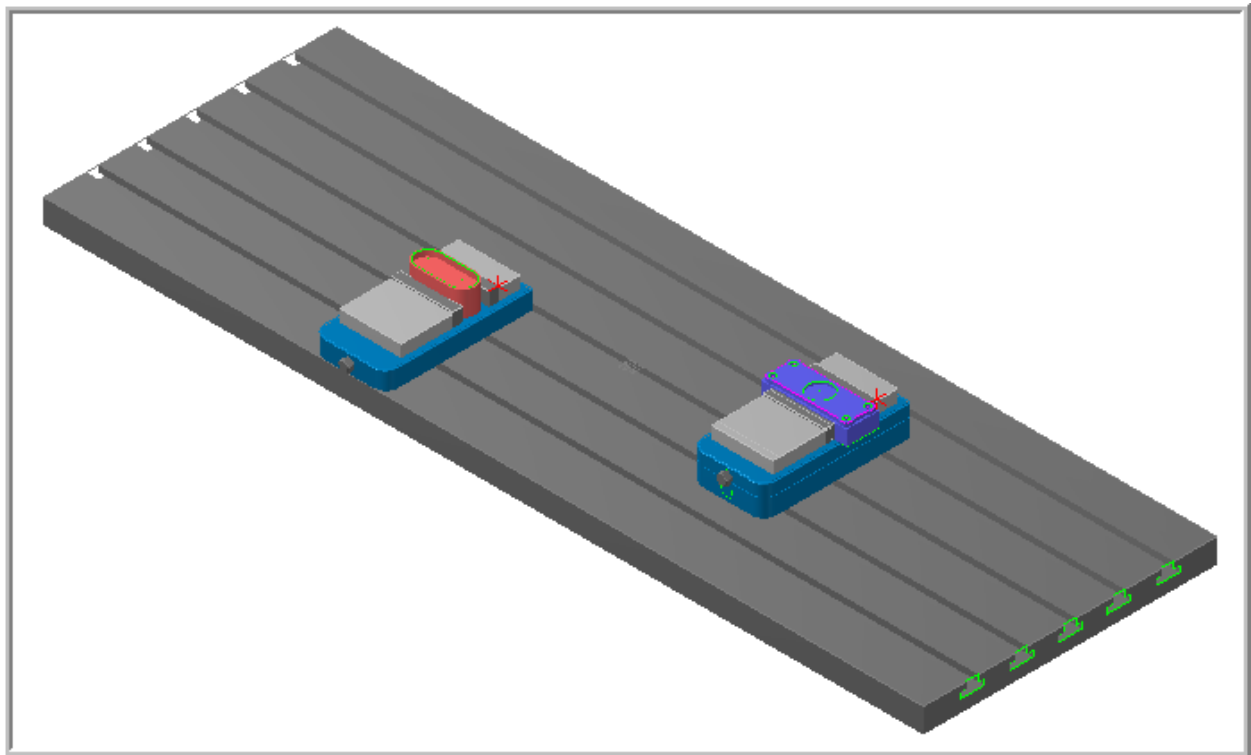


Mastercam X⁸

TRAINING

GUIDE



WCS-WORK OFFSETS PART-4

Mastercam® X3 Training Guide Mill 2D

Published by

CamInstructor Incorporated

330 Chandos Crt.

Kitchener, Ontario

N2A 3C2

www.caminstructor.com

Date: July. 1, 2008

Author: Matthew Manton and Duane Weidinger

ISBN: 978-1-897466-29-2

Copyright © 2008 CamInstructor Inc. - All rights reserved.

This book is protected under the copyright laws of Canada and the United States. All rights are reserved. This document may not, in whole or part, be copied, photocopied, reproduced, translated or reduced to any electronic medium or machine-readable form without prior consent, in writing, from CamInstructor Inc..

National Library of Canada Cataloguing in Publication

To order additional copies of the book contact CamInstructor Inc. at:

Canadian Office Phone 1-877-873-6867

330 Chandos Crt Fax 1-866-741-8421

Kitchener, ON email sales@caminstructor.com

N2A 3C2

Limit of Liability/Disclaimer of Warranty: While the Publisher and Author have used their best efforts in preparing this book, they make no representations or warranties with respect to the accuracy or completeness of the contents of this book and specifically disclaim any implied warranties of merchantability or fitness for a particular purpose. No warranty may be created or extended by representatives. The advice and strategies contained in this book may not be suitable for the readers or users situation. Neither the publisher nor author shall be liable for any damage, loss or any other damages, including but not limited to special, incidental, consequential, or other damages including personal.

Notice

CamInstructor Inc. reserves the right to make improvements to this book at any time and without notice.

Trademarks

Mastercam is a registered trademark of CNC Software, Inc.

All brands are the trademark of their respective owners.

Printed in Canada

Requirements

Use of this book requires Mastercam X3 Mill Level 1. Use of the Multi-media CD requires a computer with speakers, and CD ROM.

Objectives

The learner will create the pocket and contour toolpaths for WCS-Part-4 using two different work offsets.

This Lesson will cover the following topics:

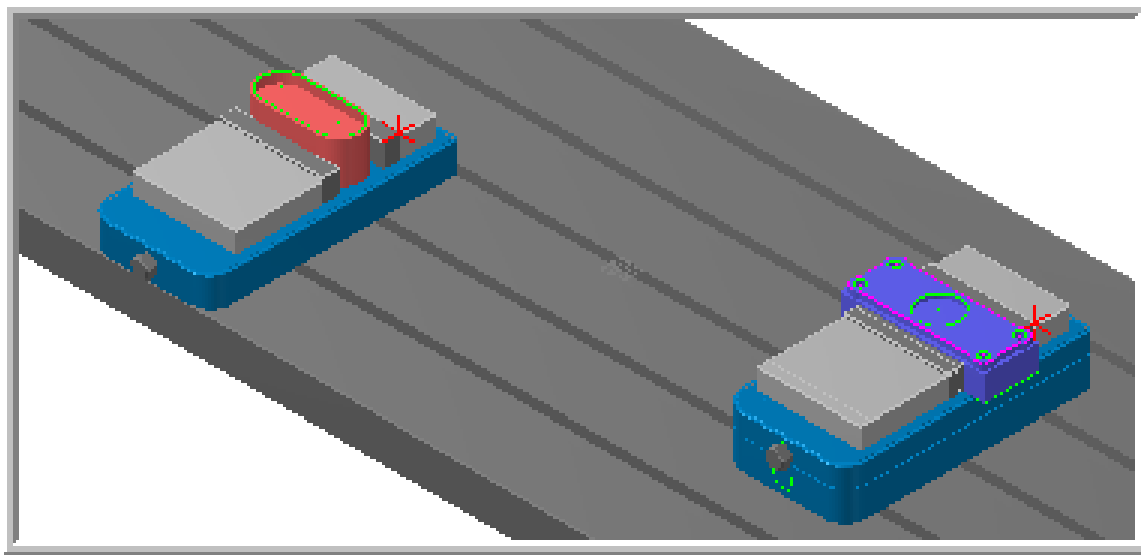
➤ **Create a 2-dimensional part by:**

Using the View Manager

Using Views, Tool Planes and Construction Planes

Create Pocket and Contour toolpaths

WCS-PART-4



WCS-PART-4 - THE PROCESS

- TASK 1:** Setting the environment
- TASK 2:** Introduction - Watch the video
- TASK 3:** Open an Existing file from the Multimedia CD
- TASK 4:** Use View manager to set up a new WCS
- TASK 5:** Machine the pocket left hand setup
- TASK 6:** Use View manager to set up right vice
- TASK 7:** Machine the contour right hand setup
- TASK 8:** Backplot toolpath
- TASK 9:** Post and create the CNC code file

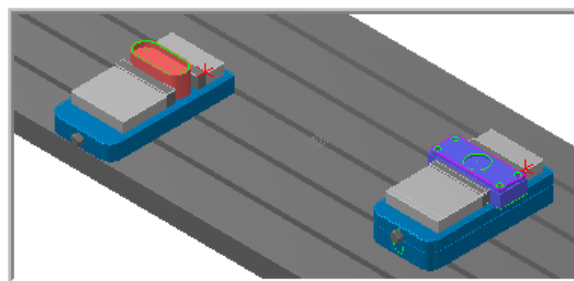
TASK 1: SETTING THE ENVIRONMENT

Before starting the geometry creation you should set up the grid, toolbars and machine type as outlined in the **Setting up the Environment** section at the beginning of this text:

1. Set up the Grid. This will help identify the location of the origin.
2. Customize the toolbars to machine a 2D part.
3. Set the machine type to a Haas Vertical Spindle CNC machine.

TASK 2: INTRODUCTION – WATCH THE VIDEO

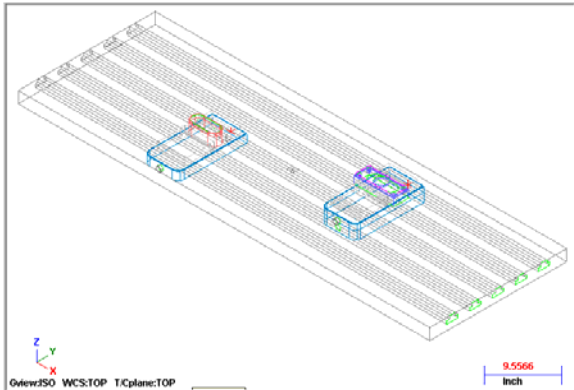
1. Before you start to work on this Lesson review the video on the multimedia CD that came with this text. You will find the video in the “**Tips and Techniques**” section it is entitled **WCS - Part 4 - Moving the WCS - 6 Minutes**
 - In this lesson you will machine two different parts located in two machine vices with the same CNC code file.
 - What you will be setting up is a different work offset for each part. You will be creating two different tool paths and include the offset number in each toolpath. Each toolpath is going to be based on an offset number instead of a coordinate position.
 - The CNC operator can run these parts without concern of how the fixtures are positioned on the table.
 - All the operator needs to do is touch of the two parts correctly before running the machine and input each offset position in the CNC control.
 - For this lesson I will assume that the machine uses G54 to G59 for work offsets.
 - You will use Offset G57 for the left fixture and G58 for the right.



TASK 3: OPEN EXISTING FILE FROM THE MULTIMEDIA CD

➤ On the multimedia CD that came with this text is a folder called **Mastercam-Files**. Open the file **WCS-PART-4.MCX**

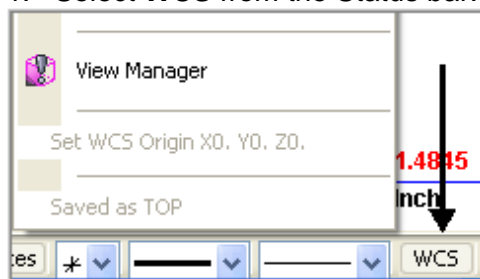
1. **File>Open>WCS-PART-4.MCX**. The file should appear as below.



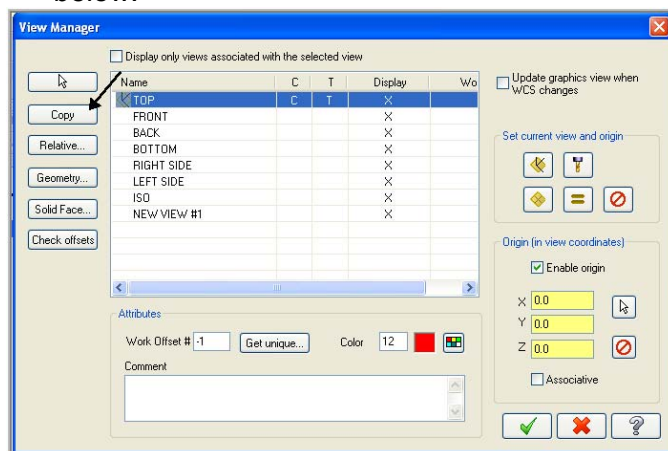
TASK 4: USE VIEW MANAGER TO SET UP A NEW WCS

➤ The two views you are going to create are in the same plane as Top, so you will copy Top and then change the origin.

1. Select **WCS** from the Status bar.

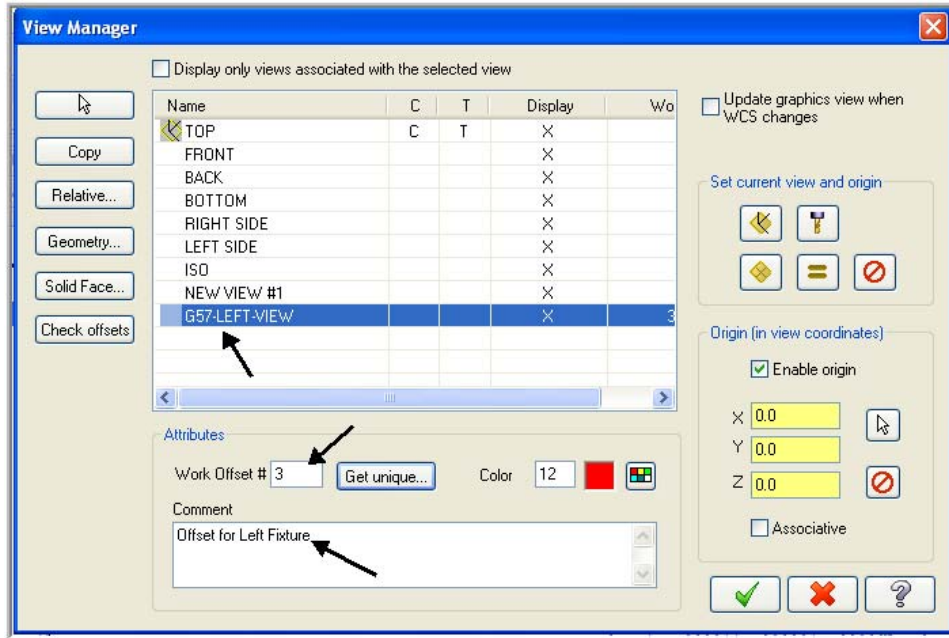


2. From the View Manager Dialog box select the **TOP** view and then select **Copy** as shown below.

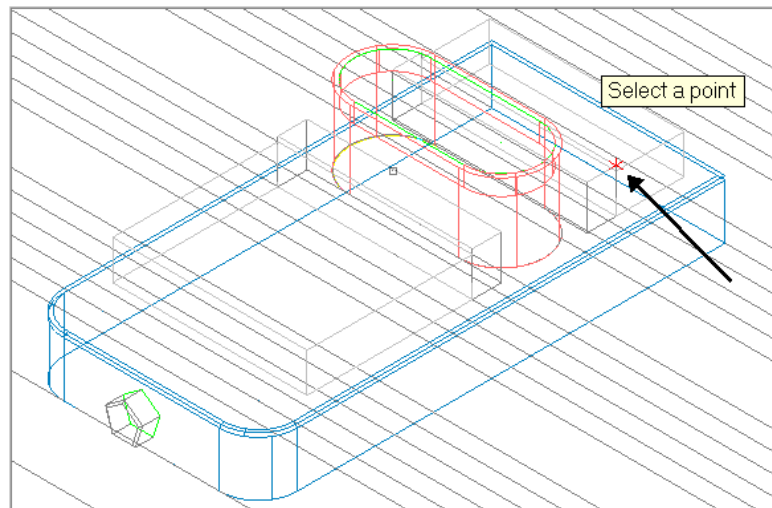
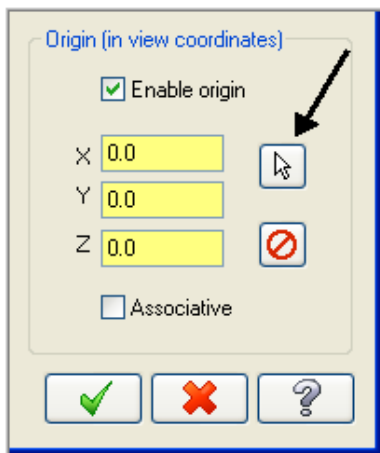


3. Right Click on **COPY OF TOP** and rename this view to **G57-LEFT-VIEW**.
4. Type **Offset for Left Fixture** in the Comment section.
5. In the Work Offset column type in **3** this tells Mastercam to output a **G57**.

☞ 0=G54, 1=G55, 2=G56

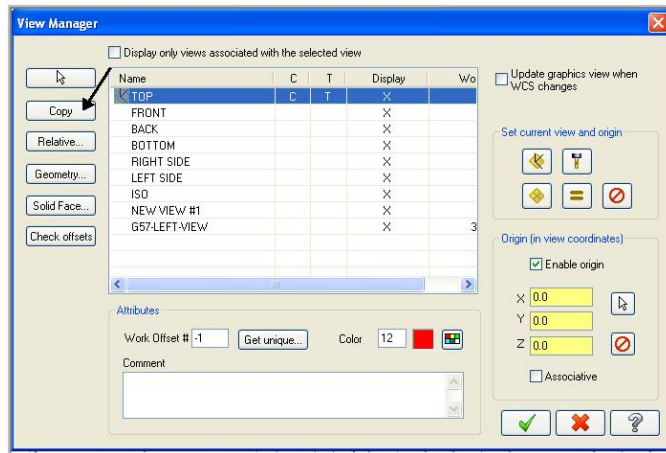


6. In the lower right corner of the View Manger dialog box click on the **Select Origin** button as shown below left:
7. Set the origin for the left setup by clicking on the red point on the **left vice** as shown below right:



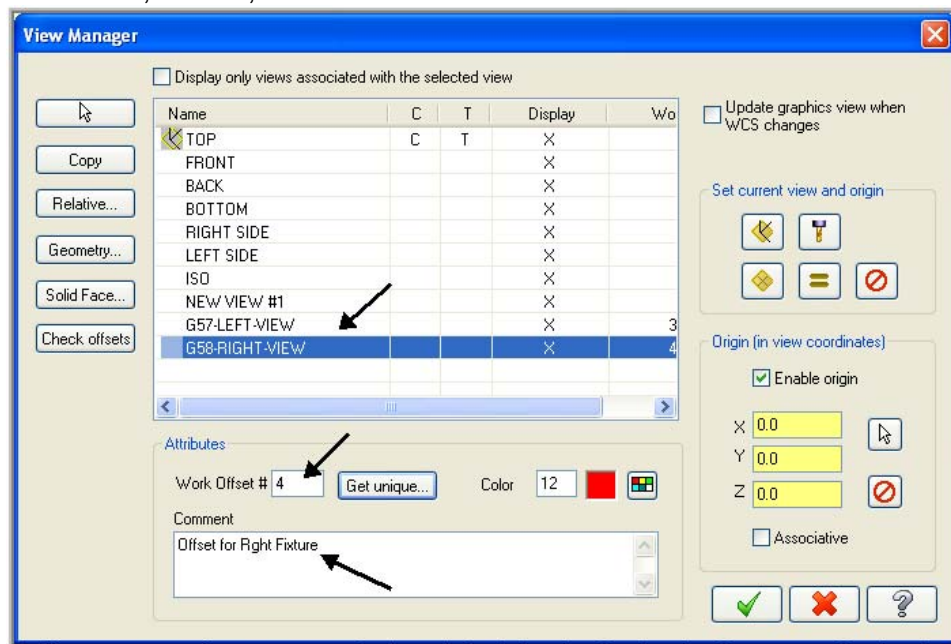
Mastercam Training Guide

8. From the View Manager Dialog box select the **TOP** view and then select **Copy** as shown below.

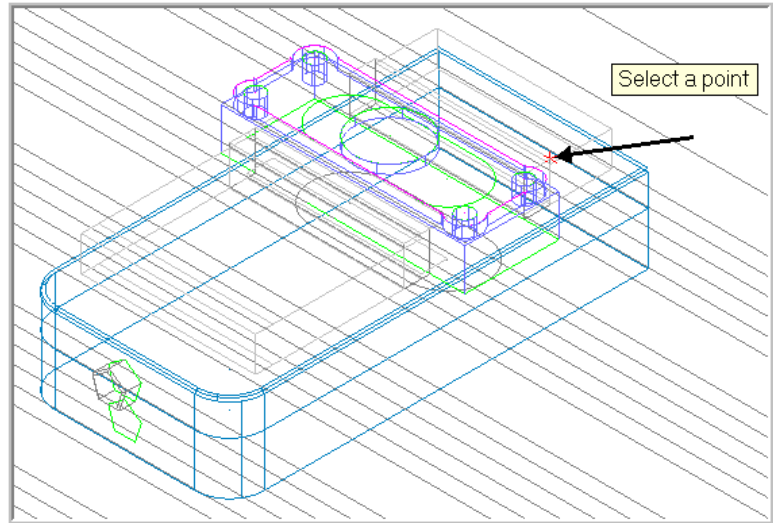
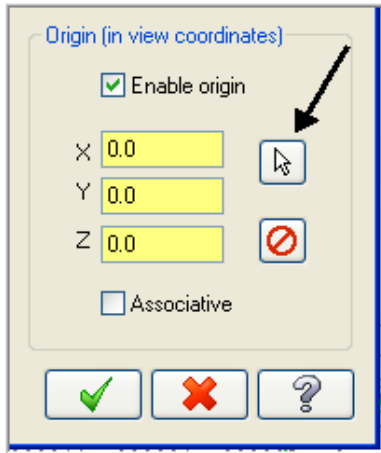


9. Click on **COPY OF TOP** and rename this view to **G58-RIGHT-VIEW**.
10. Type **Offset for Right Fixture** in the Comment section.
11. In the Work Offset column type in **4** this tells Mastercam to output a **G58**.

⇒ 0=G54, 1=G55, 2=G56




12. In the lower right corner of the View Manger dialog box click on the **Select Origin** button as shown below left:
13. Set the origin for the left setup by selecting the red point on the **right vice** as shown below right:



14. Now you are back in the View Manager dialog box select in the **T (Tplane)** column of the **G57-LEFT-VIEW** row as shown below.

Name	C	T	Display	Wo
TOP	C		X	
FRONT			X	
BACK			X	
BOTTOM			X	
RIGHT SIDE			X	
LEFT SIDE			X	
ISO			X	
NEW VIEW #1			X	
G57-LEFT-VIEW		T	X	3
G58-RIGHT-VIEW			X	4

15. Click on the OK icon  to complete this feature. Take note of the **Tplane** status at the bottom of the screen: **G57-LEFT-VIEW**.

TASK 5:

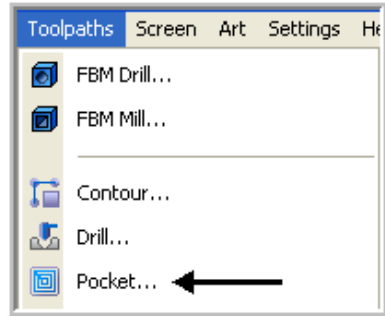
MACHINE THE POCKET LEFT HAND SETUP

- In this task you machine the pocket in the left hand vice with a **0.750 diameter 2 flute end mill**.
- The pocket depth is 0.5.

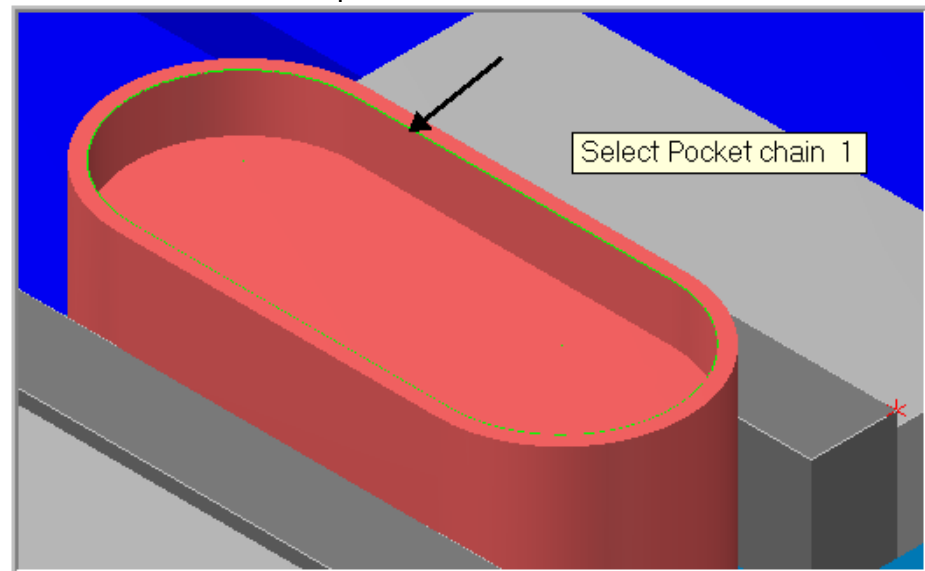
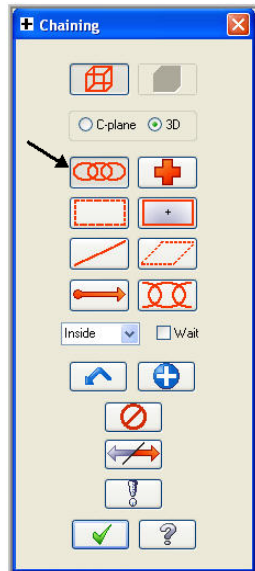
1. Activate a shaded view by selecting the icon at the top of the screen.



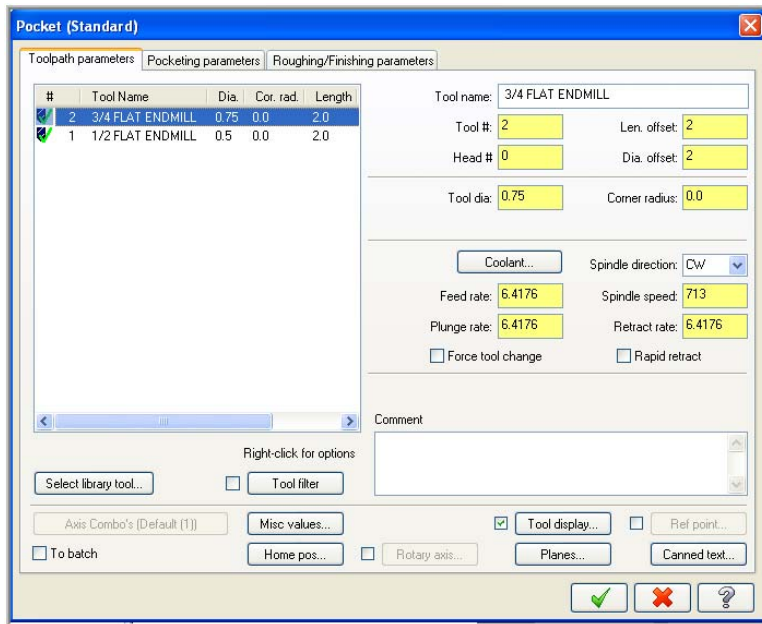
2. From the menu bar select **Toolpaths>Pocket...**



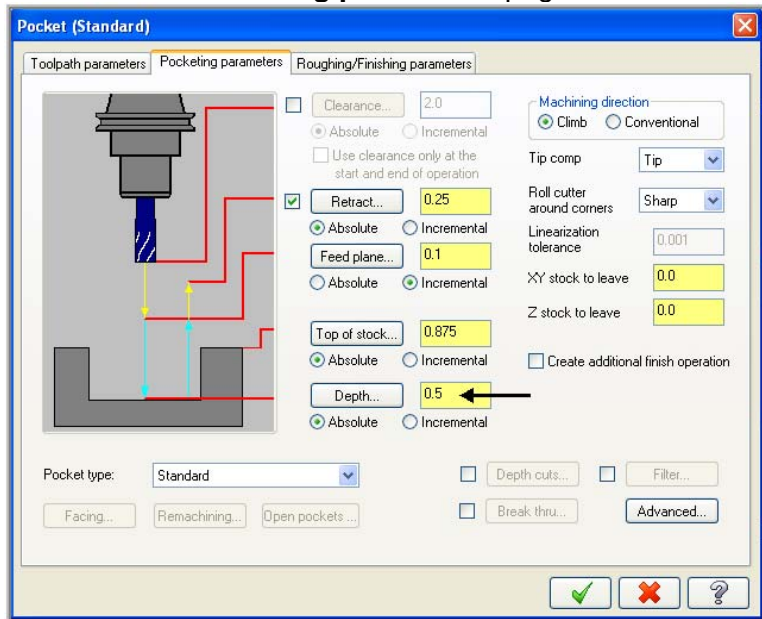
3. On the screen you will now see the **Chaining dialog box** with **Chain set** and in the graphics screen a prompt to **Select Pocket chain 1**.
4. Zoom in on the **left hand Vice**.
5. Select the line as shown below to chain the pocket.



6. Select **0.750** diameter end mill.



7. Select the **Pocketing parameters** page and make changes to this page as shown below:

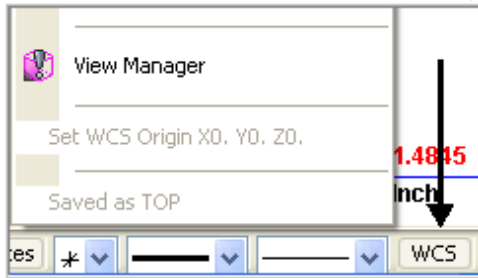


8. Select the **OK** button  to exit Pocket parameters.

TASK 6: USE VIEW MANAGER TO SET UP RIGHT VICE


☞ In this task we will setup for the right side vice using the view created earlier.

1. Select **WCS** from the Status bar, then select **View Manager**.



2. Now you are back in the **View Manager** dialog box. Select in the **T (Tplane)** column of the **G58-RIGHT-VIEW** row as shown below.

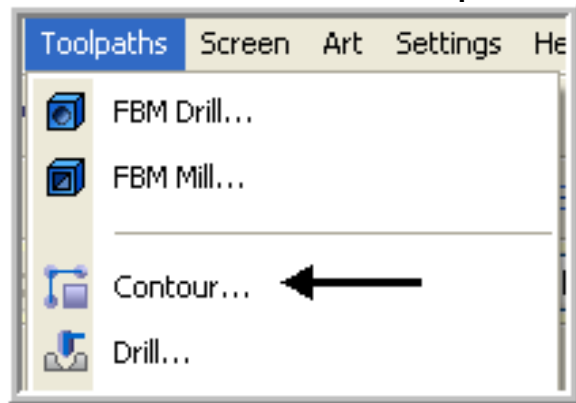
Name	C	T	Display	Wo
TOP	C		X	
FRONT			X	
BACK			X	
BOTTOM			X	
RIGHT SIDE			X	
LEFT SIDE			X	
ISO			X	
NEW VIEW #1			X	
G57-LEFT-VIEW			X	3
G58-RIGHT-VIEW		T	X	4

3. Click on the **OK** icon  to complete this feature. Take note of the **Tplane** status at the bottom of the screen **Tplane: G58-RIGHT-VIEW**.

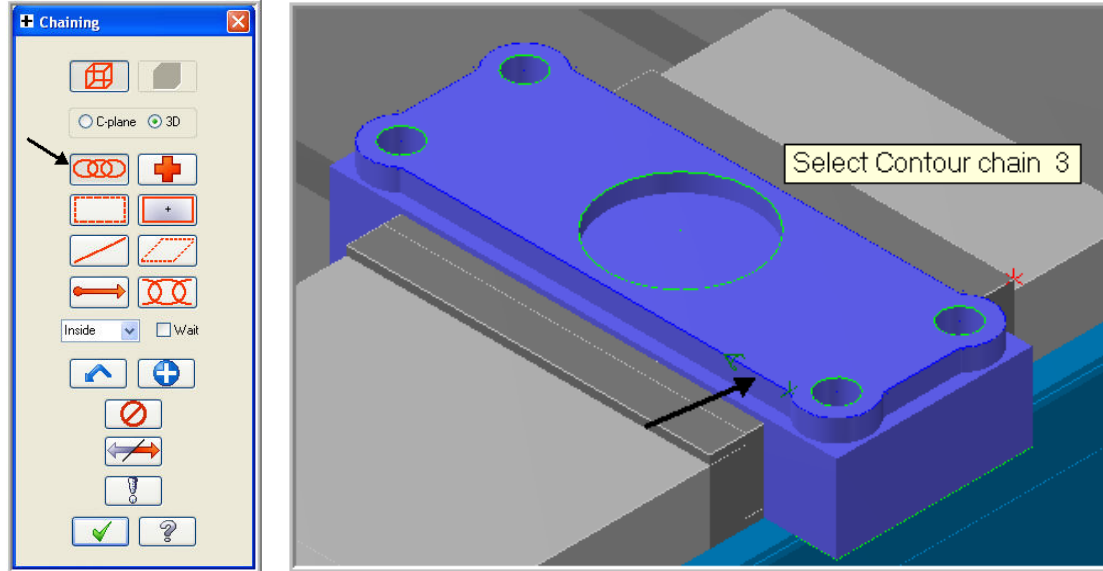
TASK 7: MACHINE THE CONTOR RIGHT HAND SETUP

- In this task you machine the contour on the right hand vice with a **0.500 diameter 2 flute end mill**.
- The pocket depth is 0.1.

1. From the menu bar select **Toolpaths>Contour...**




2. On the screen you will now see the **Chaining dialog box** with **Chain set** and in the graphics screen a prompt to **Select Contour chain 1**.
3. Zoom in on the **right hand Vice**.
4. Select the line as shown below:

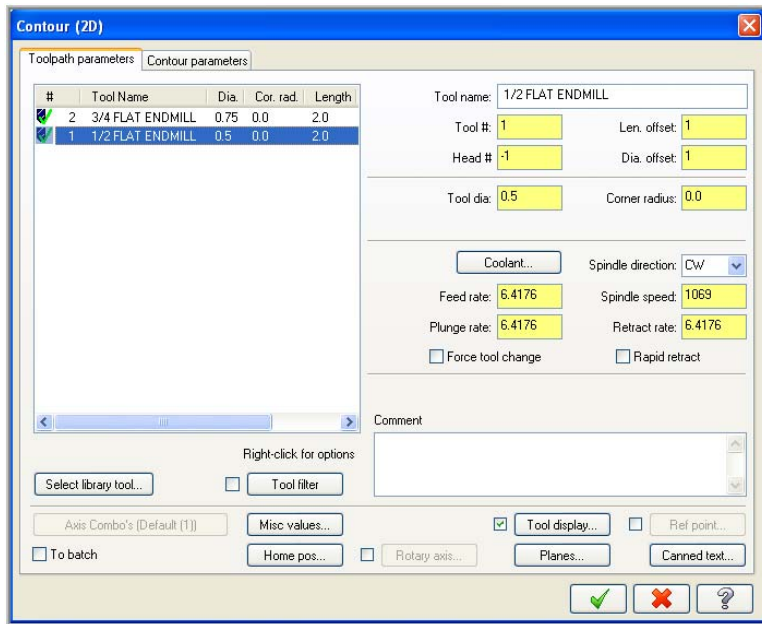


9. After selecting the line your graphics screen should look like the screenshot above, with the **green arrow pointing to the left, if not see below**.
10. If the arrow is not pointing to the left select the **arrow** from the Chaining dialog box shown below to reverse the direction.

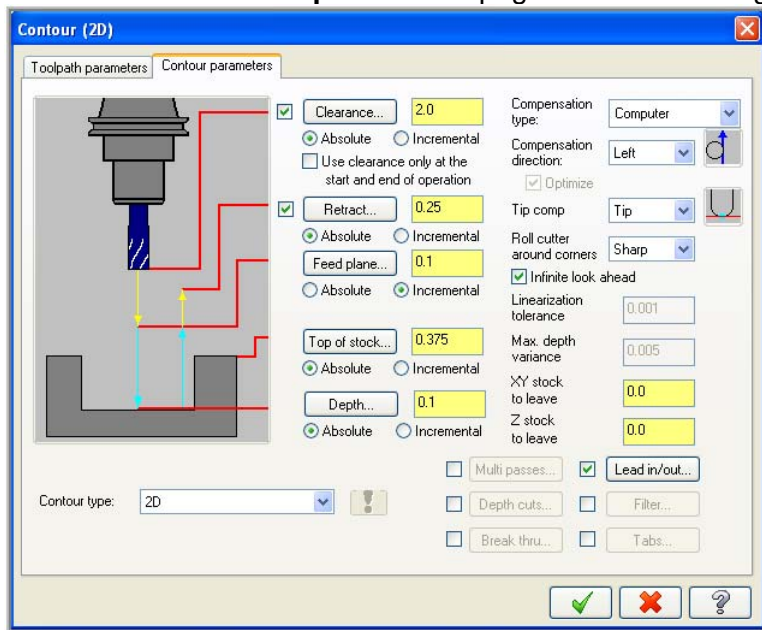


11. After the pocket has been successfully chained select the **OK** button  at the bottom of the Chaining dialog box.

5. Ensure the **0.5 diameter flat end mill** is selected.




6. Select the **Contour parameters** page and make changes to this page as shown below.




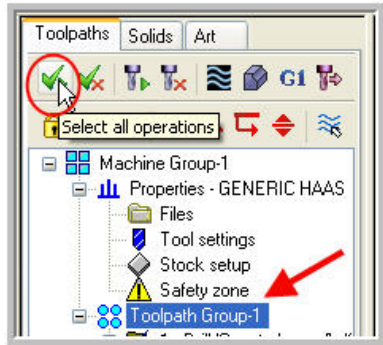
7. Select the **OK** button  to exit Contour parameters.

TASK 8: BACKPLOT THE TOOLPATHS

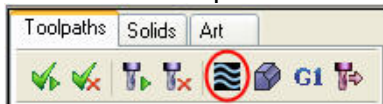
☞ In this task you will use Mastercam's Backplot function to view the path the tools take to cut this part.

1. Fit to Screen .

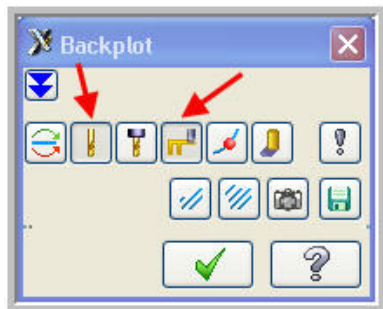
2. To pick all the operations to backplot pick the **Select All** icon  circled below:



3. The next step is to select the Backplot selected operations icon shown below:




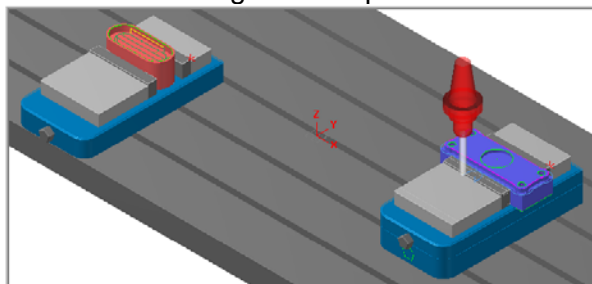
4. Before you Backplot the toolpath ensure the two buttons shown below are activated. The option on the left will **Display Tool** and the option on the right will **Display rapid moves**. These buttons act like a toggle switch, pressed in activates the function.




5. Set the run speed on the Backplot VCR midway along the speed bar as shown by the arrow below and then select the play button.

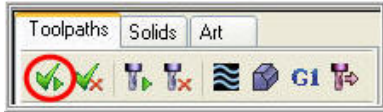


6. After reviewing the backplot select the OK button  to exit Backplot.

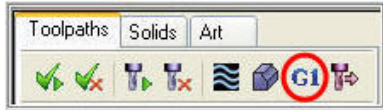


TASK 9: POST AND CREATE THE CNC CODE FILE

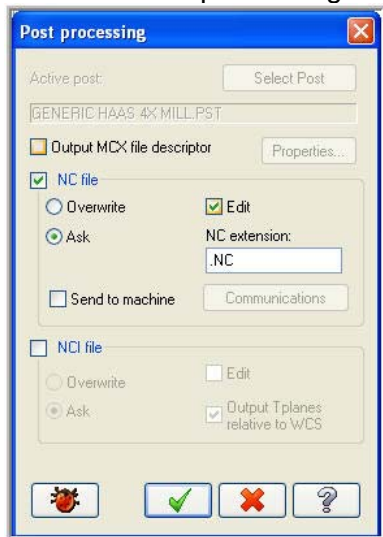
1. Ensure all the operations are selected by picking the **Select All** icon  from the Toolpath manager.



2. Select the **Post selected operations** button from the Toolpath manager.
Please Note: If you cannot see **G1** click on the right pane of the Toolpath manager window and expand the window to the right.

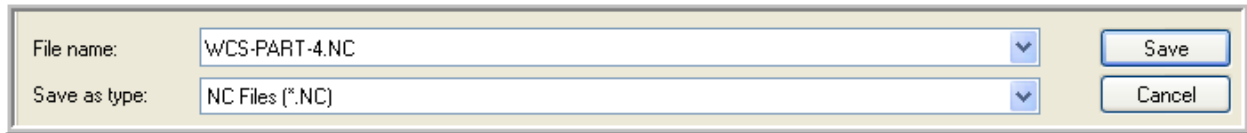


3. In the Post processing window, make the necessary changes as shown below:

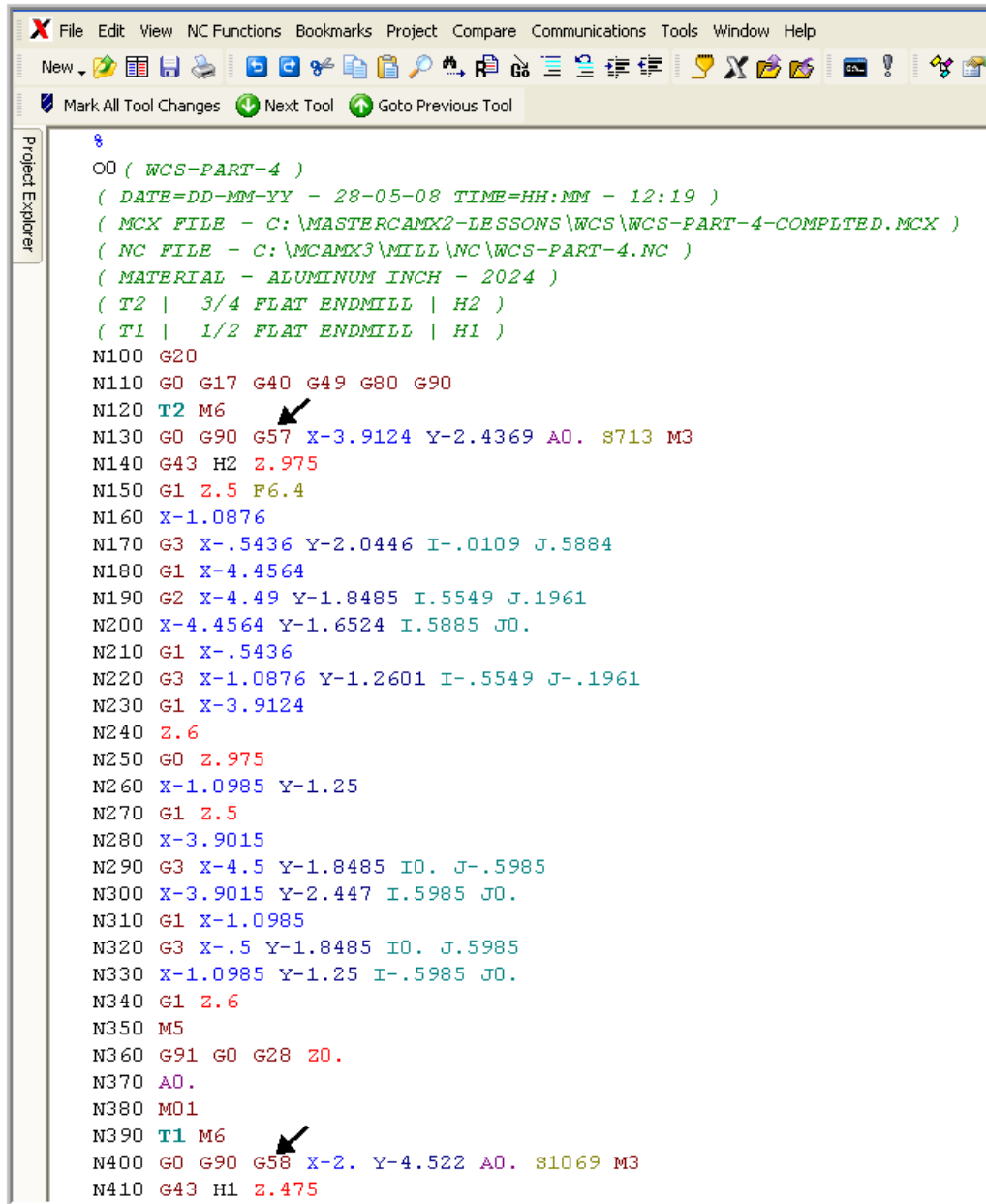


4. Select the OK button  to continue.

5. Enter the same name as your Mastercam part file name in the NC File name field **WCS-Part-4**.



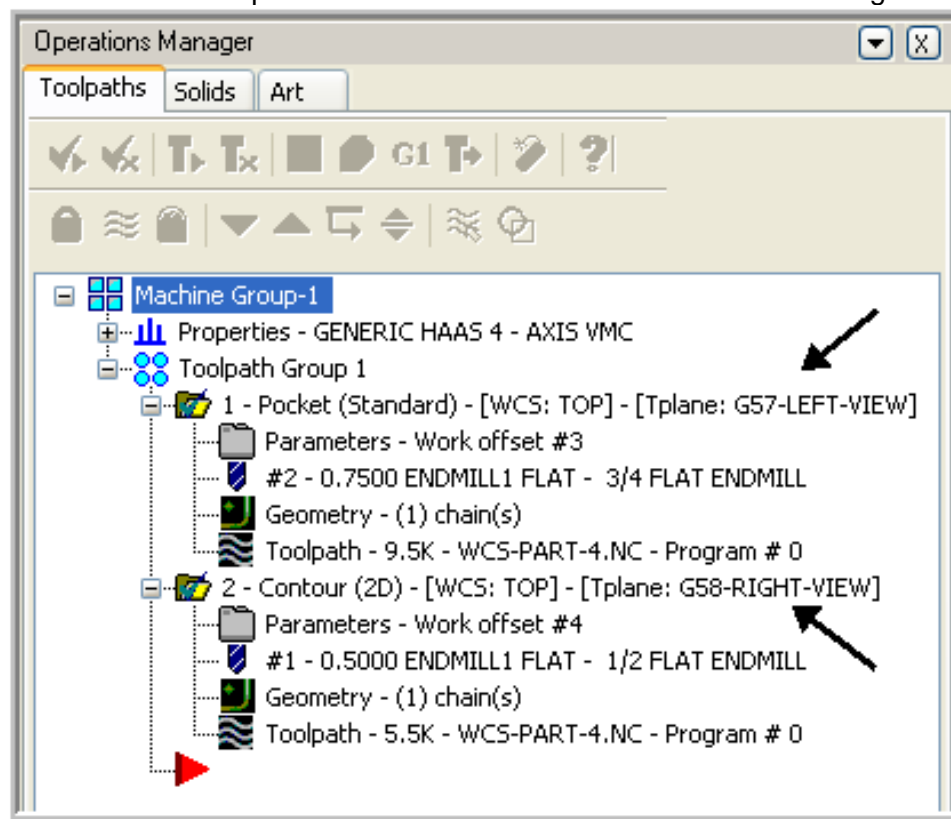
2. Select the Save button.
3. The CNC code file opens up in the default editor. Notice the G57 and G58 in the NC file below:



4. Select the  in the top right corner to exit the CNC editor.

Mastercam Training Guide

5. Enlarge the Toolpath manager window to the right. To do this left mouse button click on the right hand pane hold and extend to the right.
6. Note how the operations are identified with the two offsets being used.



This completes WCS-Part-4.