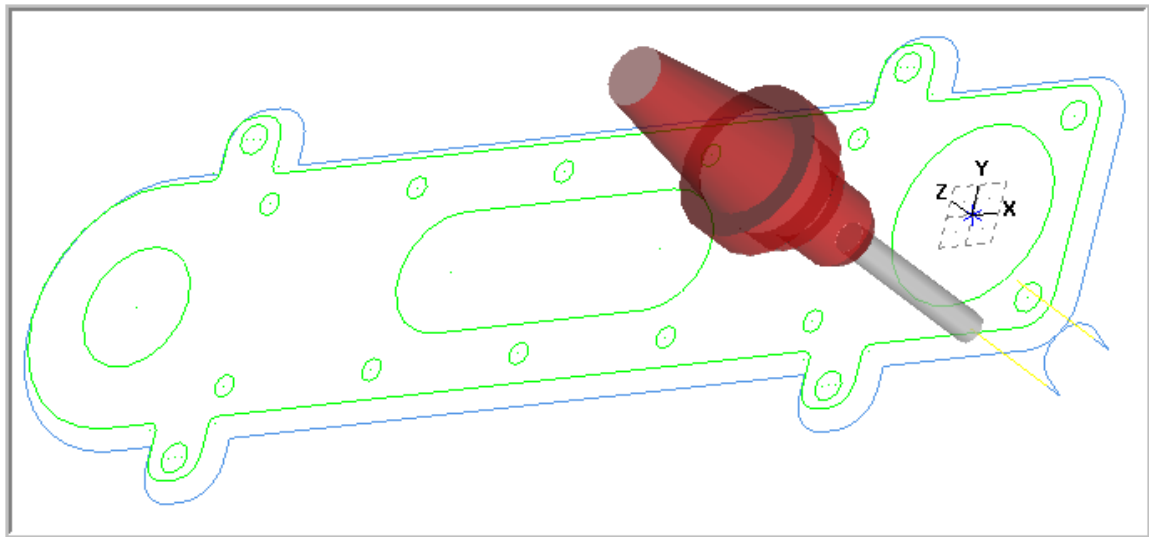# Mastercam X³

TRAINING
GUIDE

WCS - MOVING THE WCS - PART-3

camInstructor

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

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 contour toolpaths for Mill-Lesson-WCS-Part-3 This Lesson will cover the following topics:

⮩ **Create a 3-dimensional drawing by:**
USE VIEW MANAGER TO SET UP A NEW WCS Creating Lines
Creating Circles
Trimming Geometry
Using the View Manager
Using Views, Tool Planes and Construction Planes
Create Face and Pocket toolpaths

WCS-PART-3
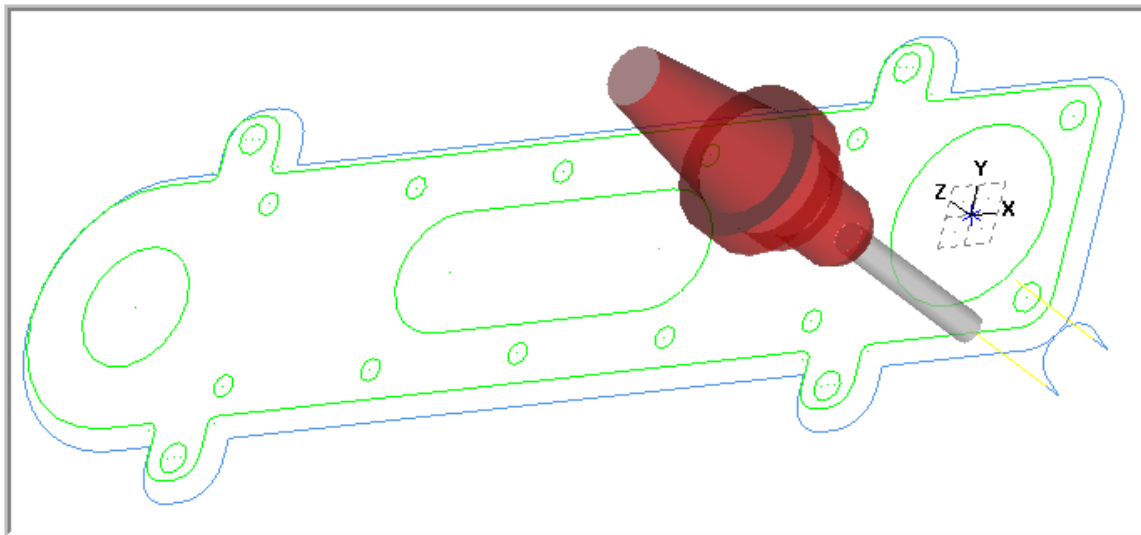
WCS-PART-3 - THE PROCESS

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 contour
TASK 6: Backplot toolpath
**TASK 7:** 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 3 - Moving the WCS - 6 Minutes.**

➲ In this lesson you will machine the contour of a part on a vertical spindle machining centre. Only problem is you need to machine the part lying flat on the machine table. But the drawing in the Mastercam file is oriented in a non-standard plane. No problem – what you are going to do is change the WCS.

➲ Changing the WCS is useful for parts that have been drawn as part of a CAD drawing assembly but cannot be moved by coordinates.

➲ It allows the user to define a new part work origin and system view to create toolpaths on it.
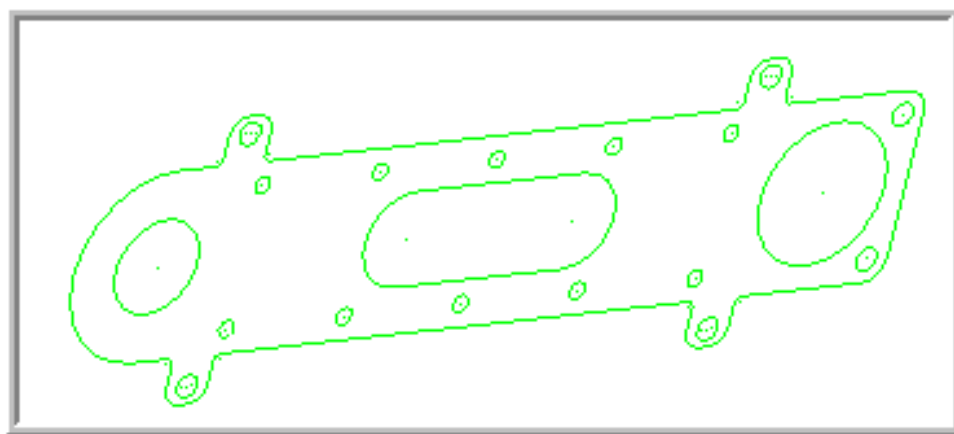
TASK 3:
OPEN EXISTING FILE FROM THE MULTIMEDIA CD

➲ On the multimedia CD that came with this text is **a folder called Mastercam-Files**. Open the file **WCS-PART-3.MCX**
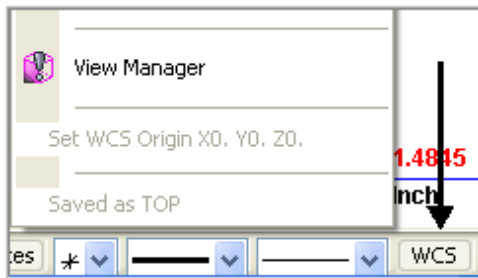
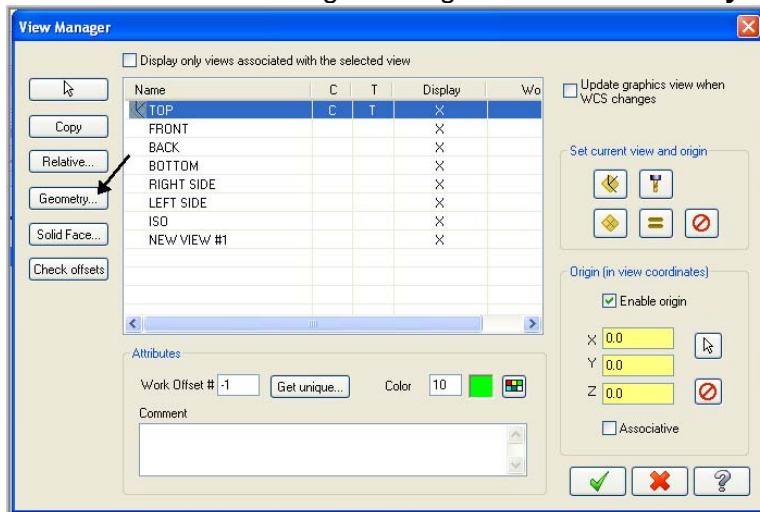1. Select **File>Open>WCS-PART-3.MCX.** The file should appear as below.
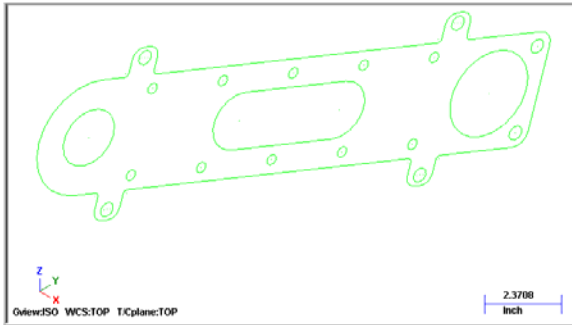
TASK 4:
USE VIEW MANAGER TO SET UP A NEW WCS
➲ In this task you will create a new view aligned with the part geometry.

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

2. From the View Manager Dialog box select **Geometry** as shown below.

3. You will be returned to the graphics screen. Select the **circle** as shown below left and make sure the **XYZ tripod** is the same as the example below right. The Z axis should be pointing up from the top surface of the part. If required use the arrow buttons to cycle through to the correct coordinate system.
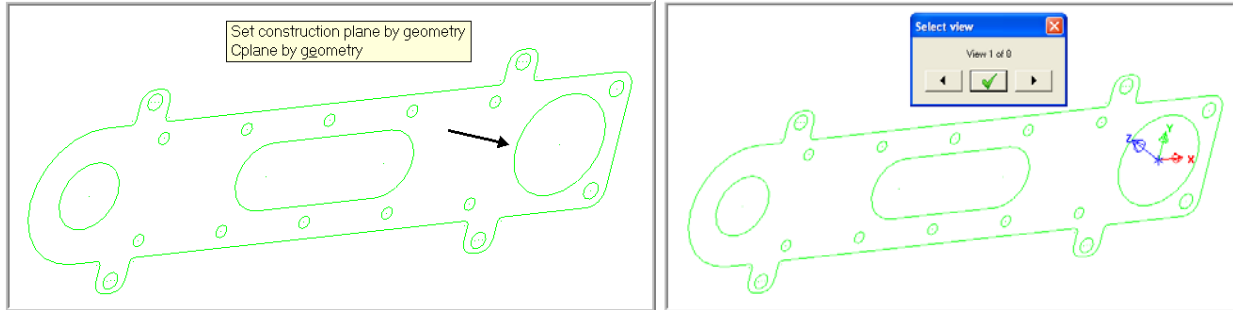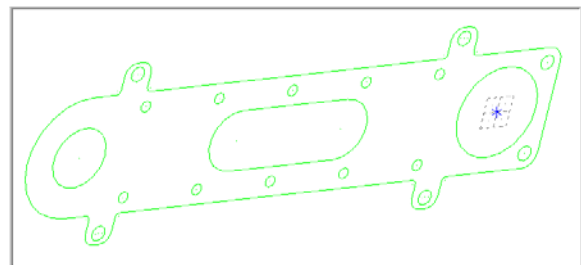

4. Select the green check mark for **OK**.
5. In the **New View** dialog box enter **PART-WCS** in the name section. Activate to **Set as** WCS.
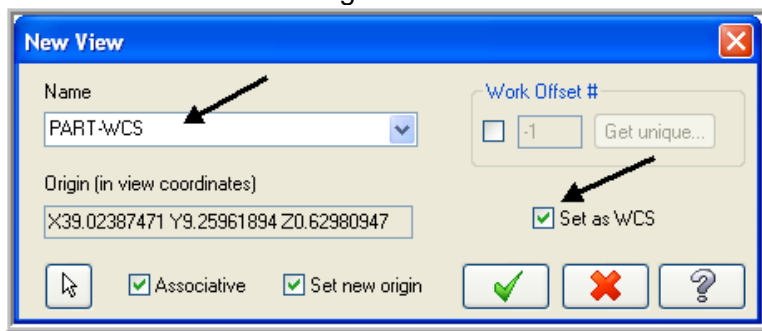
6. Click on the **OK** icon to complete this feature.
7. Now you are back in the View Manager dialog box select in the **C (Cplane) column** and **T (Tplane) column** of the **PART-WCS row** as shown below.


8. Click on the **OK** icon to complete this feature. Note the location and orientation of the grid now as shown above right.
9. Select on your keyboard **ALT and F9**. This Displays on the screen the orientation of the **Cplane**, **Tplane** and the **WCS**.
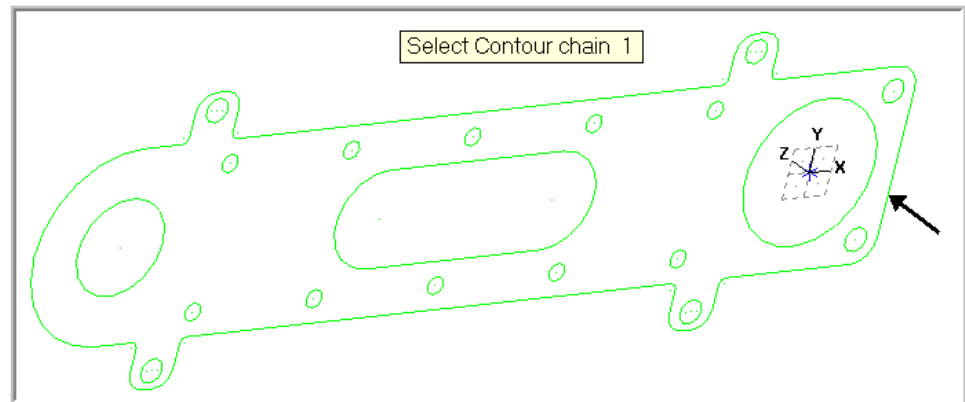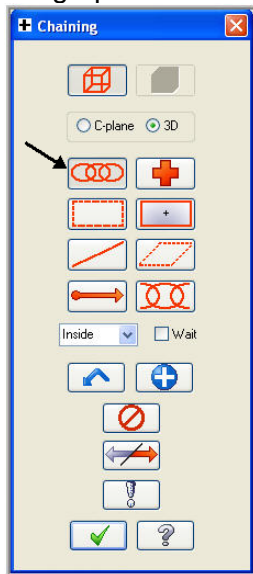
TASK 5:
MACHINE THE CONTOUR
➲ In this task you machine the contour with a **0.5 diameter 2 flute end mill**.

1. From the menu bar select **Toolpaths>Contour…**
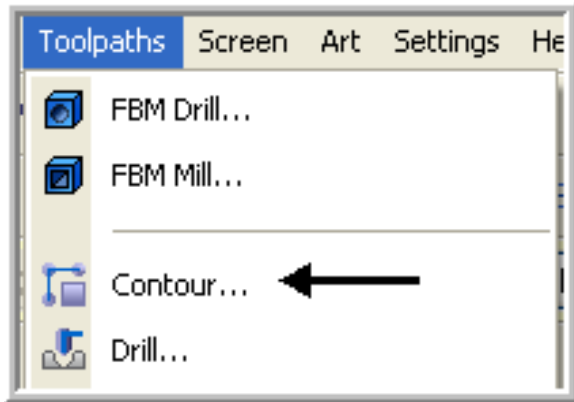
2. On the screen you will now see the **Chaining dialog box** with **Chain set** and in the graphics screen a prompt to **Select Contour chain 1**. Select the line as shown below:

3. After selecting the line your graphics screen should look like the screenshot below, with the **green arrow pointing downwards** in a clockwise direction.
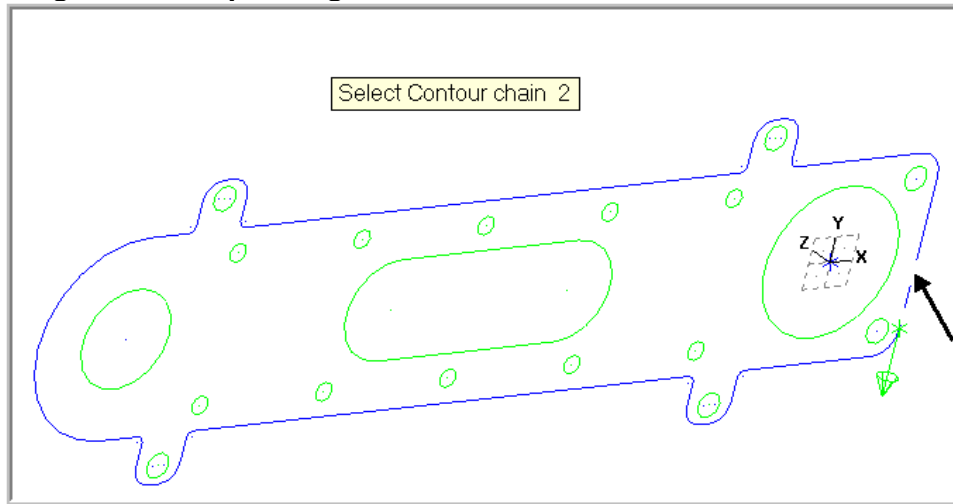

4. If the arrow is not pointing **downwards** select the arrow from **the Chaining dialog box** shown below to reverse the direction.

5. After the contour has been successfully chained select the **OK** button at the bottom of the Chaining dialog box.
6. Ensure the **0.5 diameter flat end mill** is selected.
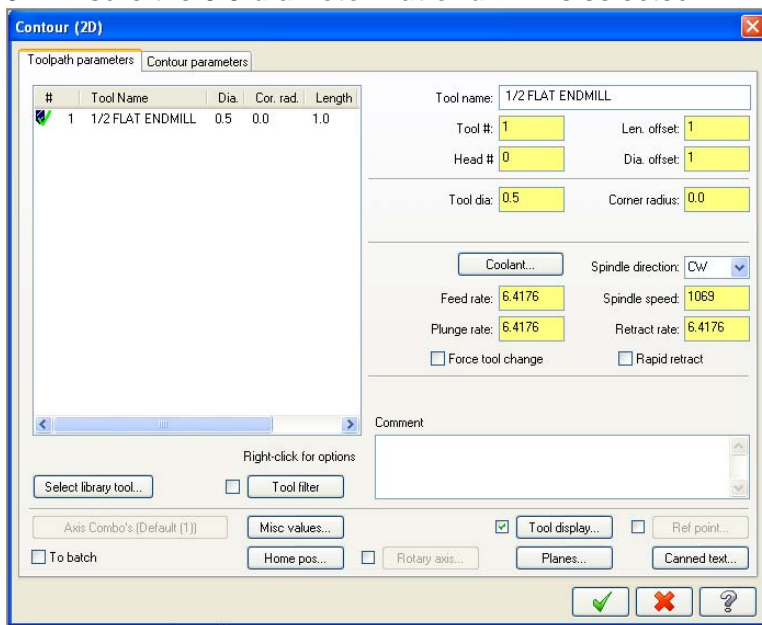
7. Select the **Contour parameters** page and make changes to this page as shown below.
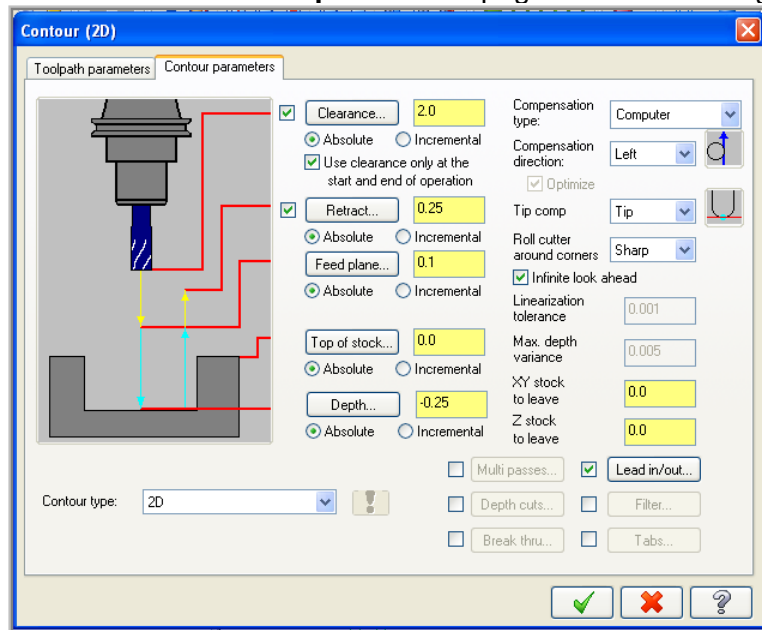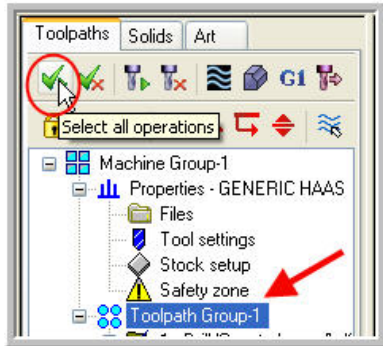

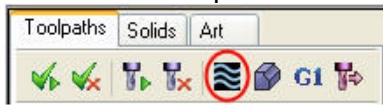8. Select the **OK** button to exit Contour parameters.

TASK 6:
BACKPLOT THE TOOLPATH

➲ In this task you will use Mastercam's Backplot function to view the path the tool takes to cut this part.

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

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

3. 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.
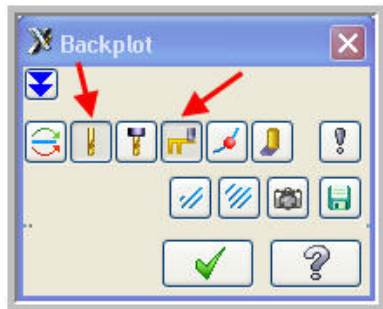
4. Set the run speed on the Backplot VCR midway along the sped bar as shown by the arrow below and then select the play button.
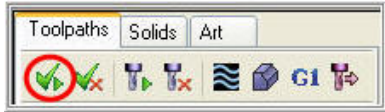

5. After reviewing the backplot select the OK button to exit Backplot.
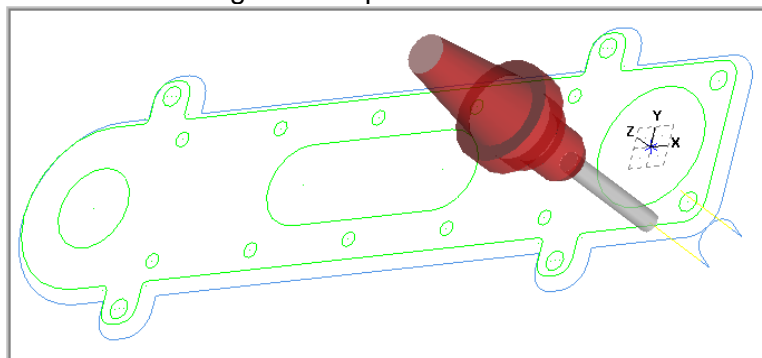
TASK 12:
POST AND CREATE THE CNC CODE FILE

1. Ensure all the operations are selected by picking the **Select All** icon [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 manger window and expand the window to the right.
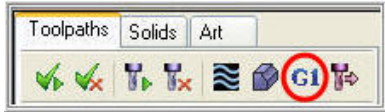
3. In the Post processing window, make the necessary changes as shown below:

4. Select the OK button [icon] to continue.

5. Enter the same name as your Mastercam part file name in the NC File name field **WCS-Part-3**.

File name:	WCS-PART-3.NC	⌄	Save
Save as type:	NC Files (*.NC)	⌄	Cancel

6. Select the **Save** button.
7. The CNC code file opens up in the default editor.

X File Edit View NC Functions Bookmarks Project Compare Communications Tools Window Help

New ⌄

Mark All Tool Changes   Next Tool   Goto Previous Tool

```
%
O0 ( WCS-PART-3 )
( DATE=DD-MM-YY - 28-05-08 TIME=HH:MM - 12:09 )
( MCX FILE - C:\MASTERCAMX2-LESSONS\WCS\WCS-PART-3.MCX )
( NC FILE - C:\MCAMX3\MILL\NC\WCS-PART-3.NC )
( MATERIAL - NONE )
( T1 |  1/2 FLAT ENDMILL | H1 )
N100 G20
N110 G0 G17 G40 G49 G80 G90
N120 T1 M6
N130 G0 G90 G54 X3.091 Y-2.3839 A0. S1069 M3
N140 G43 H1 Z2.
N150 Z.1
N160 G1 Z-.25 F6.4
N170 X2.7374 Y-2.0303
N180 G3 X2.3839 Y-1.8839 I-.3535 J-.3536
N190 X2.0303 Y-2.0303 I0. J-.5
```

8. Select the ☒ in the top right corner to exit the CNC editor.
This completes Mill-Lesson-WCS-Part-3.