for entities that you select, without affecting system default settings.

Point style – Assigns the current point style. You can choose between star, point, cross, X, circle or square styles.

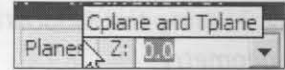
Line style – Assigns the current line style. You can choose between solid, hidden, center, phantom and Zbreak styles.

Line width – Assigns the current line width. You can choose between 5 different widths.

WCS – Allows you to redefine the World Coordinate System, and groups the functionality of Tool, Construction and Graphic Views.

Groups – Defines a collection of entities or operations that can be manipulated as a single entity.

Hovering the cursor over each function displays the current setting.



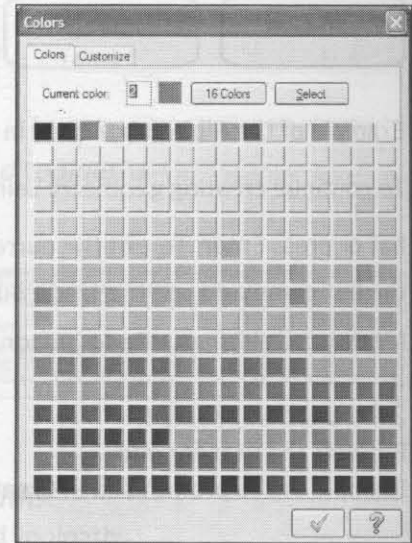
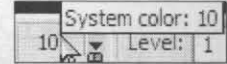
Example

Changing the current **Color**.

Left-click on **Color**.

Select the desired color from the dialog box.

Select the **OK** button to exit.



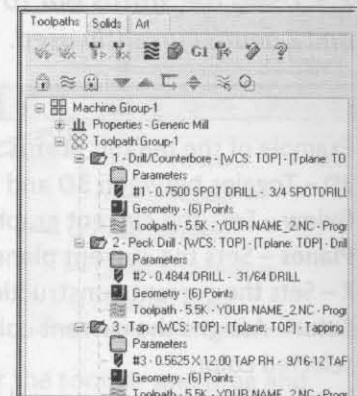
2.6. Using the Toolpaths Manager.

The **Toolpaths Manager** lists all operations in the current job.

Use this dialog box to sort, edit, regenerate, verify and post any operations.

For more information on the **Toolpaths Manager**, please check the **General Notes**.

Toolpaths/Solids manager can be hidden to gain more space in the graphic area for design. Press **Alt + O** to hide it.

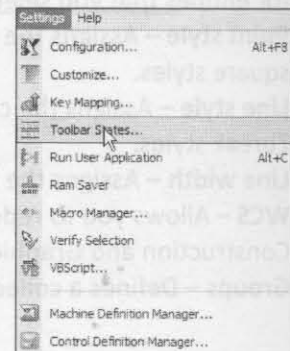


3. SETTING THE TOOLBAR STATES FOR MILL

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

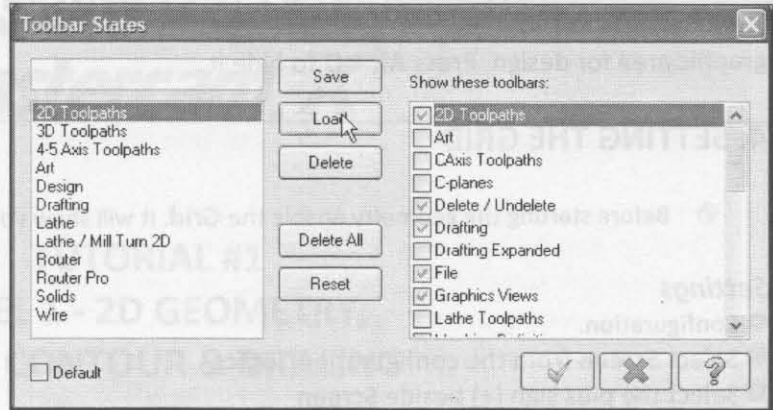
• **Toolbar states**



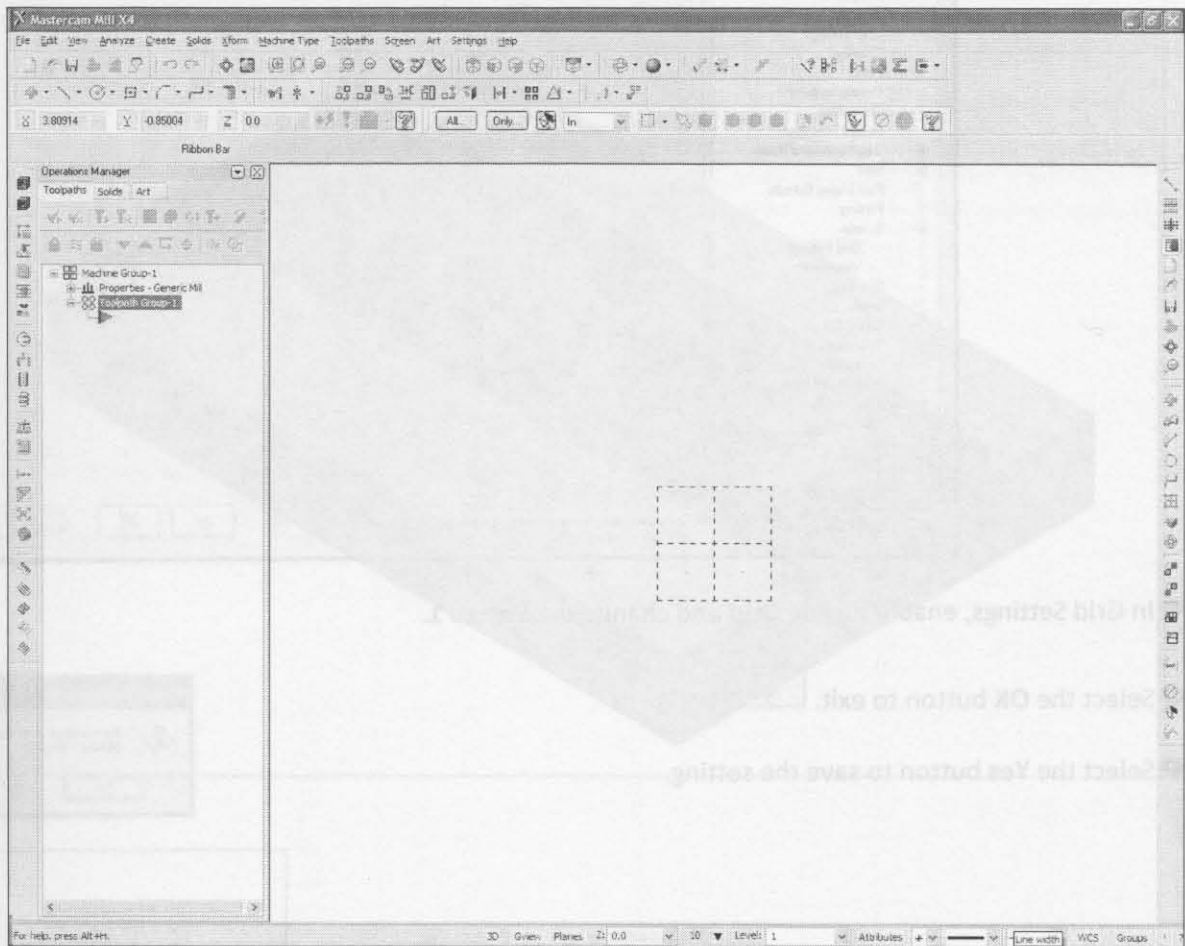
Select **2D-Toolpaths**.

Select the **Load** button.

Select the **OK** button to accept it.



The 2D toolpath toolbar is then displayed as shown, to the left of the **Operations Manager**.



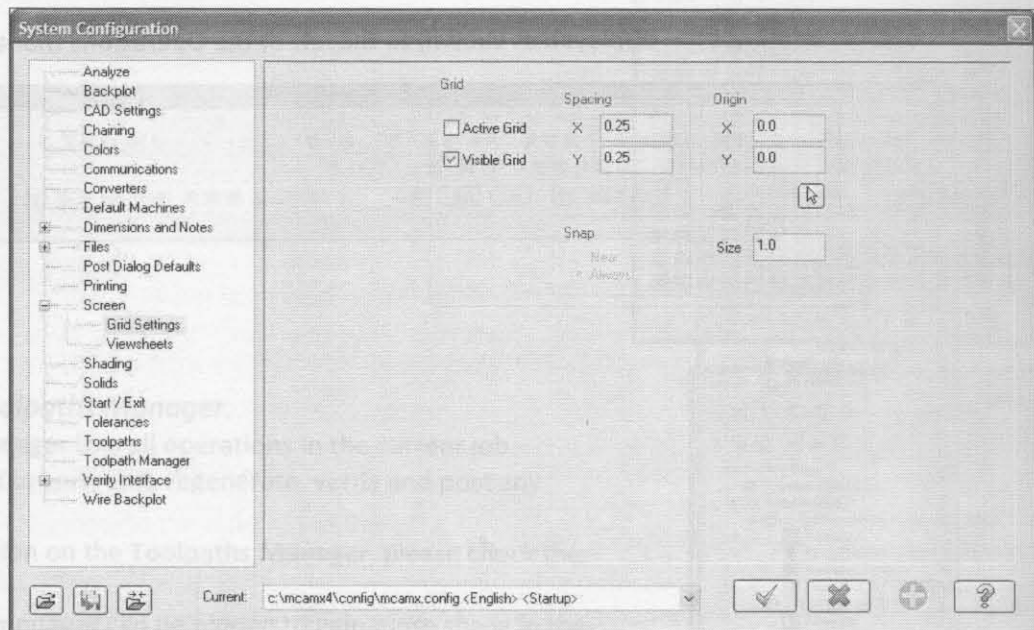
The **Toolpaths/Solids/Art Manager** to the left of the screen can be hidden to gain more space in the graphic area for design. Press **Alt + O** to hide it.

4. SETTING THE GRID


- Before starting the geometry enable the **Grid**. It will show you where the part origin is.

Settings

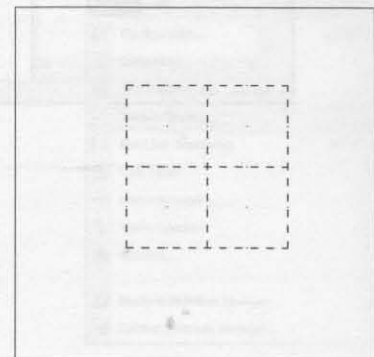
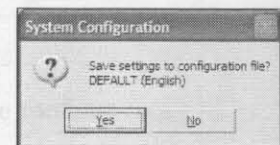
- **Configuration.**
- Select **Screen** from the configuration **Topics**.
- Select the plus sign (+) beside **Screen**



- In Grid Settings, enable **Visible Grid** and change the **Size** to 1.

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

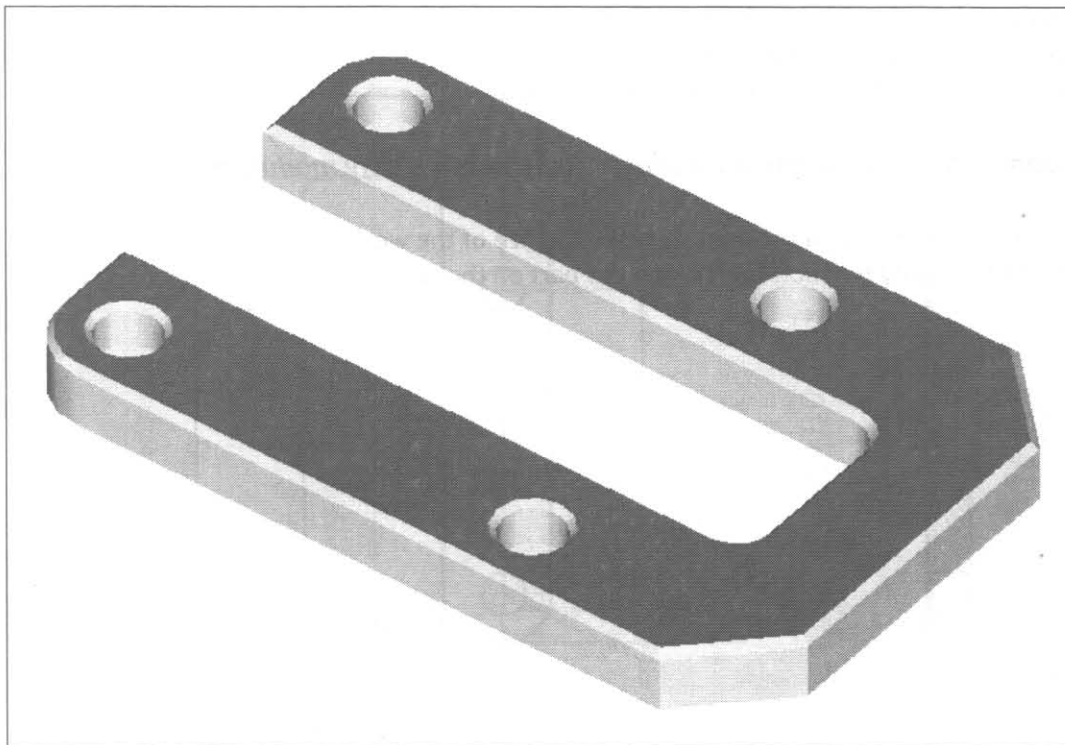
- Select the **Yes** button to save the setting.



TUTORIAL SERIES FOR

Mastercam X⁴

**TUTORIAL #2
LEVEL 1 - 2D GEOMETRY;
DRILLING, OPEN POCKET, OPEN CONTOUR & CHAMFER CONTOUR
TOOLPATHS**



Mill Level 1

Objectives:

The Student will design a 2-dimensional wireframe drawing by:

- Creating a rectangle knowing the width and the height of the rectangle.
- Creating parallel lines defining the offset direction and distance.
- Trimming 1 entity to another existing entity.
- Creating fillet radii.
- Creating 45 degree chamfers.
- Creating arcs knowing the diameter and the center point.
- Mirroring existing geometry to complete the part.

The Student will create 2-dimensional milling toolpaths consisting of:

First Setup

- 4 holes drilled through the part.

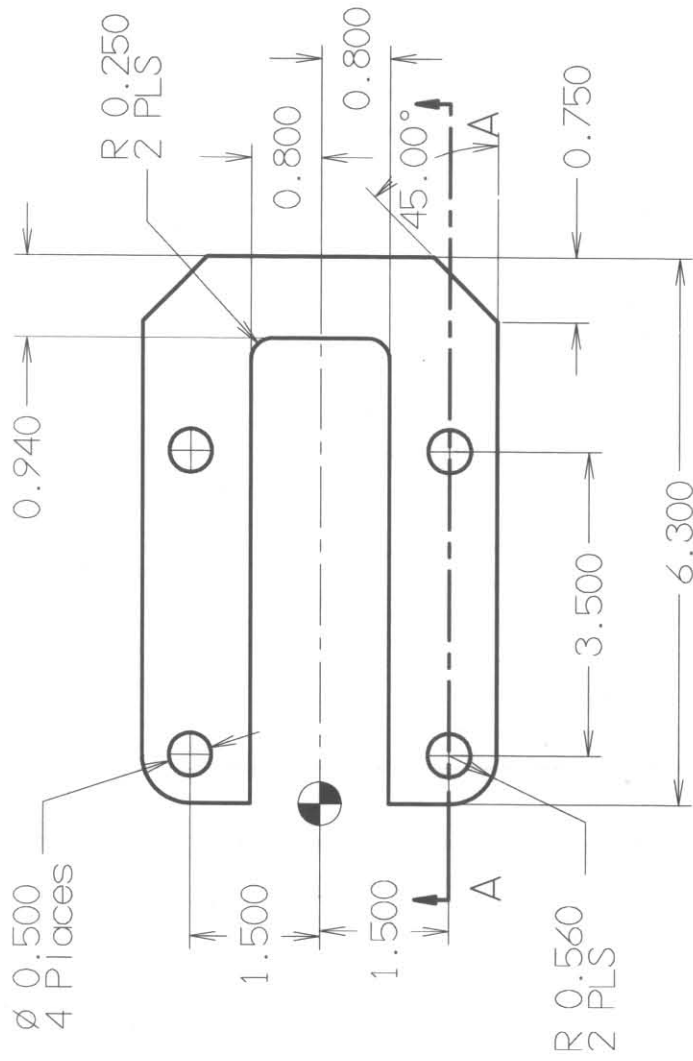
Second Setup

- An open pocket to machine the inside material.
- A contour using ramp motion.
- Chamfer the top of profile using chamfer contour.

The Student will check the toolpath using Mastercam's Verify module by:

- Defining a 3-dimensional rectangular block the size of the workpiece.
- Running the Verify function to machine the part on the screen.

ALL DIMENSIONS IN INCHES



45 degrees X 0.05 Chamfer
4 Places



SECTION A - A

TITLE TUTORIAL 2

MATERIAL ALUMINUM T6061

DATE: JUNE 12, 2008 eMastercam.com

Mill Level 1

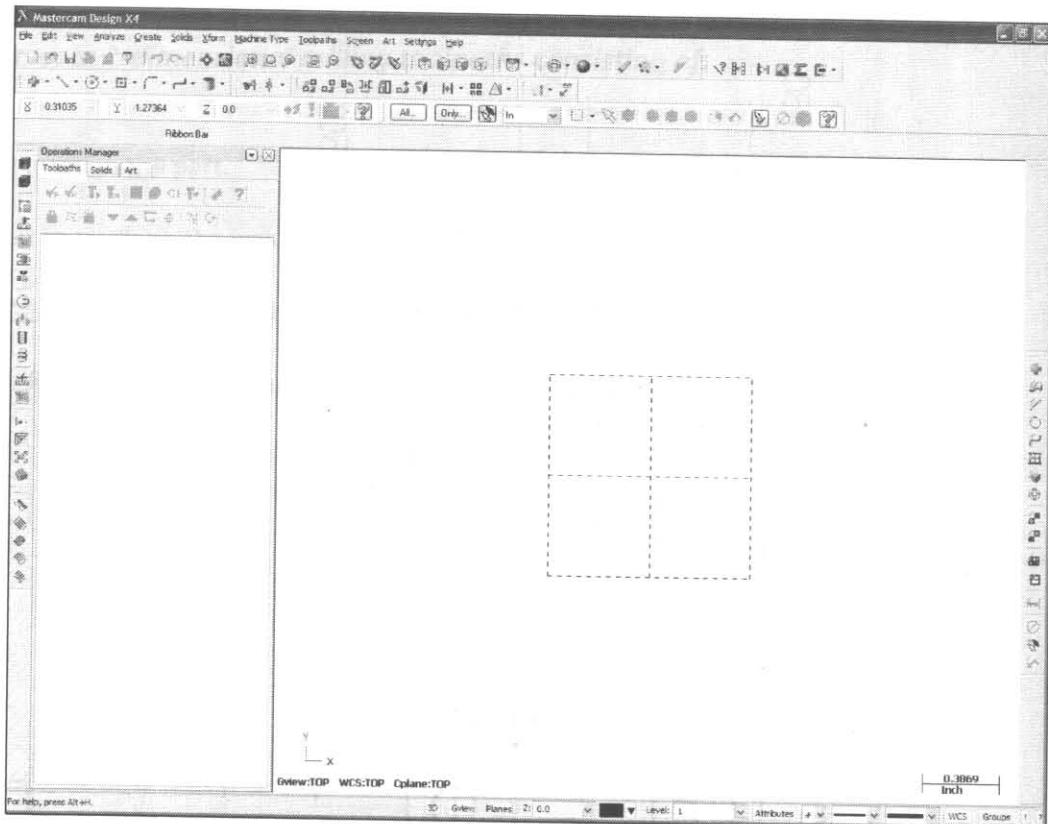
GEOMETRY CREATION

Setting up the Graphic User Interface

Before starting the geometry creation we should customize the toolbars to see the toolbars required to create the geometry and machine a 2D part. See **Getting started** page A-4 for details.

Make sure that the **Grid** is enabled. It will show you where the part origin is. See **Getting started** page A-6 for further information.

The **Operations Manager** to the left of the screen can be hidden to gain more space in the graphic area for design. From the keyboard, press **Alt + O** keys simultaneously to hide it. Repeat this command to make it visible again.



- ◆ Since this drawing is symmetrical in the X-axis you will only draw half of the total part and use the mirror function to complete the part.

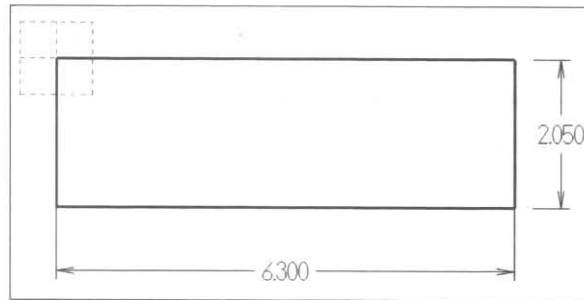
Start a new file

File

● **New**

STEP 1: CREATE A RECTANGLE KNOWING THE WIDTH, HEIGHT AND BASE POINT.

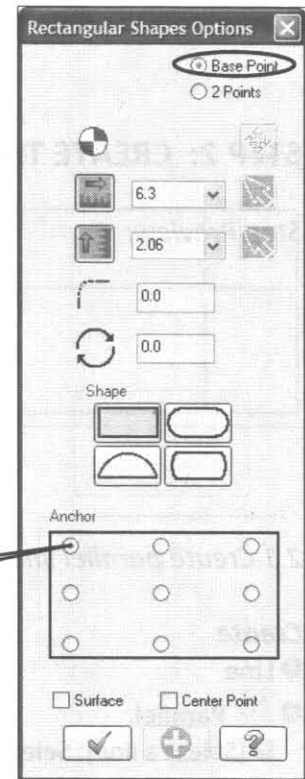
Step Preview:



Create

Rectangular Shapes

- Make sure that the radio button in front of **Base Point** is enabled.
- Enter the **Width** and the **Height** as shown in the screenshot to the right.
- Make sure that the rectangular shape is selected.

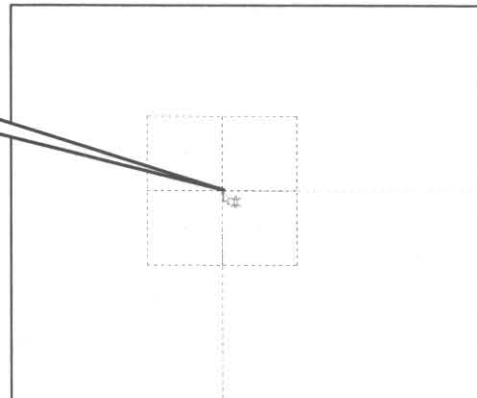


- Select the upper left corner radio button as the anchor.

Select the Anchor here

- [Select position for base point]: Select the center location of the grid (the origin).



Select the Origin

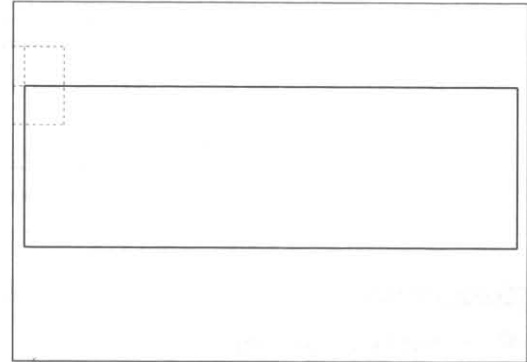


Mill Level 1

☛ Select the **OK** button to exit the **Rectangular Shapes Options** dialog box. 

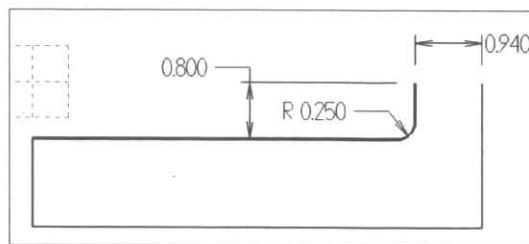
☛ Use the **Fit** icon to fit the drawing to the screen. 

☛ 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. 



STEP 2: CREATE THE INSIDE GEOMETRY

Step Preview:



2.1 Create parallel lines.

Create

☛ **Line**


☛  **Parallel**

☛ [Select a line]: Select Entity A.

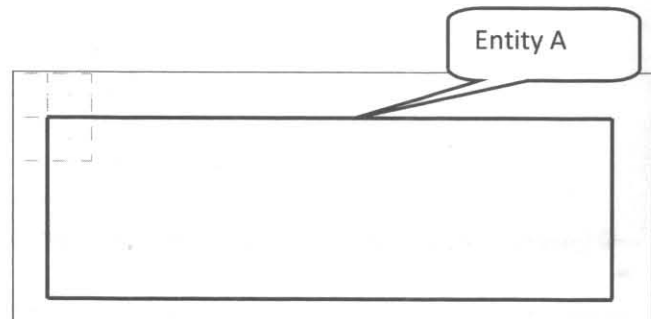
☛ [Select the point to place a parallel line through]: Pick a point below the selected line.

☛ Note that the color of the geometry is cyan which means that the entity is "live" and you can still change the line parameters.


☛ Enter the **Distance**  0.8 (Press Enter).

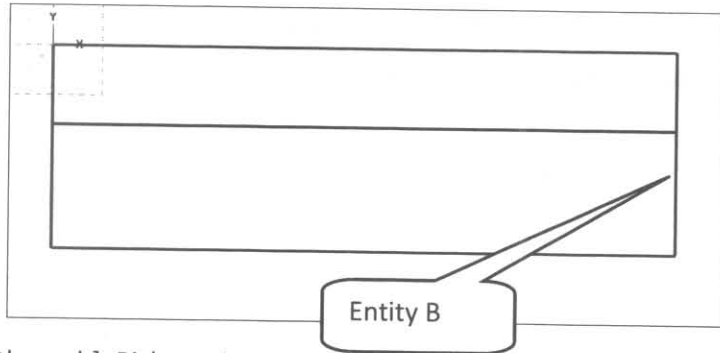
☛ Note that to continue using the same command you have to select the **Apply** button  or press Enter.



To exit the command you can either start a new command or select the **OK** button .



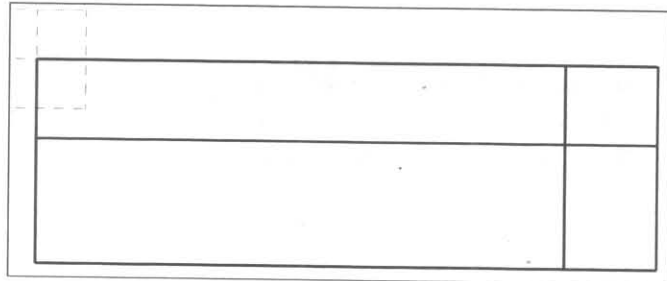
Mill Level 1

- Select the **Apply** button to continue. 
- [Select a line]: Select Entity B.



- [Select the point to place a parallel line through]: Pick a point to the left of the selected line.
- Type the **Distance**  0.94 (Enter).
- Select the **OK** button. 

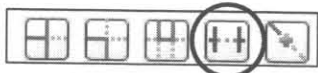
The drawing should look as shown to the right.



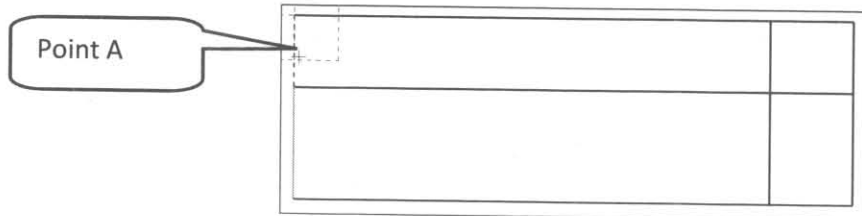
2.2 Trimming the lines.


Edit

- **Trim/Break**
-  **Trim/Break/Extend**




- Select **Divide**.
- [Select the entity to trim/extend]: Select the entity at Point A (select all entities exactly as shown in the drawing).

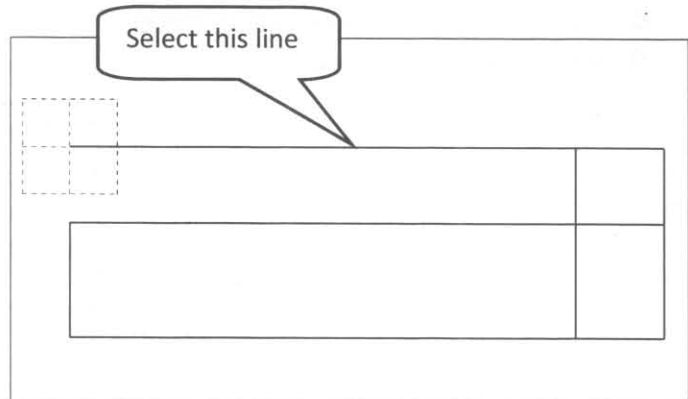


- Note that the dashed line represents what will be removed.
- Select the **OK** button. 

Mill Level 1

2.3 Deleting a line.

- Select the horizontal line as shown.
- Select the **Delete entity** icon. 




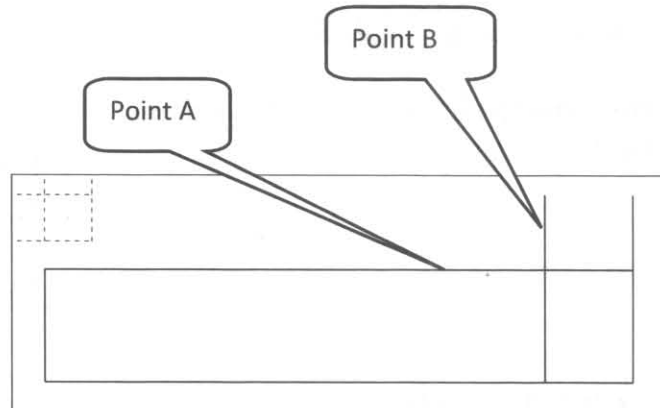
2.4 Create the 0.25 radius fillet.

Create

➤ **Fillet**

➤  **Entities**

- Enter the fillet **Radius**  0.25.
- [Select an entity]: Select Point A.
- [Select another entity]: Select Point B.



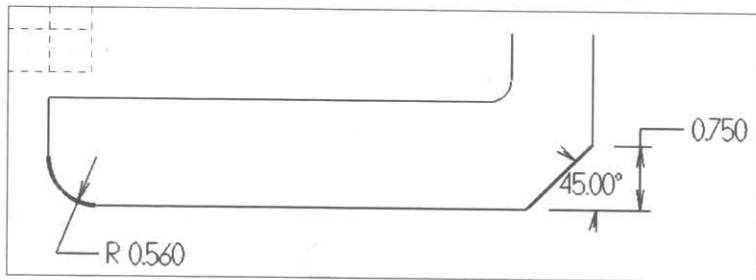
- Note that a fillet preview will be automatically drawn depending on where you move the cursor around the entities.

- Select the **OK** button. 

Mill Level 1

STEP 3: COMPLETE THE OUTSIDE PROFILE

Step Preview:







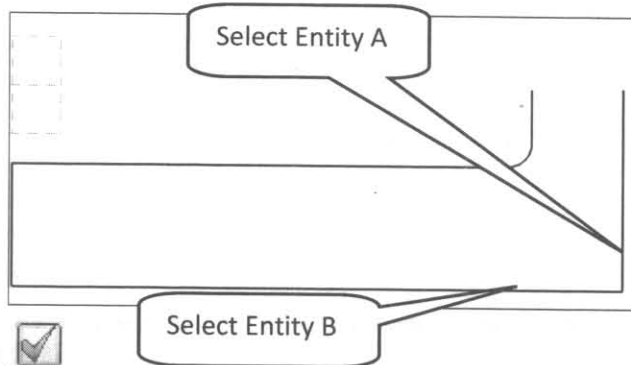
3.1 Create the 45 degree chamfer.

Create

☛ Chamfer

☛ Entities

- ☛ Make sure that **1 Distance**  1 Distance  is selected and the **Trim**  option is enabled.
- ☛ Enter the **Distance 1**  0.75 (Press Enter).
- ☛ [Select line or arc]: Select Entity A.
- ☛ [Select line or arc]: Select Entity B.




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

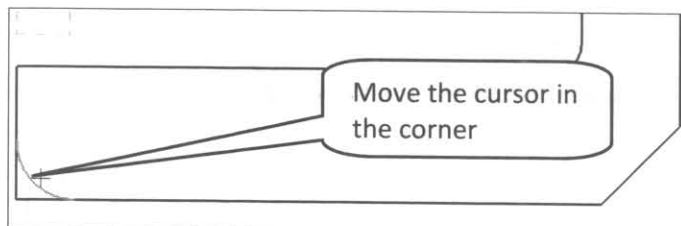
3.2 Create the 0.56 radius fillet.

Create

☛ Fillet

☛ Entities

- ☛ Enter the fillet **Radius**  0.56 (Enter).
- ☛ [Select an entity]: Move the cursor in the corner area until the desired fillet appears. Left-mouse click to accept it.

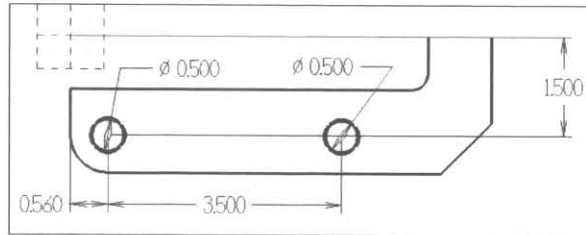


- ☛ Select the **OK** button. 

Mill Level 1

STEP 4: CREATE THE TWO 0.5" DIAMETER CIRCLES KNOWING THE CENTER POINT AND THE DIAMETER

Step Preview:



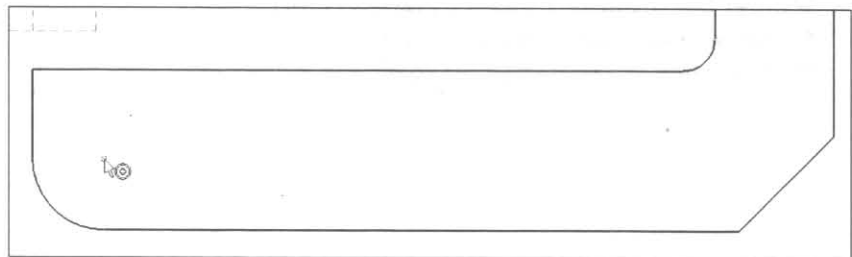
Create

➤ Arc

➤ Circle Center Point

➤ Enter the **Diameter** value 0.5 (Enter).

➤ [Enter the center point]: Move the cursor to the center location of the 0.56 radius fillet. Select the point when the circle center icon appears.



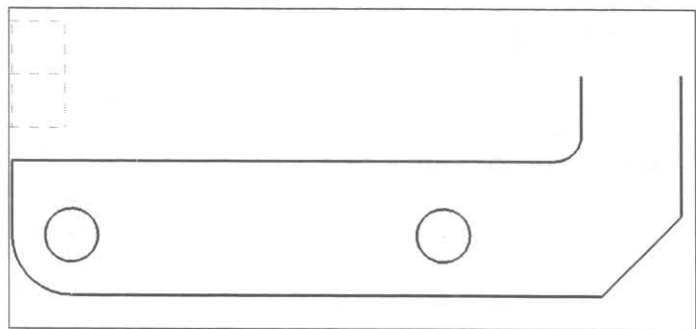
➤ To create more arcs with the same diameter click on the diameter icon to lock the values. The diameter and radius values will be highlighted in red. To unlock the values, click again on the diameter or radius icons.

➤ [Enter the center point]: Select the **Fast Point** icon.

➤ Enter the coordinates: 0.56+3.5, -1.5 (Enter).

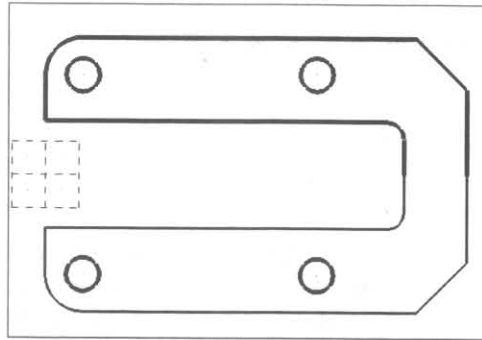
➤ Note that Mastercam will perform basic math operations (+, -, *, /). You can enter the values without any of the corresponding coordinate letters (X, Y, Z) as long as you enter them in this order and separate them by commas.

➤ Select the **OK** button to exit



STEP 5: TRANSFORM THE GEOMETRY TO REPRESENT THE WHOLE PART

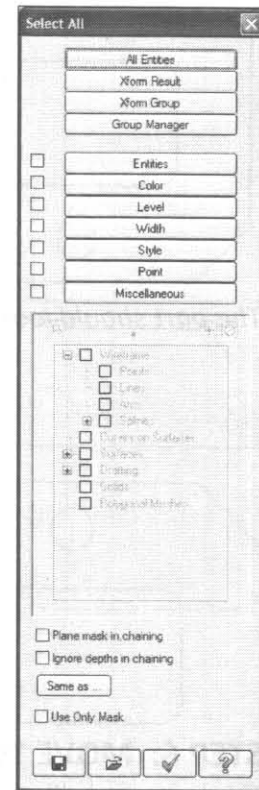
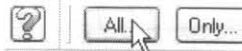
Step Preview:



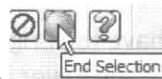
Xform

 **Mirror**

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




Select the **OK** button in the **Select All** dialog box.

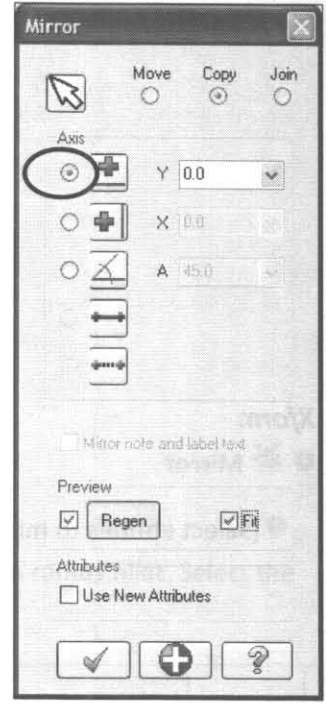


Click on the **End Selection** button.

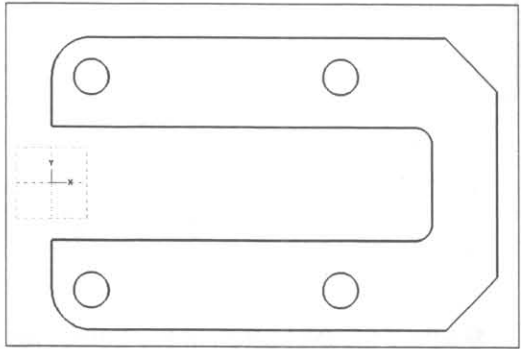
Mill Level 1

- Set the parameters to match the following dialog box (keep the original entities **Copy**, to mirror about the **X-axis**, **Fit** the geometry to the screen).
- Select the **OK** button to exit the **Mirror** window. 

• Note that although a Y value (in our case 0.0) can be also added to define the horizontal axis location of the plane, the first radio button enables mirror about **X-axis**. The tool tip within Mastercam will confirm that this indeed is the setting for X axis.



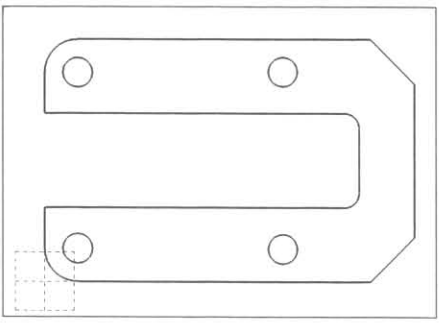
The part should look as shown in the following picture.



STEP 6: MOVE TO ORIGIN

This command allows you to quick move the part to have the origin in the same location that you will set the part zero at the machine. In our case we will move the lower left corner to the origin.

Step Preview:



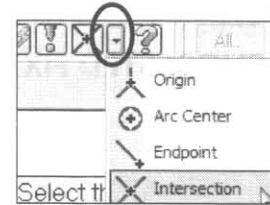
Mill Level 1

Xform

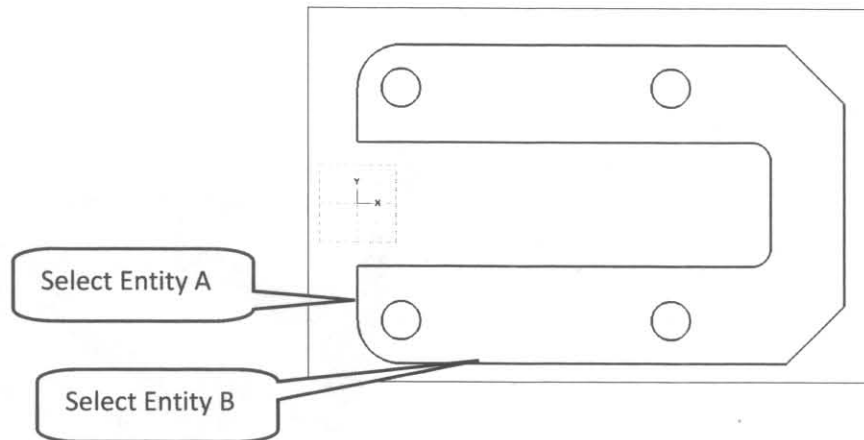
Move to Origin

[Select the point to translate from]: Select the drop-down arrow next to the **AutoCursor's override** and select from the list **Intersection** as shown.

- Note that the **AutoCursor Override list** allows you to choose only one type of point to detect and snap to. In our case we do not have an intersection point to snap to, so we will let Mastercam determine the intersection point of two existing lines.



Select the two lines as shown.

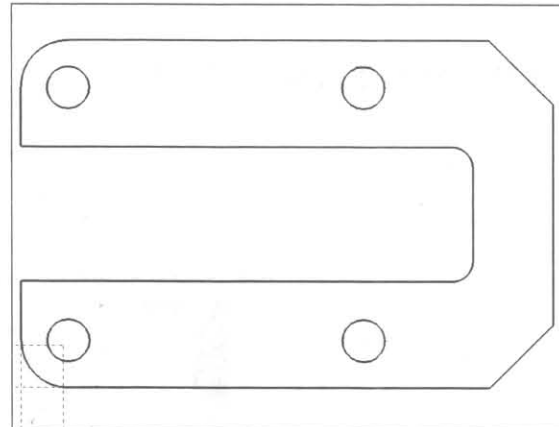


Screen

Clear Colors

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

The final geometry should look as shown.



- Note the origin is set to the lower left corner.

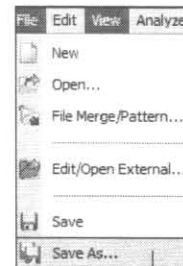
STEP 7: SAVE THE FILE

File

Save As

File name: "Your Name_2"

Select the **OK** button.

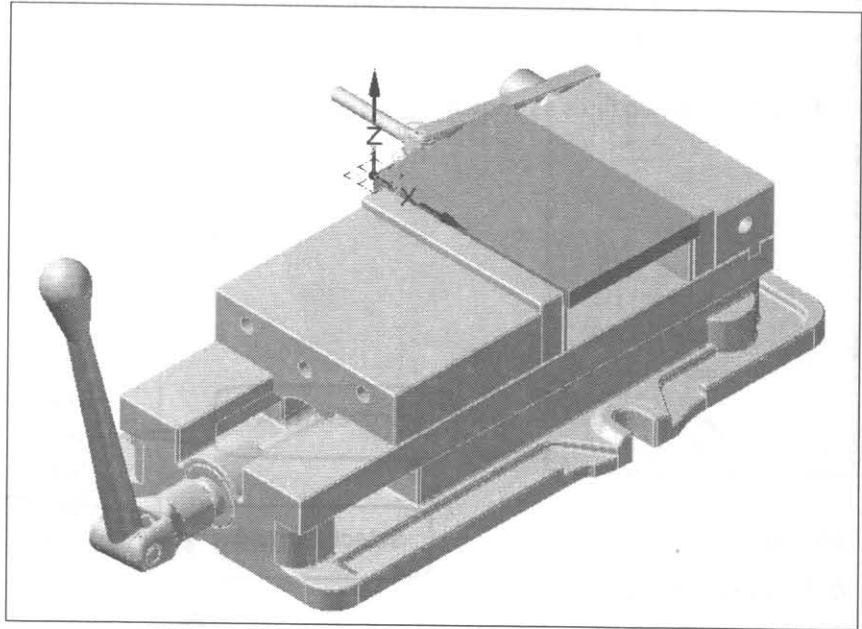


Mill Level 1



TOOLPATH CREATION - SETUP 1

This part will be machined in two different setups. To accomplish that, we will create two separate programs. To be able to post process separate operations of each setup, we will create them under different toolpath groups with different NC names.

SUGGESTED FIXTURE FOR SETUP 1:




SETUP SHEET FOR SETUP 1:

Operation List of YOUR NAME 2.MCX			
Proj./Part No.:	0	Date:	04/22/09
Drawing No.:	1	Customer:	C:\MCMX4\MILL\NCI\YOUR NAME 1.NCI
Prog. No.:	2	Programmer:	0
<hr/>			
	Tool type :	0.75 Spot Drill 3/4 SPOTDRILL	
	Manufact.code :		
	Chuck :		
	Tool Number :	1	Operation time: 0h 0m 44s
	Diameter :	0.75	RPM : 4000
	Corner radius :	0	Tip angle : 118
	Flute length :	2	Material : ALUMINUM ...
	Comment :	Canned Cycle Spot drill all holes	
	Tool type :	0.5 Drill 1/2 DRILL	
	Manufact.code :		
	Chuck :		
	Tool Number :	2	Operation time: 0h 0m 24s
	Diameter :	0.5	RPM : 3200
	Corner radius :	0	Tip angle : 0
	Flute length :	2	Material : ALUMINUM ...
	Comment :	Canned Cycle Drill through holes	
(Post processor specific average tool change time 0.0, rapid feedrate 250.0)			
Total machining time:		0h 1m 8s	

Mill Level 1

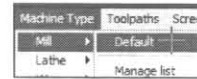
STEP 8: SELECT THE MACHINE AND SET UP THE STOCK TO BE MACHINED

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).

- To display the **Toolpaths Manager** press **Alt + O**.
- Use the **Fit** icon to fit the drawing to the screen. 
- For the purpose of this tutorial, we will be using the default mill machine. If the default machine is already selected, disregard the next step.

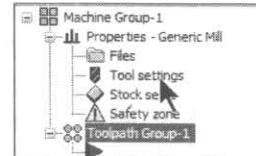
Machine type

- Mill
- Default



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

Select the plus



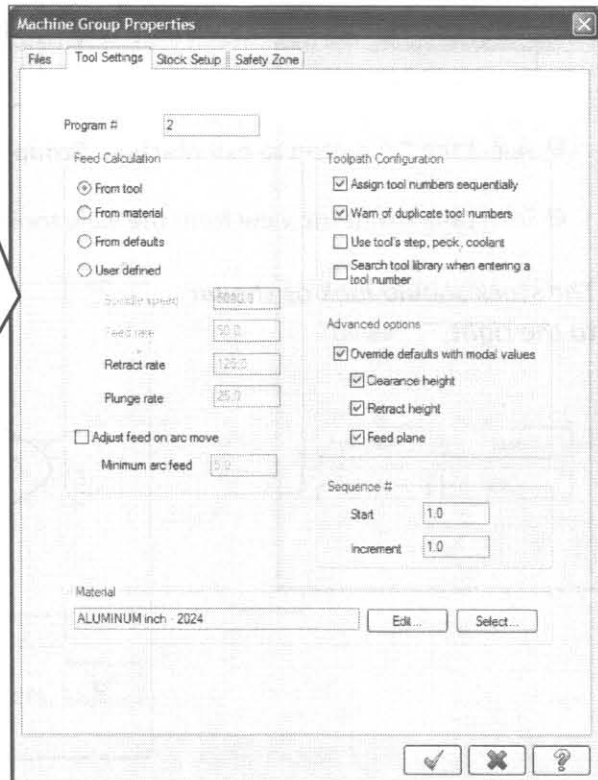
- Select **Tool Settings** and change the parameters to match the screenshot below.

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.

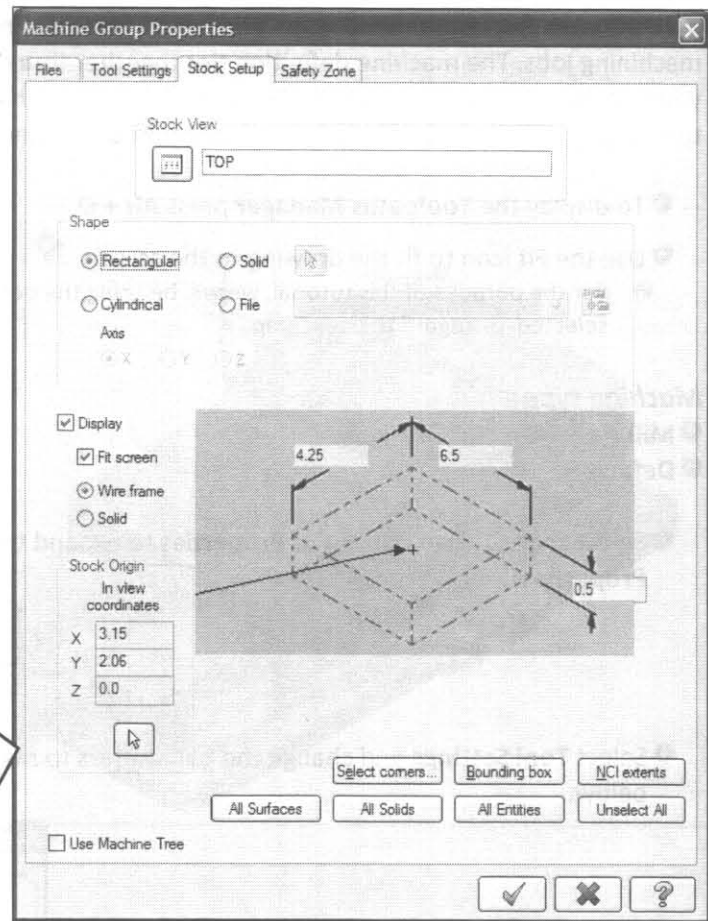


Mill Level 1

- Select **Stock setup** tab and define the stock by setting the stock shape (Rectangular) and entering the stock dimensions as shown in the screenshot below.

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

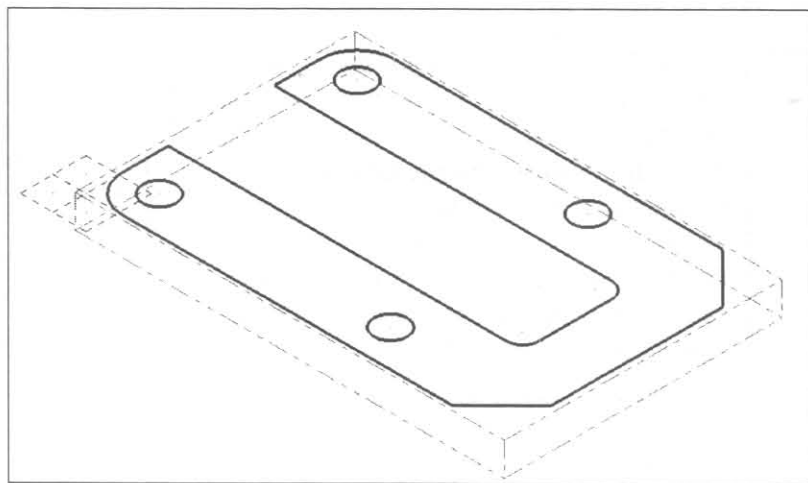
Display options allow you to set the stock as **Wireframe** and to fit the stock to the screen (**Fit Screen**) to better visualize the stock.



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

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

The stock should look as shown to the right.



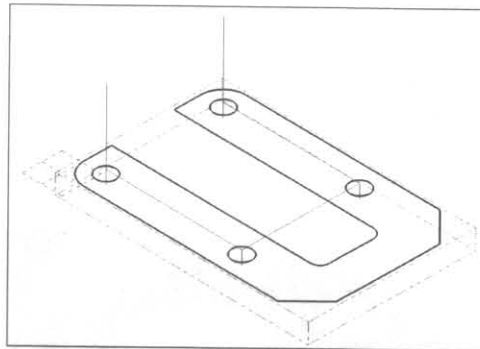
Mill Level 1

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



STEP 9: SPOT DRILL THE FOUR 1/2" DIAMETER HOLES

Toolpath Preview:

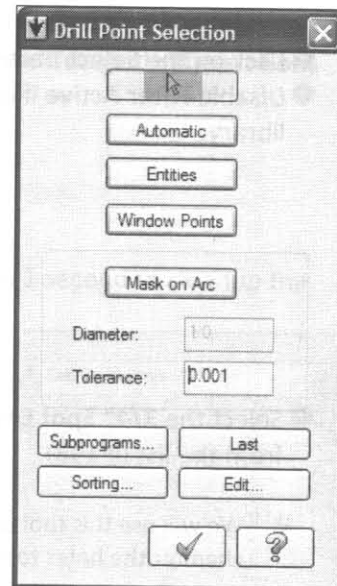
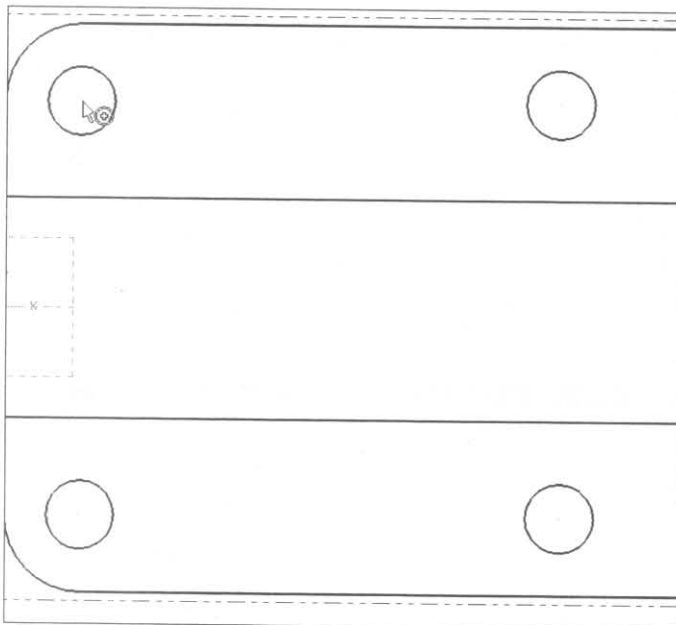


9.1 Drill center points selection

Toolpaths

Drill

- Select the **OK** button to accept the NC file name the same as the geometry name.
- Select the four circle centers by clicking within the center of the circles as shown in the following picture.

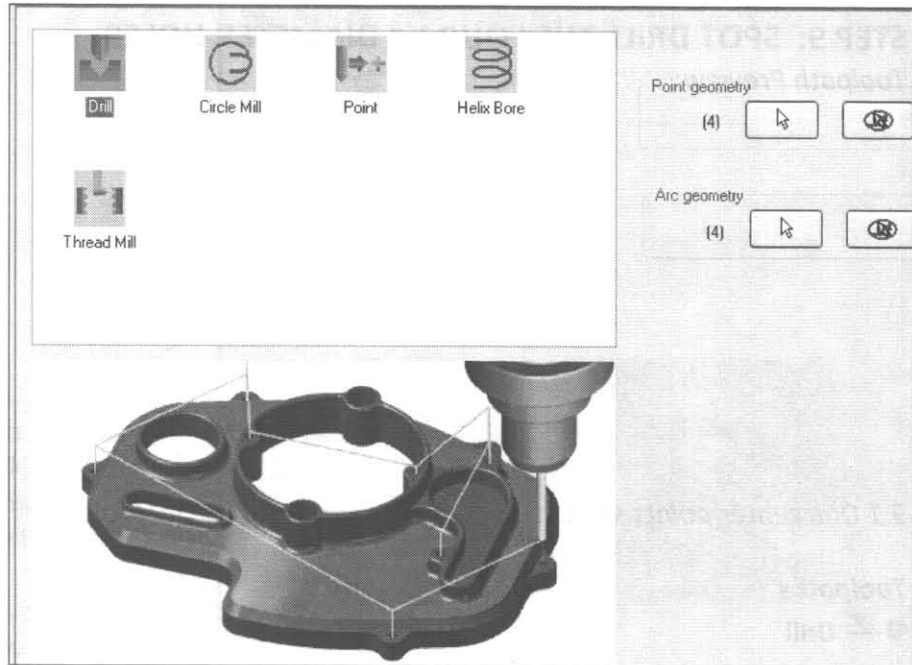


- Select the **OK** button, once you select all four centers.

Mill Level 1

9.2 Check the Toolpath Type page parameters

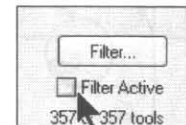
- In the **Toolpath Type** page, note that the **Drill** icon should be selected and in the **Point/Arc geometry** you should have 4 points as shown.



9.3 Select the 3/4" Spot drill

- From the **Tree view list** select **Tool**.
- Click on the **Select library tool** button.
- Disable **Filter Active** if needed to see all the tools available in the current tool library.

Select library tool...



- Select the **3/4" Spot Drill** from the list (# 198).

- We will use this tool to chamfer the holes too.

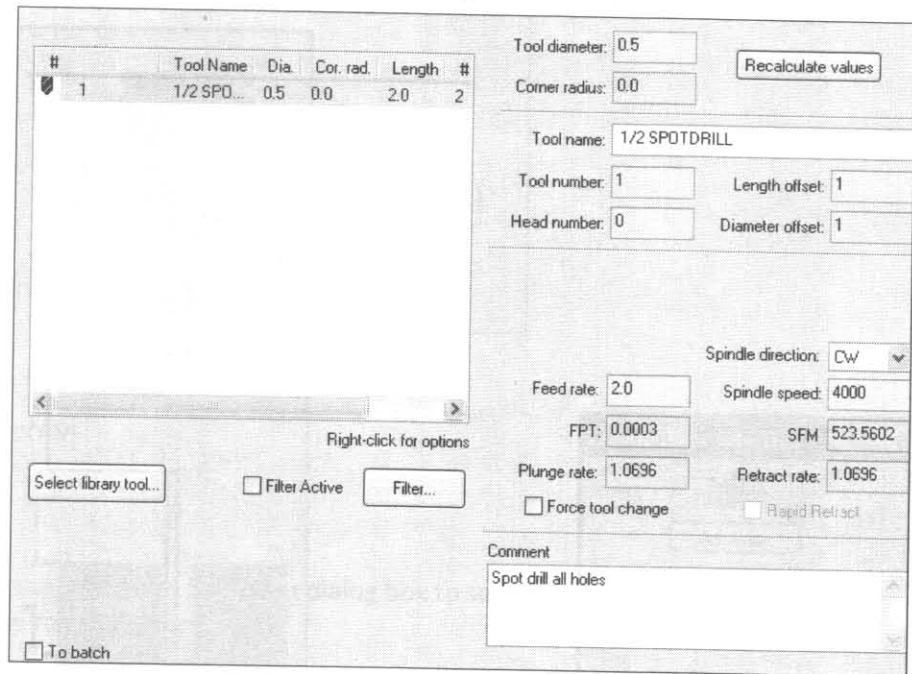
#	Tool Name	Dia.	Cor. rad.	Length	# Flutes	Type	Rad. Type
192	NO. 8-36...	0...	0.0	2.0	1	Tap RH	None
193	NO. 10-2...	0...	0.0	2.0	1	Tap RH	None
194	NO. 10-3...	0...	0.0	2.0	1	Tap RH	None
195	NO. 12-2...	0...	0.0	2.0	1	Tap RH	None
196	NO. 12-2...	0...	0.0	2.0	1	Tap RH	None
197	17/32 F...	0...	0.0	1.0	4	Endmill Flat	None
198	3/4 SPO...	0.75	0.0	2.0	4	Spot Drill	None
199	1/4-20 T...	0...	0.0	2.0	1	Tap RH	None
200	1/4-28 T...	0...	0.0	2.0	1	Tap RH	None
201	5/16-18 ...	0...	0.0	2.0	1	Tap RH	None
202	5/16-24 ...	0...	0.0	2.0	1	Tap RH	None
203	3/8-16 T...	0...	0.0	2.0	1	Tap RH	None
204	3/8-24 T...	0...	0.0	2.0	1	Tap RH	None
205	7/16-14 ...	0...	0.0	2.0	1	Tap RH	None
206	7/16-20 ...	0...	0.0	2.0	1	Tap RH	None
207	1/2-13 T...	0...	0.0	2.0	1	Tap RH	None
208	1/2-20 T...	0...	0.0	2.0	1	Tap RH	None

- Select the **OK** button.

Mill Level 1

9.4 Set the parameters of the spot drilling operation

- Make the changes in the **Tool** parameters as shown in the following screenshot.

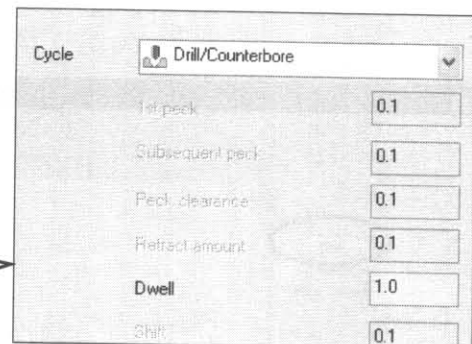


- Note that the **Feed rate**, **Plunge rate**, **Retract rate** and **Spindle speed** are roughly based on the part material aluminium and the tool material HSS. Change them if needed.
- In the **Comment** field enter a comment to help identify the toolpath in the **Toolpaths/Operations Manager**

9.5 Set the Cut Parameters

- Make sure that the **Cycle** is set to **Drill /Counterbore** and set the **Dwell** to 1 second to cleanup the chamfer.

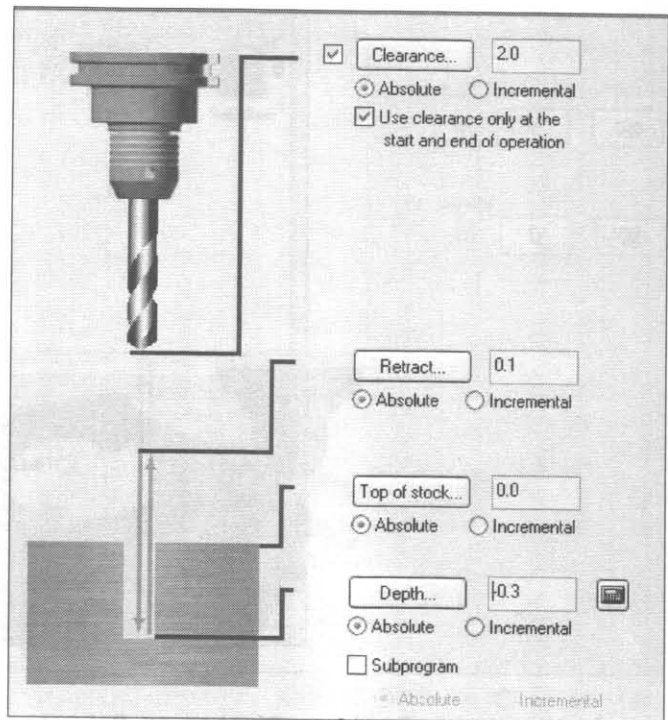
Drill/ Counterbore cycle (G81) is recommended for drilling holes with depths of less than three times the tool diameter.




Mill Level 1

9.6 Set the Linking Parameters

- From the **Tree view list**, select **Linking Parameters**.
- Make sure the parameters are set as shown.



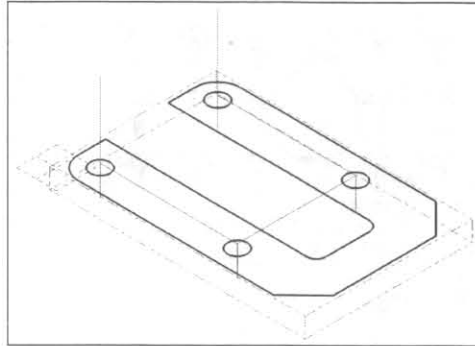
Clearance value sets the height at which the tool rapids to or from the part.
Retract value sets the height at which the tool rapids/feed-rates up to, before the next step down.
Depth value sets the final machining depth for the drilling operation.

- Select the **OK** button to exit drilling parameters. 
- Press **Alt + T** keys to remove the toolpath display from the screen.

Mill Level 1

STEP 10: DRILL ALL FOUR 1/2" DIAMETER THROUGH HOLES

Toolpath Preview:

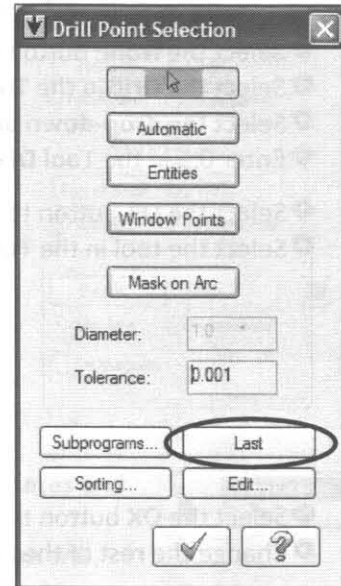


10.1 Drill center points selection

Toolpaths

Drill

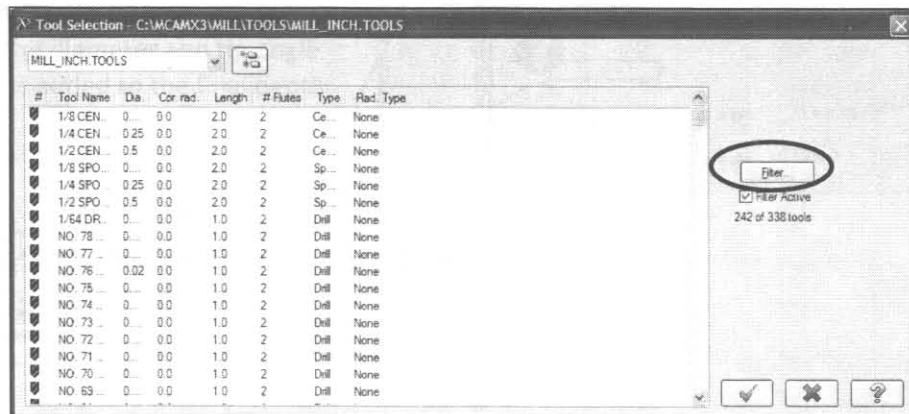
- Select the **Last** button in the **Drill Point Selection** dialog box to select the same center points as you did before.

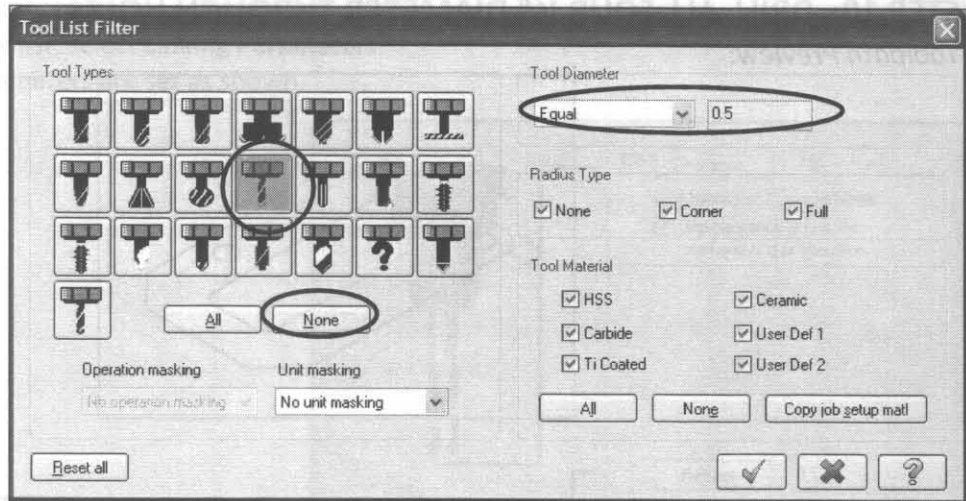



- Select the **OK** button to exit **Drill Point Selection**.

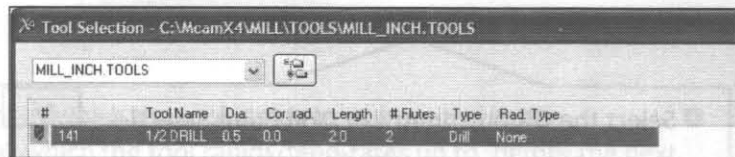
10.2 Select the 1/2" diameter drill and set the Tool parameters

- From the **Tree view list** select **Tool**.
- Click on the **Select library tool** button.
- Select the **Filter** button in the **Tool Selection** dialog box.

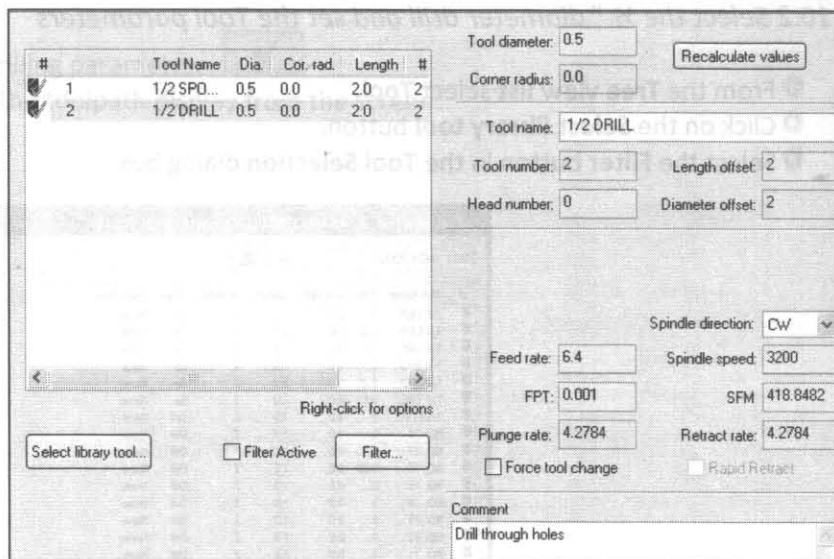




- Select the **None** button to disable any previous tool selection as shown.
- Select the **Drill** in the **Tool Types** list.
- Select the **drop-down arrow** in the **Tool Diameter** field and select **Equal**.
- Enter 0.5 in the **Tool Diameter** value box.
- Select the **OK** button to exit **Tool List Filter**. 
- Select the tool in the **Tool Selection** window.

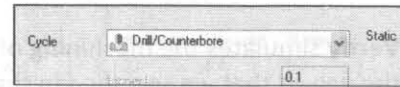


- Select the **OK** button to exit **Tool Selection**. 
- Change the rest of the parameters in the **Tool** page as shown.



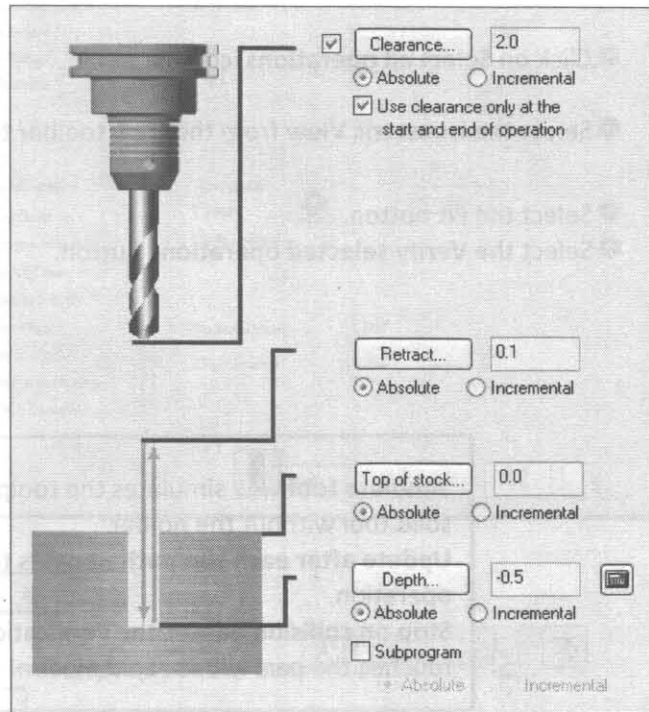
Mill Level 1

- Click on the **Cut parameters** and make sure the drilling **Cycle** is set to **Drill/Counterbore**.



10.3 Set the Linking Parameters

- Change the **Depth** to -0.50 as shown.



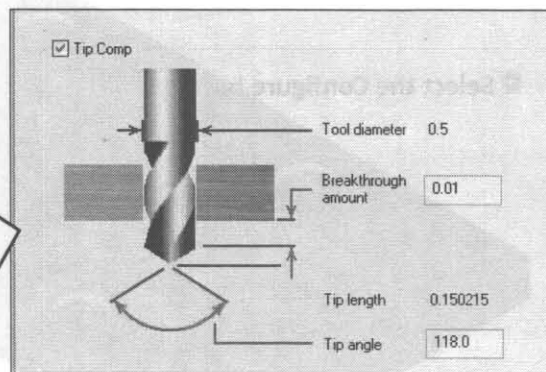
- From the Tree view list, select the plus sign in front of the **Linking Parameters** and then select **Tip Comp**.



- Enable the **Tip Comp** to cut deeper than the final depth with the tip of the drill.

Breakthrough amount value allows you to give an extra amount for the tool to go deeper than the final depth to prevent any remaining material for the cut-outs.

Tip length value is automatically calculated by the system based on the diameter and tip angle of the tool. The value is added to the final depth.



- Select the **OK** button to exit **2D Toolpaths – Drill/CirclesSimple drill**.



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

STEP 11: VERIFY THE TOOLPATH

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

- Click on **Select all operations** icon.



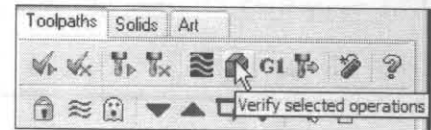
- Select the **Isometric View** from the view toolbar to see the stock.




- Select the **Fit** button.

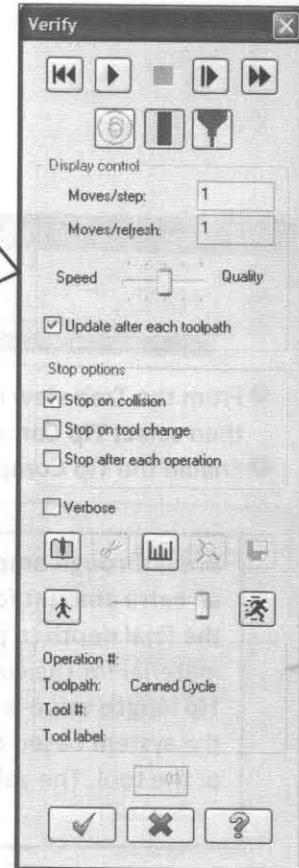


- Select the **Verify selected operations** button.



Simulate tool  simulates the toolpaths, displaying the solid tool without the holder
Update after each toolpath updates the stock after each operation.
Stop on collision pauses the verification when the tool touches the part with a rapid move.

- Select the **Configure** button.

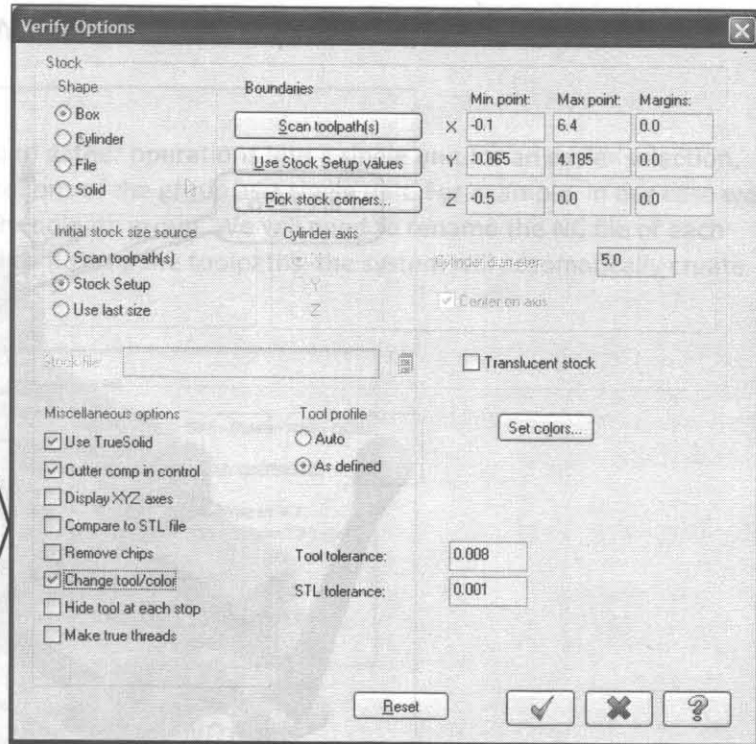



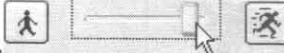

Initial stock size source should be set to **Stock Setup** to use the stock information from Stock Setup.

Use True Solid allows you, after verifying the part, to rotate and magnify the part to more closely check features, surface finish or scallops.

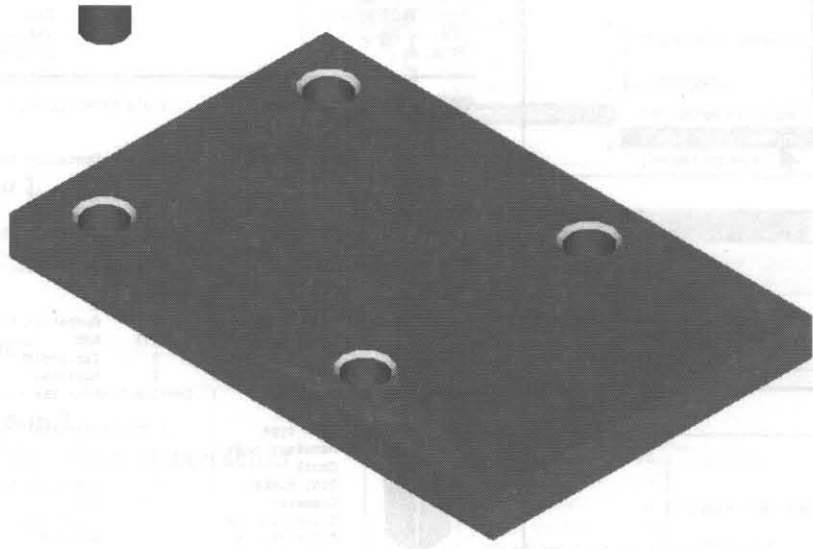
Cutter comp in control allows Verify to use the information regarding the tool diameter and to simulate the cutter compensation.

Change tool/color to change the color of the cut stock to indicate tool changes in the toolpath.

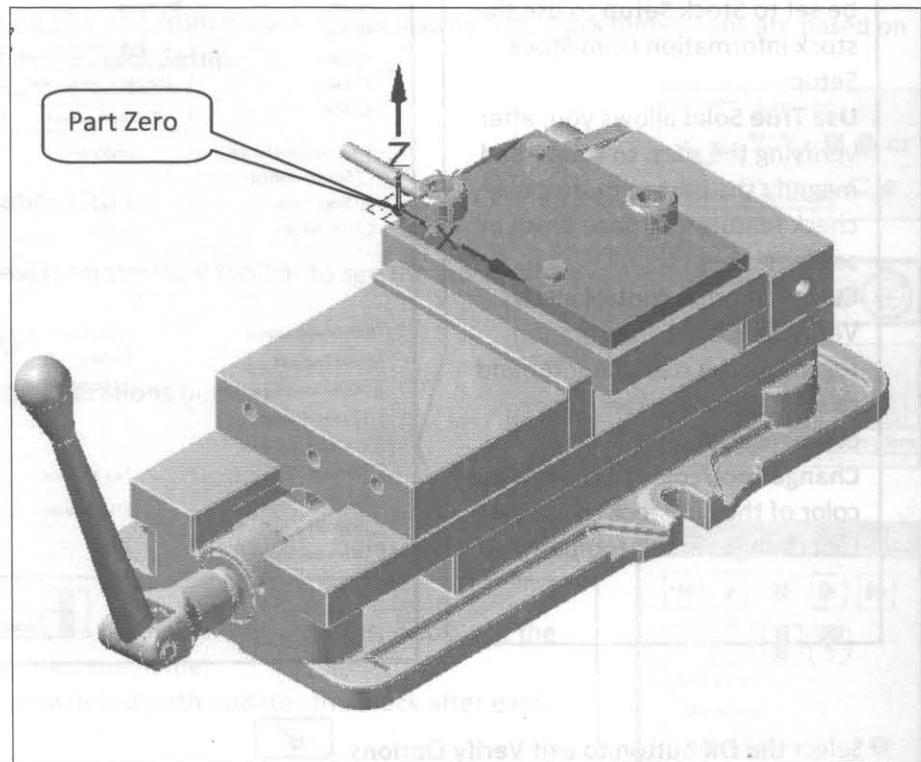


- Select the **OK** button to exit **Verify Options**. 
- Set the **Verify speed** by moving the slider bar in the speed control bar. 
- Select the **Play** button to start simulation. 




The part should appear as shown in the following picture.



SUGGESTED FIXTURE (SETUP 2):



SETUP SHEET (SETUP 2)

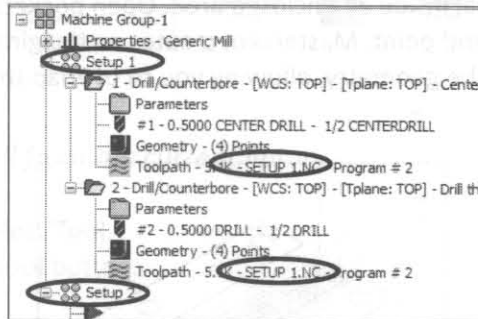
Operation List of YOUR NAME 2.MCX	
Proj./Part No. : 0	Date : 03/23/09
Drawing No. : 1	Customer : C:\MCMX4\MILL\NCI\YOUR NAME 1.NCI
Prog. No. : 2	Programmer : 0
	Tool type : 0.375 Endmill1 Flat 3/8 FLAT ENDMILL Manufact.code : Chuck : Tool Number : 3 Operation time: 0h 5m 15s Feedrate : 30 Diameter : 0.375 RPM : 8500 Plunge feed r. : 6.332 Corner radius : 0 Tip angle : 0 Diam. offset : 3 Flute length : 0.75 Material : ALUMINUM ... Length offset : 3 Comment : Pocket Machine the open pocket.
	Tool type : 0.75 Endmill1 Flat 3/4 FLAT ENDMILL Manufact.code : Chuck : Tool Number : 4 Operation time: 0h 2m 10s Feedrate : 36 Diameter : 0.75 RPM : 4500 Plunge feed r. : 6.4176 Corner radius : 0 Tip angle : 0 Diam. offset : 4 Flute length : 2 Material : ALUMINUM ... Length offset : 4 Comment : Contour Contour the outside profile using ramp cutting method.
	Tool type : 0.75 Chamfer mill 3/4 CHAMFER MILL Manufact.code : Chuck : Tool Number : 5 Operation time: 0h 4m 6s Feedrate : 8.5568 Diameter : 0.75 RPM : 713 Plunge feed r. : 8.5568 Corner radius : 0 Tip angle : 45 Diam. offset : 5 Flute length : 1 Material : ALUMINUM ... Length offset : 5 Comment : Contour Chamfer the top of the part.
(Post processor specific average tool change time 0.0, rapid feedrate 250.0)	
Total machining time: 0h 11m 31s	

Mill Level 1

STEP 12: CREATING AND RENAMING TOOLPATH GROUPS; RENAMING THE NC FILE

Working with **toolpath groups** allows you to gather operations into a single unit for an easier selection, and to perform a function on all the operations of the group as a single unit. For example, in our case we want to have a separate program for each toolpath group. We will need to rename the NC file of each group with a unique name, and then, while posting the toolpaths, the system will automatically create a different program for each NC file name.

Step Preview:

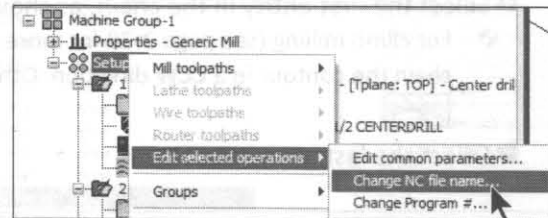


12.1 Rename the current Toolpath Group-1 and the NC file

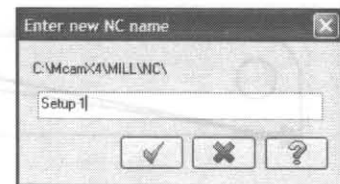
- Double-click the Toolpath Group -1 to highlight it and rename it **Setup 1**.



- Right click on the toolpath group and select **Edit selected operations** and then, select **Change NC file name**.

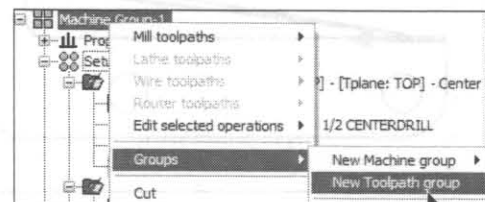


- Enter the new NC name: **Setup 1**
- Select the **OK** button.



12.2 Create a new Toolpath Group

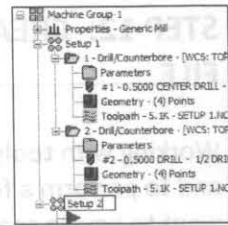
- Right click on the **Machine Group-1**
- Select **Groups** from the drop down list and then select **New Toolpath group**.



Mill Level 1

➤ Rename it: **Setup 2**

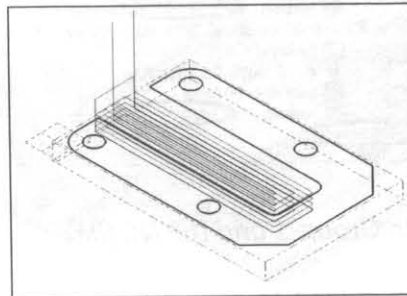
- Make sure that the red insert arrow is below the **Setup 2 group**; otherwise, click on it and drag it below. This ensures that the next operation will be added in the proper group.



STEP 13: REMOVE THE INSIDE MATERIAL USING OPEN POCKET

Pocket toolpaths remove material inside an enclosed area. Open pockets are created from chains that do not have the same start and end point. Mastercam creates an imaginary line between the start and end points of the chain to close the geometry, allowing you to overlap the open area with a percentage of the tool diameter.

Toolpath Preview:

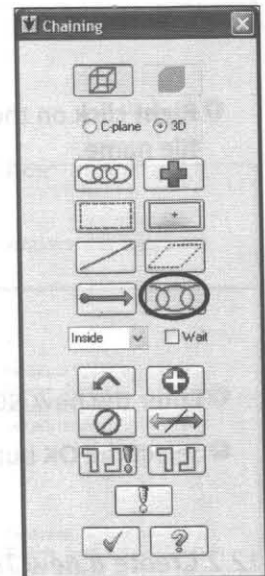
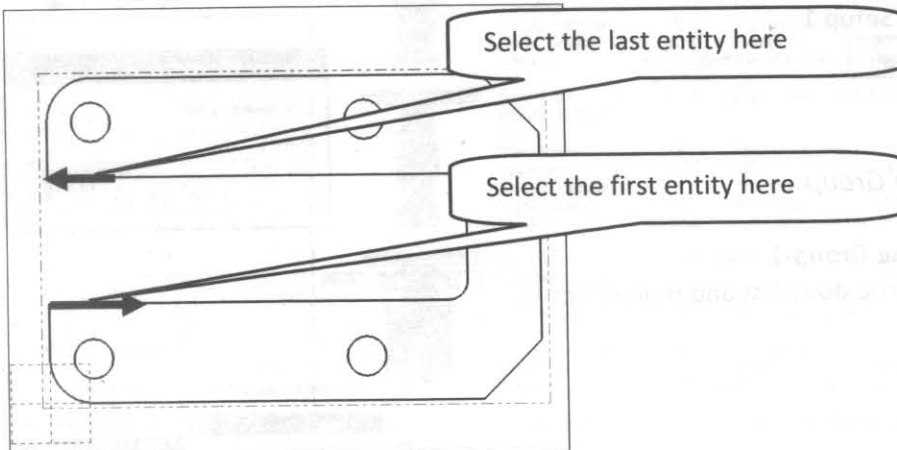


13.1 Partial Chain selection

Toolpaths

➤ **Pocket**

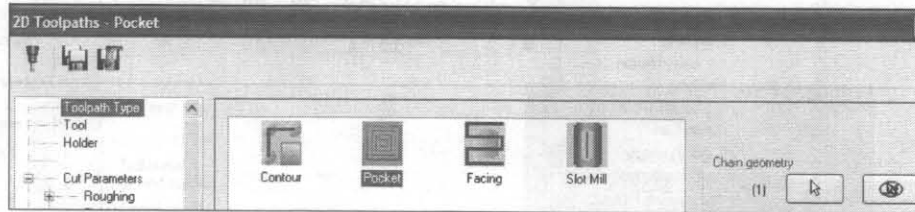
- Enable **Partial** button in the chaining dialog box to be able to select the first and the last entity of the chain.
- Select the first entity in the chain, as shown.
 - For climb milling (see page 2-28 for more details on climb milling), be sure to chain the contour in a **CCW** direction. Otherwise select the **Reverse** button.
- Select the last entity as shown.



Mill Level 1

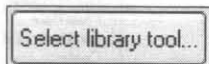
- Select the **OK** button to exit **Chaining**. 

- Note that in the **Toolpath Type** page the **Pocket** icon is automatically selected by **Mastercam** and the **Chain geometry** shows one (1) chain.

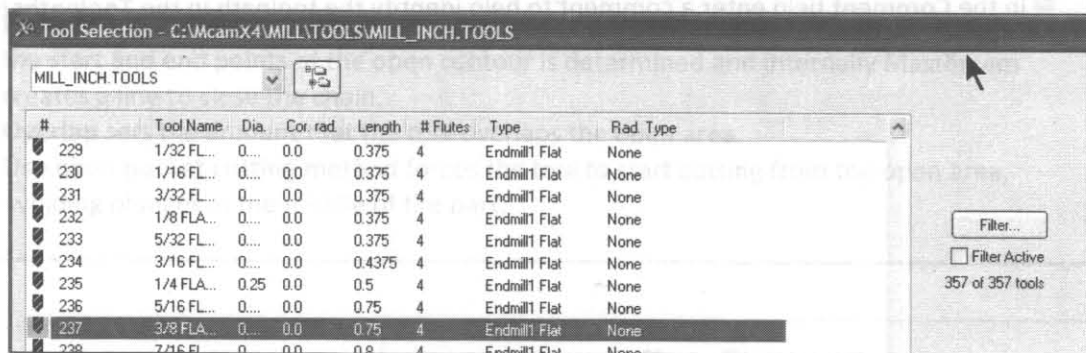


13.2 Select a 3/8" Flat endmill from the current library

- From the **Tree view list**, select **Tool**.
- Click on the **Select library tool** button.



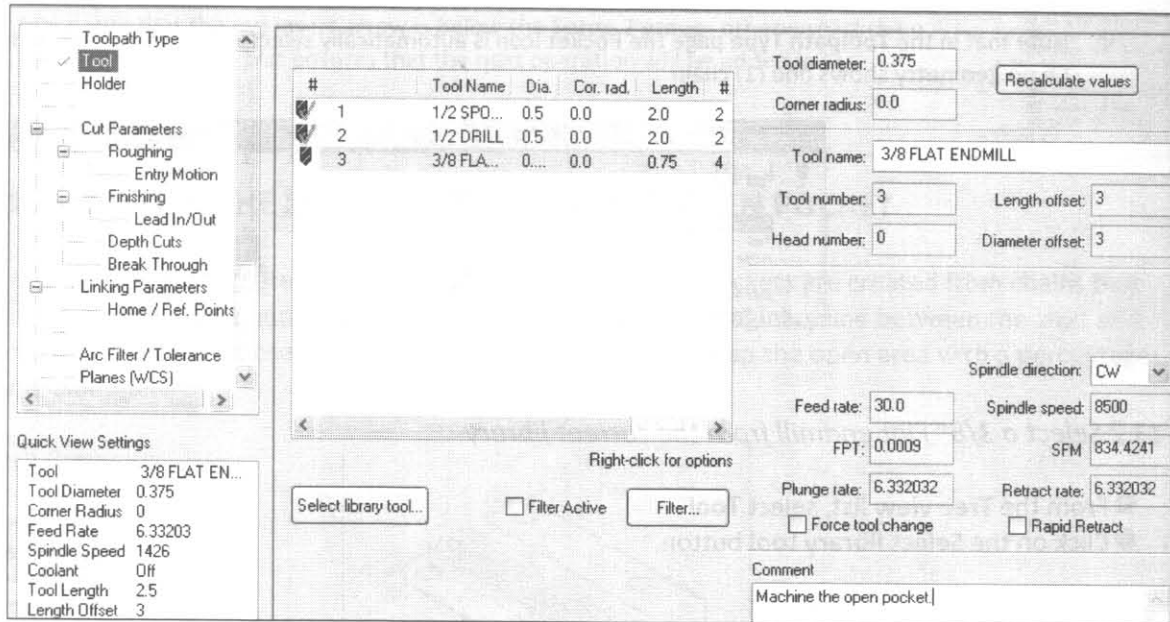
- Disable **Filter Active** to see all the tools available in the current tool library.
- Scroll down and select the 0.375 diameter **Endmill** from the **Tool Selections** list.



- Select the **OK** button to exit **Tool Selection**. 

Mill Level 1

- Make all the necessary changes as shown below.

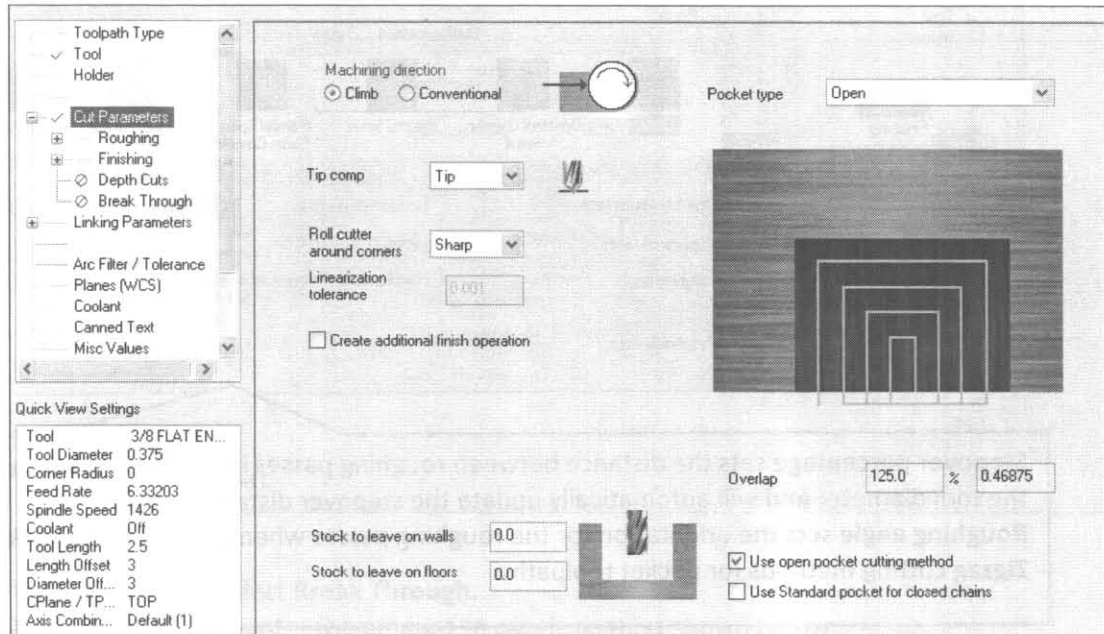


- Note that the **Feed rate**, **Plunge rate**, **Retract rate** and **Spindle speed** are roughly based on Aluminum part material and a carbide tool. Change them if needed.
- In the **Comment** field enter a comment to help identify the toolpath in the **Toolpaths/Operations Manager**

Mill Level 1

13.3 Set the Cut Parameters

- Make all the necessary changes as shown below.

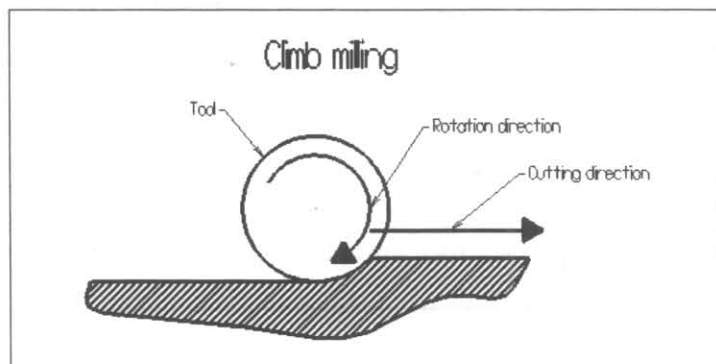


Machining Direction set to **Climb** cuts in one direction with the tool rotating in the opposite direction of the tool motion. See the graphic below.

Pocket type set to **Open** can be used with open and closed chains. The distance between the start and end points of the open contour is determined and internally Mastercam creates a line to close the chain.

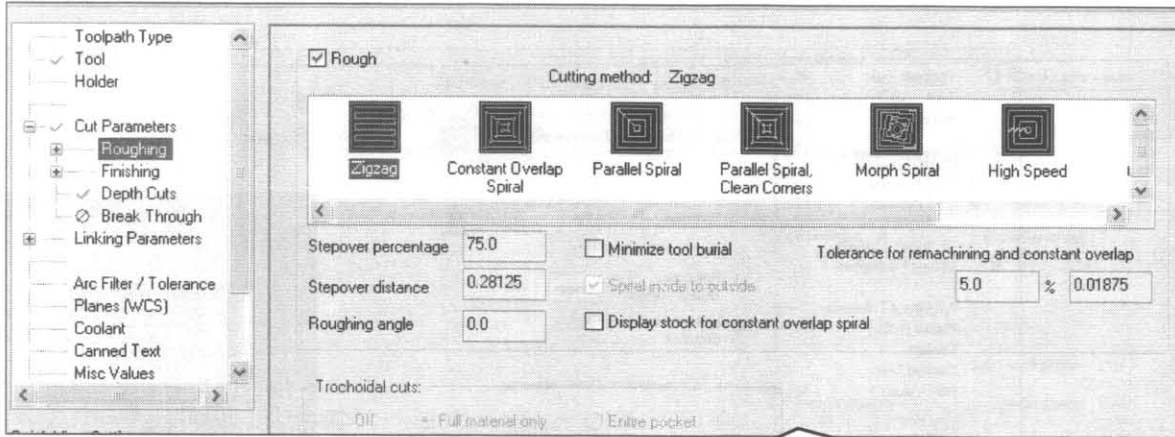
Overlap sets the amount that the tool overlaps the open area.

Use open pocket cutting method forces the tool to start cutting from the open area, avoiding plunges in the middle of the part.



Mill Level 1

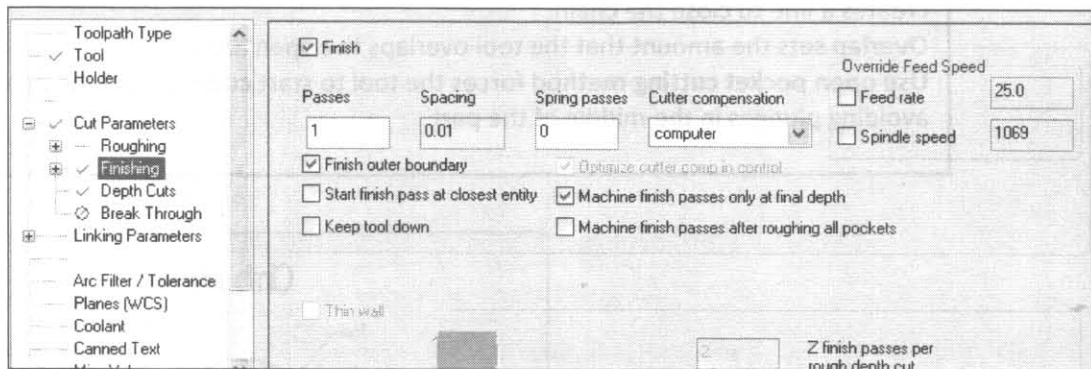
- From the **Tree view list**, select **Roughing**.



Stepover percentage sets the distance between roughing passes in the XY axis as a percentage of the tool diameter and will automatically update the stepover distance.
Roughing angle sets the orientation for the roughing passes when you use the **One Way** or **Zigzag** cutting methods for pocket toolpaths.

- Note that the **Zigzag** cutting method will not be used in this toolpath; instead, the open pocket cutting method will be used as set in the **Cutting Parameters** page.

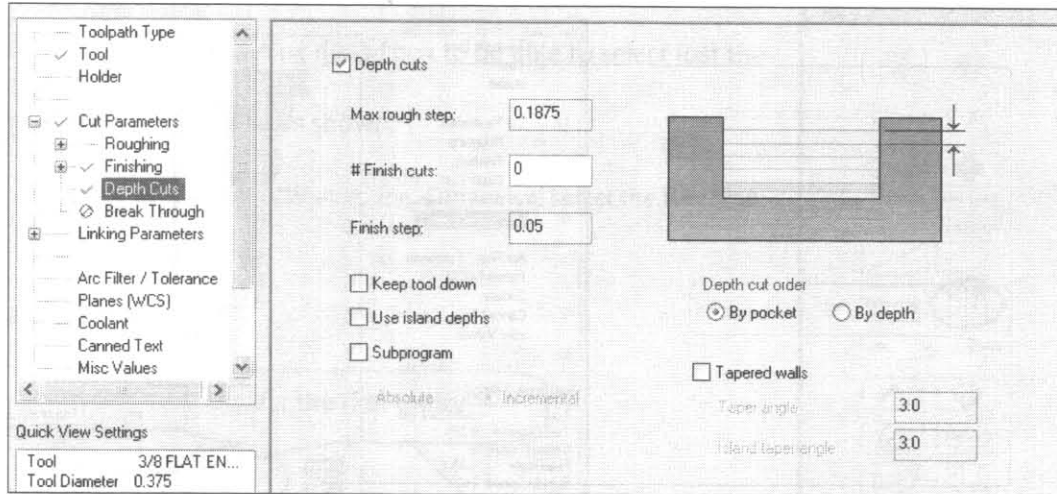
- From the **Tree view list**, select **Finishing**.



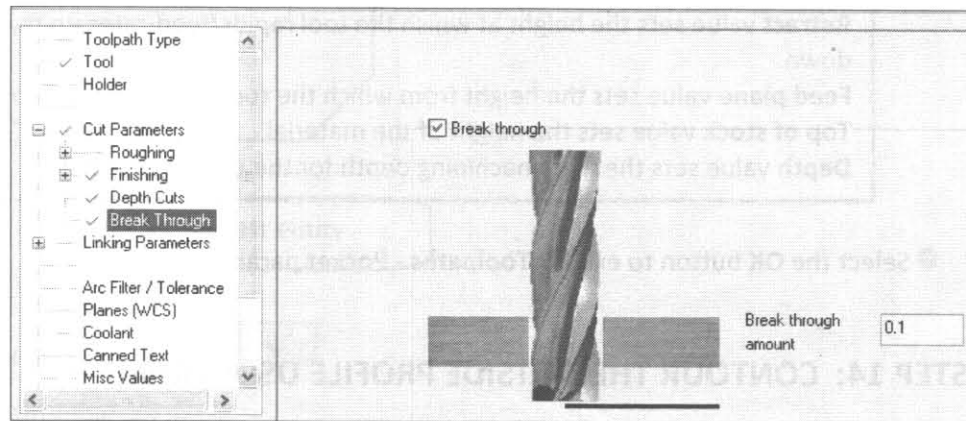
The current settings in the **Finish** area enables one finish pass of a 0.01 amount, performed around the pocket walls at the final depth.

Mill Level 1

- From the **Tree view list**, select the **Depth Cuts**.
- Enable the box in front of the **Depth cuts** and change the parameters to divide the total depth in increments not bigger than the tool radius as shown below.



- From the **Tree view list**, select **Break Through**.
- Enable **Break through** and enter the amount to cut through as shown below.

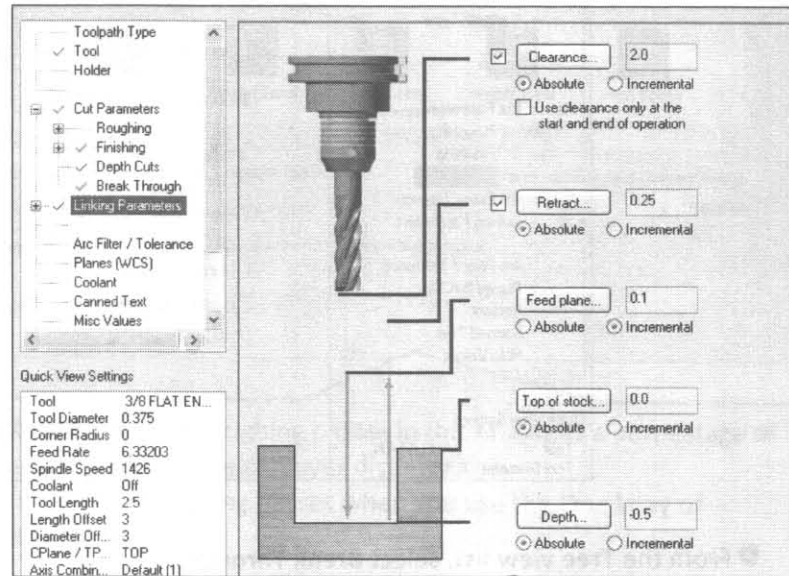


Break Through causes the tool to cut through the material. Always enter the break through amount as a positive number. Mastercam adds the break through amount to the final depth of the toolpath to ensure through-cutting.

Mill Level 1

13.4 Set the Linking Parameters

- From the **Tree view list**, select **Linking Parameters**.
- Enable **Clearance** and enter the **Depth** value as shown below.



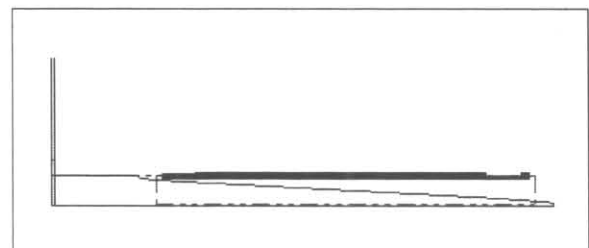
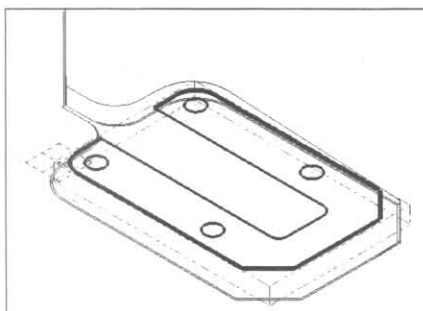
Clearance value sets the height at which the tool rapids to or from the part.
Retract value sets the height at which the tool rapids/feed-rates up to, before the next step down.
Feed plane value sets the height from which the tool plunge-rates into the part.
Top of stock value sets the height of the material.
Depth value sets the final machining depth for the pocket operation.

- Select the **OK** button to exit **2D Toolpaths - Pocket** parameters. 

STEP 14: CONTOUR THE OUTSIDE PROFILE USING RAMP

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

Toolpath Preview:




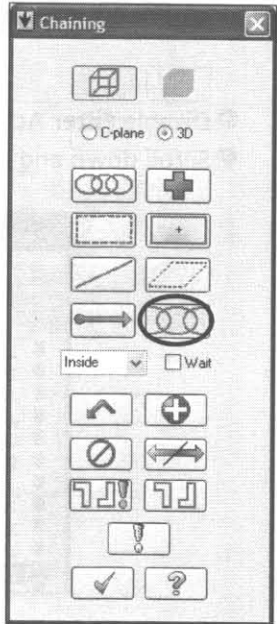
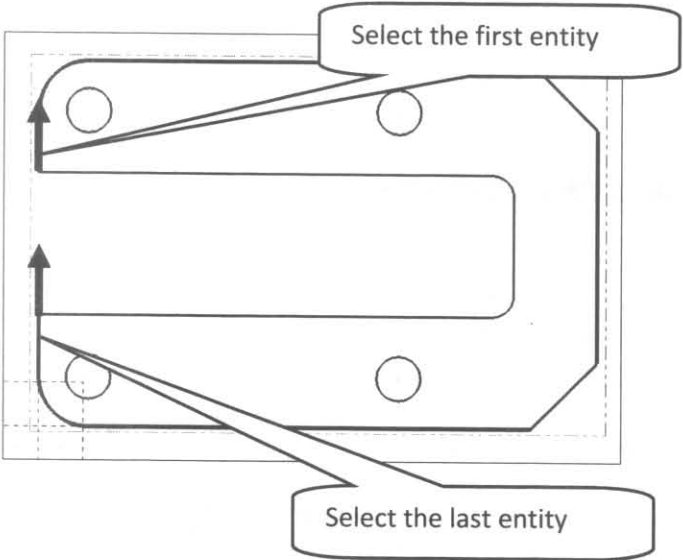
Mill Level 1

14.1 Chain selection

Toolpaths

➤  **Contour**

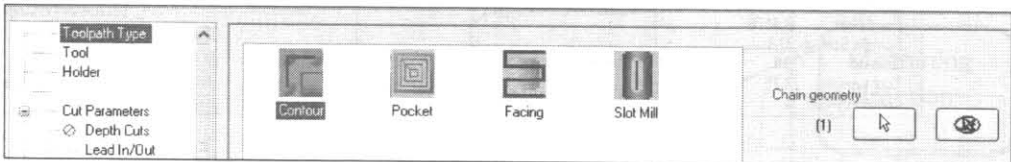
- Enable **Partial** button in the chaining dialog box to be able to select just the outside contour.
- Select the first entity in the chain, as shown.
- Be sure to chain the contour in a **CW** direction. Otherwise, select the **Reverse** button. 
- Select the last entity as shown.



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

14.2 Set the Tool parameters and select the 3/4" Flat endmill from the library

- Check the **Toolpath Type** to ensure that the **Contour** icon is selected as shown.



Mill Level 1

- From the **Tree view list**, select **Tool**.
- Click on the **Select library tool** button.

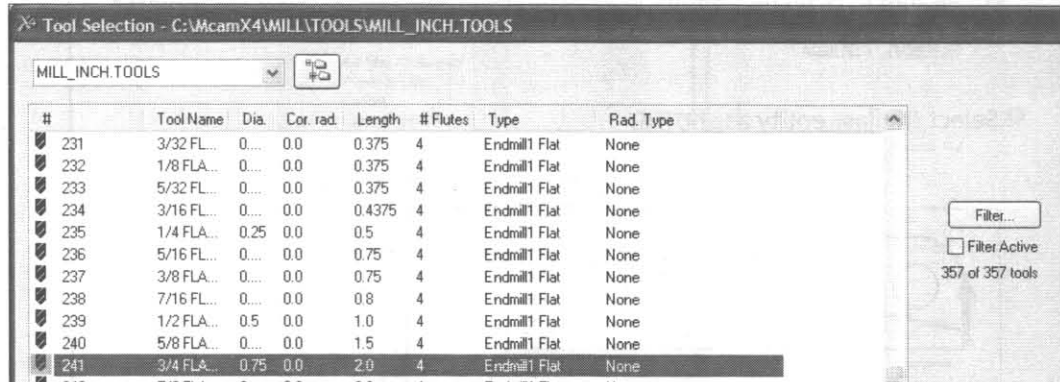
Select library tool...

Filter...

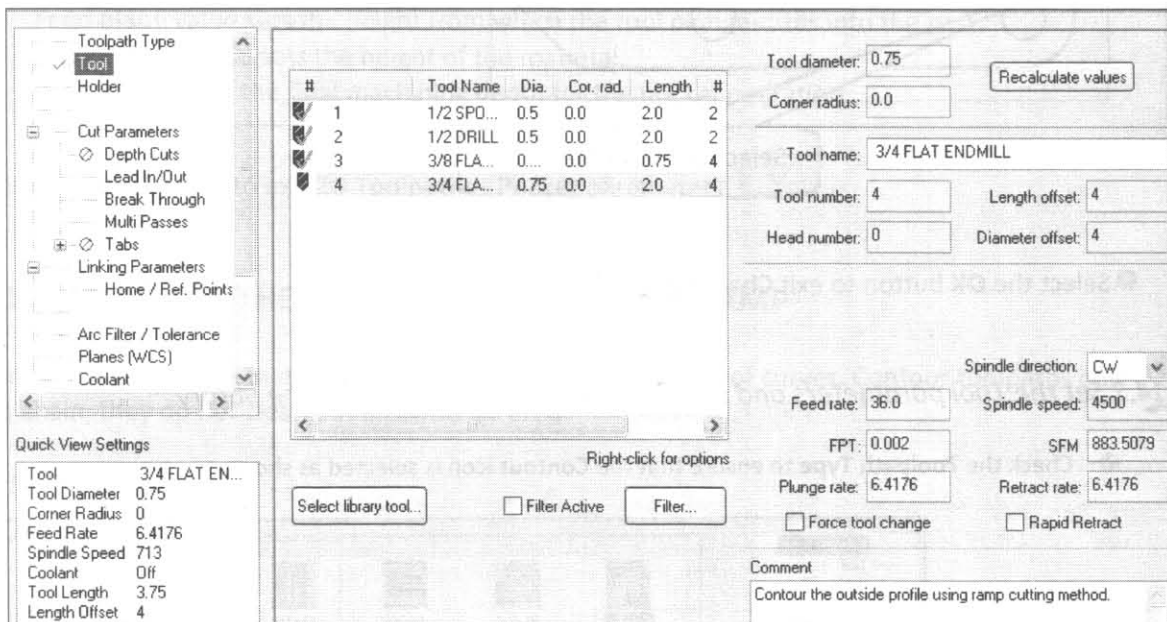
Filter Active

357 of 357 tools

- Disable **Filter Active** to see all the tools available in the current tool library.
- Scroll down and select the 3/4" diameter **Endmill Flat** from the **Tool Selections** list.



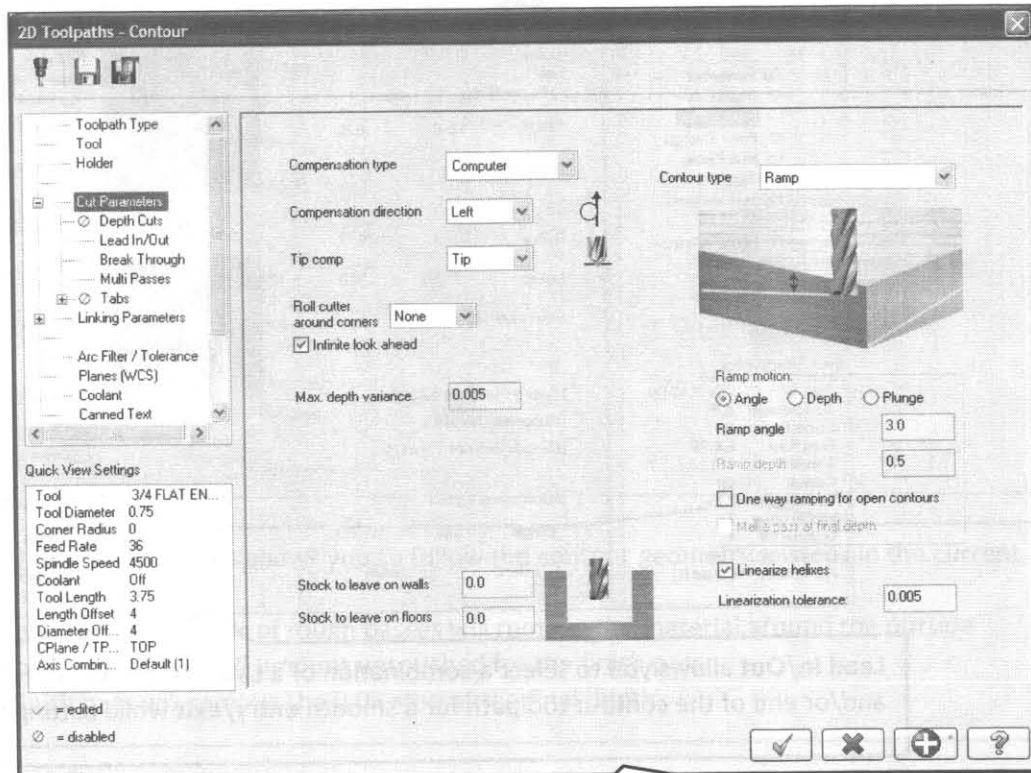
- Select the **OK** button to exit **Tool Selection**.
- Change the parameters to match the following screenshot.



Mill Level 1

14.3 Set the Cut Parameters

- Make all the necessary changes as shown below.



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.

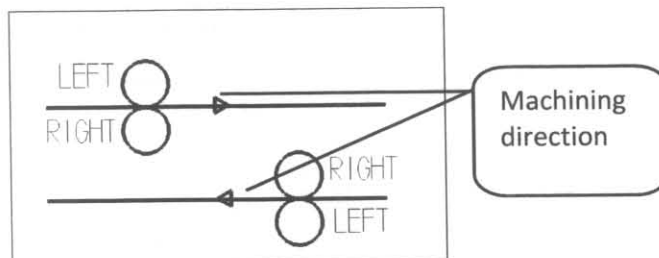
Compensation direction set to **Left** compensates the toolpath to the left of the chain based on the chaining direction. See the graphic below.

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. Set to **None** to not create any extra arcs.

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

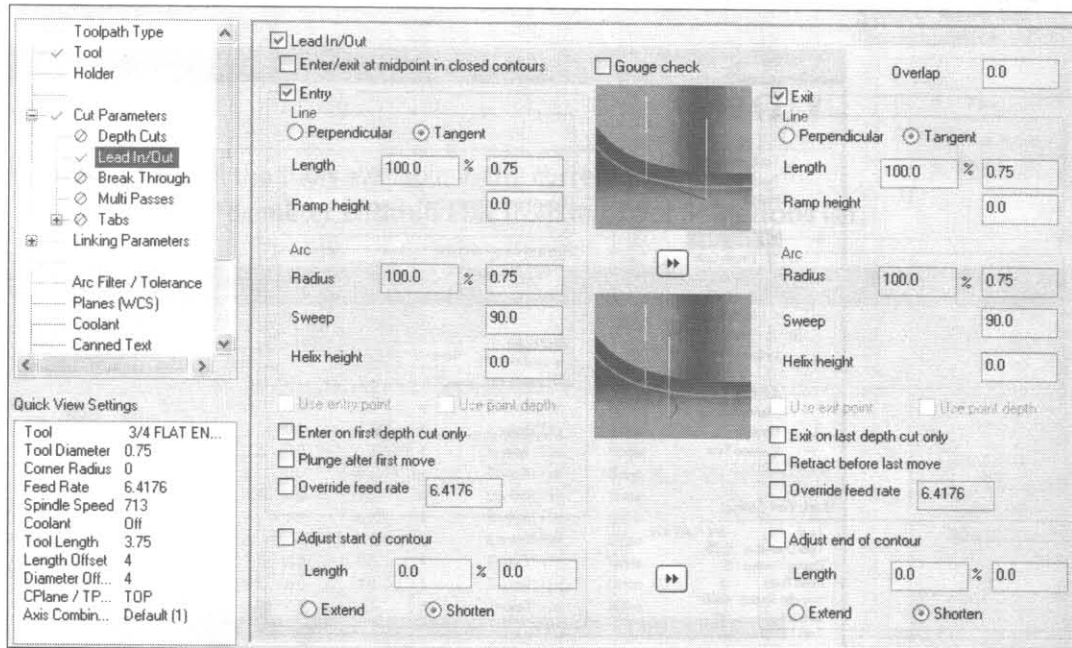
Contour ramp type allows you to use a continuous ramp to transition smoothly between depth cuts, instead of individual plunge cuts.

Cutter compensation:



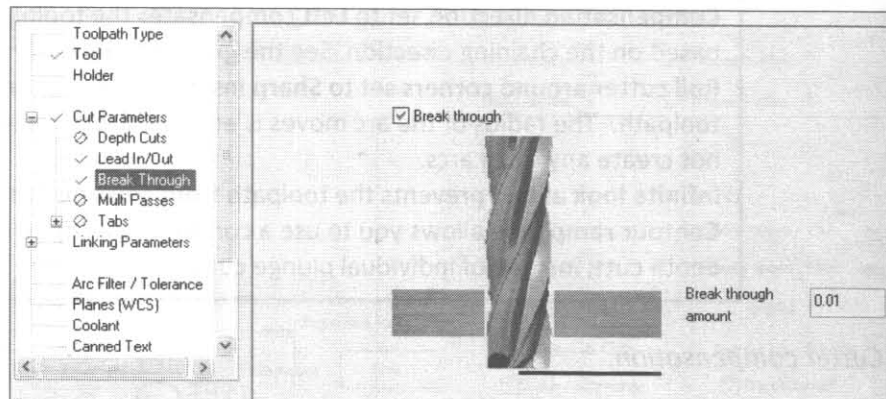
Mill Level 1

- From the Tree view list, select **Lead In/Out** and set the parameters as shown below.



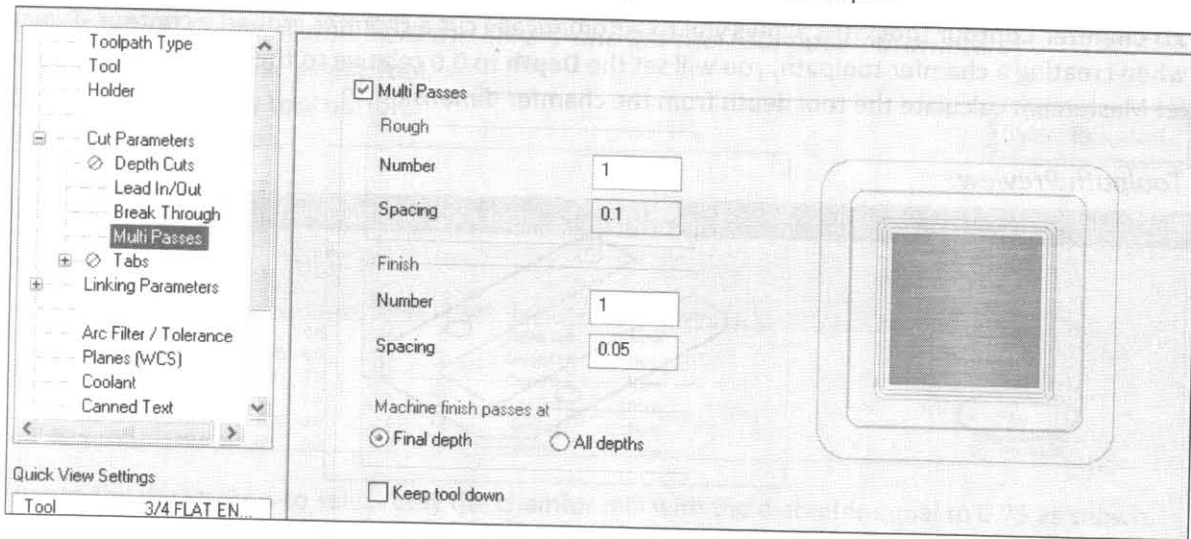
Lead In/Out allows you to select a combination of a **Line** and an **Arc** at the beginning and/or end of the contour toolpath for a smooth **entry/exit** while cutting the part.

- From the **Tree view** list, select **Break Through**, enable the option and enter the **Break through amount** as shown below.



Mill Level 1

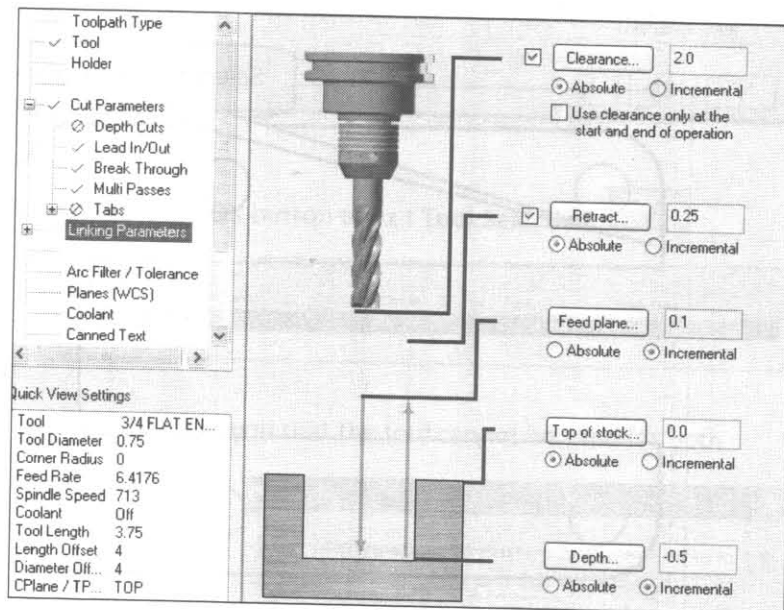
- From the **Tree view list**, select **Multi Passes**
- Enable **Multi passes** to set a rough pass and a finish pass at the final depth.



Multiple cutting passes allows you to follow the contour geometry in steps in the current cutting plane (XY).
 Setting to **1** the **Number** of rough passes will remove the material around the outside contour leaving the 0.05 amount untouched for the finish pass.
 The finish pass will remove the 0.05 step at the final depth.

14.4 Set the Linking Parameters

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



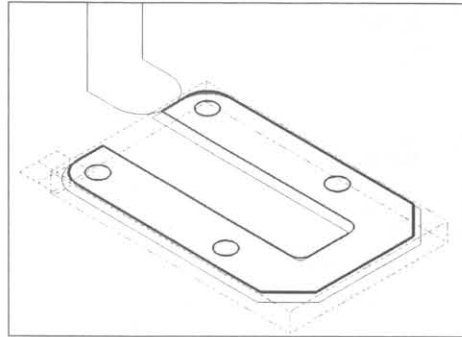
- Select the **OK** button to exit **2D Toolpaths - Contour** parameters.

Mill Level 1

STEP 15: CHAMFER THE PROFILE USING 2D CHAMFER CONTOUR TYPE

2D Chamfer Contour toolpaths allows you to automatically cut a chamfer around a contour. Typically, when creating a chamfer toolpath, you will set the **Depth** to 0.0 relative to the chained geometry, and let Mastercam calculate the tool depth from the chamfer dimensions.


Toolpath Preview:

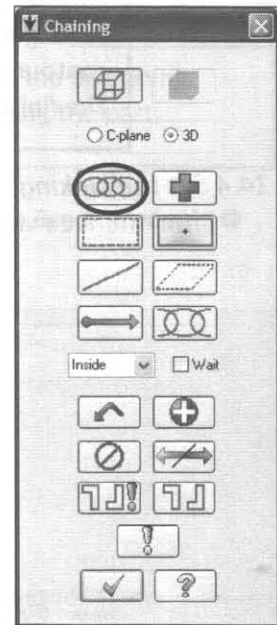
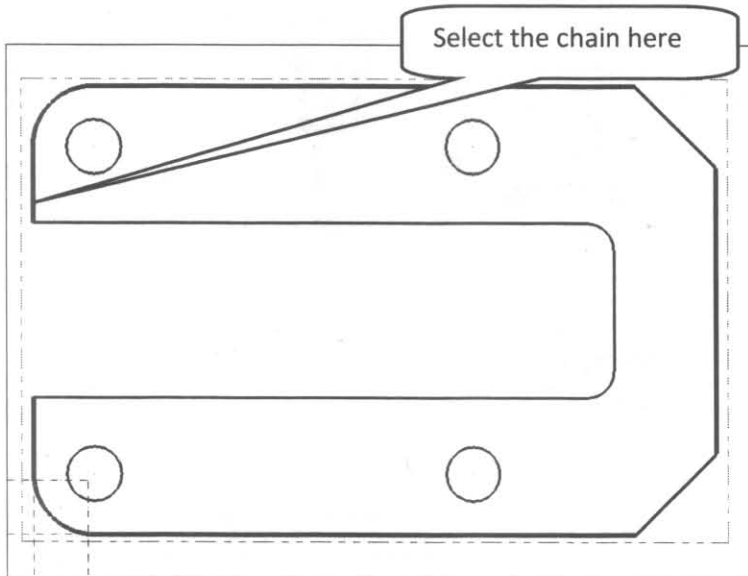


15.1 Chain selection

Toolpaths

Contour

- Make sure that **Chain** button in the chaining dialog box is enabled to be able to select the entire contour.
- Select the contour, as shown.
- Be sure to chain the contour in a **CW** direction. Otherwise, select the **Reverse** button. 

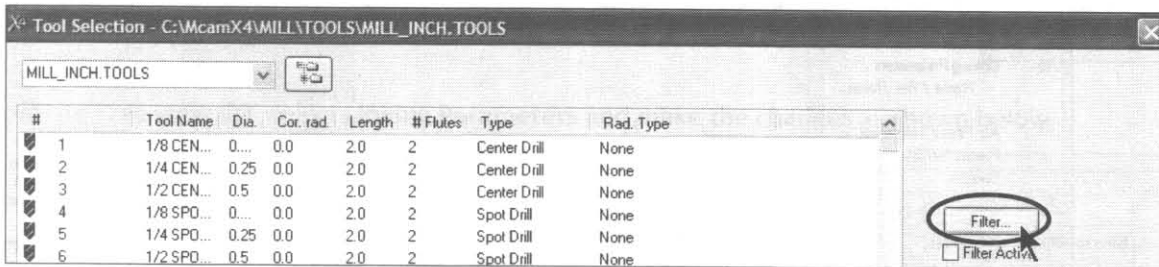
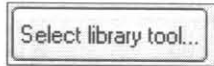


Mill Level 1

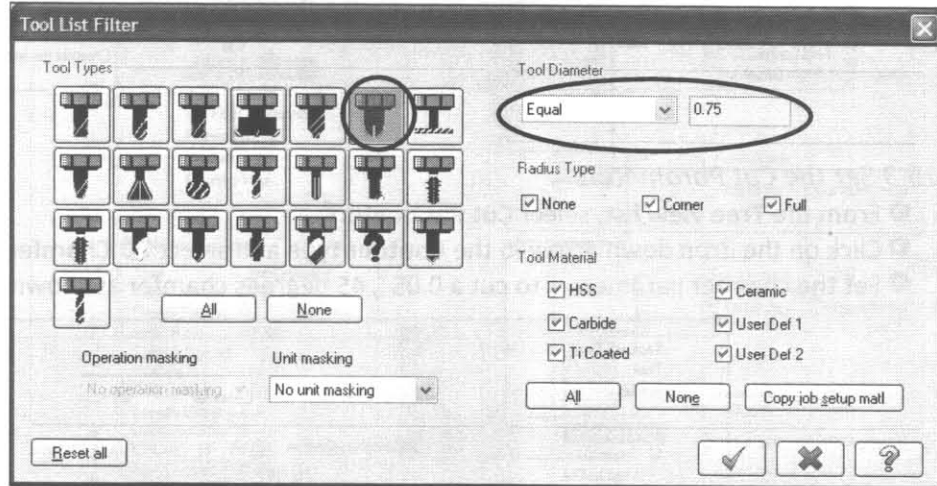
- Select the **OK** button to exit **Chaining**. 

15.2 Select the 3/4" Chamfer mill from the library and set the Tool page parameters

- From the **Tree view list**, select **Tool**.
- Click on **Select library tool** button.
- Select the **Filter** button.

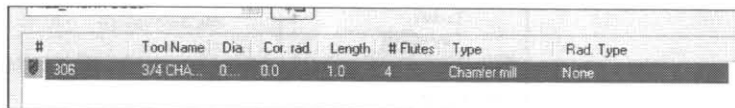


- Change the parameters to select only the chamfer mill with the diameter equal to 0.75 as shown.

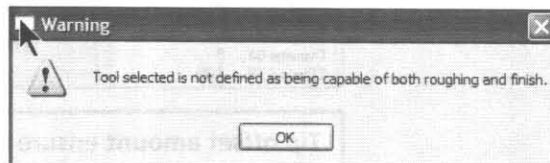


- Select the **OK** button to exit. 

- Click on the tool to highlight it, and then select the **OK** button to exit **Tool Selection**. 

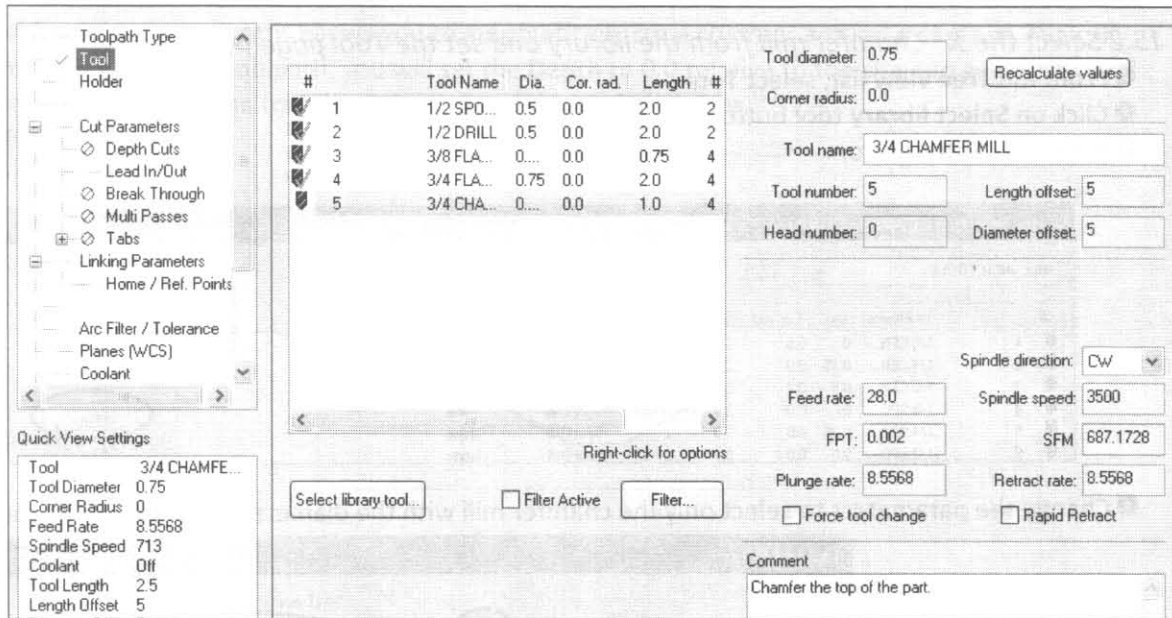


- Select the **OK** button to accept the warning that tells you that the tool cannot be used for both roughing and finish.



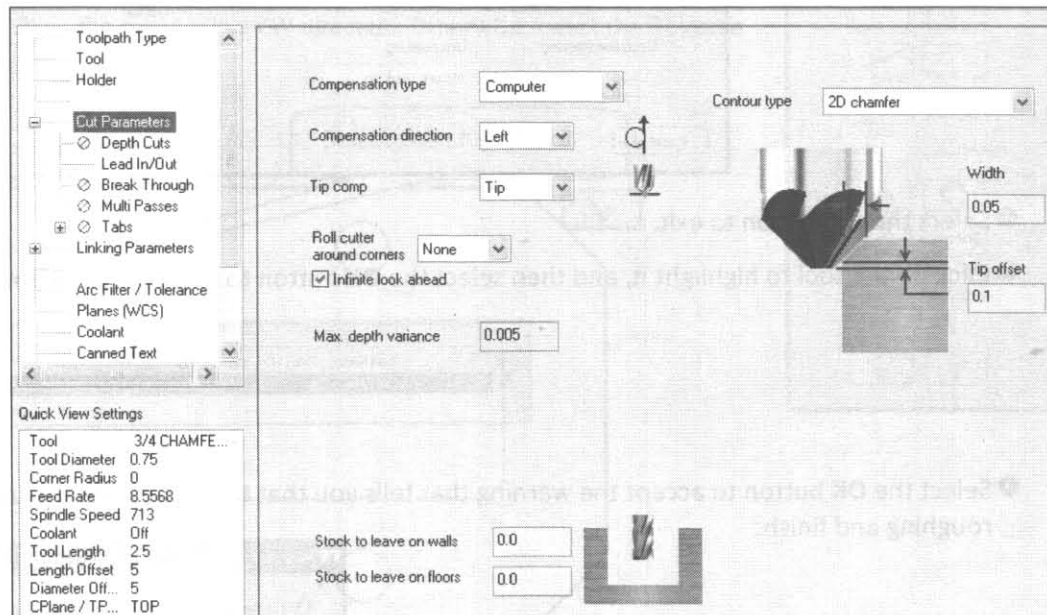
Mill Level 1

- Set the **Tool** page parameters as shown.



15.3 Set the Cut Parameters

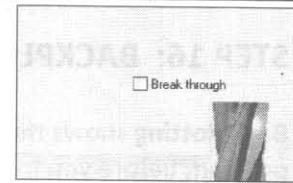
- From the **Tree view list**, select **Cut Parameters**
- Click on the drop down arrow in the **Contour type** and select **2D Chamfer**.
- Set the chamfer parameters to cut a 0.05", 45 degrees chamfer as shown.



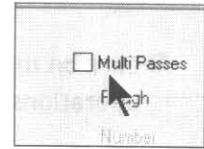
Tip offset amount ensure that the tip of the tool clears the bottom of the chamfer.

Mill Level 1

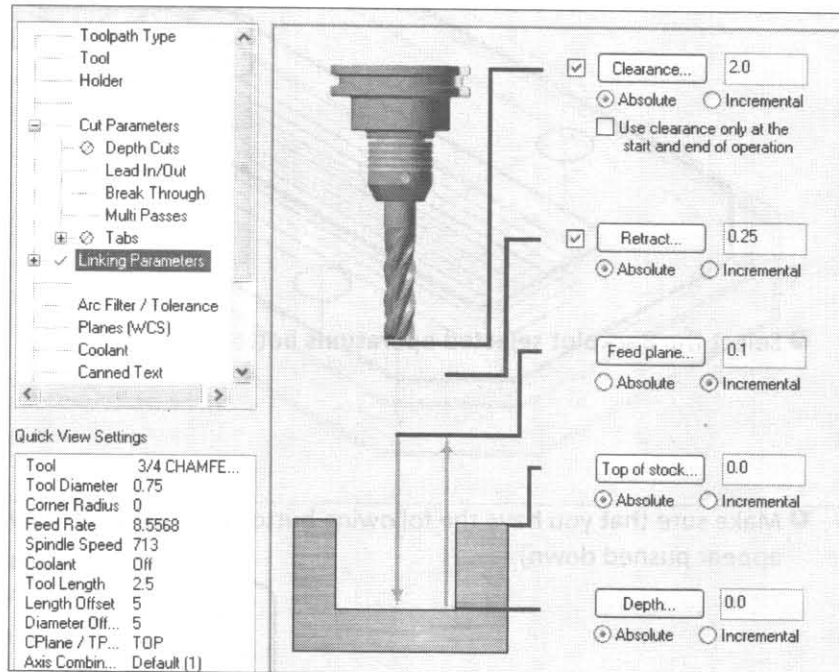
- From the **Tree view list**, select **Break through** and disable the option.



- From the **Tree view list**, select **Multi Passes** and disable the option.



- From the **Tree view list**, select **Linking Parameters** and make the changes as shown below.



- Note that the **Depth** is set to 0 relative to the top of the stock to let Mastercam to calculate the tool depth from the chamfer dimensions.

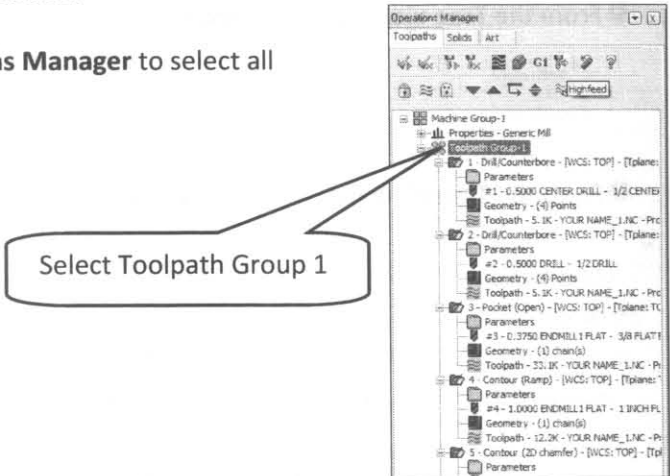
- Select the **OK** button to exit **2D Toolpaths - Contour** parameters.

Mill Level 1

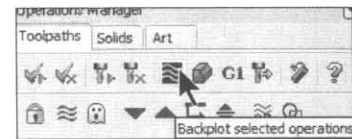
STEP 16: BACKPLOT THE TOOLPATHS

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.

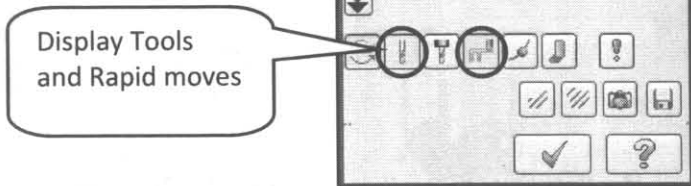
- 1 Click on the **Toolpath Group** in the **Toolpaths Manager** to select all operations.



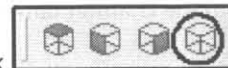
- 2 Select the **Backplot selected operations** button.



- 3 Make sure that you have the following buttons turned on (they will appear pushed down).



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



- 5 Select the **Fit** button.



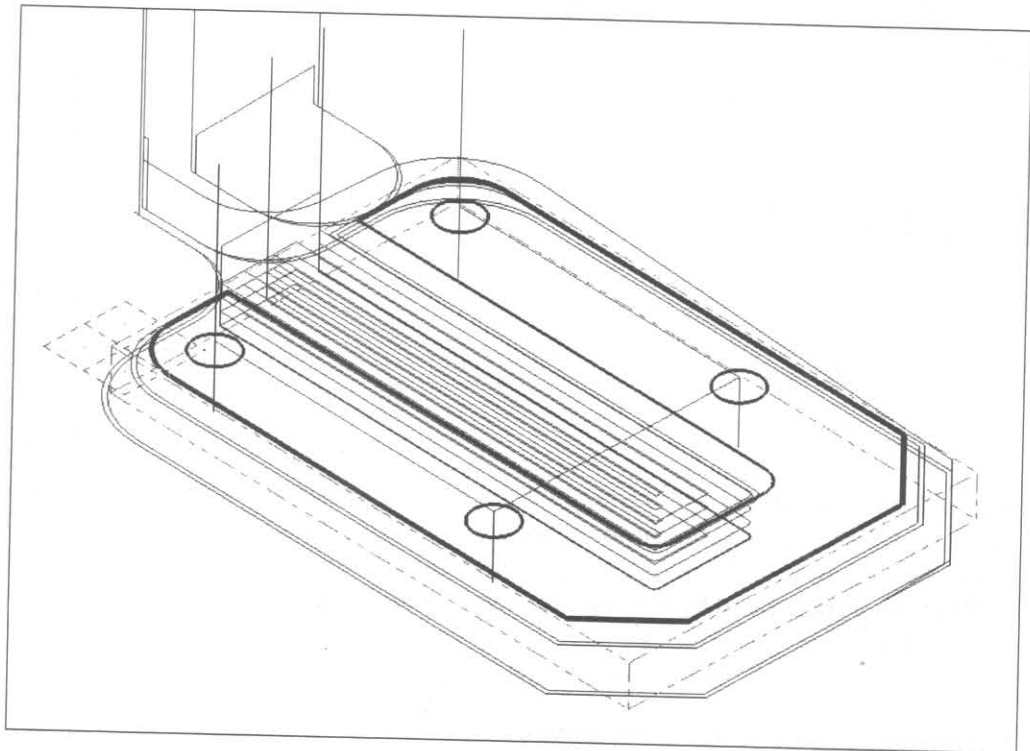
- 6 You can adjust the speed of the backplot.



- 7 You can step through the Backplot by using the **Step forward** or **Step back** buttons.



• Select the **Play** button in the VCR bar.

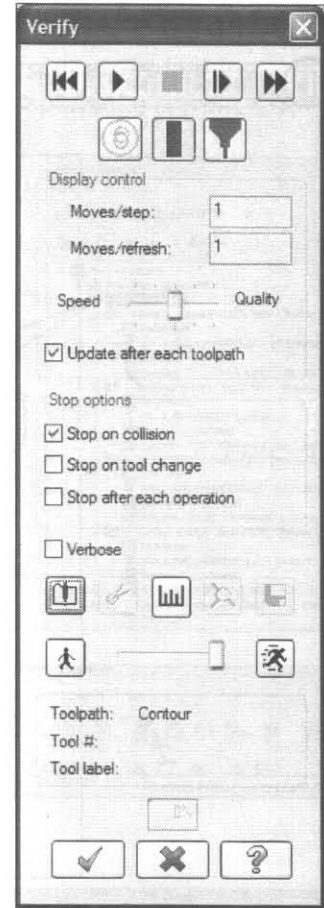


• Select the **OK** button to exit **Backplot**.

STEP 17: VERIFY THE TOOLPATH

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





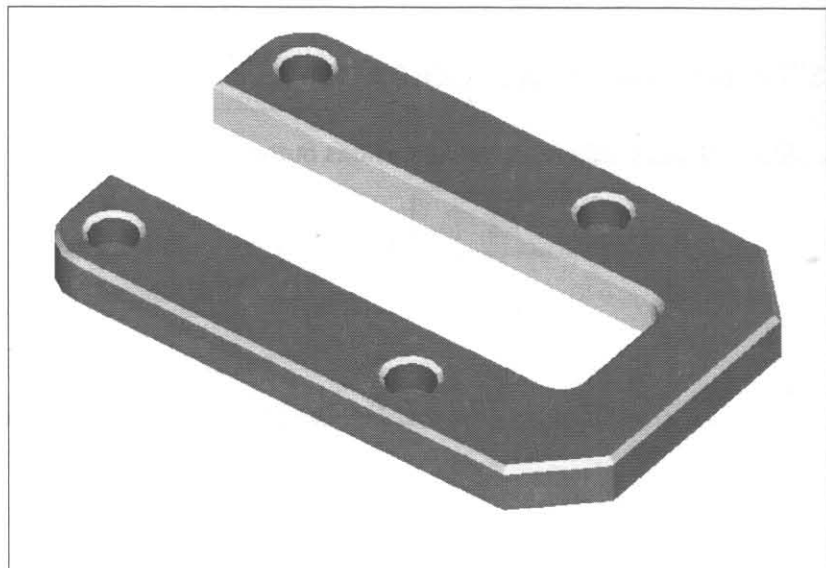
- Set the **Verify speed** by moving the slider bar in the speed control



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



The finished part should appear as shown in the following picture.

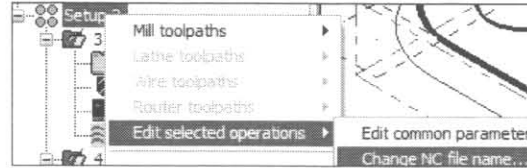


- Select the **OK** button to exit

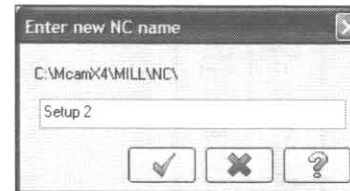


STEP 18: RENAME THE NC FILE FOR THE SETUP 2 TOOLPATH GROUP

- Right click on the **Setup 2** group, select **Edit selected operations** and then select **Change NC file name**.



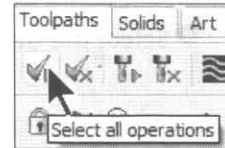
- Enter the new NC name: Setup 2
- Select the **OK** button.



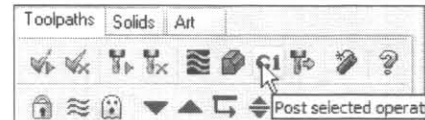
STEP 19: POST THE FILE

Post processing, or **posting**, refers to the process by which the toolpaths in the Mastercam part files are converted to a format that can be understood by the machine tool's control such as G-codes. Generally, every machine controller will require its own post processor, customized to produce code formatted to meet its exact requirements.

- Make sure that all operations are selected, otherwise, **Select all operations**.

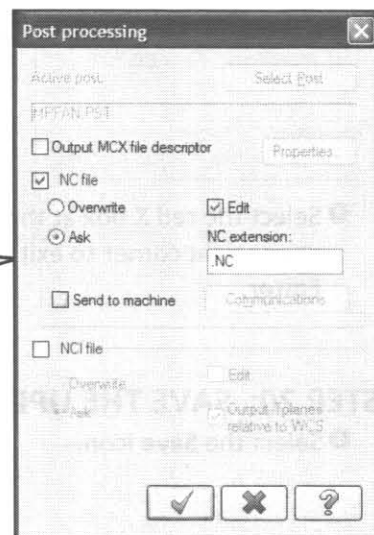


- Select the **Post selected operations** button from **Toolpath Manager**.



- In the **Post processing** window, make all the necessary changes as shown to the right.

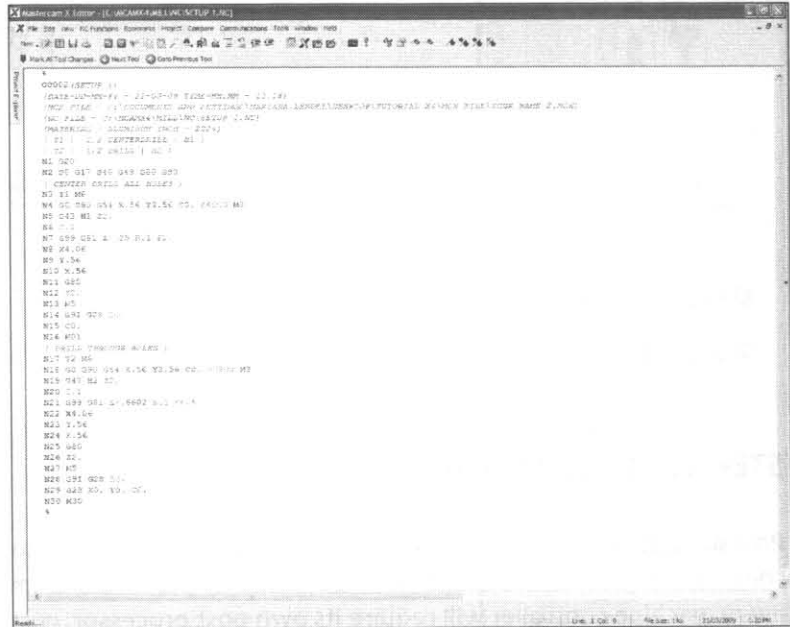
NC file enabled allows you to keep the NC file and to assign the same name as the MCX file.
Edit enabled allows you to automatically launch the default editor.



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

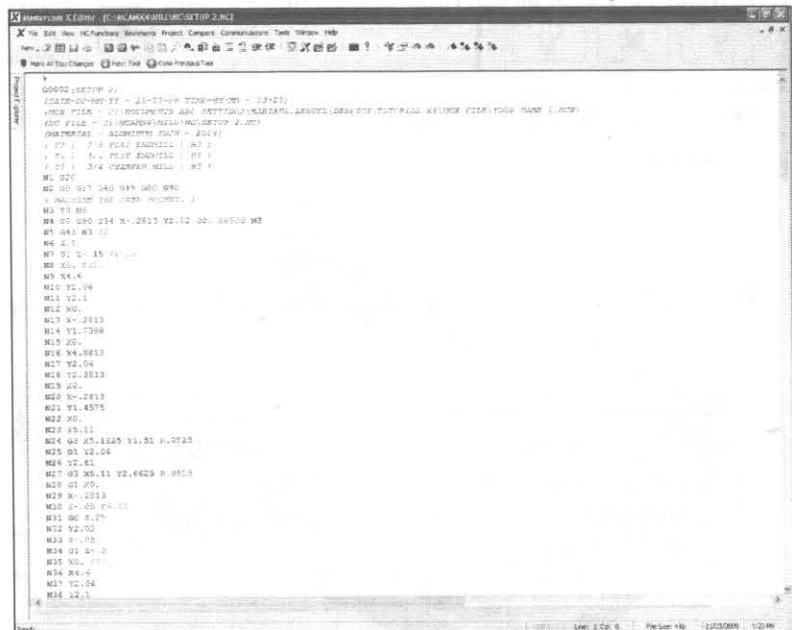
Mill Level 1

- Select the **OK** button to accept in the **File name** field the name as "Setup 1". 
- The "Setup 1" program will be displayed in **Mastercam Editor** as shown.



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

- Select the **OK** button to start the second program "Setup 2". 



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

STEP 20: SAVE THE UPDATED MCX FILE

- Select the **Save** icon.

