

Mastercam® 2018

IMPERIAL – TRAINING TUTORIAL SERIES



CAD IMPORT & MILL ESSENTIALS

Mastercam® 2018

CAD Import & Mill Essentials Toolpaths Training Tutorial

To order more books:

Call 1-800-529-5517 or

Visit www.emastercam.com or

Contact your Mastercam dealer

Mastercam 2018 CAD Import & Mill Essentials Toolpaths Training Tutorial

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

Software: Mastercam 2018

Author: Mariana Lendel

ISBN: 978-1-77146-659-2

Date: June 8, 2017

Notice

In-House Solutions Inc. reserves the right to make improvements to this manual at any time and without notice.

Disclaimer Of All Warranties And Liability

In-House Solutions Inc. makes no warranties, either express or implied, with respect to this manual or with respect to the software described in this manual, its quality, performance, merchantability, or fitness for any particular purpose. In-House Solutions Inc. manual is sold or licensed "as is." The entire risk as to its quality and performance is with the buyer. Should the manual prove defective following its purchase, the buyer (and not In-House Solutions Inc., its distributor, or its retailer) assumes the entire cost of all necessary servicing, repair, of correction and any incidental or consequential damages. In no event will In-House Solutions Inc. be liable for direct, indirect, or consequential damages resulting from any defect in the manual, even if In-House Solutions Inc. has been advised of the possibility of such damages. Some jurisdictions do not allow the exclusion or limitation of implied warranties or liability for incidental or consequential damages, so the above limitation or exclusion may not apply to you.

Copyrights

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

Trademarks

Mastercam is a registered trademark of CNC Software, Inc.

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

MASTERCAM SHORTCUTS

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

CUSTOMIZE MASTERCAM

Create Your Own Keyboard Shortcuts

- ♦ Right mouse click in the **Ribbon**.
- ♦ **Customize the Ribbon**.
- ♦ **Keyboard shortcuts**, select **Customize** button.

- ♦ Select the **Category**.
- ♦ Select a command and under **Press new shortcut key** enter the key combinations you want to assign to it.

Customize Quick Access Toolbar

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

Customize the Ribbon

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

Customize the right-click menu

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

WAYS TO GET THE MOST FROM MASTERCAM

Mastercam Training

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

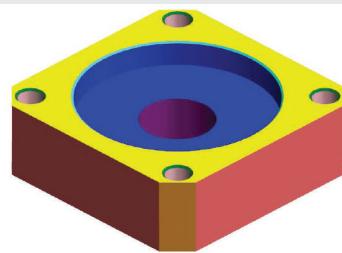
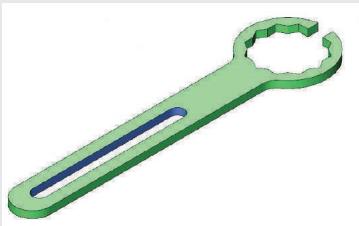
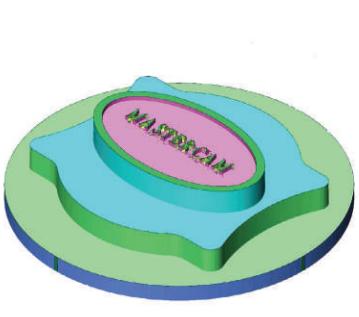
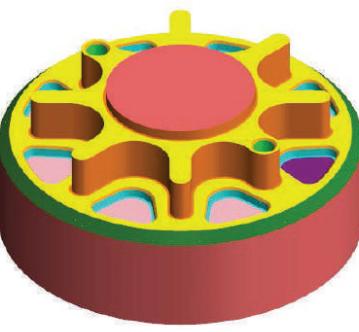
Our library of **Mastercam Training Solutions** consists of several product lines that cater to any learning style. Learn Mastercam at your own pace with our **Training Tutorials**, teach your students with the help of our **Instructor Kits**, learn the theory behind Mastercam with our **Handbooks**,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

Mastercam Community

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

For downloaded pdf please visit

www.emastercam.com/qrc

Tutorial	CAD Import and Toolpath Creation
#1	 <p>Import an AutoCAD DWG file. Facing Toolpath. Circle Mill Toolpath. Spot Drill Toolpath. Drill Toolpath. 2D Contour Chamfer Toolpath. 2D Contour Toolpath.</p>
#2	 <p>Import an AutoCAD DWG file. Setup 1 Slot Mill Toolpath. 2D HS Dynamic Mill Toolpath. Contour Toolpath. 2D HS Dynamic Contour Toolpath. Setup 2 Facing Toolpaths.</p>
#3	 <p>Import an AutoCAD DWG file. 2D HS Area Mill Toolpath. 2D HS Dynamic Mill Toolpath. 2D HS Area Mill with Island Toolpath. Pocket Remachine Toolpath.</p>
#4	 <p>Import a SolidWorks file. Setup 1 2D HS Dynamic Mill Toolpath. Contour Toolpath 2D HS Area Mill Toolpath. Transform Toolpath. Contour Toolpath Drill Toolpath. Contour Chamfer Toolpath. Setup 2 2D HS Dynamic Area Mill Toolpath. Contour Toolpath.</p>

CAD IMPORT & MILL ESSENTIALS TOOLPATHS PROJECTS

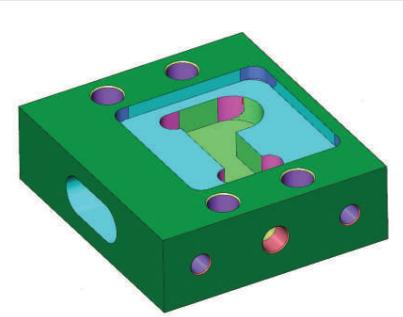
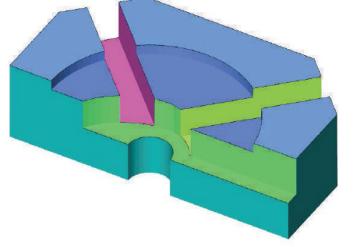
Tutorial	Toolpath Creation
#5 	<p>Import a SolidWorks file. Setup 1- Top Tool Planes. 2D HS Area Mill Toolpath. 2D HS Area Mill Rest Material Toolpath. Spot Drill, Drill & Tap Toolpaths.</p> <p>Setup 2 - Front Tool Plane. Spot Drill, Drill & Tap Toolpaths.</p> <p>Setup 3 - Left Tool Plane. Slot Mill Toolpath.</p>
#6 	<p>Import a SolidWorks file. 2D HS Dynamic Mill Toolpath. 2D HS Area Mill Toolpath. 2D HS Blend Mill Toolpath.</p>
#7 	<p>Import a SolidWorks file. 2D HS DynamicToolpath. Feature Based Drilling Toolpath. 2D HS DynamicToolpath. Contour Toolpath.</p>

Table of Contents

Getting Started	1
Tutorials:	
Tutorial #1 - Import a DWG file, Facing, Circle Mill, Drill & Contour-Chamfer, Contour Toolpaths	15
Tutorial #2 - Import a DWG file, 2 Setups, Slot Mill, 2D HS Dynamic Mill & Contour Toolpaths	111
Tutorial #3 - Import a DWG file, 2D HS Area & Dynamic Mill, 2D HS Area with Island & Pocket Remachining.....	207
Tutorial #4 - Import a SolidWorks file, 2D HS Area Mill & Dynamic Mill, Transform Toolpaths	303
Tutorial #5 - Import a SolidWorks file, 3 Setups, 2D HS Area Mill - Rest Material Toolpath.....	423
Tutorial #6 - Import a SolidWorks file, 2D HS Dynamic, Blend & Area Mill Toolpaths	561
Tutorial #7 - Import a SolidWorks file & Feature Based Drilling Toolpath	627
Creating/Editing Tools	695
Quiz Answers	707

TABLE OF CONTENTS

GETTING STARTED

Objectives:

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

STEP 1: STARTING MASTERCAM

For Windows 7

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

For Windows 8

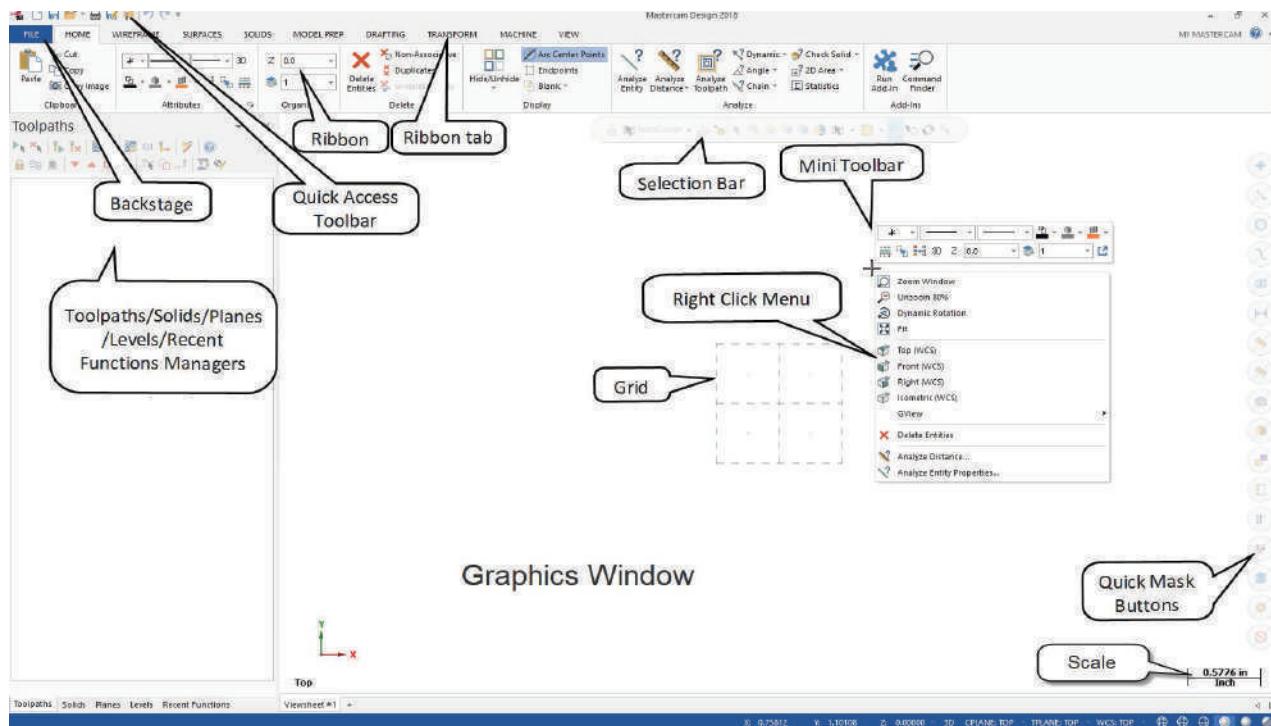
- ◆ Select the **Start** button.
- ◆ Click on the drop down arrow to open **Apps**.
- ◆ Find and click on **Mastercam 2018**.

For Windows 10

- ◆ Select the **Start** button.
 - ◆ Click on the drop down arrow to open **Apps**.
 - ◆ Find and click on **Mastercam 2018**.
-
- ◆ To start the software, from **Desktop**, click on the shortcut icon as shown.



STEP 2: GUI - GRAPHICAL USER INTERFACE



Quick Access Toolbar	QAT contains a fully customizable set of functions that can be quickly accessed by the user.
Backstage (FILE)	Allows you to manage files. You can insert information about files, start a new file, open an existing one or merge files together. You can also save, convert or print files as well as access the help resources.
Tabs	Contain all the functionality within Mastercam.
Ribbon	Displays the commands available for a selected Tab.
Selection Bar	Allows you to set the AutoCursor modes and to switch between wireframe or solid selections.
Quick Mask Buttons	Let you select all entities of a specific type. Clicking on the left side of the button or right side of the button toggles between select all or only.
Right Click Menu	Right click menu allows quick access to functions such as zoom, graphic views or recent functions used. A mini toolbar will also appear that allows you to quickly change the attributes.
Toolpaths/Solids/Planes Manager	Lists the history of the toolpath operations and solids.
Graphics Window	Workspace area in Mastercam where the geometry is displayed.
Scale	Shows you a scale of the object on the screen.
WCS: TOP T/Cplane:	Displays the current WCS and T/Cplane information.

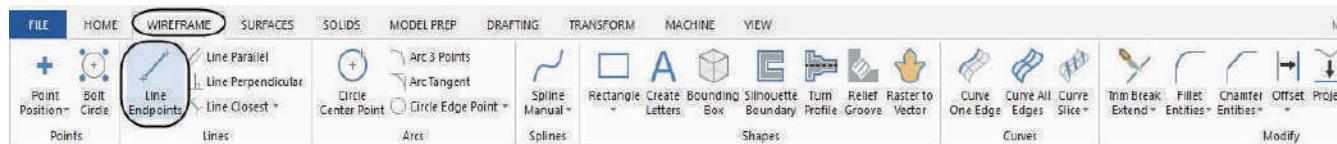
STEP 3: NAVIGATE THROUGH MASTERCAM

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

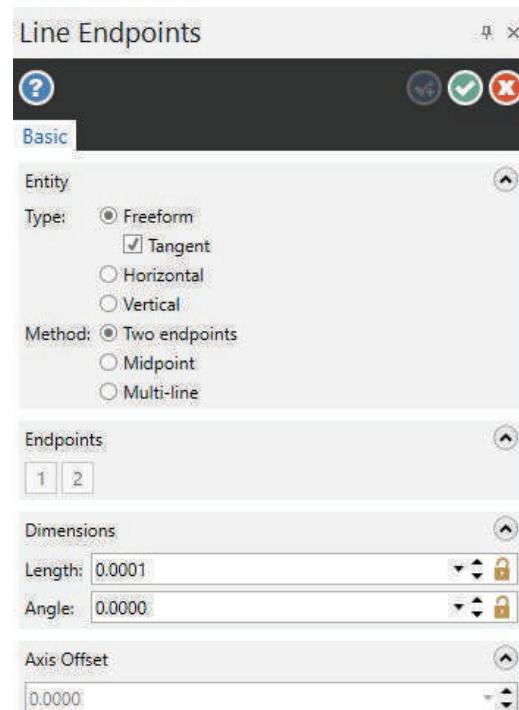
3.1 Using the Wireframe tab to select the command to create Line Endpoints

- ♦ Left click on **WIREFRAME**.
- ♦ Left click on the **Line Endpoints** icon as shown in [Figure: 3.1.1](#).

Figure: 3.1.1



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



Sketching a line

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

Creating a line knowing the endpoint coordinates

- ♦ To make a line knowing the two endpoint coordinates, select the **AutoCursor Fast Point** icon from the **General Selection** toolbar.



- ♦ In the coordinates field that opens in the upper left corner enter the coordinates of the first endpoint as shown.

0,1

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

Creating a line knowing an endpoint, the length and the angle

- ♦ You can also enter the coordinates of the first endpoint, then enter the **Length** and **Angle** if necessary.
- ♦ To continue making lines, choose the **OK** and **Create New Operation** button from the dialog box or press

Enter.



- ♦ To exit the current command, select the **OK** button or press the **Esc** button.
- ♦ To undo the last command, from the **QAT** (Quick Access Toolbar) select the **Undo** button. The **Undo** button can be used to go back to the beginning of geometry creation or to the last point of the saved file.

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

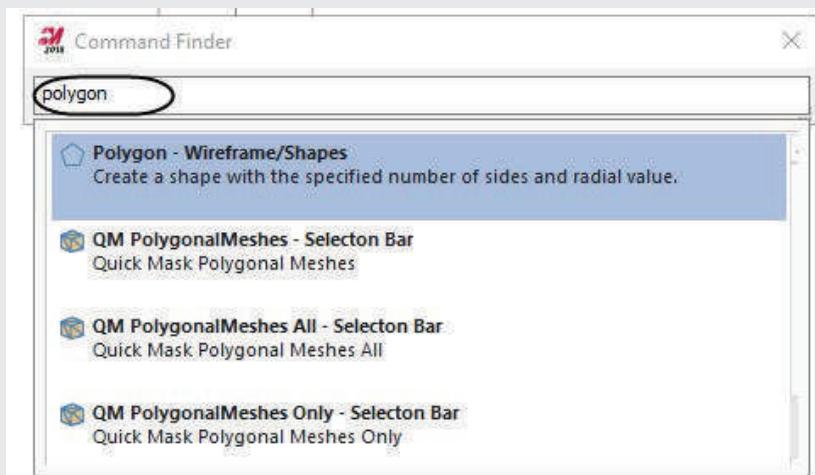
3.2 Function Prompt

Prompts the user to execute a command.

- ◆ Example: this prompt is used in the **Line Endpoints** command. Specify the first endpoint

NOTE: To find a command, from the **HOME** ribbon, select the **Command Finder** icon and type the function name in the field that opens up.

For example, to find the **Polygon** command type "polygon" in the text field.



From the list, select the desired command.

STEP 4: SET THE ATTRIBUTES

Mastercam attributes are point style, line style, line thickness, color and levels. Before starting to create geometry, you should set the attributes.

4.1 'Attributes' Group

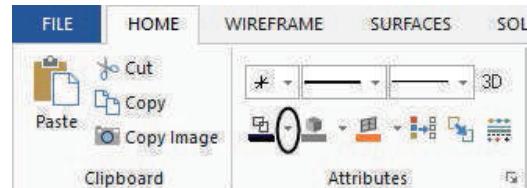
Point Style	Displays and sets the system's point style.
Line Style	Displays and sets the system's line style.
Line Width	Displays and sets the current system's line width.
Color	Assigns the current color to wireframe, solid and surface entities. To change the current color, click in the specific color field and select a color from the color pallet. To change an existing geometry color, select the geometry first and then click in the color field and select a color from the color pallet.
Clear Color	When performing a transform function (Xform), Mastercam creates a temporary group from the originals (red) and a result (purple) from the transformed entities. These system groups appear in the Groups dialog box. However, they stay in effect only until you use the Clear Colors function or perform another transform function.
2D / 3D Construction Mode	Toggles between 2D and 3D construction modes. In 2D mode, all geometry is created parallel to the current Cplane at the current system Z depth. In 3D mode, you can work freely in various Z depths, unconstrained by the current system Z depth and Cplane setting.

4.2 'Organize' Group

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

Change the Wireframe Color

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



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



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

To change the color of existing geometry, select the entities first and then click on the drop down arrow next to the **Wireframe Color** and select the desired color.

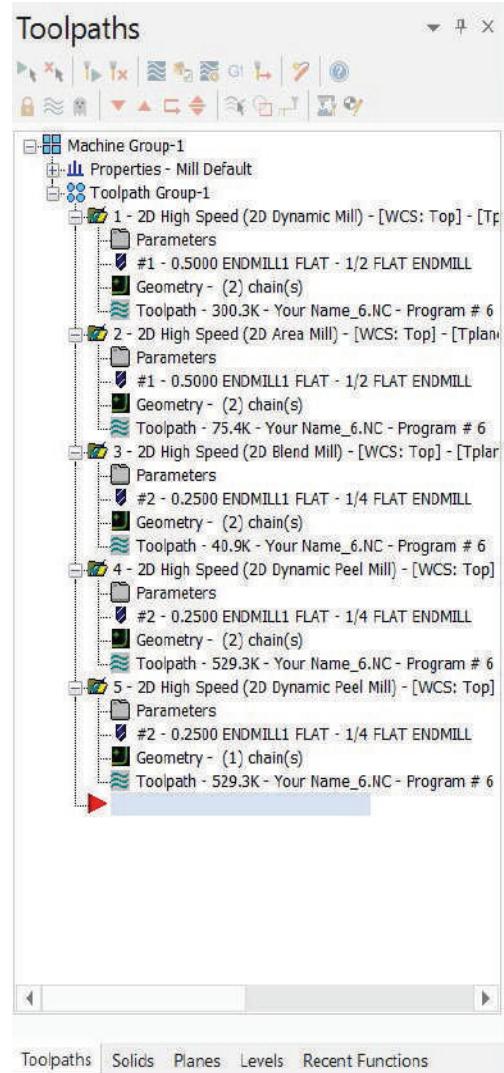
The same method can be applied for any other attribute that you want to set or change.

STEP 5: MANAGER PANELS

5.1 The Toolpaths Manager

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

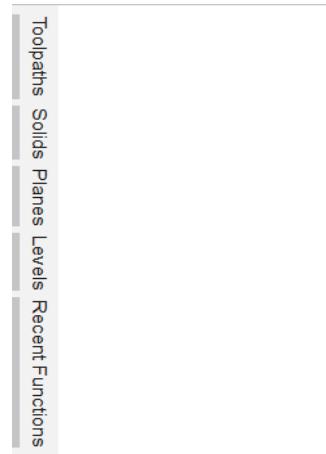
Figure: 5.1.1



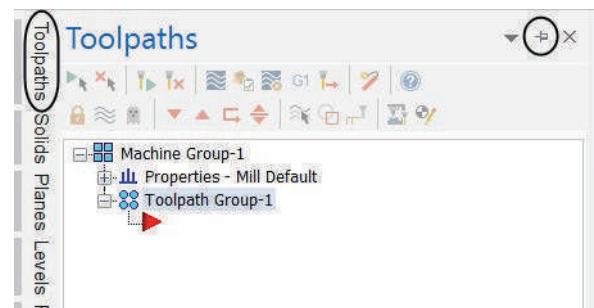
- The **Toolpaths Manager**, **Solids Manager**, or **Planes Manager** can be hidden to gain more space in the graphics area for creating geometry. Use **Auto Hide** icon to close all **Toolpaths**, **Solids**, **Planes** and **Levels Manager** panels.



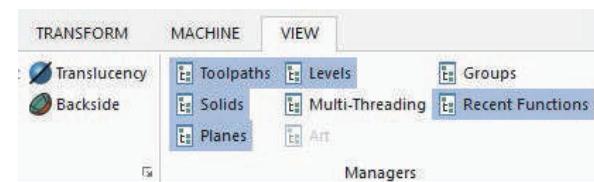
- The panels will be hidden to the left of the graphics window as shown.



- To un-hide them, click on one of the managers to open it and then click again on the **Auto Hide** icon as shown.



- Selecting the X (Close icon) instead of the Auto Hide, you will close the manager panel. To re-open them, from the **VIEW** tab, select **Toolpaths**, **Solids**, **Planes** or **Levels** as shown.



STEP 6: SETTING MASTERCAM TO IMPERIAL

In this step you will learn how to switch the system to imperial and how to set the imperial system as your default. You will have to select the **Backstage** options and select the system configuration.

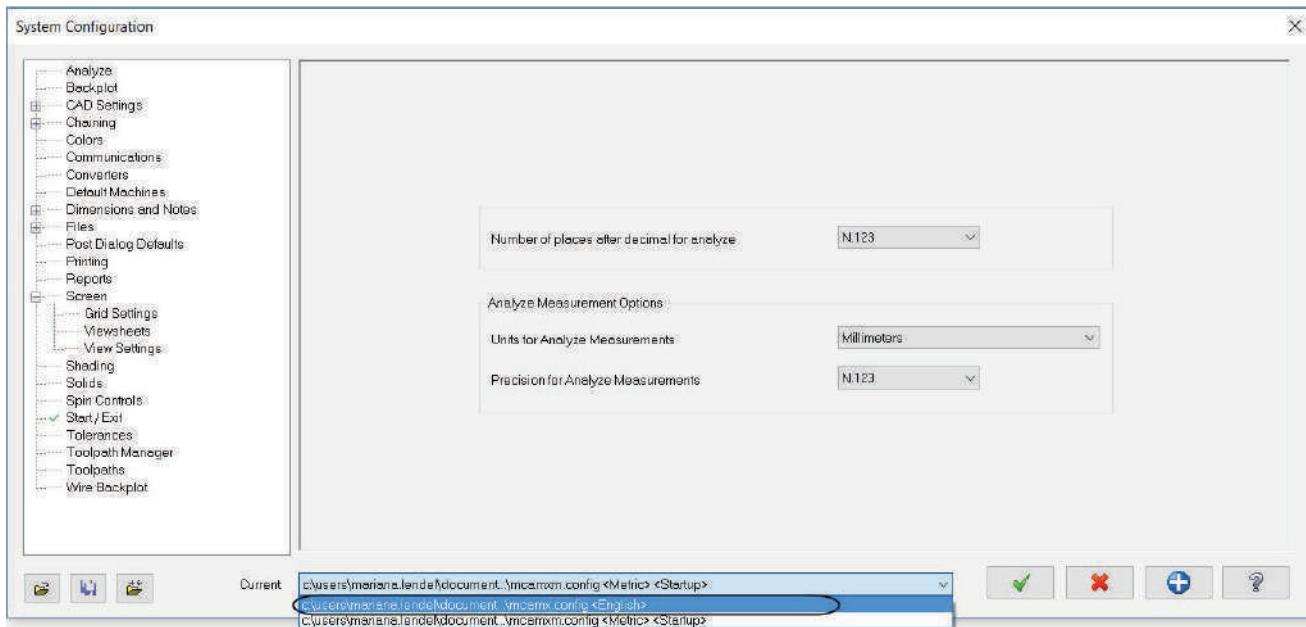
6.1 Setting Mastercam to imperial for the current session only

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

FILE

- ◆ **Configuration.**
- ◆ Select the drop down arrow beside **Current** as shown in [Figure: 6.1.1](#).
- ◆ Select **mcamxm.config <English>** as shown.

Figure: 6.1.1



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

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

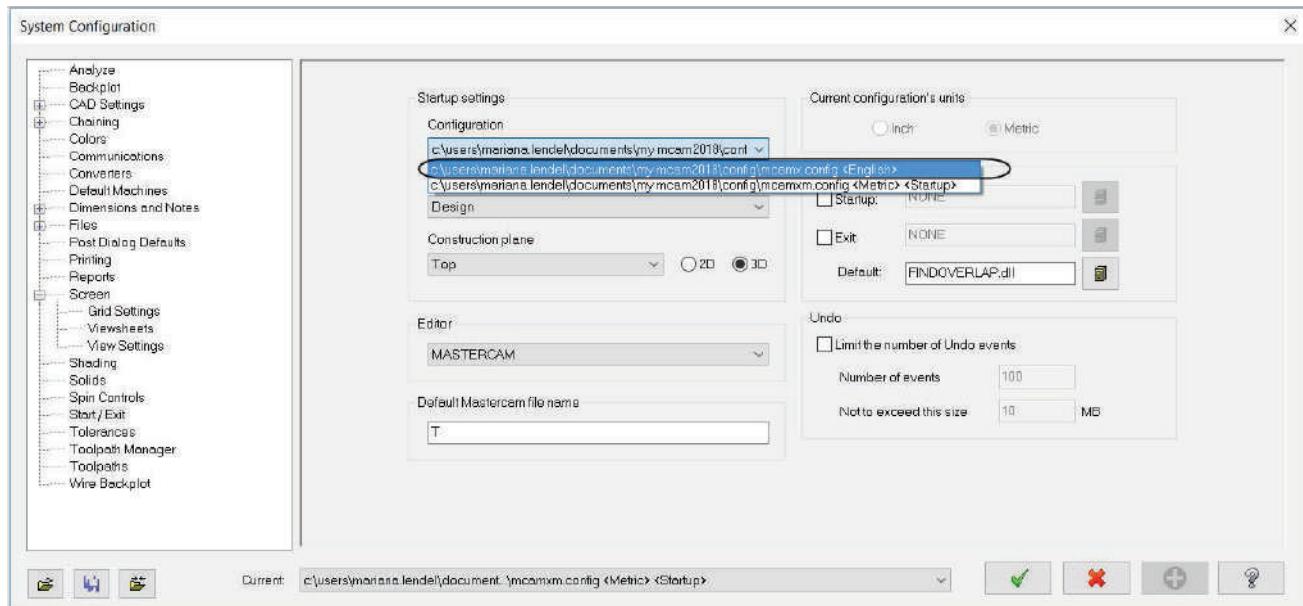
6.2 Setting Mastercam to imperial as a default

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

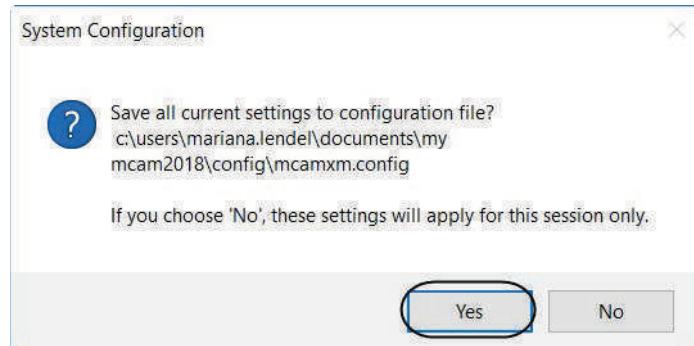
FILE

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

Figure: 6.2.1



- ◆ Select the **OK** button to exit the **System Configuration** dialog box.
- ◆ Mastercam will then prompt you to save these settings to your current configuration file, select **Yes**.



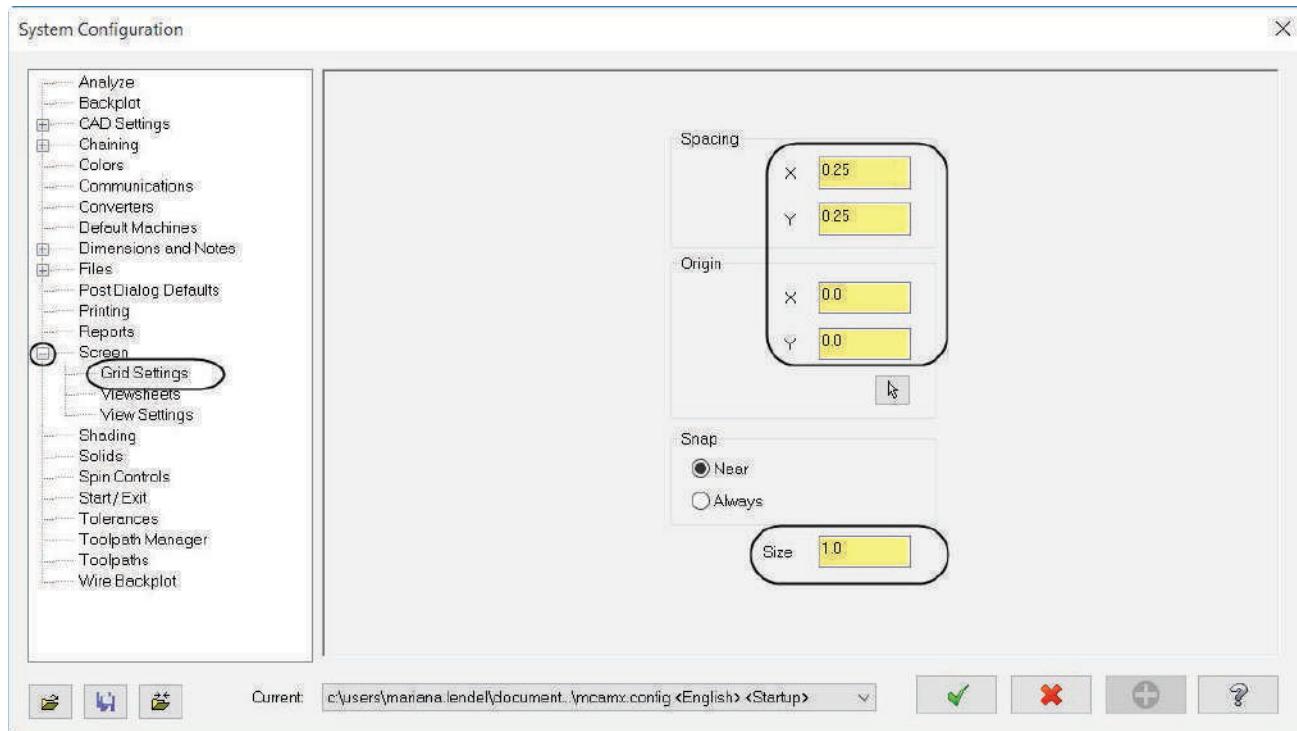
STEP 7: SET THE GRID

Before beginning to create geometry, it is highly recommended to enable the **Grid**. The grid will show you where the origin is and the orientation of the grid gives you a quick preview of the plane you are working in.

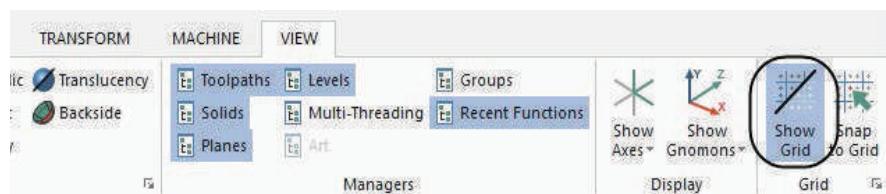
FILE

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

Figure: 7.0.1



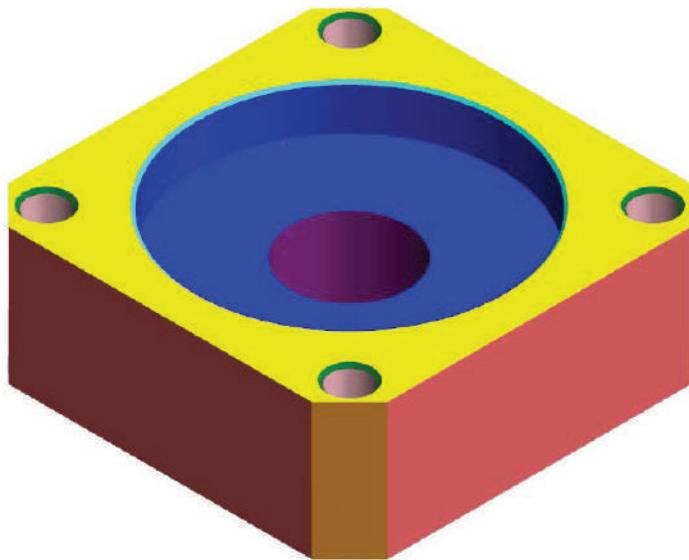
- ◆ In **Grid Settings**, change the **Spacing** to **X = 0.25** and **Y = 0.25**.
- ◆ Set the **Size** to **1.0**.
- ◆ Choose the **OK** button to exit.
- ◆ Select the **Yes** button to save the settings in the **System Configuration**.
- ◆ To see the **Grid** in the graphics window, from the **VIEW**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

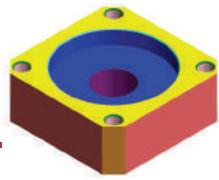


- ◆ The grid should look as shown.



TUTORIAL #1



**OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:****Import the 2D CAD Model and prepare it to generate Toolpaths from:**

- ◆ The student will open the AutoCAD DWG file in Mastercam.
- ◆ The student will move the geometry needed to create the toolpaths on a different level.
- ◆ The student will check for problems in the geometry.
- ◆ The student will move the geometry to set the origin to the lower left corner.
- ◆ The student will change the geometry color.

Create the necessary Toolpaths to machine the part:

- ◆ The student will set up the stock size and the clamping method to be used.
- ◆ A Facing toolpath will be created to machine the top of the part.
- ◆ A Circle Mill toolpath will remove the material inside of the large hole.
- ◆ A Circle Mill toolpath will remove the material inside of the smaller hole.
- ◆ A Drilling toolpath will be created to spot drill the four holes.
- ◆ A Drilling toolpath will be created to machine the through holes.
- ◆ A Contour toolpath with 2D chamfer option will be created to chamfer the top of the boss.
- ◆

Backplot and Verify the file:

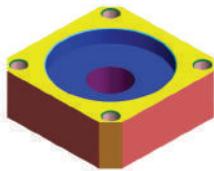
- ◆ The Backplot will be used to simulate a step-by-step process of the tool's movements.
- ◆ The Verify will be used to watch a tool machine the part out of a solid model.

Post Process the file to generate the G-code:

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



This tutorial takes approximately one hour to complete.



GEOMETRY CREATION

STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

Please refer to the **Getting Started** section to set up the graphical user interface. Make sure that the system is set to inch.

STEP 2: IMPORTING THE DWG FILE GEOMETRY

Mastercam lets you read (import) a variety of CAD file types into the Mastercam database. You can also write (export) Mastercam files to a variety of different file formats.

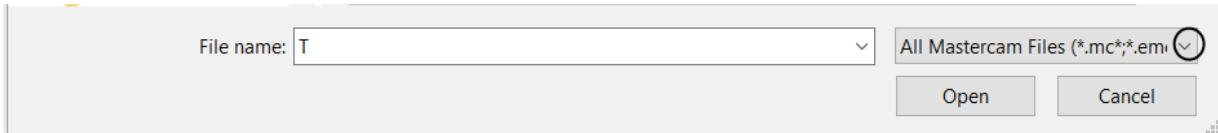
To import an AutoCAD file in Mastercam, you have to use the Open function and then select from the type of file list AutoCAD files.

Download the files from www.emastercam.com/trainingfiles.

Save the file at a preferred location.

FILE

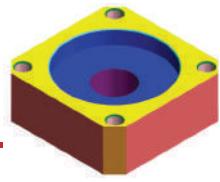
- ♦ Open.
- ♦ In the file name extension, click on the drop down arrow as shown.



- ♦ From the list of file types, select **AutoCAD Files (*.DWG;*.DXF;*.DWF;*.DWFX)** as shown.

Mastercam Educ X7 Files (*.emcx-7)
Mastercam Educ X6 Files (*.emcx-6)
Mastercam Educ X5 Files (*.emcx-5)
Mastercam Educ pre-X5 Files (*.emcx)
Mastercam V9 Files (*.mc9)
Mastercam V8 Files (*.mc8)
All Mastercam Files (*.mc*;*.emc*)
IGES Files (*.igs;*.iges)
AutoCAD Files (*.dwg;*.dxf;*.dwf;*.dwfx)
Parasolid Files (*.x_t;*.x_b;*.xmt;*.txt)
Pro/E/Creo Files (*.prt;*.asm;*.prt;*.asm;*.asm;*.asm)
ACIS Kernel SAT Files (*.sat;*.sab)
STEP Files (*.stp;*.step)
VDA Files (*.vda)
Rhino 3D Files (*.3dm)
SOLIDWORKS Files (*.sldprt;*.sldasm)
SOLIDWORKS Drawing Files (*.slddrw)
Solid Edge Files (*.par;*.psm;*.asm)
Autodesk Inventor Files (*.ipt;*.iam)
Autodesk Inventor Drawing Files (*.idw)
KeyCreator Files (*.ckd)

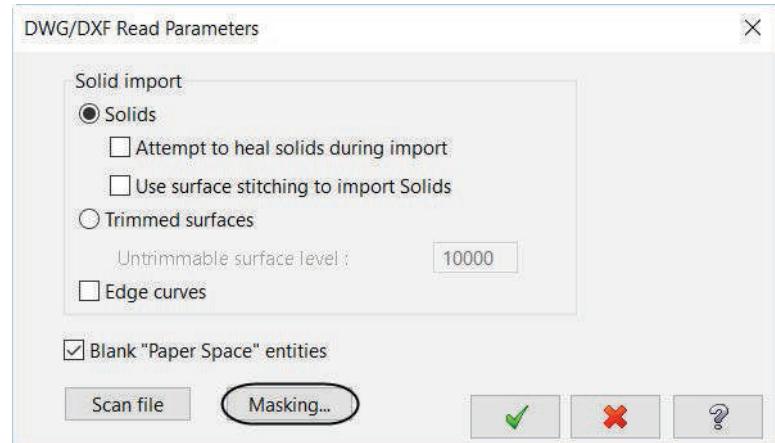
TUTORIAL #1 IMPORTING THE DWG FILE GEOMETRY



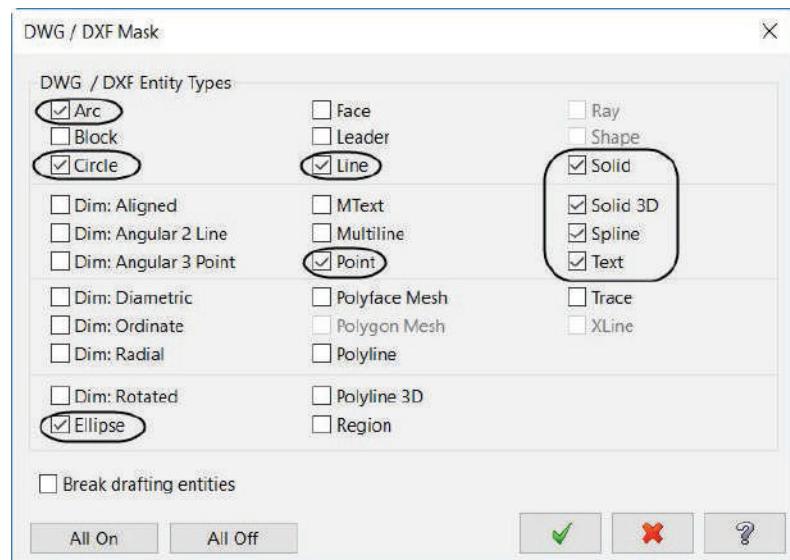
- ♦ Find and select **TUTORIAL_1.DWG**.
- ♦ Select the **Options** button to filter the type of entities that you bring in.



- ♦ From the **DWG/DXF Read Parameters** dialog box, select the **Masking** button as shown.

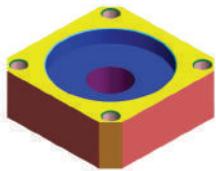


- ♦ Make sure that you only have the following entity types selected to avoid bringing in unnecessary entities.

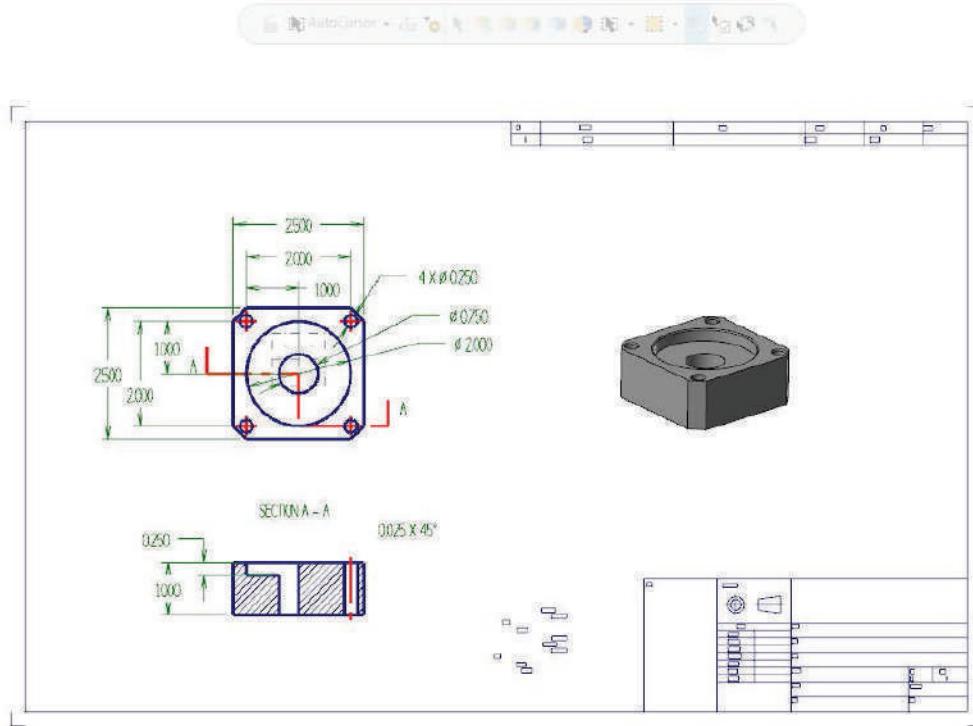


NOTE: Select the **Help** button to view descriptions of all the entity types in this image.

- ♦ Select the **OK** button to exit **DWG/DXF Mask**.
- ♦ Select the **OK** button to exit **DWG/DXF Read Parameters**.



- ♦ Open the file.
- ♦ Press **Alt + F1** to fit the geometry to the screen.
- ♦ The geometry should look as shown.



NOTE: To better display the geometry in the book, the background color in Mastercam is set to white and the title box and the geometry are in black color. On your screen you should see the default Mastercam background which is a grey gradient color and the geometry will be shown in white.

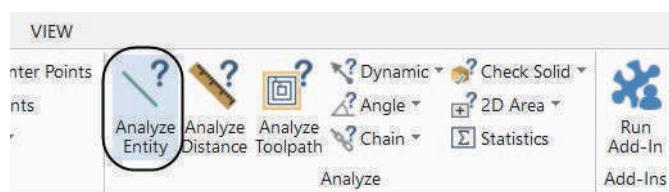
The dimensions, title block, and some of the geometry are not required for machining purposes. To be able to see only the geometry needed for toolpath creation, you will move it to a different level. This will be covered in the following steps of this tutorial.

STEP 3: ANALYZE THE GEOMETRY

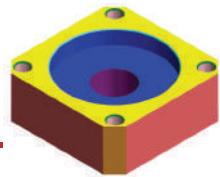
To machine this part, the geometry has to be created at a 1:1 scale. In order to check the geometry's accuracy, you will analyze one of the entities that is dimensioned and check if the value is the same as the one in the drawing.

HOME

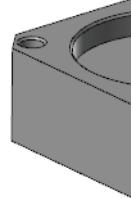
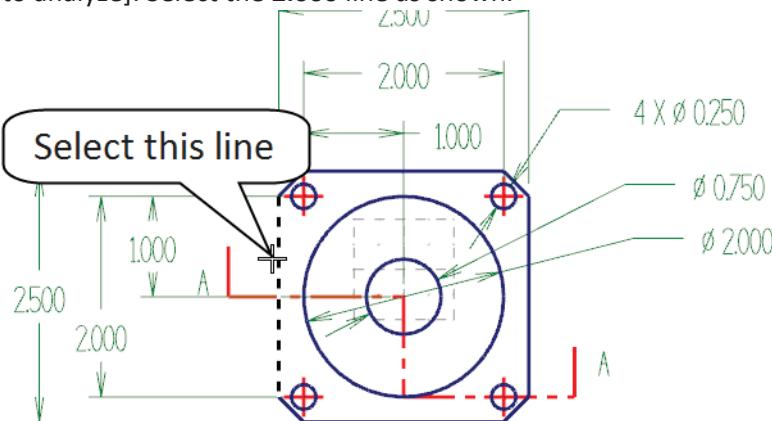
- ♦ From the **Analyze** group, select **Analyze Entity**.



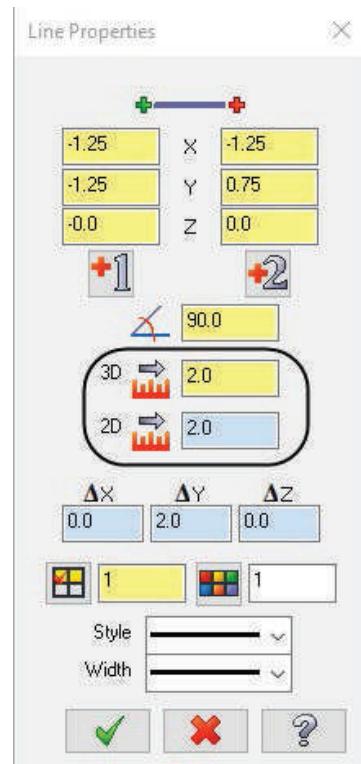
TUTORIAL #1 ANALYZE THE GEOMETRY



- [Select entities to analyze]: Select the **2.000** line as shown.



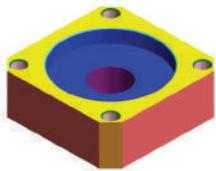
- In the **Line Properties**, the **3D** and **2D** are both **2.0** as shown.



NOTE: The geometry was created at the proper scale 1:1. If the entity value was different than the dimension value, you can use the **Transform Scale** command to increase or decrease the size of the entities by a factor relative to a defined point using the uniform method.

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





STEP 4: MOVE THE GEOMETRY FOR THE TOOLPATHS TO A DIFFERENT LEVEL

Levels is a primary organizational tool in Mastercam. A Mastercam file can contain separate levels for wireframe, surfaces, drafting entities, and toolpaths. By organizing your files into levels, you can more easily control which areas of the drawing are visible at any time and which parts are selectable so that you do not inadvertently make changes to areas of the drawing you do not want to change.

In this step you will select the geometry required for the toolpath and move it to a different level.

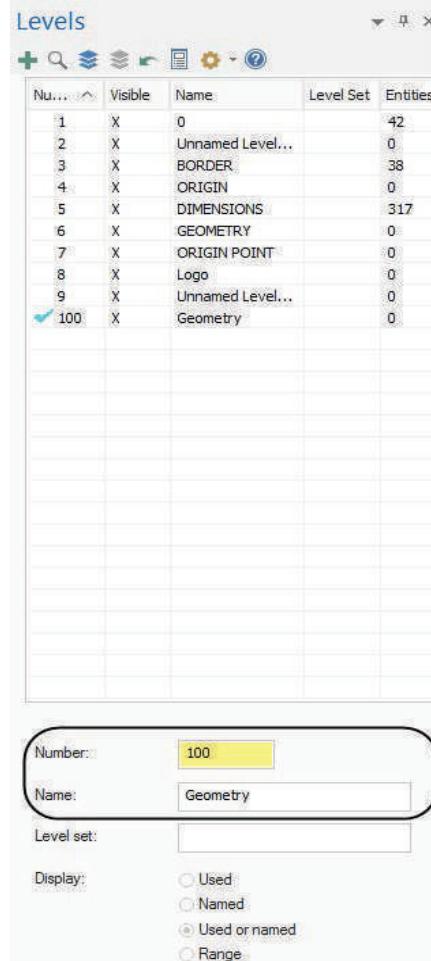
4.1 Set the Main Level and name it

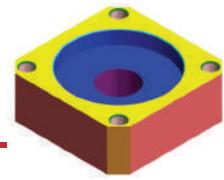
The **Main Level** is the current working level. Any geometry that you create is always placed on the main level. There can be only one main level at a time. The number of the main level appears on the Level button in the **Status bar**. In the **Levels Manager** dialog box, the main level is highlighted in yellow.

- From the bottom left corner of the screen, click on the **Levels** tab as shown.



- Change the **Number** in the **Main Level** to **100** and enter the name **Geometry** as shown.





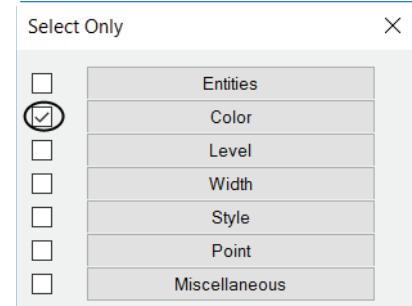
4.2 Move the geometry to the Main Level 100

NOTE: To select the geometry you want to keep, use the window selection, a default selection method in Mastercam. In order to create the window, simply click and drag until the window covers the selected parts completely. In order to only select the desired geometry lines, enter the **General Selection** toolbar and select only the color blue geometry inside of the window.

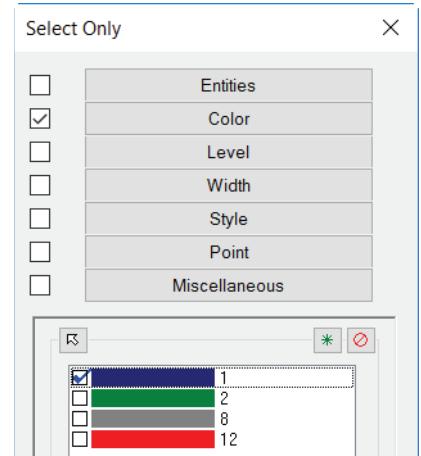
- From the **Quick Mask** buttons, choose **Select only advanced** as shown.



- In the **Select Only** dialog box, enable **Color** as shown.



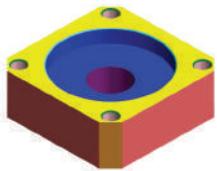
- Click on the **Color** button and enable color blue number 1 as shown.



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



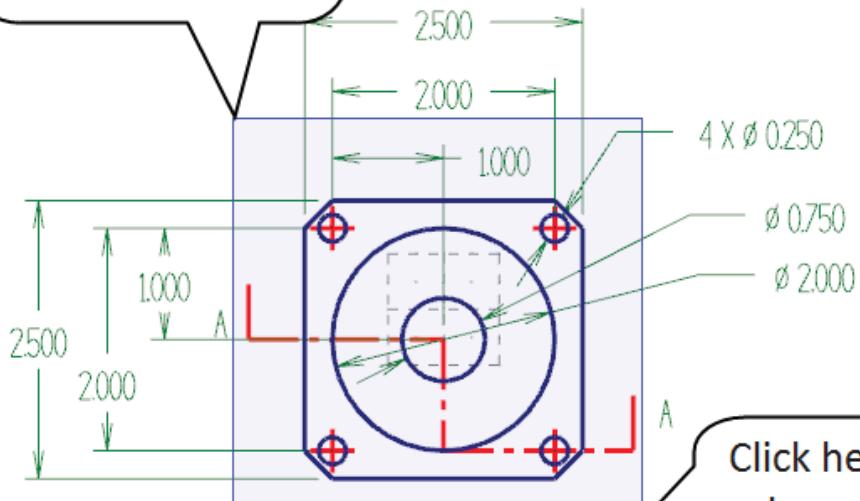
NOTE: The **Only** button in the **General Selection** is highlighted. It will stay like this while the color mask is on.



- Select the upper corner and the lower corner of the window as shown in [Figure: 4.2.1](#).

Figure: 4.2.1

Click here for the
upper corner



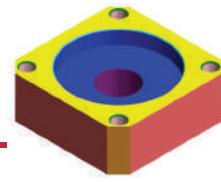
Click here for the
lower corner

NOTE: The entities selected will be shown with hidden lines.

- Right mouse click on the graphics window and select **Change Levels** as shown.



TUTORIAL #1 MOVE THE GEOMETRY FOR THE TOOLPATHS TO A DIFFERENT LEVEL



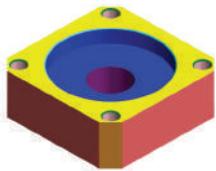
- The Change Levels dialog box should appear on the screen as shown.



NOTE: The parameters in the **Change Levels** dialog box are already set to move the geometry to the **Main Level**.

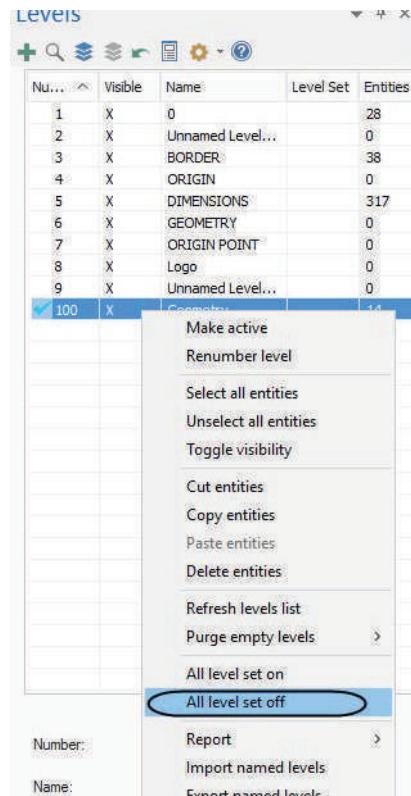
- Leave the default settings in the **Change Levels** and select the **OK** button to continue.



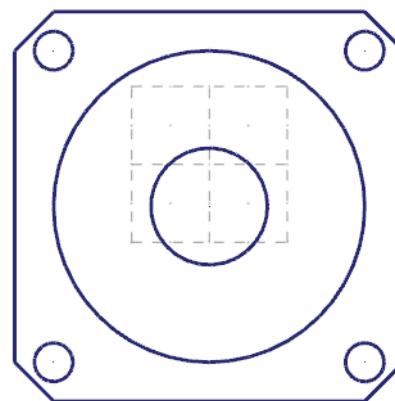


4.3 Make the Main Level the only visible level

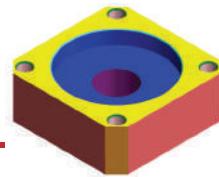
- In the **Levels Manager**, right mouse click on the number **100** and select **All level set off** to make all the levels invisible except the **Main level** as shown.



- Press **Alt + F1** to fit the geometry in the graphics window.
- The geometry should look as shown.

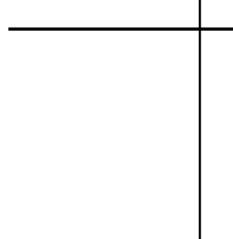


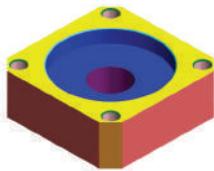
NOTE: When you import geometry from a different CAD software, before starting to make any toolpath, you should check for any problems in the geometry that will be selected in the toolpath.



STEP 5: CHECKING FOR CHAINING PROBLEMS

Chaining is a process used to select a number of entities (lines, arcs, points, splines, etc.) when creating a toolpath. The entities form a chain when the distance between two consecutive entity endpoints is less than the chaining tolerance. The following table illustrates the problems that can be encountered and the selection stops.

Description	Check	Fix
Chain stops: Lines do not touch. 	Zoom in to inspect.	Trim lines together using trim 1 entity.
Chain stops: Lines cross. 	Zoom in to inspect.	Trim lines using trim 2 entities.
Lines do not touch. 	Look at geometry in ISO Gview.	Use TRANSFORM/Project.



Description	Check	Fix
Duplicate entities: Identical entities on top of each other.	When you delete the line, another one appears in its place. 	Delete Duplicate entities.
Overlapping entities: Entities of different lengths on top of each other.	Move the cursor over the top of the entities, you may see different entities highlight. 	Use Findoverlap.dll hook.

5.1 Check for Duplicate Entities

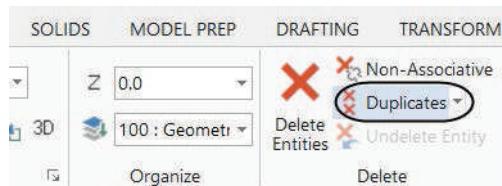
- ♦ Make sure that no entity is selected (highlighted in yellow color).

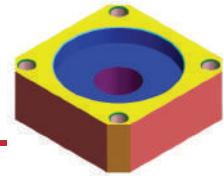
NOTE: To unselect geometry, click on it or click on **Clear selection** in the **Quick Mask** buttons.



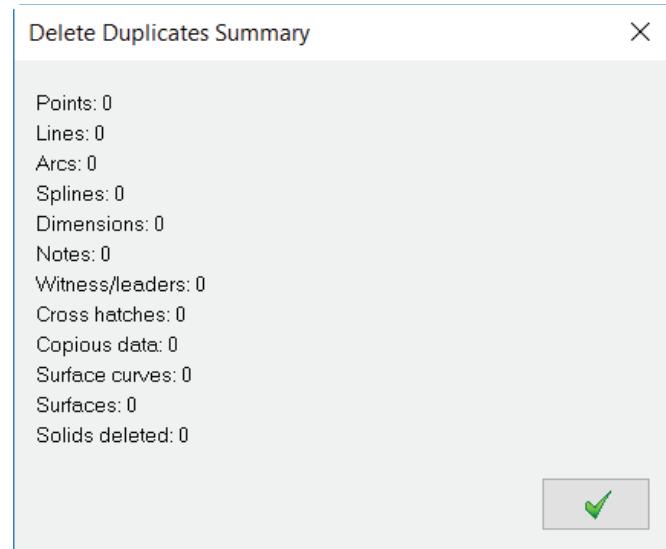
HOME

- ♦ From the **Delete** group, select **Duplicates** to delete duplicate entities.





- ♦ A dialog box should appear on the screen listing any type of duplicate entities that were found and deleted as shown.



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



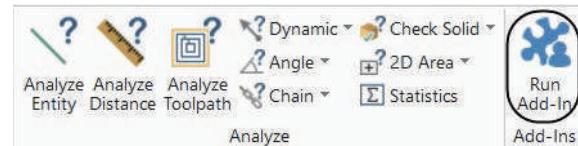
5.2 Check for Overlapping Entities Using FindOverlap C-Hook

C-Hooks and NET-Hooks are add-in applications or utilities that customize, enhance, or extend Mastercam's functionality. FindOverlap C-Hook allows you to find, display, and clean up overlapping lines and arcs. Overlapping entities can interfere with chaining, and are often difficult to see and delete.

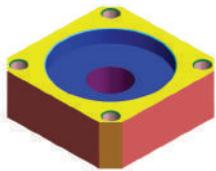
NOTE: The **C-Hooks** are not available with **Mastercam Home Learning Edition**. Skip this step if you do not have an educational or industrial seat.

HOME

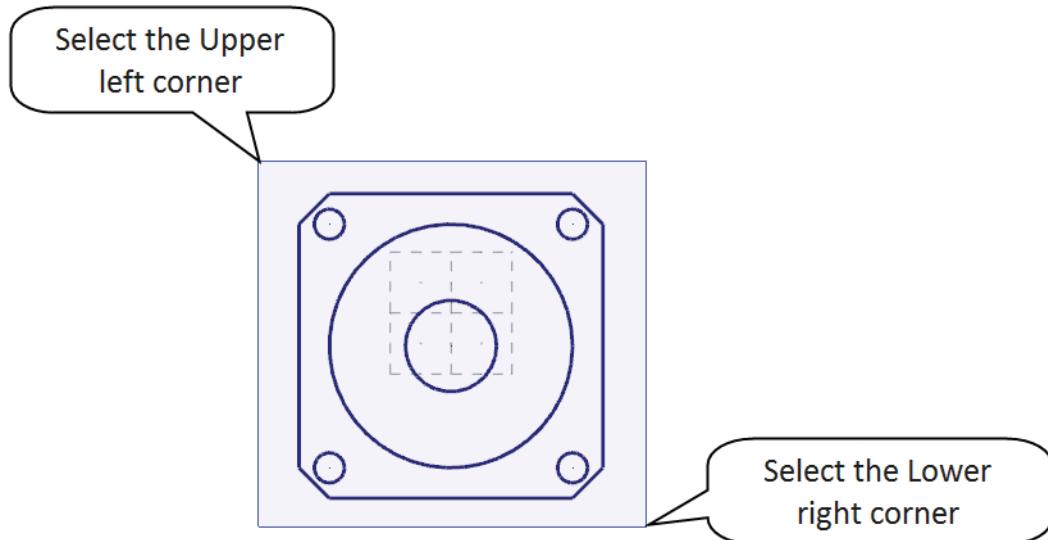
- ♦ To open the **C-hook** directory, from the **Add-Ins** group select **Run Add-In** as shown.



- ♦ Select the **FindOverlap.dll** and open it.

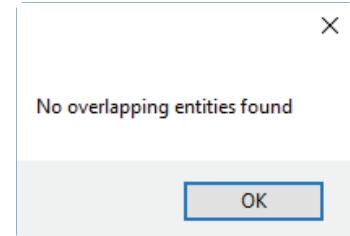


- ♦ [Select entities to check for overlap]: Make a window around the entire geometry as shown.

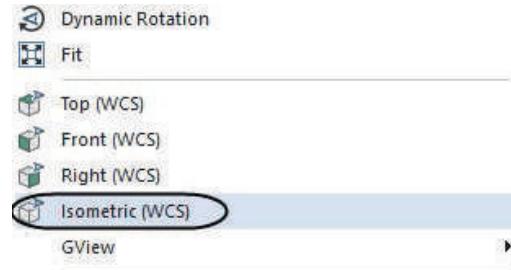


- ♦ Click on **End Selection** to finish the selection.
- ♦ This dialog box should appear showing that the geometry has no overlapping entities as shown in [Figure: 5.2.1](#).

Figure: 5.2.1

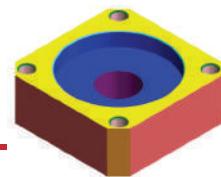


- ♦ Select the **OK** button to continue.
- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.

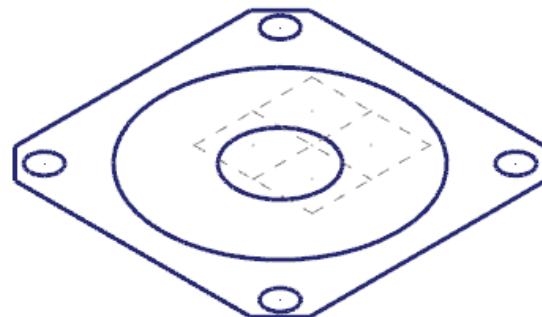


TUTORIAL #1

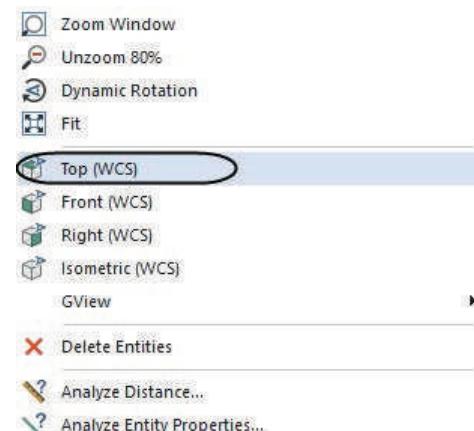
CHECKING FOR CHAINING PROBLEMS

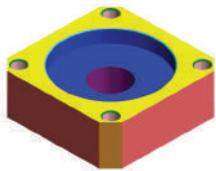


- ◆ The geometry should look as shown.



- ◆ Change the **Graphic view** to **Top** as shown.





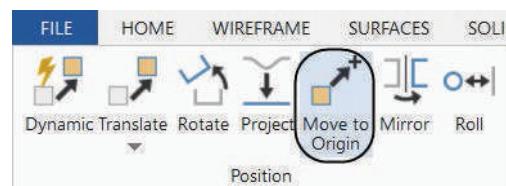
STEP 6: MOVE THE GEOMETRY TO THE ORIGIN

In this step you will use the **Move to Origin** command to quickly move all visible geometry to the current **WCS (Work Coordinate System)** origin based on a point that you select with the **AutoCursor**. This origin will be the origin you will set on your part at the machine.

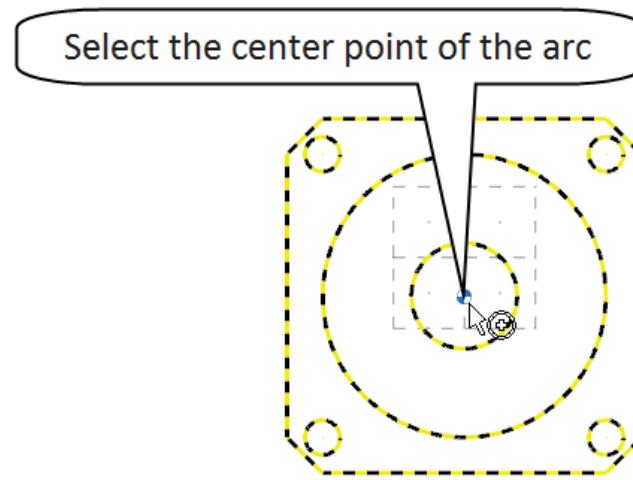
Mastercam uses a 3D Cartesian coordinate system to locate your work in three-dimensional space. This means that geometry and toolpath positions are expressed in terms of three coordinate axes: X, Y, and Z. Each axis is signed, which means that it has a positive and a negative direction.

TRANSFORM

- From the **Position** group, select **Move to Origin** as shown.



- [Select the point to translate from]: Select the arc center point as shown.

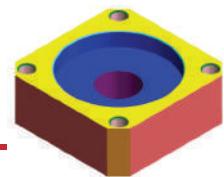


NOTE: Make sure that the arc center point cue tip appear on the screen when selecting it as shown.

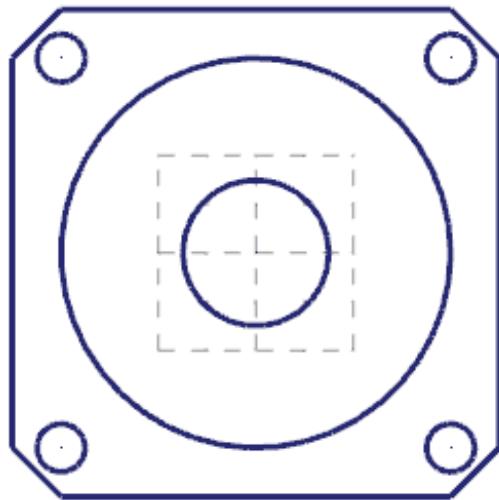
- Right click on the graphics window and select **Clear Colors** as shown to remove the result color.



- Press **Alt + F1** to fit the geometry to the screen. Scroll up and down the mouse wheel to zoom in or out.
- Hold down the **Shift** key and the mouse wheel to drag the part in the center of the graphics window.



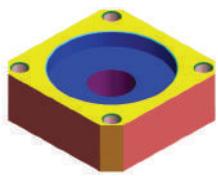
- ◆ The geometry should look as shown.



STEP 7: SAVE THE FILE

FILE

- ◆ **Save As.**
- ◆ File name: "Your Name_1".

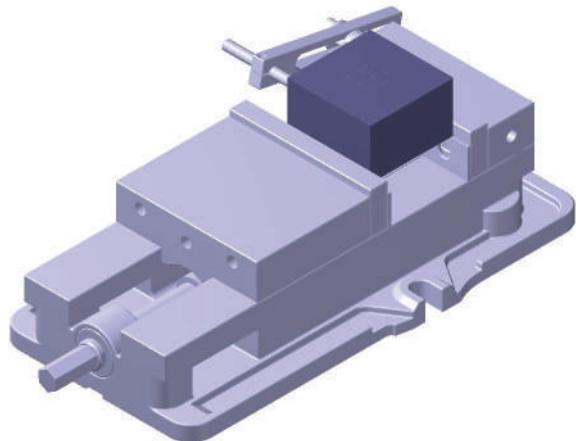


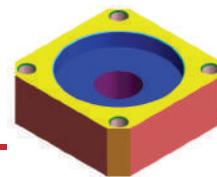
SUGGESTED FIXTURE:

TUTORIAL #1

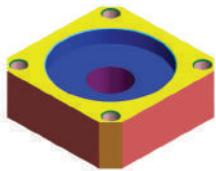
TOOLPATH CREATION

SUGGESTED FIXTURE:



TUTORIAL #1**SETUP SHEET:****SETUP SHEET:****TOOL LIST**

	TYPE: Face mill DIA OFFSET: 1 HOLDER: DEFAULT HOLDER NUMBER: 1 LENGTH OFFSET: 1 #1 - 2.0000 FACE MILL - 2" FACE MILL	FLUTE LENGTH: 1.575 OVERALL LENGTH: 2.1 CORNER RAD: 0.0 # OF FLUTES: 2
	TYPE: Endmill 1 Flat DIA OFFSET: 2 HOLDER: DEFAULT HOLDER NUMBER: 2 LENGTH OFFSET: 2 #2 - 0.7500 ENDMILL FLAT - 3/4 FLAT ENDMILL	FLUTE LENGTH: 2.0 OVERALL LENGTH: 3.75 CORNER RAD: 0.0 # OF FLUTES: 4
	TYPE: Endmill 1 Flat DIA OFFSET: 3 HOLDER: DEFAULT HOLDER NUMBER: 3 LENGTH OFFSET: 3 #3 - 0.2500 ENDMILL FLAT - 1/4 FLAT ENDMILL	FLUTE LENGTH: 0.5 OVERALL LENGTH: 2.5 CORNER RAD: 0.0 # OF FLUTES: 4
	TYPE: Spot Drill DIA OFFSET: 4 HOLDER: DEFAULT HOLDER NUMBER: 4 LENGTH OFFSET: 4 #4 - 0.7500 SPOT DRILL - 3/4 SPOTDRILL	FLUTE LENGTH: 2.0 OVERALL LENGTH: 3.0 CORNER RAD: 0.0 # OF FLUTES: 4
	TYPE: Drill DIA OFFSET: 5 HOLDER: DEFAULT HOLDER NUMBER: 5 LENGTH OFFSET: 5 #5 - 0.2500 DRILL - 1/4 DRILL	FLUTE LENGTH: 2.0 OVERALL LENGTH: 3.0 CORNER RAD: 0.0 # OF FLUTES: 2
	TYPE: Chamfer mill DIA OFFSET: 6 HOLDER: DEFAULT HOLDER NUMBER: 6 LENGTH OFFSET: 6 #6 - 0.5000 CHAMFER MILL - 1/2 CHAMFER MILL	FLUTE LENGTH: 0.75 OVERALL LENGTH: 2.5 CORNER RAD: 0.0 # OF FLUTES: 4
	TYPE: Endmill 1 Flat DIA OFFSET: 7 HOLDER: DEFAULT HOLDER NUMBER: 7 LENGTH OFFSET: 7 #7 - 0.5000 ENDMILL FLAT - 1/2 FLAT ENDMILL	FLUTE LENGTH: 1.0 OVERALL LENGTH: 3.0 CORNER RAD: 0.0 # OF FLUTES: 4



STEP 8: SELECT THE MACHINE AND SET UP THE STOCK

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

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

VIEW

- ♦ To display the **Toolpaths Manager**, from the **Mangers** group select **Toolpaths** as shown.



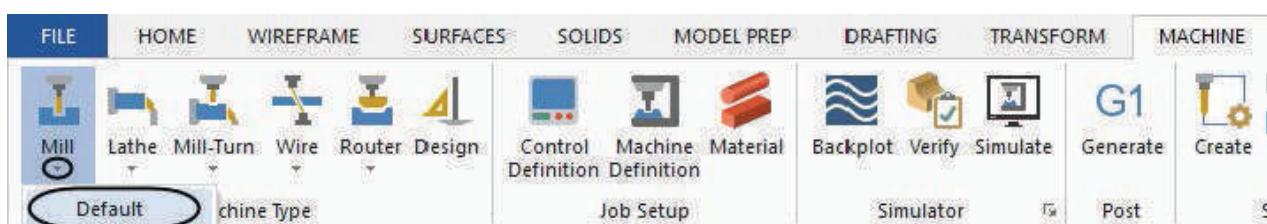
- ♦ Pin the **Toolpaths Manager** by clicking on the **Auto Hide** icon as shown.



NOTE: Select the **Mill Default** only if there is no **Machine Group** in the **Toolpaths Manager**.

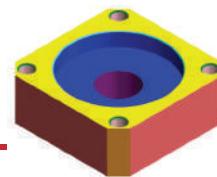
MACHINE

- ♦ From the **Machine Type** group, select the drop down arrow below **Mill**.
- ♦ Select the **Default**.



NOTE: Once you select the **Mill Default**, the ribbon bar changes to reflect the toolpaths that could be used with **Mill Default**.

TUTORIAL #1 SELECT THE MACHINE AND SET UP THE STOCK

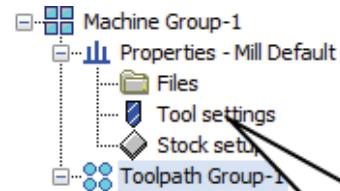


- ♦ Select the plus sign in front of **Properties** in the **Toolpaths Manager** to expand the **Toolpaths Group Properties**.



Select the plus sign

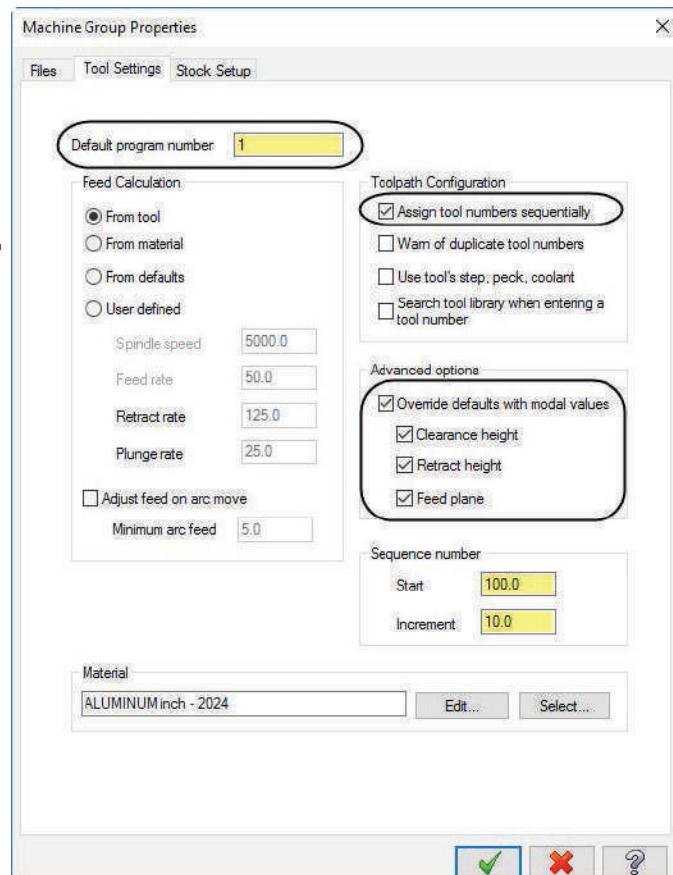
- ♦ Select **Tool settings** to set the tool parameters.



Select Tool settings

- ♦ Change the parameters to match the screen shot as shown in [Figure: 8.0.1](#).

Figure: 8.0.1



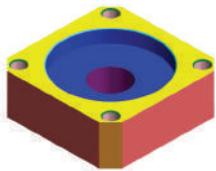
Default program number is used to enter a number if your machine tool requires a number for a program name.

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

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

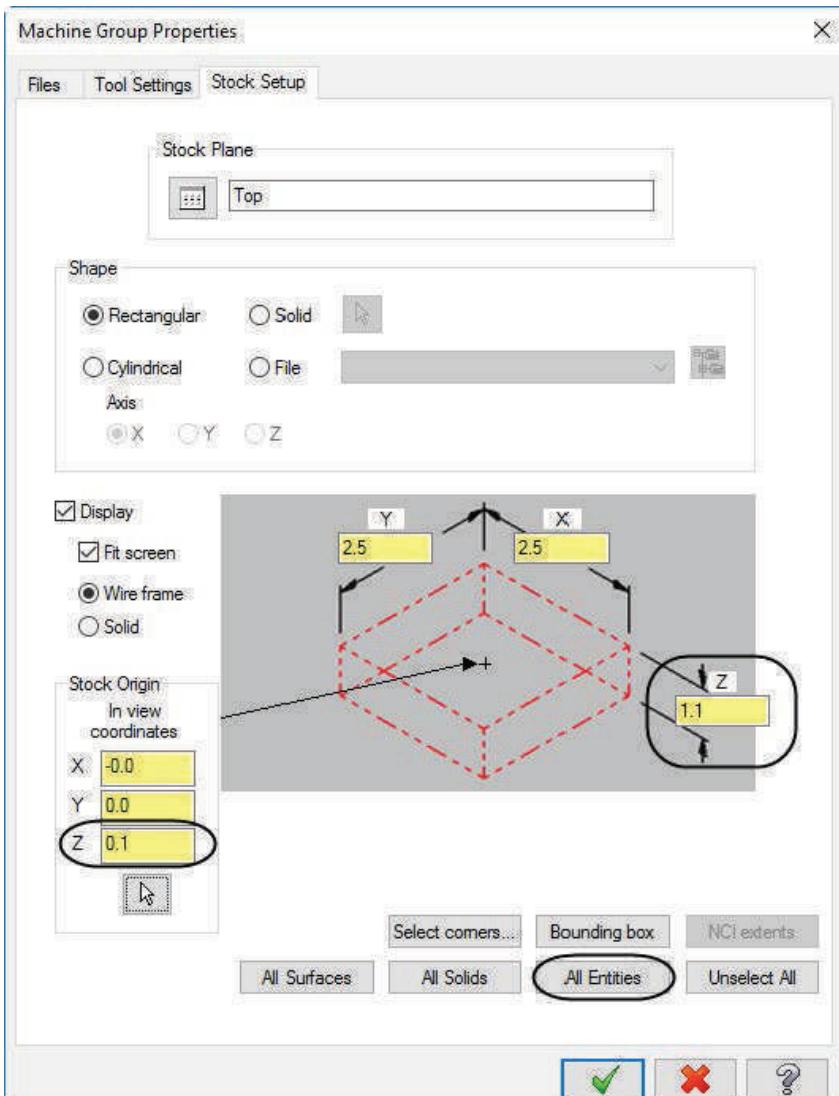
Override defaults with modal values enables the system to keep the values that you enter.

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



- ♦ Select the **Stock Setup** tab to define the stock.
- ♦ Select the **All Entities** button near the bottom of the **Stock Setup** page as shown.
- ♦ In the **Stock Setup**, enter in the **Z** field **1.1** and the **Z Stock Origin 0.1**. Make sure that the rest of the parameters are as shown in [Figure: 8.0.2](#).

Figure: 8.0.2



The X, Y, Z values in the graphics area are the dimensions of the stock model. They are always positive values.

The **Stock Origin** values adjust the positioning of the stock, ensuring that you have an equal amount of extra stock around the finished part. In the graphics, the plus sign (+) shows you where the stock origin is. The default position is the middle of the stock.

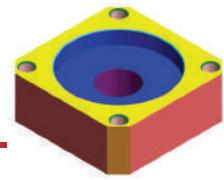
Display options allow you to set the stock as **Wireframe** and to fit the stock to the screen. (Fit Screen)

NOTE: The stock model that you create is used when viewing the file or the toolpaths, during backplot, or while verifying toolpaths.

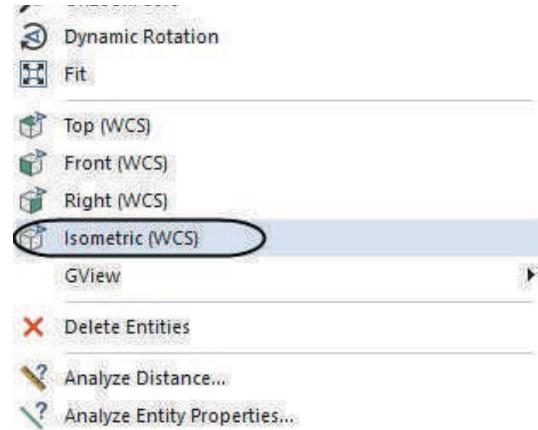
- ♦ Select the **OK** button to exit **Machine Group Properties**.



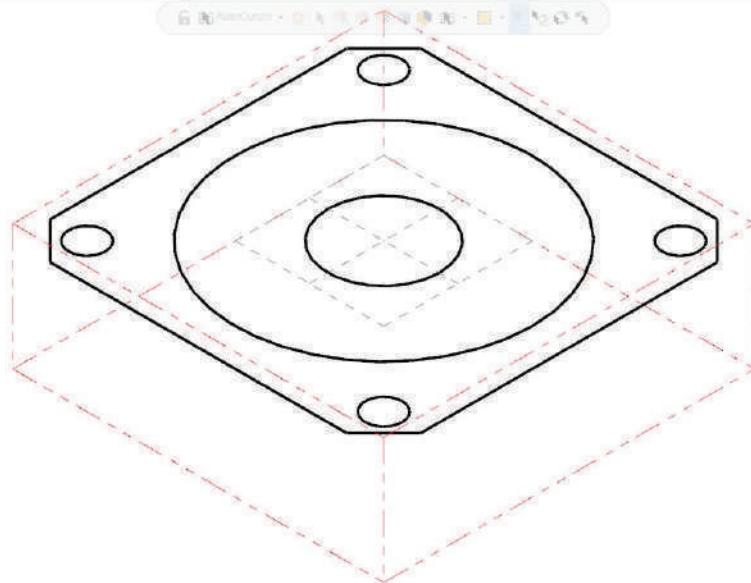
TUTORIAL #1 SELECT THE MACHINE AND SET UP THE STOCK



- Right mouse click in the graphics window and select **Isometric** to see the stock.



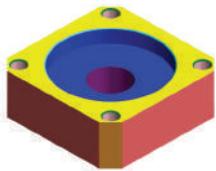
- Press **Alt + F1** to fit the geometry to the screen.
- The stock model will appear as shown.



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

- Right mouse click on the graphics window and select the **Top** view to see the part from the top.

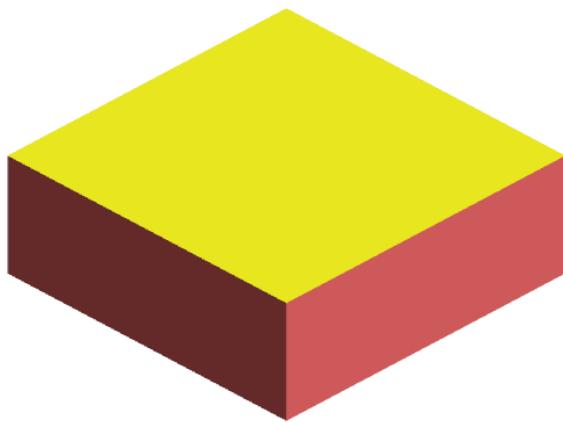




STEP 9: FACE THE PART

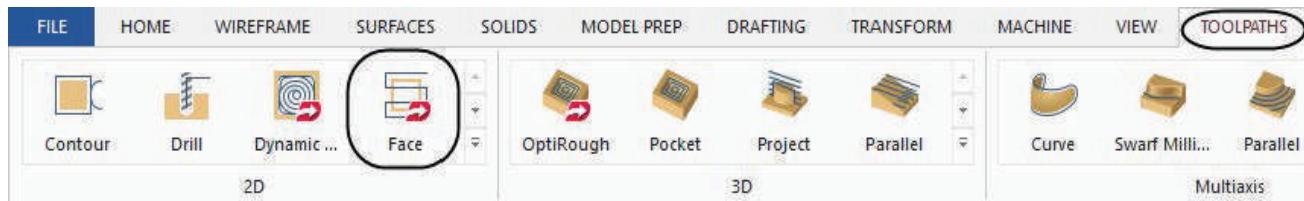
A **Facing** toolpath quickly removes material from the top of the part to create an even surface for future operations.

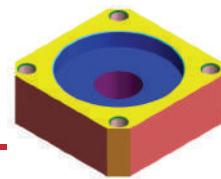
Toolpath Preview:



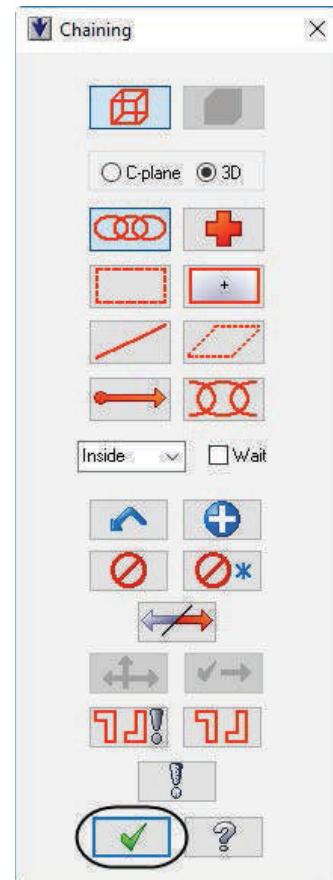
TOOLPATHS

- From the **2D** group, select **Face** as shown.



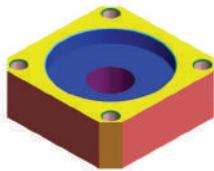


- When the chaining dialog box appears, choose the **OK** button to use the defined stock and exit the **Chaining** dialog box.



NOTE: Mastercam will create the **Facing** toolpath defined from the stock setup.

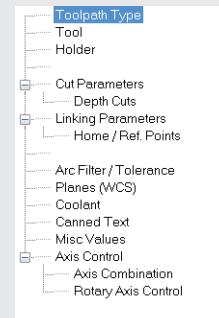
For more information on the **Chaining** button and **Options**, click on the **Help** button.



- In the Toolpath Type page, the **Facing** icon will be automatically selected.

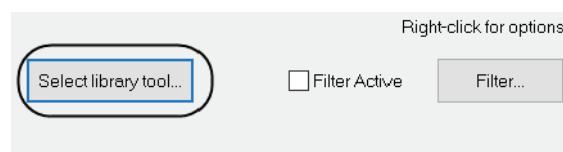


NOTE: Mastercam updates the pages as you modify them and then marks them in the **Tree View list** with a green check mark. Pages that are not enabled are marked with a red circle and slash.

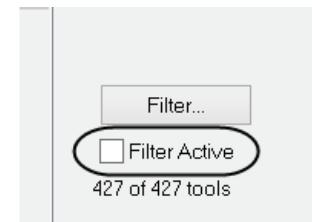


9.1 Select a 2.0" Face Mill from the library and set the Tool parameters

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

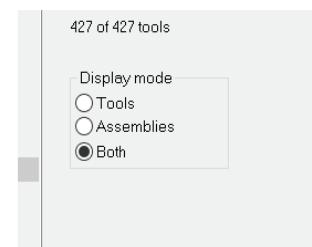


- To be able to see all the tools from the library, disable **Filter Active**.



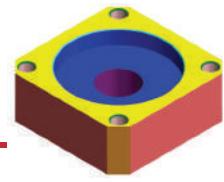
- Select the **2"** Face Mill as shown.

318	-	1/4 CHA...	-	0.25...	0.0	0.5	4	Cha...	None
319	-	1/2 CHA...	-	0.5-45	0.0	0.75	4	Cha...	None
320	-	3/4 CHA...	-	0.75...	0.0	1.0	4	Cha...	None
321	-	1 INCH C...	-	1.0-45	0.0	1.0	4	Cha...	None
322	-	2" FACE ...	-	2.0	0.0	1.575	2	Fac...	None
323	-	2-1/2" FA...	-	2.5	0.0	1.575	4	Fac...	None
324	-	3" FACE ...	-	3.0	0.0	1.969	4	Fac...	None
325	-	4" FACE ...	-	4.0	0.0	1.969	2	Fac...	None
326	-	5" FACE ...	-	5.0	0.0	2.48	4	Fac...	None
327	-	6" FACE ...	-	6.0	0.0	2.48	2	Fac...	None



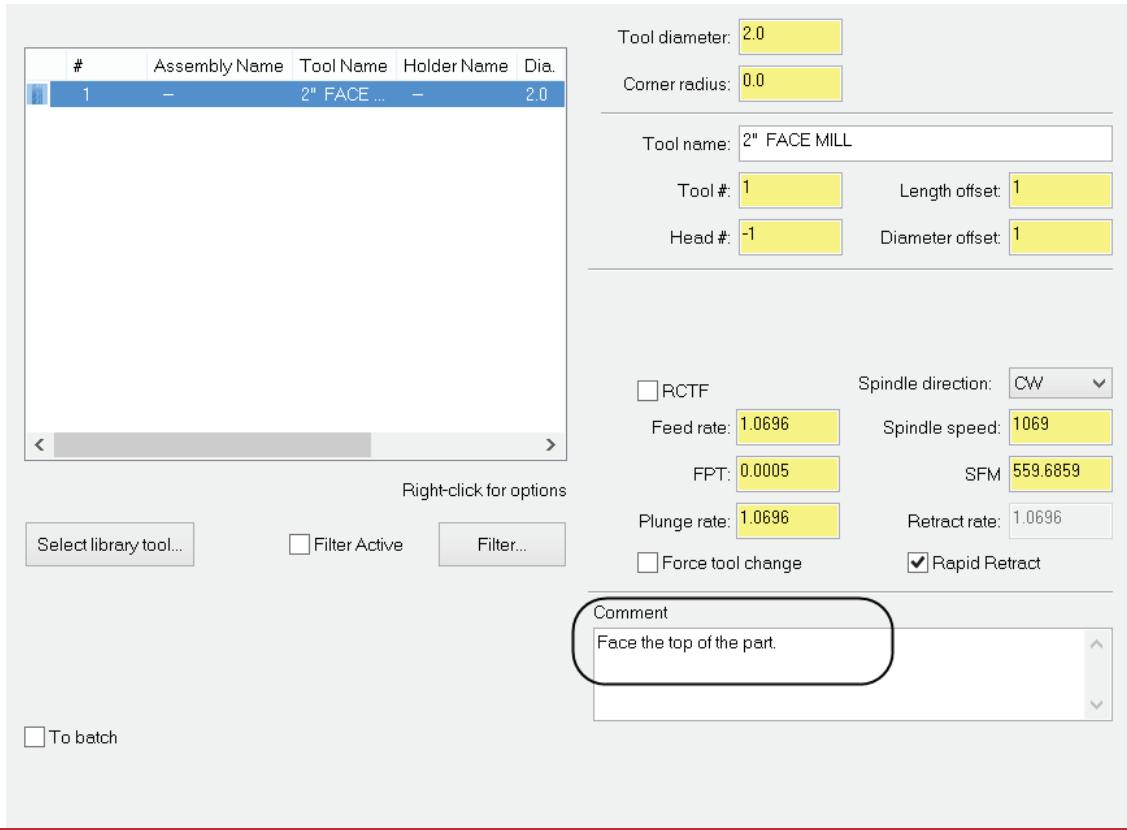
- Select the tool in the **Tool Selection** page and then select the **OK** button to exit.





- ♦ Make all the necessary changes as shown in [Figure: 9.1.1](#).

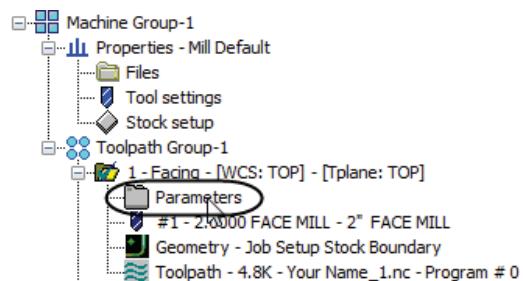
Figure: 9.1.1

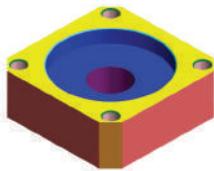


The **Feed rate**, **Plunge rate**, **Retract rate** and **Spindle speed** are based on the tool definition as set in the **Tool Settings**. You may change these values as per your part material and tools.

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

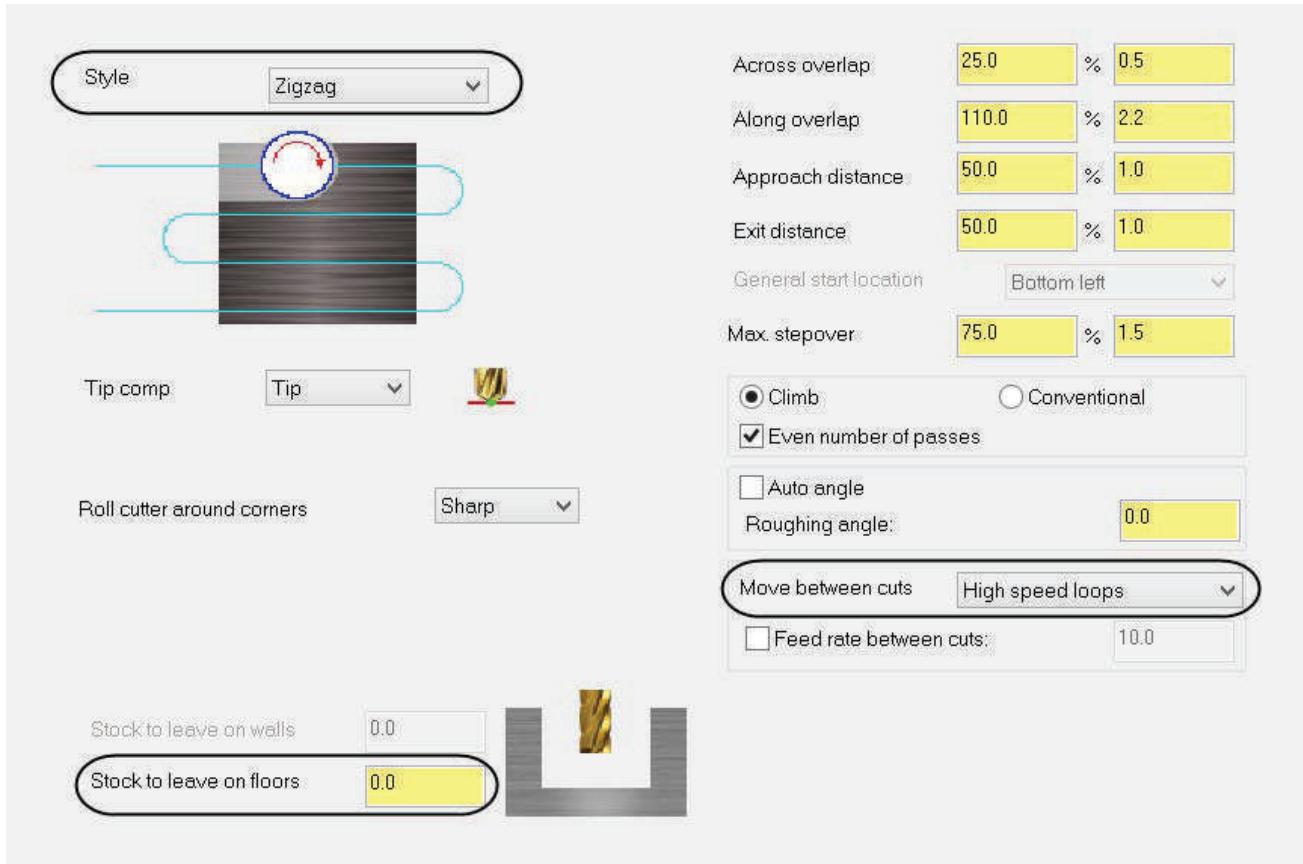
NOTE: If by mistake you click the **OK** button, the toolpath will be generated without all the parameters set properly. To go back in the parameters area, in the **Toolpaths Manager**, click on the **Parameters** as shown.





- From the Tree View list, select **Cut Parameters** and make the necessary changes as shown in [Figure: 9.1.2](#).

Figure: 9.1.2

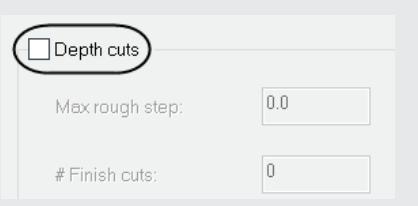


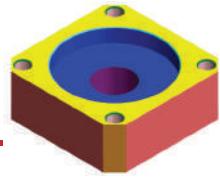
The **Style** (facing cutting method) **Zigzag** creates a back and forth cutting motion.

Move between cuts determines how the tool moves between each cut. This is only available if you select the zigzag cutting method.

High speed loops create 180 degree arcs between each cut.

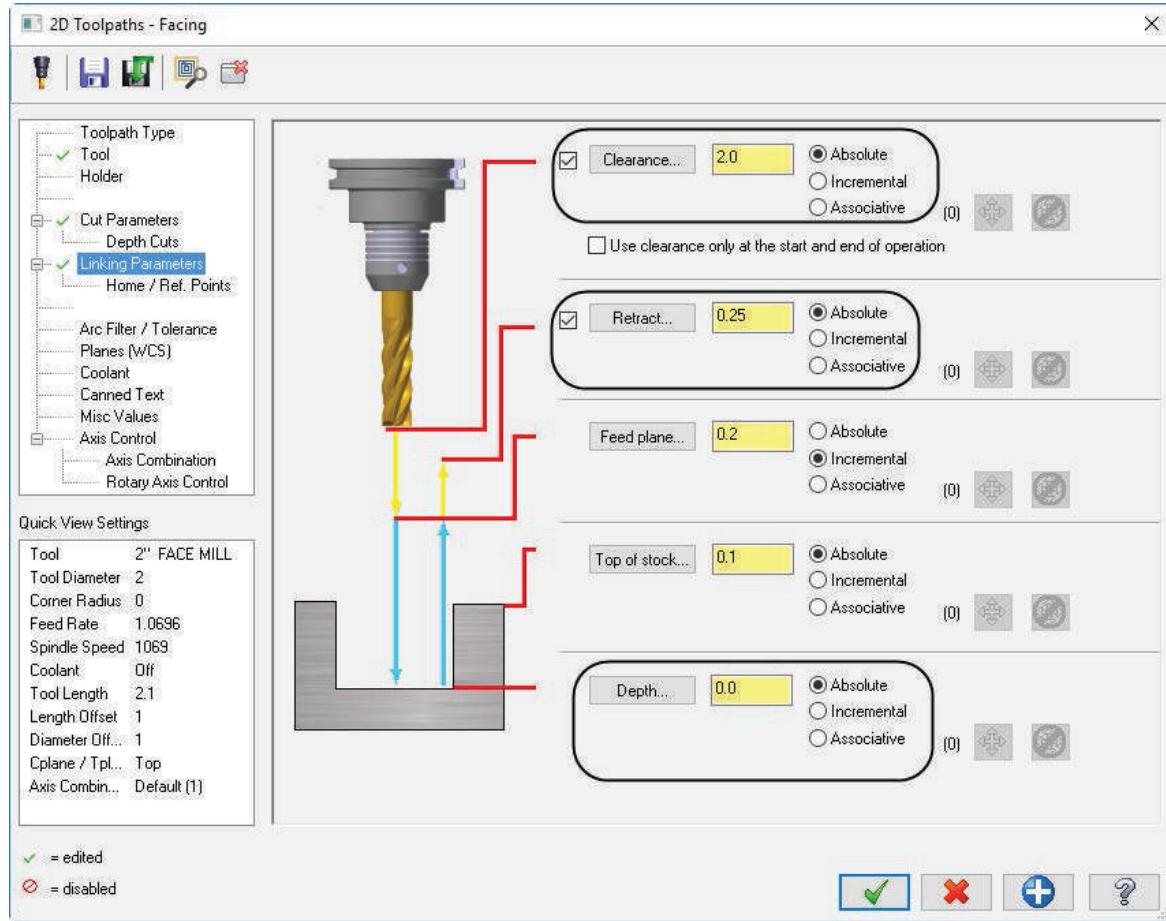
NOTE: Ensure that depth cuts is disabled. If not, select **Depth Cuts** from the **Tree View list** and disable it as shown.





- From the Tree View list, select the **Linking Parameters** page and make the necessary changes as shown in [Figure: 9.1.3](#).

Figure: 9.1.3



Clearance sets the height at which the tool moves to and from the part.

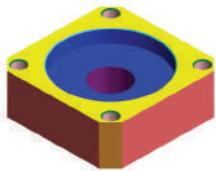
Retract sets the height that the tool moves up to before the next tool pass.

Feed plane sets the height that the tool rapids to before changing to the plunge rate to enter the part.

Top of stock sets the height of the material in the Z axis.

Depth determines the final machining depth that the tool descends into the stock.

NOTE: The **Top of stock** is set to **0.1"** because the **Stock Origin** was set to **0.1"** above the origin. The depth is set to **0.0"** because this is the finish depth. The majority of the values are set to absolute (measured from Z zero which is set at the top of the finished part). **Feed plane** set to incremental is measured from the **Top of stock**.



9.2 Preview the Toolpath

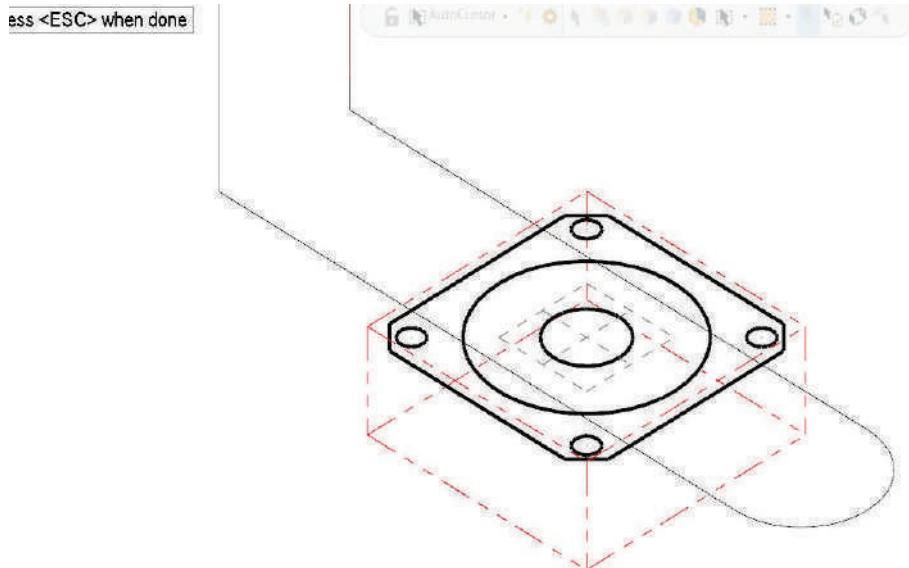
- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.
- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.



- ♦ Press **Esc** key to exit the preview.

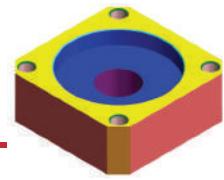
NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to exit the **Facing Parameters**.



TUTORIAL #1

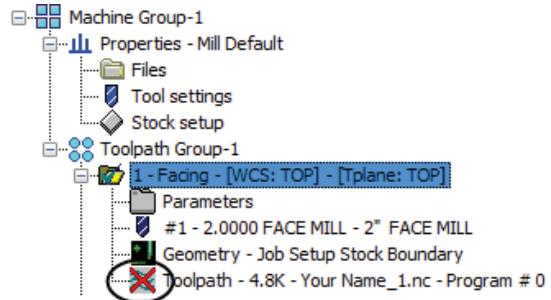
CIRCLE MILL THE LARGE HOLE



NOTE: If you exit the toolpath in the middle of setting the parameters, in the Toolpaths Manager you will have a red X on the Face Toolpath as shown in [Figure: 9.2.1](#). This shows that you modified the toolpath and you need to update it. You will have to select the **Regenerate all dirty operations** icon each time you change something in the toolpath parameters.



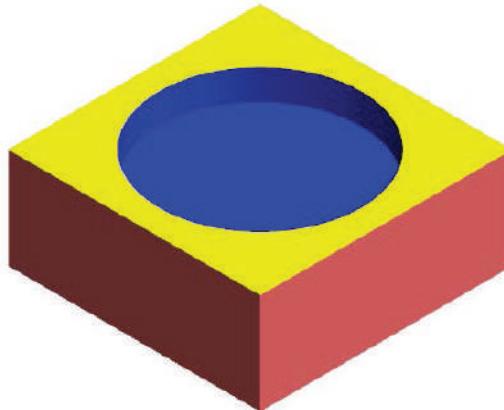
Figure: 9.2.1

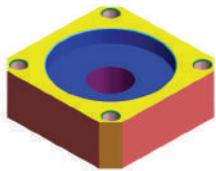


STEP 10: CIRCLE MILL THE LARGE HOLE

The **Circle Mill Toolpath** removes circular pockets based on a single point. You can select either point entities, or the center points of arcs. Mastercam will then pocket out a circular area of the diameter and to the depth that you specify.

Toolpath Preview:



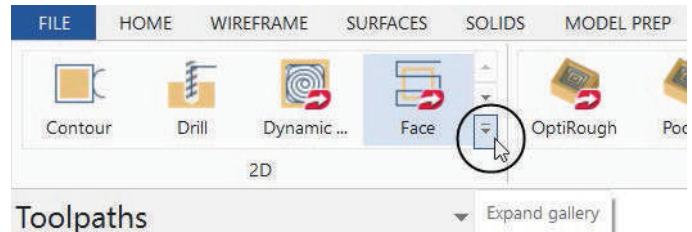


10.1 Drill Point Selection

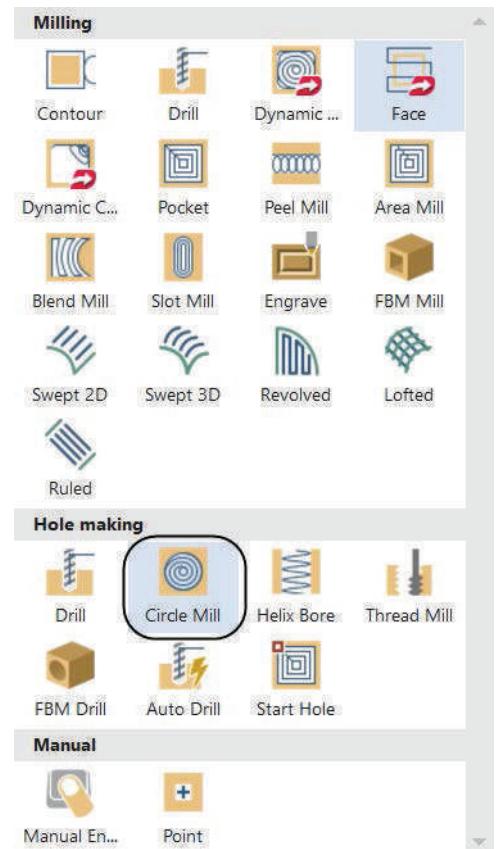
- ◆ Press **Alt + T** to remove the toolpath display.

TOOLPATHS

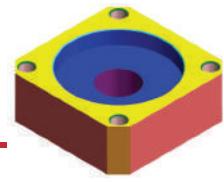
- ◆ From the **2D** group, click on the **Expand gallery** button as shown.



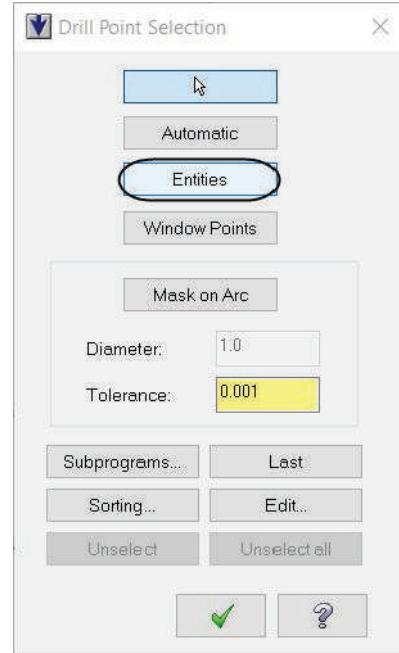
- ◆ Select **Circle Mill**.



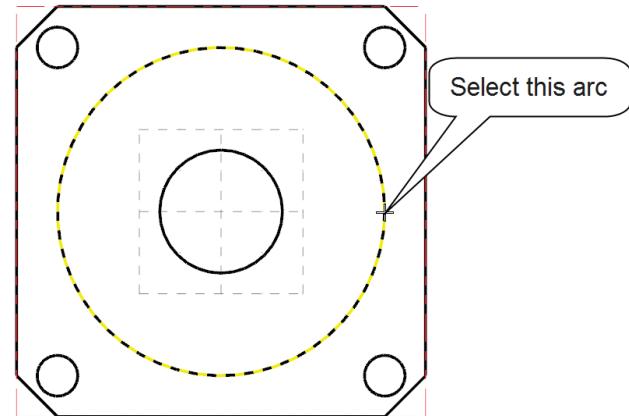
TUTORIAL #1 CIRCLE MILL THE LARGE HOLE



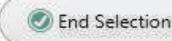
- From the Drill Point Selection, click on Entities.



- Select the 2.0" diameter circle as shown.



- Click on the End Selection button or press Enter to finish the selection.

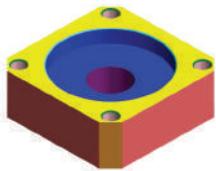


- Select the OK button to exit Drill Point Selection.



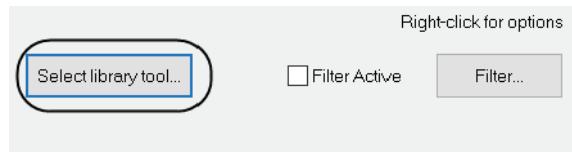
- In the Toolpath Type page, the Circle Mill icon will be selected.





10.2 Select a 3/4" Flat Endmill from the library and set the Tool parameters

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

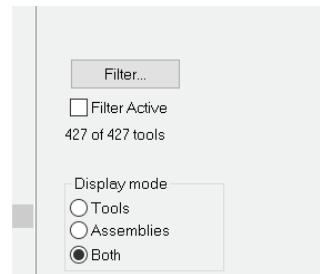


- ♦ To be able to see all the tools from the library, disable **Filter Active**.



- ♦ Scroll down and select the **3/4" Flat Endmill (#294)** as shown.

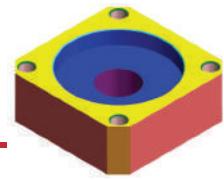
289	-	7/16 FLAT ENDMILL	-	0.4375	0.0	0.8	4	End...	None
290	-	1/2 FLAT ENDMILL	-	0.5	0.0	1.0	4	End...	None
291	-	17/32 FLAT ENDMILL	-	0.5312	0.0	1.0	4	End...	None
292	-	5/8 FLAT ENDMILL	-	0.625	0.0	1.5	4	End...	None
293	-	23/32 FLAT ENDMILL	-	0.71...	0.0	1.5	4	End...	None
294	-	3/4 FLAT ENDMILL	-	0.75	0.0	2.0	4	End...	None
295	-	13/16 FLAT ENDMILL	-	0.8125	0.0	2.0	4	End...	None
296	-	7/8 FLAT ENDMILL	-	0.875	0.0	2.0	4	End...	None
297	-	1 INCH FLAT ENDMILL	-	1.0	0.0	2.0	4	End...	None
298	-	1-3/16 FLAT ENDMILL	-	1.1875	0.0	2.0	4	End...	None
299	-	1-1/2 FLAT ENDMILL	-	1.5	0.0	2.5	4	End...	None
300	-	2 INCH FLAT ENDMILL	-	2.0	0.0	2.75	4	End...	None



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



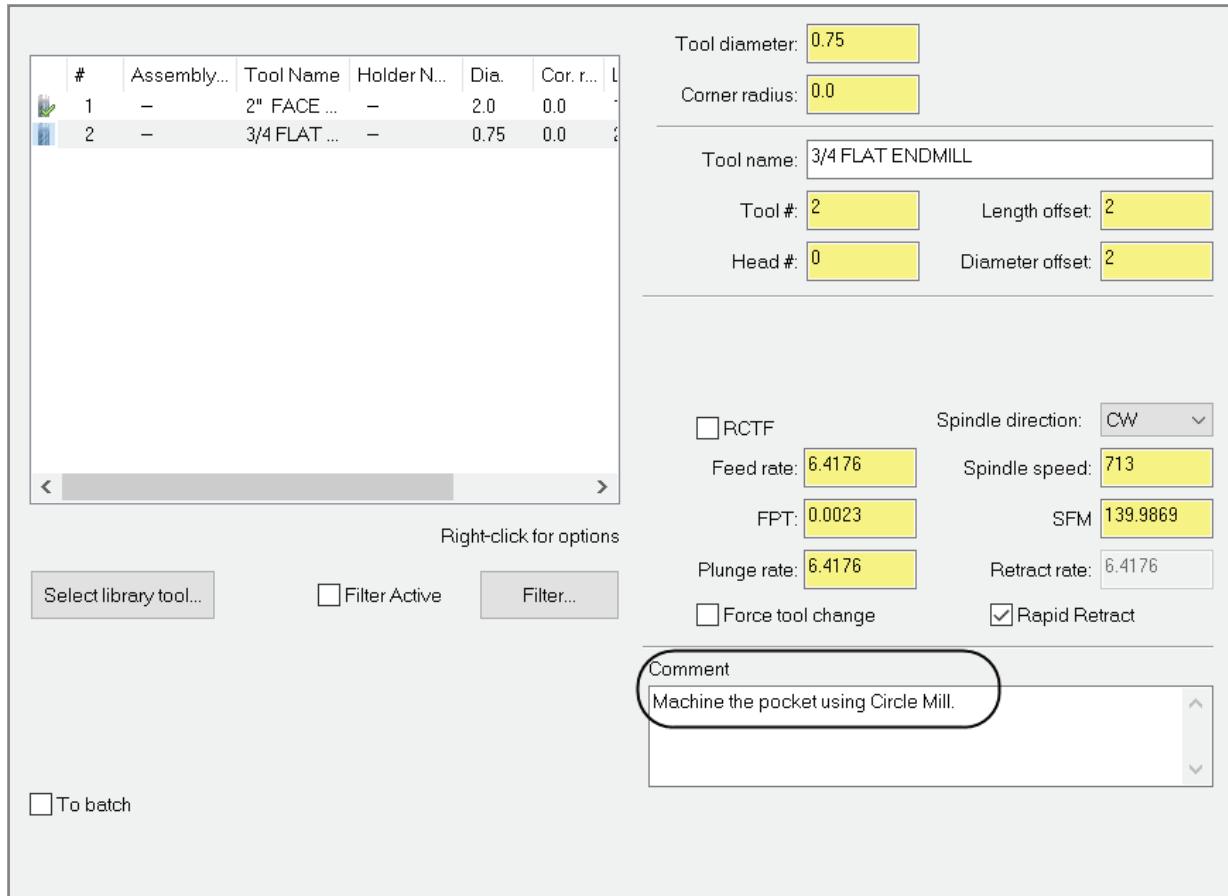
TUTORIAL #1 CIRCLE MILL THE LARGE HOLE

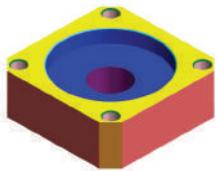


- Make all the necessary changes as shown in [Figure: 10.2.1](#).

NOTE: The **Feed rate**, **Plunge rate**, **Retract rate**, and **Spindle speed** are based on the tool definition as set in the **Tool Settings**. You may change these values as per your part material and tools.

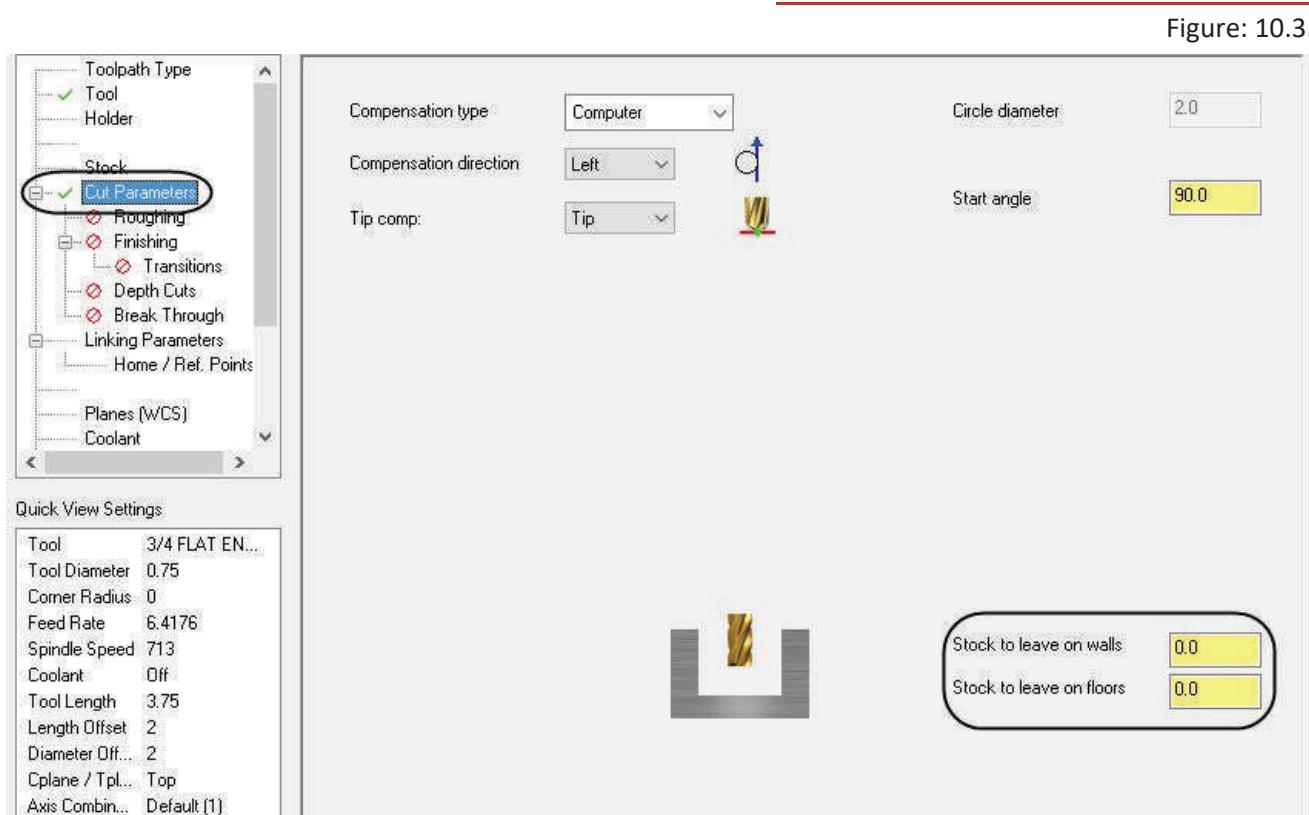
Figure: 10.2.1

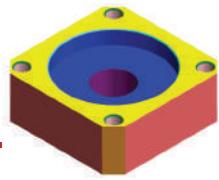




10.3 Cut Parameters

- From the Tree View list, select **Cut Parameters** and ensure the settings appear as shown in [Figure: 10.3.1](#).

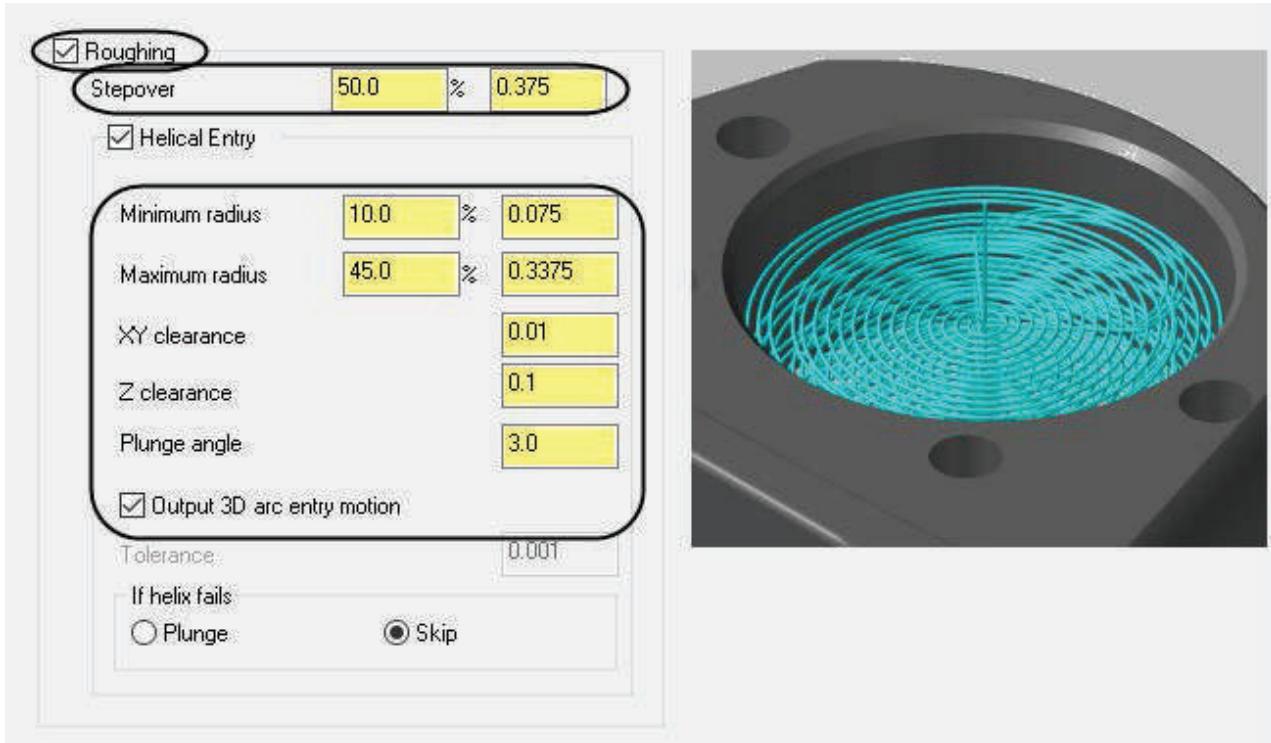




10.4 Roughing

- From the **Tree View** list, select **Roughing** and enable it. Set the **Stepover** to 50% and enable **Helical Entry** and set the parameters as shown in [Figure: 10.4.1](#).

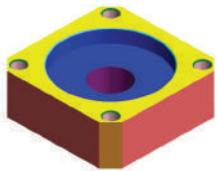
Figure: 10.4.1



Stepover sets the distance between cutting passes in the X and Y axes as a percentage of the tool diameter.

Helical entry creates a helix at the center of the circle to begin the roughing motion. If off, the tool plunges to start the toolpath.

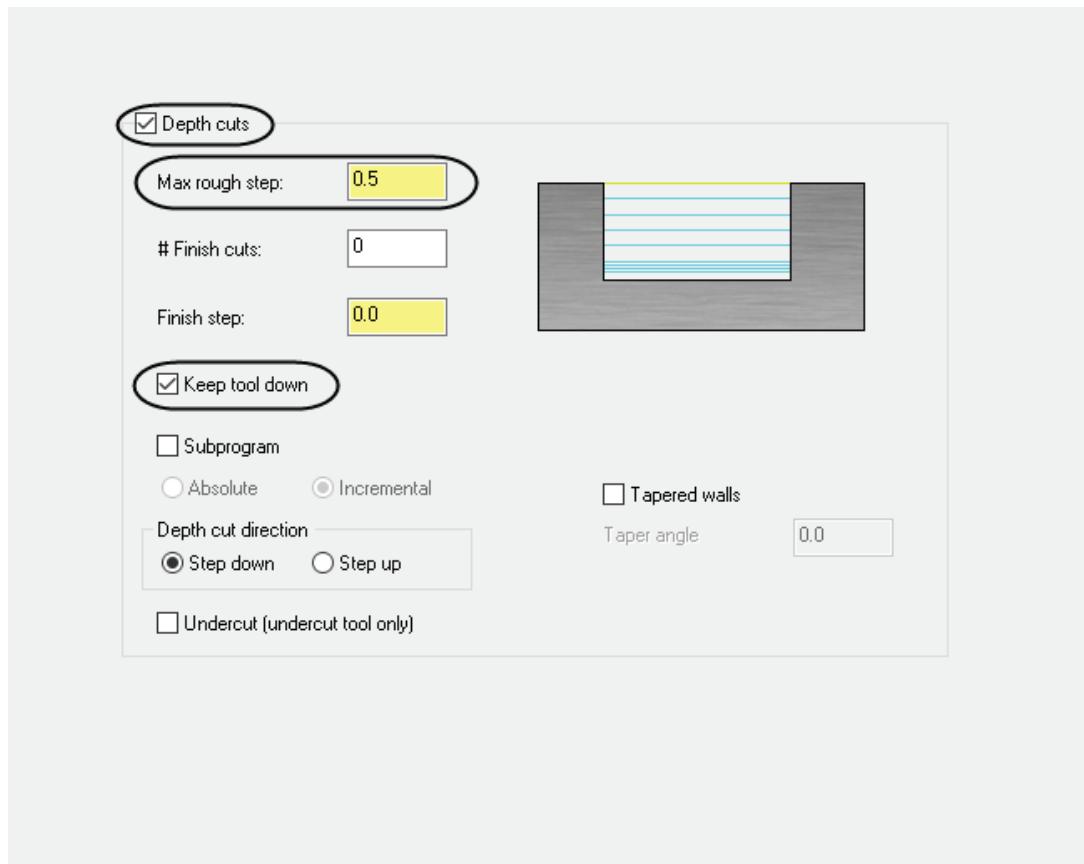
NOTE: The images in the toolpaths change depending on the parameter last selected in the page.



10.5 Depth Cuts

- From the Tree View list, select **Depth Cuts**. On the **Depth Cuts** page, enable **Depth cuts** and set the **Max rough step** to **0.5** and enable **Keep tool down**.
- Make any necessary changes as shown in [Figure: 10.5.1](#).

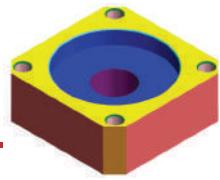
Figure: 10.5.1



Depth Cuts: Mastercam will take the total depth and divide it into separate depth cuts. Mastercam never performs unequal depth cuts.

Max rough step sets the maximum amount of material removed in the Z axis with each rough cut. Mastercam will calculate equal rough cuts no larger than the maximum rough step until it reaches the final Z depth.

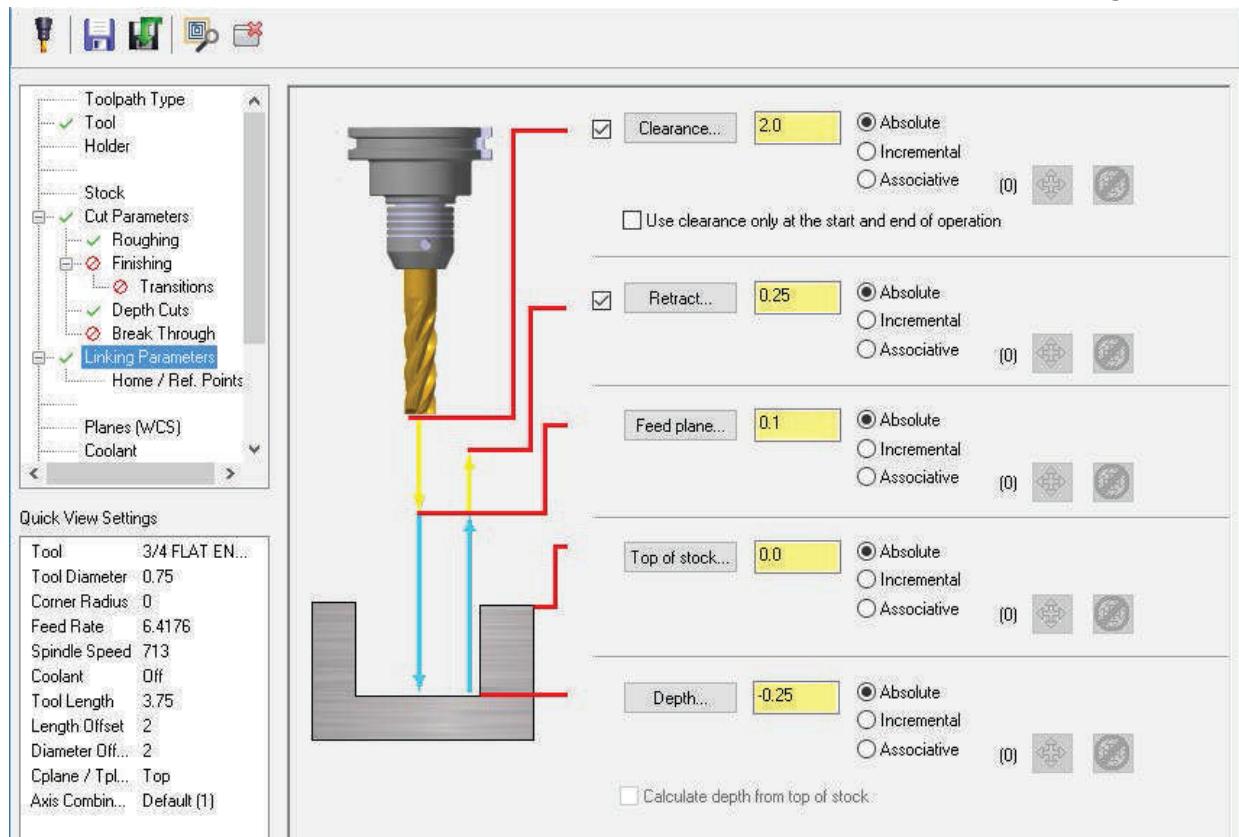
Keep tool down determines whether or not to retract the tool between depth cuts.



10.6 Linking Parameters

- ♦ Select **Linking Parameters** from the **Tree View** list.
- ♦ Change the top of stock to **0.0** and set the depth to **-0.25**. Ensure all the values are set to **Absolute** as shown in [Figure: 10.6.1](#).

Figure: 10.6.1



Absolute values are always measured from the origin 0,0,0.

Incremental values are relative to other parameters or chained geometry.

Associative option allows you to select points from the existing geometry from where the values will be measured.

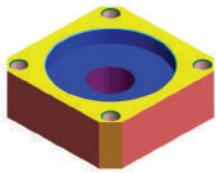
10.7 Preview the Toolpath

- ♦ To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

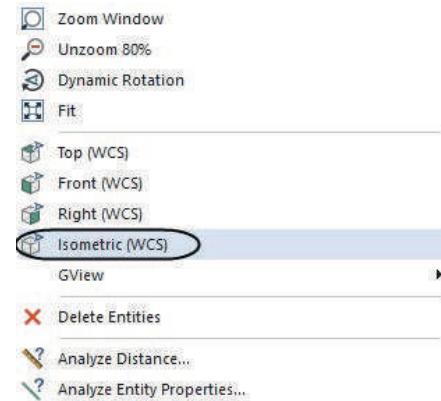


- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.

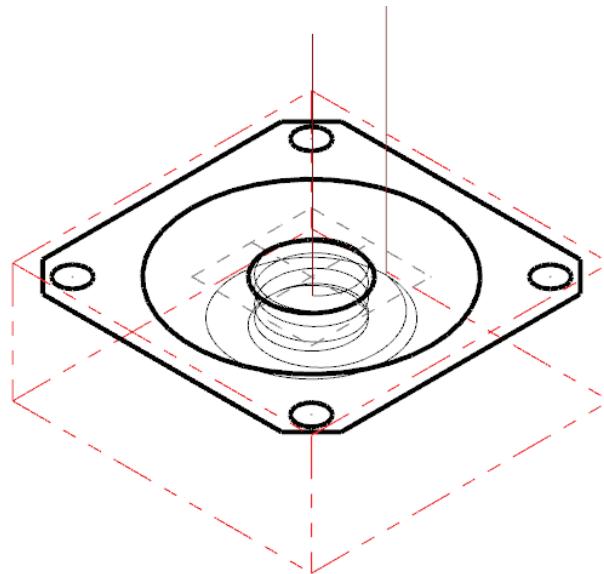




- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.

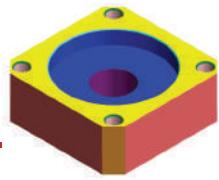


- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to exit the **2D Toolpath - Circle Mill** parameters.





STEP 11: 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 additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates, and the cycle time. It also shows any collisions between the workpiece and the tool.

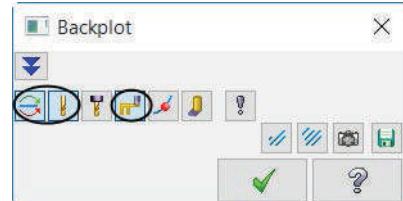
- ♦ Make sure that the toolpaths are selected (signified by the green check mark on the folder icon). If both operations are not selected, choose the **Select all operations** icon.



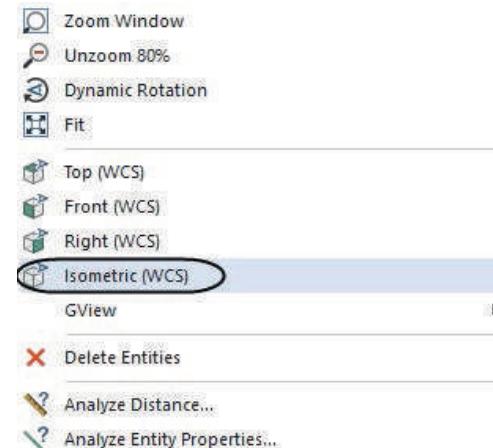
- ♦ Select the **Backplot selected operations** button.



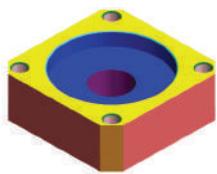
- ♦ In the **Backplot** panel, enable **Display with color codes**, **Display tool** and **Display rapid moves** icons as shown.



- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ To fit the workpiece to the screen, if needed, right mouse click in the graphics window again and select **Fit**.
- ♦ You can step through the **Backplot** by using the **Step forward** or **Step back** buttons.



- ◆ You can adjust the speed of the backplot.

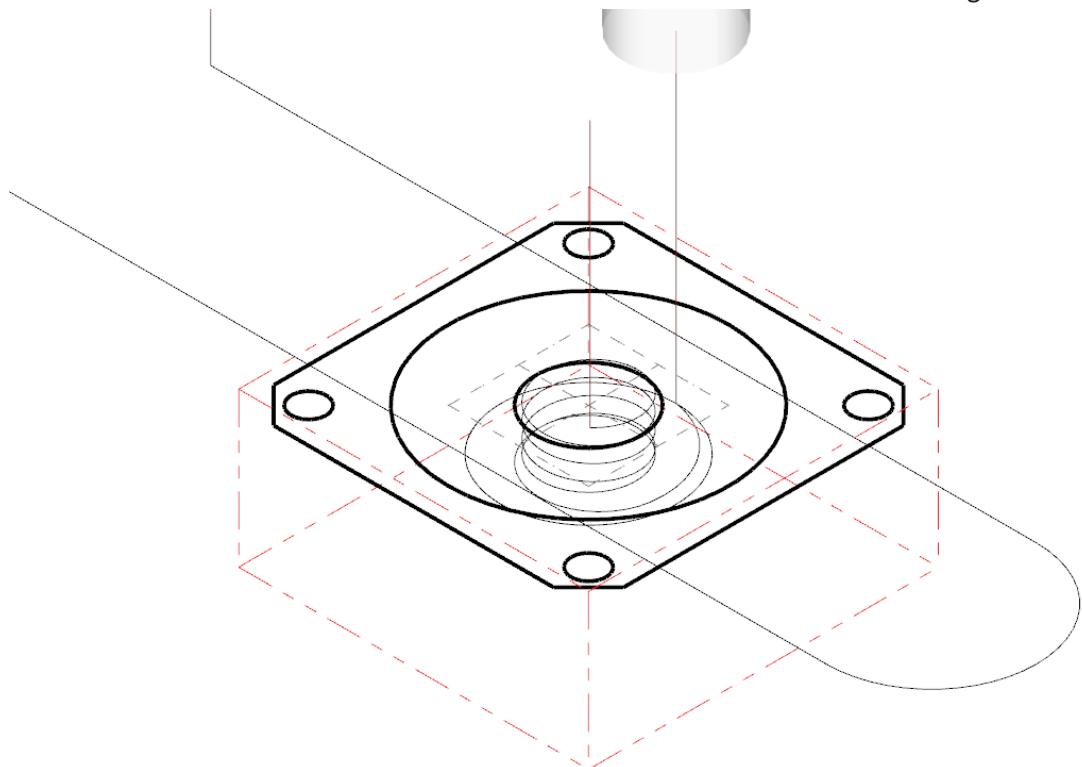


- ◆ Select the **Play** button to run Backplot.



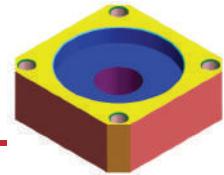
- ◆ The toolpath should look as shown in [Figure: 11.0.1](#).

Figure: 11.0.1



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





STEP 12: SIMULATE THE TOOLPATH IN VERIFY

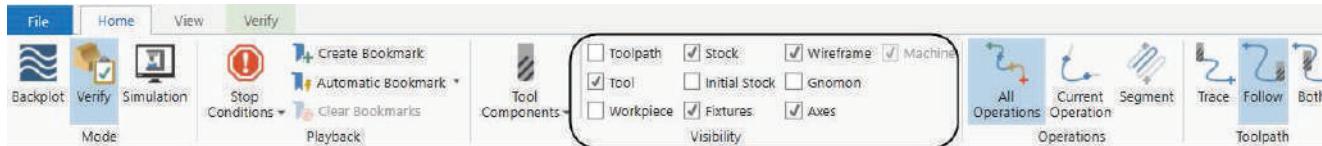
Verify Mode shows the path the tools take to cut the part with material removal. This display lets you spot errors in the program before you machine the part. As you verify toolpaths, Mastercam displays additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates, and the cycle time. It also shows any collisions between the workpiece and the tool.

- From the **Toolpaths Manager**, select the **Verify selected operations** icon as shown.



NOTE: Mastercam launches a new window that allows you to check the part using **Verify**.

- In **Mastercam Simulator**, change the settings as shown.

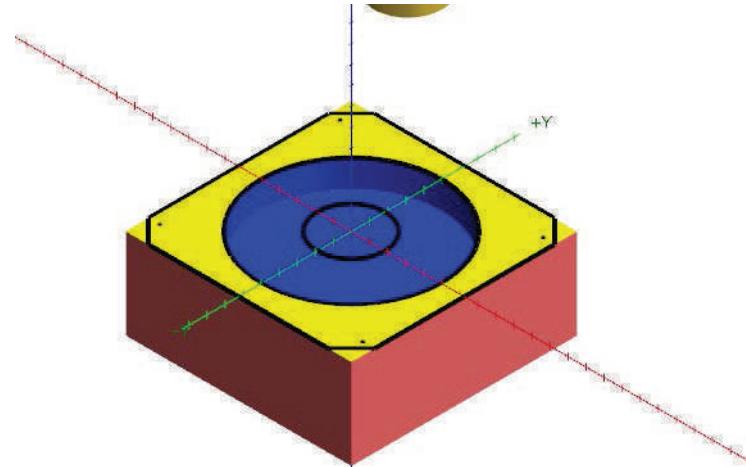


- Select the **Play** button to run **Verify**.



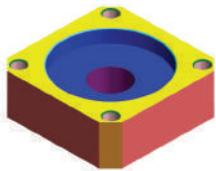
- The part should appear as shown in [Figure: 12.0.1](#).

Figure: 12.0.1



NOTE: To rotate the part, move the cursor to the center of the part and click and hold the mouse wheel and slowly move it in one direction.

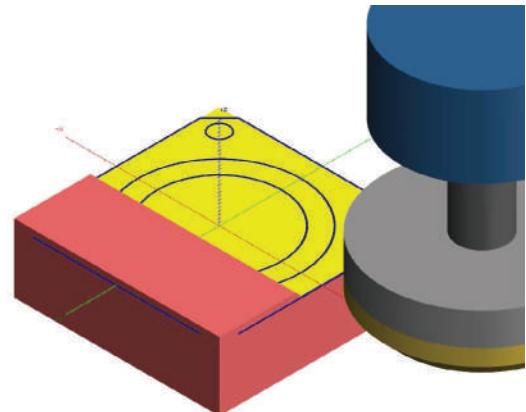
To **Zoom In** or **Out**, hold down the mouse wheel and scroll up or down as needed.



- ♦ Right mouse click in the graphics window and select **Isometric** and then right mouse click again and select **Fit** to see the part in the original position.



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



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



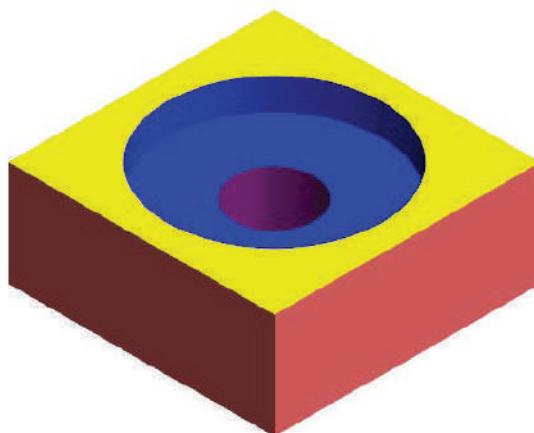
- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.
- ♦ Click in the **Toolpaths Manager** and press **Alt + T** until you remove the toolpath display.



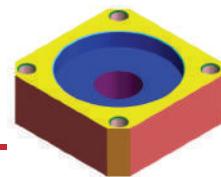
STEP 13: CIRCLE MILL THE INSIDE HOLE

Circle Mill Toolpaths remove circular pockets based on a single point. You can select either point entities or center points of arcs. Mastercam will then pocket out a circular area of the diameter to the depth that you specify.

Toolpath Preview:



TUTORIAL #1 CIRCLE MILL THE INSIDE HOLE



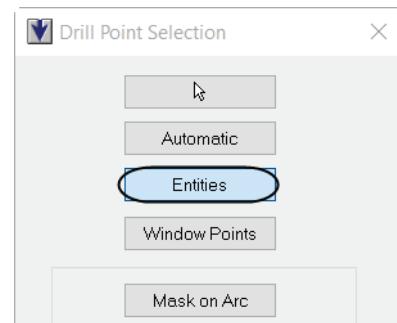
13.1 Drill Point Selection

TOOLPATHS

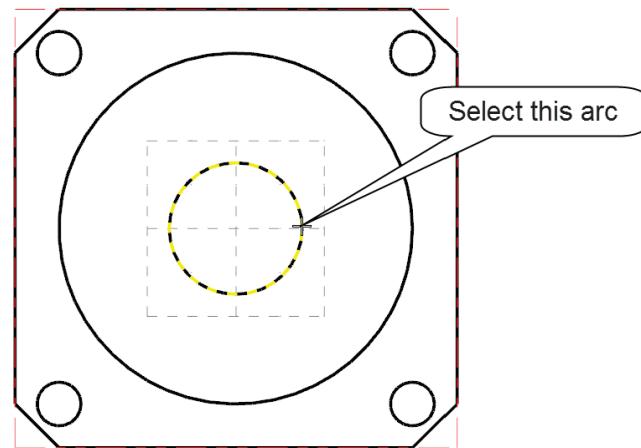
- From the **2D** group, click on the **Circle Mill** icon.



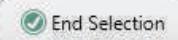
- From the **Drill Point Selection**, click on **Entities**.



- Select the **0.75"** diameter circle as shown.



- Click on the **End Selection** button or press **Enter** to finish the selection.

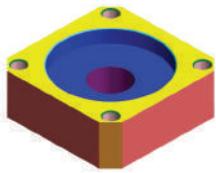


- Select the **OK** button to exit **Drill Point Selection**.



- In the **Toolpath Type** page, the **Circle Mill** icon will be selected.





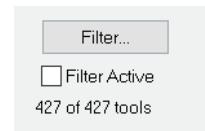
13.2 Select a 1/4" Flat Endmill from the library and set the Tool parameters

- ♦ Select **Tool** from the **Tree View** list.

- ♦ Click on **Select library tool** button.

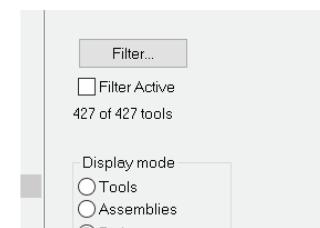
Select library tool...

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



- ♦ Scroll down and select the **1/4" Flat Endmill (#285)** as shown.

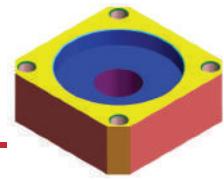
278	-	1 INCH COUNTERSINK ...	-	1.0	0.0	2.0	2	CSink	None
279	-	1/32 FLAT ENDMILL	-	0.03...	0.0	0.375	4	End...	None
280	-	1/16 FLAT ENDMILL	-	0.0625	0.0	0.375	4	End...	None
281	-	3/32 FLAT ENDMILL	-	0.09...	0.0	0.375	4	End...	None
282	-	1/8 FLAT ENDMILL	-	0.125	0.0	0.375	4	End...	None
283	-	5/32 FLAT ENDMILL	-	0.15...	0.0	0.375	4	End...	None
284	-	3/16 FLAT ENDMILL	-	0.1875	0.0	0.4375	4	End...	None
285	-	1/4 FLAT ENDMILL	-	0.25	0.0	0.5	4	End...	None
286	-	5/16 FLAT ENDMILL	-	0.3125	0.0	0.75	4	End...	None
287	-	3/8 FLAT ENDMILL	-	0.375	0.0	0.75	4	End...	None



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

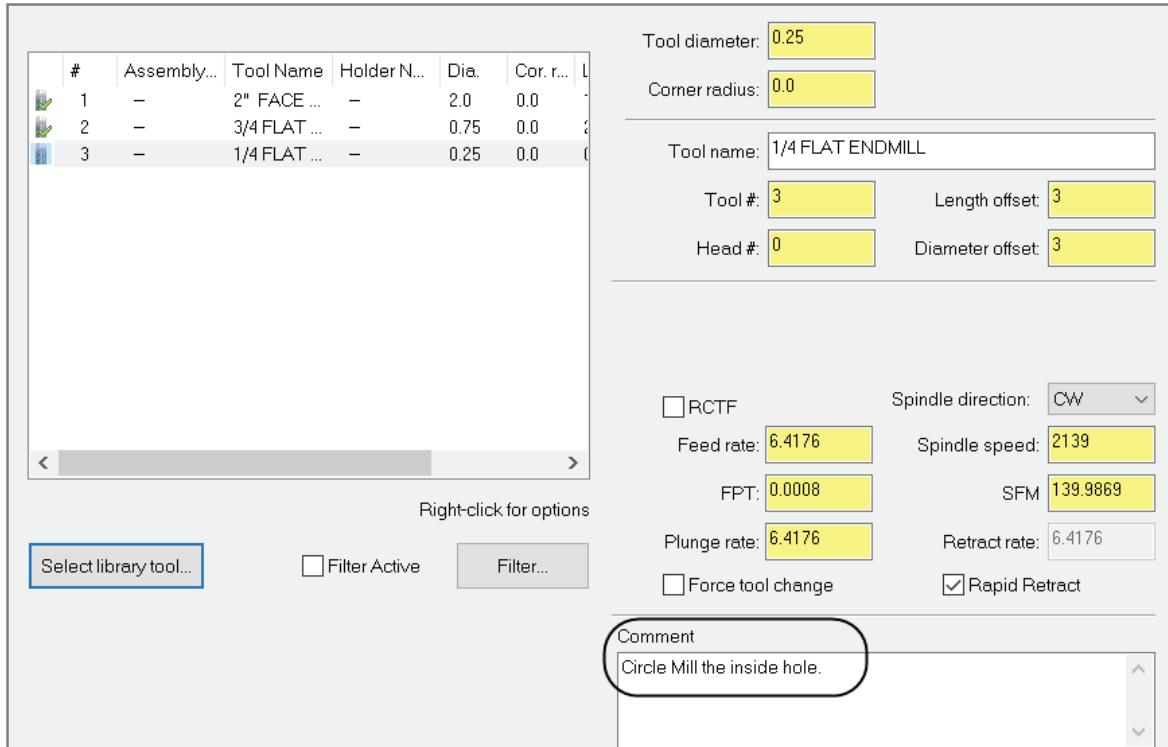


TUTORIAL #1 CIRCLE MILL THE INSIDE HOLE



- Input a comment and make all the necessary changes, as shown in [Figure: 13.2.1](#).

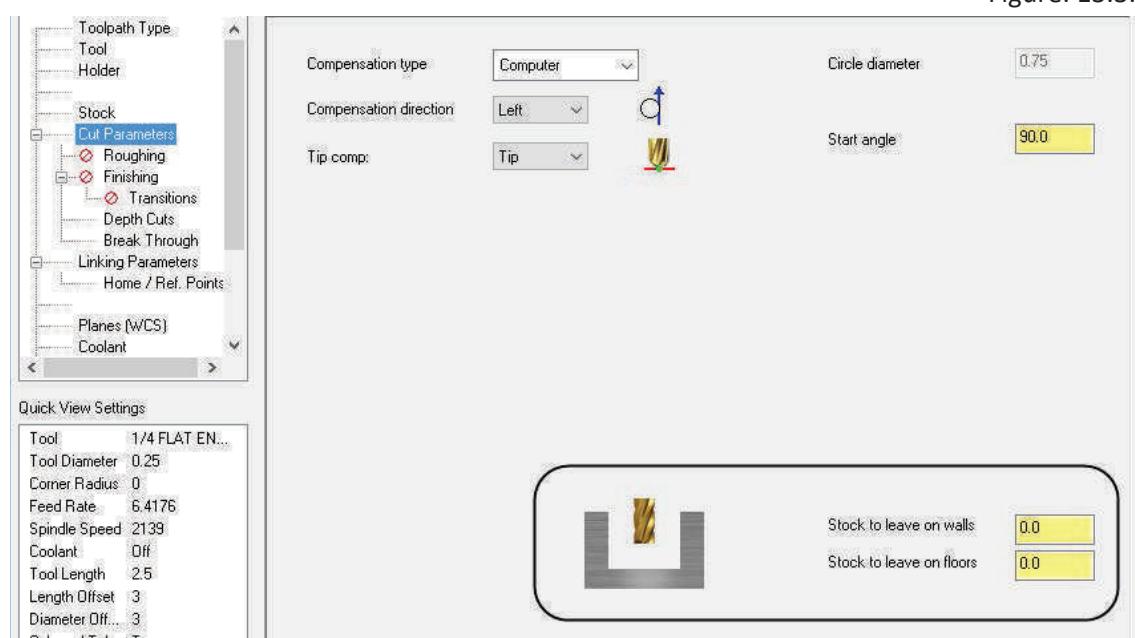
Figure: 13.2.1

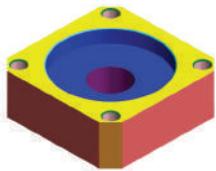


13.3 Cut Parameters

- From the Tree View list, select **Cut Parameters** and ensure the settings appear as shown in [Figure: 13.3.1](#).

Figure: 13.3.1

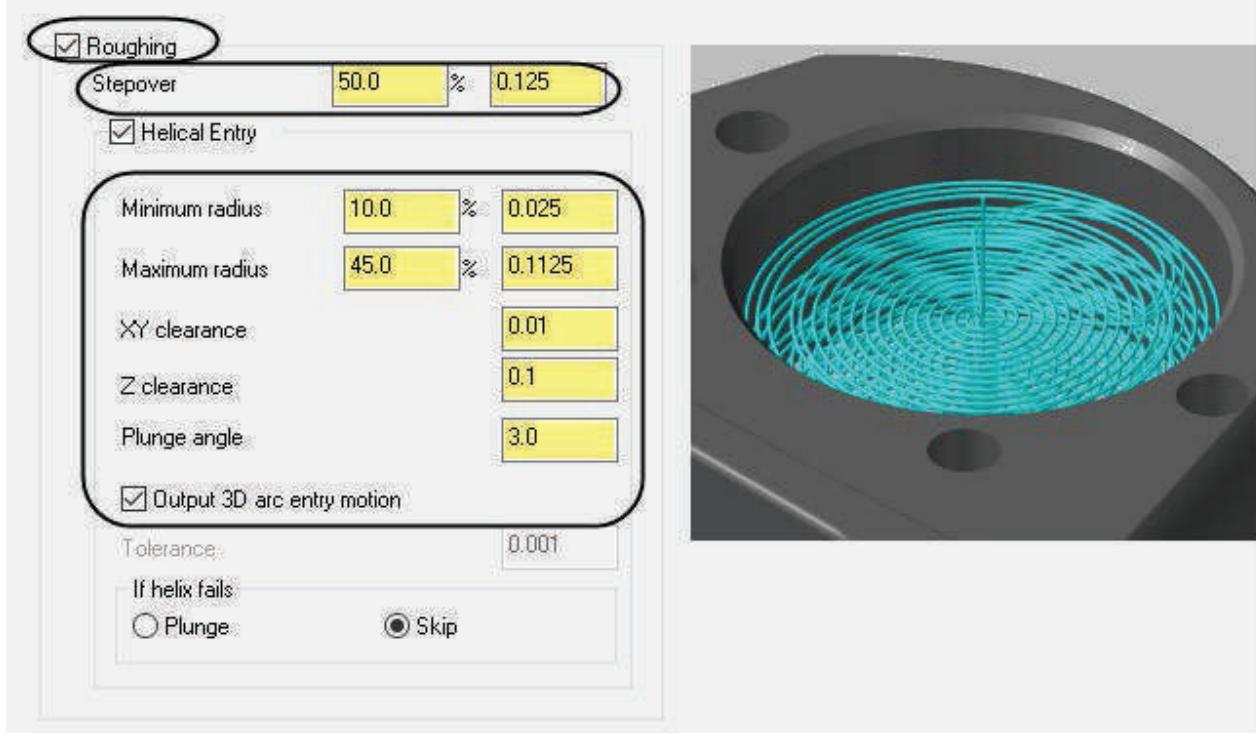




13.4 Roughing

- From the **Tree View** list, select **Roughing** and enable it. Set the **Stepover** to 50%, enable **Helical Entry**, and specify the other parameters as shown in [Figure: 13.4.1](#).

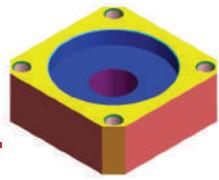
Figure: 13.4.1



Stepover sets the distance between cutting passes in the X and Y axes as a percentage of the tool diameter.

Helical Entry creates a helix at the center of the circle to begin the roughing motion. If this option is turned off, the tool plunges to start the toolpath.

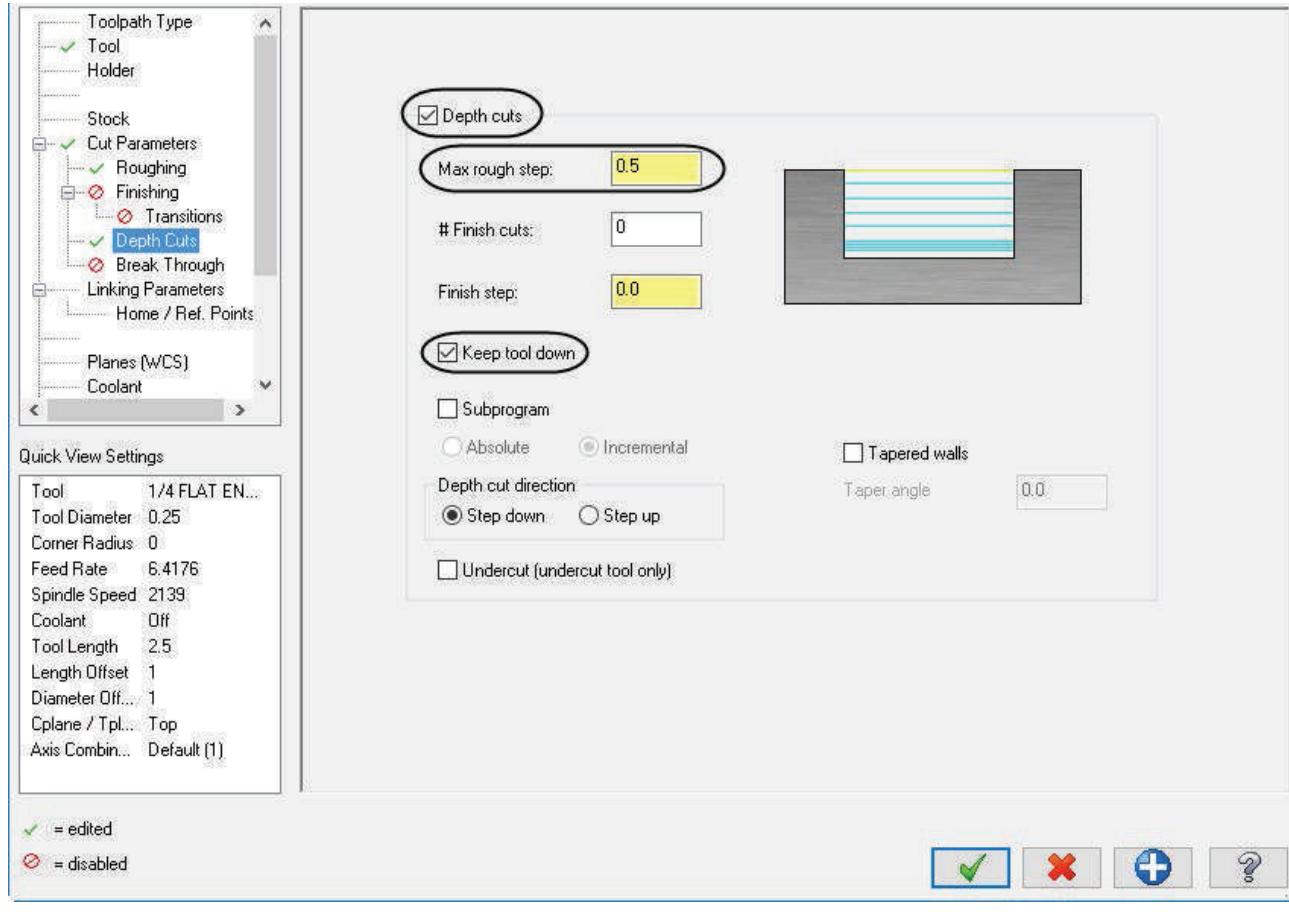
NOTE: The images in the toolpaths change depending on the parameter that you last selected in the page.



13.5 Depth Cuts

- ♦ Make any necessary change as shown in [Figure: 13.5.1](#).

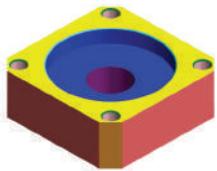
Figure: 13.5.1



Depth Cuts sets the steps the tool takes along the **Z axis**. Mastercam will take the total depth and divide it into separate depth cuts. Mastercam never performs unequal depth cuts.

Max rough step sets the maximum amount of material removed in the Z axis with each rough cut. Mastercam will calculate equal rough cuts no larger than the maximum rough step until it reaches the final Z depth.

Keep tool down determines whether or not to retract the tool between depth cuts.



13.6 Set the Break Through

- From the **Tree View** list, select **Break Through** and set the parameters to completely cut through the material by an amount that you specify as shown in [Figure: 13.6.1](#).

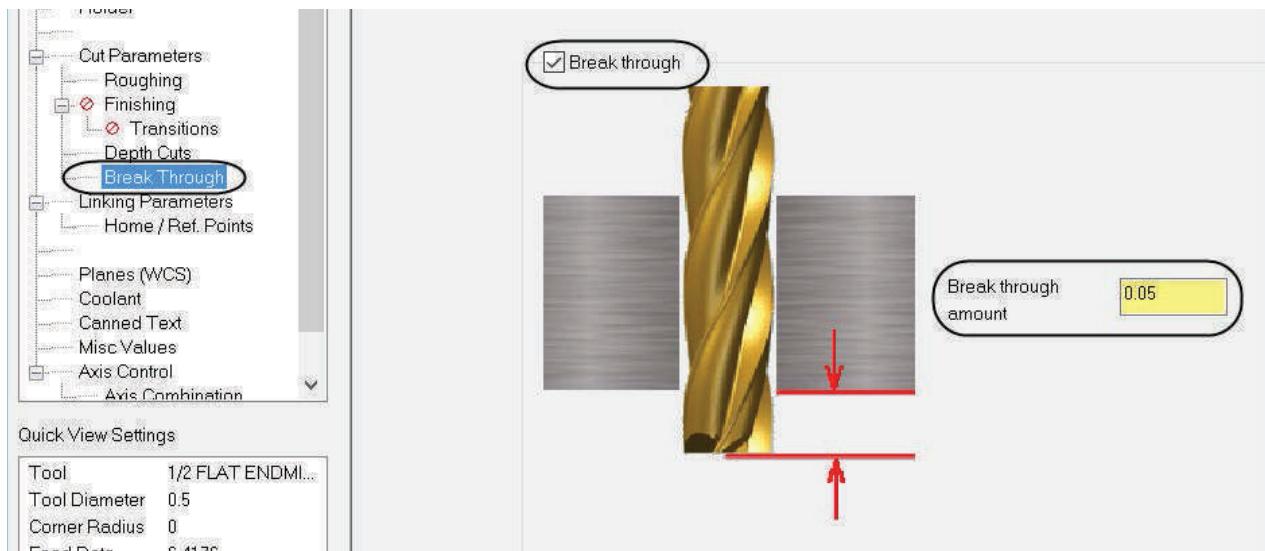
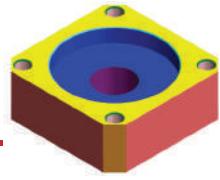


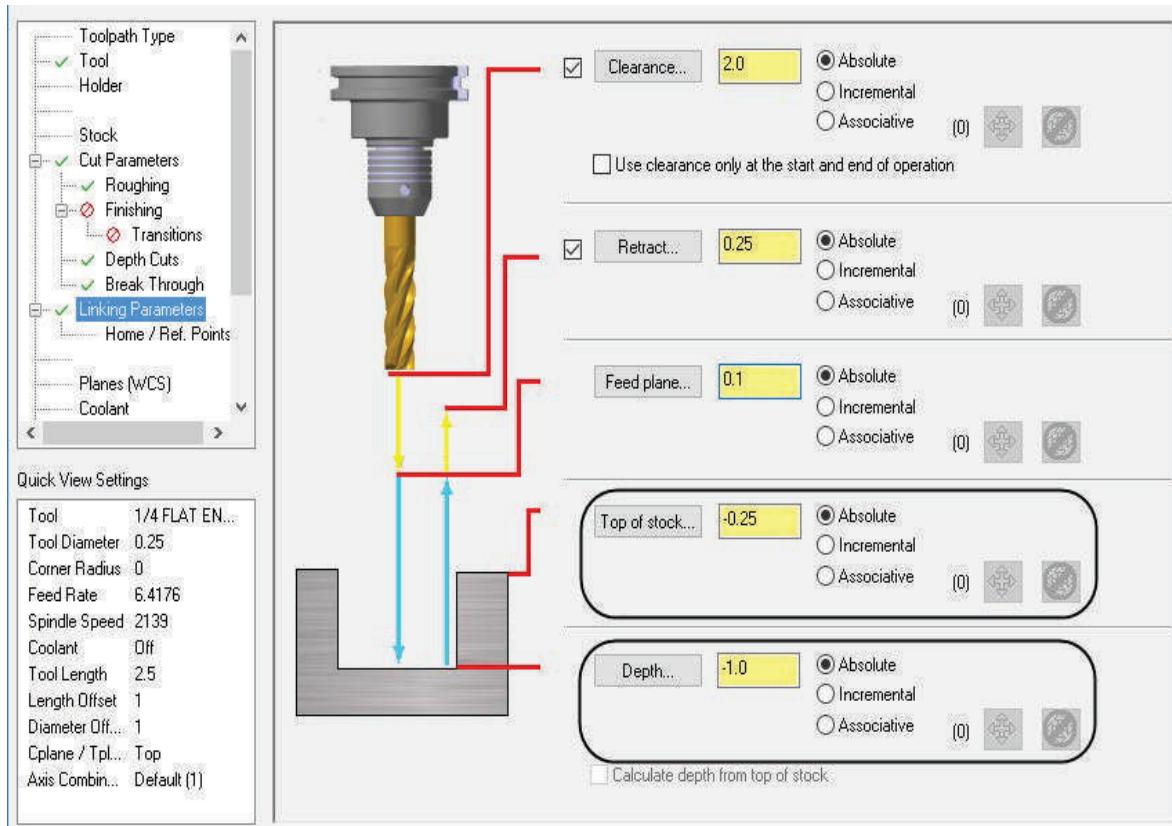
Figure: 13.6.1



13.7 Linking Parameters

- ♦ Select **Linking Parameters** from the **Tree View** list.
- ♦ Change the **Top of stock** to **-0.25** and set the **Depth** to **-1.0**. Ensure all the values are set the same as shown in [Figure: 13.7.1](#).

Figure: 13.7.1



Absolute values are always measured from the origin 0,0,0.

Incremental values are relative to other parameters or chained geometry.

Associative option allows you to select points from the existing geometry from where the values will be measured.

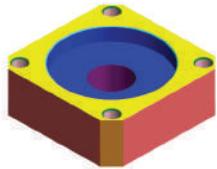
13.8 Preview the Toolpath

- ♦ To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

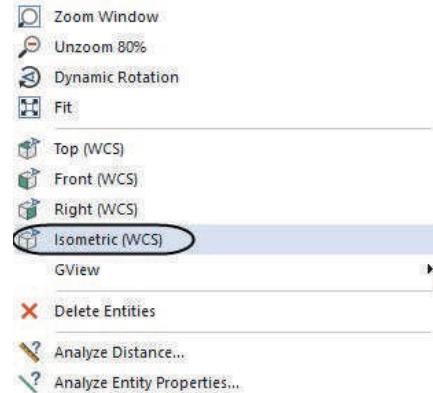


- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.

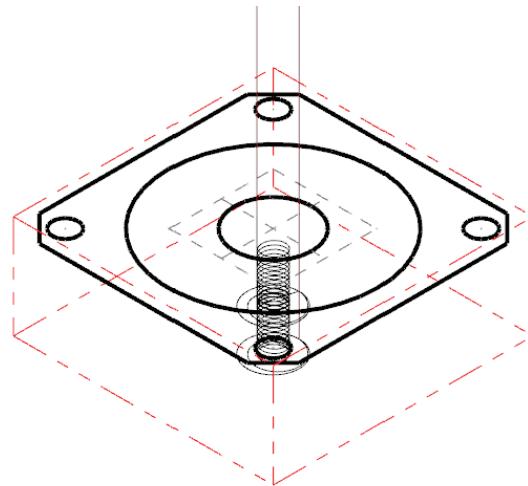




- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.

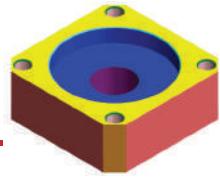


- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

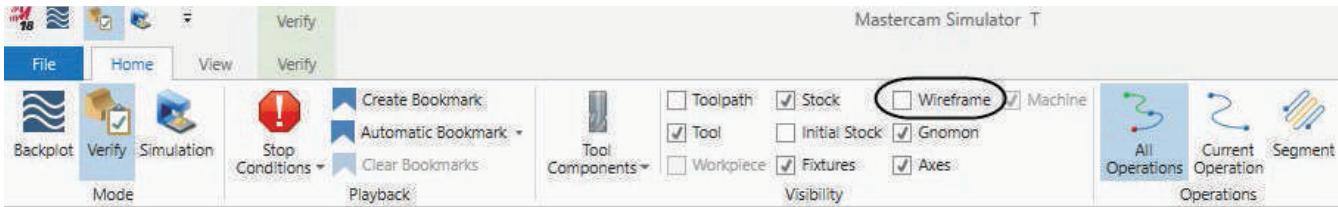
- ♦ Select the **OK** button to exit the **Circle Mill** parameters.





13.9 Verify the toolpaths

- From the **Toolpaths Manager**, click on the **Select all operations** icon.
- Click on the **Verify selected operations** icon.
- For information on how to set the verify parameters and to simulate the toolpath, please check [page 58](#).
- Disable **Wireframe** as shown.

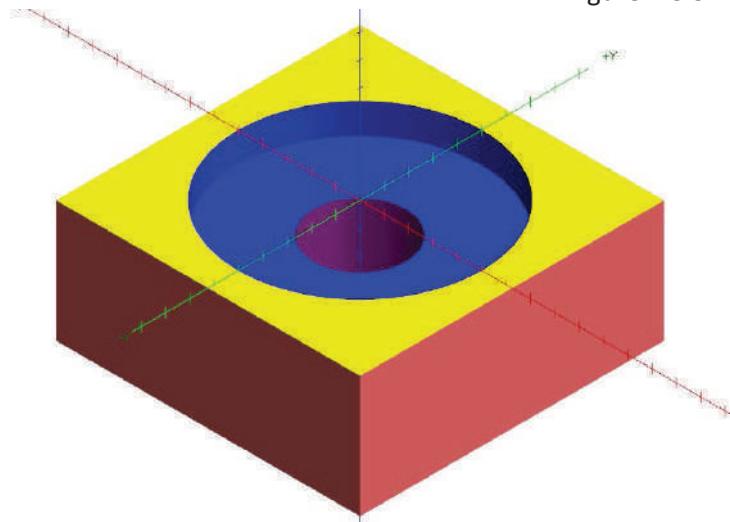


- Select the **Verify** tab, and enable **Color Loop** as shown.

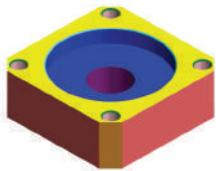


NOTE: This option will change the material removal color. This can be set based on the operation or on the tool number used to machine the part. This makes it easier to spot if you forget to leave the stock in the finish operations.

- Select the **Play** button as shown.
- The part will appear as shown in [Figure: 13.9.1](#).



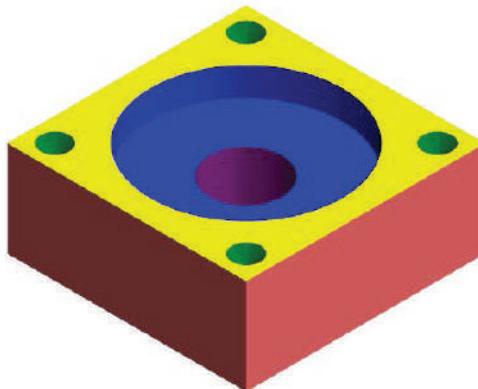
- To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.



STEP 14: SPOT DRILL THE 0.25" HOLES

Spot Drilling the holes allows you to start the hole. In this operation, we will use the spot drill to chamfer the hole before drilling it.

Toolpath Preview:



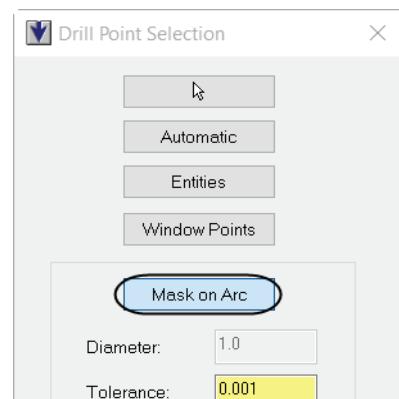
- ♦ Select all toolpaths and press **Alt + T** to remove the toolpath display if needed.

TOOLPATHS

- ♦ In the **2D** group, select the **Drill** icon as shown.

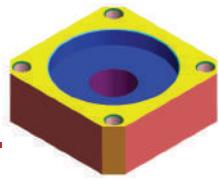


- ♦ In the **Drill Point Selection** panel, choose the option **Mask on Arc**.



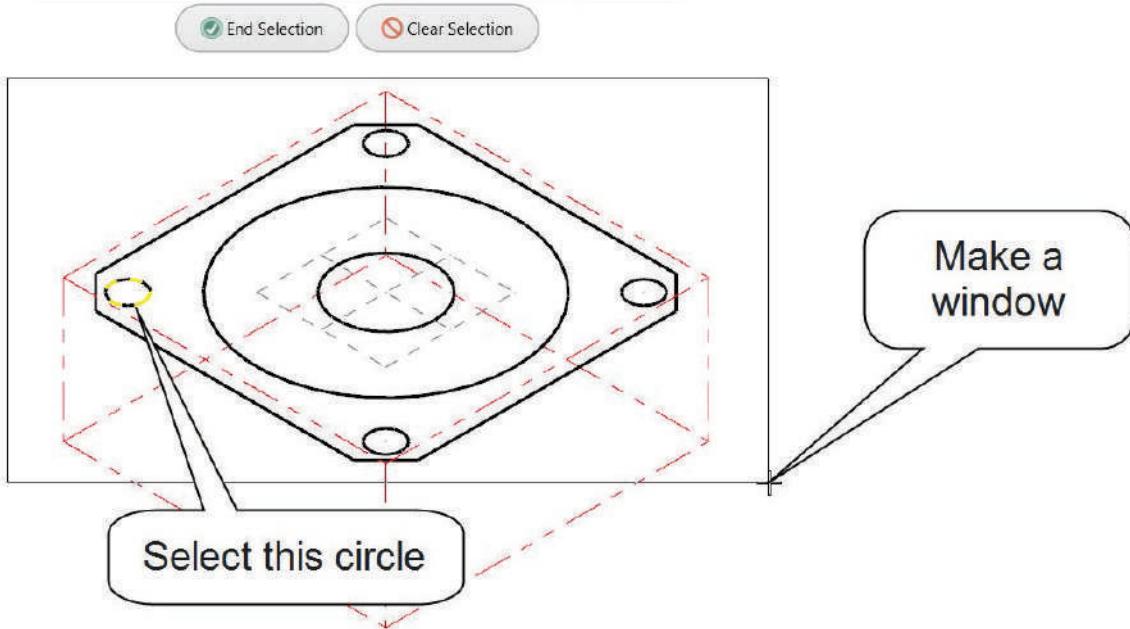
NOTE: Mask on Arc is a tool for selecting arcs whose diameters match the one that you select within a specified tolerance.

TUTORIAL #1 SPOT DRILL THE 0.25" HOLES



- ♦ Hover the cursor above the center of the geometry and scroll down the mouse wheel to unzoom the geometry as shown in [Figure: 14.0.1](#).
- ♦ Select one of the four arcs as shown in [Figure: 14.0.1](#).
- ♦ Left click in the upper left corner of the graphics window, hold the left button down and drag a rectangle to the lower right corner of the part to include all entities, as shown in [Figure: 14.0.1](#).

Figure: 14.0.1

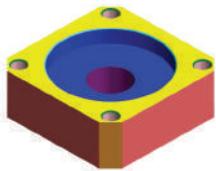


- ♦ Release the left mouse button and click it again once you have created a window encompassing the entire part.

NOTE: All the arcs inside of the window will be selected. Once you hit the **Enter** key or **End Selection** button, only the circles with the same diameter as the original selected circle will be selected.

- ♦ Click on the **End Selection** button or press **Enter** to finish the selection.
- ♦ Select the **OK** button in the **Drill Point Selection** panel to accept the 4 drill points.
- ♦ In the **Toolpath Type** page, the **Drill** toolpath should already be selected.





14.1 Select a 3/4" Spot Drill from the library and set the Tool Parameters

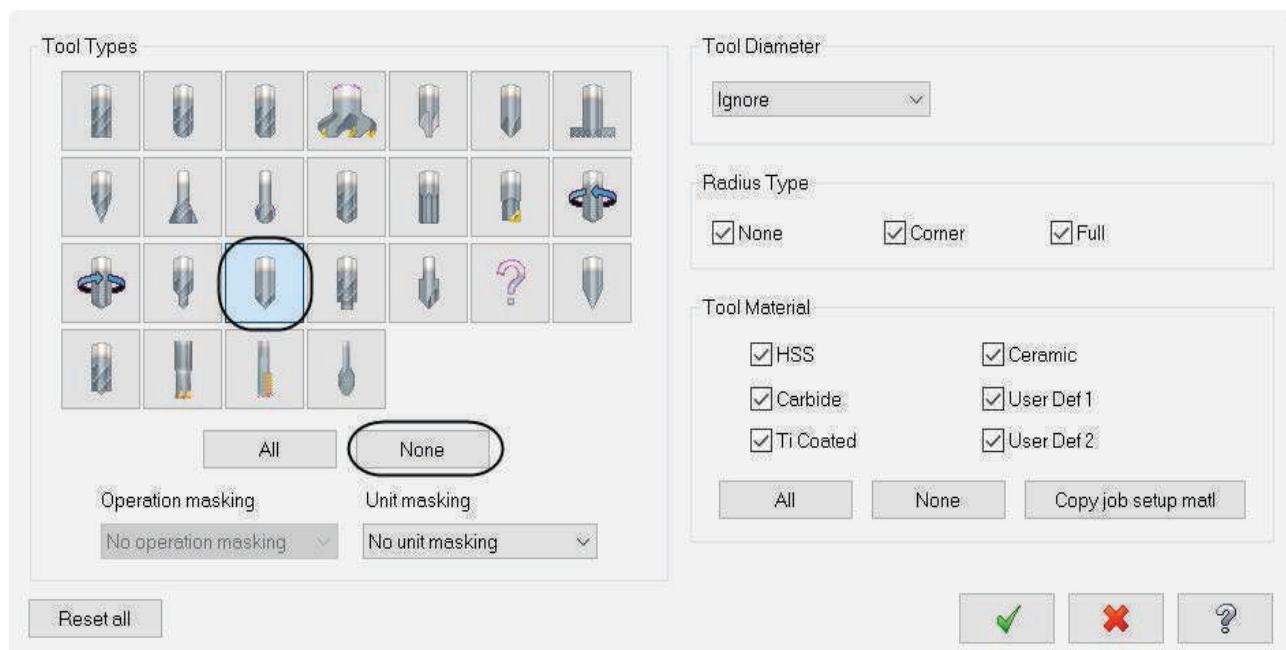
- ♦ Select **Tool** from the **Tree View** list.

- ♦ Click on the **Select library tool...** button.
- ♦ To view only the spot drill, select the **Filter** button.



- ♦ Under **Tool Types**, select the **None** button to unselect any unwanted tool.
- ♦ Hover the cursor over each icon and the tool type will be displayed; choose the **Spot Drill** icon as shown in [Figure: 14.1.1](#).

Figure: 14.1.1

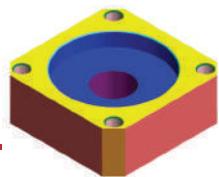


- ♦ Select the **OK** button to exit the **Tool List Filter** dialog box.
- ♦ At this point you should only see **Spot Drills**.
- ♦ From that list select the **3/4" Spot Drill** as shown.

#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	Length	# Flut...	Type	Rad...
21	-	1/8 SPOT...	-	0.125	0.0	2.0	2	Spot...	None
22	-	1/4 SPOT...	-	0.25	0.0	2.0	2	Spot...	None
23	-	3/8 SPOT...	-	0.375	0.0	2.0	4	Spot...	None
24	-	1/2 SPOT...	-	0.5	0.0	2.0	2	Spot...	None
25	-	3/4 SPOT...	-	0.75	0.0	2.0	4	Spot...	None
26	-	1. SPOT...	-	1.0	0.0	2.0	4	Spot...	None

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

TUTORIAL #1 SPOT DRILL THE 0.25" HOLES



- Input a comment and make the necessary changes to the Tool page as shown in Figure: 14.1.2.

Figure: 14.1.2

2D Toolpaths - Drill/Circles Simple drill - no peck

Toolpath Type: Tool

Holder:

Stock:

Cut Parameters:

Linking Parameters:

Tip Comp:

Home / Ref. Points:

Planes (WCS):

Coolant:

Canned Text:

Misc Values:

Axis Control:

Axis Combination:

Rotary Axis Control:

Quick View Settings

Tool	3/4 SPOTDRIL...
Tool Diameter	0.75
Corner Radius	0
Feed Rate	32.5973
Spindle Speed	2037
Coolant	Off
Tool Length	3
Length Offset	4
Diameter Off...	4
Cplane / Tpl...	Top
Axis Combin...	Default (1)
Tip comp	Off

Toolpath Type: Tool

Holder:

Stock:

Cut Parameters:

Linking Parameters:

Tip Comp:

Home / Ref. Points:

Planes (WCS):

Coolant:

Canned Text:

Misc Values:

Axis Control:

Axis Combination:

Rotary Axis Control:

Tool diameter: 0.75

Corner radius: 0.0

Tool name: 3/4 SPOTDRILL

Tool #: 4 Length offset: 4

Head #: 0 Diameter offset: 4

Spindle direction: CW

Feed rate: 32.597333 Spindle speed: 2037

FPT: 0.004 SFM: 399.9346

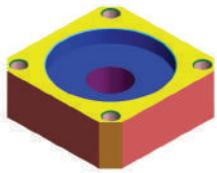
Plunge rate: 32.597333 Retract rate: 32.597333

Force tool change: Rapid Retract:

Comment: Spot drill the holes.

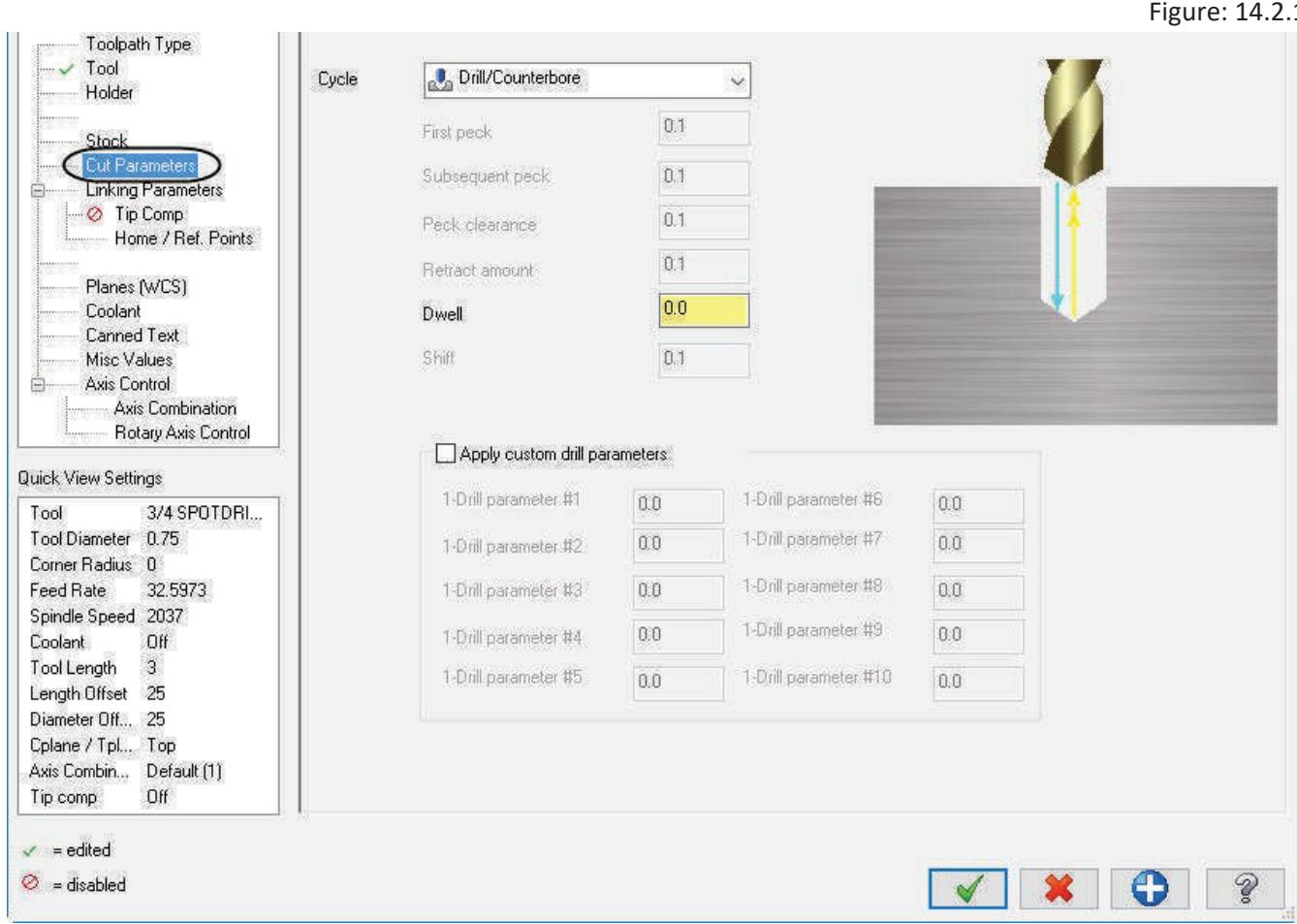
To batch:

✓ = edited ✘ = disabled



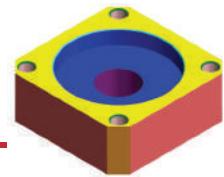
14.2 Set the Cut Parameters

- ♦ Select **Cut Parameters** and make sure the parameters are set as shown in [Figure: 14.2.1](#).



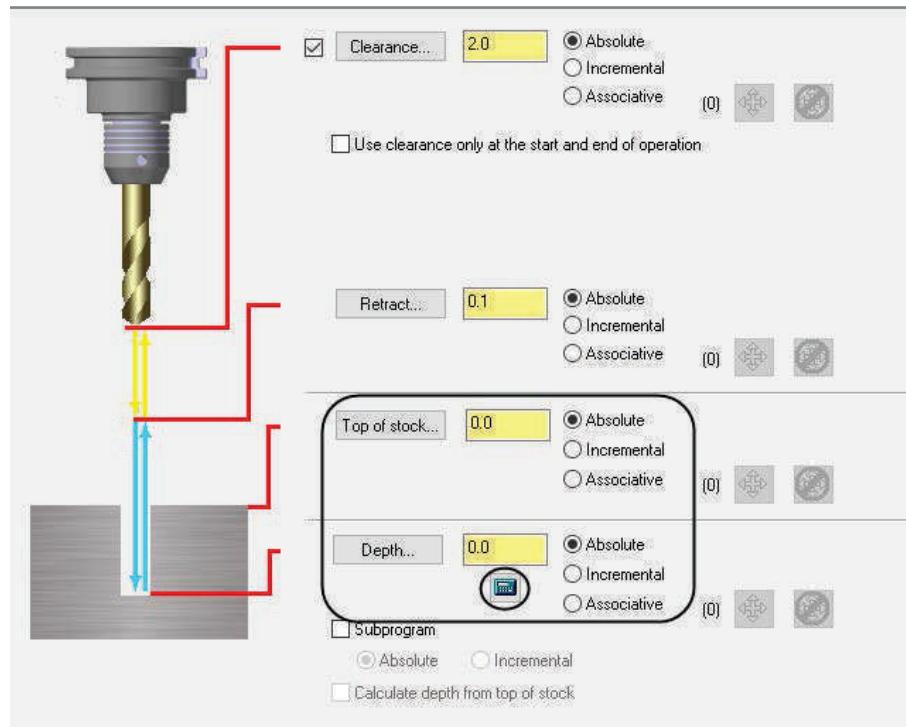
Drill/Counterbore is recommended for drilling holes with depths of less than three times the tool's diameter.

Dwell sets the amount of time in seconds that the tool remains at the bottom of a drilled hole.



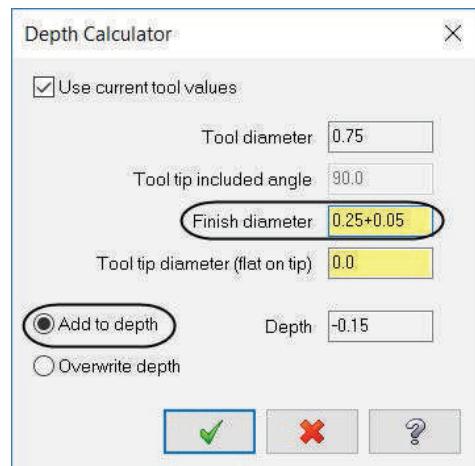
14.3 Linking Parameters

- Choose **Linking Parameters** and ensure **Clearance** is enabled. Set the **Top of stock** and the **Depth** to **Absolute** and **0.0** as shown.
- Select the **Calculator** icon on the right hand side of the **Depth** icon as shown.

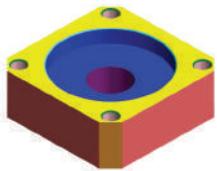


- To generate a **0.025** chamfer, input the following equation in the **Finish diameter** area: **$0.25 + 0.05$** (diameter of the finished hole + 2 X the chamfer size) and hit **Enter** to calculate the **Depth**, as shown in [Figure: 14.3.1](#). Make sure that **Add to depth** is enabled.

Figure: 14.3.1

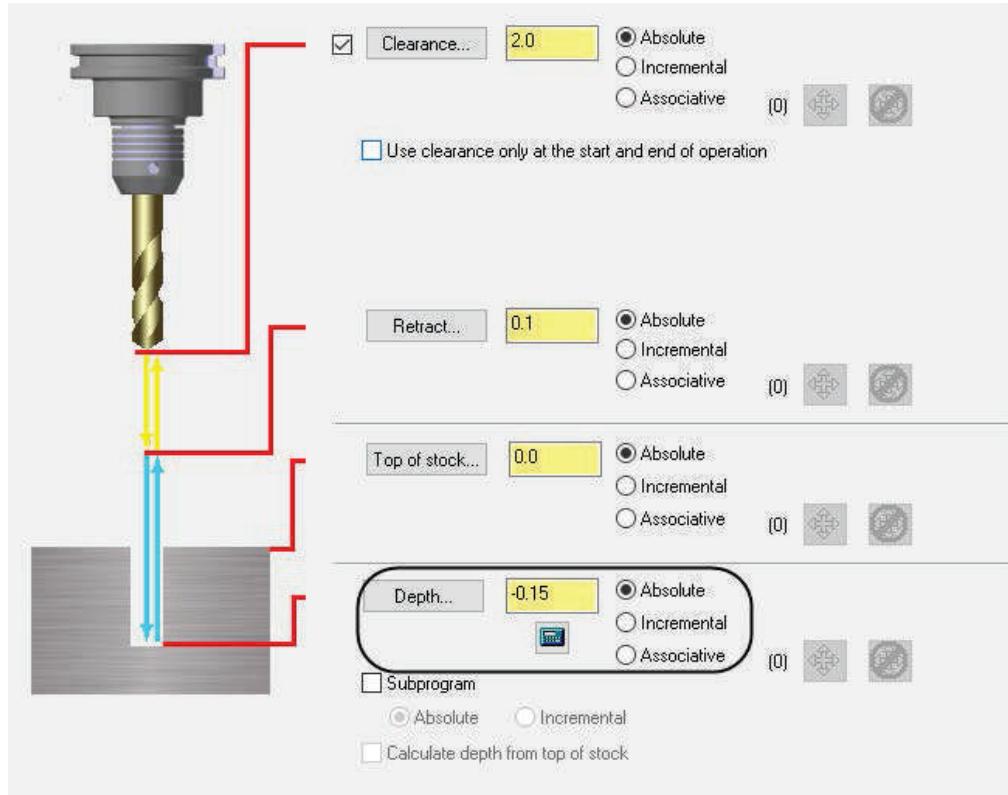


- Select the **OK** button to exit the **Depth Calculator**.



- You will now see the **Depth** for this spot drilling operation is updated after we specify the finish diameters of the holes including the chamfer. Change the rest of the parameters as shown in [Figure: 14.3.2](#).

Figure: 14.3.2



14.4 Preview the Toolpath

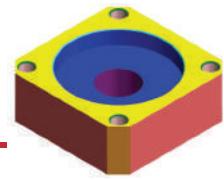
- To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



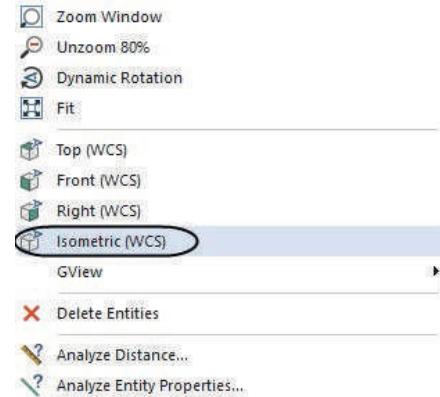
- To hide the dialog box, click on the **Hide dialog** icon as shown.



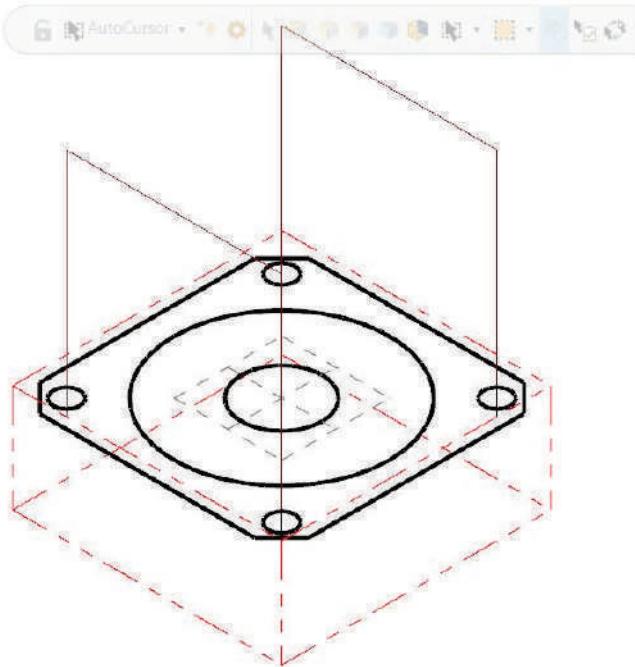
TUTORIAL #1 SPOT DRILL THE 0.25" HOLES



- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



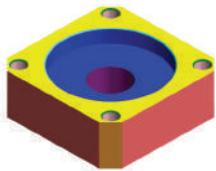
- ♦ The toolpath should look as shown.



- ♦ Press **Esc** key to exit the preview.

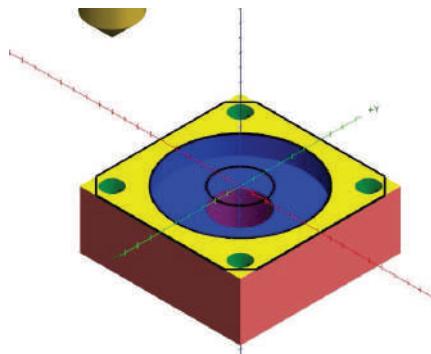
NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to exit the toolpath parameters.



14.5 Verify the toolpaths

- ♦ See [page 58](#) to review the procedure.

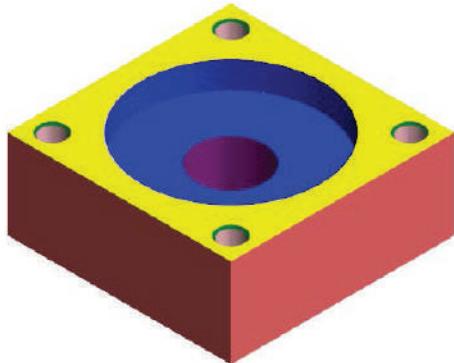


- ♦ To go back to the Mastercam window, minimize the Mastercam Simulator window as shown.

STEP 15: DRILL THE 0.25" HOLES

In this step, we will drill the holes to a specified depth.

Toolpath Preview:



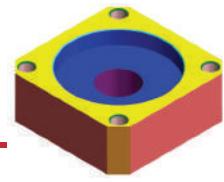
- ♦ Move the cursor in the **Toolpaths Manager** and press **Alt + T** until the toolpath display is removed.

TOOLPATHS

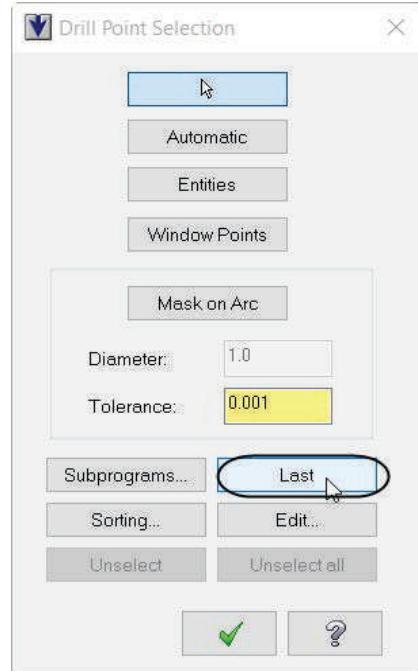
- ♦ From the **2D** group, select **Drill**.



TUTORIAL #1 DRILL THE 0.25" HOLES



- In the Drill Point Selection panel, choose the option Last.



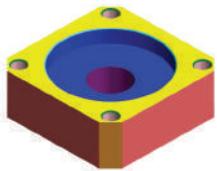
- This option will automatically select the 4 holes from the previous drill operation.
- Select the OK button in the Drill Point Selection panel to accept the 4 drill points.
- In the Toolpath Type page, the Drill toolpath will be selected as shown in [Figure: 15.0.1](#).



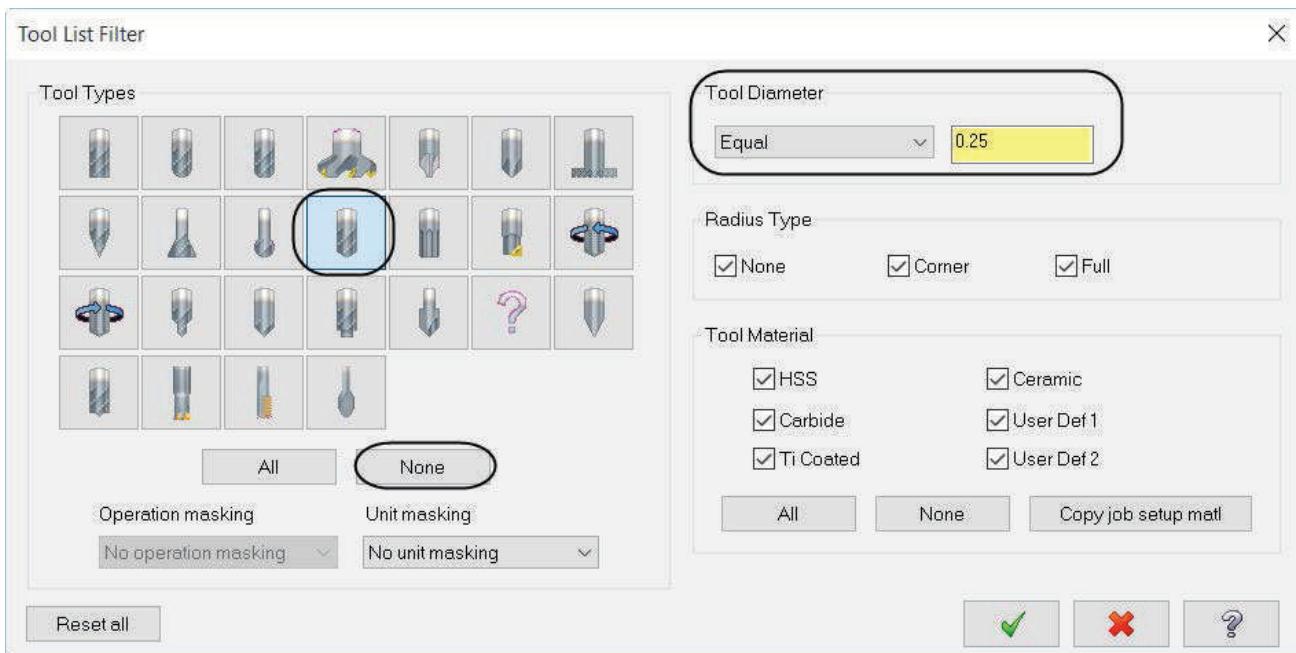
15.1 Select a 1/4" Drill from the library and set the Tool Parameters

- Select Tool from the Tree View list.
- Click on the Select library tool... button.
- To view only the drill tools, select the Filter button.





- Under **Tool Types**, select the **None** button and then choose the **Drill** icon. Under **Tool Diameter** section, select **Equal** and input a value of **0.25**.



- Select the **OK** button to exit the **Tool List Filter** panel.



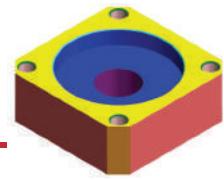
- At this point you should see a **1/4" Drill**.

#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	Length	# Flut...	Type	Rad....
124	-	1/4 DRILL	-	0.25	0.0	2.0	2	Drill	None

- Select the tool in the **Tool Selection** page and then choose the **OK** button to exit.

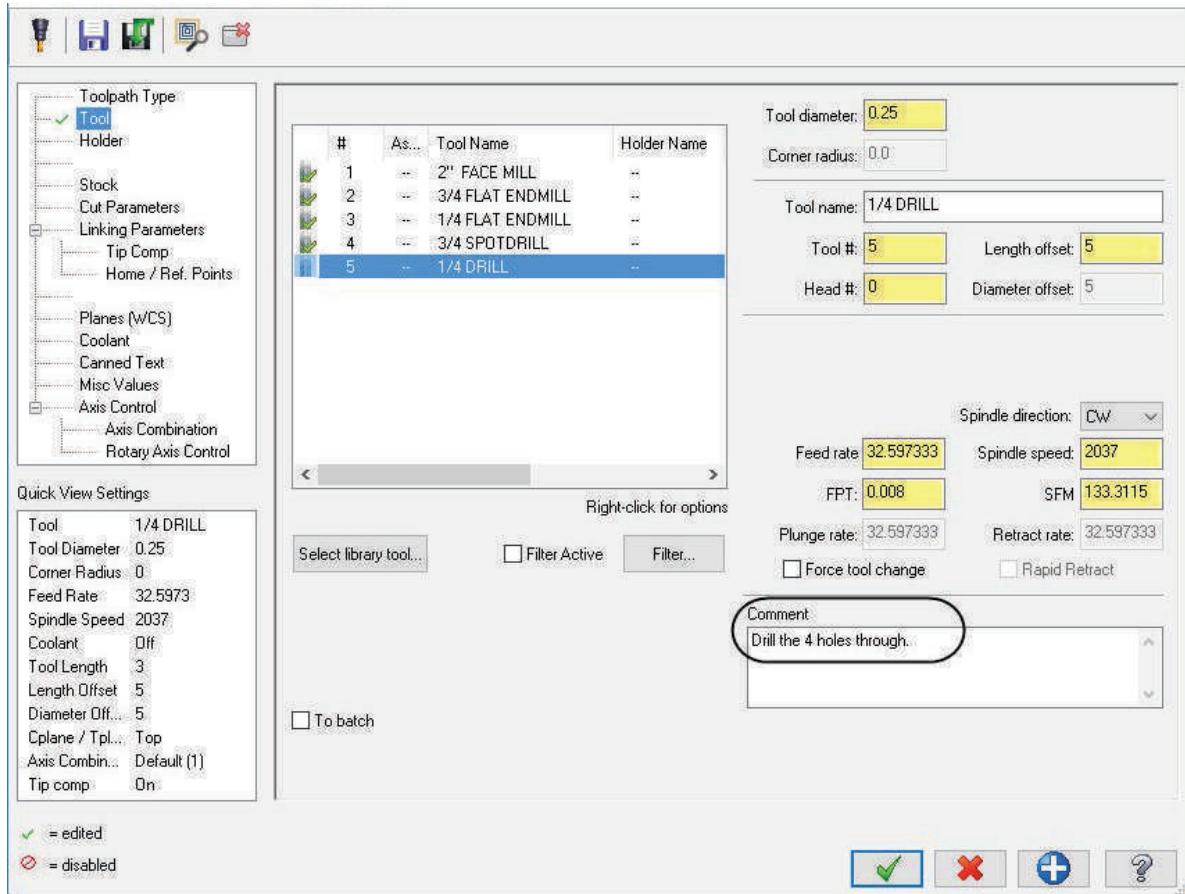


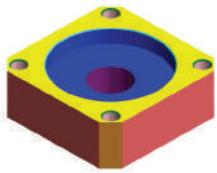
TUTORIAL #1 DRILL THE 0.25" HOLES



- Make the necessary changes to the Tool page as shown in [Figure: 15.1.1](#).

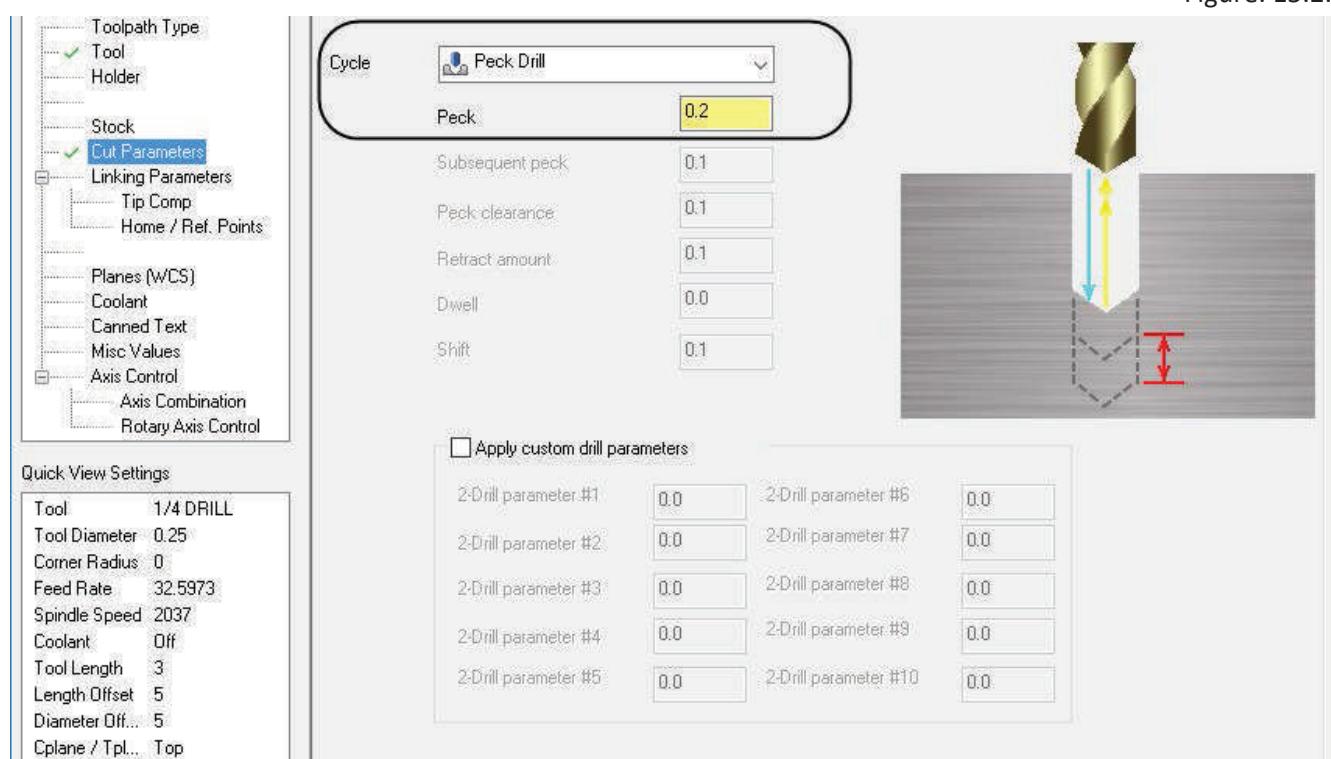
Figure: 15.1.1

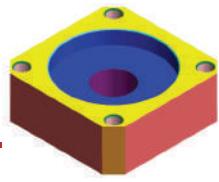




15.2 Cut Parameters

- ♦ Select **Cut Parameters** and change the drill **Cycle** to **Peck Drill** as shown in [Figure: 15.2.1](#).

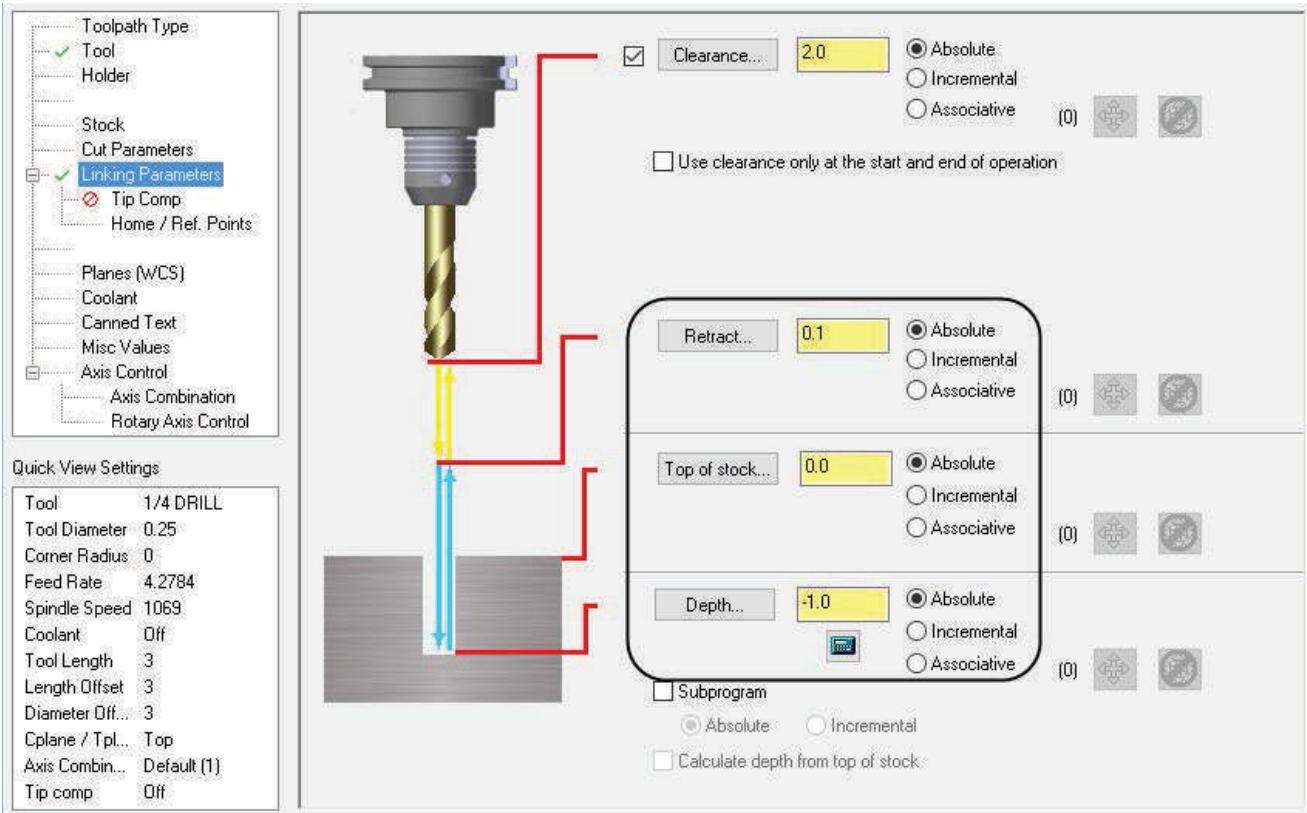


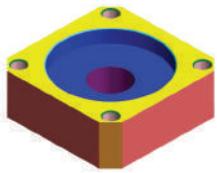


15.3 Linking Parameters

- Choose **Linking Parameters** and set the **Top of stock** to **0.0**. Input a **Depth** value of **-1.0** as shown in Figure: [15.3.1](#).

Figure: 15.3.1

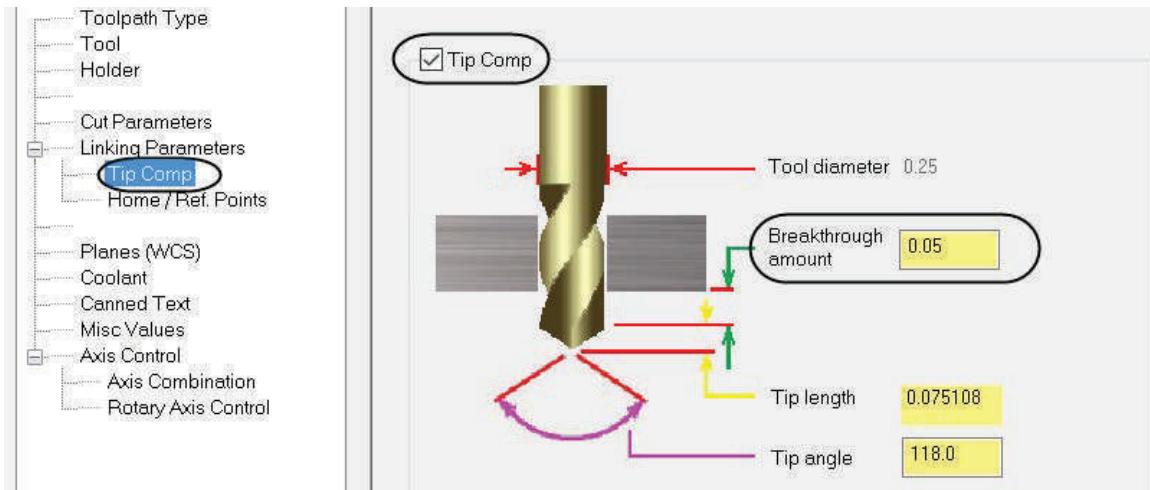




15.4 Set the Tip Compensation

- ♦ Select **Tip Comp** and enable it.
- ♦ Set the **Breakthrough amount** to **0.05** as shown in [Figure: 15.4.1](#).

Figure: 15.4.1

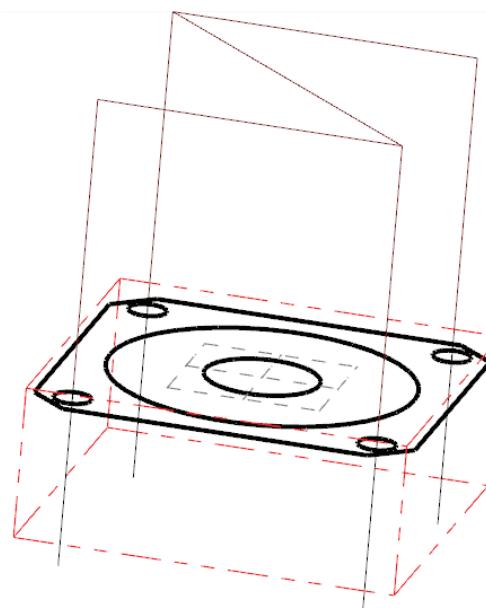


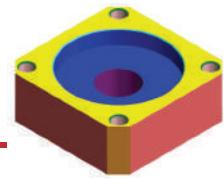
15.5 Preview the Toolpath

- ♦ To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.
- ♦ The toolpath should look as shown.





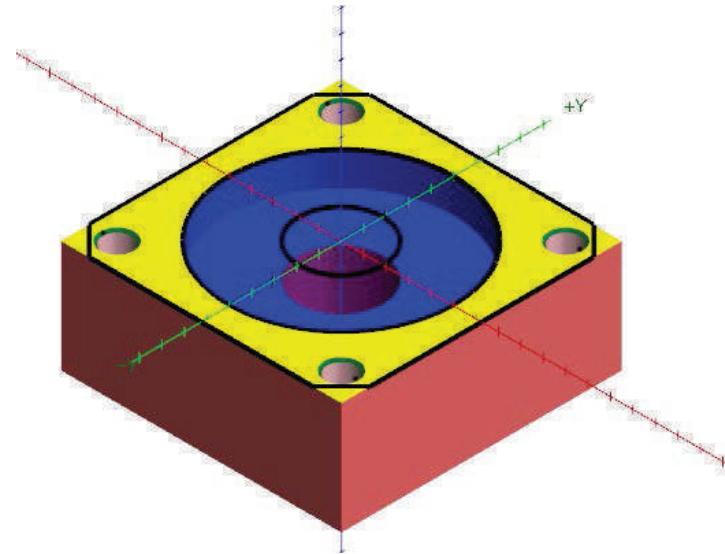
- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

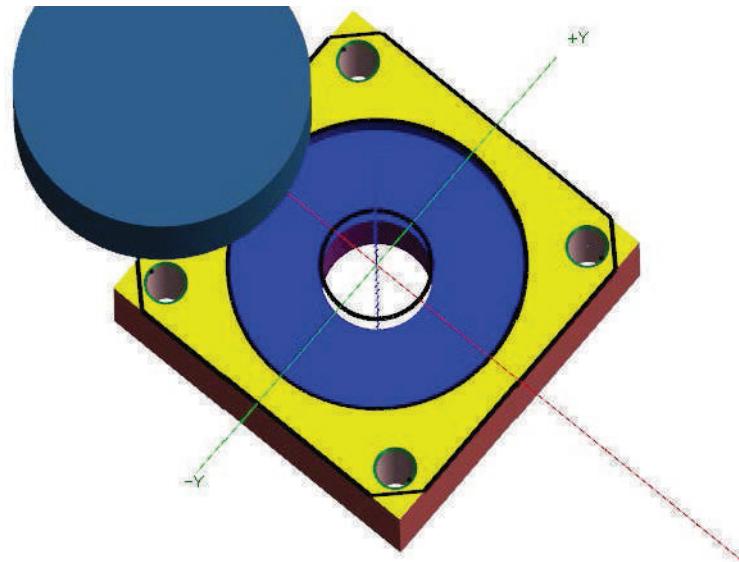
- ♦ Select the **OK** button to exit the **2D Toolpaths - Drill/Circles Simple Drill - no peck** parameters.

15.6 Verify the toolpaths

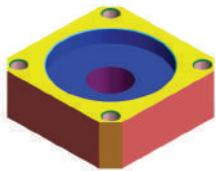
- ♦ To **Verify** the toolpaths, see [page 58](#) to review the procedure.



- ♦ To rotate the part, click in the center of the part with the mouse wheel. Hold down the mouse wheel and slightly drag the cursor to rotate.



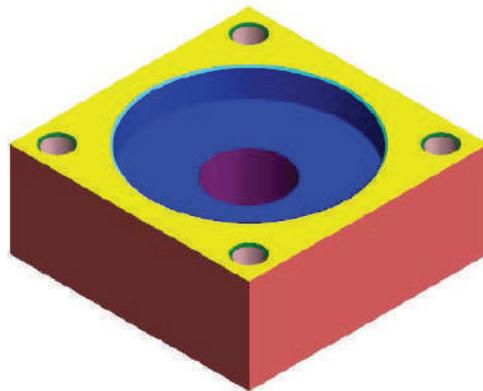
- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.



STEP 16: CHAMFER THE LARGE HOLE

Chamfer Toolpath automatically cuts a chamfer around a contour using a chamfer mill.

Toolpath Preview:

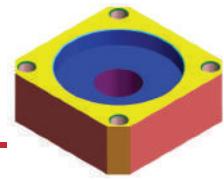


TOOLPATHS

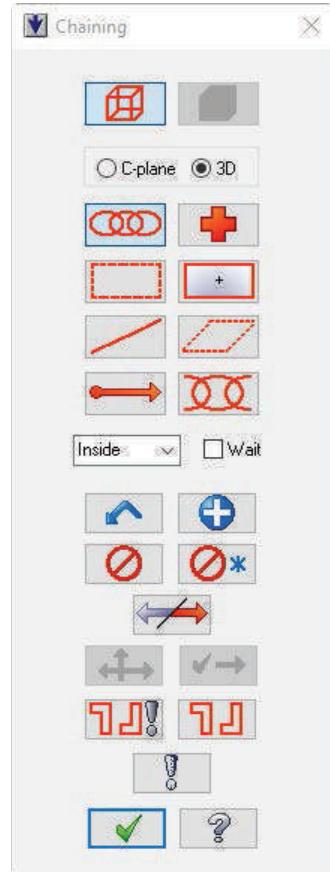
- ♦ In the **2D** group, select the **Contour** icon as shown.



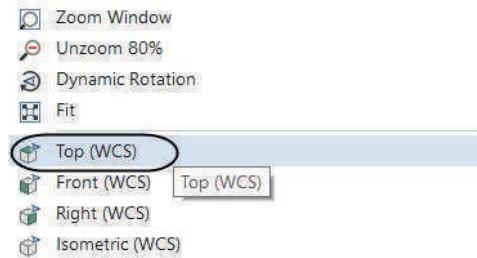
TUTORIAL #1 CHAMFER THE LARGE HOLE

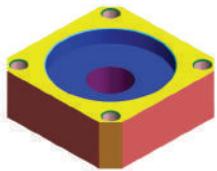


- Leave the default settings in the **Chaining** dialog box as shown.



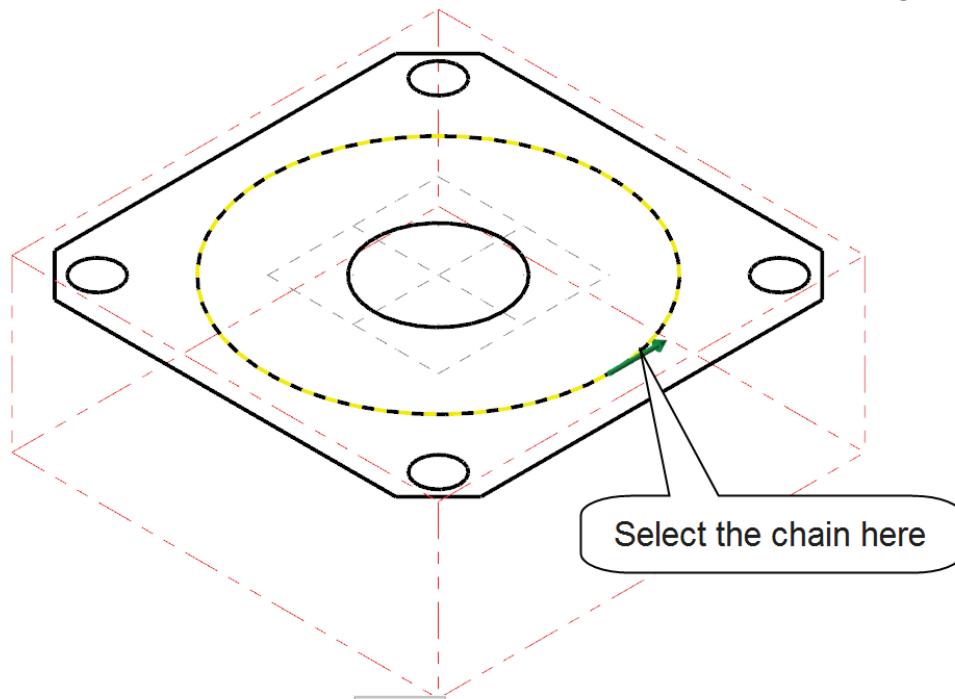
- If needed, right mouse click and select the **Top** view.





- ♦ Select the chains and ensure the chaining direction is the same as shown in [Figure: 16.0.1](#).

Figure: 16.0.1



- ♦ Select the **OK** button to exit the **Chaining** dialog box.
- ♦ In the **Toolpath Type** page, the **Contour** toolpath will be selected.



Contour



Pocket

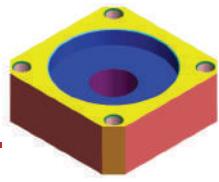


Facing



Slot Mill

TUTORIAL #1 CHAMFER THE LARGE HOLE



16.1 Select a 1/2" Chamfer Mill from the library and set the Tool parameters

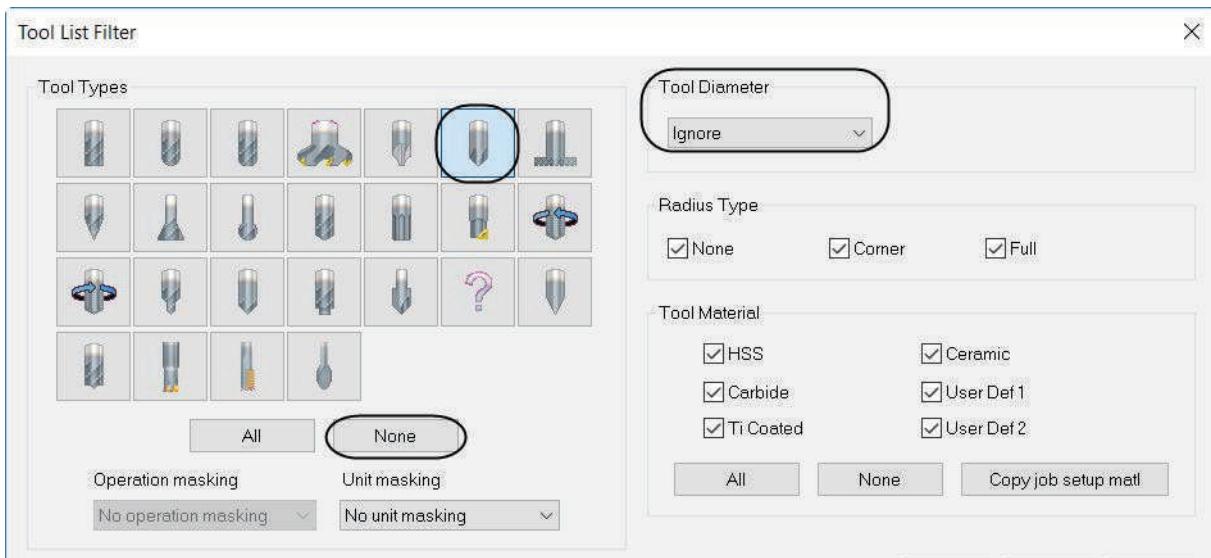
- ♦ Select **Tool** from the **Tree View list**.
- ♦ Click on the **Select library tool** button.



- ♦ To be able to see just the chamfer mill, select the **Filter** button.



- ♦ Under **Tool Types**, select the **None** button and then choose the **Chamfer Mill** icon.



- ♦ Select the **OK** button to exit the **Tool List Filter** dialog box.



- ♦ At this point you should only see a list of chamfer mills.

- ♦ From the **Tool Selection** list, select the **1/2" Chamfer Mill**.

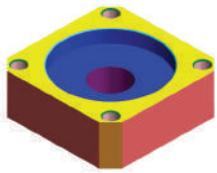
#	Assembly...	Tool Name	Holder Name	Dia.	Cor. r...	Length	# Flut...	Type	Rad....
318	-	1/4 CHAMFER MILL	-	0.25...	0.0	0.5	4	Cha...	None
319	-	1/2 CHAMFER MILL	-	0.5-45	0.0	0.75	4	Cha...	None
320	-	3/4 CHAMFER MILL	-	0.75...	0.0	1.0	4	Cha...	None
321	-	1 INCH CHAMFER MILL	-	1.0-45	0.0	1.0	4	Cha...	None

- ♦ In the **Tool Selection** page, choose the **OK** button to exit.



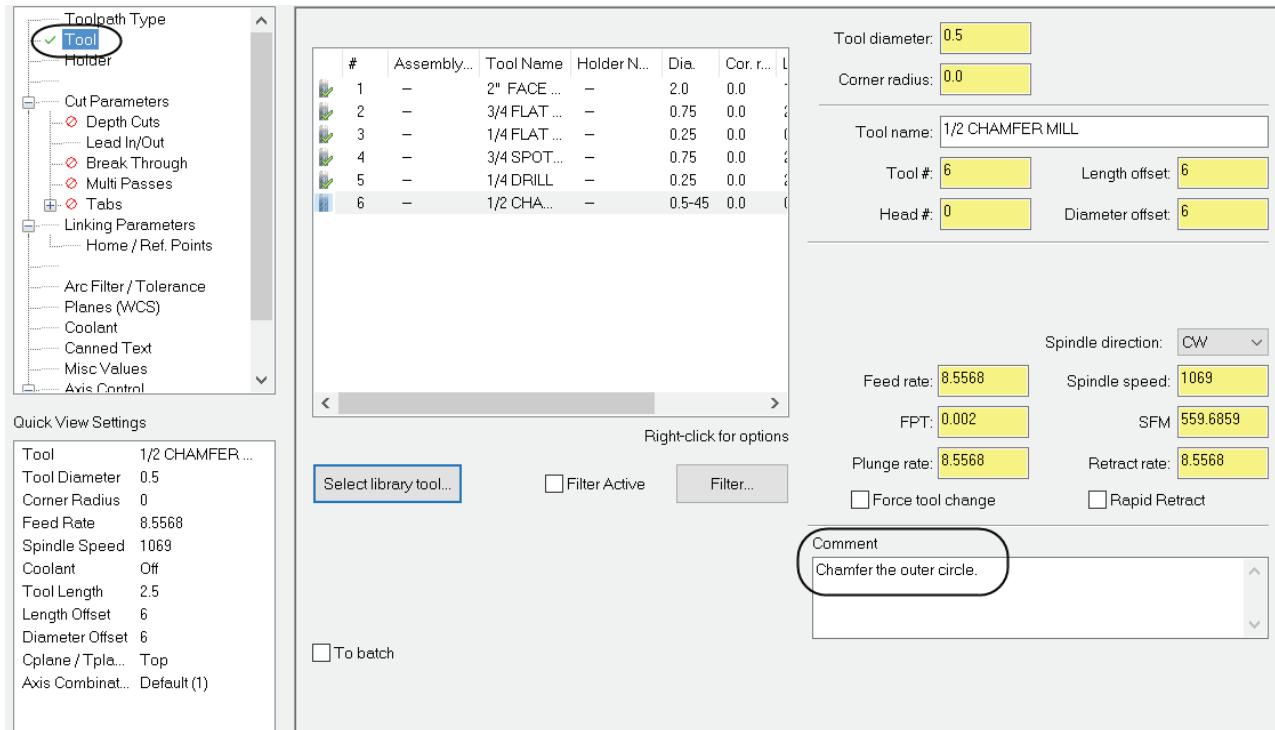
- ♦ A warning message that the tool selected is not defined as being capable of both roughing and finish may appear on the screen.

NOTE: The chamfer mill is defined for finish operations only. For a chamfer toolpath, we only need a finish operation.

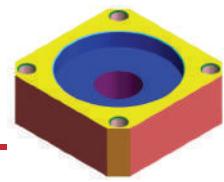


- ♦ Select the **OK** button to continue.
- ♦ Make all the necessary changes as shown in [Figure: 16.1.1](#).

Figure: 16.1.1



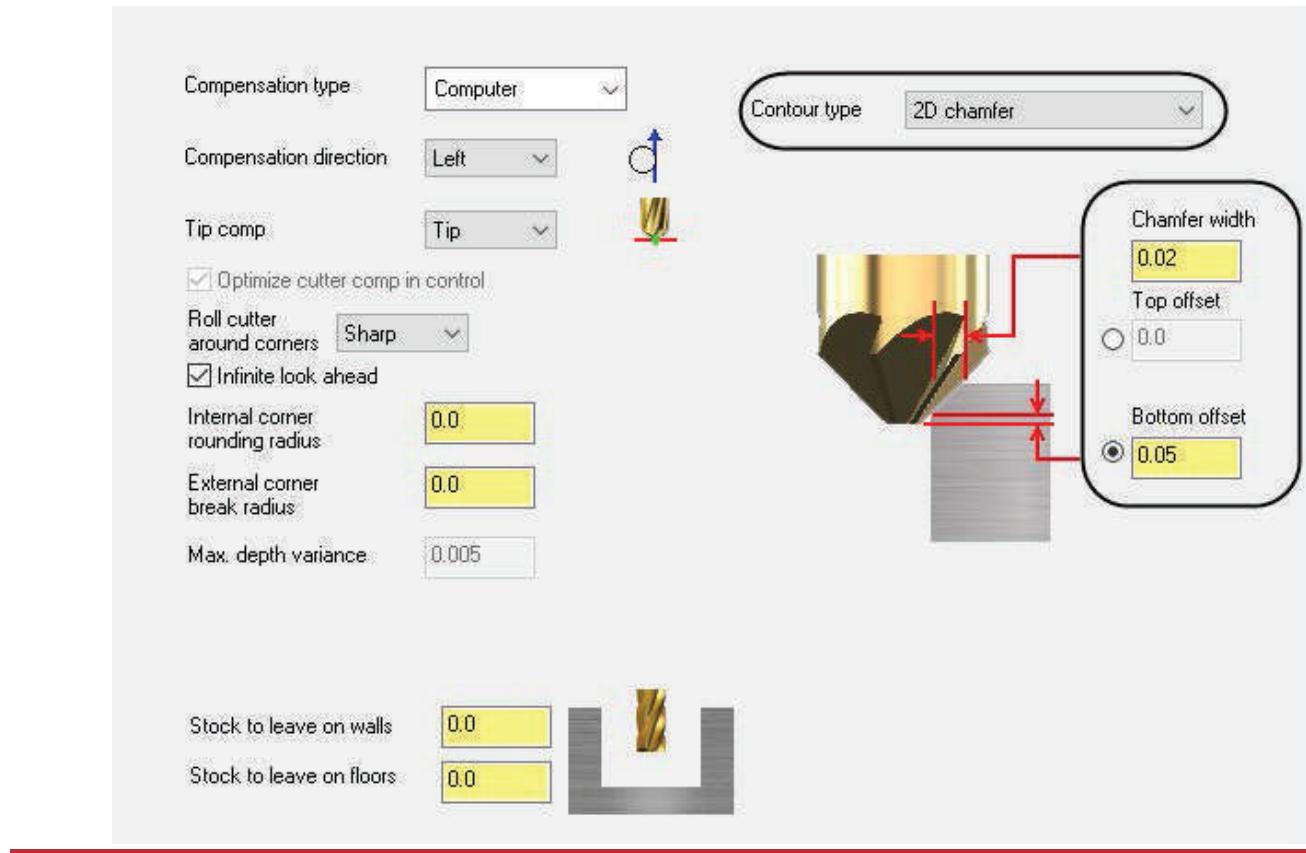
TUTORIAL #1 CHAMFER THE LARGE HOLE



16.2 Cut Parameters

- From the Tree View list, select the **Cut Parameters** page and change the **Contour type** to **2D chamfer**.
- Input a **Chamfer width** of **0.02** and a **Bottom offset** of **0.05** as shown in [Figure: 16.2.1](#).

Figure: 16.2.1



2D chamfer cuts chamfers around a contour.

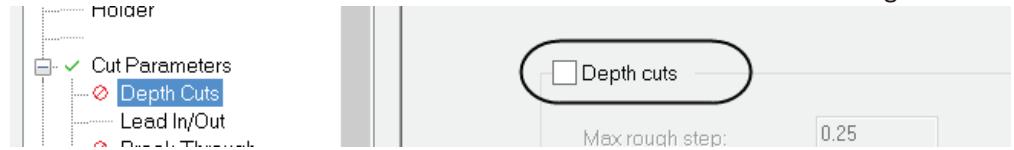
Chamfer width sets the chamfer width. Mastercam measures the width from the chained geometry adjusted by the cut depths defined on the **Linking Parameters** page.

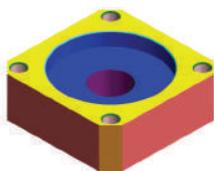
Bottom offset is an amount to ensure that the tip of the tool clears the bottom of the chamfer.

16.3 Depth Cuts

- From the Tree View list, select **Depth Cuts** and **disable** it as shown in [Figure: 16.3.1](#).

Figure: 16.3.1

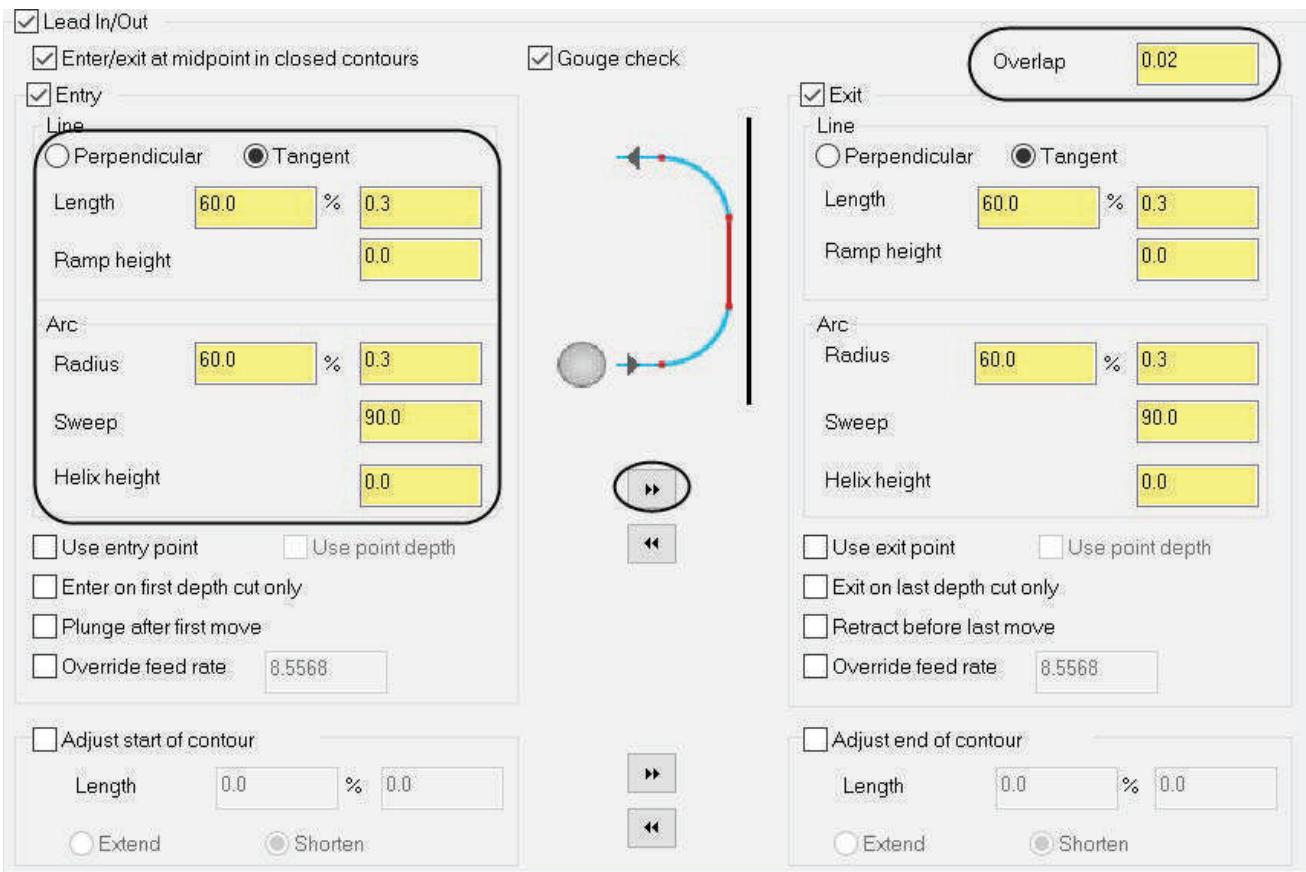




16.4 Lead In/Out

- From the **Tree View** list, choose the option **Lead In/Out** and input an **Overlap** value as shown.
- Make any other necessary change as shown in [Figure: 16.4.1](#).

Figure: 16.4.1

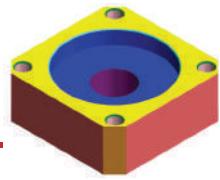


Lead In/Out allows you to select a combination of a Line and an Arc at the beginning and/or end of the contour toolpath for a smooth entry/exit while cutting the part.

Length is set to 60% of the tool diameter to ensure that the linear movement is greater than the tool radius in case **Cutter Compensation** in **Control** was used.

Radius is set to 60% of the tool diameter to ensure that the arc movement is greater than the tool radius to generate an arc output.

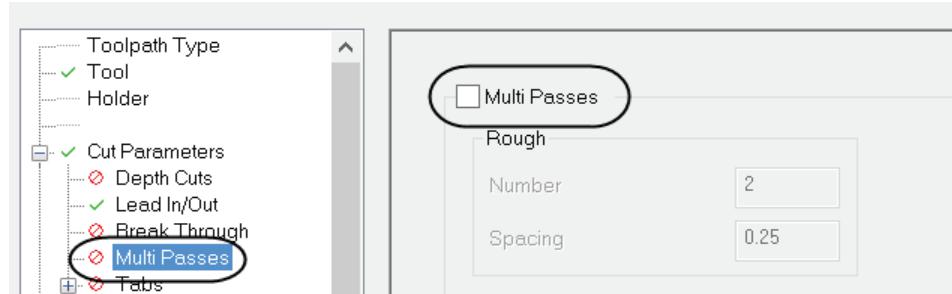
Overlap sets how far the tool goes past the end of the toolpath before exiting for a cleaner finish.



16.5 Multi Passes

- From the Tree View list, select **Multi Passes** and disable this option as shown in [Figure: 16.5.1](#).

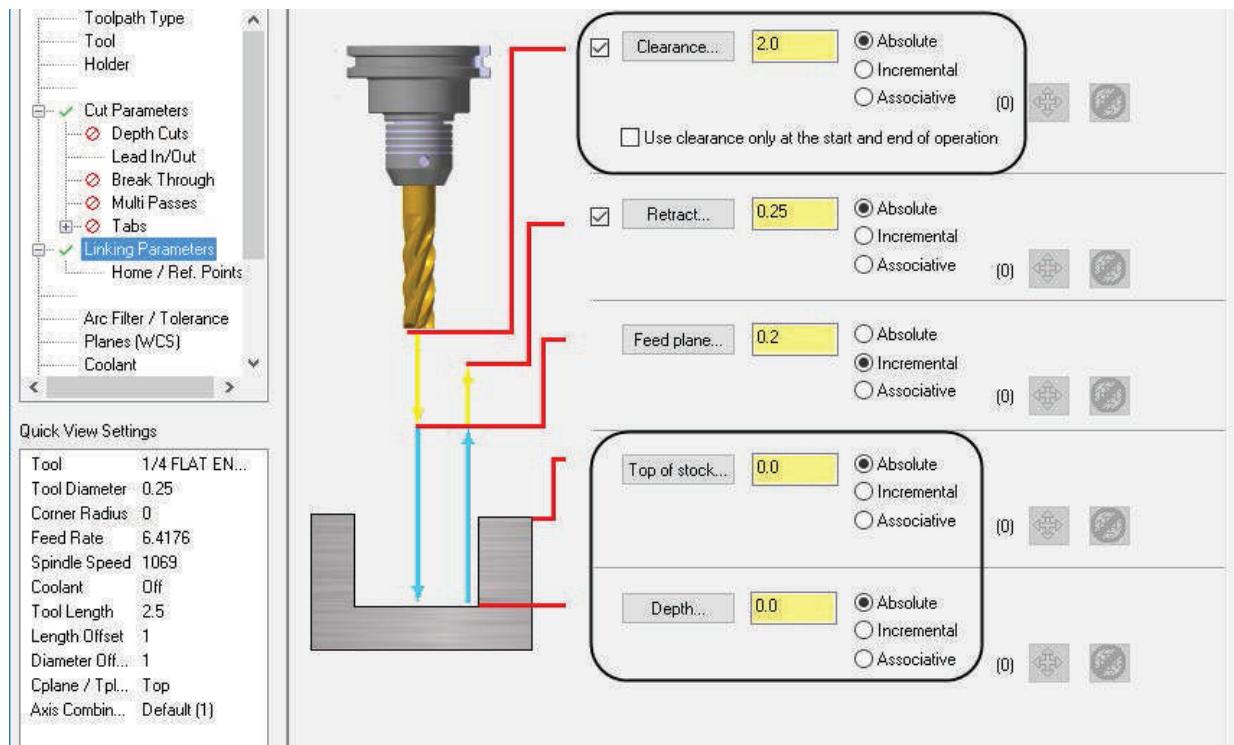
Figure: 16.5.1



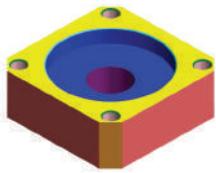
16.6 Linking Parameters

- From the Tree View list, select the **Linking Parameters**. Set the **Top of stock** to **0.0** and the **Depth** to **0.0** as shown in [Figure: 16.6.1](#).

Figure: 16.6.1



NOTE: The depth of the chamfer is based on the width and tip offset set in the **Cut Parameters** page. This is why we set the depth here to zero.

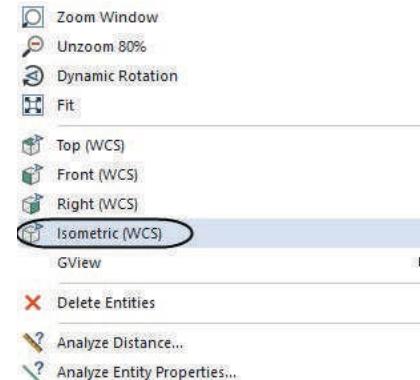


16.7 Preview the Toolpath

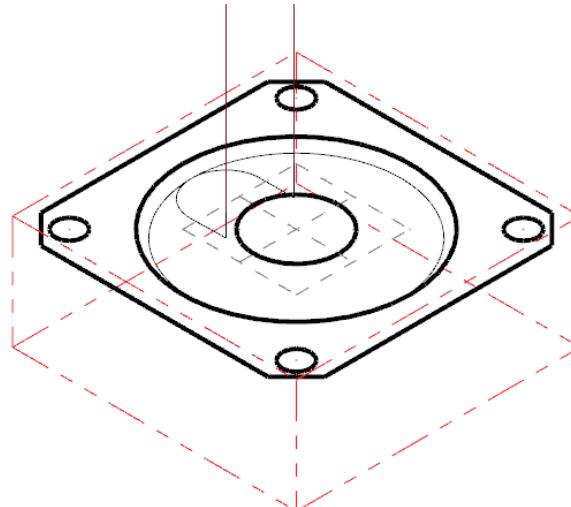
- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.
- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.



- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to exit the **2D Toolpaths - Contour** dialog box.

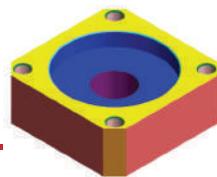


16.8 Verify the toolpaths

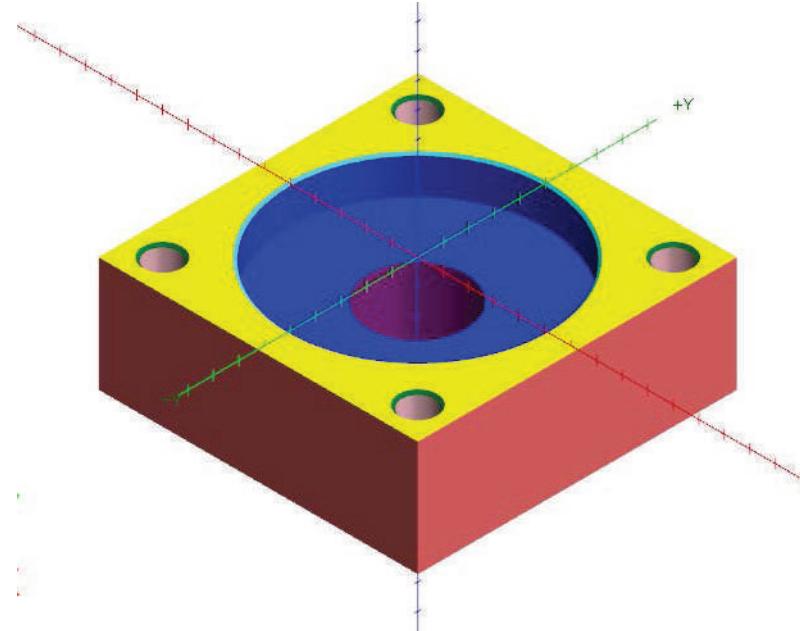
- ♦ To **Verify** the toolpaths, see [page 58](#) to review the procedure.

TUTORIAL #1

MACHINE THE CHAMFERS AT THE CORNERS USING CONTOUR TOOLPATH



- ♦ Ensure all operations are selected. If they are not, use the button **Select all operations** in the **Toolpaths Manager**.
- ♦ Your part will appear as shown.

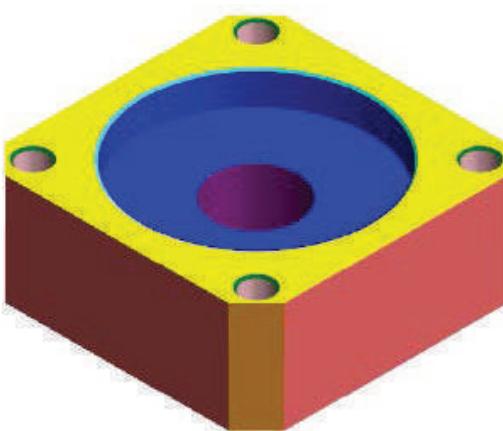


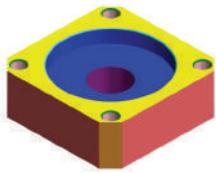
- ♦ To go back to Mastercam window, minimize **Mastercam Simulator** window as shown.

STEP 17: MACHINE THE CHAMFERS AT THE CORNERS USING CONTOUR TOOLPATH

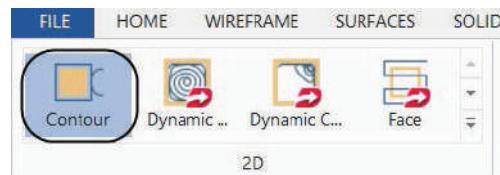
In this step you will machine the corners of the part using **Contour Toolpath**.

Toolpath Preview:

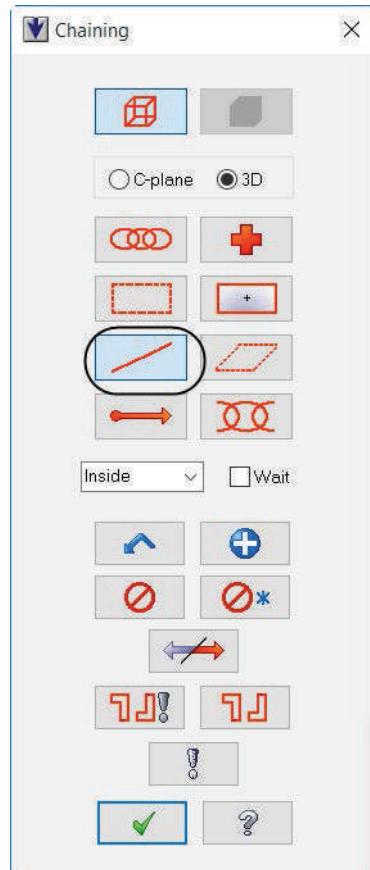


**TOOLPATHS**

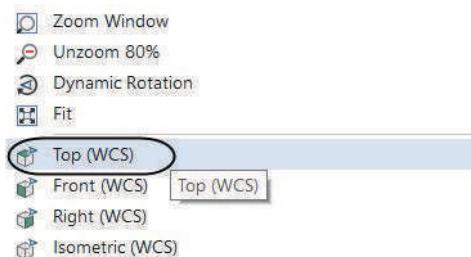
- From the **2D** group, select **Contour** as shown.



- To select only one entity at a time, select the **Single** button in the **Chaining** dialog box as shown below.

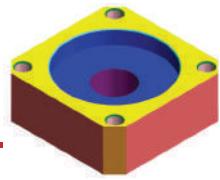


- Right mouse click in the graphics window and select the **Top** view as shown.



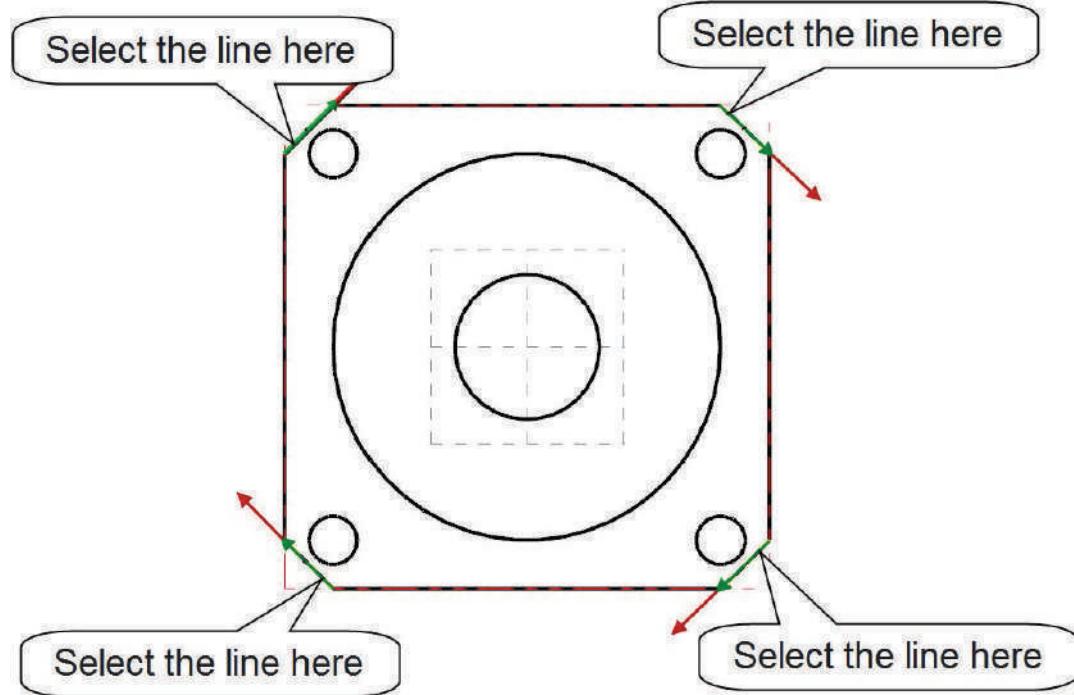
TUTORIAL #1

MACHINE THE CHAMFERS AT THE CORNERS USING CONTOUR TOOLPATH



- ♦ Select the chains and ensure the chaining direction is the same as shown in [Figure: 17.0.1](#).

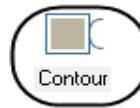
Figure: 17.0.1



NOTE: Select the contour as shown in [Figure: 17.0.1](#) to ensure that the chaining directions for all four chains are correct. Use the **Reverse** button to reverse the chains if needed.

The green color arrow shows the chain start location and the red color arrow shows the end of the chain. The chain selection arrows disappear as you select the next chamfer.

- ♦ Select the **OK** button to exit the **Chaining** dialog box.
- ♦ In the **Toolpath Type** page, the **Contour** toolpath will be selected.



Contour



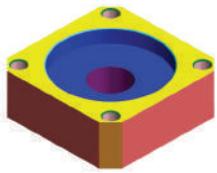
Pocket



Facing



Slot Mill



17.1 Select the 1/2" Flat Endmill and set the Tool parameters

- ♦ Click on **Tool** in the **Tree View** list and make all the necessary changes as shown in [Figure: 17.1.1.](#)

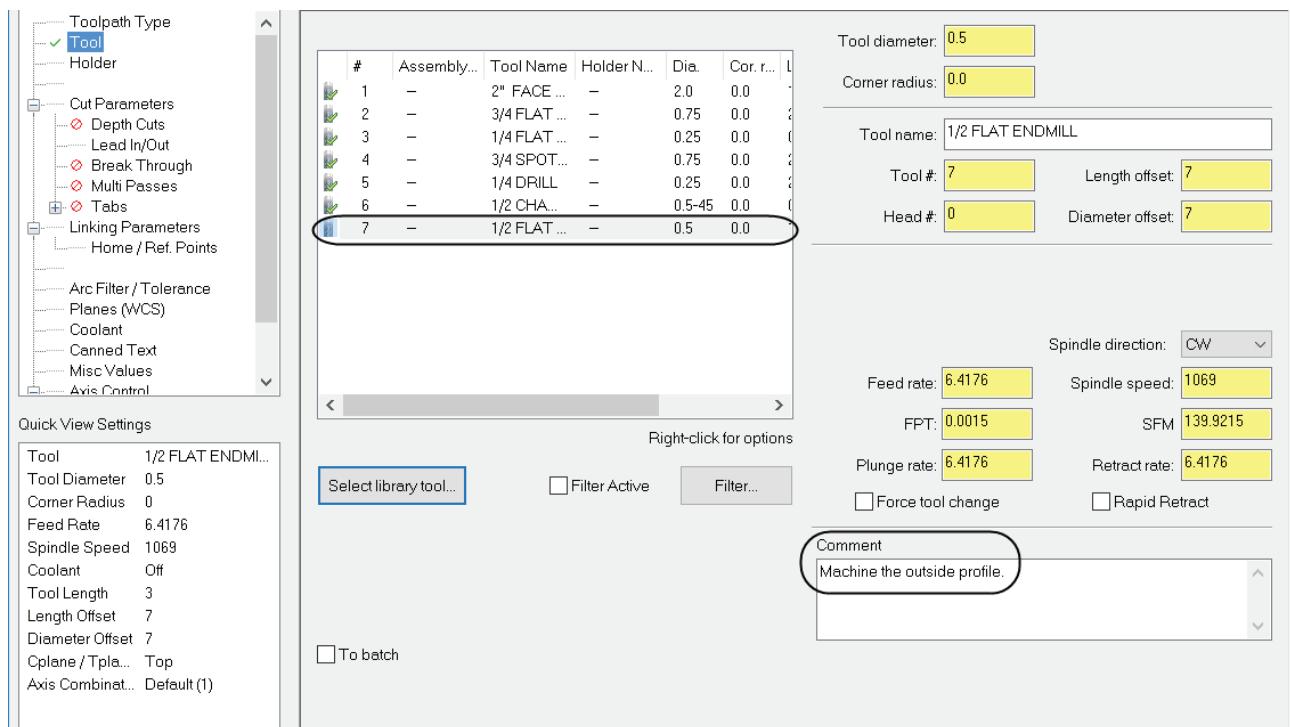
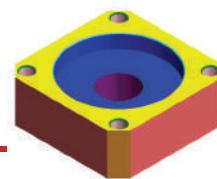


Figure: 17.1.1

TUTORIAL #1

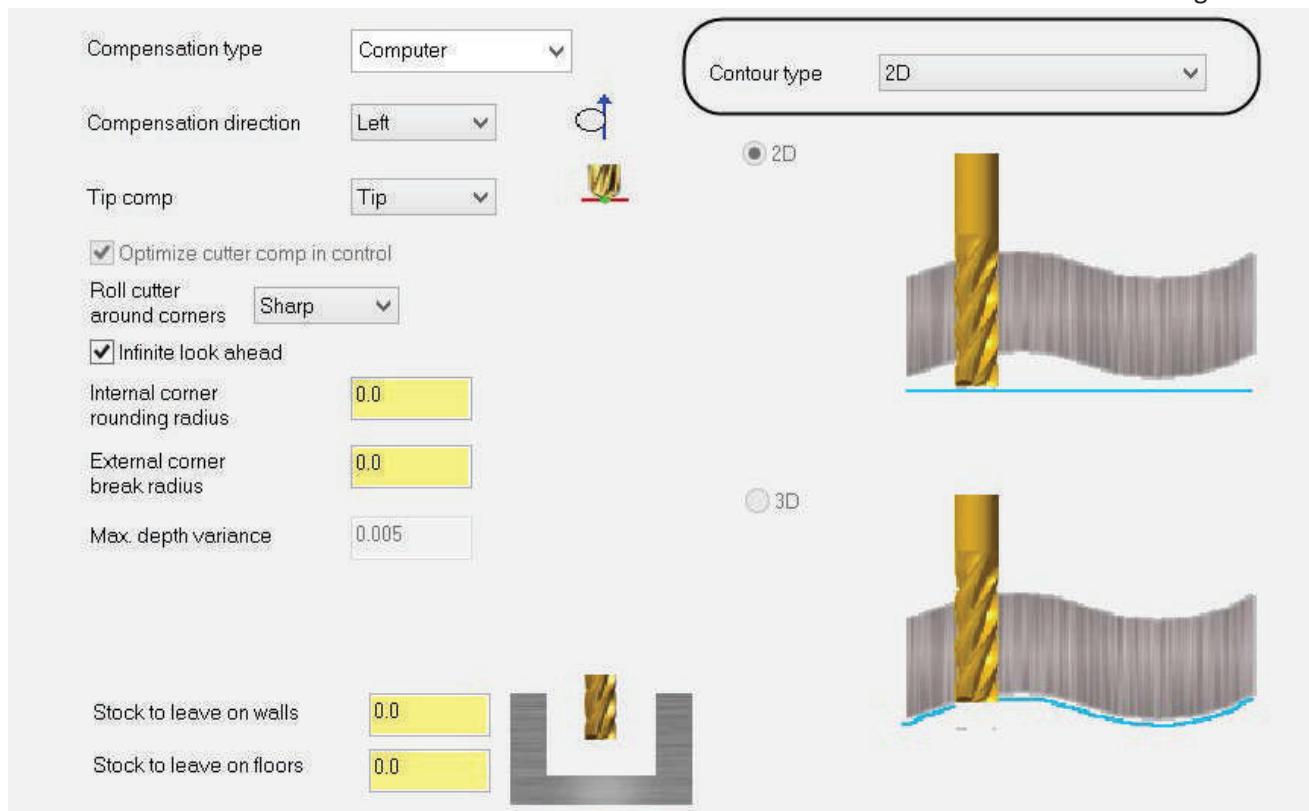
MACHINE THE CHAMFERS AT THE CORNERS USING CONTOUR TOOLPATH

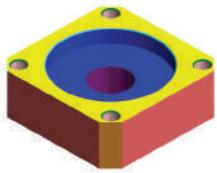


17.2 Cut Parameters

- From the Tree View list, select the **Cut Parameters** page and change the **Contour type** to **2D** as shown in [Figure: 17.2.1](#).

Figure: 17.2.1

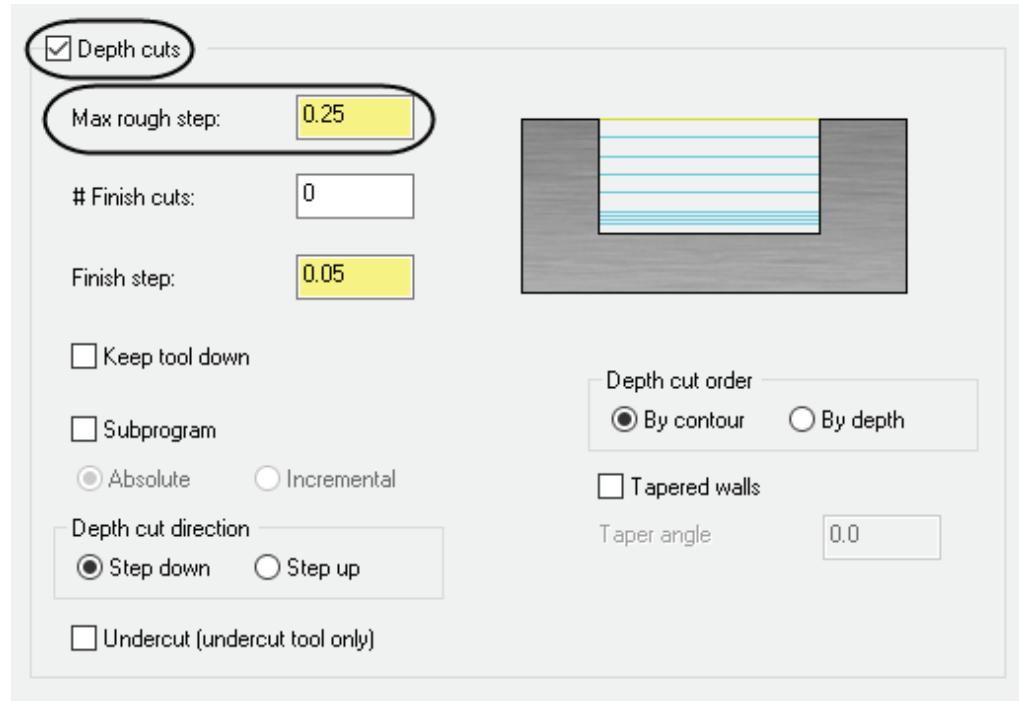


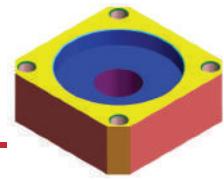


17.3 Depth Cuts

- From the **Tree View** list, select **Depth Cuts** and enable it as shown in [Figure: 17.3.1](#).
- Make sure that the parameters are set as shown in [Figure: 17.3.1](#).

Figure: 17.3.1

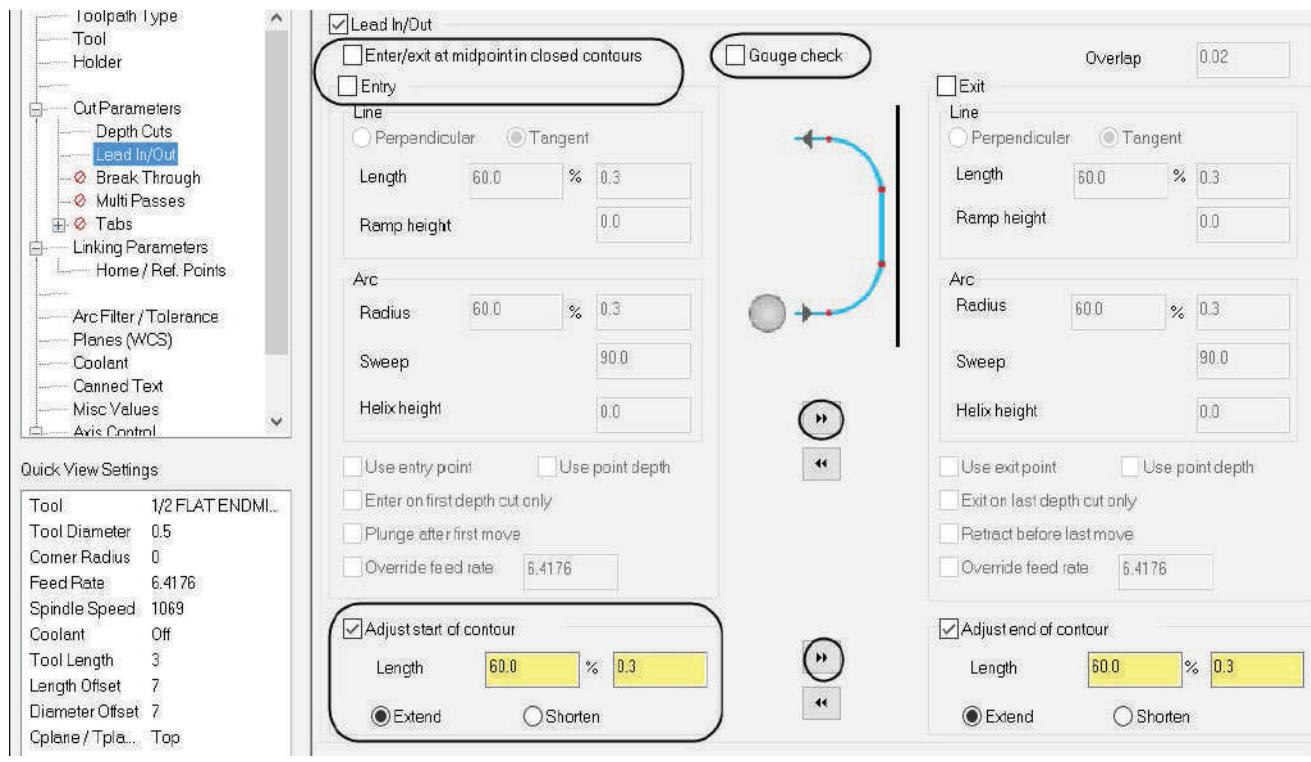




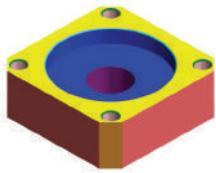
17.4 Lead In/Out

- Choose the option **Lead In/Out** and make sure the parameters are set as shown in [Figure: 17.4.1](#).

Figure: 17.4.1



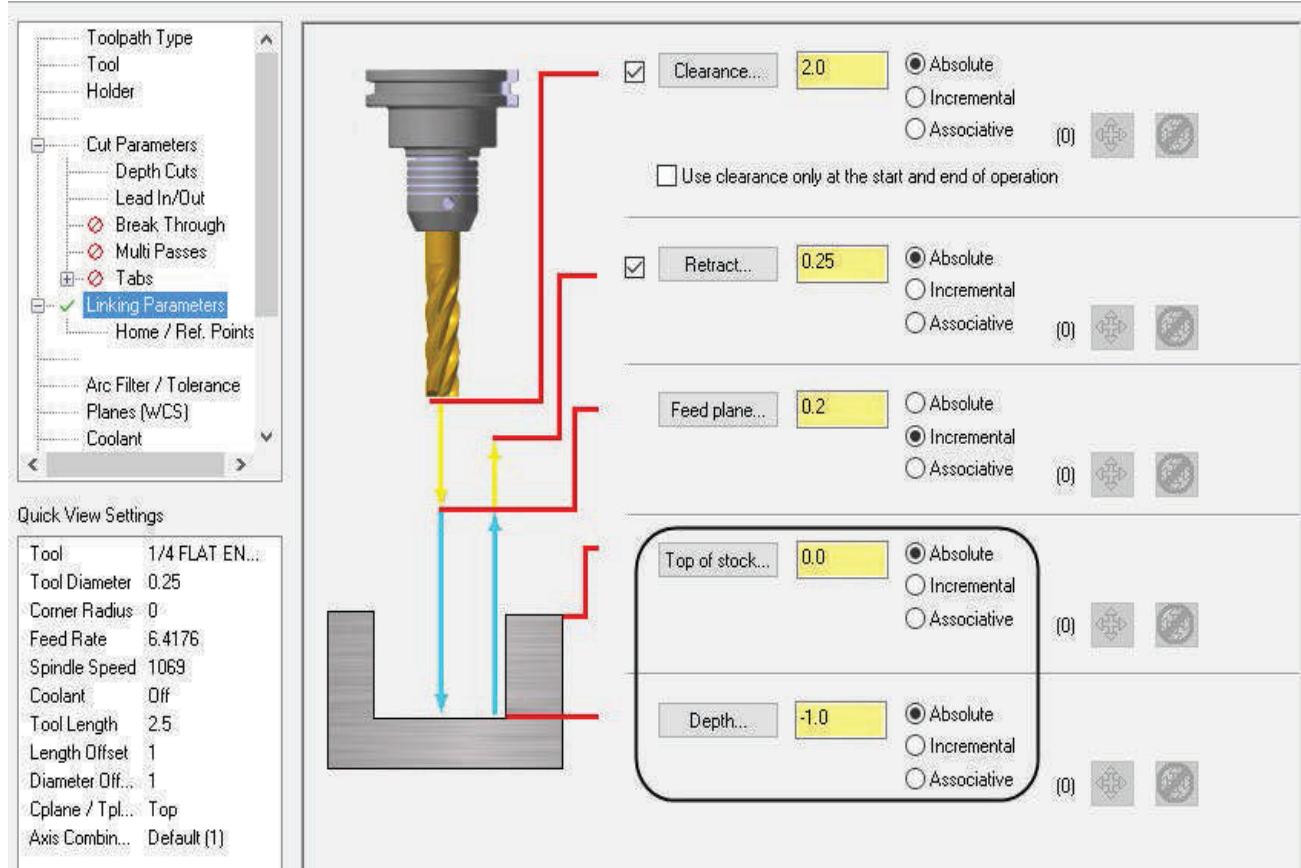
Adjust start/end of contour moves the starting/ending position in open contours by adding (**Extend**) or removing (**Shorten**) the specified length.



17.5 Linking Parameters

- ♦ Select the **Linking Parameters** from the **Tree View list**. Set the **Top of stock** and the **Depth** as shown in [Figure: 17.5.1](#).

Figure: 17.5.1



17.6 Preview the Toolpath

- ♦ To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

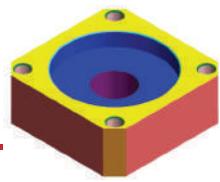


- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.

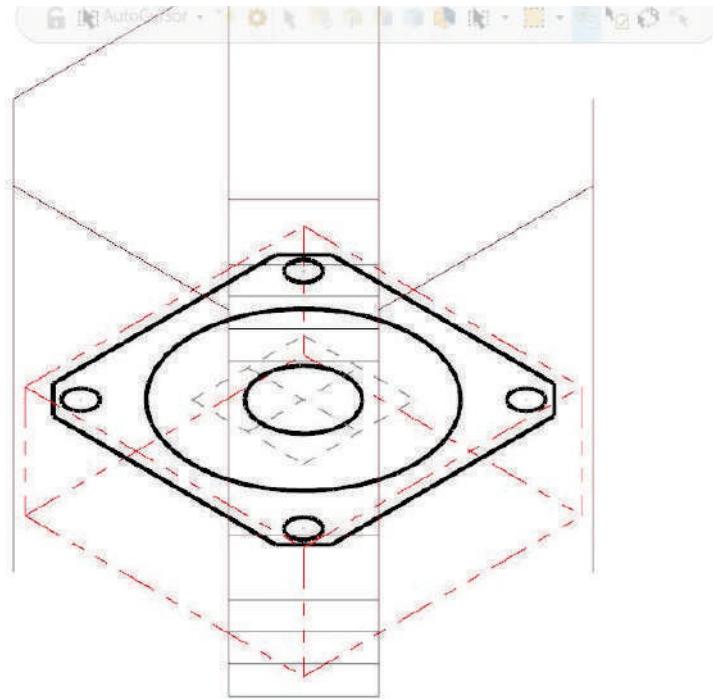


TUTORIAL #1

MACHINE THE CHAMFERS AT THE CORNERS USING CONTOUR TOOLPATH



- The toolpath should look as shown.

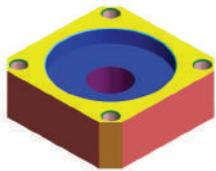


- Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

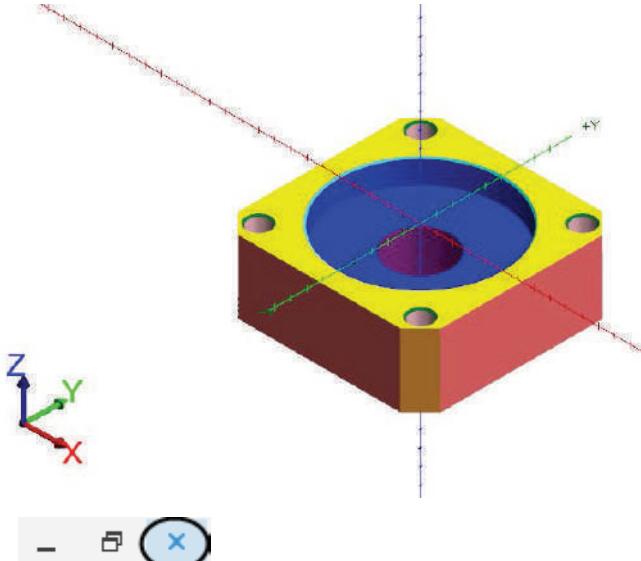
- Select the **OK** button to exit the **2D Toolpath - Contour** toolpath parameters.





17.7 Verify the toolpaths

- ♦ To **Verify** the toolpaths, see [page 58](#) to review the procedure.
- ♦ Ensure all operations are selected. If they are not, use the button **Select all operations** in the **Toolpaths Manager**.
- ♦ Your part will appear as shown.

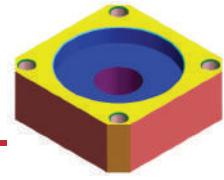


- ♦ To exit the **Mastercam Simulator**, click on the **Close** icon.

STEP 18: POST THE FILE

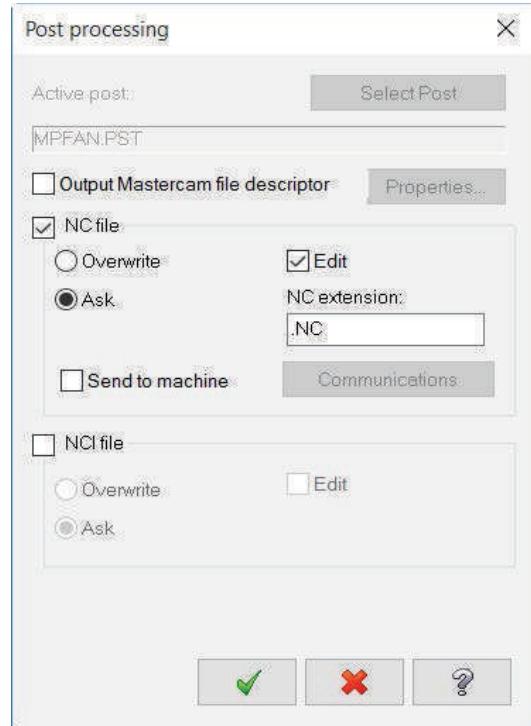
- ♦ Ensure all operations are selected. If they are not, use the button **Select all operations** in the **Toolpaths Manager**.
- ♦ Select the **Post selected operations** icon from the **Toolpaths Manager** as shown.





- ♦ In the Post processing window, make the necessary changes as shown in [Figure: 18.0.1](#).

Figure: 18.0.1



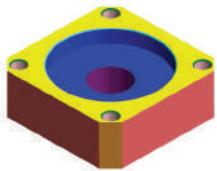
NC file enabled allows you to keep the NC file and to assign the same name as the MCAM file.

Edit enabled allows you to automatically launch the default.

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



- ♦ Save the NC file.



SAVE THE UPDATED MCAM FILE

TUTORIAL #1

- ♦ A window with **Mastercam Code Expert** will be launched and the NC program will appear as shown in [Figure: 18.0.2](#).

Figure: 18.0.2

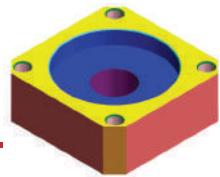
The screenshot shows the Mastercam 2018 Code Expert interface. The title bar reads "Your Name_.INC - Mastercam 2018 Code Expert". The menu bar includes File, Home, View, and NC Functions. The toolbar has buttons for Insert Block Numbers, Insert Block Skip, Remove Block Numbers, Remove Block Skip, Remove Spacing, Remove Comments, Send File, Receive, Sync, Tools, Multi-Stream, and Configuration. The main area displays an NC program:

```
1 *  
2 OOOOO (YOUR NAME_1)  
3 (DATE=05-09-17 TIME=09:00 - 15:34)  
4 (NC FILE = \\THSREVER06\BOCNCIEV\PRODUCTION\BOOKS\2018\TRAINING TUTORIALS\IT MILL ESSENTIALS\MCM FILES\TUTORIAL TOOLPATH INCH\TUTORIAL #1.MCAM)  
5 (NC FILE - C:\USERS\BINGYOU.MAO\Desktop\YOUR NAME_1.NC)  
6 (MATERIAL - ALUMINUM INCH - 2024)  
7 ( T1 3" FACE MILL | H6 )  
8 ( T2 3/4 FLAT ENDMILL | H2 )  
9 ( T3 1/4 FLAT ENDMILL | H3 )  
10 ( T4 3/8 SQUARING H4 )  
11 ( T5 1/8 DRILL | H8 )  
12 ( T6 1/2 CHAMFER MILL | H6 )  
13 ( T7 1/2 FLAT ENDMILL | H7 )  
14 N100 G20  
15 N110 G0 G17 G40 G69 G80 G90  
16 ( FACE THE TOP OF THE PART. )  
17 N120 T1 M6  
18 N130 G0 G50 G54 X-3.45 Y-7599 A0. S1065 M3  
19 N140 G43 R1 Z2.  
20 N150 Z3.  
21 N160 G1 Z0. F1.09  
22 N170 X2.45  
23 N180 G2 X3.1999 Y0. Z0. J-.7499  
24 N190 X2.45 Y-.7489 I-.7499 J0.  
25 N200 G1 X-3.45  
26 N210 G0 Z2.  
27 N220 M5  
28 N230 G91 G21 Z0.  
29 N240 A0.  
30 N250 M01  
31 ( MACHINE THE POCKET USING CIRCLE MILL. )  
32 N260 T2 M6  
33 N270 G0 G50/G54 X.2434 Y-.1338 A0. S713 M3  
34 N280 G43 R2 Z2.  
35 N290 Z1.  
36 N300 G0 X.3375 Y0. Z.0065 I-.2434 J-.1338 F6.42  
37 N310 X0. Y.3375 Z.0587 I-.3375 J0.  
38 N320 X-.3375 Y0. Z.0309 I0. J-.3375  
*** M0 T0 G0 G40 G41 G42 G43 G44 ***
```

At the bottom left is a "Find Extents" button, and at the bottom right are zoom controls (12.28KB, 100%, - +).

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

STEP 19: SAVE THE UPDATED MCAM FILE



REVIEW EXERCISE - STUDENT PRACTICE

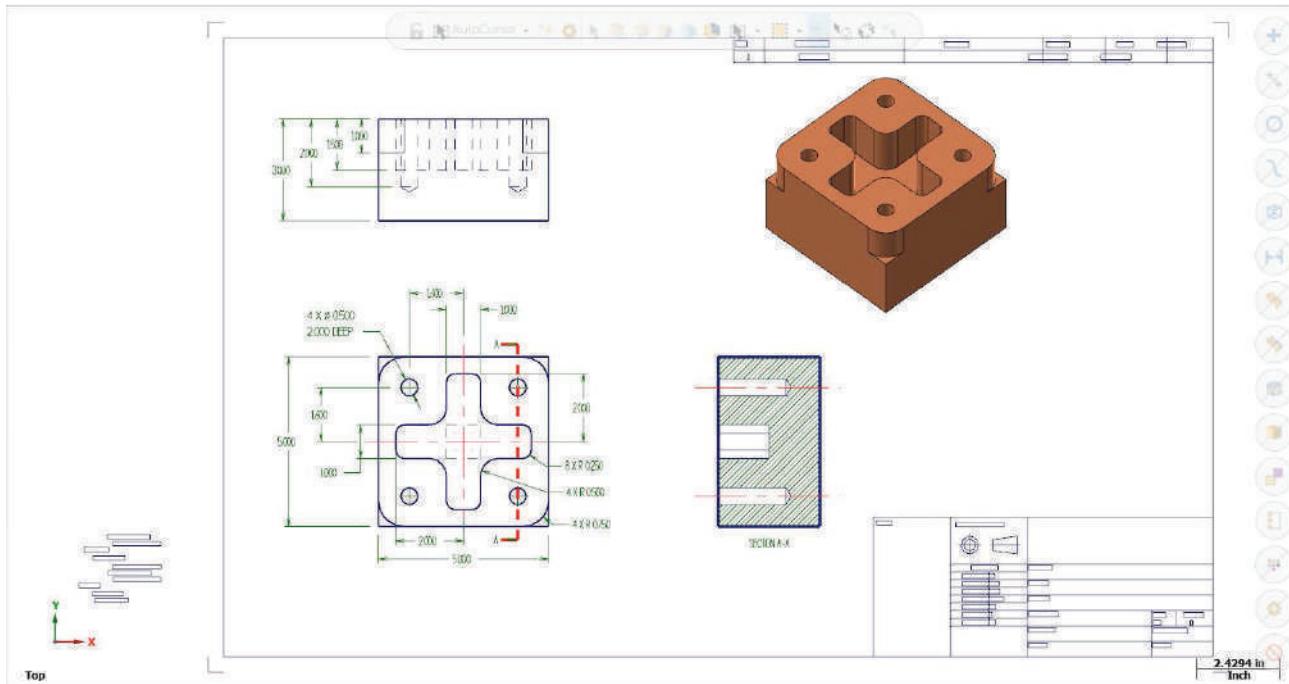
IMPORT THE DWG FILE FOR TUTORIAL #1 EXERCISE

Download the files from www.emastercam.com/trainingfiles.

Save the file in a preferred location.

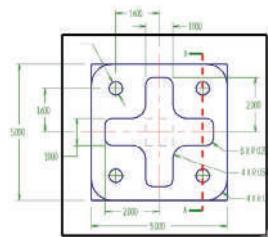
Import TUTORIAL_1 EXERCISE.DWG.

- ♦ Use file **Open** and change the file type to AutoCAD (*.dwg; *dxf...).

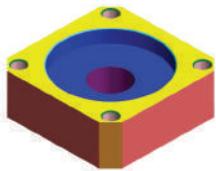


Move the geometry to level 100.

- ♦ Create the **Main Level 100** and name it **Geometry**.
- ♦ Use the **Select Only Advanced** option to select color blue number **1** and make a window around the top view shown as the rectangle with thicker lines.
- ♦ Right mouse click on the graphics window and select the **Change Level** button to move the geometry on **Level 100**.



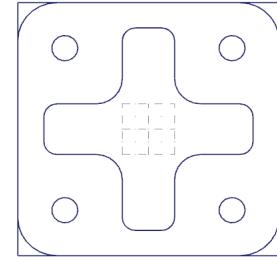
Make **Level 100** the only visible level.



- ♦ In the **Levels Manager**, right mouse click on the **Level 100** and select **All level set off**.

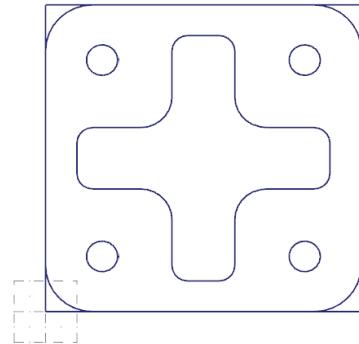
Check for any chaining problem.

- ♦ Use **Delete Duplicates** to delete the identical entities.
- ♦ Use **Findoverlap C-Hook** to delete overlapping entities.



Move the lower corner into the origin.

- ♦ Use **Transform Move to Origin** and select the endpoint of the entity at the lower left corner.



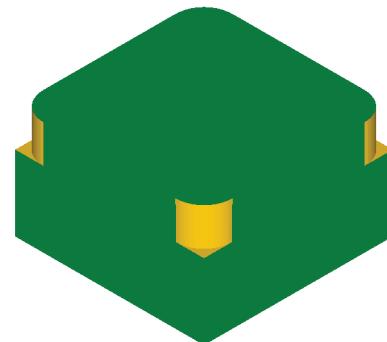
CREATE THE TOOLPATHS FOR TUTORIAL #1 EXERCISE

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

Set the machine properties including the stock.

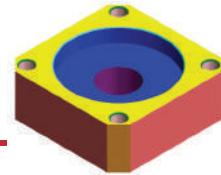
Remove the material on the outside of the part Contour (2D).

- ♦ Use a **1/2" Flat Endmill**.
- ♦ Based on your chaining direction, ensure the **Compensation direction** is set correctly.
- ♦ Enable **Depth Cuts** and set the **Max rough step** to **0.25"**.
- ♦ **Lead In/Out** set **Length** and **Radius** to **60%** with a **90** degree sweep.
- ♦ **No Break Through, Multi Passes**. See page 91 for details.
- ♦ Set the depth according to the drawing.



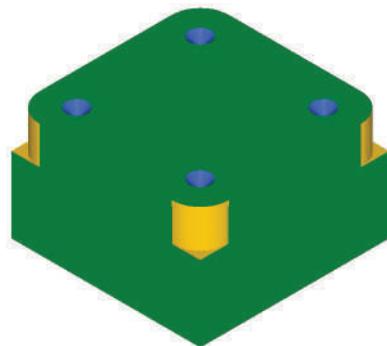
TUTORIAL #1

CREATE THE TOOLPATHS FOR TUTORIAL #1 EXERCISE



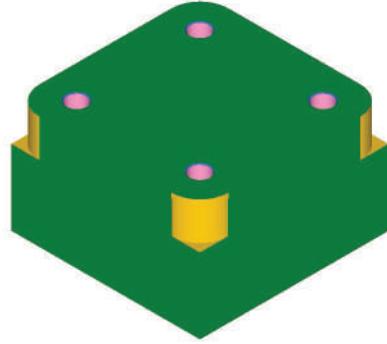
Spot drill the holes.

- ◆ Use a **3/4"** Spot Drill.
- ◆ Set the **Cycle** to **Drill/Counterbore** and set a **Dwell** to **1.0** second.
- ◆ Use the depth calculator to set a **0.05"** chamfer on the hole.



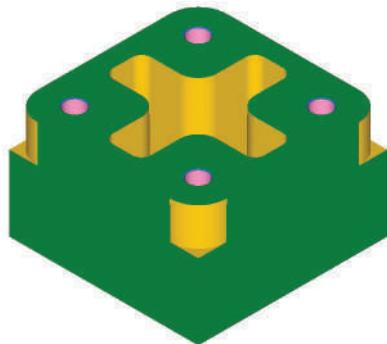
Drill the holes.

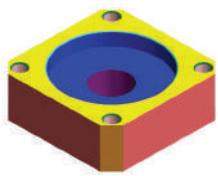
- ◆ Use a **1/2"** Drill.
- ◆ Set the **Cycle** to **Peck Drill** and set your peck values.
- ◆ Set the depth according to the drawing.



Remove the material in the center of the part (Pocket Standard).

- ◆ Use a **1/2"** Flat Endmill.
- ◆ Choose to leave no stock on the walls.
- ◆ In the **Roughing** parameters, set the **Cutting method** to **Constant Overlap Spiral**.
- ◆ The **Entry Motion** will be **Helix**.
- ◆ Enable **Finishing** and set the parameters.
- ◆ **Lead In/Out** set **Length** and **Radius** to **60%** with a **90** degree sweep.
- ◆ Enable **Depth Cuts** and set the **Max rough step** to **0.25"**.
- ◆ Disable **Break Through**.
- ◆ Set the depth according to the drawing.

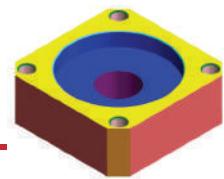




NOTES:

TUTORIAL #1

NOTES:

**TUTORIAL #1 QUIZ**

♦ What is a Contour Toolpath used for?

♦ What is a Facing Toolpath used for?

♦ What does Backplot do?

♦ What does Verify allow you to do?

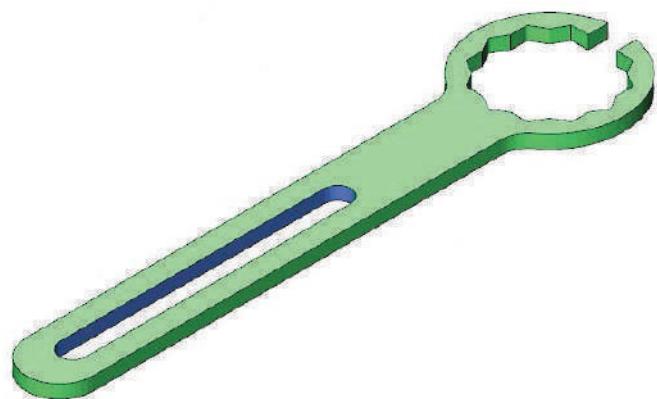


Automatically sign me into this document in the future.

(Do not select this when using a public computer)

Document ID: 0000-156C-204CF-00021E37

TUTORIAL #2



**OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:****Import the 2D CAD Model and prepare it to generate the Toolpaths from:**

- ◆ The student will open the AutoCAD DWG file in Mastercam.
- ◆ The student will move the geometry needed to create the toolpaths on a different level.
- ◆ The student will check for problems in the geometry.
- ◆ The student will move the geometry to set the origin to the lower left corner.
- ◆ The student will change the geometry color.

Create the necessary Toolpaths to machine the part:

- ◆ The student will set up the stock size to be used and the clamping method used.

Setup 1

- ◆ A Slot Mill toolpath will be created to machine the slot.
- ◆ 2D High Speed Dynamic Mill toolpath will be created to rough out the outside profile.
- ◆ A Contour toolpath will be created to finish the outside profile.
- ◆ 2D High Speed Dynamic Contour Mill toolpath will be created to machine the small radii.

Setup 2

- ◆ A Facing toolpath will be used to face the bottom of the part.

Backplot and Verify the file:

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

Post Process the file to generate the G-code:

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



This tutorial takes approximately one hour to complete.



GEOMETRY CREATION

STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

Please refer to the **Getting Started** section to set up the graphical user interface. Ensure the grid is set to inch.

STEP 2: IMPORTING THE DWG FILE GEOMETRY

Mastercam lets you read (import) a variety of CAD file types into the Mastercam database. You can also write (export) Mastercam files to a variety of different file formats.

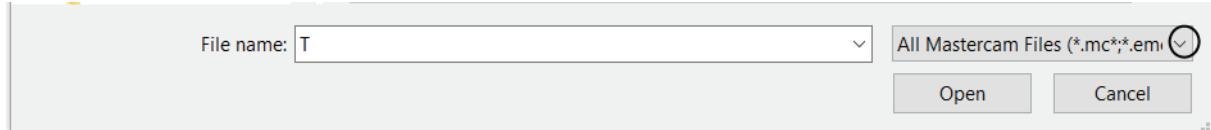
To import an AutoCAD file in Mastercam, you have to use the Open function and then select from the list of file types the AutoCAD files.

Download the files from www.emastercam.com/trainingfiles.

Save the file at a preferred location.

FILE

- ♦ Open.
- ♦ In the file name extension, click on the drop down arrow as shown.



- ♦ From the list of file types, select **AutoCAD Files (*.DWG;*.DXF;*.DWF;*.DWFX)** as shown.

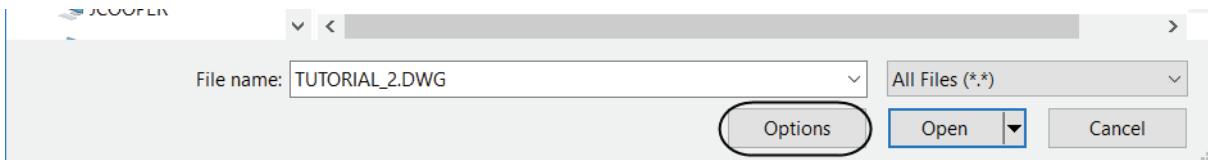
Mastercam Educ X7 Files (*.emcx-7)
Mastercam Educ X6 Files (*.emcx-6)
Mastercam Educ X5 Files (*.emcx-5)
Mastercam Educ pre-X5 Files (*.emcx)
Mastercam V9 Files (*.mc9)
Mastercam V8 Files (*.mc8)
All Mastercam Files (*.mc*;*.emc*)
IGES Files (*.igs;*.iges)
AutoCAD Files (*.dwg;*.dxf;*.dwf;*.dwfx)
Parasolid Files (*.x_t;*.x_b;*.xmi;*.txt)
ProE/Creo Files (*.prt;*.asm;*.prt;*.asm;*.asm;*.asm)
ACIS Kernel SAT Files (*.sat;*.sab)
STEP Files (*.stp;*.step)
VDA Files (*.vda)
Rhino 3D Files (*.3dm)
SOLIDWORKS Files (*.sldprt;*.sldasm)
SOLIDWORKS Drawing Files (*.slddrw)
Solid Edge Files (*.par;*.psm;*.asm)
Autodesk Inventor Files (*.ipt;*.iam)
Autodesk Inventor Drawing Files (*.idw)
KeyCreator Files (*.ckd)

TUTORIAL #2

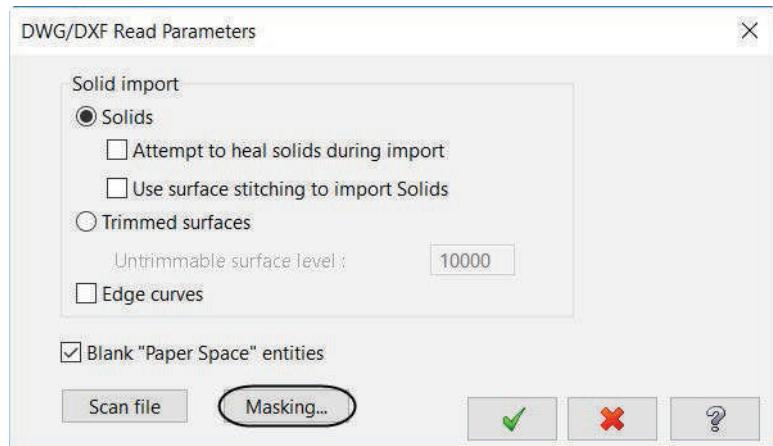
IMPORTING THE DWG FILE GEOMETRY



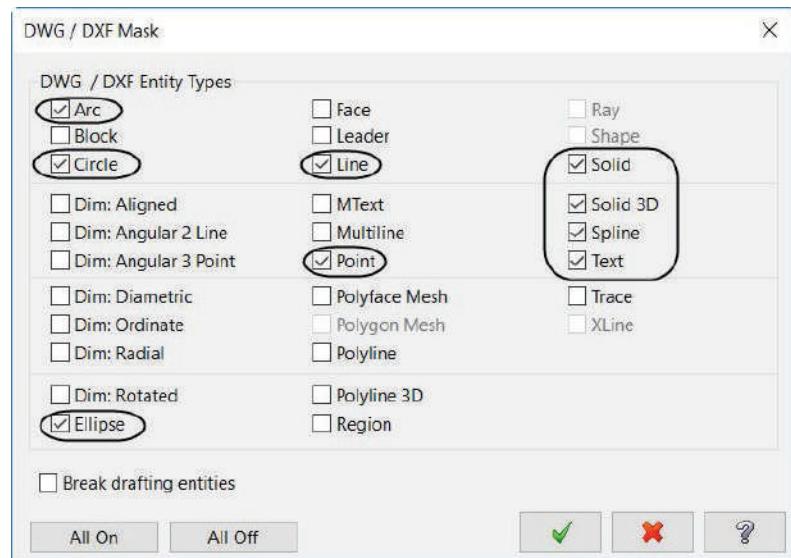
- ♦ Find and select **TUTORIAL_2.DWG**.
- ♦ Select the **Options** button to filter the type of entities that you bring in.



- ♦ From the **DWG Read Parameters** dialog box, select the **Masking** button as shown.



- ♦ Make sure that you only have the following entity types selected to avoid bringing in unnecessary entities.



NOTE: Select the **Help** button to view descriptions of all the entity types in this image.



- ♦ Select the **OK** button to exit **DWG/DXF Mask**.



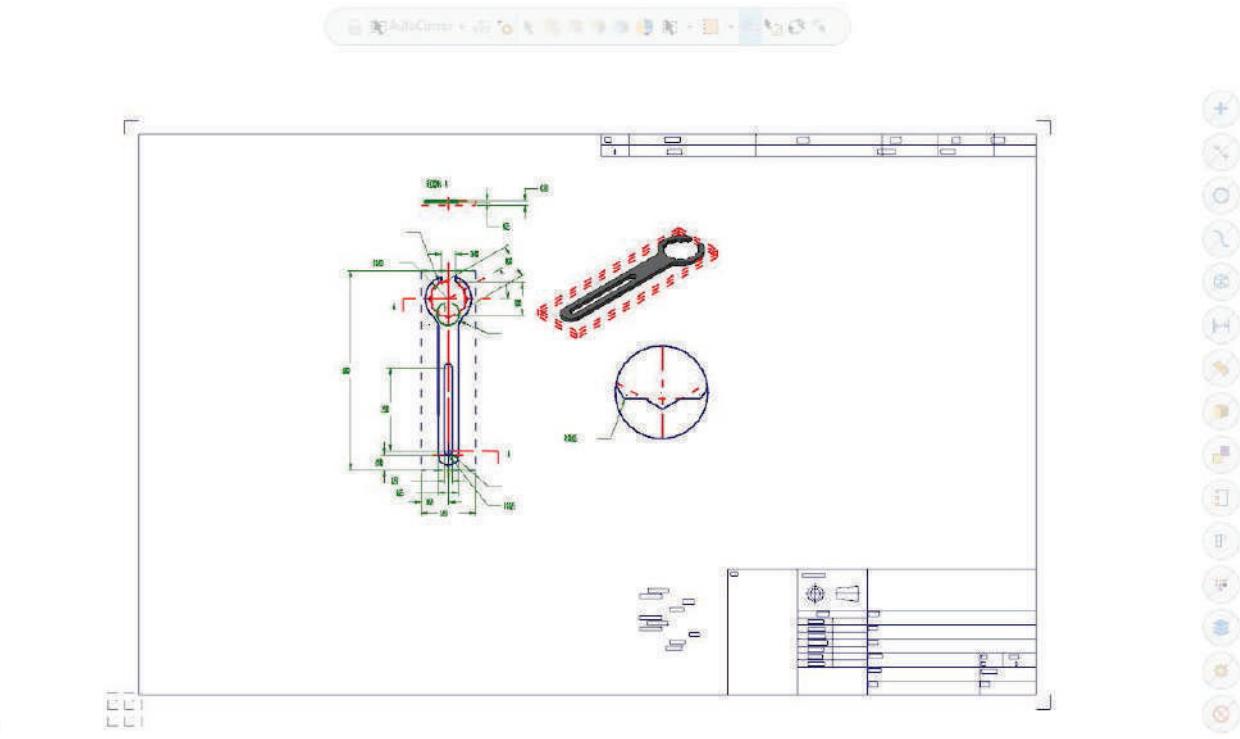
- ♦ Select the **OK** button to exit **DWG/DXF Read Parameters**.



- ♦ Open the file.



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



NOTE: To better display the geometry in the book, the background color in Mastercam is set to white and the title box and the geometry are in black color. On your screen you should see the default Mastercam background which is a grey gradient color and the geometry will be shown in white.

The dimensions, title block, and some of the geometry are not required for machining purposes. To be able to see only the geometry needed for toolpath creation, you will move it to a different level. This will be covered in the following steps of this tutorial.

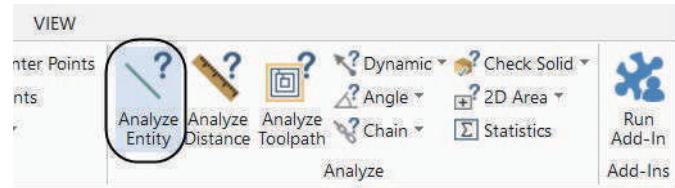


STEP 3: ANALYZE THE GEOMETRY

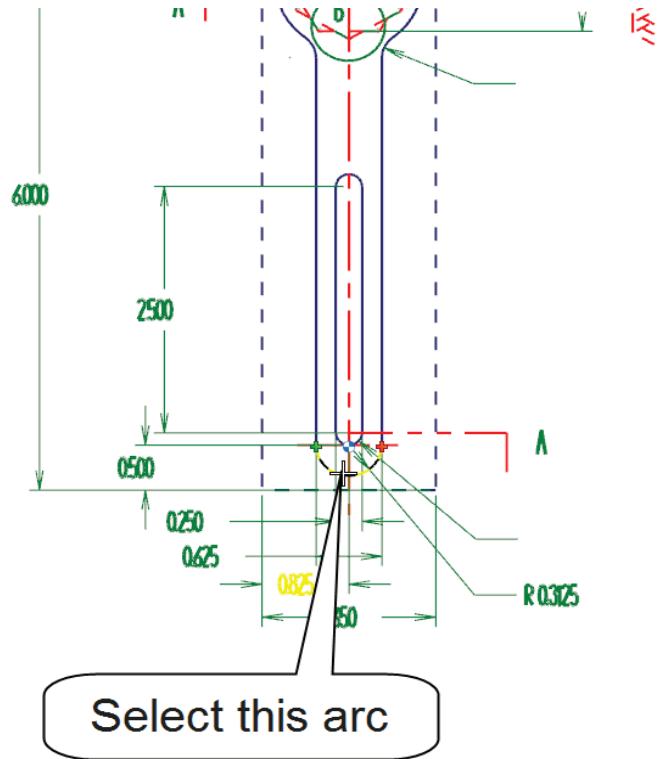
To machine this part, the geometry has to be created at a 1:1 scale. In order to check the geometry's accuracy, you will analyze one of the entities that is dimensioned and check if the value is the same as the one in the drawing.

HOME

- From the **Analyze** group, select **Analyze Entity**.

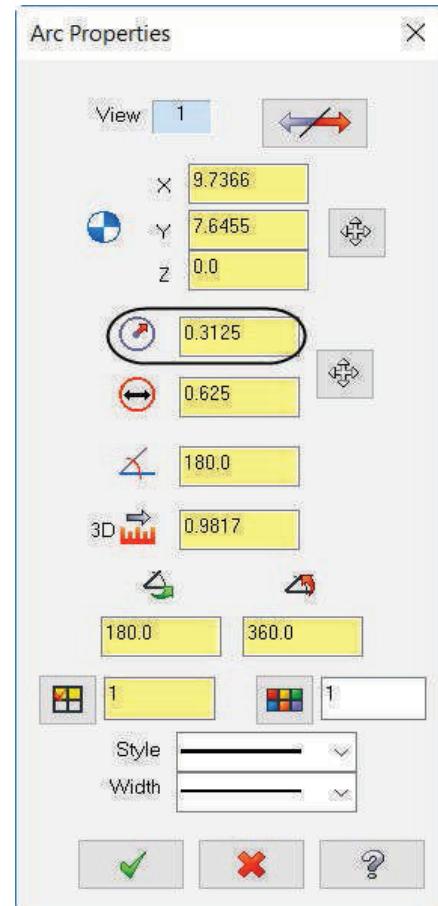


- [Select entities to analyze]: Hover the mouse above the top view and scroll up the mouse wheel to zoom in and select the arc as shown.





- In the **Arc Properties**, the **radius** is **0.3125** as shown.



NOTE: The geometry was created at the proper scale 1:1. If the entity value was different from the dimension value, you can use the **TRANSFORM/Scale** command to increase or decrease the size of the entities by a factor relative to a defined point using the uniform method.

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





STEP 4: MOVE THE TOOLPATH GEOMETRY TO A DIFFERENT LEVEL

Levels are a primary organizational tool in Mastercam. A Mastercam file can contain separate levels for wireframe, surfaces, drafting entities, and toolpaths. By organizing your files into levels, you can more easily control which areas of the drawing are visible at any time and which parts are selectable so that you do not inadvertently make changes to areas of the drawing you do not want to change.

In this step you will select the geometry required for the toolpath and move it to a different level.

4.1 Set the Main Level and name it

The **Main Level** is the current working level. Any geometry that you create is always placed on the main level. There can be only one main level at a time. The number of the main level appears on the **Levels** button in the **Status** bar. In the **Levels Manager** dialog box, the main level is highlighted in yellow.

- From the bottom left corner of the screen, click on the **Levels** tab as shown.



- Change the **Number** in the **Main Level** to **100** and enter the name **Geometry** as shown.





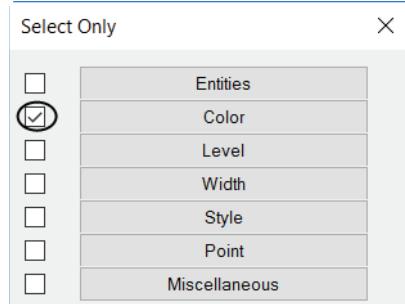
4.2 Move the geometry to the Main Level 100

NOTE: To select the geometry you want to keep, use the window selection, a default selection method in Mastercam. In order to create the window, simply click and drag until the window covers the selected part completely. In order to only select the desired geometry lines, enter the **General Selection** toolbar and select only the color blue geometry inside of the window.

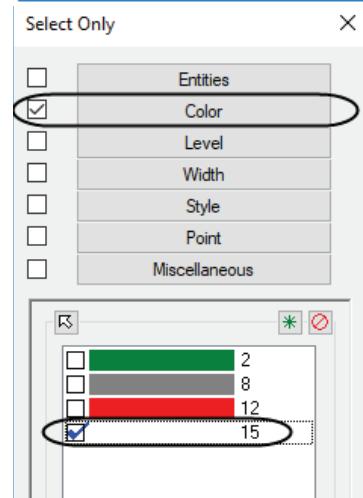
- From the **Quick Mask** buttons, choose **Select only advanced** as shown.



- In the **Select Only** dialog box, enable **Color** as shown.



- Click on the **Color** button and enable color white number **15** as shown.



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



NOTE: The **Only** button in the **General Selection** is highlighted. It will stay like this while the color mask is on.

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

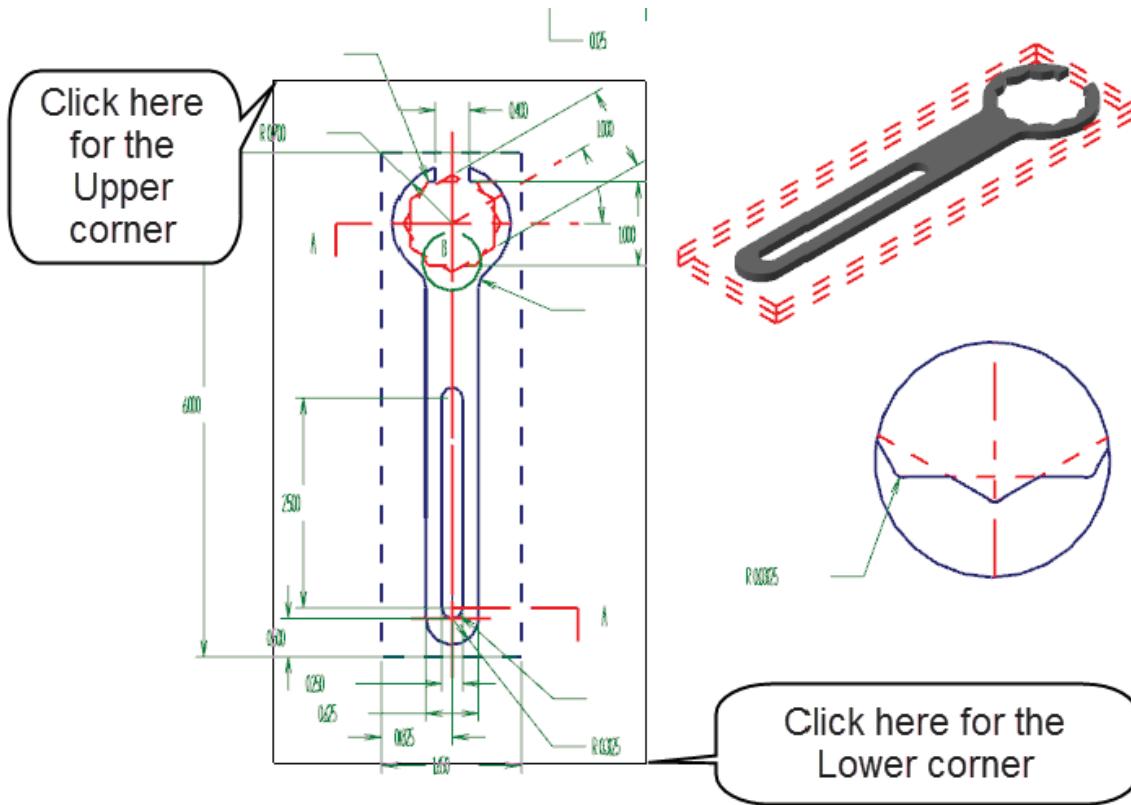
TUTORIAL #2

MOVE THE TOOLPATH GEOMETRY TO A DIFFERENT LEVEL



- ♦ To zoom in, move the cursor in the center of the top view and scroll the mouse wheel upwards.
- ♦ Select the geometry using window selection.
- ♦ Select the upper corner and the lower corner of the window as shown in [Figure: 4.2.1](#).

Figure: 4.2.1



NOTE: The entities selected will be shown with hidden lines.

- ♦ Right mouse click on the graphics window and select **Change Level** as shown.





- The Change Levels dialog box should appear on the screen as shown.



NOTE: The parameters in the **Change Levels** dialog box are already set to move the geometry to the **Main Level**.

- Leave the default settings in the **Change Levels** and select the **OK** button to continue.



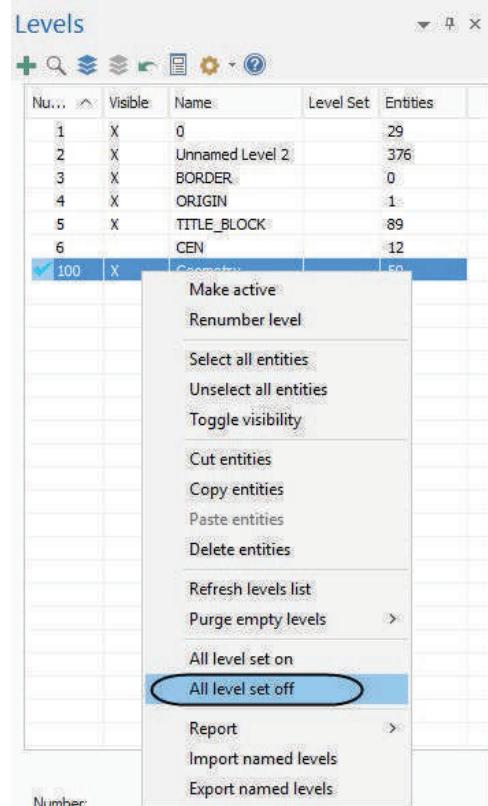
TUTORIAL #2

MOVE THE TOOLPATH GEOMETRY TO A DIFFERENT LEVEL

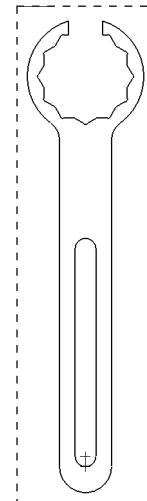


4.3 Make the Main Level the only visible level

- In the **Levels Manager**, right mouse click on the number **100** and select **All level set off** to make all the levels invisible except the **Main level** as shown.



- Press **Alt + F1** to fit the geometry in the graphics window.

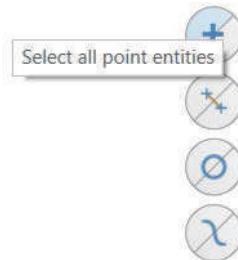


- Hover the cursor close to the center of the geometry and scroll down the mouse wheel to unzoom the part slightly.

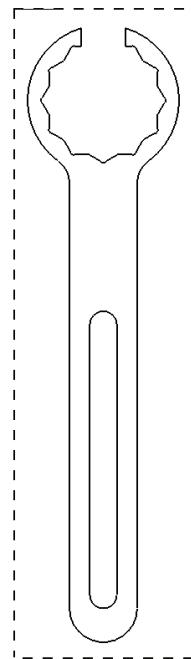


4.4 Delete the extra geometry

- From the right side of the graphics window, choose **Select all point entities** as shown.



- Press **Delete** from the keyboard.
- The geometry should look as shown.



NOTE: When you import geometry from a different CAD software, before starting to make any toolpath, you should check for any problems in the geometry that will be used in the toolpath.



STEP 5: CHECKING FOR CHAINING PROBLEMS

Chaining is a process used to select a number of entities (lines, arcs, points, splines, etc.) when creating a toolpath. The entities form a chain when the distance between two consecutive entity endpoints is less than the chaining tolerance. The following steps illustrate the problems that can be encountered and the selection stops.

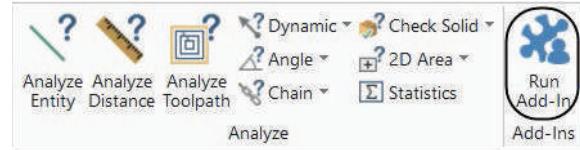
NOTE: Please check **Tutorial 1 page 26** to see how you can check and fix the problems in the geometry. In this example, the outside rectangle has overlapping entities which can be removed using Findoverlap.dll C-Hook.

5.1 Check for Overlapping Entities Using FindOverlap C-Hook

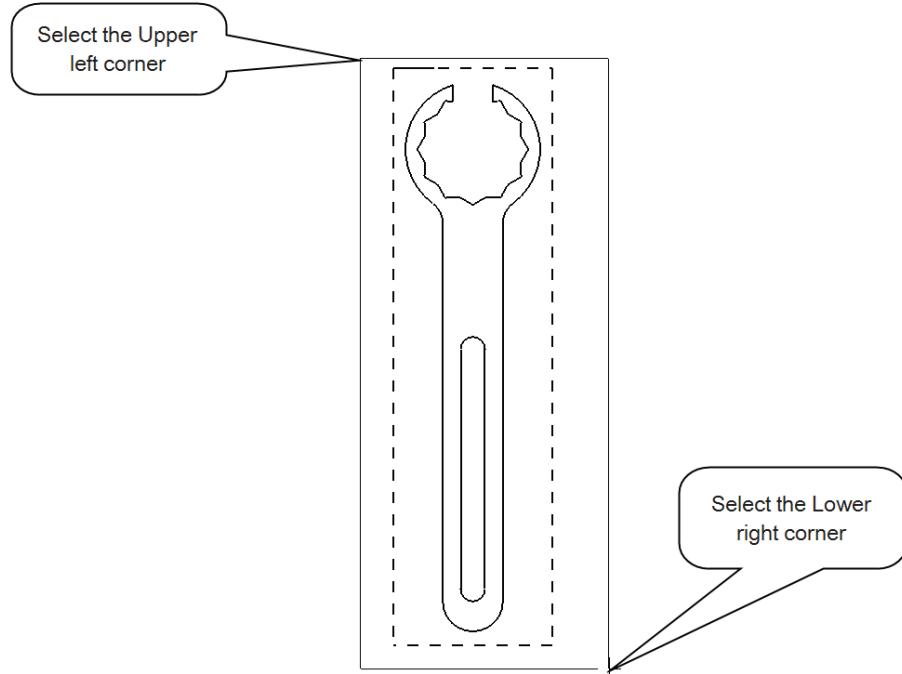
C-Hooks and NET-Hooks are add-in applications or utilities that customize, enhance, or extend Mastercam's functionality. FindOverlap C-Hook allows you to find, display and clean up overlapping lines and arcs. Overlapping entities can interfere with chaining and are often difficult to see and delete.

HOME

- ♦ To open the **C-hook** directory, from the **Add-Ins** group select **Run Add-In** as shown.

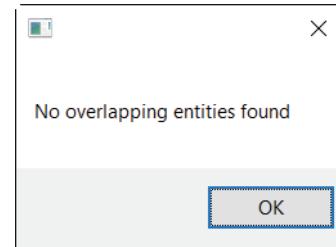


- ♦ Select the **FindOverlap.dll** and open it.
- ♦ [Select entities to check for overlap]: Make a window around the entire geometry as shown.





- ♦ Click on **End Selection** to finish the selection. 
- ♦ If no overlapping entities were found, select the **OK** button on the dialog box that states this.



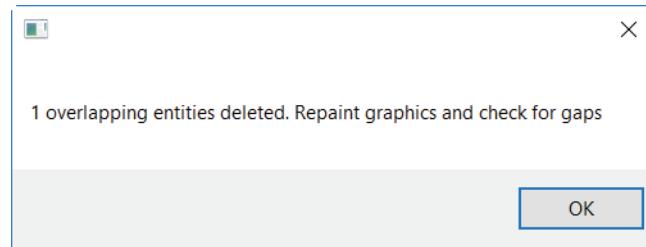
NOTE: You need to follow the next steps only if overlaps are found.

- ♦ Otherwise, the **FindOverlap C-Hook** dialog box should appear showing that the geometry has duplicate entities as shown in [Figure: 5.1.1](#).
- ♦ Click on the **Cleanup** button for the system to automatically remove them as shown in [Figure: 5.1.1](#).

Figure: 5.1.1



- ♦ This message should appear on the screen.



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



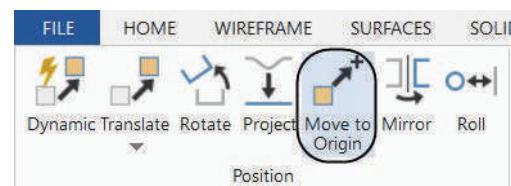
STEP 6: MOVE THE GEOMETRY TO THE ORIGIN

In this step you will use the **Move to Origin** command to quickly move all visible geometry to the current WCS (Work Coordinate System) origin based on a point that you select with the AutoCursor. This origin will be the origin you will set on your part at the machine.

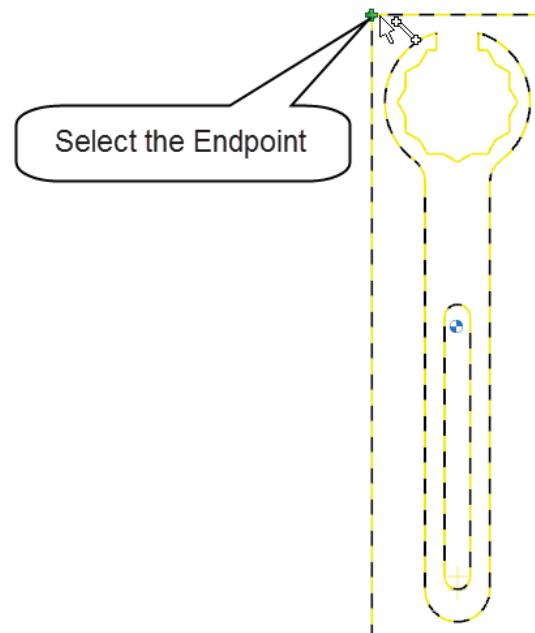
Mastercam uses a 3D Cartesian coordinate system to locate your work in three-dimensional space. This means that geometry and toolpath positions are expressed in terms of three coordinate axes: X, Y, and Z. Each axis is signed, which means that it has a positive and a negative direction.

TRANSFORM

- From the **Position** group, select **Move to Origin** as shown.

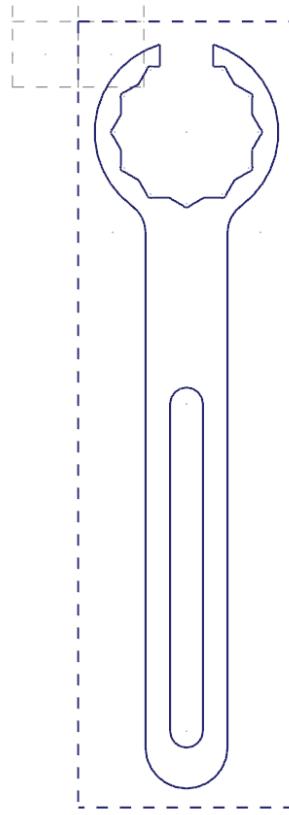


- [Select the point to translate from]: Select the Endpoint of the line as shown.

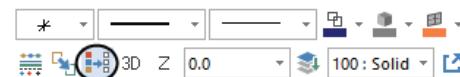




- ♦ Press **Alt + F1** to fit the geometry inside the graphics window.



- ♦ If the geometry is off-center, hold down the **Shift** key and with the mouse wheel pressed, drag the geometry approximately to the center of the graphics window. Hover the cursor near the center of the screen and scroll up to zoom in if needed.
- ♦ Right mouse click in the graphics window and select **Clear Colors** as shown to remove the result color.



STEP 7: SAVE THE FILE

FILE

- ♦ **Save As.**
- ♦ File name: "Your Name_2".

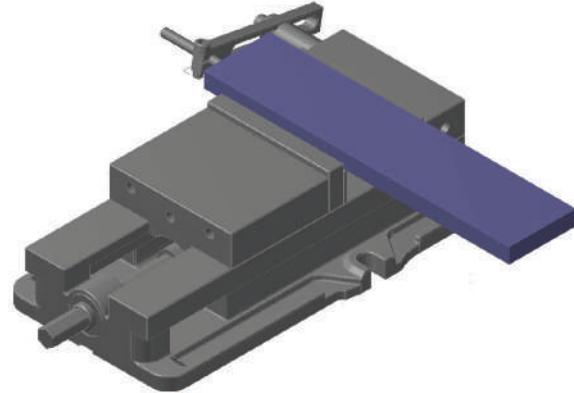
TUTORIAL #2

SUGGESTED FIXTURE:



TOOLPATH CREATION - SETUP 1

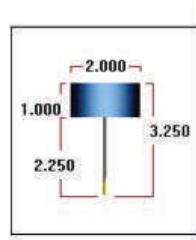
SUGGESTED FIXTURE:



NOTE: In order to machine this part, we will have 2 setups and output 2 NC files. To view the second setup, see page 182.

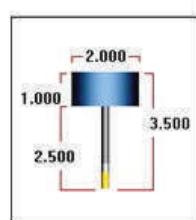
SETUP SHEET:

TOOL LIST



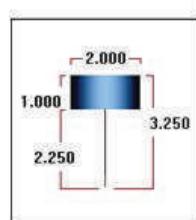
TYPE: Endmill1 Flat
DIA OFFSET: 1
HOLDER: Default Holder
NUMBER: 1
LENGTH OFFSET: 1
#1 - 0.1250 ENDMILL1 FLAT - 1/8 FLAT ENDMILL

FLUTE LENGTH: 0.375
OVERALL LENGTH: 2.25
CORNER RAD: 0.0
OF FLUTES: 4



TYPE: Endmill1 Flat
DIA OFFSET: 2
HOLDER: Default Holder
NUMBER: 2
LENGTH OFFSET: 2
#2 - 0.2500 ENDMILL1 FLAT - 1/4 FLAT ENDMILL

FLUTE LENGTH: 0.5
OVERALL LENGTH: 2.5
CORNER RAD: 0.0
OF FLUTES: 4



TYPE: Endmill1 Flat
DIA OFFSET: 3
HOLDER: Default Holder
NUMBER: 3
LENGTH OFFSET: 3
#3 - 0.0625 ENDMILL1 FLAT - 1/16 FLAT ENDMILL

FLUTE LENGTH: 0.375
OVERALL LENGTH: 2.25
CORNER RAD: 0.0
OF FLUTES: 4



STEP 8: SELECT THE MACHINE AND SET UP THE STOCK

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

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

VIEW

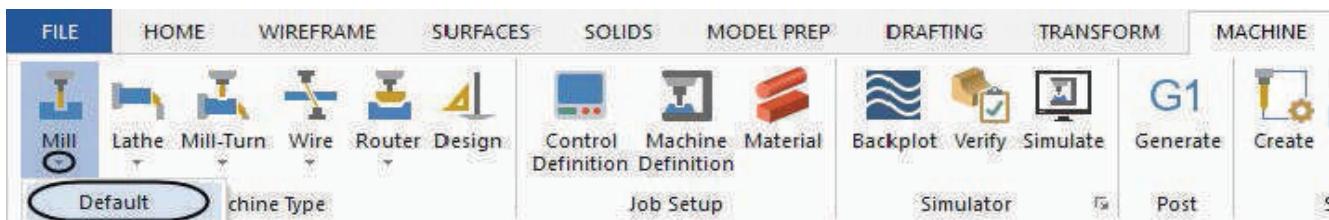
- ♦ To display the **Toolpaths Manager**, from the **Managers** group select **Toolpaths** as shown.



NOTE: Select the **Mill Default** only if there is no **Machine Group** in the **Toolpaths Manager**.

MACHINE

- ♦ From the **Machine Type** group, click on the drop down arrow below **Mill** and select the **Default**.



NOTE: Once you select the **Mill Default**, the ribbon bar changes to reflect the toolpaths that could be used with **Mill Default**.

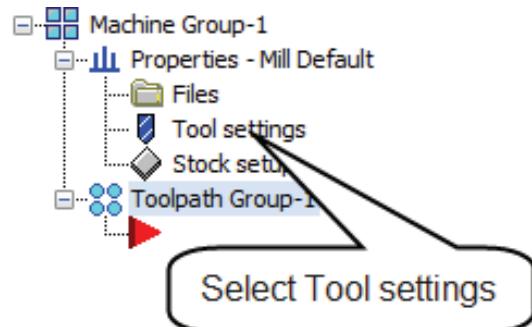
- ♦ Select the plus sign in front of **Properties** in the **Toolpaths Manager** to expand the **Toolpaths Group Properties**.



Select the plus sign

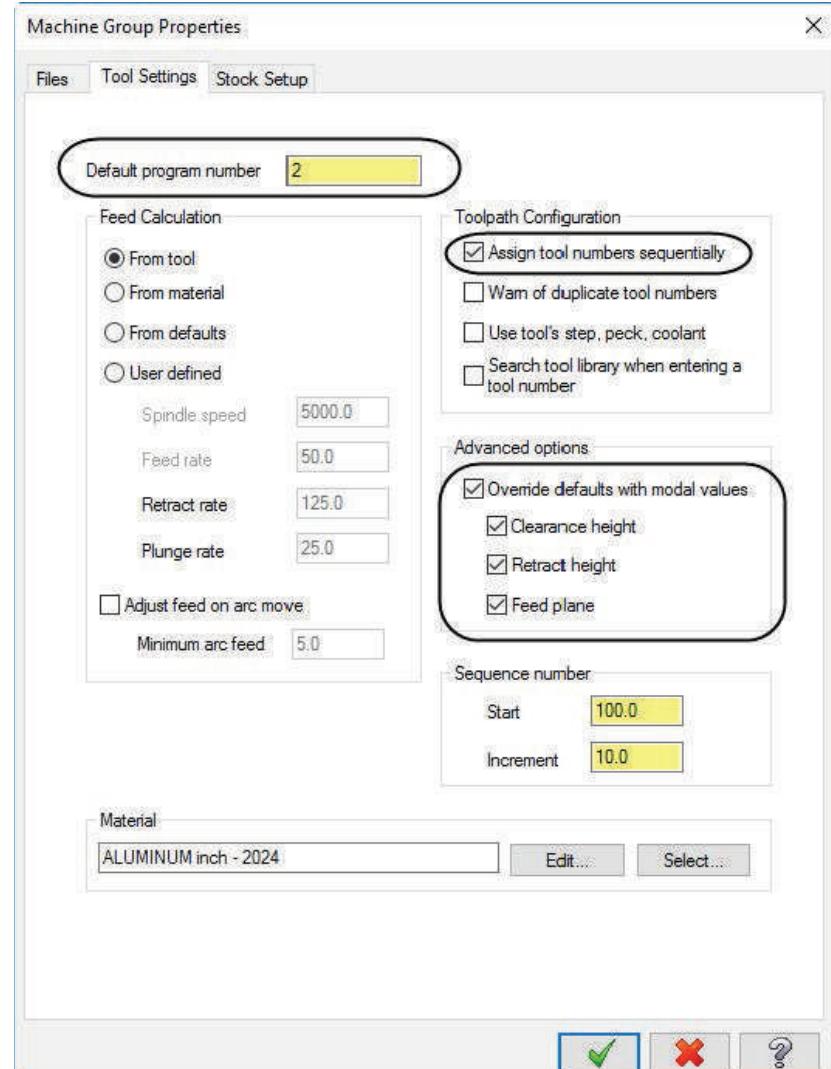


- ♦ Select **Tool settings** to set the tool parameters.



- ♦ Change the parameters to match the screen shot as shown in [Figure: 8.0.1](#).

Figure: 8.0.1



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

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

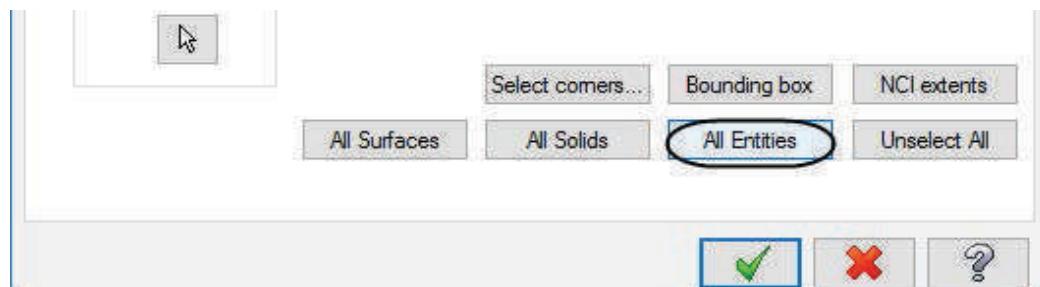
Warn of duplicate tool numbers allows you to get a warning if you enter two tools with the same number.

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

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

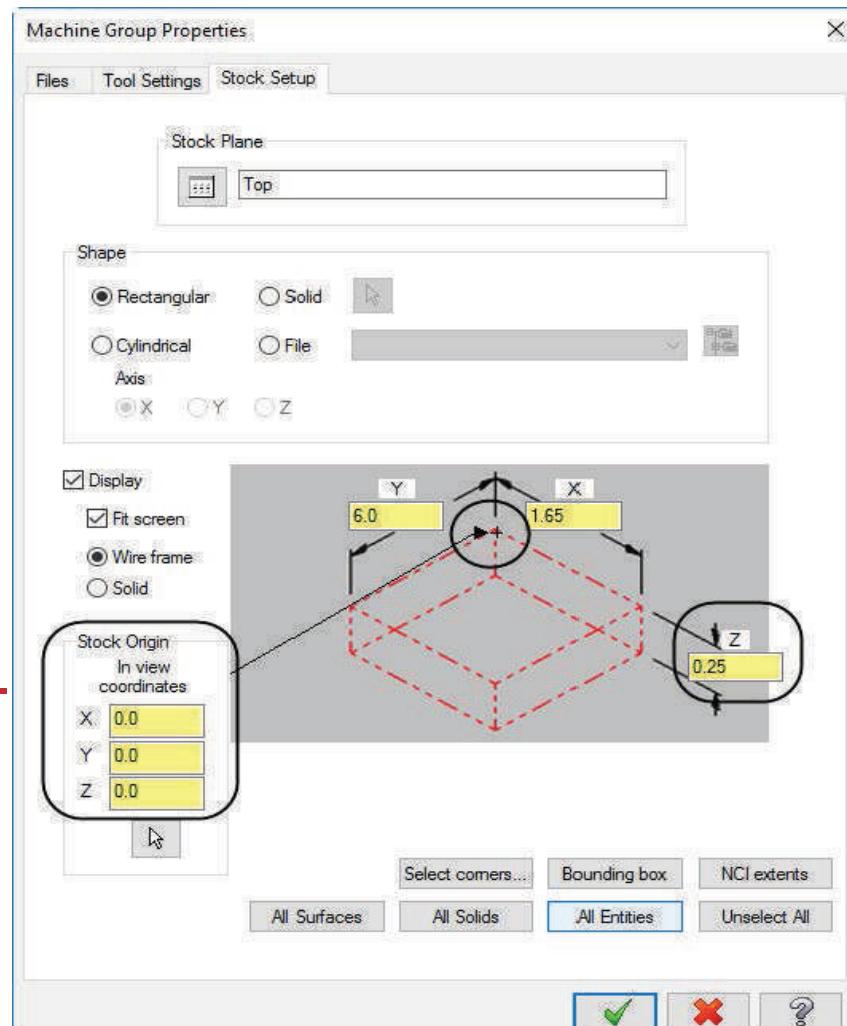


- ♦ Select the **Stock Setup** tab to define the stock.
- ♦ Select the **All Entities** button near the bottom of the **Stock Setup** page as shown.



- ♦ In the **Z** field, enter **0.25** as shown in [Figure: 8.0.2](#). This will add **0.125"** of stock on the bottom of the model.
- ♦ Click in the **Stock Setup window** at the upper left corner of the stock to move the arrow where the origin is set and then change the **Stock Origin** values to **0.0** as shown in [Figure: 8.0.2](#).

Figure: 8.0.2

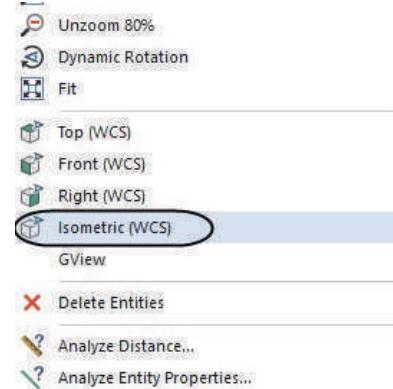


Stock Origin values adjust the positioning of the stock, ensuring that you have an equal amount of extra stock around the finished part.

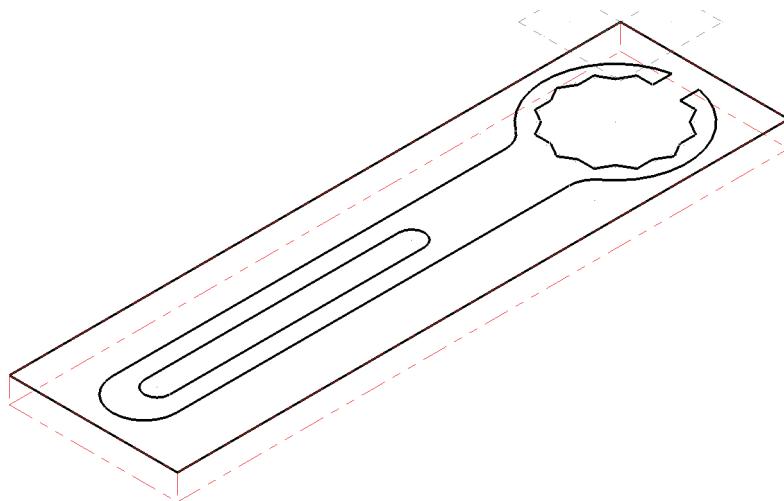
Display options allow you to set the stock as **Wireframe** and to fit the stock to the screen. (Fit Screen)



- ♦ Select the **OK** button to exit **Machine Group Properties**. 
- ♦ Right mouse click in the graphics window and select the **Isometric** view to see the stock.



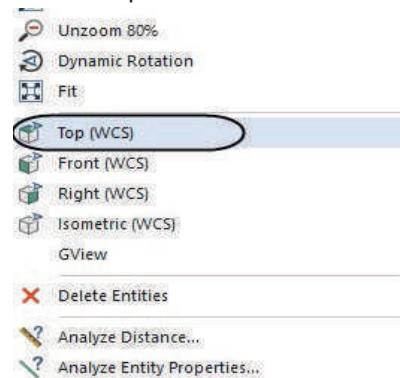
- ♦ Press **Alt + F1** to fit the drawing to the screen.
- ♦ The stock model will appear as shown.



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



- Right mouse click again and select the **Top** view from the list to see the part from the top.

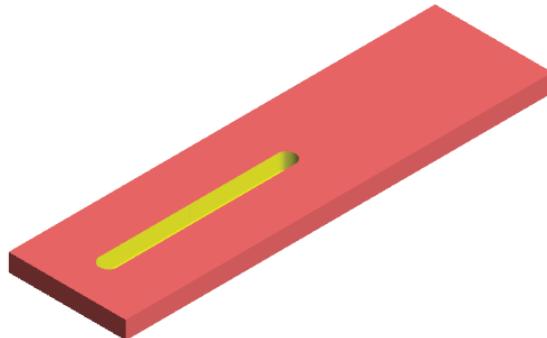


NOTE: There will not be a facing toolpath because the stock is already to size.

STEP 9: SLOT MILLING

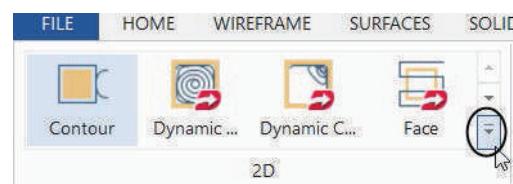
Slot Mill toolpath allows Mastercam to efficiently machine obround slots. These are slots that consist of 2 straight lines and two 180-degree arcs at the ends.

Toolpath Preview:



TOOLPATHS

- From the **2D** group, select the **Expand gallery** arrow as shown.

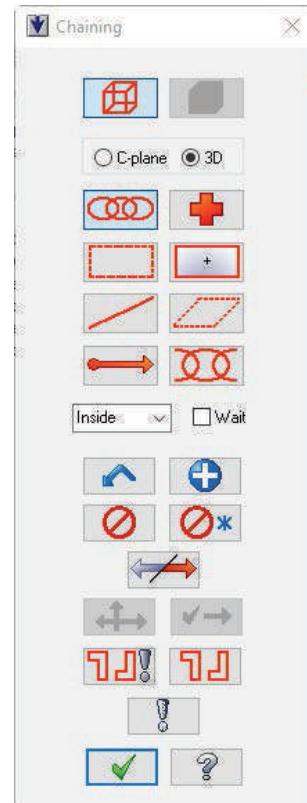




- ♦ Select the **Slot Mill** as shown.



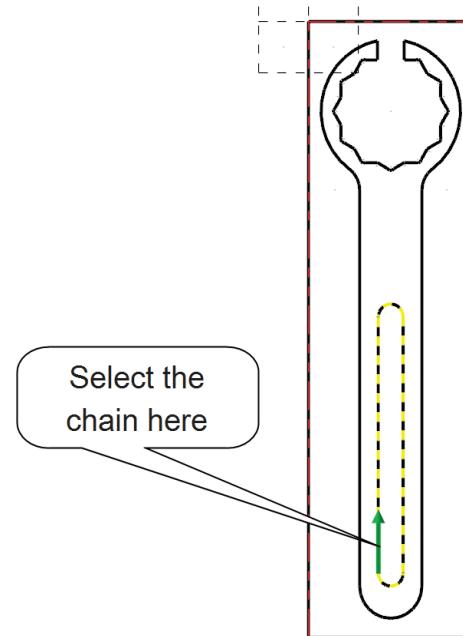
- ♦ When the **Chaining** dialog box appears, leave **Wireframe** and **Chain** as the only options selected.





- ♦ Select the chain as shown in [Figure: 9.0.1](#).

Figure: 9.0.1



- ♦ Choose the **OK** button to exit the **Chaining** dialog box.
- ♦ In the **Toolpath Type** page, the **Slot Mill** icon will be selected.



Contour



Pocket



Facing



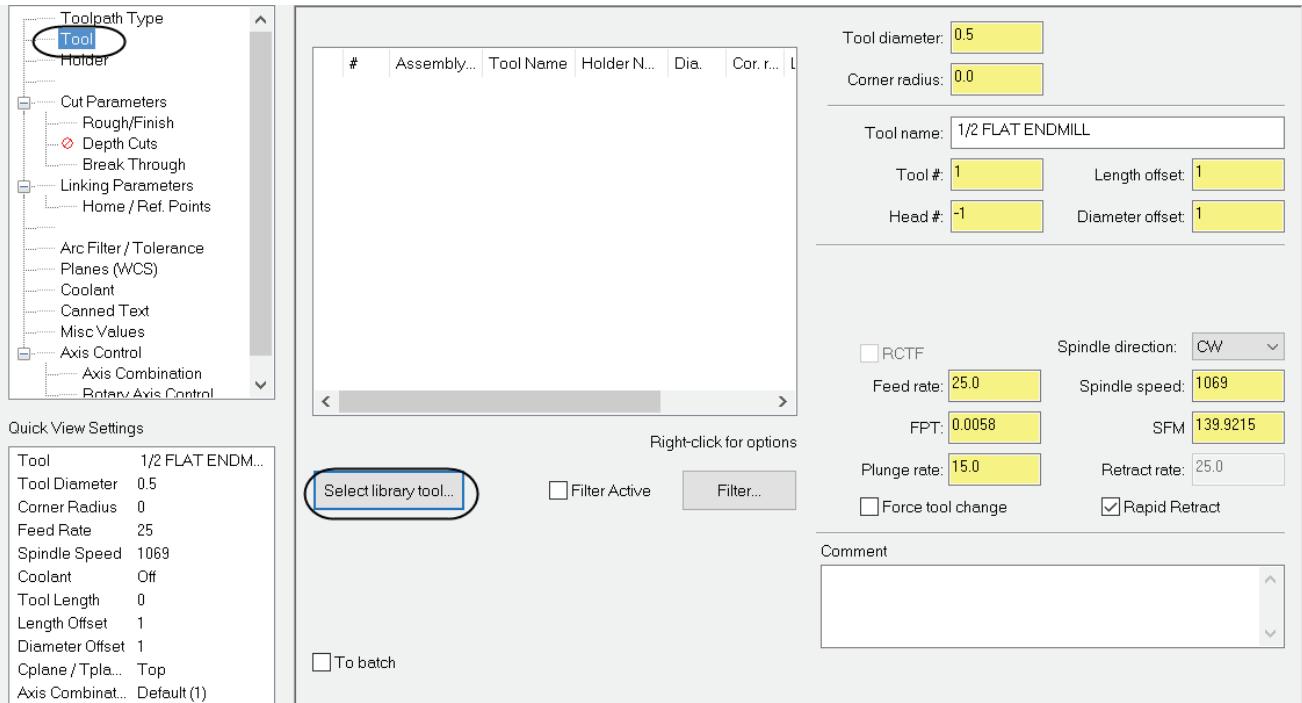
Slot Mill

NOTE: Mastercam updates the pages as you modify them and then marks them, in the **Tree View list**, with a green check mark. Pages that are disabled are marked with a red circle and slash.



9.1 Select a 1/8" Flat Endmill and set the Tool Parameters

- ♦ Select **Tool** from the **Tree View** area.
- ♦ Click on the **Select library tool** button.



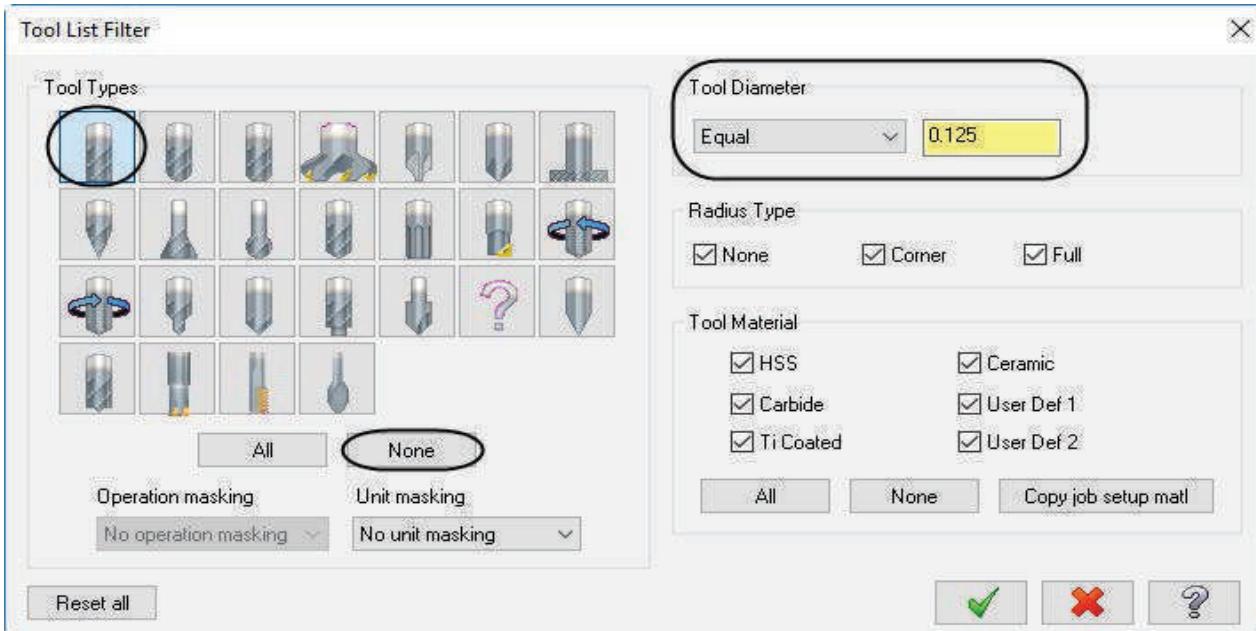
- ♦ Select the **Filter** button in the Tool Selection dialog box.





- ♦ Select the **None** button and then under **Tool Types**, choose the **Flat Endmill** icon.
- ♦ Under tool diameter, pick **Equal** and input a value **0.125** as shown in [Figure: 9.1.1](#).

Figure: 9.1.1



- ♦ Select the **OK** button to exit the **Tool List Filter**.
- ♦ In the **Tool Selection** dialog box, you should only see a **1/8" Flat Endmill**.

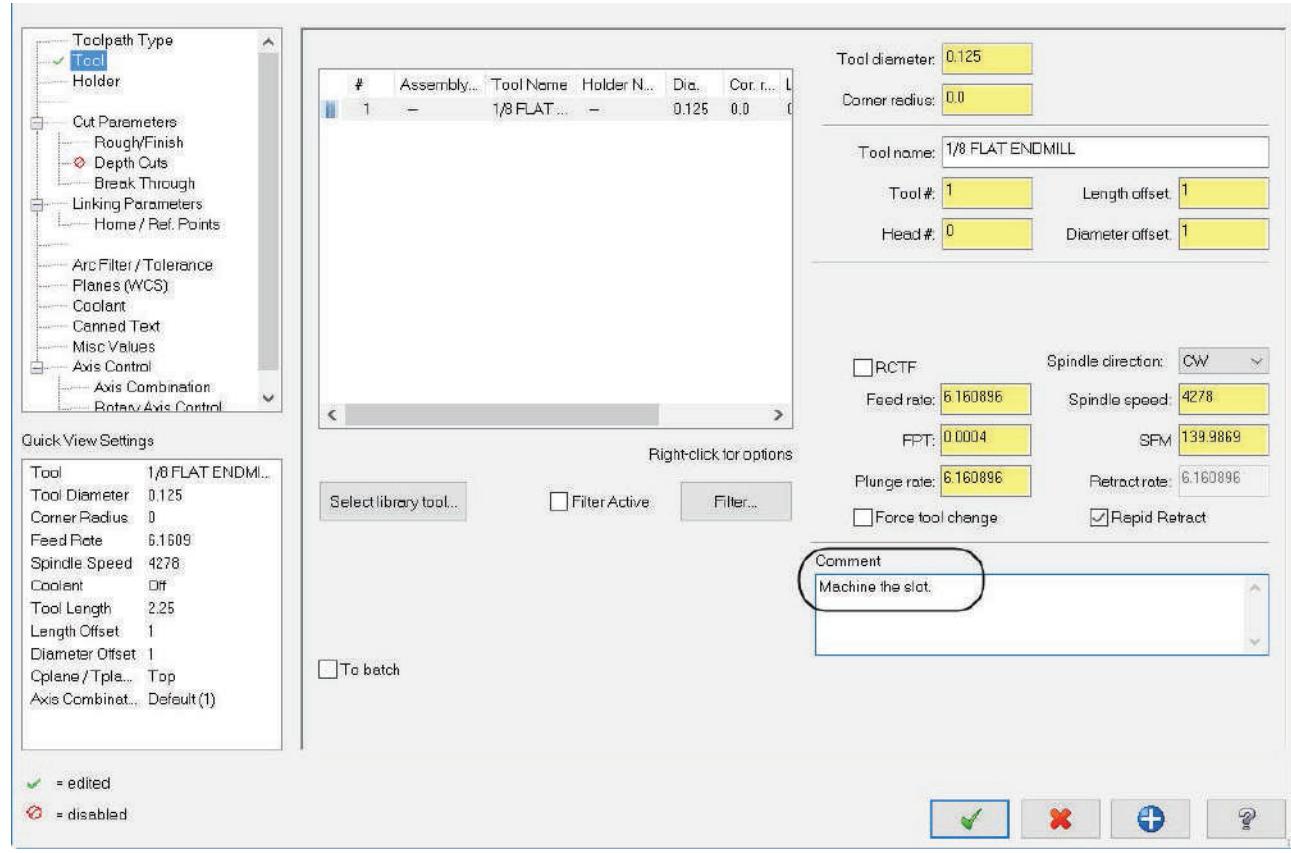
#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type	Ra...
282	-	1/8 FLAT ...	-	0....	0.0	0.375	4	En...	No...

- ♦ Select the **1/8" Flat Endmill** in the **Tool Selection** page and then select the **OK** button to exit.



- Make all the necessary changes as shown in [Figure: 9.1.2.](#)

Figure: 9.1.2



The **Feed rate**, **Plunge rate**, **Retract rate** and **Spindle speed** are based on the tool definition as set in the **Tool Settings**. You may change these values as per your part material and tools.

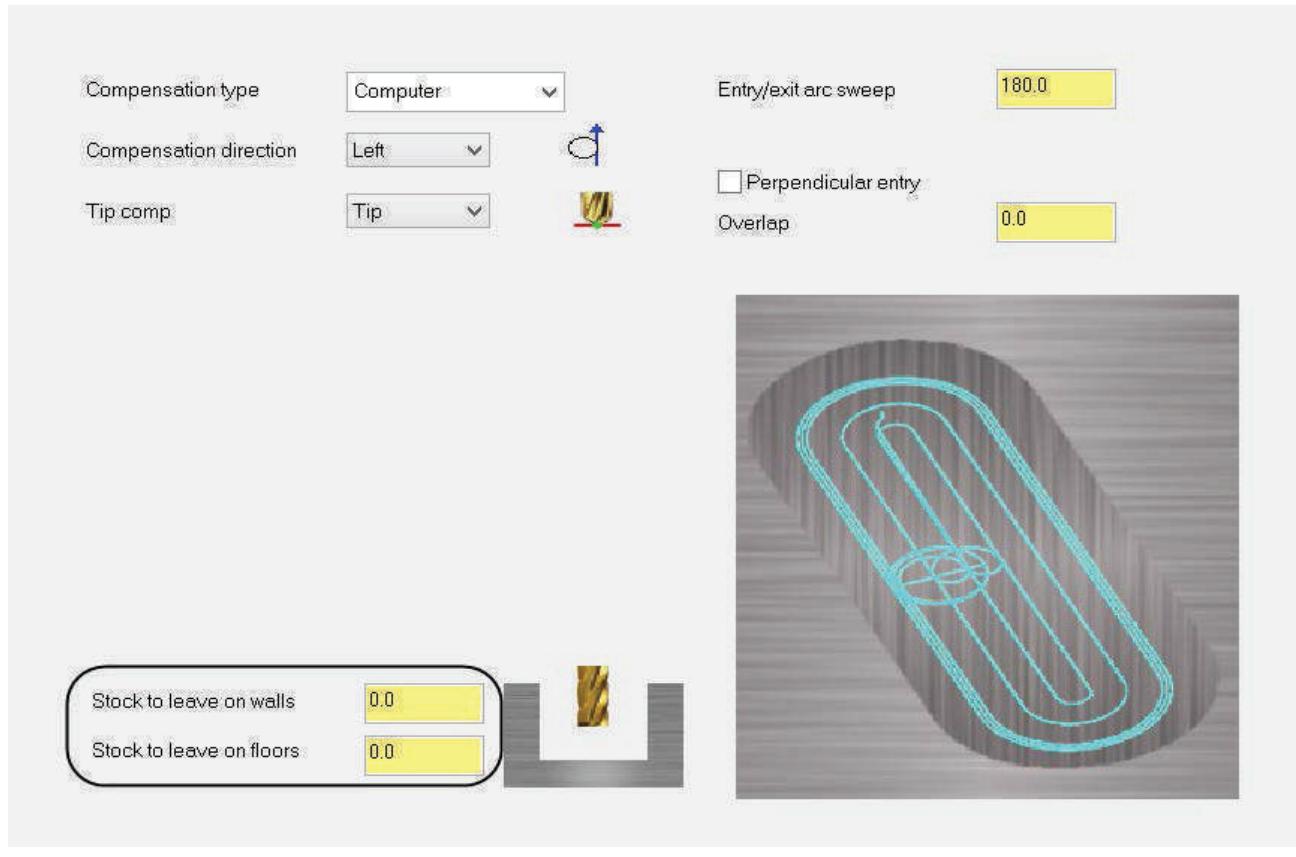
In the **Comment** field, enter a comment to help identify the toolpath in the **Toolpaths Manager** such as the one shown above.



9.2 Cut Parameters

- From the Tree View list, select **Cut Parameters** and make the necessary changes as shown in [Figure: 9.2.1](#).

Figure: 9.2.1



Compensation type allows you to choose how you want to handle cutter compensation. **Computer** sets Mastercam to compute the compensated toolpath and does not output control codes for compensation.

Entry/exit arc sweep sets the included angle of each entry and exit arc. If the entry/exit arc sweep is less than 180 degrees, the system applies an entry/exit line.

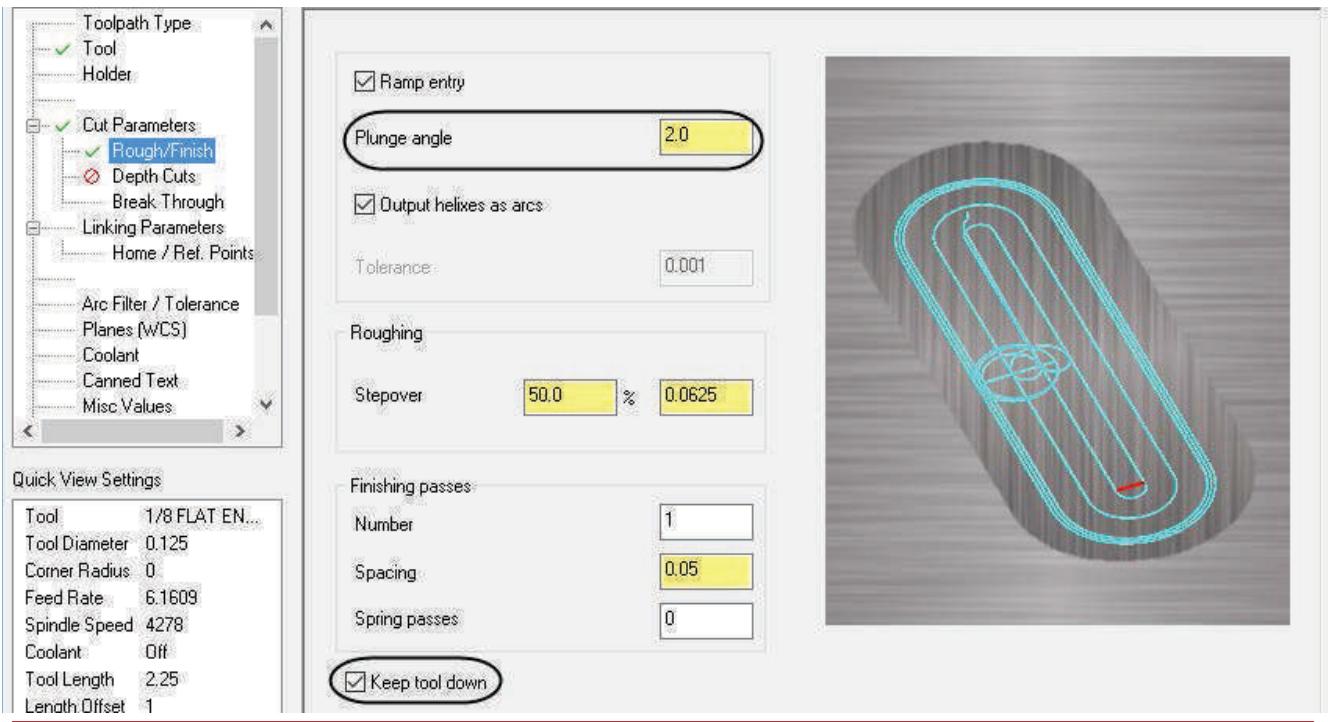
Perpendicular entry enters the toolpath perpendicular to the first tool move.



9.3 Rough/Finish

- From the Tree View list, select **Rough/Finish** and make the necessary changes as shown in [Figure: 9.3.1](#).

Figure: 9.3.1



Ramp entry creates a smoother entry motion rather than plunging directly.

Plunge angle sets the angle of descent for the entry move and determines the pitch. A smaller plunge angle means that the entry move takes longer to descend in the Z axis. A recommended angle is 3 to 5 degrees.

Output helixes as arcs writes the entry helix to the NCI file as arcs. Using this option can create shorter NC files. If you turn off this option, the helix breaks into linear segments in the NCI file.

Roughing Stepover sets the distance between cutting passes in the X and Y axes. Enter a percentage of the tool diameter or a distance.

Finishing passes allows you to set the finish cuts for the toolpath. This **Number** multiplied by the finish **Spacing** value equals the total amount of stock cut by the finish passes. Setting the number of finish cuts to 0 creates no finish cuts.

Keep tool down enabled does not allow the tool to retract between multipasses.

NOTE: Toolpath display in the window may be different depending on the last field selected.



9.4 Depth Cuts

- From the **Tree View** list, choose **Depth Cuts** and enable this option. Input a **Max rough step** of **0.1**.
- Enable the option **Keep tool down** as shown in [Figure: 9.4.1](#).

Figure: 9.4.1

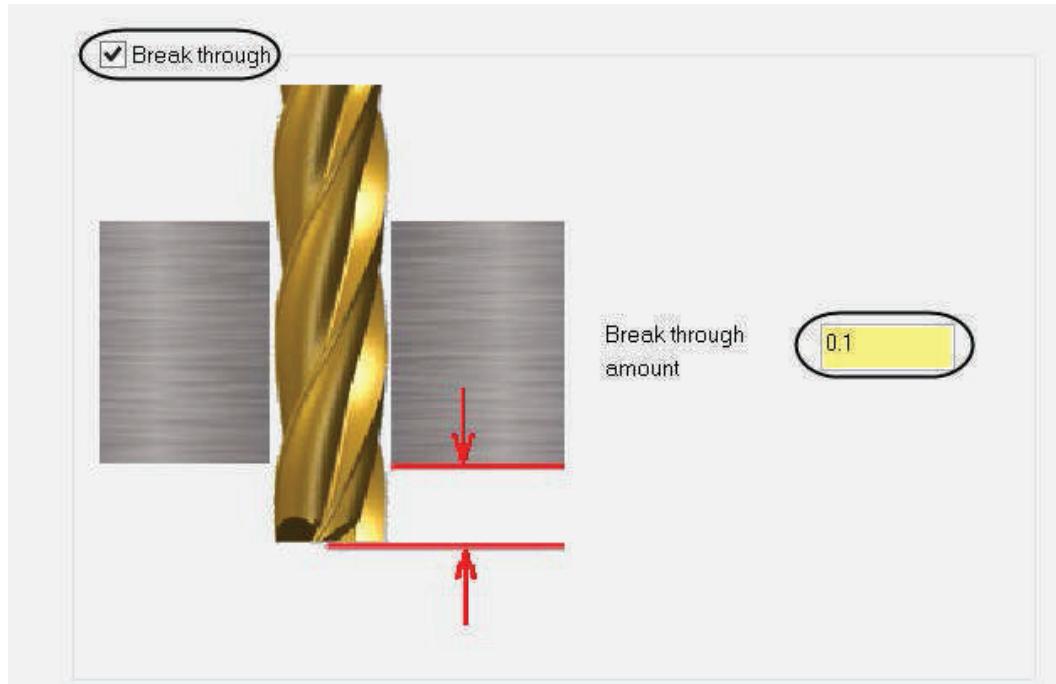




9.5 Break Through

- ♦ Select **Break Through** from the **Tree View** list. Enable this option and input a **Break through amount** of **0.1** as shown in [Figure: 9.5.1](#).

Figure: 9.5.1



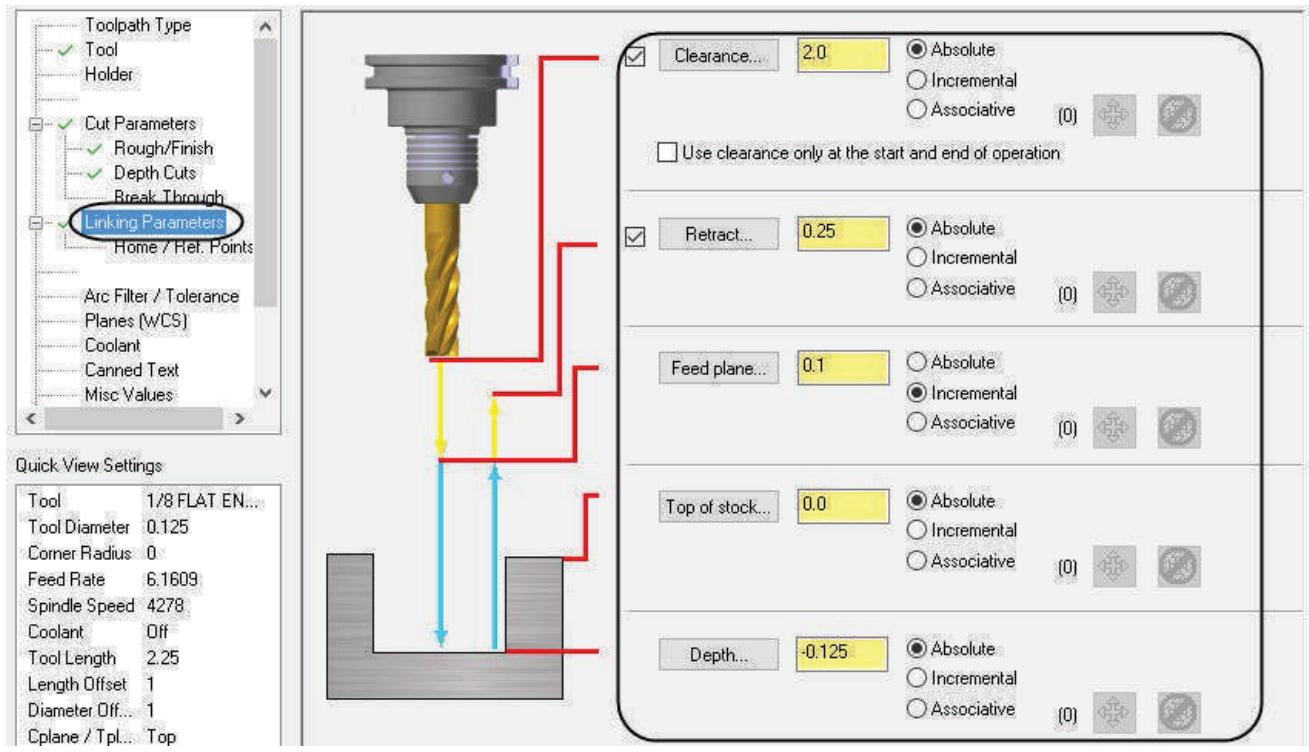
Break Through allows you to specify an amount that the tool will completely cut through the material by. This value is always a positive number.



9.6 Linking Parameters

- From the Tree View list, select **Linking Parameters** and make the necessary changes as shown in [Figure: 9.6.1](#).

Figure: 9.6.1



Clearance sets the height at which the tool moves to and from the part.

Retract sets the height that the tool moves up to before the next tool pass.

Feed plane sets the height that the tool rapidts to before changing to the plunge rate to enter the part.

Top of stock sets the height of the material in the Z axis.

Depth determines the final machining depth that the tool descends into the stock.

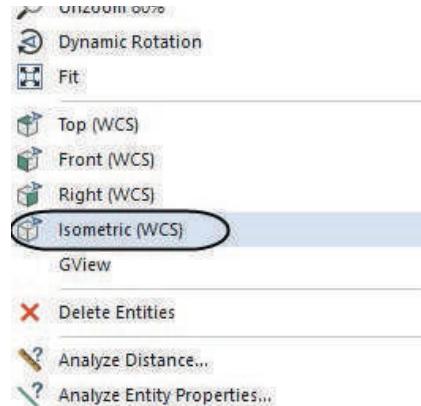


9.7 Preview the Toolpath

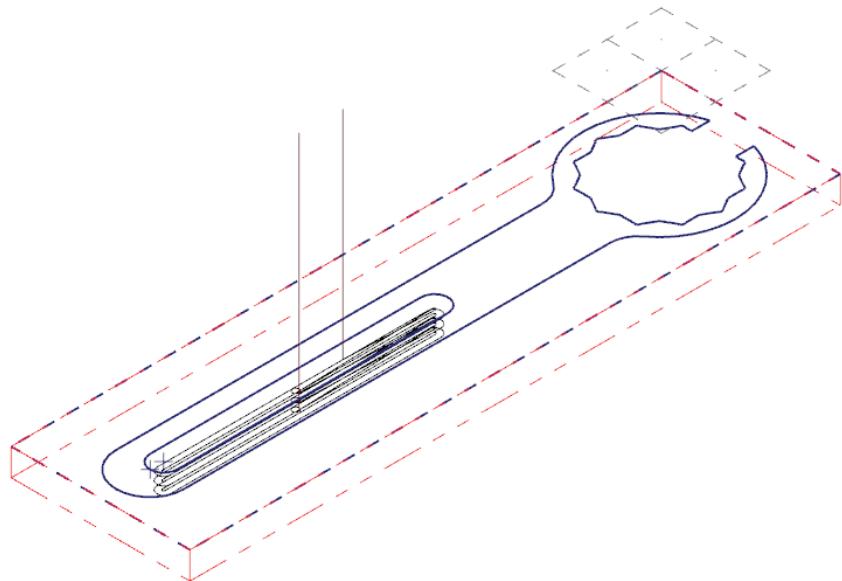
- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.
- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.



- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to exit the **2D Toolpaths - Slot Mill** parameters.





STEP 10: BACKPLOT THE TOOLPATHS

Backplotting shows the path the tools take to cut the part. This display lets you spot errors in the program before you machine the part. As you backplot toolpaths, Mastercam displays additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates, and the cycle time. It also shows any collisions between the workpiece and the tool.

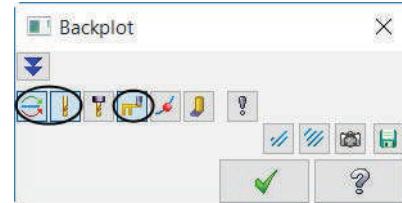
- ♦ Make sure that the toolpaths are selected (signified by the green check mark on the folder icon). If the operation is not selected, choose the **Select all operations** icon.



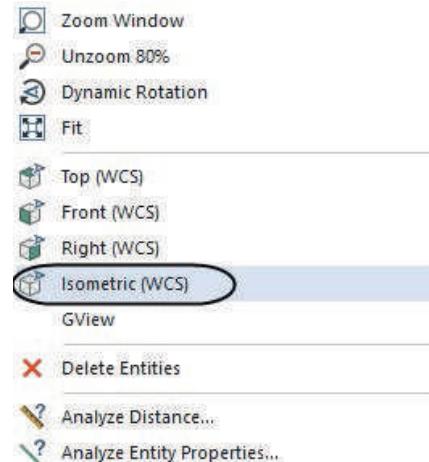
- ♦ Select the **Backplot selected operations** button.



- ♦ In the **Backplot** panel, enable **Display with color codes**, **Display tool** and **Display rapid moves** icons as shown.



- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ To fit the workpiece to the screen, if needed, right mouse click in the graphics window again and select **Fit**.
- ♦ You can step through the **Backplot** by using the **Step forward** or **Step back** buttons.
- ♦ You can adjust the speed of the backplot.

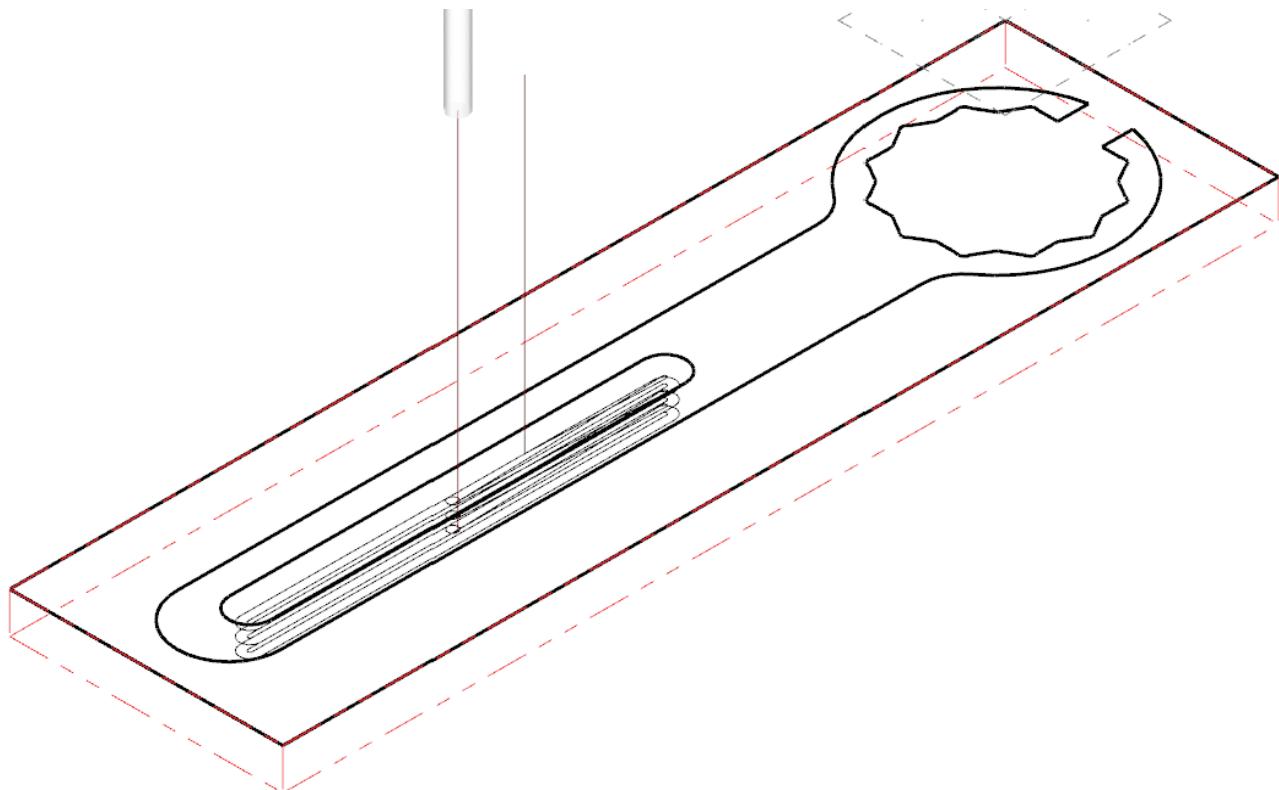
TUTORIAL #2

BACKPLOT THE TOOLPATHS



- ♦ Select the **Play** button to run **Backplot**.
- ♦ The toolpath should look as shown in [Figure: 10.0.1.](#)

Figure: 10.0.1.



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



STEP 11: SIMULATE THE TOOLPATH IN VERIFY

Verify Mode shows the path the tools take to cut the part with material removal. This display lets you spot errors in the program before you machine the part. As you verify toolpaths, Mastercam displays additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates, and the cycle time. It also shows any collisions between the workpiece and the tool.

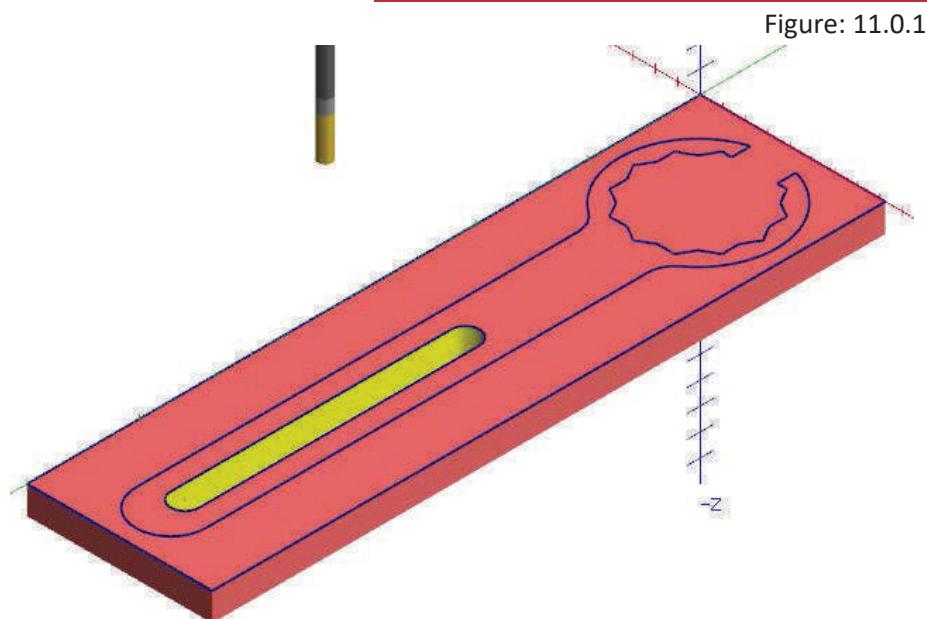
- From the **Toolpaths Manager**, select the **Verify selected operations** icon as shown.



NOTE: Mastercam launches a new window that allows you to check the part using **Verify**.



- Select the **Play** button to run **Verify**.
- The part should appear as shown in [Figure: 11.0.1](#).



- To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.





STEP 12: ROUGH THE OUTSIDE USING HIGH SPEED DYNAMIC MILL

In this step you will machine the outside profile using a **2D HS Dynamic Mill** toolpath.

Dynamic Mill Toolpath machines cores or pockets using the entire flute length. The toolpath supports many powerful entry methods, including a custom entry method. Entry methods and micro lifts support custom feeds and speeds to optimize and generate safe tool motion.

The toolpath depends on the **Machining strategy** that you choose in the **Chain Options**. If the strategy chosen is **From outside**, the toolpath starts at the outmost chain and works its way in taking on the final shape of the part as it approaches the final pass. You can also machine pockets in which case the strategy selected is **Stay inside** which keeps the tool inside the machining regions.

Toolpath Preview:



12.1 Chain selection

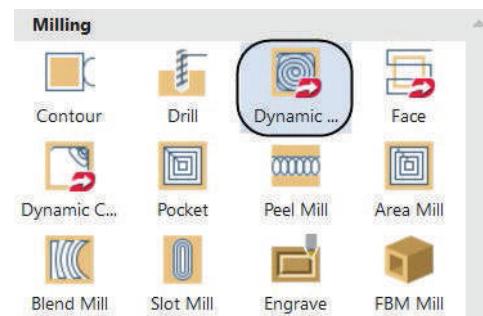
- ♦ Press **Alt + T** to remove the toolpath display.

TOOLPATHS

- ♦ In the **2D** group, click on the **Expand gallery** arrow as shown.

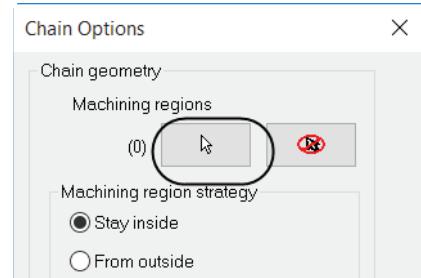


- ♦ Select the **Dynamic Mill** icon as shown.

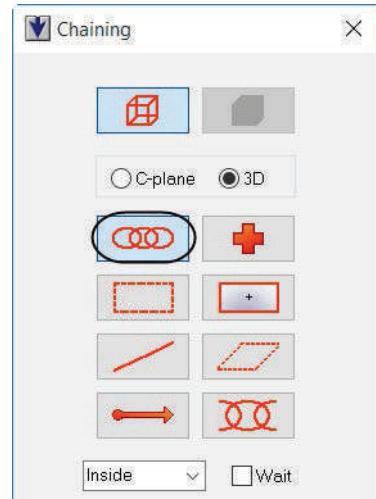




- In the **Chain Options** dialog box, under **Machining regions**, click on the **Select machining chains** button as shown to define the area to be machined.



- The **Chaining** dialog box will open and leave the **Chain** button enabled as shown.



- Right mouse click on the graphics window and change the graphics view to **Top**.



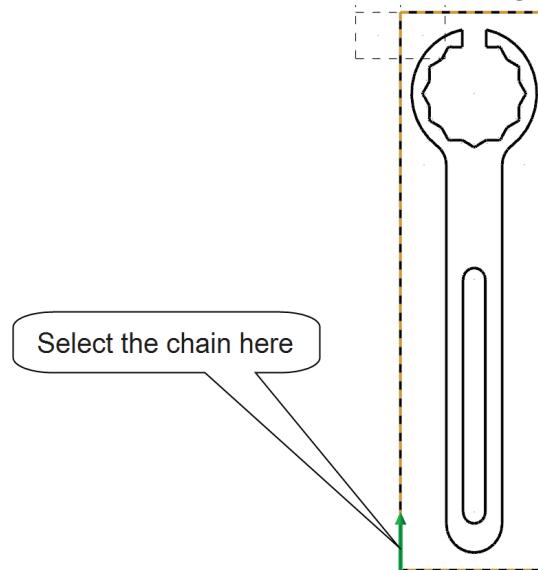
TUTORIAL #2

ROUGH THE OUTSIDE USING HIGH SPEED DYNAMIC MILL

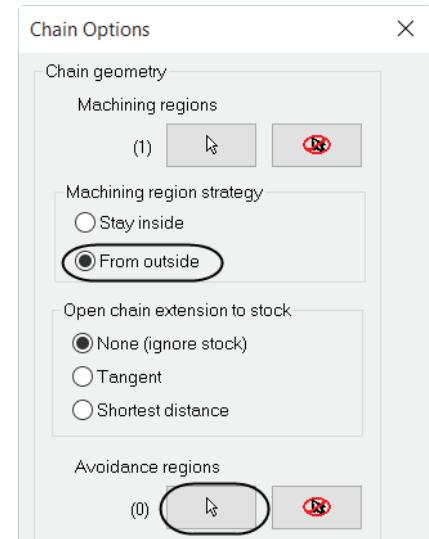


- ♦ [Select 2D HST machining chain 1]: Select the rectangle as shown in [Figure: 12.1.1](#).

Figure: 12.1.1



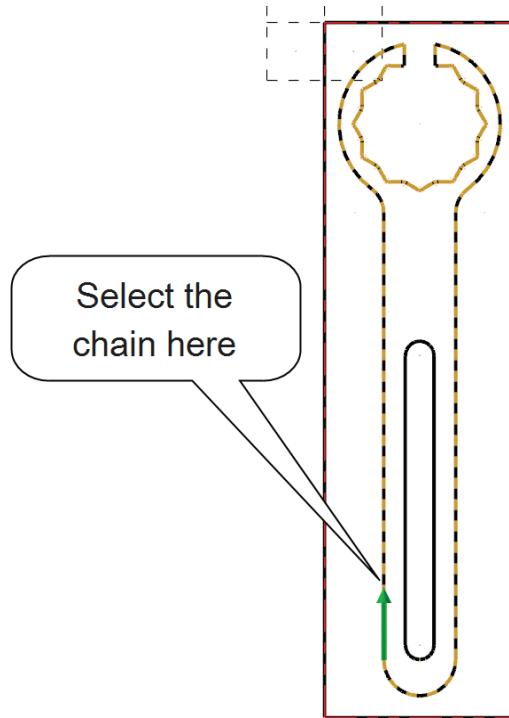
- ♦ Select the **OK** button to exit the **Chaining** dialog box.
- ♦ To start the toolpath from the outside, in the **Machining region strategy**, make sure that **From outside** is enabled.
- ♦ In the **Avoidance regions**, click on the **Select** button as shown.





- ♦ [Select 2D HST avoidance chain 1]: Select the profile as shown in [Figure: 12.1.2.](#)

Figure: 12.1.2



- ♦ Select the **OK** button to exit the **Chaining** dialog box.
- ♦ Select the **OK** button to exit the **Chain Options** panel.
- ♦ In the **Toolpath Type** page, **Dynamic Mill** will be selected as shown in [Figure: 12.1.3.](#)

Figure: 12.1.3



12.2 Preview Chains

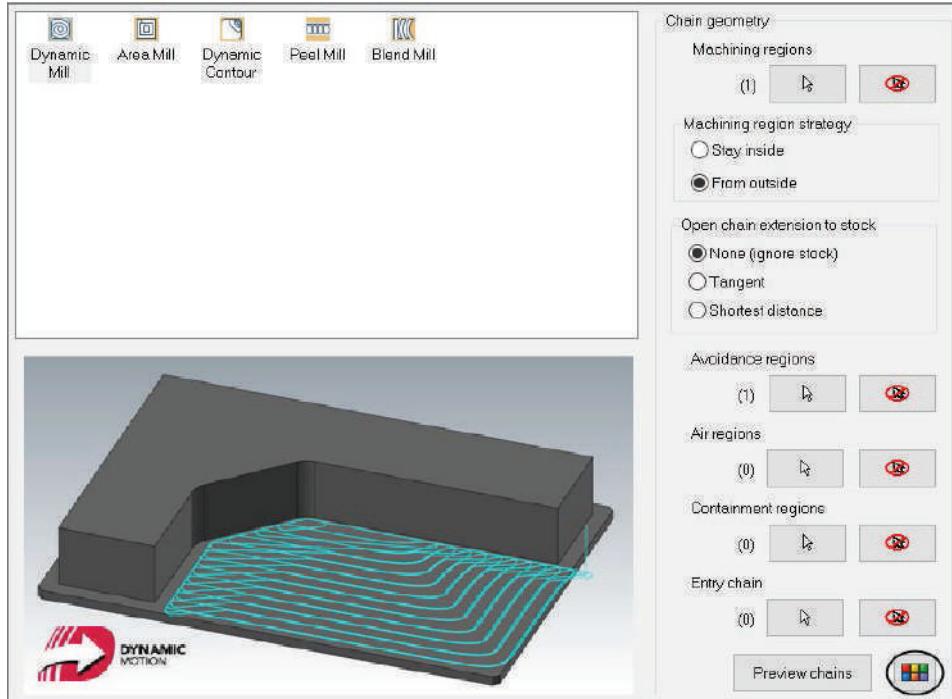
The **Preview Chains** function is intended to give the user a quick visual representation of how Mastercam sees the various pieces of geometry that have been selected, how they interact with one another, and a general overview of how the toolpath will be calculated with the selections presently made.

TUTORIAL #2

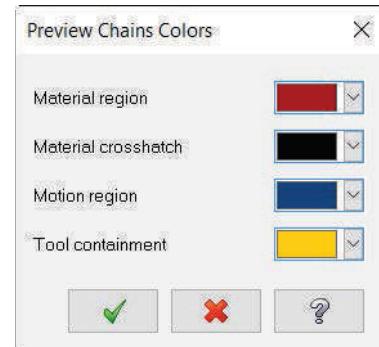
ROUGH THE OUTSIDE USING HIGH SPEED DYNAMIC MILL



- Click on the **Color** icon to see the legend for **Preview chains** as shown.



- The **Preview Chains Colors** dialog box should look as shown.



The **Material region** and **Material crosshatch** are the two colors that are used to define the material to be cut. The default colors are red for the background and black for the crosshatch.

The **Motion region** displays the area that Mastercam is making available to the toolpath for motion if it needs it. The color to represent it is dark blue. The primary reason for the display of the entire available (but not necessarily used) motion region is to help the user visualize how the tool may move near or interact with any adjacent geometry.

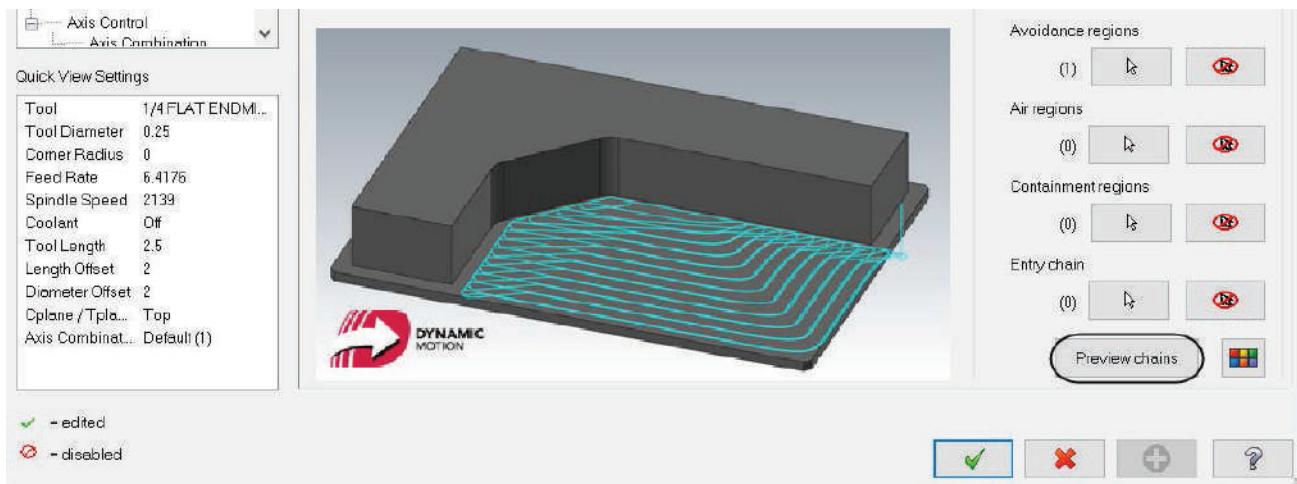
The **Tool containment** is what you have selected as the containment region in the chain geometry. If you have not selected a containment region, it will default to the outside of the motion region since that is currently the default area the toolpath is being contained to. The color used to represent the tool containment is yellow.

- Select the **OK** button to exit **Preview Chains Colors**.





- ♦ Select the **Preview chains** button as shown.



- ♦ Select the **Hide dialog** button to see the preview in the graphics window.
- ♦ The **Preview chains** should look as shown.



- ♦ Press **Esc** key to return to the toolpath parameters.
- ♦ Click on the **Preview chains** button again to clear the **Preview chains** display.



TUTORIAL #2

ROUGH THE OUTSIDE USING HIGH SPEED DYNAMIC MILL



12.3 Select a 1/4" Flat Endmill from the library and set the Tool Parameters

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

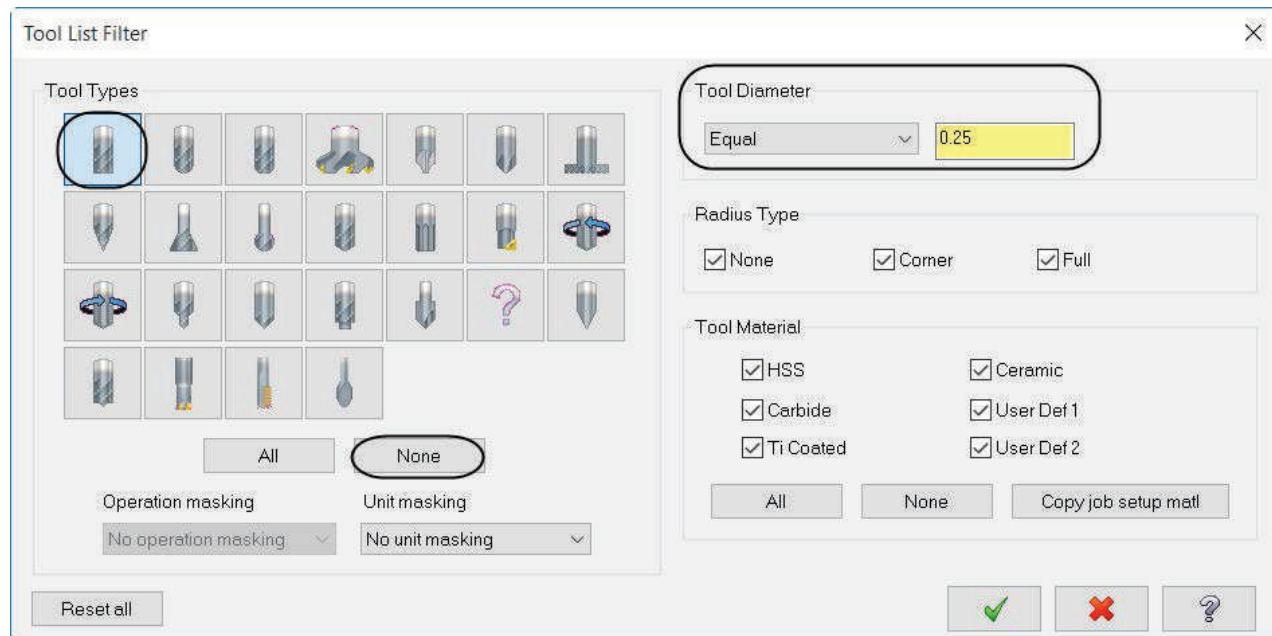


- ♦ Select the **Filter** button.



- ♦ Select the **None** button and then under **Tool Types**, choose the **Flat Endmill** Icon.
- ♦ Under **Tool Diameter**, pick **Equal** and input a value **0.25** as shown in [Figure: 12.3.1](#).

Figure: 12.3.1



- ♦ Select the **OK** button to exit the **Tool List Filter**.
- ♦ In the **Tool Selection** dialog box, you should only see a **1/4" Flat Endmill**.

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Ra...	Type
285	-	1/4 FLAT ...	-	0.25	0.0	0.5	4	No...	En...

- ♦ Select the **1/4" Flat Endmill** in the **Tool Selection** page and then select the **OK** button to exit.



- Make all the necessary changes as shown in [Figure: 12.3.2.](#)

Figure: 12.3.2

The screenshot shows the Mastercam software interface for tool setup. On the left, a tool library table lists two entries:

#	Assembly Name	Tool Name	Holder Name	Dia.
1	-	1/8 FLAT ...	-	0...
2	-	1/4 FLAT ...	-	0.25

On the right, specific tool settings are defined:

- Tool diameter: 0.25
- Corner radius: 0.0
- Tool name: 1/4 FLAT ENDMILL
- Tool #: 2
- Length offset: 2
- Head #: 0
- Diameter offset: 2

Below these settings, machining parameters are listed:

- RCTF:
- Spindle direction: CW
- Feed rate: 6.4176
- Spindle speed: 2139
- FPT: 0.0008
- SFM: 139.9869
- Plunge rate: 6.4176
- Retract rate: 6.4176
- Force tool change:
- Rapid Retract:

A comment box at the bottom contains the text: "Machine the outside profile."

At the bottom left, there is a checkbox labeled "To batch".



12.4 Set the Cut Parameters

- From the Tree View list, select **Cut Parameters**. Change the settings as shown in Figure: 12.4.1.

Figure: 12.4.1



Stepover sets the distance between cutting passes in the X and Y axis.

Approach distance adds the specified absolute distance to the beginning of the toolpath's first cut.

First pass offset offsets out the machining region with a user defined distance for the tool to safely engage from the outside of the material.

First pass feed reduction allows you to slow the feed for the first pass on machining region material approached from the outside.

Min toolpath radius reduces sharp corner motion between cut passes.

Micro lift distance enters the distance the tool lifts off the part on the back moves. Microlifts are slight lifts that help clear chips and minimize excessive tool heating.

Back feedrate controls the speed of the backfeed movement of the tool.

Motion > Gap Size, retract controls retracts in the toolpath when making a non-cutting move within an area where the tool can be kept down or microlifted.

Cut order optimization defines the cut order Mastercam applies to different cutting passes in the dynamic mill toolpath.



12.5 Set the Entry Motion

Entry Motion configures an entry method for the dynamic mill toolpath which determines not only how and where the tool enters the part, but the cutting method/machining strategy used by the toolpath.

- From the **Tree View list**, select **Entry Motion** and set the **Entry method** to **Helix only**. Make all necessary changes as shown in [Figure: 12.5.1](#).

Figure: 12.5.1



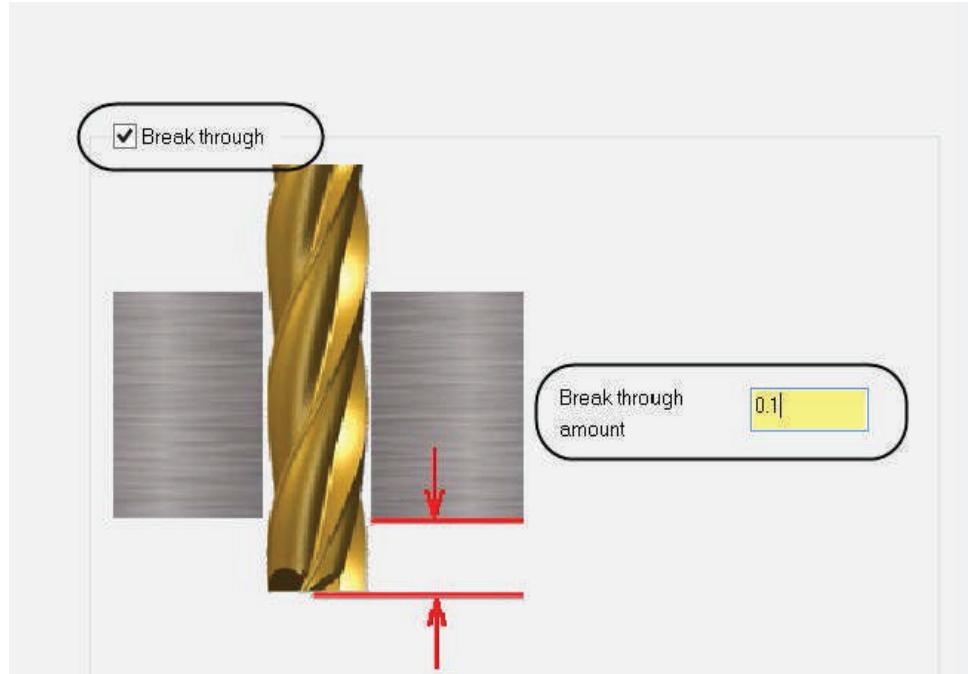
Entry method set to **Helix only** creates a helical entry into the part.



12.6 Set the Break through

- From the **Tree View** list, select and enable **Break Through** to cut completely through the material by an amount that you specify as shown in [Figure: 12.6.1](#).

Figure: 12.6.1

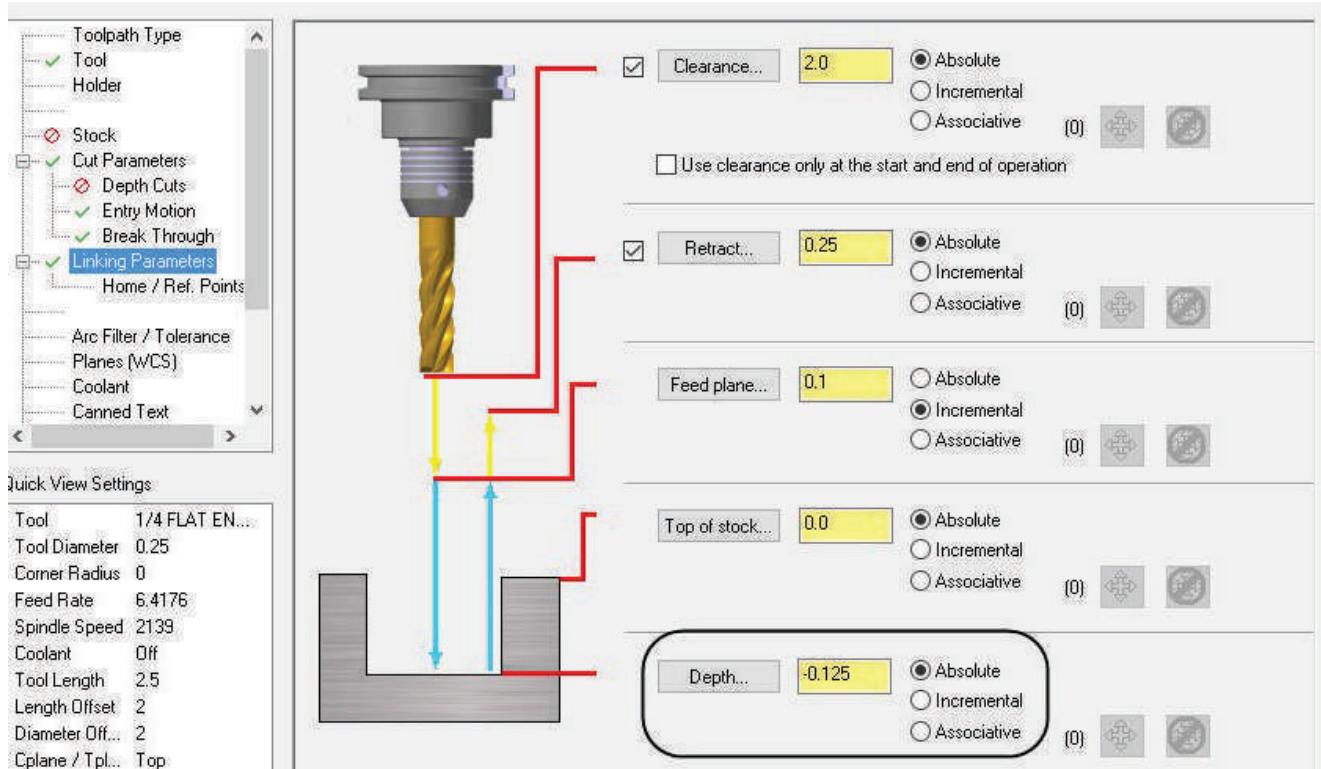




12.7 Set the Linking Parameters

- From the Tree View list, select **Linking Parameters** and change the **Depth** to **-0.125** as shown in [Figure: 12.7.1](#).

Figure: 12.7.1



12.8 Preview the Toolpath

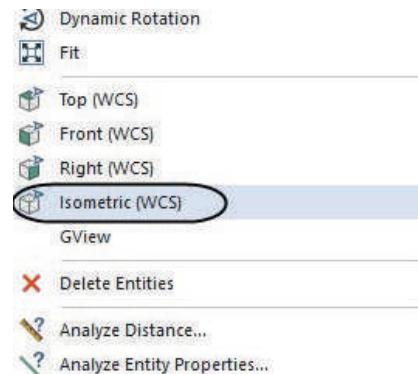
- To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- To hide the dialog box, click on the **Hide dialog** icon as shown.



- To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.

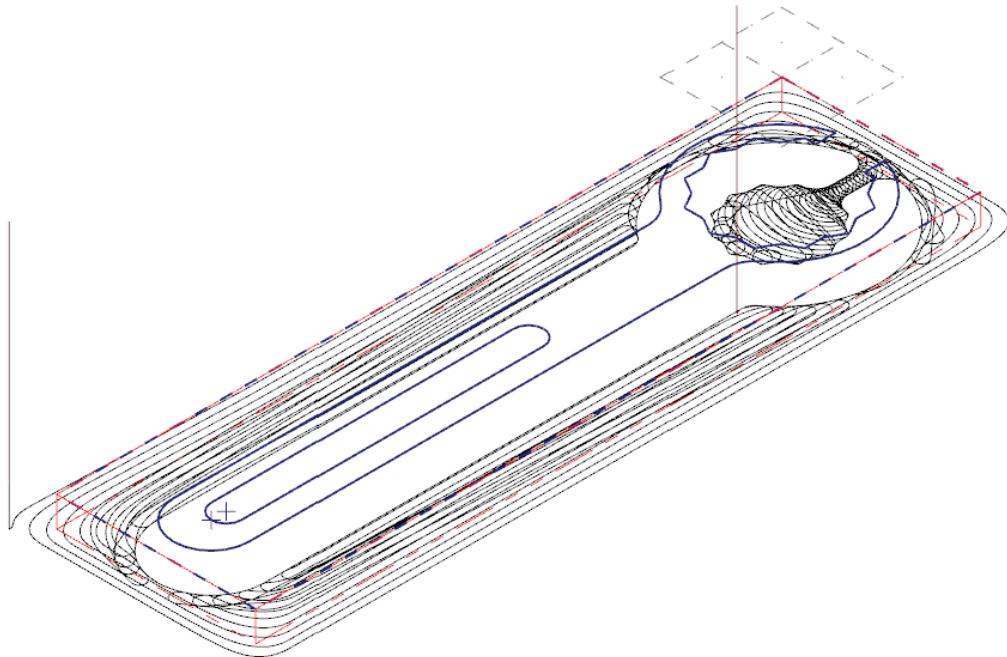


TUTORIAL #2

ROUGH THE OUTSIDE USING HIGH SPEED DYNAMIC MILL



- The toolpath should look as shown.



- Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

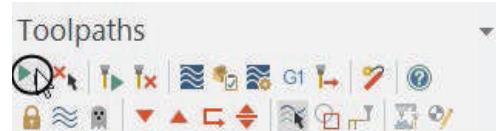
- Select the **OK** button to exit the **2D High Speed Toolpath - Dynamic Mill** parameters.
- To remove the toolpath display, if needed, press **Alt + T**.





12.9 Verify the toolpath

- From the **Toolpaths Manager**, click on the **Select all operations** icon.

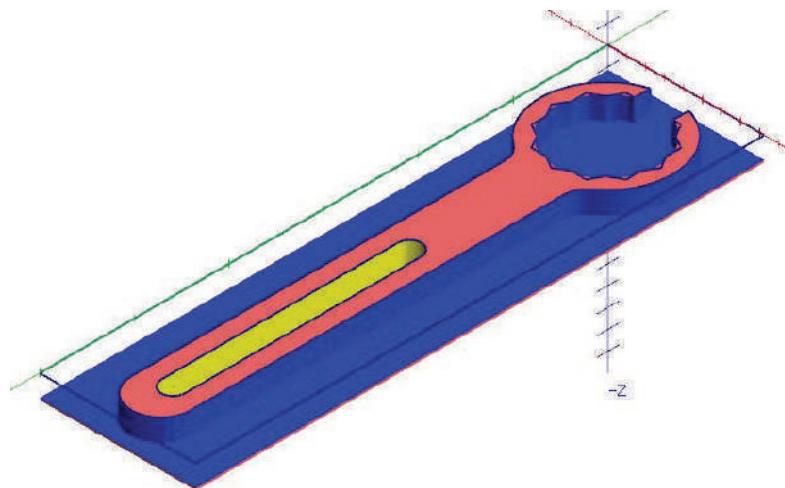


- Select the **Verify selected operations** icon.



- Follow **Mastercam Simulation** procedures as shown on [page 147](#).
- The part will appear as shown in [Figure: 12.9.1](#).

Figure: 12.9.1



- To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.
- To remove or view the toolpath display, press **Alt + T** or click on the **Toggle display on selected operations**.

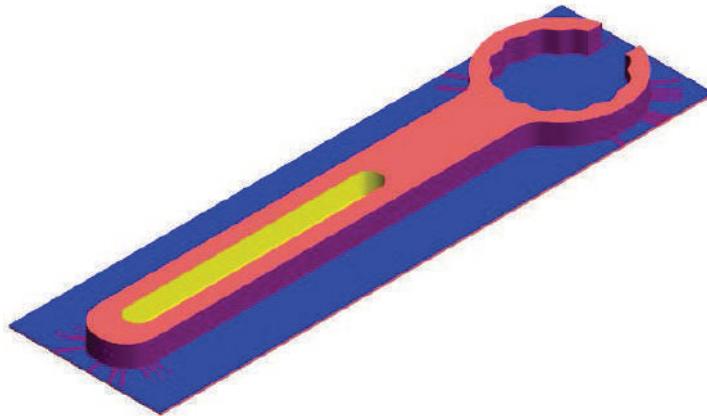




STEP 13: FINISH THE OUTSIDE PROFILE USING CONTOUR TOOLPATH

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:



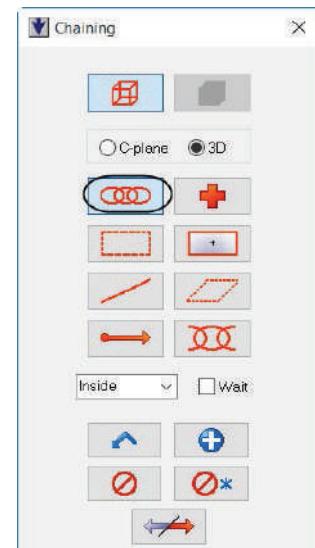
13.1 Chain selection

TOOLPATHS

- From the **2D** group, select the **Contour** icon.



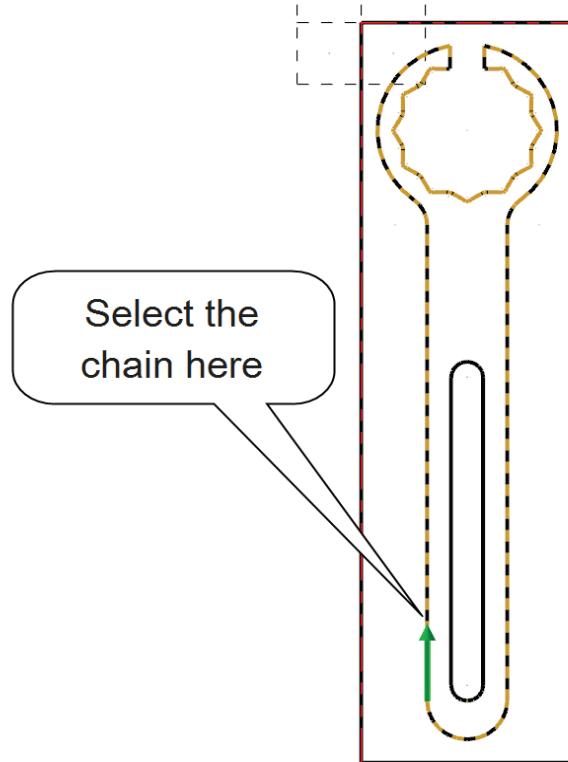
- Leave the **Chain** button enabled in the **Chaining** dialog box as shown.





- ♦ If needed, change the graphics view to **Top**.
- ♦ Select the profile as shown in [Figure: 13.1.1](#).

Figure: 13.1.1



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



TUTORIAL #2

FINISH THE OUTSIDE PROFILE USING CONTOUR TOOLPATH



13.2 Select the 1/4" Flat Endmill from the list and set the Tool page parameters

- From the **Tree View** list, select **Tool** and make all the necessary changes as shown in [Figure: 13.2.1](#).

Figure: 13.2.1

The screenshot shows the Mastercam software interface with the 'Tool' page selected. On the left, a tree view list displays two tools: #1 (Assembly Name: -, Tool Name: 1/8 FLAT ..., Dia.: 0...) and #2 (Assembly Name: -, Tool Name: 1/4 FLAT ..., Dia.: 0.25). The right side of the screen shows various toolpath parameters:

Tool diameter:	0.25		
Corner radius:	0.0		
Tool name:	1/4 FLAT ENDMILL		
Tool #:	2	Length offset:	2
Head #:	0	Diameter offset:	2
Spindle direction: CW			
Feed rate:	6.4176	Spindle speed:	2139
FPT:	0.0008	SFM:	139.9869
Plunge rate:	6.4176	Retract rate:	6.4176
<input type="checkbox"/> Force tool change <input type="checkbox"/> Rapid Retract			
Comment Finish the profile.			

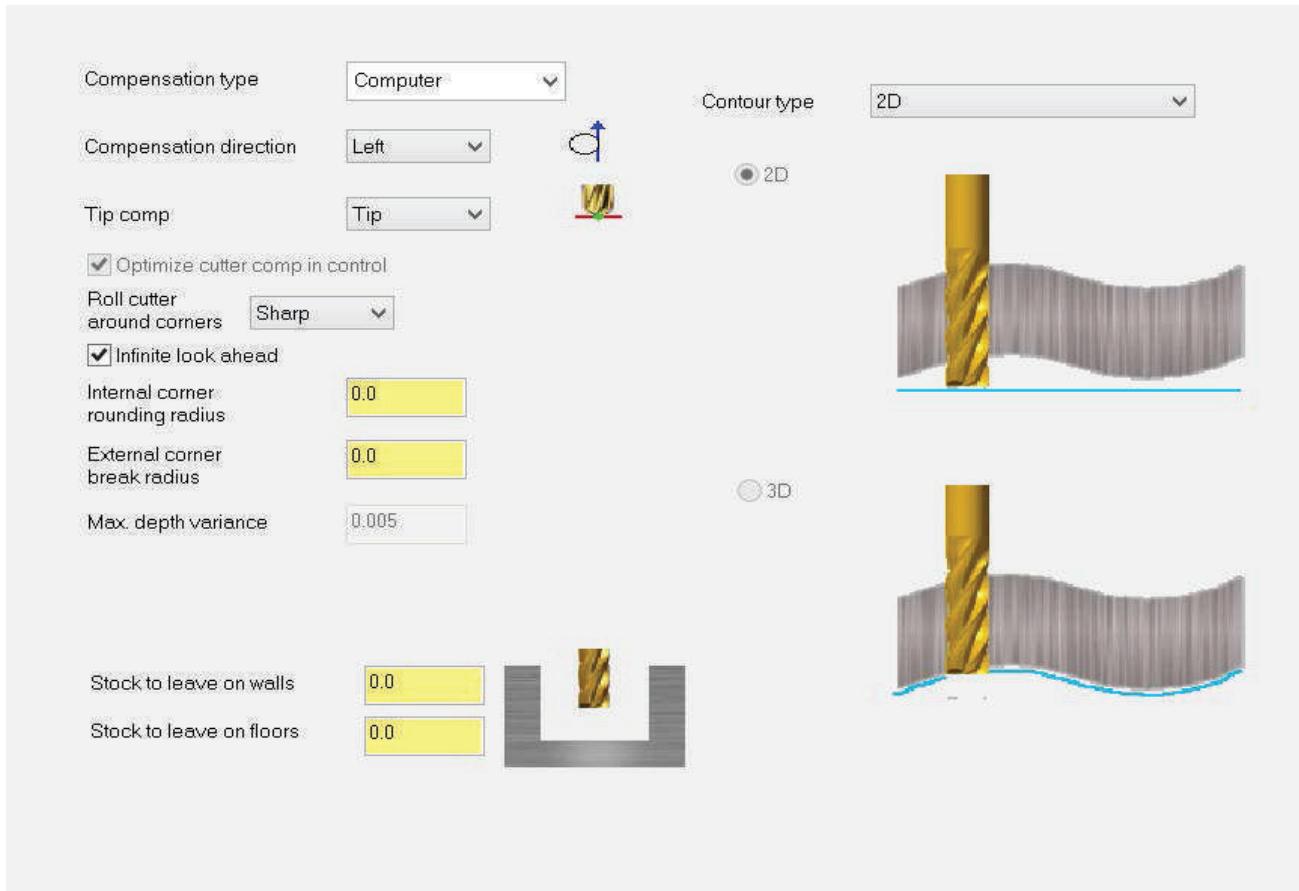
At the bottom left, there are buttons for 'Select library tool...', 'Filter Active', and 'Filter...'. A checkbox 'To batch' is also present.



13.3 Cut Parameters

- From the Tree View list, select **Cut Parameters** and ensure the settings appear as shown in [Figure: 13.3.1](#).

Figure: 13.3.1



NOTE: For more information regarding these parameters, please check Step 15 in **Tutorial #1**.

TUTORIAL #2

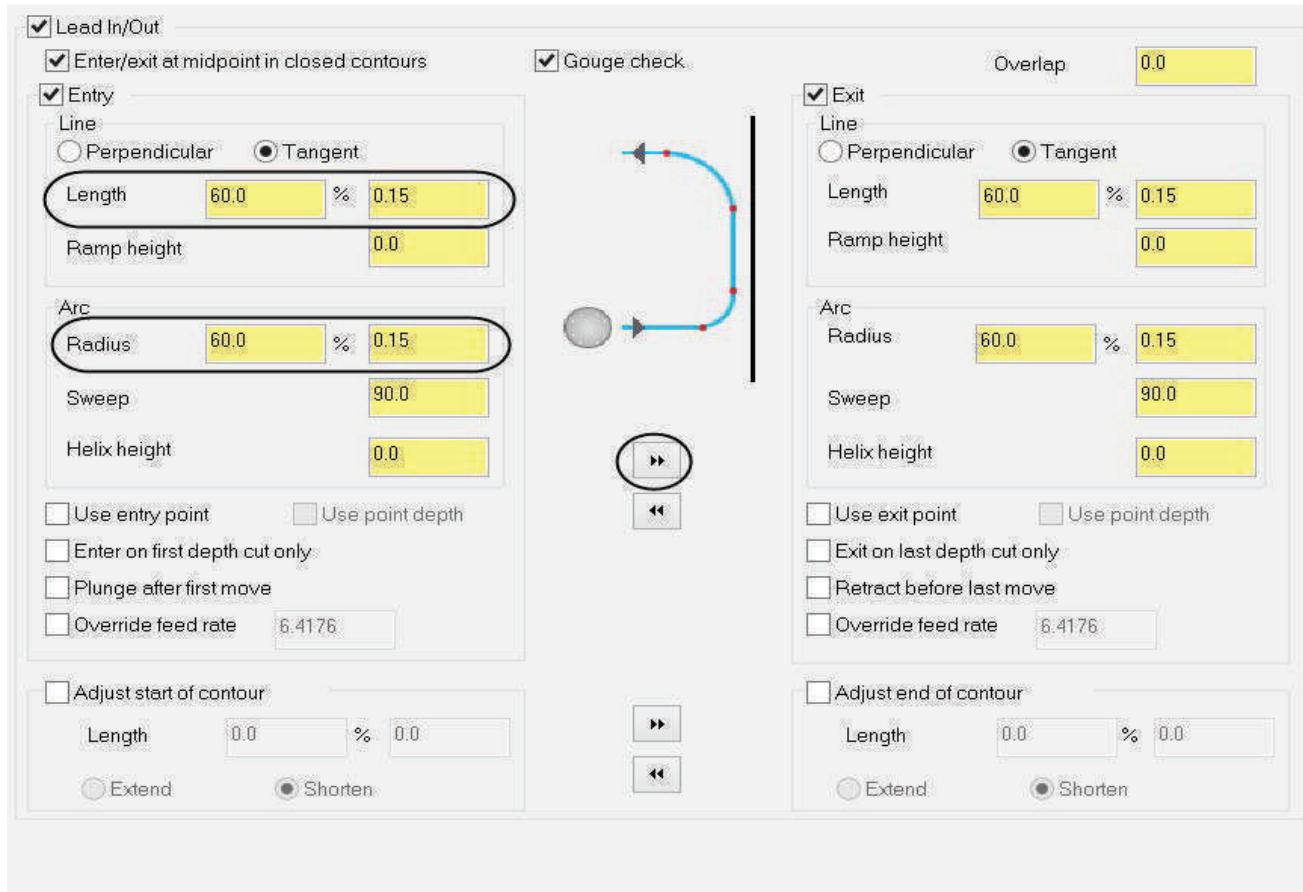
FINISH THE OUTSIDE PROFILE USING CONTOUR TOOLPATH



13.4 Lead In/Out

- ♦ Select **Lead In/Out** from the **Tree View** list.
- ♦ Change the parameters as shown in [Figure: 13.4.1](#).

Figure: 13.4.1



NOTE: Clicking on the circled arrows button near the center of the box will fill fields on the **Exit** category of the box with appropriate information copied from the **Entry** category.



13.5 Break Through

- From the Tree View list, select **Break Through** and make the necessary changes as shown in [Figure: 13.5.1](#).

Figure: 13.5.1

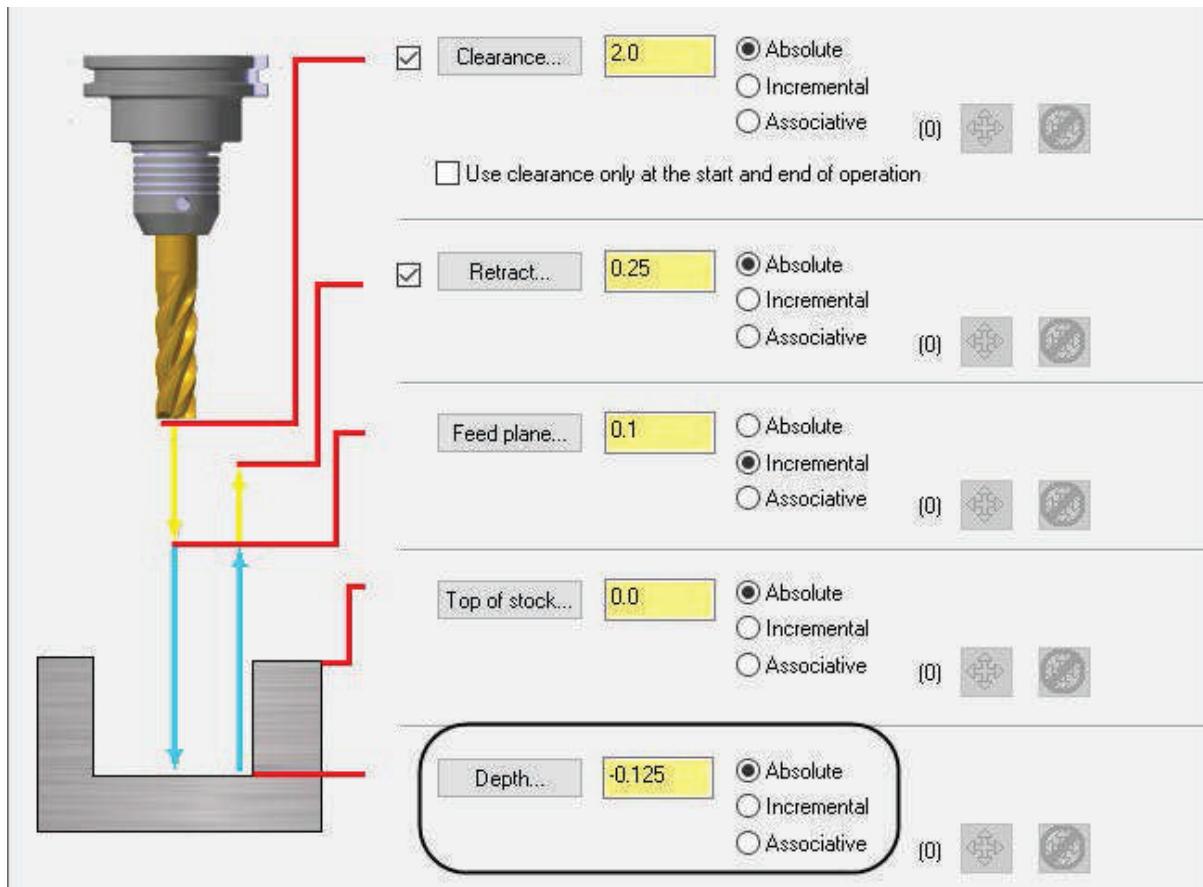




13.6 Linking Parameters

- ♦ Select **Linking Parameters** and input the **Depth** as shown in [Figure: 13.6.1](#).

Figure: 13.6.1



13.7 Preview the Toolpath

- ♦ To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

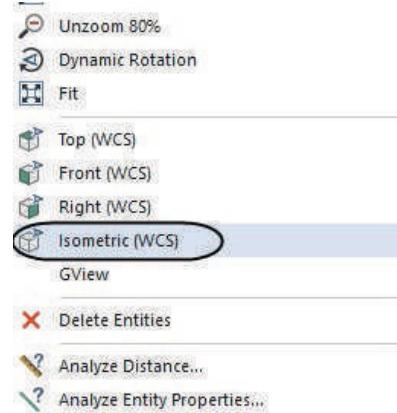


- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.

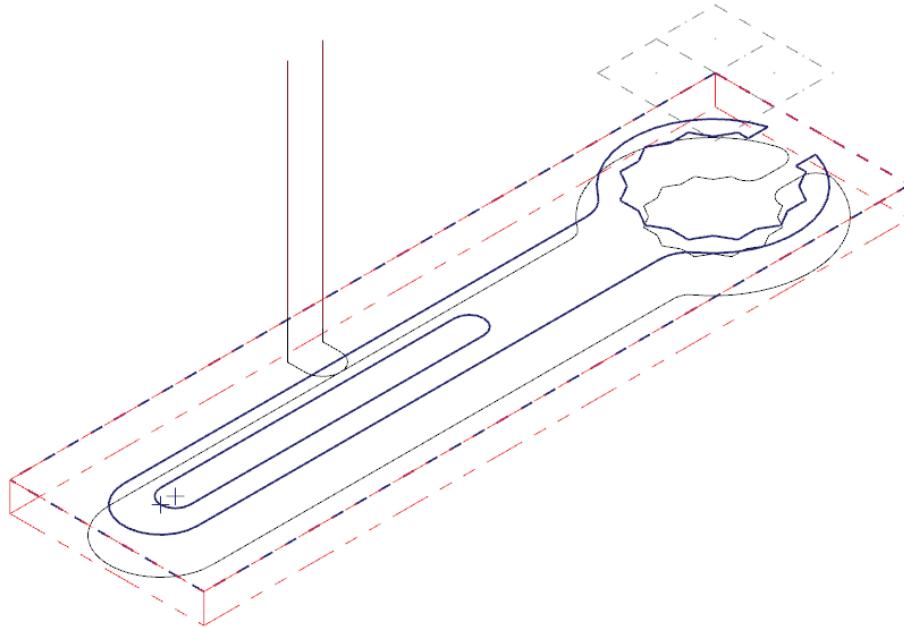




- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.



- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Once complete, select the **OK** button to exit the **2D Toolpaths - Contour** parameters.
- ♦ To remove the toolpath display, if needed, press **Alt + T**.



TUTORIAL #2

FINISH THE OUTSIDE PROFILE USING CONTOUR TOOLPATH

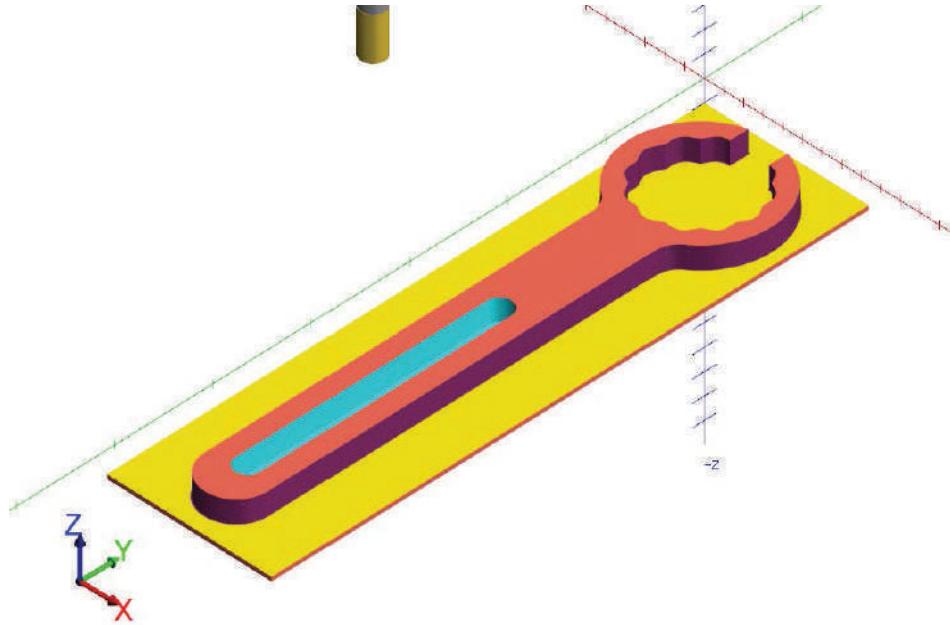


13.8 Backplot and Verify the toolpaths

- ♦ To **Backplot** and **Verify** the toolpaths, see **page 145** to review the procedures.
- ♦ To select all the operations, from the **Toolpaths Manager**, click on the **Select all operations** icon.



- ♦ After running **Verify**, the part should look as shown.



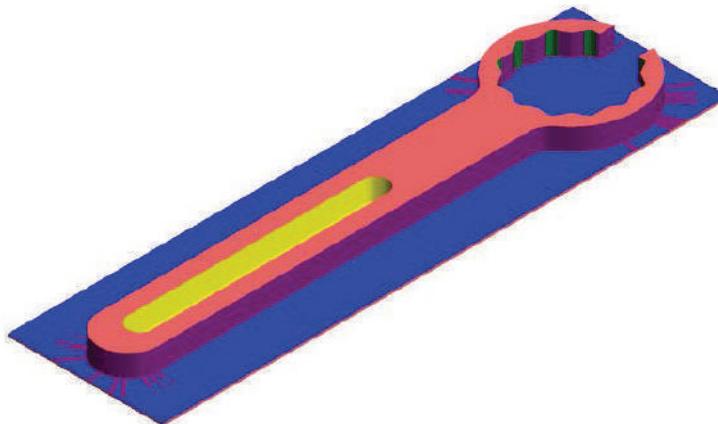
- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.



STEP 14: CLEAN THE INSIDE SHAPE USING 2D HS DYNAMIC CONTOUR

The **2D HS Dynamic Contour** toolpath utilizes the entire flute length of the cutting tools and is used to mill material off walls. It does support both closed or open chains.

Toolpath Preview:



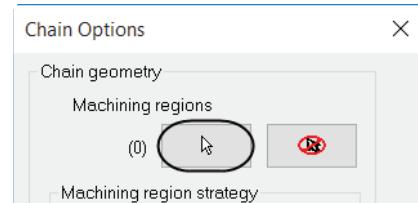
TOOLPATHS

- From the **2D** group, click on the downward arrow until you see the **Dynamic Contour** icon and select it.



14.1 Select the Geometry

- In the **Chain Options** dialog box, in the **Machining regions**, click on the **Select** button as shown.

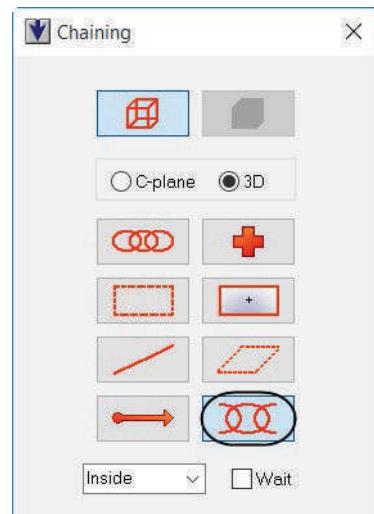


TUTORIAL #2

CLEAN THE INSIDE SHAPE USING 2D HS DYNAMIC CONTOUR

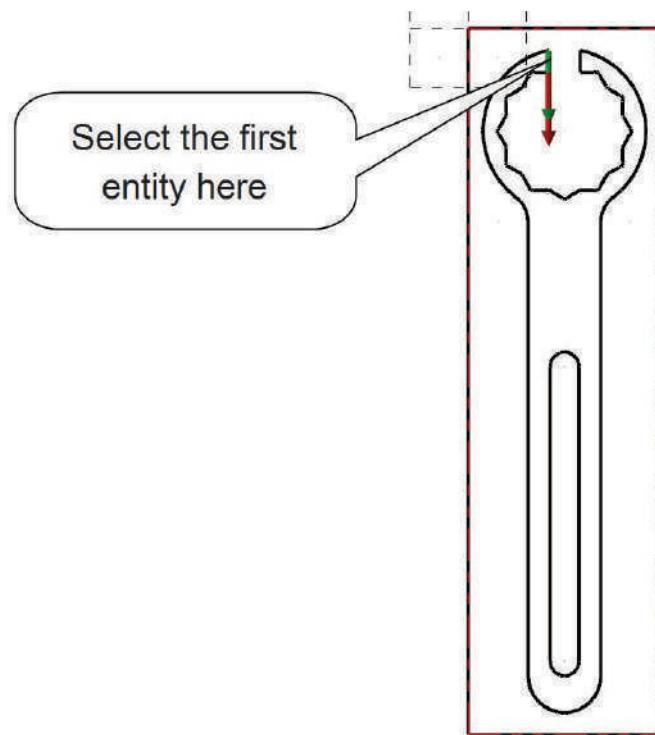


- Enable the **Partial** button in the **Chaining** dialog box as shown.



- Select the **Top** view if needed.
- Select the first entity of the chain as shown in [Figure: 14.1.1](#). Make sure that the arrows are pointing downwards as shown; otherwise select the **Reverse** button  from the **Chaining** dialog box.

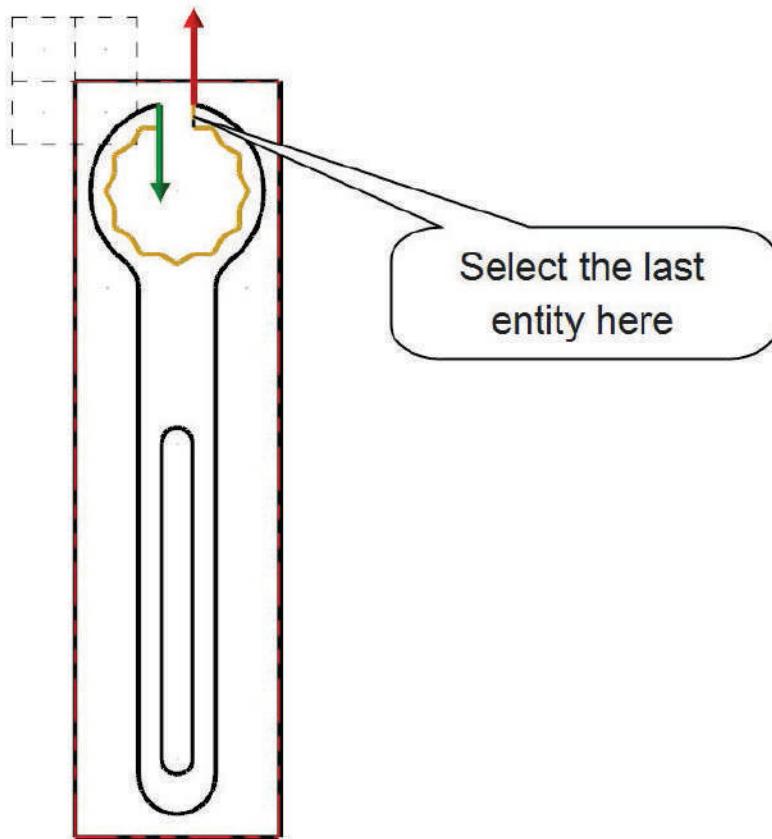
Figure: 14.1.1





- ♦ Select the last entity of the chain as shown in [Figure: 14.1.2.](#)

Figure: 14.1.2



- ♦ Choose the **OK** button to exit the **Chaining** dialog box.
- ♦ Select the **OK** button to exit the **Chain Options** dialog box and to continue.
- ♦ On the **Toolpath Type** page, **Dynamic Contour** will be picked.



TUTORIAL #2

CLEAN THE INSIDE SHAPE USING 2D HS DYNAMIC CONTOUR



14.2 Select a 1/16" Flat Endmill

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

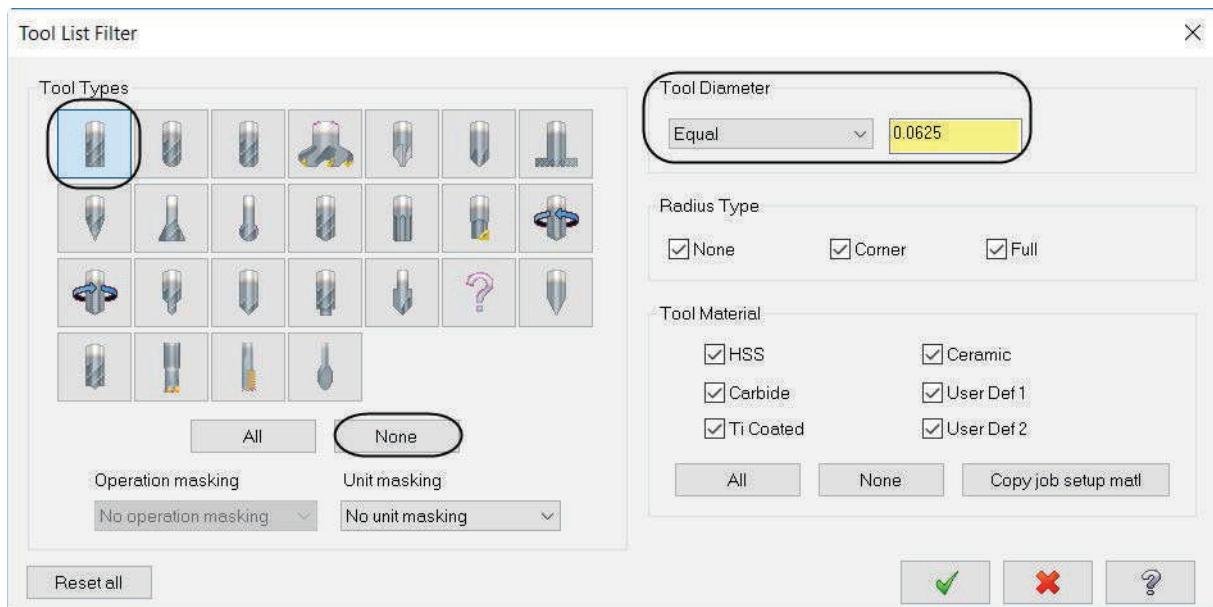


- ♦ Select the **Filter** button.



- ♦ Select the **None** button and then under **Tool Types**, choose the **Flat Endmill** icon.
- ♦ Under tool diameter, pick **Equal** and input a value **0.0625** as shown in [Figure: 14.2.1](#).

Figure: 14.2.1



- ♦ Select the **OK** button to exit the **Tool List Filter**.
- ♦ In the **Tool Selection** dialog box, you should only see a **1/16" Flat Endmill**.

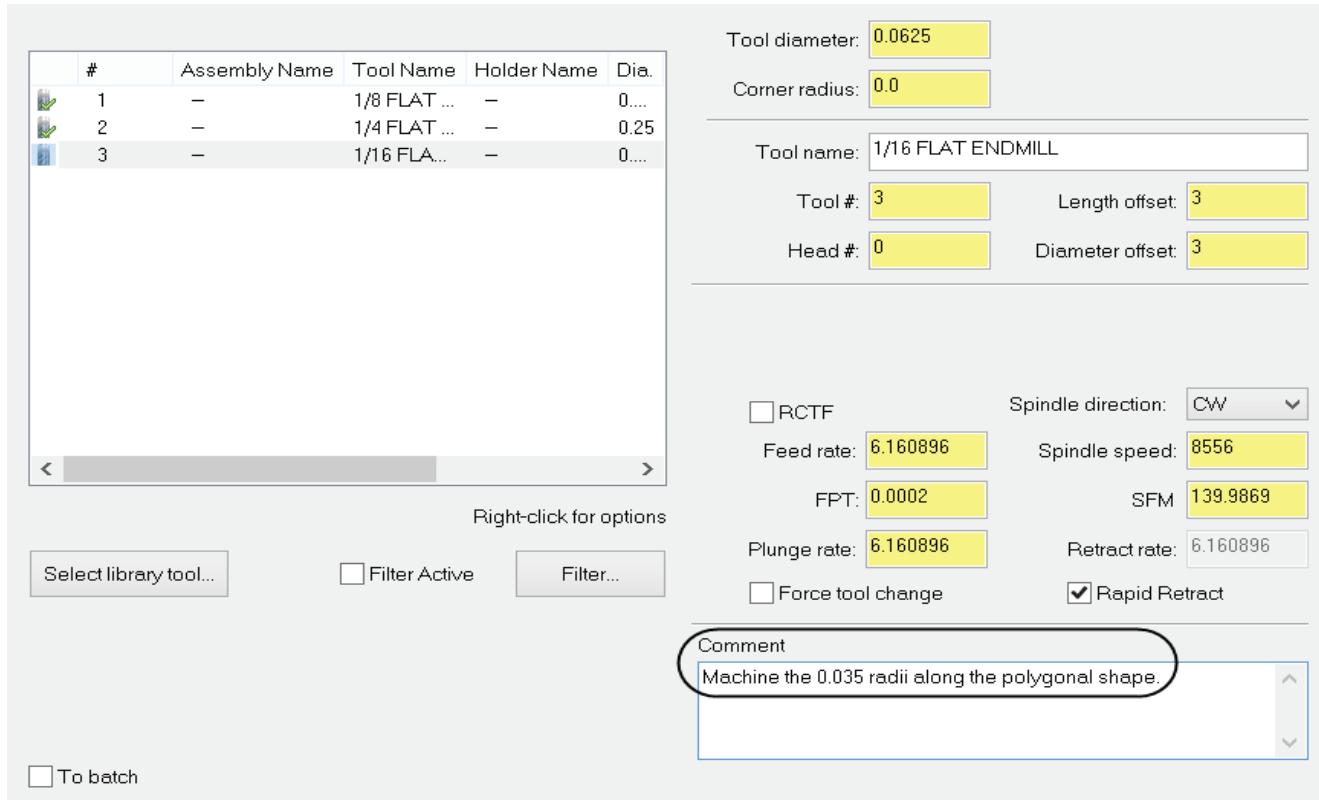
#	Assembly...	Tool Name	Holder Name	Dia.	Cor. r...	Length	# Flut...	Type	Rad...
280	-	1/16 FLAT ENDMILL	-	0.0625	0.0	0.375	4	End...	None

- ♦ Select the **1/16" Flat Endmill** in the **Tool Selection** page and then select the **OK** button to exit.



- ♦ Make any other changes as shown in [Figure: 14.2.2.](#)

Figure: 14.2.2





14.3 Cut Parameters

- From the Tree View list, select **Cut Parameters** and ensure the parameters appear the same as in [Figure: 14.3.1](#).

Figure: 14.3.1



Compensation direction offsets the tool to the **Left** in our case.

Approach distance adds the specified absolute distance to the beginning of the toolpath's first cut.

First pass offset offsets out the machining region with a user defined distance for the tool to safely engage from the outside in the material.

First pass feed reduction allows you to slow the feed for the first pass on machining region material approached from the outside.

Stepover sets the distance between cutting passes in the X and Y axes. Enter a percentage of the tool diameter or an absolute distance.

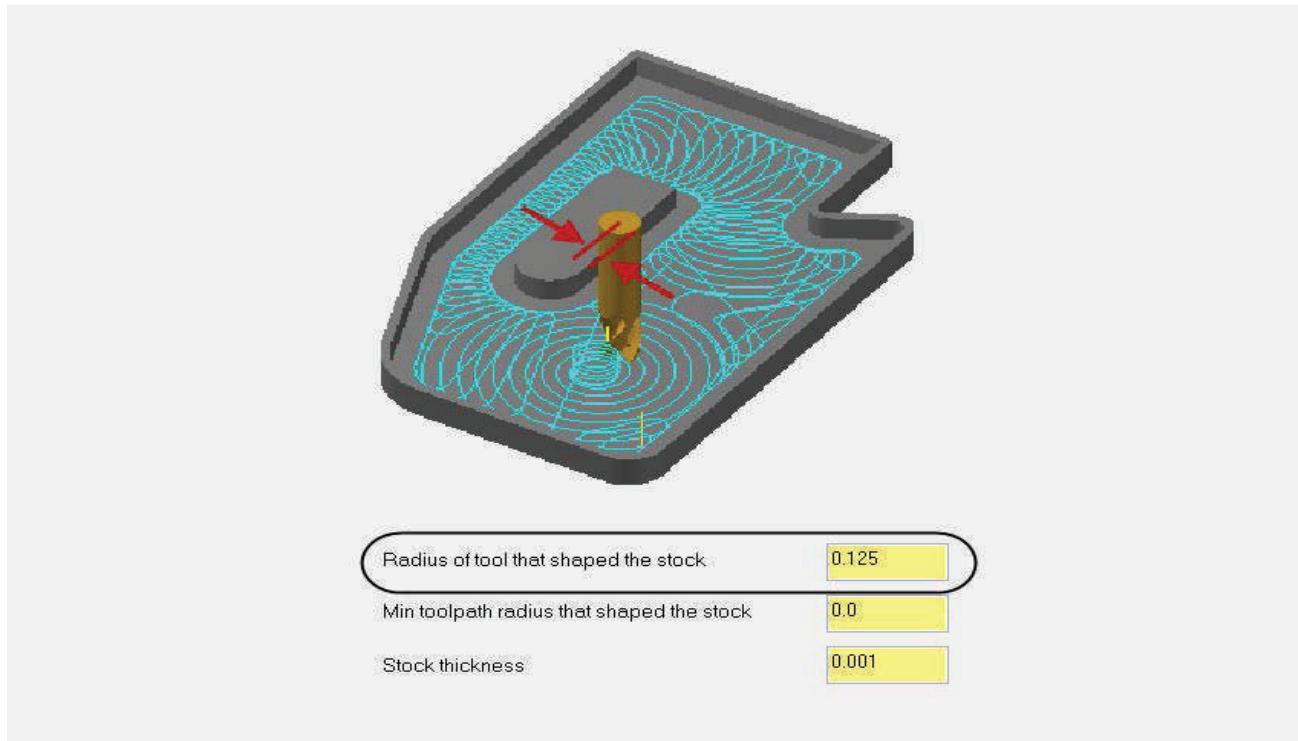
Min toolpath radius sets the minimum toolpath radius used in combination with the **Micro lift distance** and **Back feedrate** parameters to calculate 3D arc moves between cut passes.



14.4 Contour Wall

- From the Tree View list, select **Contour Wall** and ensure your parameters appear as shown in [Figure: 14.4.1.](#)

Figure: 14.4.1



Radius of tool that shaped the stock is the radius of the tool used in a toolpath that already cuts this area. Mastercam calculates the stock to remove along the contour wall using the **Stock thickness** (required) and, if provided, the **Radius of the tool that shaped the stock** and the **Min toolpath radius that shaped the stock**.

In your case, in the previous contour operation you used a 0.25" Flat Endmill and no toolpath radius was required in the toolpath. As no stock was left in the contour, the stock thickness is the value of the toolpath tolerance.

NOTE: The graphics in the toolpath pages are changing based on the parameter field you click on. Your graphics might look different.

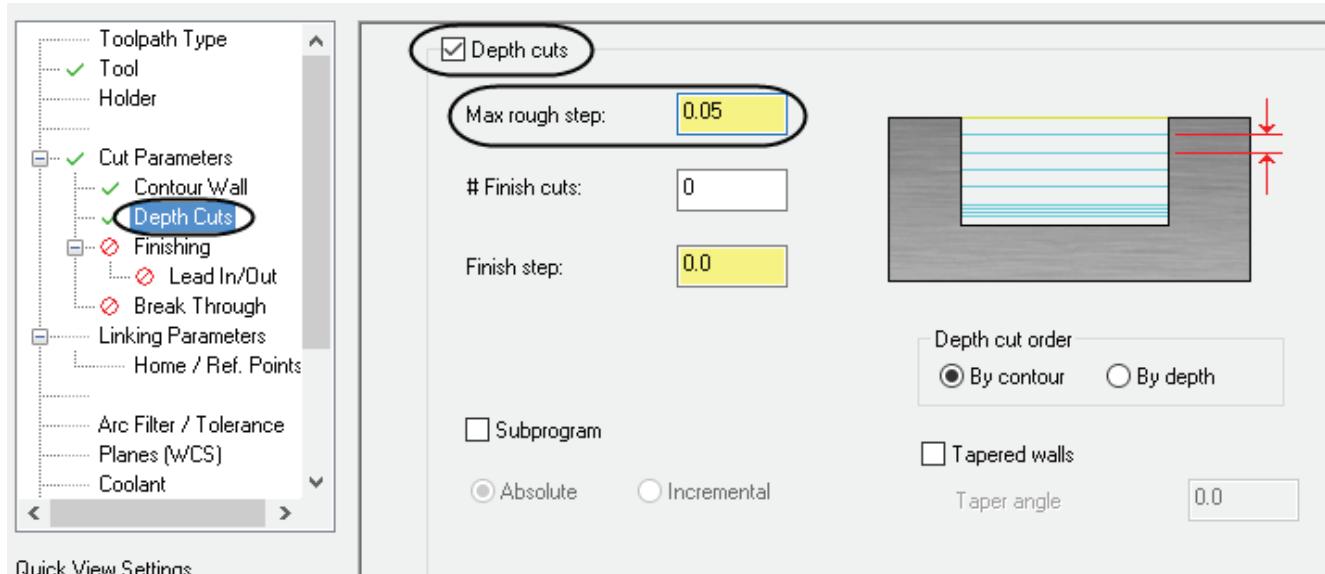
TUTORIAL #2

CLEAN THE INSIDE SHAPE USING 2D HS DYNAMIC CONTOUR



14.5 Depth Cuts

- From the **Tree View** list, choose **Depth Cuts** and enable it. Input a **Max rough step** of **0.05** as shown in [Figure: 14.5.1](#).



14.6 Break Through

- From the **Tree View** list, select **Break Through** and make the necessary changes as shown in [Figure: 14.6.1](#).

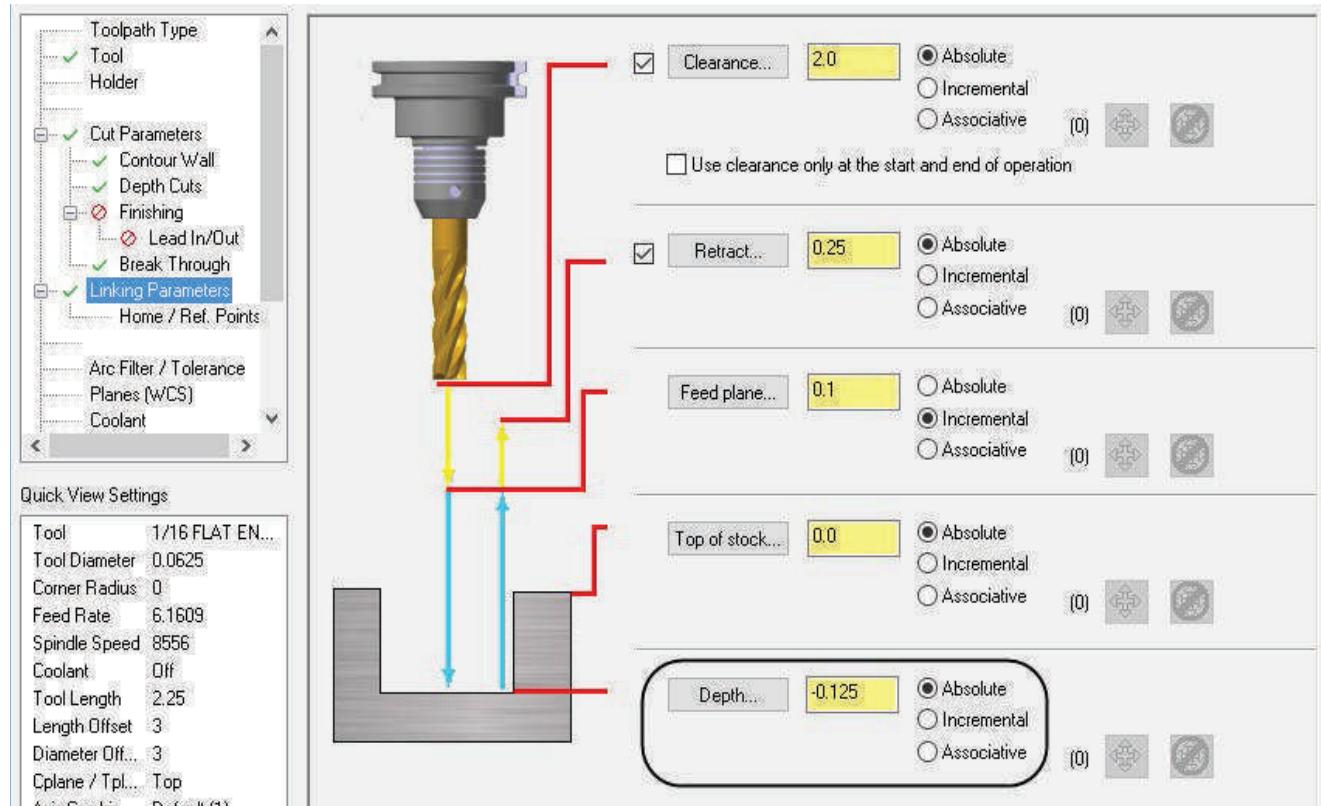




14.7 Linking Parameters

- From the Tree View list, select **Linking Parameters**.
- Set the **Depth** to **-0.125** as shown in [Figure: 14.7.1](#).

Figure: 14.7.1



14.8 Preview the Toolpath

- To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- To hide the dialog box, click on the **Hide dialog** icon as shown.

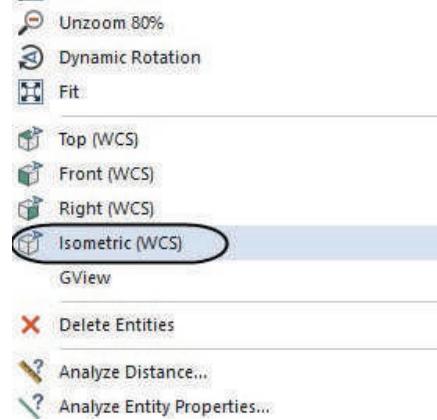


TUTORIAL #2

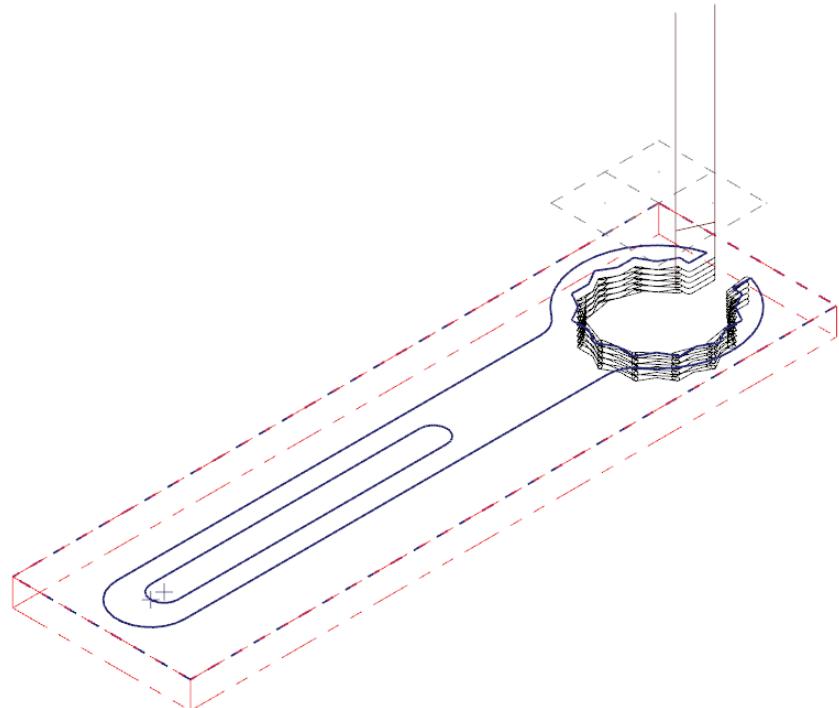
CLEAN THE INSIDE SHAPE USING 2D HS DYNAMIC CONTOUR



- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.



- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to exit the **Dynamic Contour** parameters.



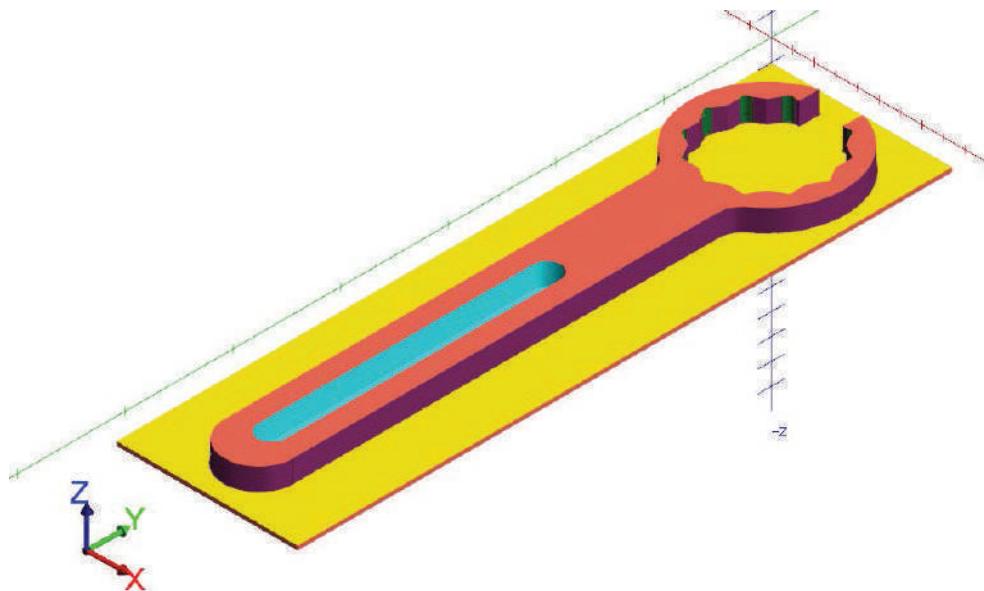


14.9 Backplot and Verify

- ♦ To **Backplot** and **Verify** your toolpath, see page 145 to review these procedures.
- ♦ To select all the operations, from the **Toolpaths Manager**, click on the **Select all operations** icon.



- ♦ The part should look as shown.



- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.

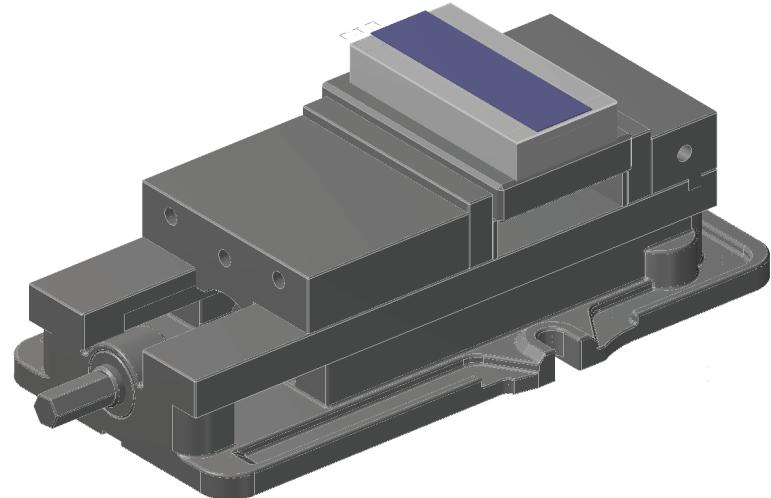
TUTORIAL #2

SUGGESTED FIXTURE:



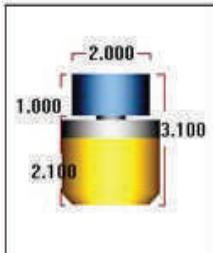
TOOLPATH CREATION - SETUP 2

SUGGESTED FIXTURE:



NOTE: In order to machine this part, we will have 2 setups and output 2 NC files.

SETUP SHEET:

TOOL LIST	
 <p>2.000 1.000 3.100 2.100</p>	<p>TYPE: Face mill DIA OFFSET: 4 HOLDER: Default Holder NUMBER: 4 LENGTH OFFSET: 4 #4 - 2.0000 FACE MILL - 2" FACE MILL</p>



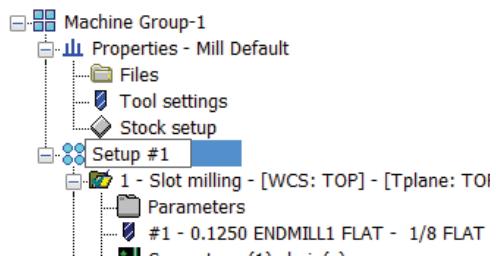
STEP 15: CREATING AND RENAMING TOOLPATH GROUPS

To machine the part in two different setups, we will need to have two separate programs. To be able to post process separately the operations of each setup, we will create them under different toolpath groups with different NC names.

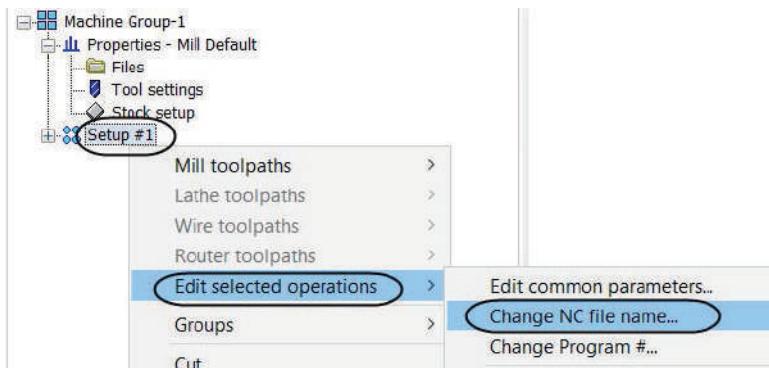
15.1 Rename the current Toolpath Group - 1 and NC file

- Click once on the **Toolpath Group - 1** to highlight it and then click again on it to rename it "Setup #1" as shown in [Figure: 15.1.1](#).

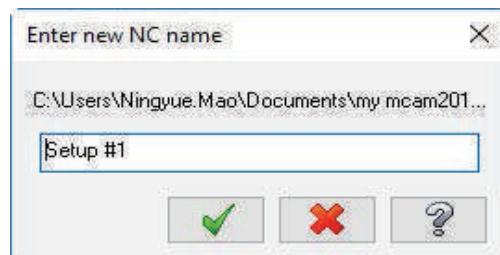
Figure: 15.1.1



- Right mouse click on the **Setup #1** Toolpath group, select **Edit selected operations**, then select **Change NC file name**.



- Enter the new NC name: **Setup #1**.

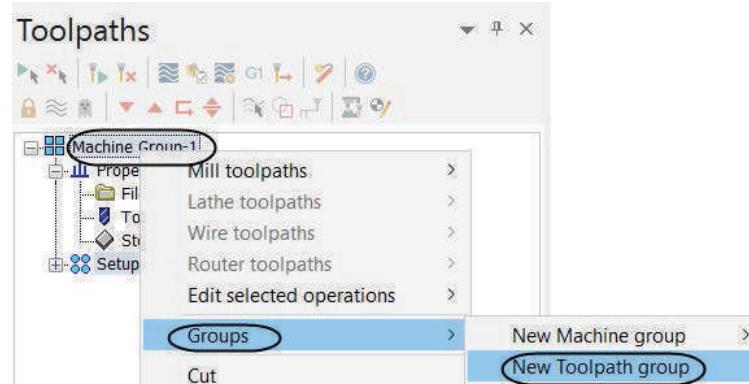


- Select the **OK** button to accept the new NC name.

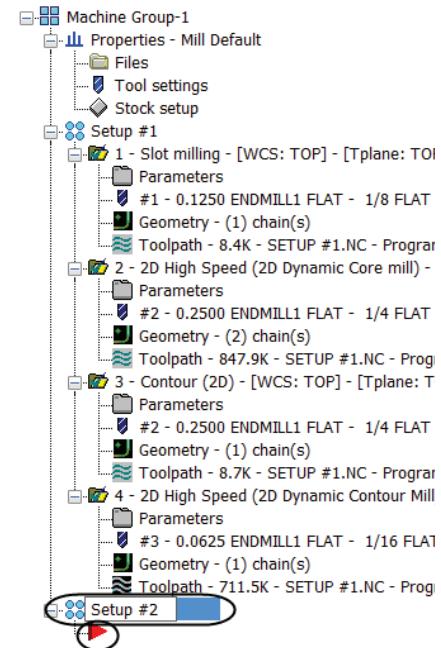


15.2 Create a new Toolpath Group

- Right mouse click on the **Machine Group-1** and select **Groups** and then **New Toolpath group** as shown.



- Rename the toolpath group "**Setup #2**" as shown.



NOTE: The red insert arrow controls where the new operation will be inserted. In our case it should be located below the **Setup #2** group.

- If the insert arrow needs to be moved, from the **Toolpaths Manager**, click on the **Move insert arrow down one item** icon.

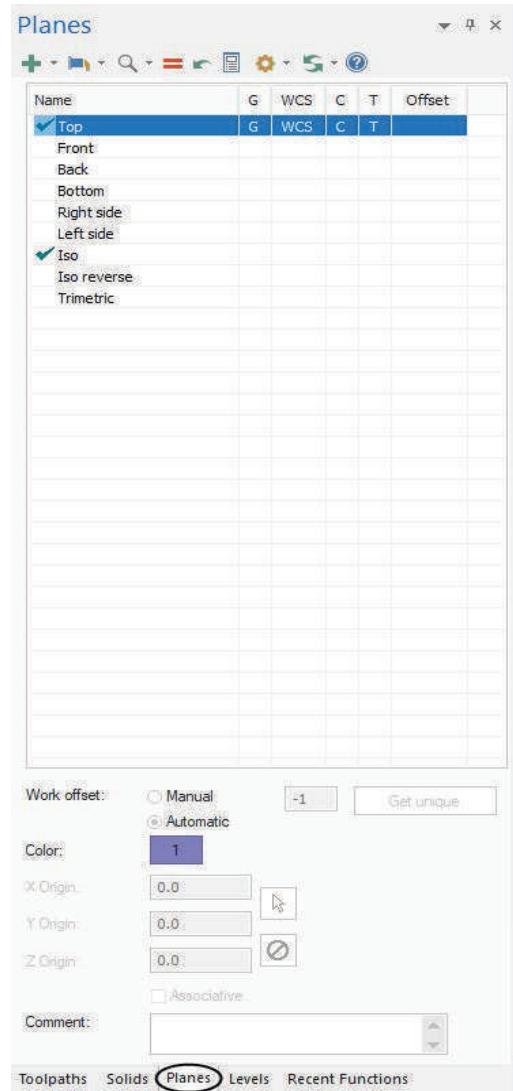




STEP 16: SET WCS TO BOTTOM

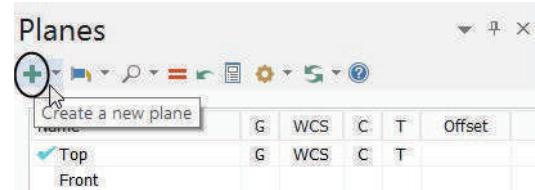
Work coordinate system (WCS) is the active coordinate system in use by Mastercam at any given time. The WCS contains the orientation of the X-Y-Z axes plus the location of the zero point (the origin). This tells Mastercam how your part is positioned or orientated in the machine.

- ♦ Select the **Planes** tab located at the bottom left corner.

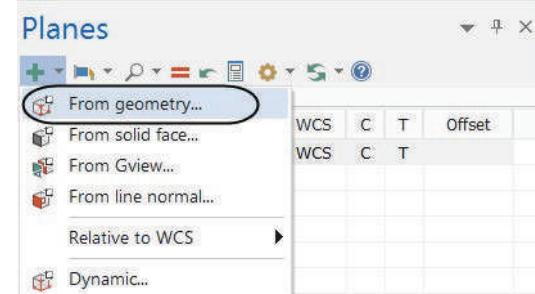




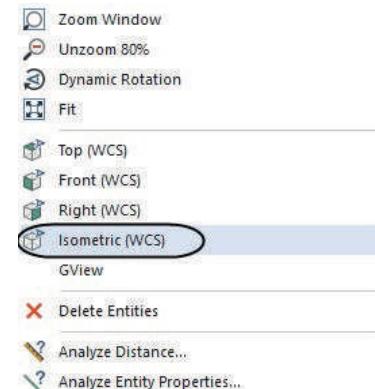
- To create a new plane based on existing geometry, click on the + sign as shown.



- Select **From geometry** as shown.



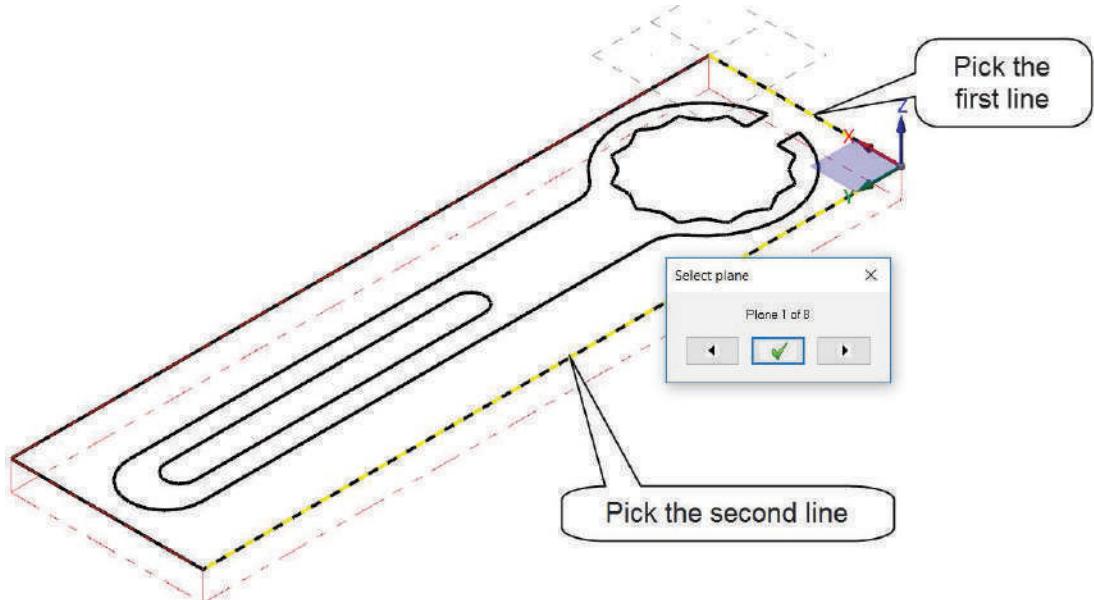
- Right mouse click in the graphics window and select the **Isometric** view.





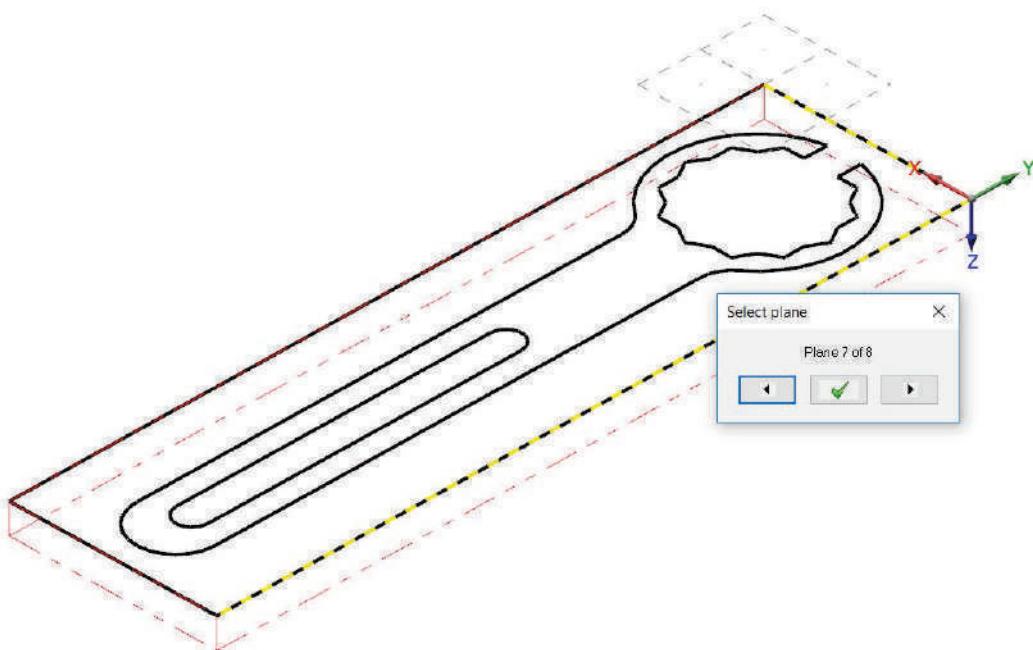
- ♦ Select the first line along the **X axis** of the new view and then select the line along the **Y axis** of the new view as shown in [Figure: 16.0.1](#).

Figure: 16.0.1



- ♦ In **Select plane**, click on the **Next plane** button until the axes are oriented as shown in [Figure: 16.0.2](#).

Figure: 16.0.2



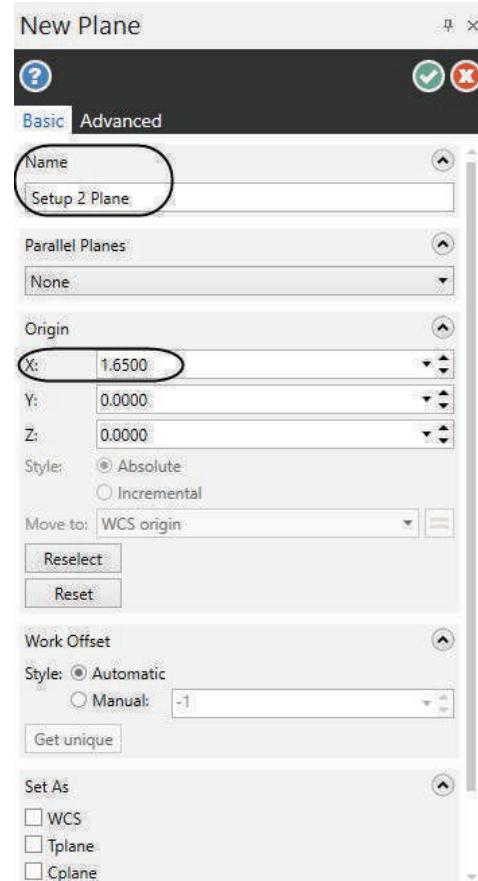
- ♦ From the **Select plane**, click on the **OK** button to continue.

TUTORIAL #2

SET WCS TO BOTTOM



- In the New Plane panel, in the Name field, input **Setup 2 Plane** as shown. Make sure that the **X coordinate** for the **Origin** is **1.65**.



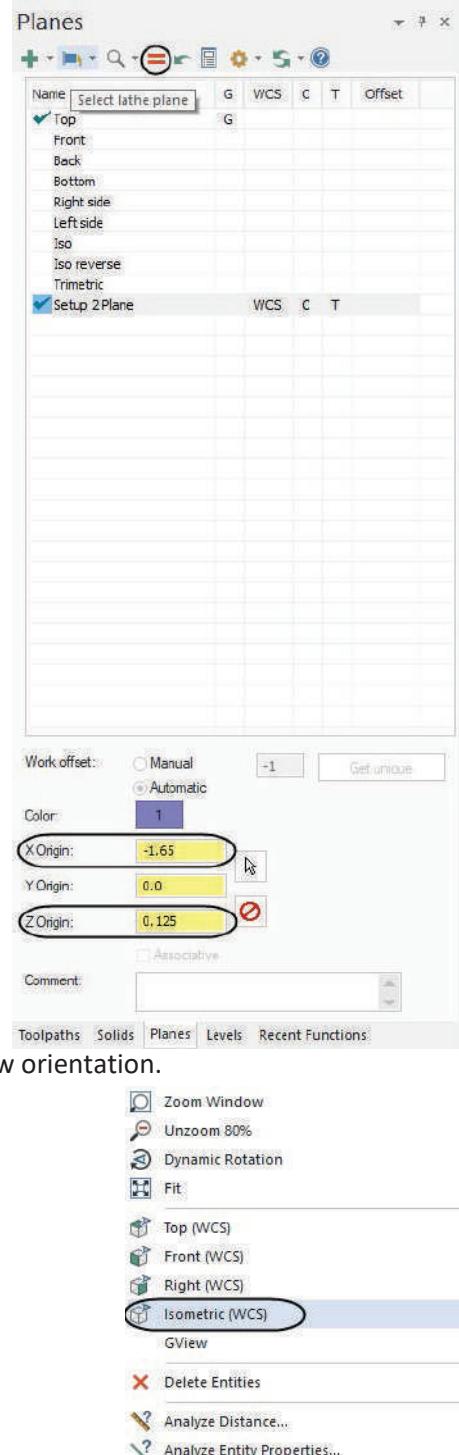
- Select the **OK** button to continue.





- Now we set the **Work Coordinate System (WCS)**, **Tool plane**, and **Construction plane** to the newly created **Bottom View** by clicking on the equal icon, with **Bottom Plane** highlighted in blue, as shown in [Figure: 16.0.3](#).
- Set the **Origin Z** to **0.125** as shown in [Figure: 16.0.3](#).

Figure: 16.0.3



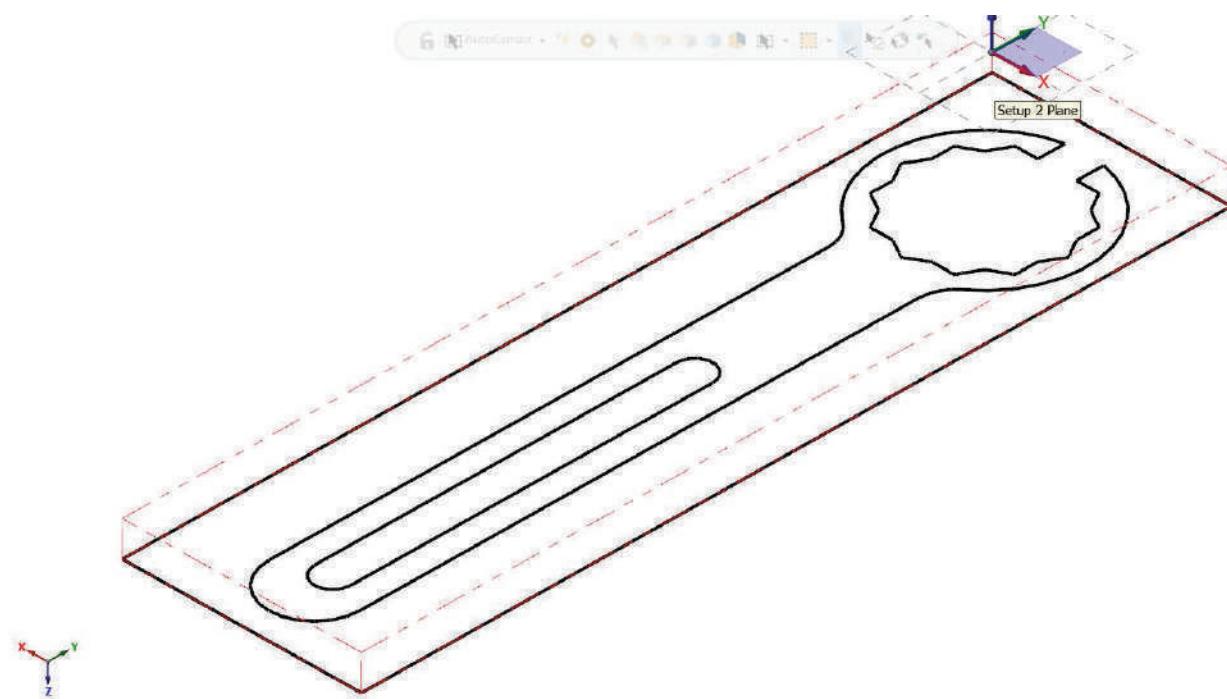
- Right mouse click and select the **Isometric** view to see the part in its new orientation.



TUTORIAL #2 SET WCS TO BOTTOM



- The part should look as shown.



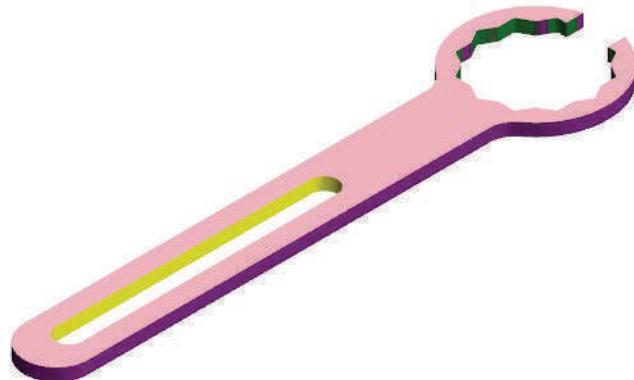
NOTE: Z zero is at **0.125** below the stock.



STEP 17: FACE THE PART

A **Facing** toolpath quickly removes material from the top of the part to create an even surface.

Toolpath Preview:



TOOLPATHS

- From the **2D** group, click on the upper arrow until you see the **Face** icon and select it as shown.



- When the **Chaining** dialog box appears, choose the **OK** button to use defined stock and exit the **Chaining** dialog box.



NOTE: Mastercam will create the **Facing** toolpath defined from the stock setup.



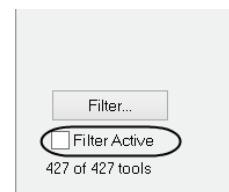
- In the Toolpath Type page, the **Facing** icon will be automatically selected.



17.1 Select a 2.0" Face Mill from the library and set the Tool parameters

- Select **Tool** from the **Tree View** list.

- Click on the **Select library tool...** button.
- To be able to see all the tools from the library, disable **Filter Active**.



- Pick the **2" Face Mill (#322)** as shown.

316	-	1-1/2 BAL...	-	1.5	0.75	2.75	En...	4	Full
317	-	2 INCH B...	-	2.0	1.0	2.75	En...	4	Full
318	-	1/4 CHA...	-	0....	0.0	0.5	Ch...	4	No...
319	-	1/2 CHA...	-	0....	0.0	0.75	Ch...	4	No...
320	-	3/4 CHA...	-	0....	0.0	1.0	Ch...	4	No...
321	-	1 INCH C...	-	1....	0.0	1.0	Ch...	4	No...
322	-	2" FACE ...	-	2.0	0.0	1.575	Fac...	2	No...
323	-	2-1/2" FA...	-	2.5	0.0	1.575	Fac...	4	No...
324	-	3" FACE ...	-	3.0	0.0	1.969	Fac...	4	No...
325	-	4" FACE ...	-	4.0	0.0	1.969	Fac...	2	No...

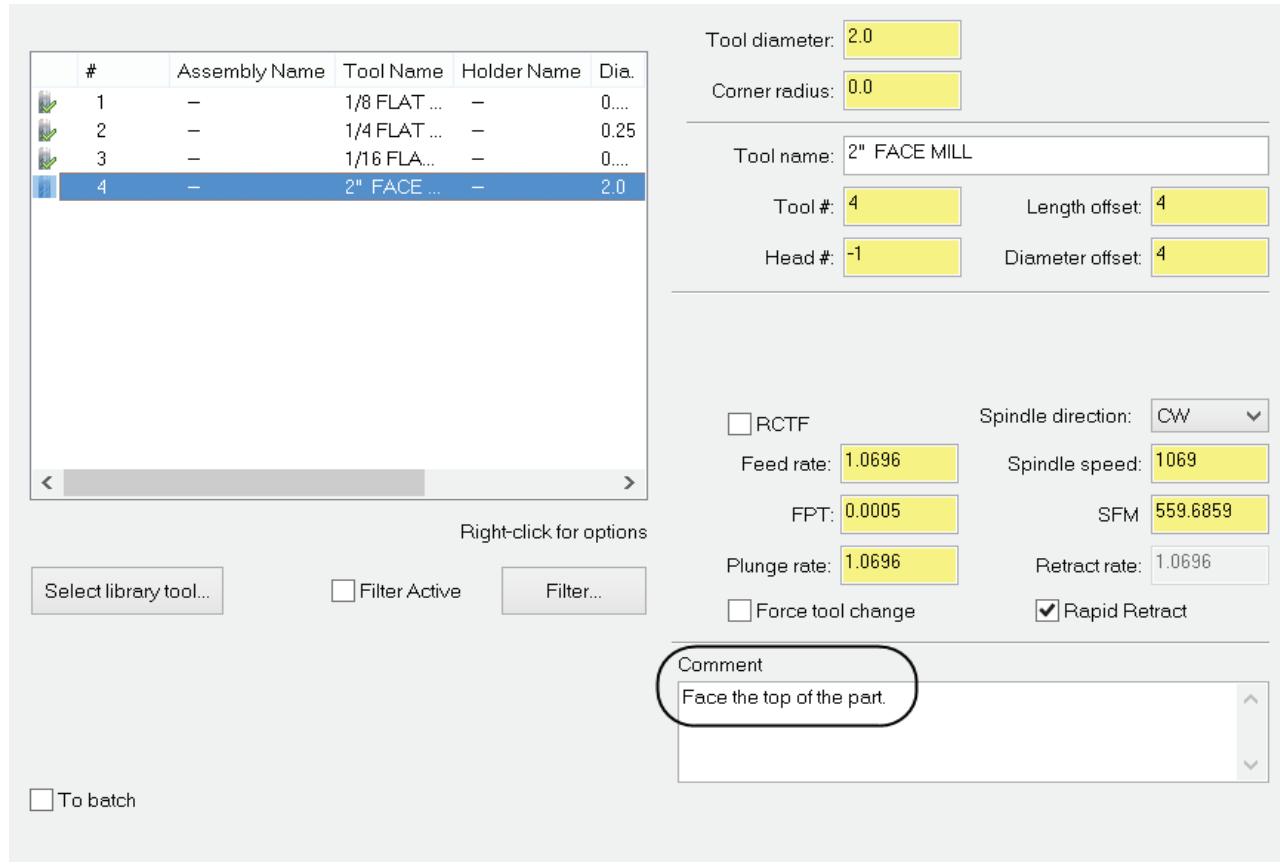
- Select the tool in the **Tool Selection** page and then select the **OK** button to exit.





- ♦ Make all the necessary changes as shown in [Figure: 17.1.1.](#)

Figure: 17.1.1



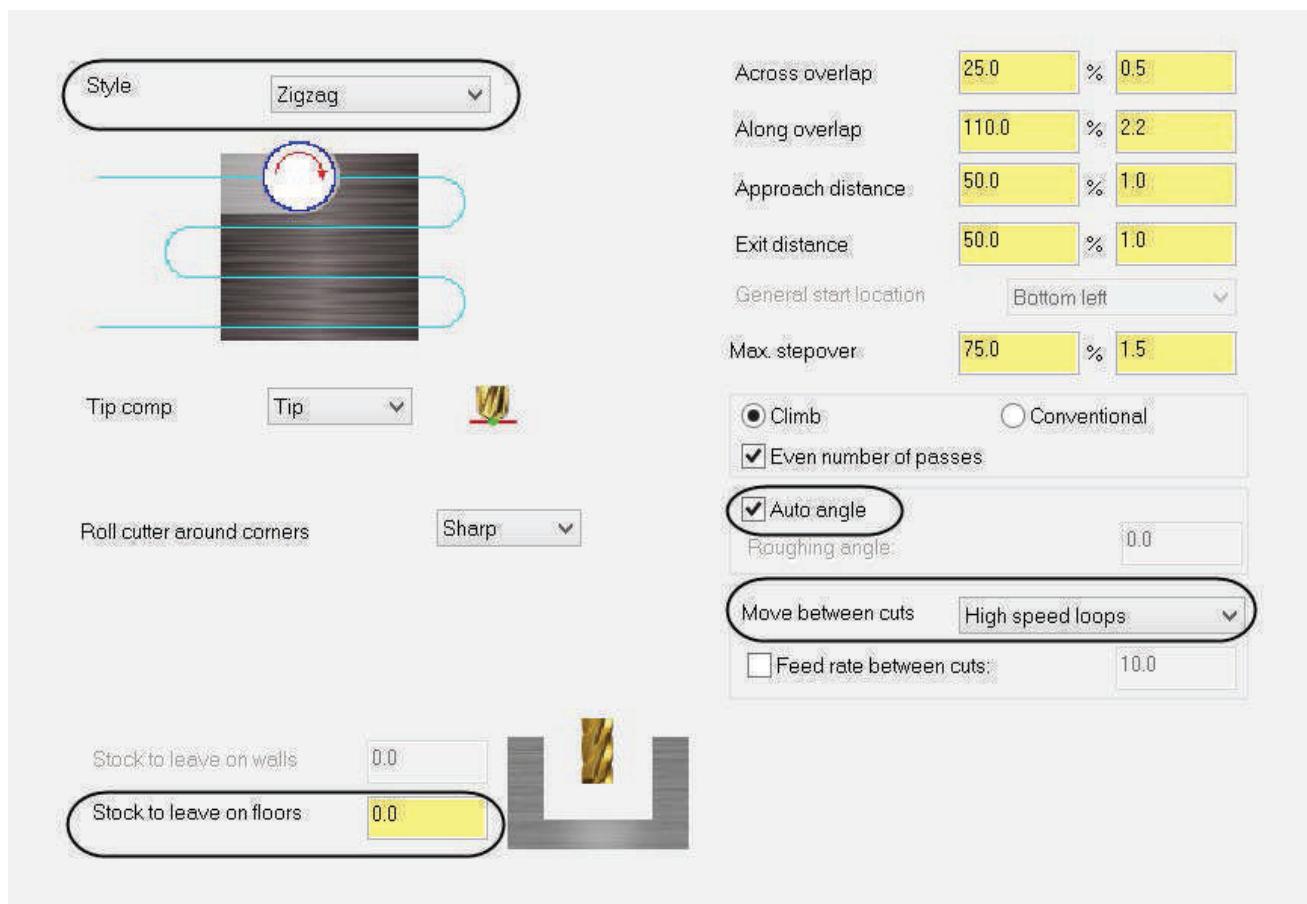
The **Feed rate**, **Plunge rate**, **Retract rate** and **Spindle speed** are based on the tool definition as set in the **Tool Settings**. You may change these values as per your part material and tools.

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



- From the Tree View list, select **Cut Parameters** and make the necessary changes as shown in [Figure: 17.1.2.](#)

Figure: 17.1.2



The **Style** (facing cutting method) **Zigzag** creates a back and forth cutting motion.

Auto angle determines the angle to machine along the larger side of the stock.

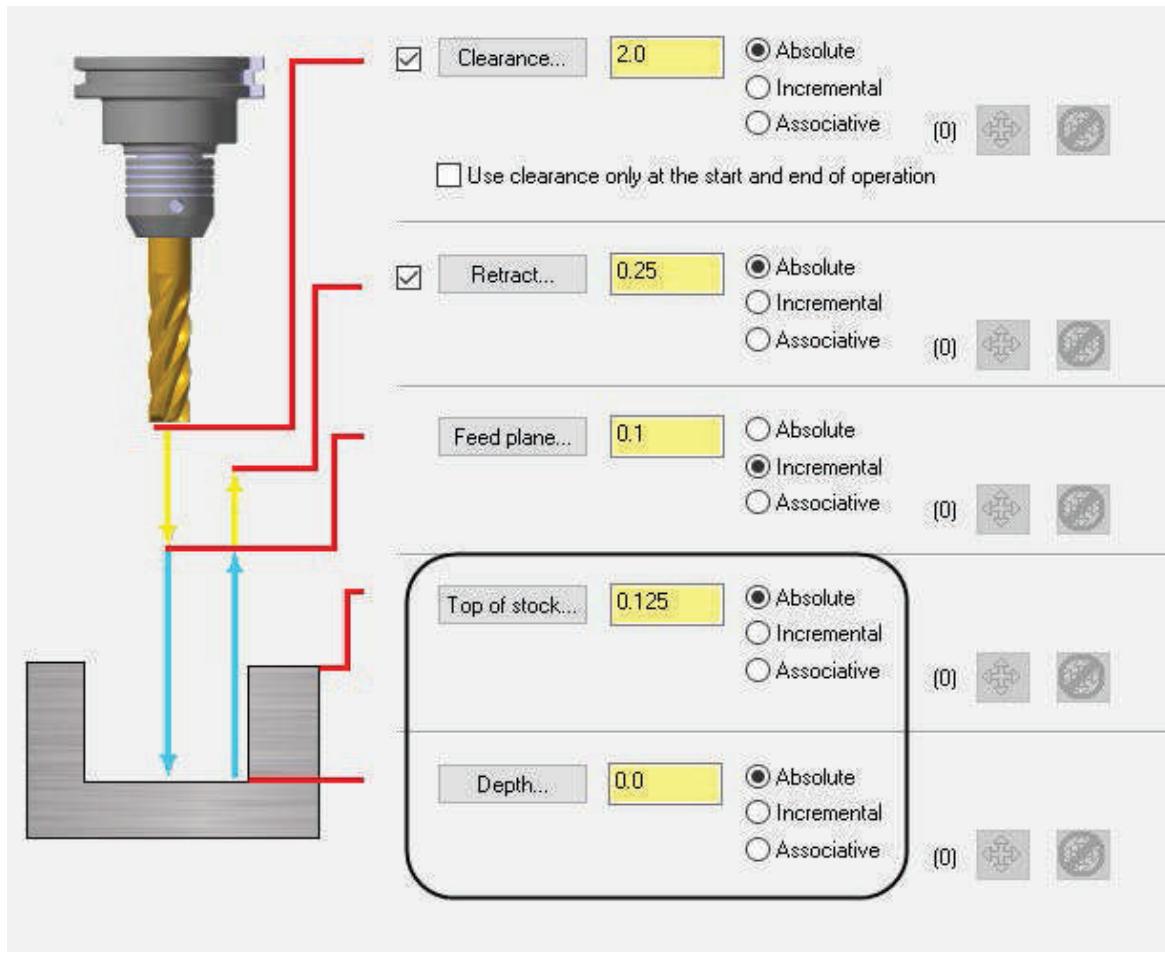
Move between cuts determines how the tool moves between each cut. This is only available if you select the zigzag cutting method.

NOTE: If **Depth Cuts** is not disabled, select it from the **Tree View list** and disable it.



- From the Tree View list, select the **Linking Parameters** page and make the necessary changes as shown in [Figure: 17.1.3](#).

Figure: 17.1.3



Clearance sets the height at which the tool moves to and from the part.

Retract sets the height that the tool moves up to before the next tool pass.

Feed plane sets the height that the tool rapidts to before changing to the plunge rate to enter the part.

Top of stock sets the height of the material in the Z axis.

Depth determines the final machining depth that the tool descends into the stock.

NOTE: The top of stock is set to **0.125"** because in our **Bottom view** we have **Origin Z** value set to **0.125"** above the first setup origin. The depth is set to **0.0"** because this is the depth of the finish part we want the tool to go to.

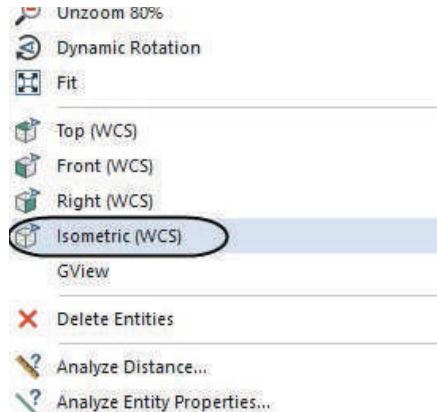


17.2 Preview the Toolpath

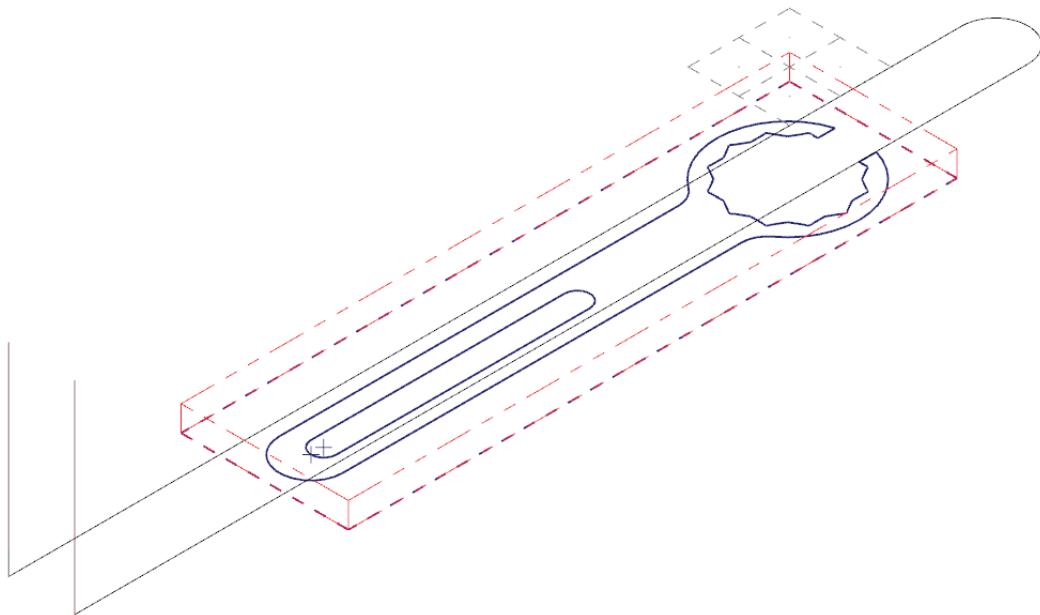
- ♦ To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.
- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.



- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

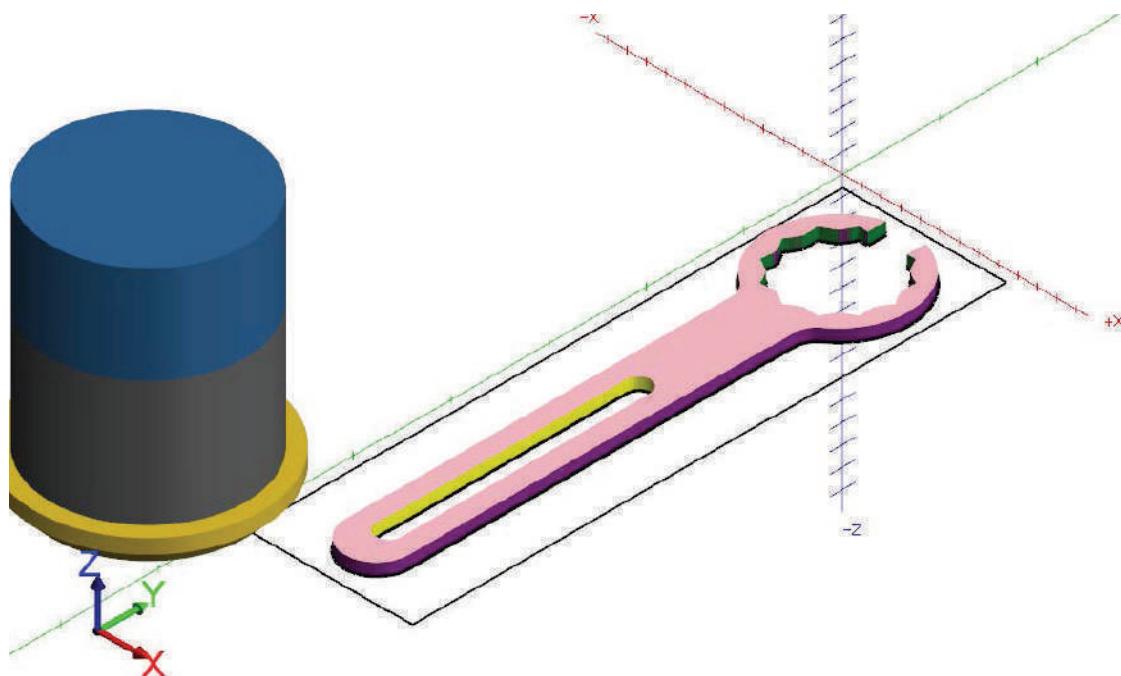
- ♦ Select the **OK** button to exit the **Facing** parameters.
- ♦ To **Backplot** and **Verify** your toolpath, see page 145 to review these procedures.



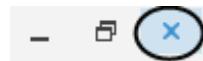
- To select all the operations, from the **Toolpaths Manager**, click on the **Select all operations** icon.



- The toolpaths should look as shown.



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

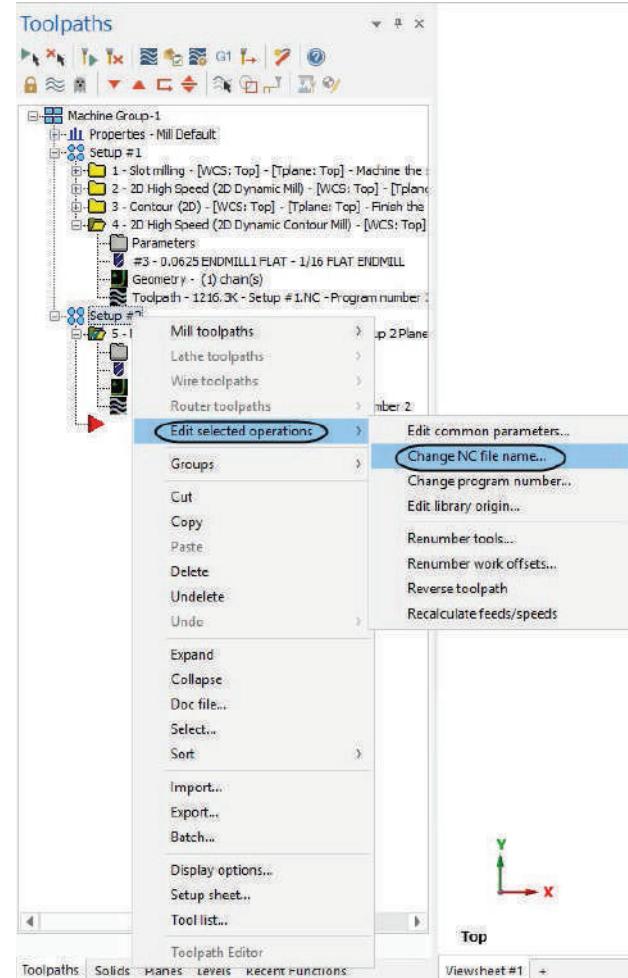




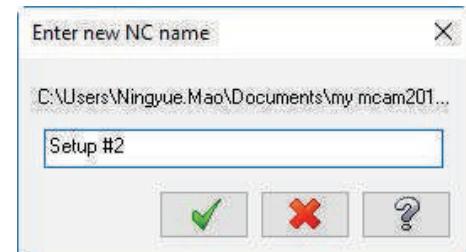
STEP 18: RENAME THE NC FILE

The Facing operation in Setup #2 kept the NC name from Setup #1. We need to rename this operation.

- ♦ Make sure that only the **Facing** is selected and right click on **Setup #2** group, choose the option **Edit selected operations** and then pick **Change NC file name**.



- ♦ When the **Enter new NC name** dialog box appears, enter "**Setup #2**" in the input field.



- ♦ Select the **OK** button to apply the change and exit the panel.





- You should see **Setup #2.NC** in the last item of text for Operation #5.



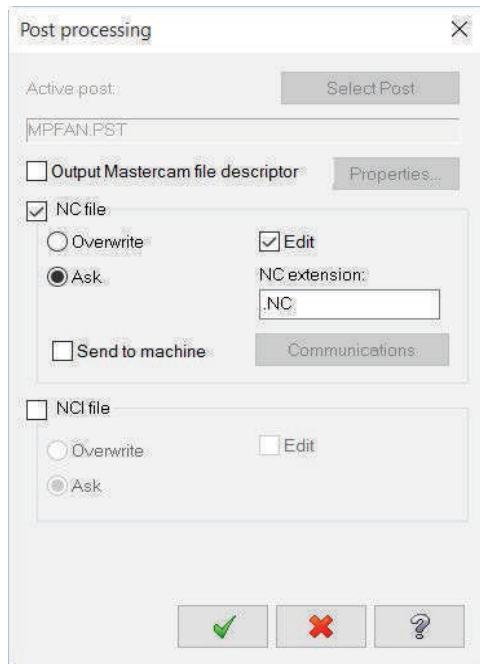
STEP 19: POST THE FILE

- Ensure all operations are selected. If they are not, use the button **Select all operations** in the **Toolpaths Manager**.



- Select the **Post selected operations** button from the **Toolpaths Manager**.
- In the **Post processing** window, make any necessary changes as shown in [Figure: 19.0.1](#).

Figure: 19.0.1



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

Edit enabled allows you to automatically launch the default editor.

- Select the **OK** button to continue.
- Save Setup #1 NC file.
- Save Setup #2 NC file.

TUTORIAL #2

SAVE THE UPDATED MCAM FILE



- ♦ A window with **Mastercam Code Expert** will be launched and the NC program will appear as shown in [Figure: 19.0.2](#).

Figure: 19.0.2

The screenshot shows the Mastercam Code Expert interface. The title bar reads "Setup #2.NC - Mastercam 2018 Code Expert". The menu bar includes File, Home, View, Editor (which is selected), NC Functions, and Utilities. The toolbar contains icons for Insert Block Numbers, Insert Block Skip, Send File, Go To, Remove Block Skip, Remove Block Skip, Remove Comments, Send, Receive, First, Previous, Next, Last, Mark, Multi-Stream, and NC Configuration. Below the toolbar are tabs for Editing, Communications, Syncs, Tools, and Utilities. The main area displays an NC program titled "Setup #1.NC" and "Setup #2.NC". The code listing is as follows:

```
1 G0002 (SETUP #2)
2 (DATE=00-MM-YY - 14-02-17 TIME=HH:MM - 10:23)
3 (NCX FILE = \\\IBSSERVER06\BOOKDEV\PRODUCTION\BOOKS\2018\TRAINING TUTORIALS\TT CAD IMPORT MILL ESSENTIALS TOOLPATHS\MCX FILES\INCH\TUTORIAL
4 (NC FILE = C:\USERS\NINGYUE.MAO\Desktop\SETUP #2.NC)
5 (MATERIAL = ALUMINUM INCH - 2024)
6 (T4 I 2" FACE MILL I H4 )
7 N100 G20
8 N110 G0 G17 G40 G49 G90 G90
9 ( FACE THE TOP OF THE PART. )
10 N120 T4 M6
11 N130 G0 G90 G54 X-2.7999 Y-5.2 Z0. S1069 M3
12 N140 G43 H4 22,
13 N150 Z-.225
14 N160 G1 Z0, F1.07
15 N170 Y1.2
16 N180 G2 X-2.475 Y1.5249 I.3249 J0.
17 N190 K-2.1501 Y1.2 I0. J-.3249
18 N200 G1 Y-5.2
19 N210 G0 22,
20 N220 M5
21 N230 G91 G28 Z0,
22 N240 G28 X0, Y0, Z0,
23 N250 M30;
24
25
26
```

The status bar at the bottom indicates "Ready", "Ln 1/26 Col 2 705 bytes 100%", and zoom controls.

- ♦ Select the "X" at the upper right corner to exit the editor.

STEP 20: SAVE THE UPDATED MCAM FILE



REVIEW EXERCISE - STUDENT PRACTICE

IMPORT THE DWG FILE FOR TUTORIAL #2 EXERCISE

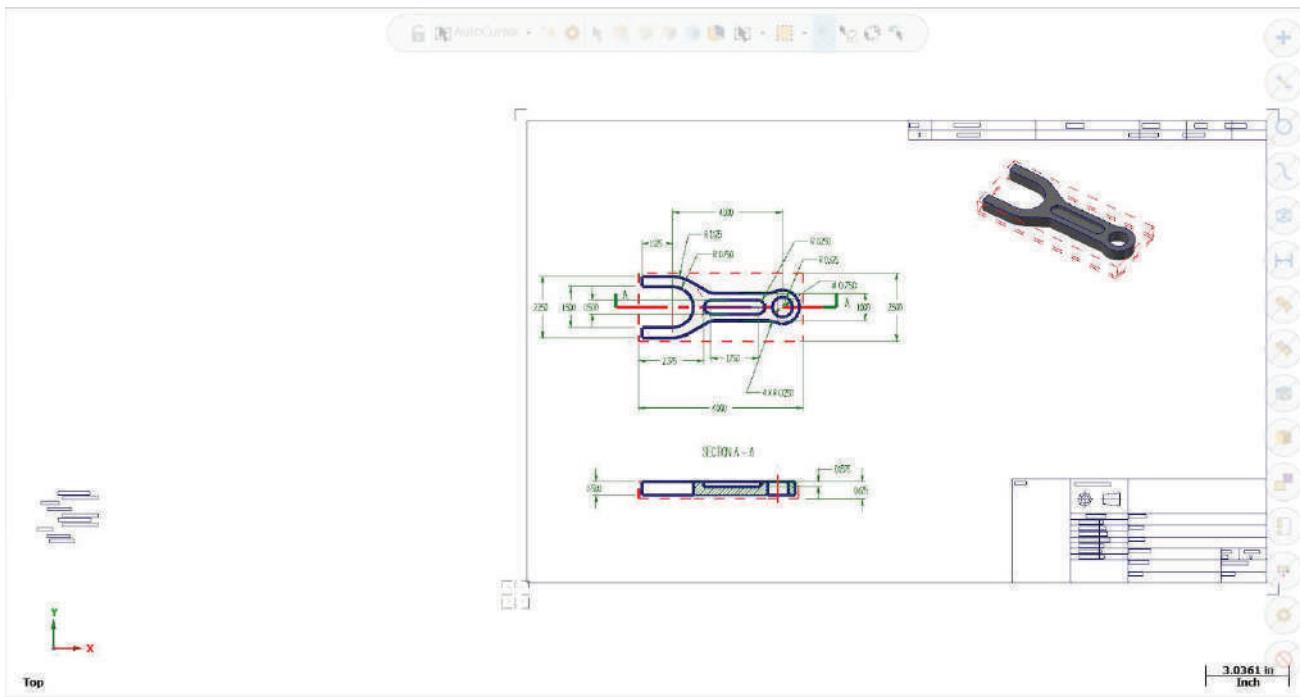
Import the dwg file and manipulate the geometry to prepare it for the toolpath creation.

Download the files from www.emastercam.com/trainingfiles.

Save the file in a preferred location.

Import **TUTORIAL_2 EXERCISE.DWG**.

- ♦ Use file **Open** and change the file type to AutoCAD (*.dwg; *.dxf...).



Move the geometry to level 100.

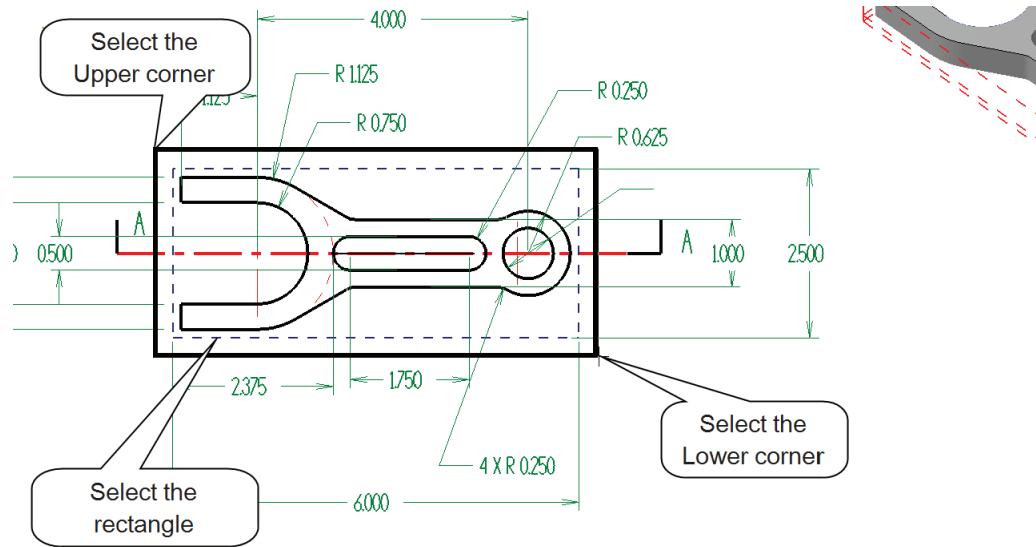
- ♦ Create the **Main Level 100** and name it **Geometry**.

TUTORIAL #2

IMPORT THE DWG FILE FOR TUTORIAL #2 EXERCISE



- ♦ Hold down the **Shift** key and click on the rectangle as shown.
- ♦ Use the **Select only advanced** option to select the color blue number **1** and make a window around the top view as shown.

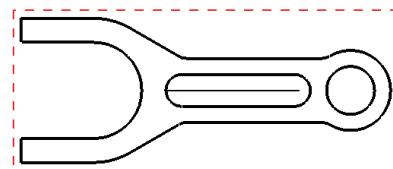


SECTION A - A

- ♦ Right mouse click on the graphics window and select the **Change Level** button to move the geometry on **Level 100**.

Make Level 100 the only visible level.

- ♦ In the **Levels Manager**, right mouse click on the **Level 100** and select **All level set off**.
- ♦ Delete the line in the middle of the slot.

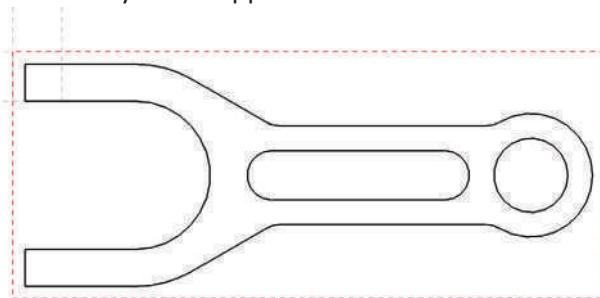


Check for any chaining problem.

- ♦ Use **Delete Duplicates** to delete the identical entities.
- ♦ Use **Findoverlap C-Hook** to delete overlapping entities.

Move the upper corner to the origin.

- ♦ Use **Transform Move to Origin** and select the endpoint of the entity at the upper left corner.





CREATE THE TOOLPATHS FOR TUTORIAL #2 EXERCISE

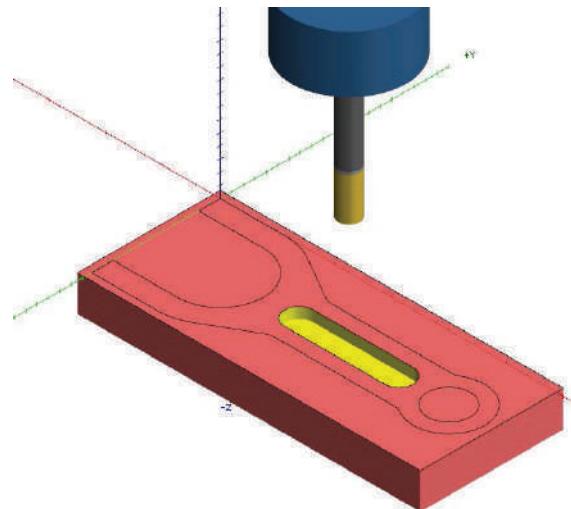
Create the Toolpaths for Tutorial #2 Exercise as per the instructions below.

Setup #1

Set the machine properties including the stock setup.

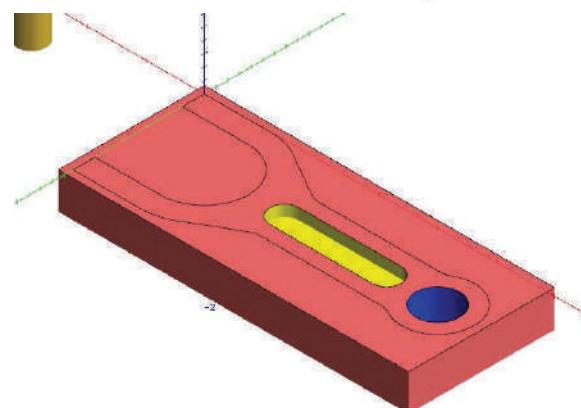
Slot Mill the Obround Slot.

- ◆ Remove the material in the slot.
- ◆ Use a **1/4" Flat Endmill**.
- ◆ **Stock to leave on walls/floors = 0.0**.
- ◆ Set the **Depth** according to the part.



Circle Mill the 3/4" Hole.

- ◆ Choose a **3/8" Flat Endmill**.
- ◆ Leave no stock on the walls or the floor.
- ◆ Enable **Roughing**.
- ◆ Set appropriate **Depth Cuts** and **Break through amount**.
- ◆ Input a **Depth** according to the part.



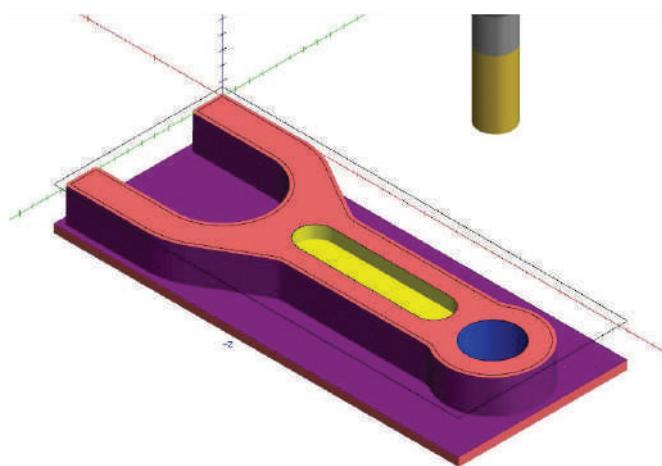
TUTORIAL #2

CREATE THE TOOLPATHS FOR TUTORIAL #2 EXERCISE



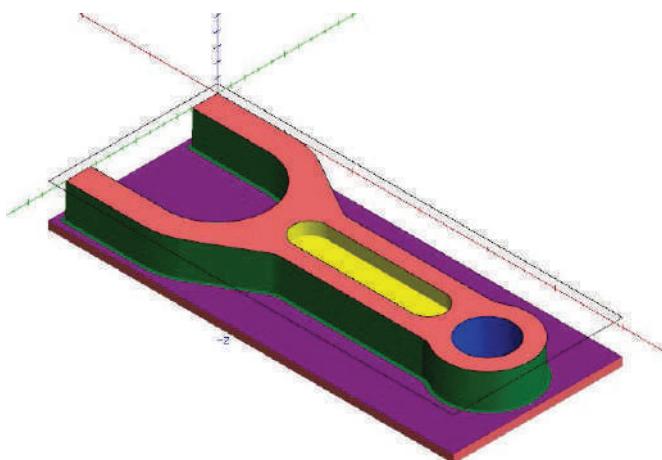
Rough the outside profile using 2D HS Dynamic Mill.

- ◆ Select the outside rectangle in the **Machining regions**.
- ◆ Enable **From outside**.
- ◆ Select the profile in the **Avoidance regions**.
- ◆ Use a **1/2" Flat Endmill**.
- ◆ Leave stock on the wall only.
- ◆ Set the **Depth** according to the part.



Finish the outside profile using Contour.

- ◆ Use a **1/2" Flat Endmill**.
- ◆ Set the final **Depth** according to the part.

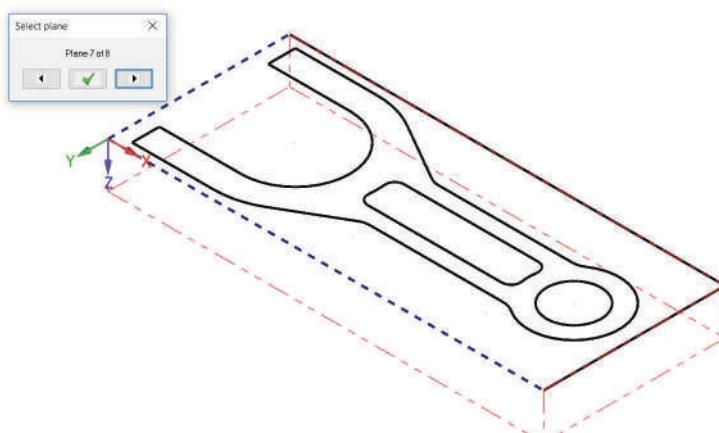


Setup #2.

- ◆ Rename the previous Toolpath Group.
- ◆ Rename the existing operation NC file.
- ◆ Create a new Toolpath Group and rename it.

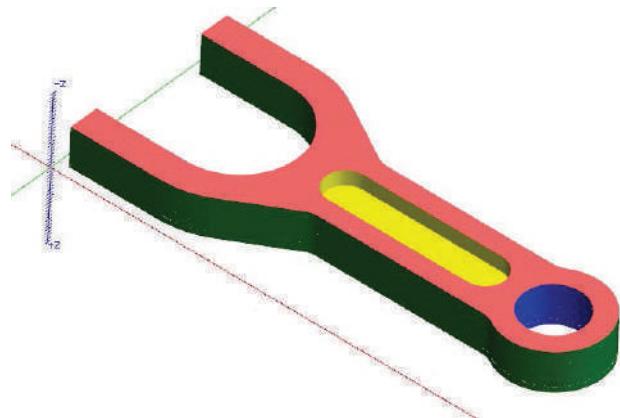
Use WCS Plane Manager and set the Bottom plane.

- ◆ Use **From geometry** to define the plane.
- ◆ Set **WCS**, **Cplane** and **Tplane** to the new view.
- ◆ Set the **Z origin** to **-0.5**.



**Face the part (Contour 2D).**

- ♦ Select the 2" Face Mill from the **Tool** page.
- ♦ Set the **Depth** according to the part.



**NOTES:**

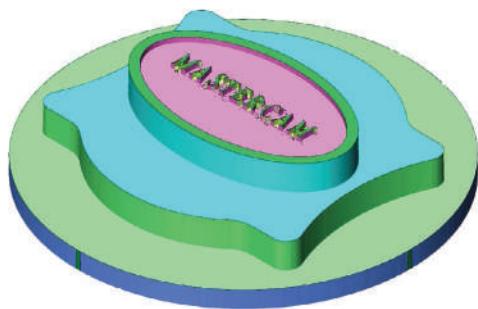


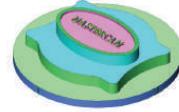
TUTORIAL #2 QUIZ

- ◆ What does the Slot Mill toolpath do?
 - ◆ What does the 2D HS Dynamic Mill toolpath do?
 - ◆ What does the 2D HS Dynamic Contour Mill toolpath do?
 - ◆ What is the process used to be able to post different operations as different programs?



TUTORIAL #3





OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

Import the 2D CAD Model and prepare it to generate the Toolpaths from:

- ◆ The student will open the AutoCAD DWG file in Mastercam.
- ◆ The student will move the solid geometry needed to create the toolpaths on a different level.
- ◆ The student will use WCS to set a new top plane.
- ◆ The student will use Transform Dynamic to rotate the part.

Create the necessary Toolpaths to machine the part:

- ◆ The student will set up the stock size to be used and the clamping method used.
- ◆ A 2D High Speed Area Mill toolpath will be created to remove the material from the outside step.
- ◆ A 2D High Speed Dynamic Mill toolpath will be created to remove the outside material.
- ◆ A 2D High Speed Area Mill with Island Facing toolpath will be created to machine the pocket with the islands and face the letters.
- ◆ A Pocket Remachining toolpath will be used to machine the remaining material.

Backplot and Verify the file:

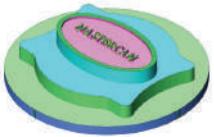
- ◆ Backplot will be used to simulate a step-by-step process of the tool's movements.
- ◆ Verify will be used to watch a tool machine the part out of a solid model.

Post Process the file to generate the G-code:

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



This tutorial takes approximately one hour to complete.



GEOMETRY CREATION

STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

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

STEP 2: IMPORTING THE DWG FILE GEOMETRY

Mastercam lets you read (import) a variety of CAD file types into the Mastercam database. You can also write (export) Mastercam files to a variety of different file formats.

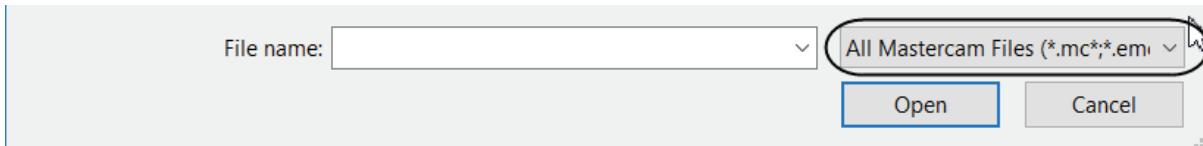
To import an AutoCAD file in Mastercam, you have to use the Open function and then select from the list of file types the AutoCAD files.

Download the files from www.emastercam.com/trainingfiles.

Save the file at a preferred location.

FILE

- ♦ **Open.**
- ♦ In the file name extension, click on the drop down arrow as shown.



- ♦ From the list of file types, select **AutoCad Files (*.DWG;*.DXF;*.DWF;*.DWFX)** as shown.

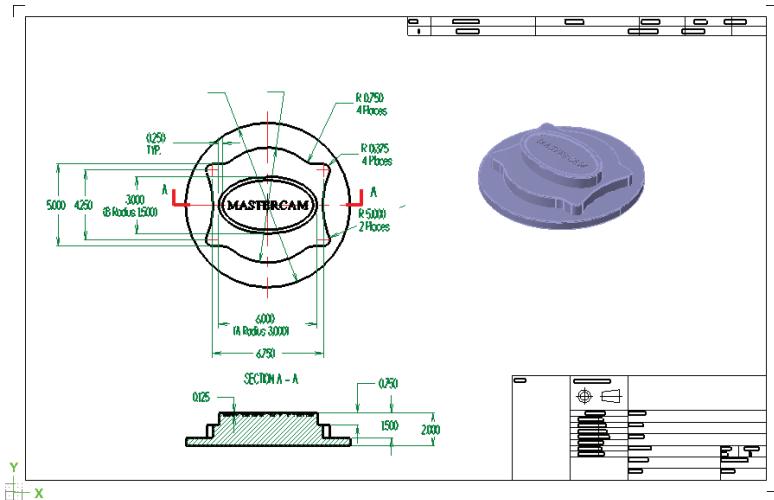
Mastercam Educ X7 Files (*.emcx-7)
Mastercam Educ X6 Files (*.emcx-6)
Mastercam Educ X5 Files (*.emcx-5)
Mastercam Educ pre-X5 Files (*.emcx)
Mastercam V9 Files (*.mc9)
Mastercam V8 Files (*.mc8)
All Mastercam Files (*.mc**;.emc*)
IGES Files (*.igs;*.iges)
AutoCAD Files (*.dwg;*.dxf;*.dwf;*.dwfx)
Parasolid Files (*.x_t;*.x_b;*.xmt;*.txt)
ProE/Creo Files (*.prt;*.asm;*.prt;*.asm;*.sat)
ACIS Kernel SAT Files (*.sat;*.sab)
STEP Files (*.stp;*.step)
VDA Files (*.vda)
Rhino 3D Files (*.3dm)
SOLIDWORKS Files (*.sldprt;*.sldasm)
SOLIDWORKS Drawing Files (*.slddrw)
Solid Edge Files (*.par;*.psm;*.asm)
Autodesk Inventor Files (*.ipt;*.iam)
Autodesk Inventor Drawing Files (*.idw)
KeyCreator Files (*.ckd)

- ♦ Find and select **TUTORIAL_3.DWG**.
- ♦ Refer to **page 113** for setting the **Options** before opening the file.
- ♦ Open the file.



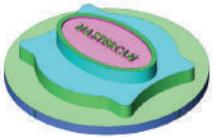
- ♦ Press **Alt + F1** to fit the geometry to the screen.
- ♦ The geometry should look as shown.

Y
C X



NOTE: To better display the geometry in the book, the background color in Mastercam is set to white and the title box and the geometry are in black color. On your screen you should see the default Mastercam background which is a grey gradient color and the geometry will be shown in white.

The dimensions, title block, and some of the geometry are not required for machining purposes. To be able to see only the geometry needed for toolpath creation, you will move it to a different level. This will be covered in the following steps of this tutorial.

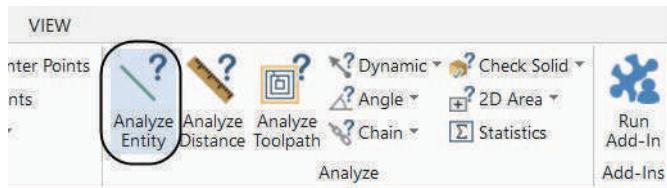


STEP 3: ANALYZE THE GEOMETRY

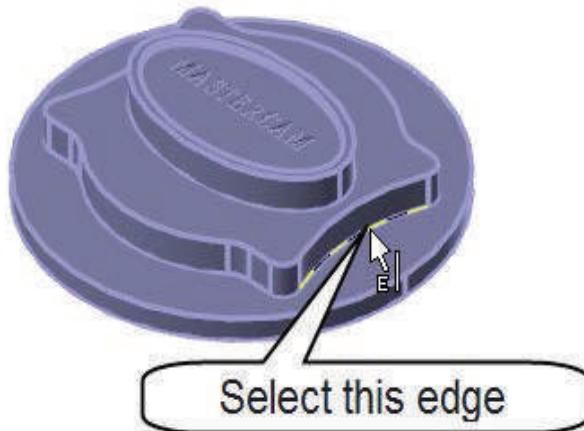
To machine this part, the geometry has to be created at a 1:1 scale. In order to check the geometry's accuracy, you will analyze one of the entities that is dimensioned and check if the value is the same as the one in the drawing. In this tutorial we will use the solid model to machine the part.

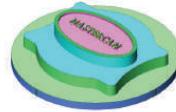
HOME

- From the **Analyze** group, select **Analyze Entity**.



- [Select entities to analyze]: Hover the mouse above the solid view and scroll up the mouse wheel to zoom in and select the arc as shown.



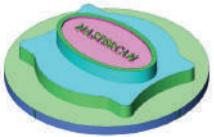


- ♦ In the **Arc Properties**, the **radius** is **5.0** as shown.



NOTE: The geometry was created at the proper scale 1:1. If the entity value was different than the dimension value, you can use the **TRANSFORM/Scale** command to increase or decrease the size of the entities by a factor relative to a defined point using the uniform method.

- ♦ Select the **OK** button to exit **Arc Edge Properties**.



STEP 4: MOVE THE TOOLPATH GEOMETRY TO A DIFFERENT LEVEL

Levels are a primary organizational tool in Mastercam. A Mastercam file can contain separate levels for wireframe, surfaces, drafting entities, and toolpaths. By organizing your files into levels, you can more easily control which areas of the drawing are visible at any time and which parts are selectable so that you do not inadvertently make changes to areas of the drawing you do not want to change.

In this step, you will select the geometry required for the toolpath, in this case the solid model, and move it to a different level.

4.1 Set the Main Level and name it

The **Main Level** is the current working level. Any geometry that you create is always placed on the main level. There can be only one main level at a time. The number of the main level appears on the **Level** button in the **Status bar**. In the **Levels Manager** dialog box, the main level is highlighted in yellow.

- From the bottom left corner of the screen, click on the **Levels** tab as shown.



TUTORIAL #3**MOVE THE TOOLPATH GEOMETRY TO A DIFFERENT LEVEL**

- ◆ Change the **Number** in the **Main Level** to **100** and enter the name **Solid** as shown.

Levels

Num...	Visible	Name	Entities	Level Set
1	X	0	0	
2	X	Geometry	1861	
3	X	Dimension	262	
4	X	Border	0	
5	X	ISO Solid	0	
6	X	Solid Wirefr...	0	
7	X	ORIGIN	0	
8	X	TITLE_BLOCK	88	
9		CEN	12	
100	X	Solid	0	

Number: 100

Name:

Level set:

Display:

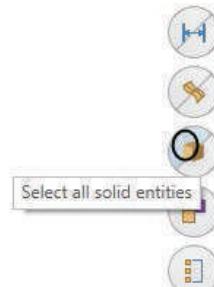
Used
 Named
 Used or named
 Range

Toolpaths Solids Planes Levels Art

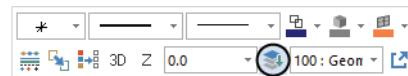


4.2 Move the geometry to the Main Level 100

- From the Quick Mask buttons, choose **Select all solid entities** as shown.



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



- The **Change Levels** dialog box should appear on the screen as shown.



NOTE: The parameters in the **Change Levels** dialog box are already set to move the geometry to the **Main Level**.

- Leave the default settings in the **Change Levels** dialog box and select the **OK** button to continue.





4.3 Make the Main Level the only visible level

- In the **Levels Manager**, right mouse click on the number **100** and select **All level set off** to make all the levels invisible except the **Main Level**.

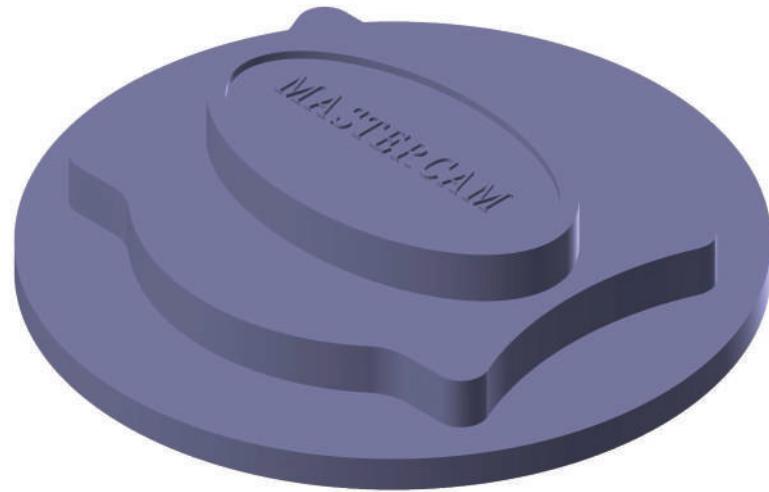
Nu...	Visible	Name	Level Set	Entities
1	X	0		0
2	X	Geometry		1861
3	X	Dimension		247
4	X	Border		0
5	X	ISO Solid		0
6	X	Solid Wireframe		0
7	X	ORIGIN		0
8	X	TITLE_BLOCK		88
9	X	CEN		12
100	X	Solid		10

Number: Name:

Right-click context menu options:

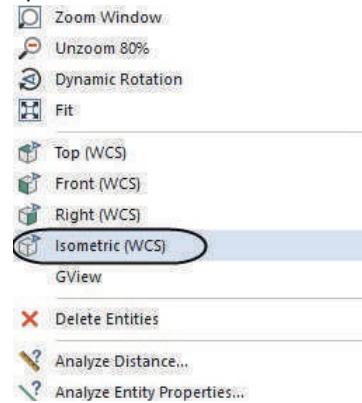
- Make active
- Renumber level
- Select all entities
- Unselect all entities
- Toggle visibility
- Cut entities
- Copy entities
- Paste entities
- Delete entities
- Refresh levels list
- Purge empty levels >
- All level set on
- All level set off** (highlighted)
- Report >
- Import named levels
- Export named levels

- Press **Alt + F1** to fit the geometry in the graphics window.
- The geometry should look as shown. You may need to shade the solid by pressing **Alt + S**.

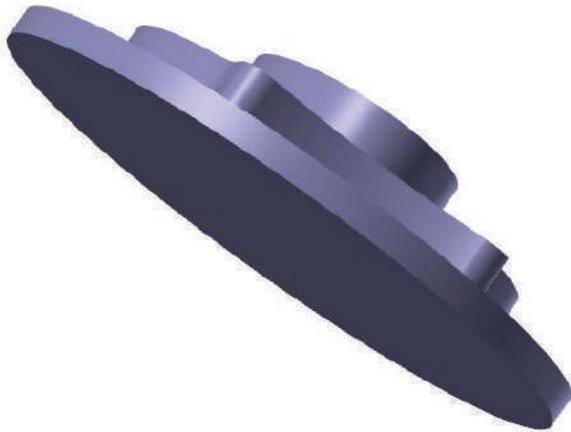




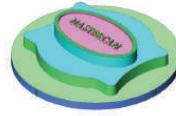
- ♦ Right mouse click in the graphics window and select the **Isometric** view to see the part.



- ♦ The geometry should look as shown.



NOTE: The solid orientation needs to be changed to match how the part will be setup at the machine. You will be using **Planes** options and **TRANSFORM/Dynamic** to align the part as it should be at the machine.

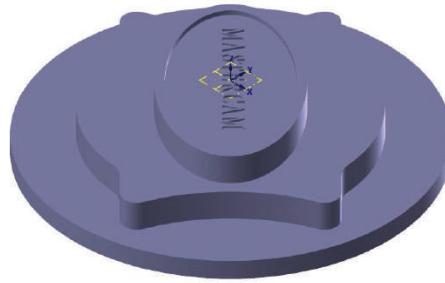


STEP 5: USE WORK COORDINATE SYSTEM TO ORIENT THE PART

Mastercam uses a 3D Cartesian coordinate system to locate your work in three-dimensional space. This means that geometry and toolpath positions are expressed in terms of three coordinate axes: X, Y, and Z. Each axis is signed, which means that it has a positive and a negative direction. You can use the WCS to align the coordinate axes and origin with the part before creating the toolpath.

Mastercam lets you create a work coordinate system (WCS), which lets you move the coordinate system to your part geometry instead of moving or transforming the part geometry. Think of the WCS view as defining the "Top plane" relative to your part.

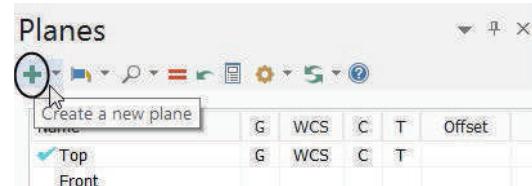
Step Preview:



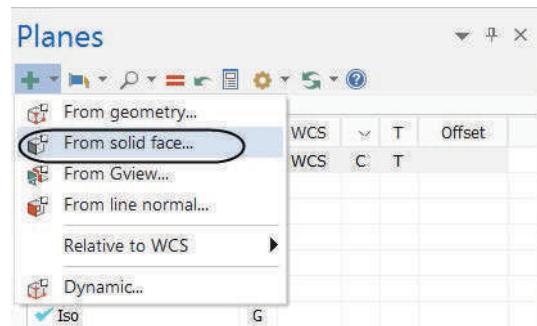
- ♦ Select the **Planes** tab located at the bottom left corner.

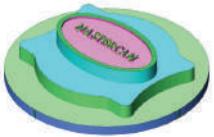


- ♦ To create a new plane based on existing geometry, click on the + sign as shown.

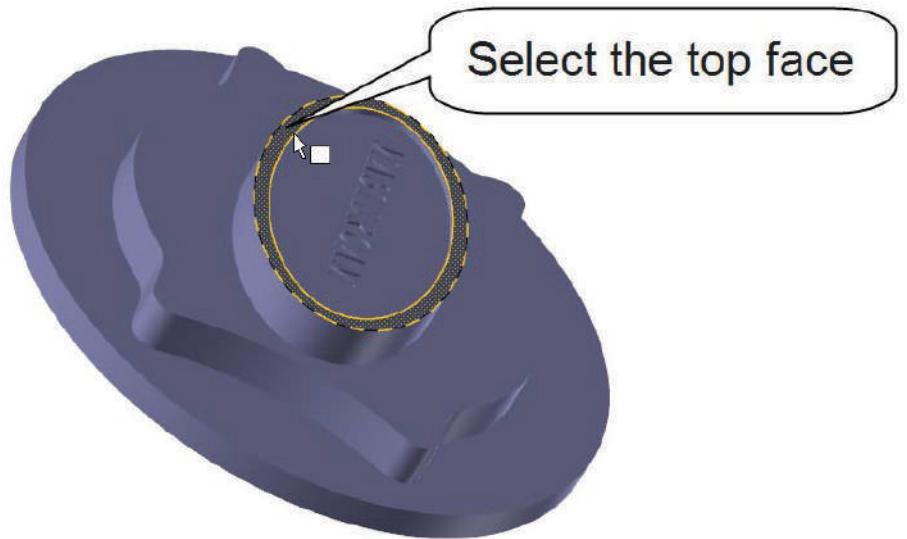


- ♦ Select **From solid face** as shown.

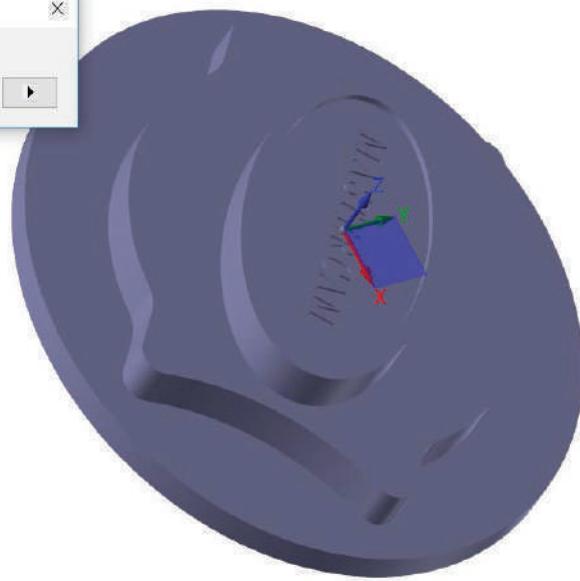
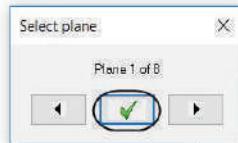




- ♦ Holding down the mouse wheel, click in the middle of the solid and rotate the part to see it as shown.
- ♦ Select the top face.



- ♦ The plane axis should look as shown.



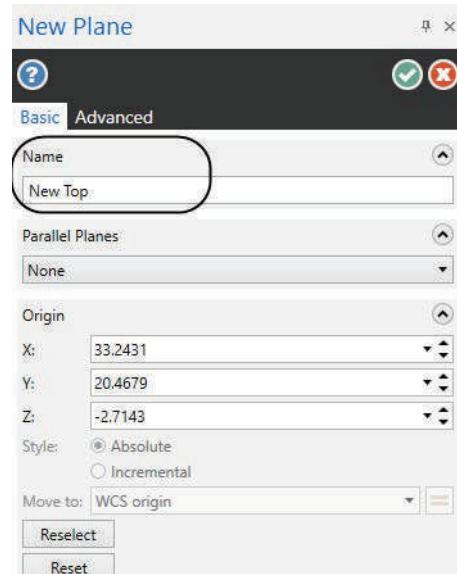
- ♦ Select the **OK** button to continue. 

TUTORIAL #3

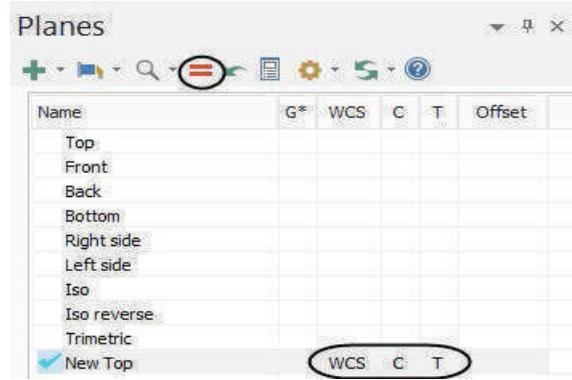
USE WORK COORDINATE SYSTEM TO ORIENT THE PART



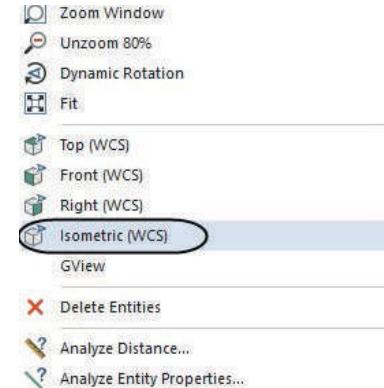
- ♦ New Plane panel will appear as shown. Enter **New Top** in the Name field as shown.



- ♦ Select the **OK** to create the new plane.
- ♦ To set the **WCS**, the **Tool plane** and the **Construction plane** to the new plane, click on the **equal** icon.

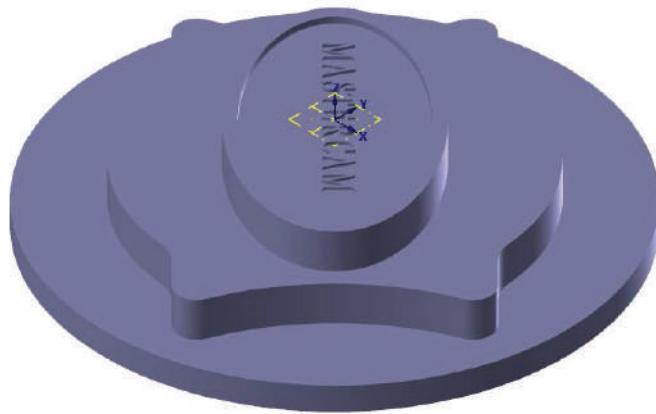


- ♦ To update the view, right mouse click in the graphics window and select the **Isometric view**.





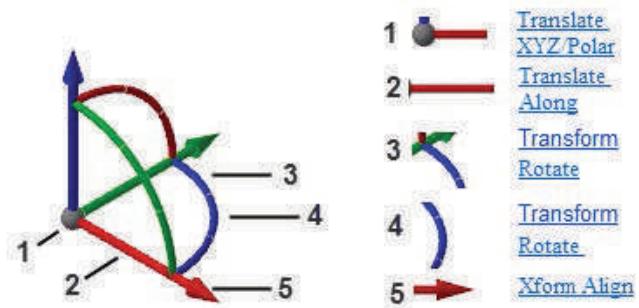
- The geometry should look as shown.



STEP 6: ALIGN THE AXIS USING TRANSFORM DYNAMIC

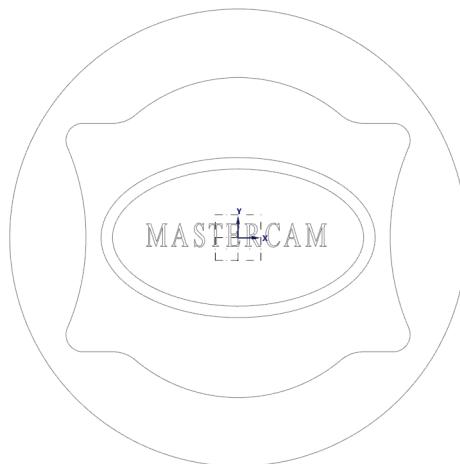
Use **TRANSFORM/Dynamic** to manipulate geometry orientation and location through the use of an interactive gnomon in the graphics window. The gnomon is made up of three axes connected at the origin. There are five selection points along each axis. Each segment of the axis line is used to produce a different type of transform motion as shown in [Figure: 6.0.1](#).

Figure: 6.0.1





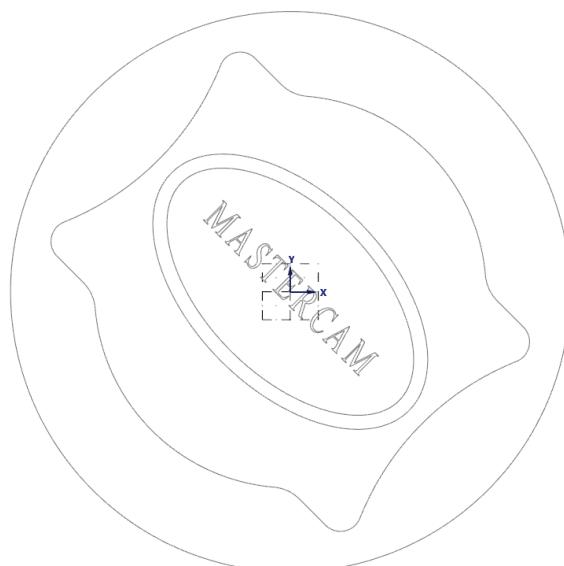
Step Preview:



- ♦ Right mouse click again and select the **Top** view from the list to see the part from the top.



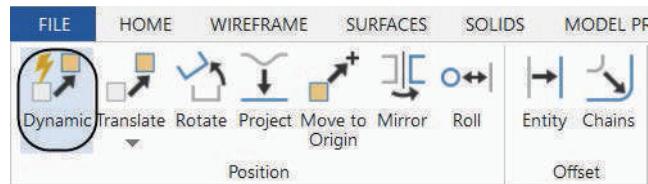
- ♦ Press **Alt + S** to unshade the solid.
- ♦ The geometry should look as shown.



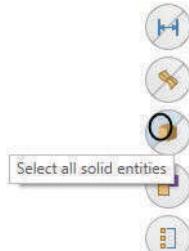


TRANSFORM

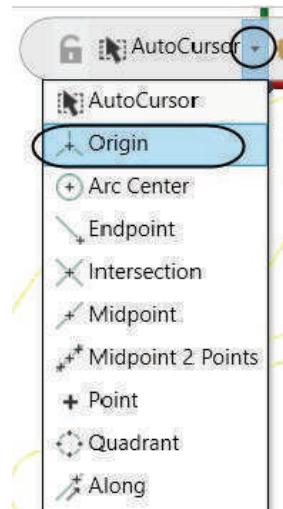
- From the **Position** group, select **Dynamic** as shown.



- [Select entities to move/copy]: Click again on **Select all solid entities** as shown.

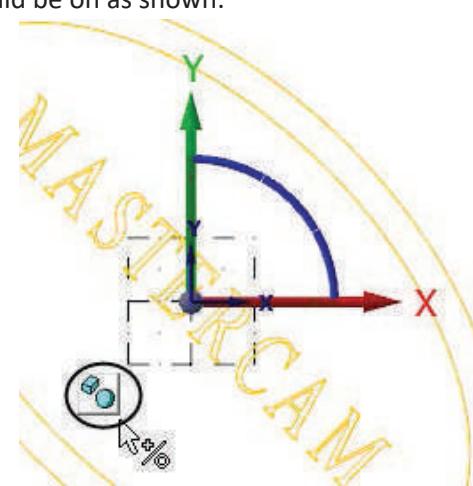


- Click on the **End Selection** button.
- [Pick gnomon origin position]: From the **General Selection toolbar**, click on the drop down arrow beside **AutoCursor** and select **Origin** as shown.

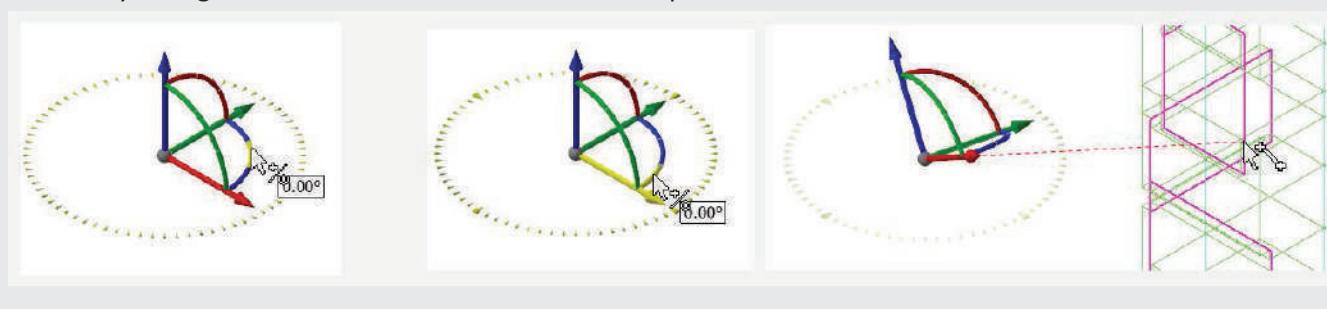




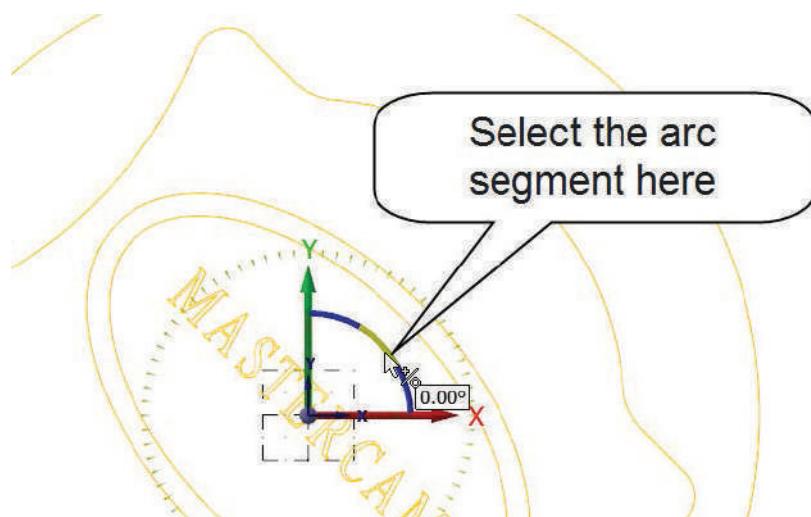
- To be able to rotate the part, the **geometry manipulation** button should be on as shown.

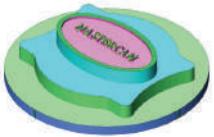


NOTE: The **Transform Rotate** mode can be entered in two ways. You can select the center of the arc between two axes, as shown below on the left, to produce a rotation about a perpendicular axis. You can also select the end of an arc with an axis, as shown below on the right, to rotate about the perpendicular axis while retaining the ability to align the selected axis with an autocursor point.

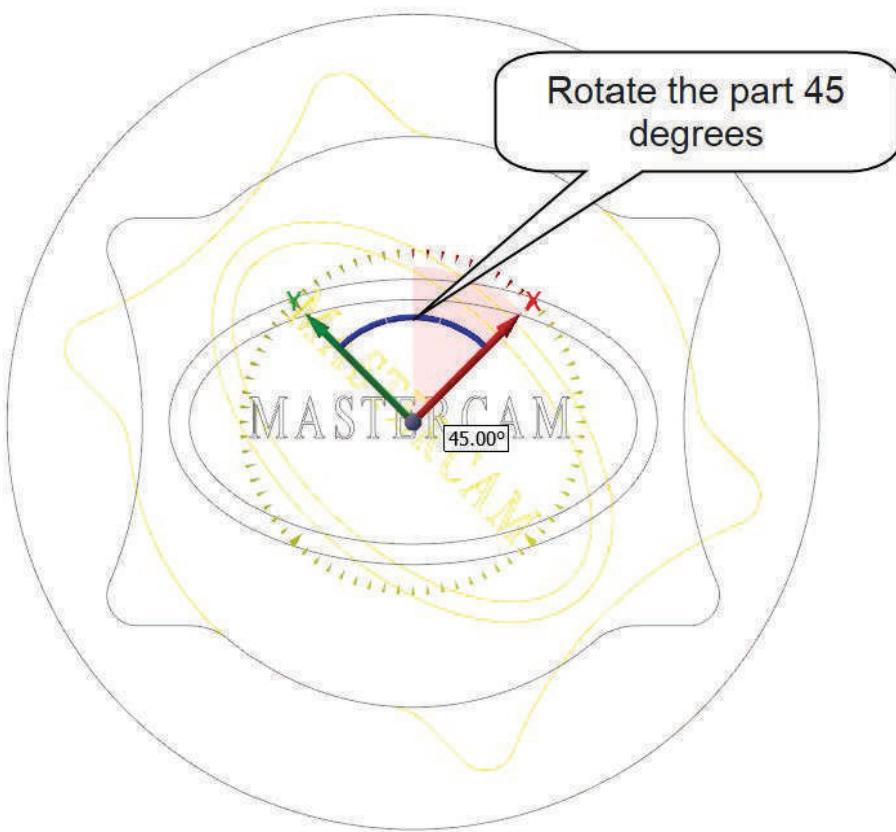


- Select the arc segment as shown.



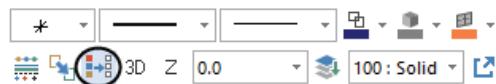


- ♦ Rotate the part counter clockwise as shown until 45 degrees are reached as shown.



- ♦ Left click the mouse to accept it.

- ♦ In the **Dynamic** panel, select the **OK** button to exit the command.
- ♦ Select the **Clear Colors** icon as shown to remove the resulting color.

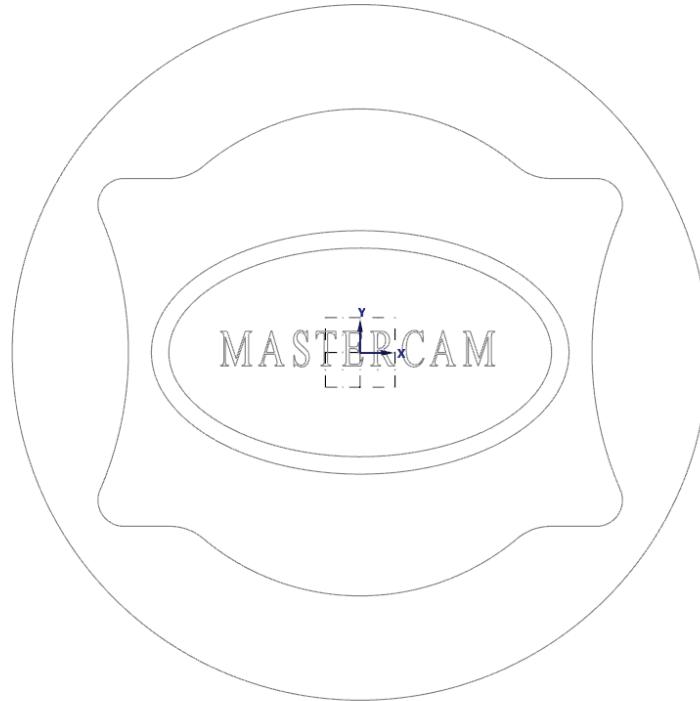


TUTORIAL #3

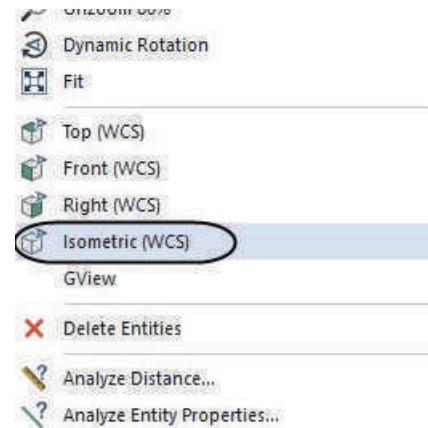
ALIGN THE AXIS USING TRANSFORM DYNAMIC

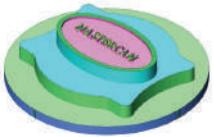


- The geometry should look as shown.



- To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.

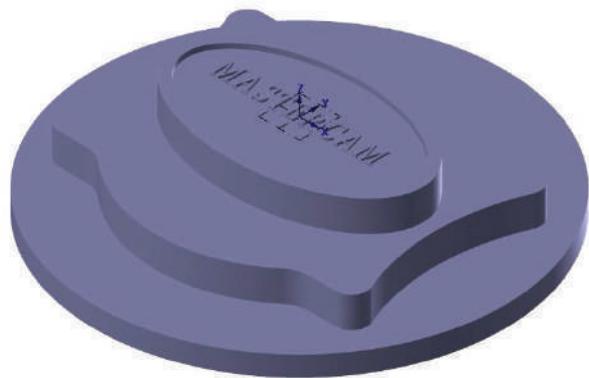




SAVE THE FILE

TUTORIAL #3

- ♦ Press **Alt + S** to see the solids in shaded mode as shown.



STEP 7: SAVE THE FILE

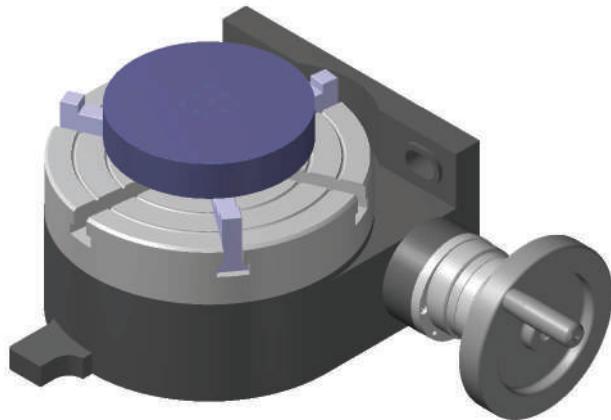
FILE

- ♦ Save As.
- ♦ File name: "Your Name_3".



TOOLPATH CREATION

SUGGESTED FIXTURE:



SETUP SHEET:

TOOL LIST

	TYPE: Endmill1 Flat DIA OFFSET: 1 HOLDER: DEFAULT HOLDER NUMBER: 1 LENGTH OFFSET: 1	FLUTE LENGTH: 2.00 OVERALL LENGTH: 3.75 CORNER RAD: 0.0 # OF FLUTES: 4
A technical drawing of a flat endmill. It has a diameter of 2.000, a height of 1.000, and a total length of 3.750. The diagram shows the tool's profile with dimensions: top width 2.000, bottom width 1.000, and overall height 3.750.	#1 - 0.7500 ENDMILL1 FLAT - 3/4 FLAT ENDMILL	
A technical drawing of a flat endmill. It has a diameter of 2.000, a height of 1.000, and a total length of 2.250. The diagram shows the tool's profile with dimensions: top width 2.000, bottom width 1.000, and overall height 2.250.	TYPE: Endmill1 Flat DIA OFFSET: 2 HOLDER: DEFAULT HOLDER NUMBER: 2 LENGTH OFFSET: 2	FLUTE LENGTH: 0.375 OVERALL LENGTH: 2.25 CORNER RAD: 0.0 # OF FLUTES: 4
A technical drawing of a flat endmill. It has a diameter of 2.000, a height of 1.000, and a total length of 3.250. The diagram shows the tool's profile with dimensions: top width 2.000, bottom width 1.000, and overall height 3.250.	TYPE: Endmill1 Flat DIA OFFSET: 3 HOLDER: DEFAULT HOLDER NUMBER: 3 LENGTH OFFSET: 3	FLUTE LENGTH: 0.375 OVERALL LENGTH: 2.25 CORNER RAD: 0.0 # OF FLUTES: 4
	#3 - 0.0313 ENDMILL1 FLAT - 1/32 FLAT ENDMILL	



STEP 8: SELECT THE MACHINE AND SET UP THE STOCK

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

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

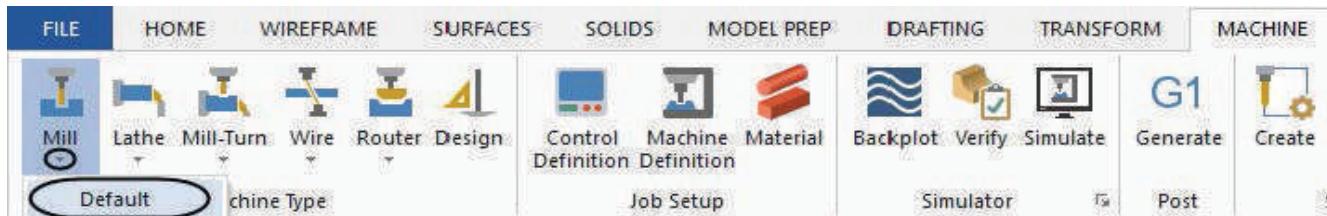
- From the bottom left side of the graphics window, click on the **Toolpaths** tab as shown.



NOTE: Select the **Mill Default** only if there is no **Machine Group** in the **Toolpaths Manager**.

MACHINE

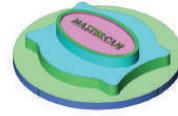
- From the **Machine Type** group, click on the drop down arrow below **Mill** and select the **Default**.



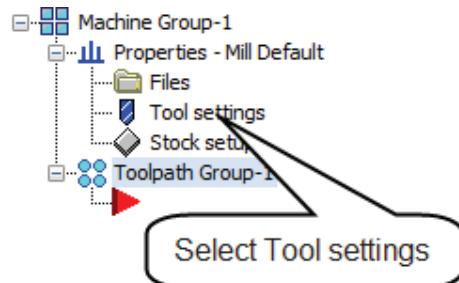
NOTE: Once you select the **Mill Default**, the ribbon bar changes to reflect the toolpaths that could be used with **Mill Default**.

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



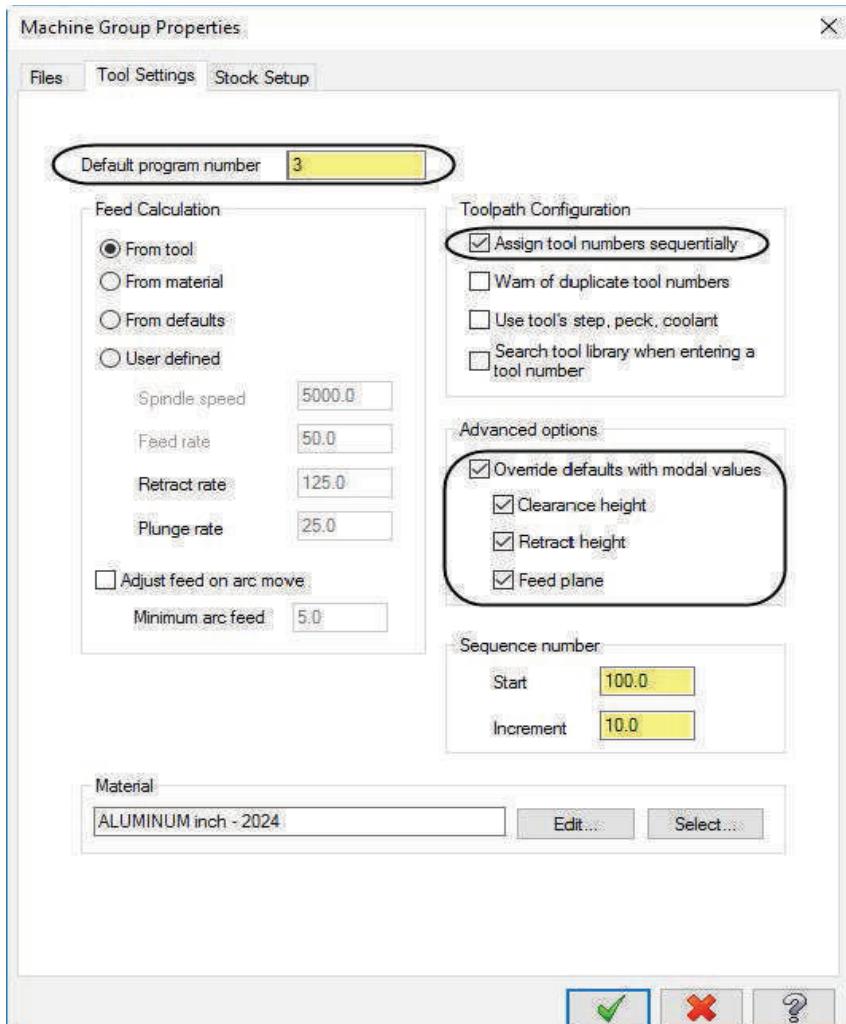


- ♦ Select **Tool settings** to set the tool parameters.



- ♦ Change the parameters to match the screenshot as shown in [Figure: 8.0.1](#).

Figure: 8.0.1



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

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

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

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

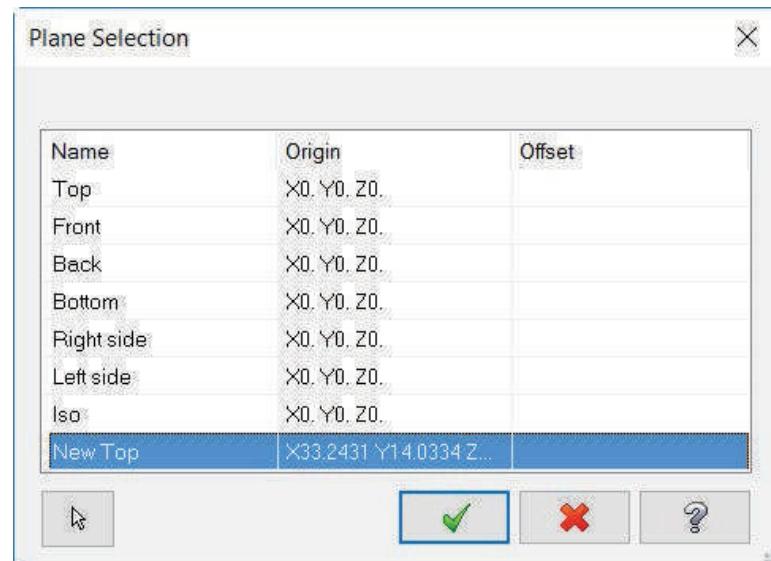
Feed Calculation set to **From tool** uses feed rate, plunge rate, retract rate and spindle speed from the tool definition.



- ♦ Select the **Stock Setup** tab to define the stock.
- ♦ Make sure that the **Stock Plane** is set as **New Top**. If not, click in the **Stock Plane** icon as shown.

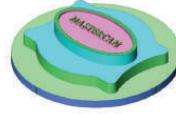


- ♦ In the **Plane Selection** dialog box, select **New Top** as shown.



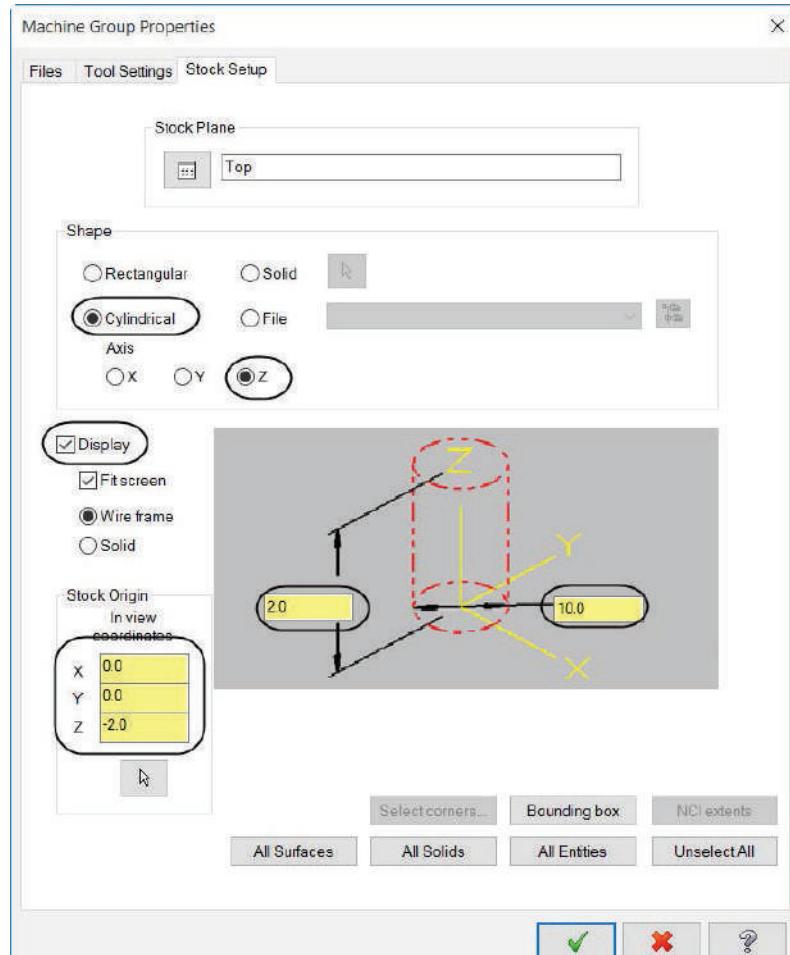
- ♦ Select the **OK** button to accept the **Plane Selection**.





- ♦ Select the **Cylindrical Shape** and enter the values as shown in [Figure: 8.0.2](#).

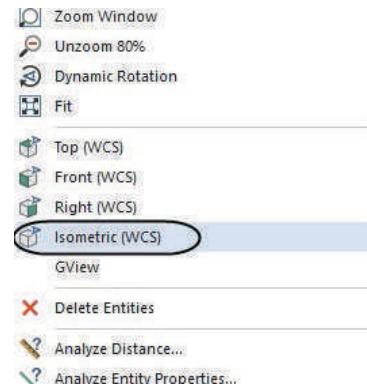
Figure: 8.0.2



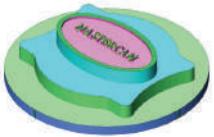
The **Stock Origin** values adjust the positioning of the stock, ensuring that you have an equal amount of extra stock around the finished part.

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

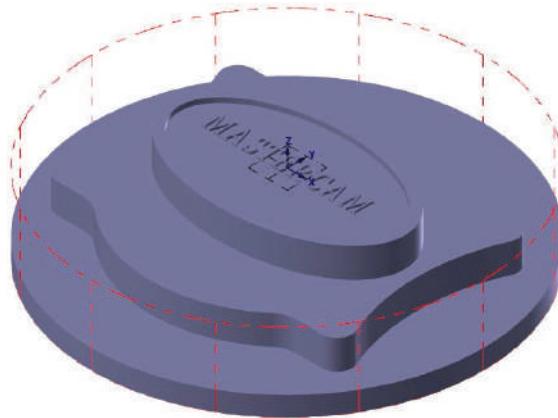
- ♦ Select the **OK** button to exit the **Machine Group Properties**.
- ♦ Right mouse click in the graphics window and select the **Isometric** view to see the stock.



- ♦ Press **Alt + F1** to fit the part to the screen.



- The stock model will appear as shown.



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

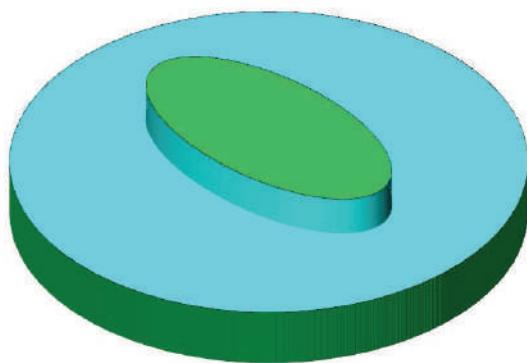
NOTE: There will not be a facing toolpath because the stock is already to the size.

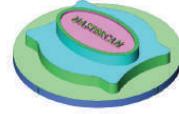
STEP 9: 2D HIGH SPEED AREA MILL

2D High Speed Area Mill generates the free-flowing motion needed to machine features, such as standing bosses and cores or pockets in a single operation. With **Area Mill High Speed**, smaller depth of cuts are recommended versus **Dynamic Mill** in which the depth cuts can be the size of the flute.

The toolpath depends on the **Machining region strategy** that you choose in the **Chain Options**. If the strategy chosen is **From outside**, the toolpath starts at the outmost chain and works its way in taking on the final shape of the part as it approaches the final pass. You can also machine pockets in which case the strategy selected is **Stay inside** which keeps the tool inside the machining regions.

Toolpath Preview:



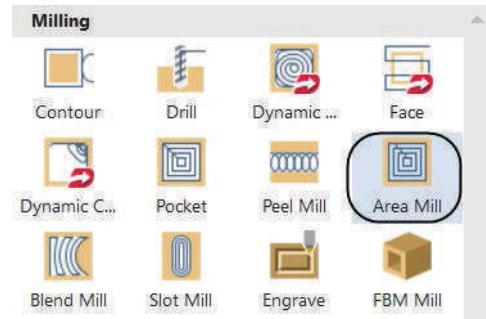


TOOLPATHS

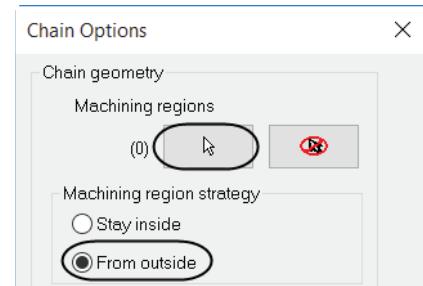
- From the 2D group, select the **Expand gallery arrow**.

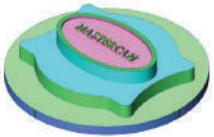


- Select the **Area Mill** icon as shown.



- In the **Chain Options**, enable **From outside** and click on the **Select machining chains** button in the **Machining regions** as shown.





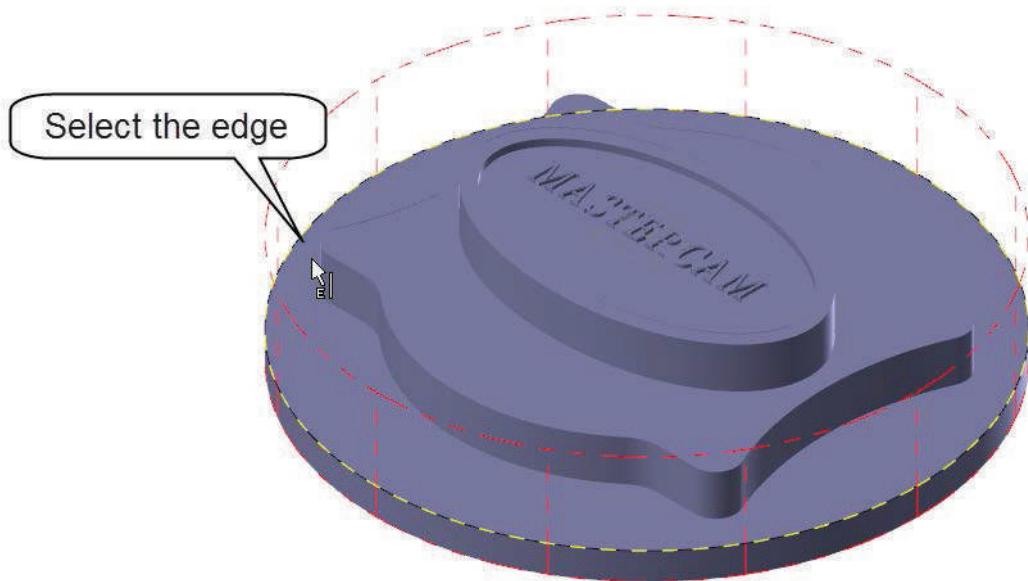
- ♦ Enable **Solids** in the **Chaining** dialog box.
- ♦ Enable only the **Loop** button as shown.



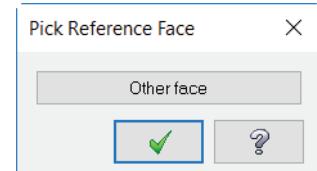


- ♦ [Solid Toolpath Chain: Select faces, edges and/or loops]: Select the edge as shown in [Figure: 9.0.1](#).

Figure: 9.0.1



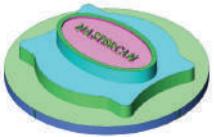
- ♦ Select the **OK** button to exit the **Pick Reference Face** dialog box.



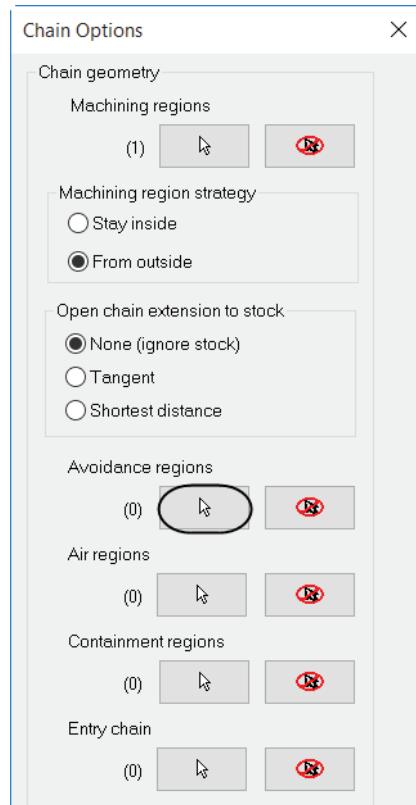
NOTE: **Pick Reference Face** dialog box allows you to cycle through possible reference faces on solids until you reach the one that you want to select.

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

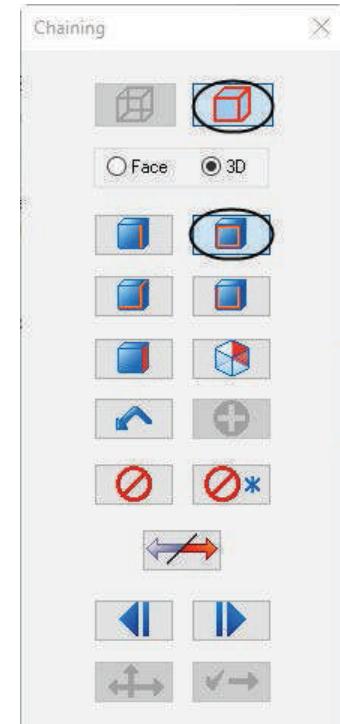




- From the **Chain Options**, click on the **Select avoidance chains** button in the **Avoidance regions** as shown.



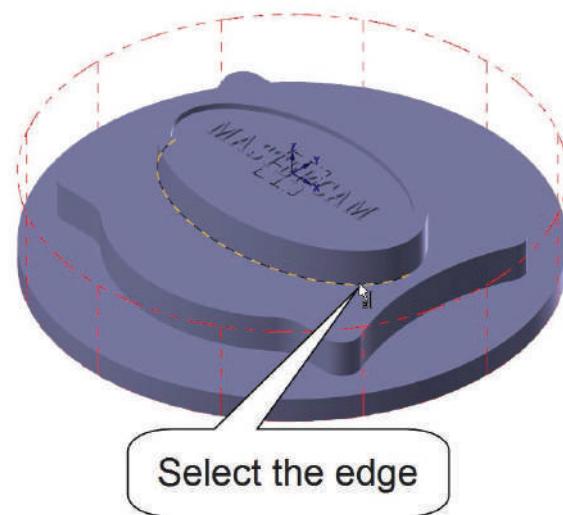
- Enable **Solids** in the **Chaining** dialog box.
- Enable only the **Loop** button as shown.





- ♦ [Solid Toolpath Chain: Select faces, edges and/or loops]: Select the edge as shown in [Figure: 9.0.2](#).

Figure: 9.0.2



- ♦ Select the **OK** button to exit the **Pick Reference Face** dialog box.
- ♦ Select the **OK** button to exit the **Chaining** dialog box.
- ♦ Select the **OK** button to exit the **Chain Options** dialog box.
- ♦ In the **Toolpath Type** page, **Area Mill** will be selected as shown.



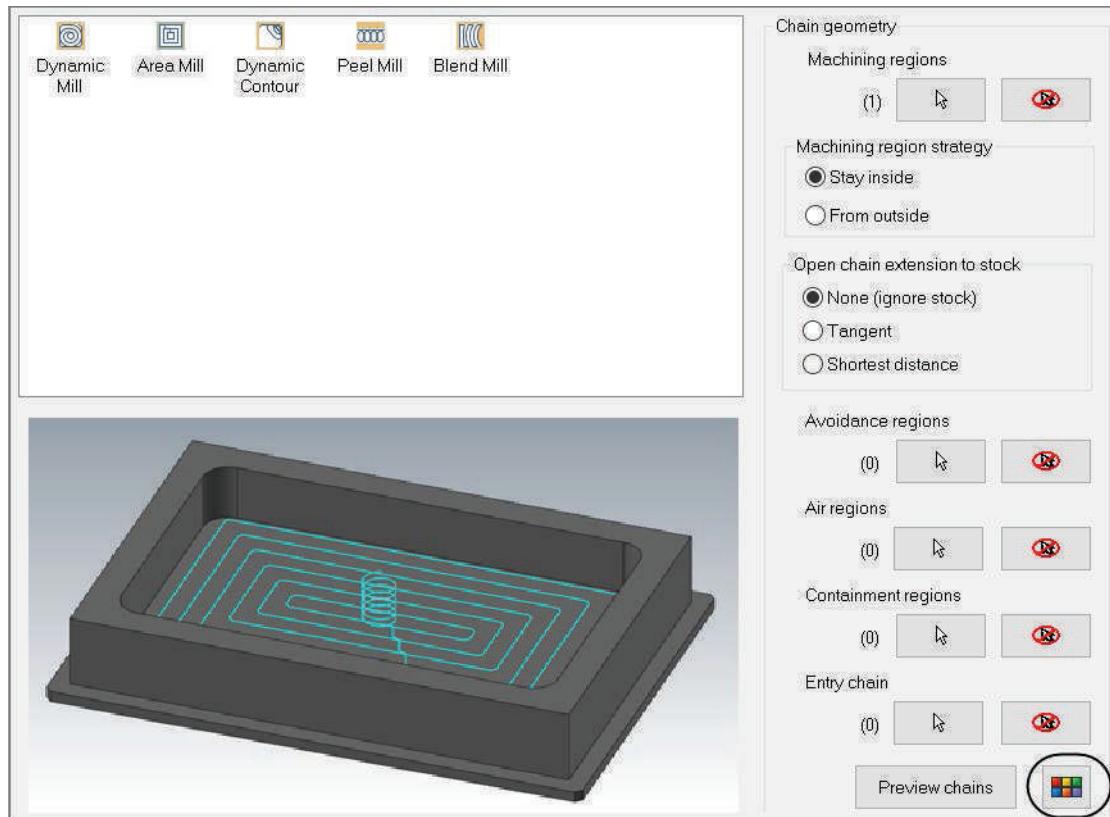
NOTE: Mastercam updates the pages as you modify them and then marks them, in the **Tree View list**, with a green check mark. Pages that are not enabled are marked with a red circle and slash.

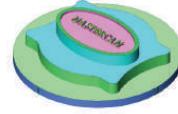


9.1 Preview Chains

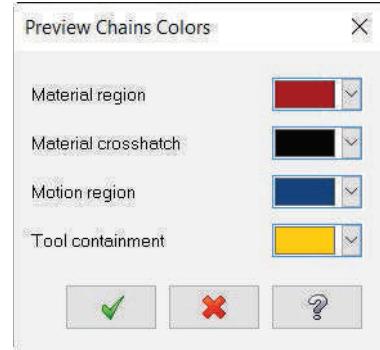
The **Preview Chains** function is intended to give the user a quick visual representation of how Mastercam sees the various pieces of geometry that have been selected, how they interact with one another, and a general overview of how the toolpath will be calculated with the selections presently made.

- ♦ Click on the **Color** icon to see the legend for **Preview chains** as shown.





- The Preview Chains Colors dialog box should look as shown.

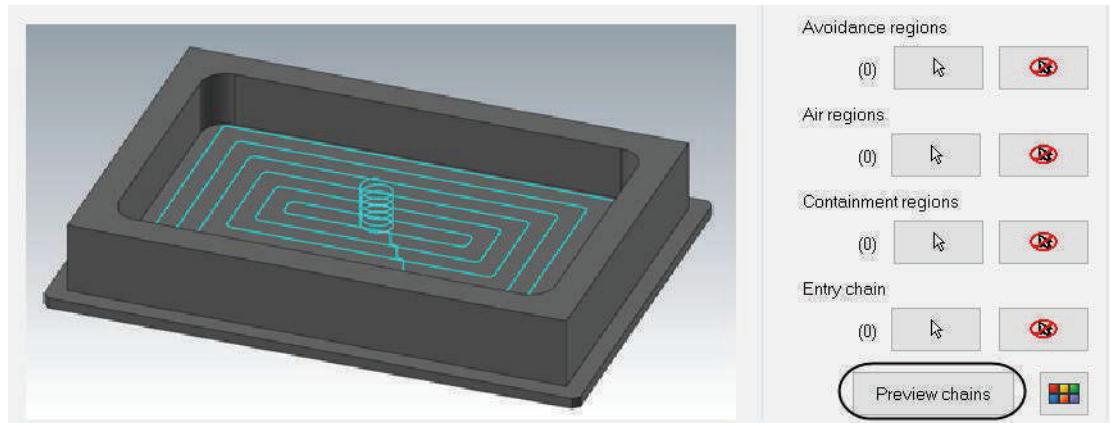


The **Material region** and **Material crosshatch** are the two colors that are used to define the material to be cut. The default colors are red for the background and black for the crosshatch.

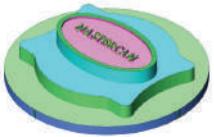
The **Motion region** displays the area that Mastercam is making available to the toolpath for motion if it needs it. The color to represent it is dark blue. The primary reason for the display of the entire available (but not necessarily used) motion region is to help the user visualize how the tool may move near or interact with any adjacent geometry.

The **Tool containment** is what you have selected as the containment region in the chain geometry. If you have not selected a containment region, it will default to the outside of the motion region since that is currently the default area the toolpath is being contained to. The color used to represent the tool containment is yellow.

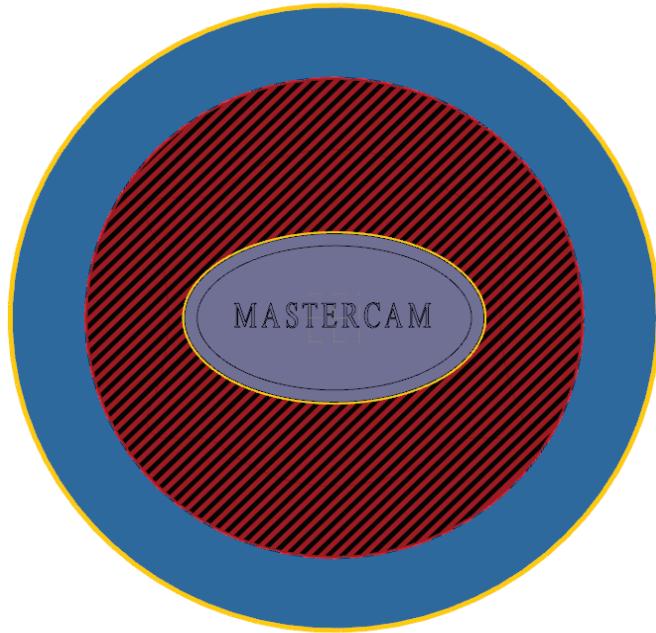
- Select the **OK** button to exit **Preview Chains Colors**.
- Select the **Preview chains** button as shown.



- Select the **Hide dialog** button to see the preview in the graphics window.



- The Preview chains should look as shown.



- Press Esc key to return to the toolpath parameters.
- Click on the Preview chains button again to clear the Preview chains display.



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

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



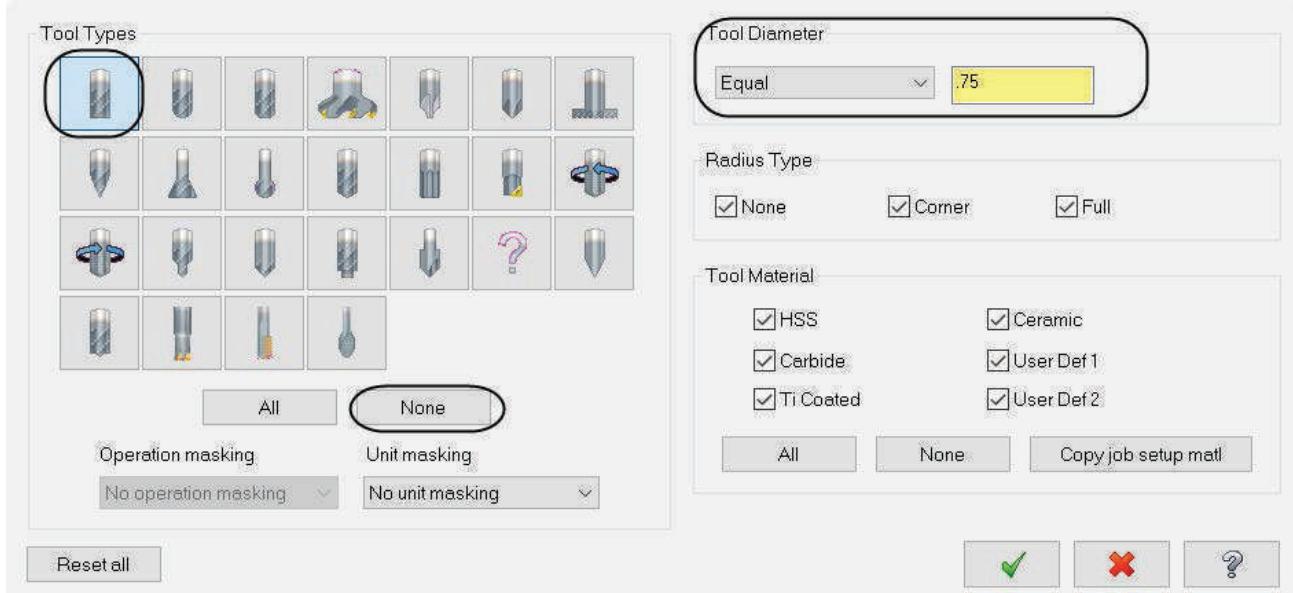
- Select the Filter button.





- ♦ Select the **None** button and then under **Tool Types**, choose the **Flat Endmill** icon.
- ♦ Under **Tool Diameter**, pick **Equal** and input a value of **0.75** as shown in [Figure: 9.2.1](#).

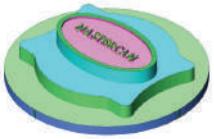
Figure: 9.2.1



- ♦ Select the **OK** button to exit the **Tool List Filter**.
- ♦ In the **Tool Selection** dialog box, you should only see a **3/4" Flat Endmill**.

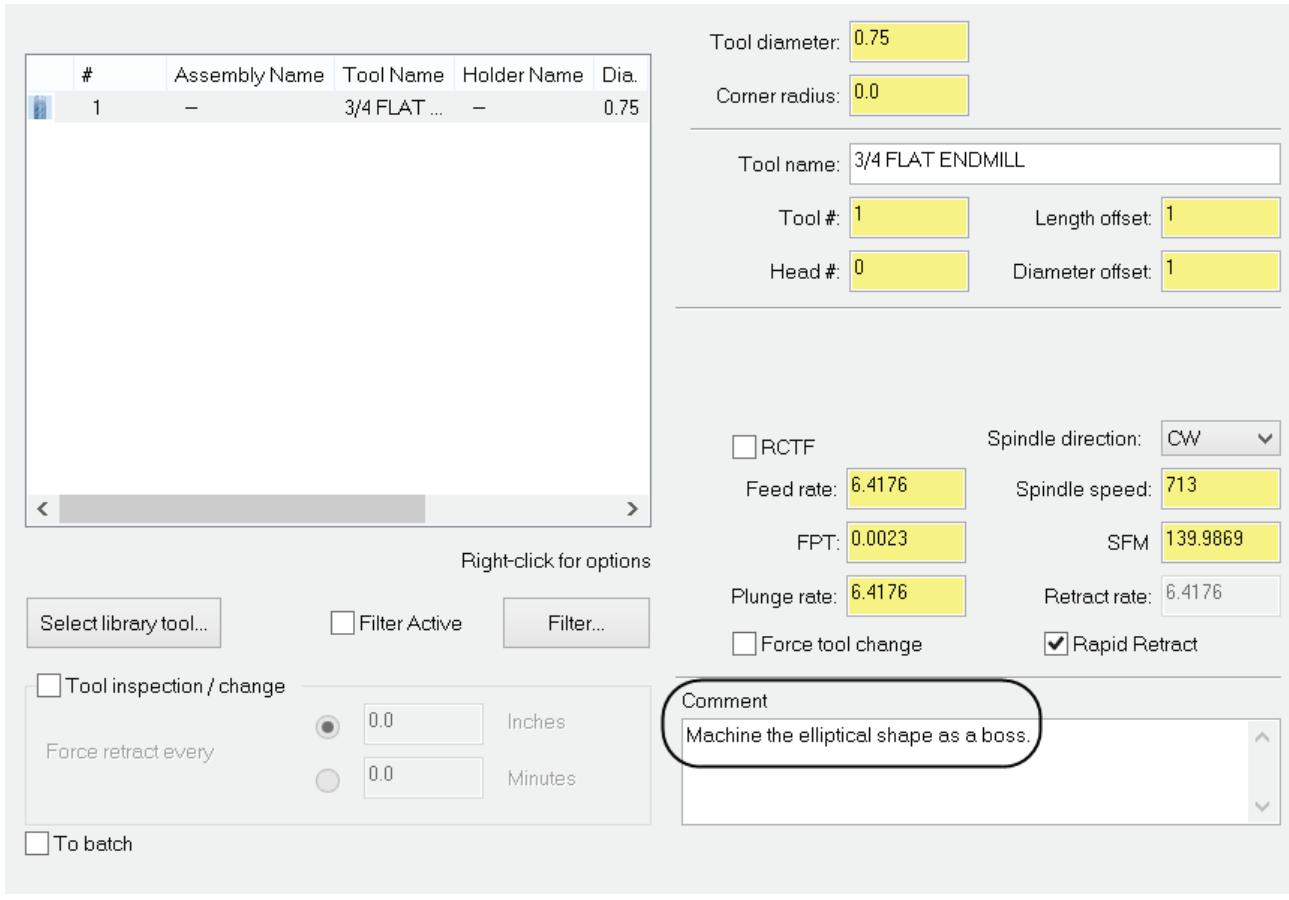
#	Assembly...	Tool Name	Holder Name	Dia.	Cor. r...	Length	# Flut...	Type	Rad...
294	-	3/4 FLAT ENDMILL	-	0.75	0.0	2.0	4	End...	None

- ♦ Select the **3/4" Flat Endmill** in the **Tool Selection** page and then select the **OK** button to exit.

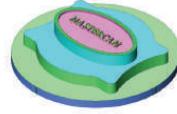


- ♦ Make all the necessary changes as shown in [Figure: 9.2.2.](#)

Figure: 9.2.2



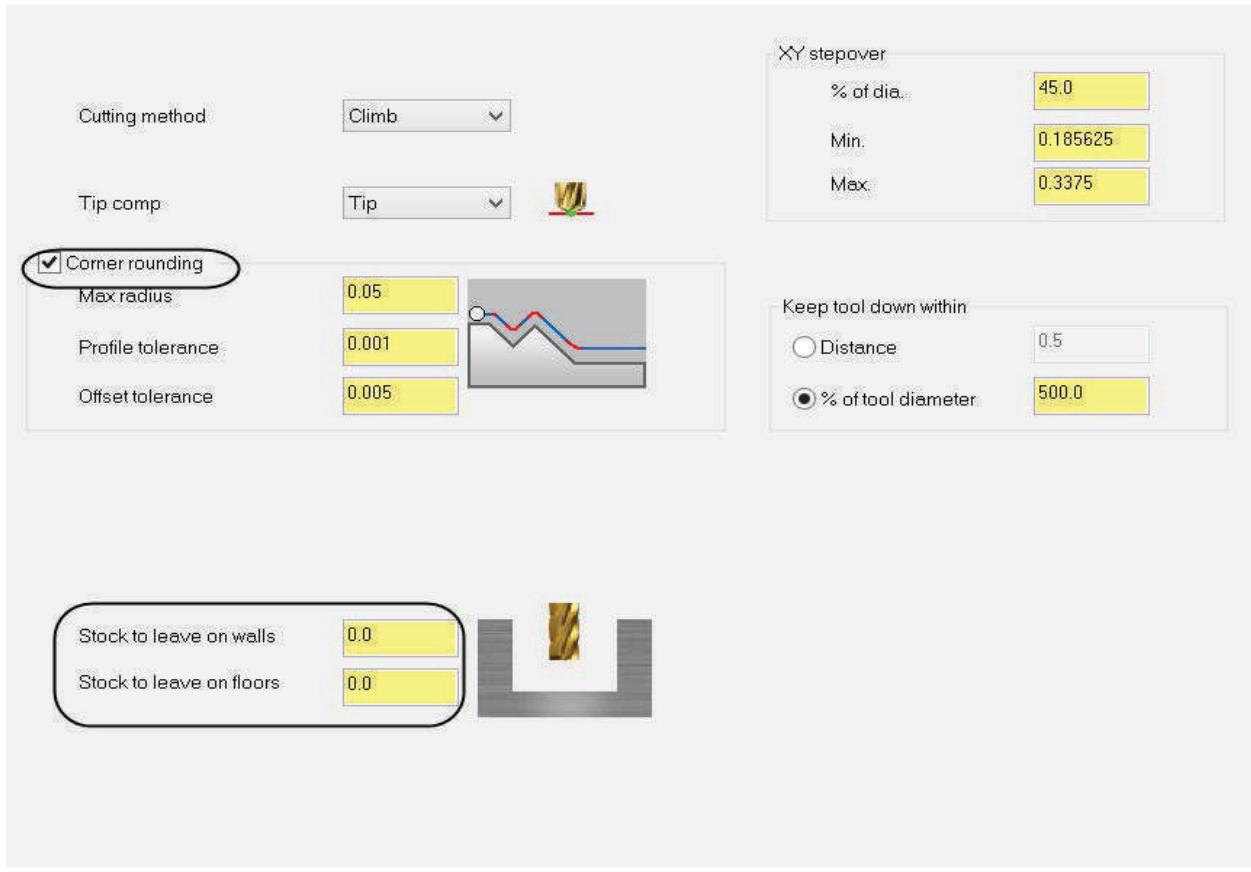
Tool Inspection/change forces a retract move at set intervals so that your machine operator can inspect the tool. When the tool reaches an inspection point, it retracts and rapids off the part to the clearance plane.



9.3 Set the Cut Parameters

- From the Tree View list, select **Cut Parameters** and enable **Corner rounding** as shown in [Figure: 9.3.1](#).

Figure: 9.3.1



Corner rounding replaces sharp corners with arcs for faster and smoother transitions in tool direction.

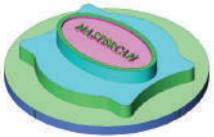
Max radius is the largest arc that you allow Mastercam to insert to replace a corner. Larger arcs will create a smoother toolpath but with greater deviation from the originally programmed toolpath.

Profile tolerance represents the maximum distance that the outermost profile of a toolpath created with a corner rounding can deviate from the original toolpath.

Offset tolerance represents the maximum distance that a profile of a toolpath created with corner rounding can deviate from the original toolpath. This is the same measurement as the profile tolerance but is applied to all the profiles except the outermost one.

XY Stepover expresses the maximum XY stepover as a percentage of the tool diameter. Mastercam will use the largest value possible that does not leave unwanted upstands of material between the passes.

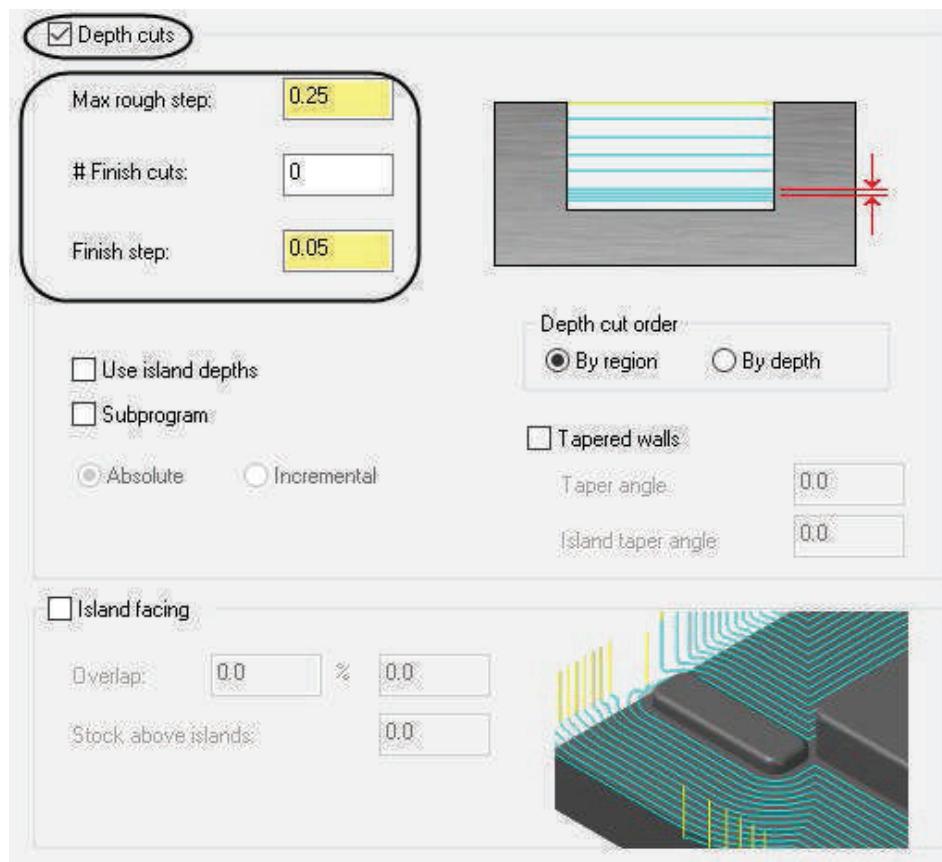
Keep tool down within keeps the tool down if the distance from the end pass to the start of the next pass is less than the value here. Mastercam will not create a retract move as defined on the **Linking Parameters** page. Instead the tool will stay down and move directly between the passes at the feed rate.



9.4 Set the Depth Cuts Parameters

- From the Tree View list, select **Depth Cuts** and make the necessary changes as shown in [Figure: 9.4.1.](#)

Figure: 9.4.1





9.5 Set the Transitions Parameters

- From the Tree View list, select **Transitions**, choose **Entry helix** as the **Entry method** and enter a **Radius** of **0.25**.
- Ensure the value set in **Skip pockets smaller than** is **0.55** as shown in [Figure: 9.5.1](#).

Figure: 9.5.1



Entry method sets the entry move that the tool makes as it transitions to new Z depths. If you choose to create a helical entry and there is not enough room, Mastercam creates a ramp entry instead.

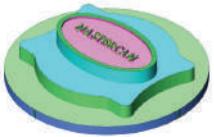
Entry feed rate sets the rate that the tool feeds into the material.

Z clearance is extra height used in the ramping motion down from a top profile. It ensures the tool has fully slowed down from the rapid speeds before touching the material.

Plunge angle sets the angle of decent for the entry move and determines the pitch. A smaller plunge angle means that the entry move takes longer to descend in the Z axis.

Preferred profile length enters a minimum size for the profile in order for a ramp to be created.

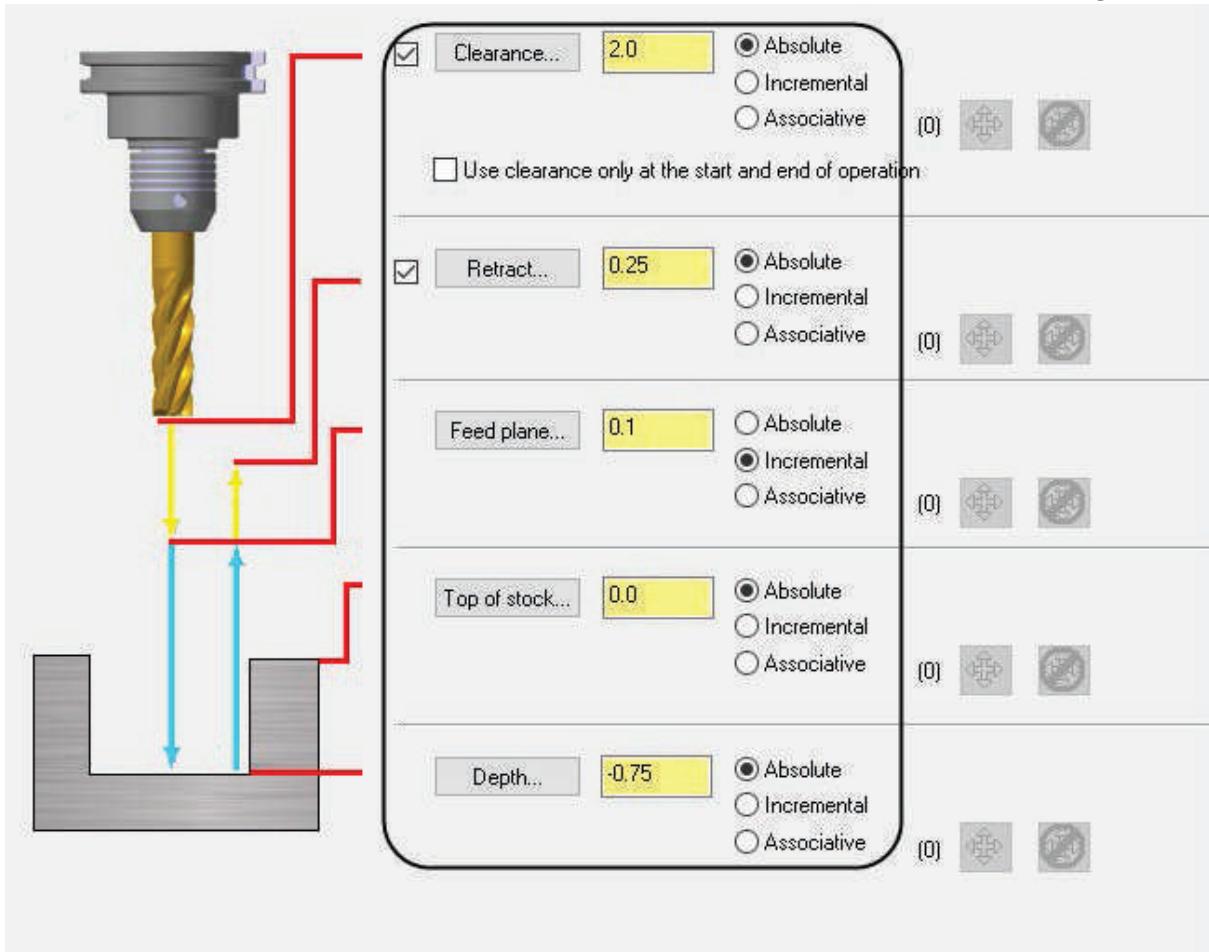
Skip pockets smaller than value is a minimum pocket size that Mastercam will consider creating a cutting pass for. Ensure the value set here is at least 100% of the tools diameter.



9.6 Set the Linking Parameters

- ♦ Select **Linking Parameters** and make any necessary changes as shown in [Figure: 9.6.1](#).

Figure: 9.6.1

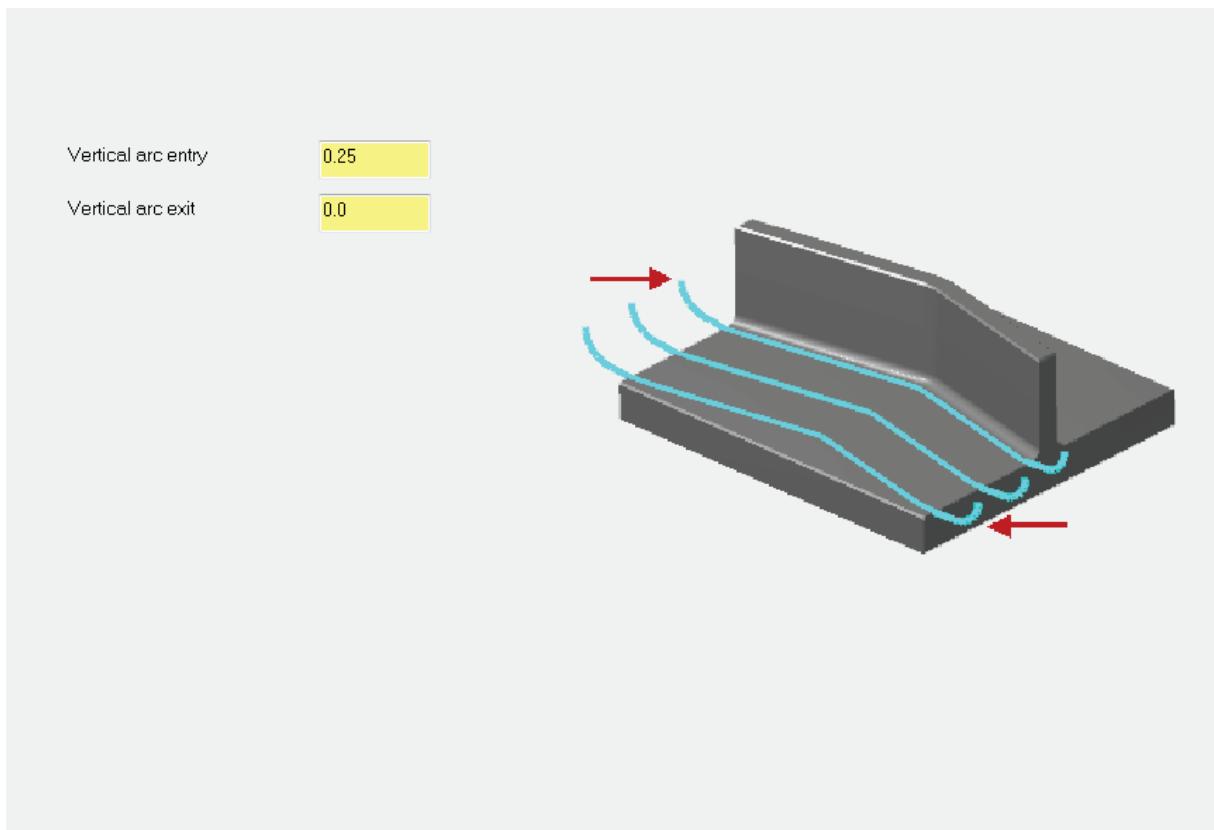




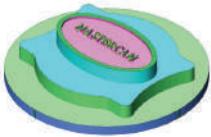
9.7 Set the HST Leads

- From the **Tree View** list, select **HST Leads** and make any necessary changes as shown in [Figure: 9.7.1](#).

Figure: 9.7.1



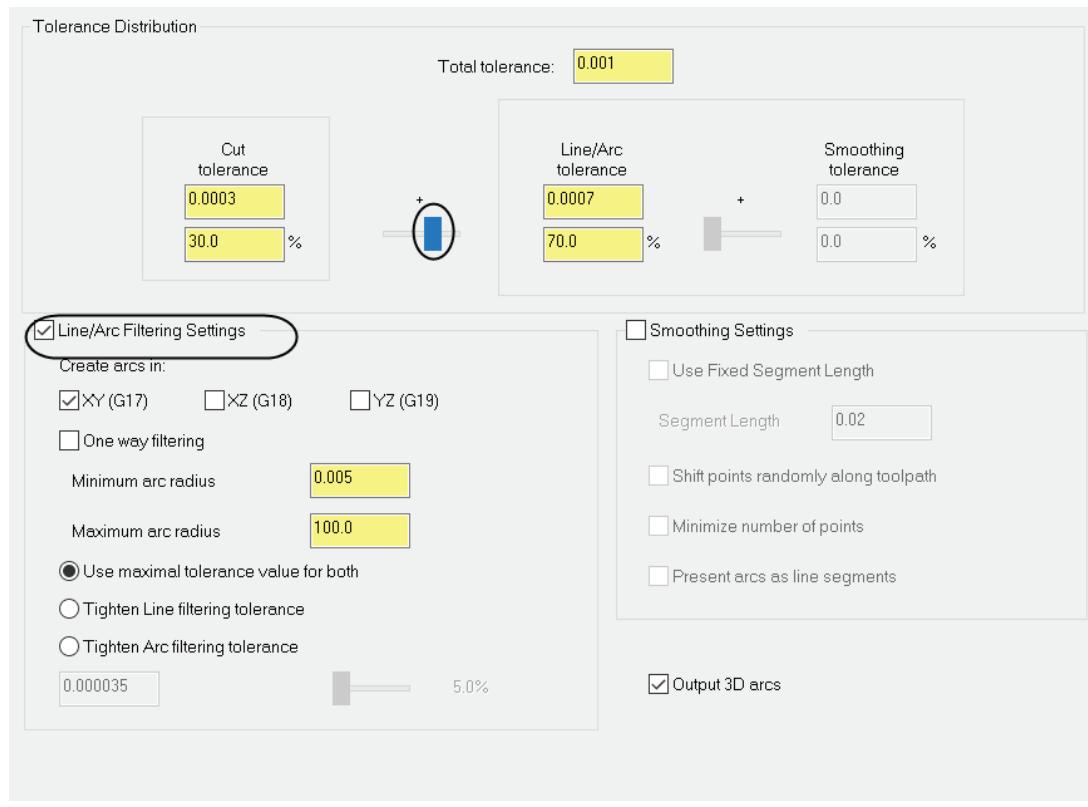
The **HST Leads** page allows you to specify an entry and exit arc radius value for the **2D High Speed Toolpaths**. The arc is created vertically to lead on and off the material.



9.8 Set the Arc Filter / Tolerance

- ◆ Choose **Arc Filter / Tolerance** from the **Tree View list**.
- ◆ If the **Warning** that **Mastercam Applet** is already set to output arcs for the selected toolpath type appears, select the **OK** button to continue with the filter.
- ◆ Make the necessary changes as shown in [Figure: 9.8.1](#).

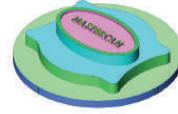
Figure: 9.8.1



Tolerance Distribution allows you to dynamically adjust the toolpath's total tolerance. Total tolerance is the sum of the **Cut tolerance** and the **Line/Arc** and **Smoothing** tolerances. Move the sliders between the **Cut tolerance**, **Line/Arc tolerance** and/or **Smoothing tolerance** fields. The ratios update in 5% increments and the toolpath's total tolerance remains at its current value.

Line/Arc Filtering Settings allows you to activate Line/Arc filtering for the toolpath and apply the settings you define in this section to the toolpath refinement. Toolpath filtering lets you replace multiple very small linear moves — within the filter tolerance — with single arc moves to simplify the toolpath. Smoothing distributes a toolpath's node points, avoiding the clustering and grouping of points that can cause marks and other imperfections.

Create arcs in creates arcs in the selected plane. Your post processor must be able to handle arcs and output the code G17, G18, G19 to select this option.

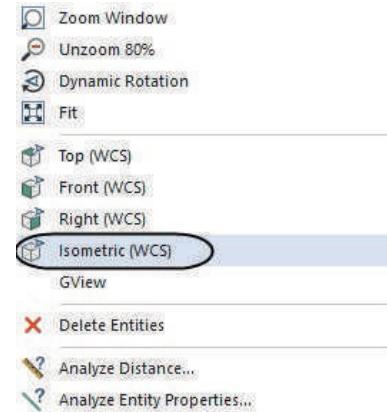


9.9 Preview the Toolpath

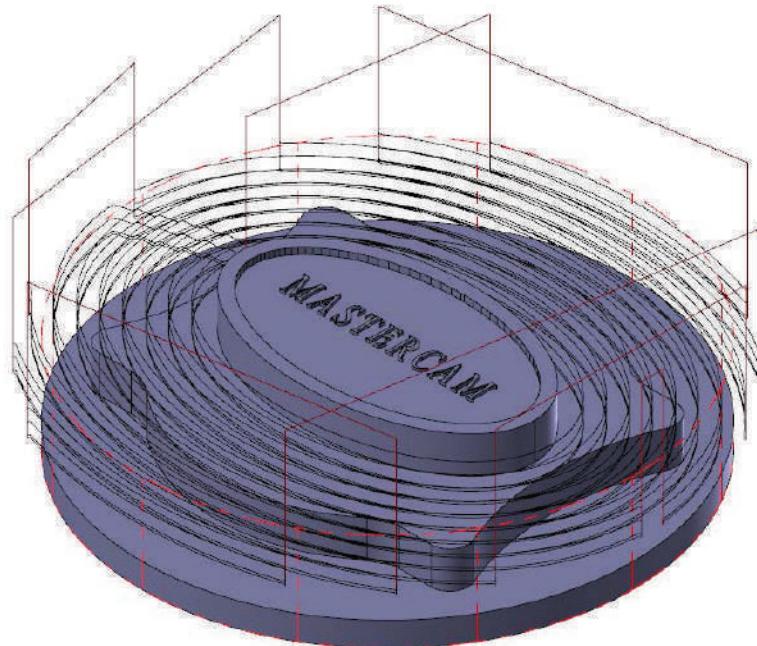
- ♦ To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.
- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.



- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to exit the **2D High Speed Toolpath - Area Mill** toolpath parameters.





STEP 10: BACKPLOT THE TOOLPATHS

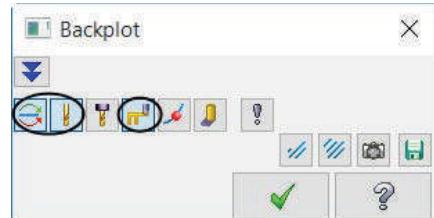
- ♦ Make sure that the toolpaths are selected (signified by the green check mark on the folder icon). If the operation is not selected, choose the **Select all operations** icon.



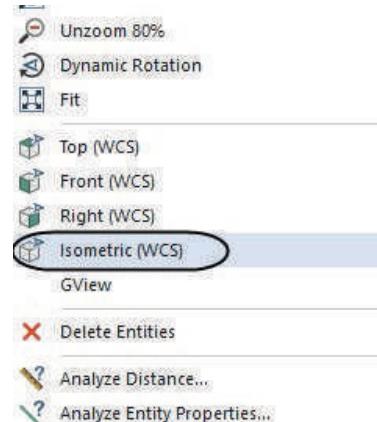
- ♦ Select the **Backplot selected operations** button.



- ♦ In the **Backplot** dialog box, enable **Display with color codes**, **Display tool** and **Display rapid moves** icons as shown.



- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ To fit the workpiece to the screen, if needed, right mouse click in the graphics window again and select **Fit**.

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

- ♦ You can adjust the speed of the backplot.

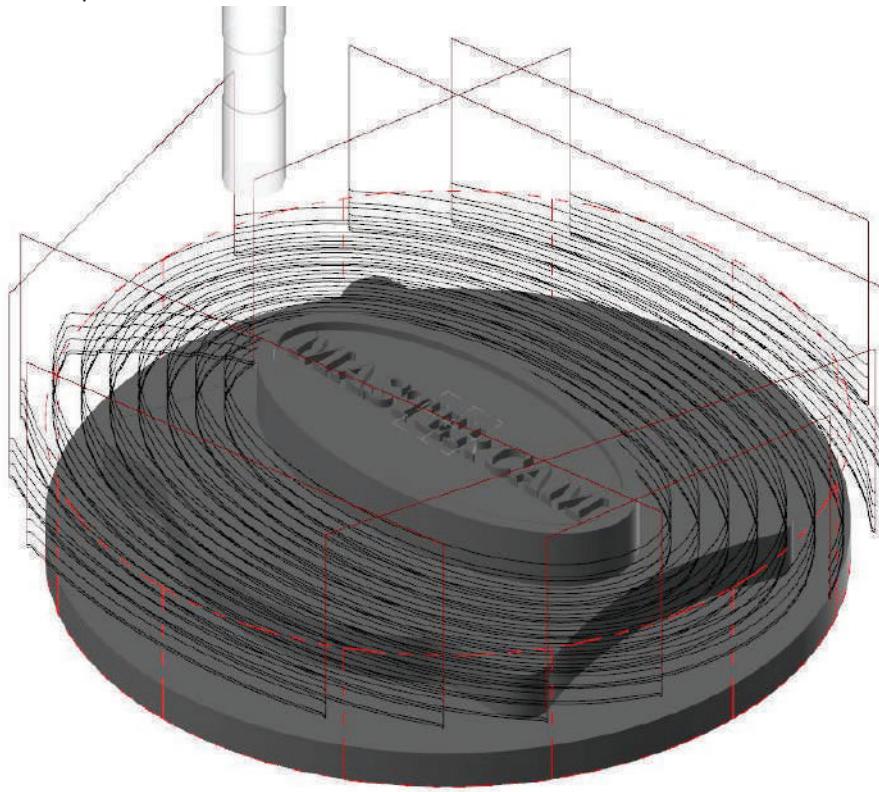


- ♦ Select the **Play** button to run **Backplot**.





- After Backplot is completed, the toolpath should look as shown.



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





STEP 11: SIMULATE THE TOOLPATH IN VERIFY

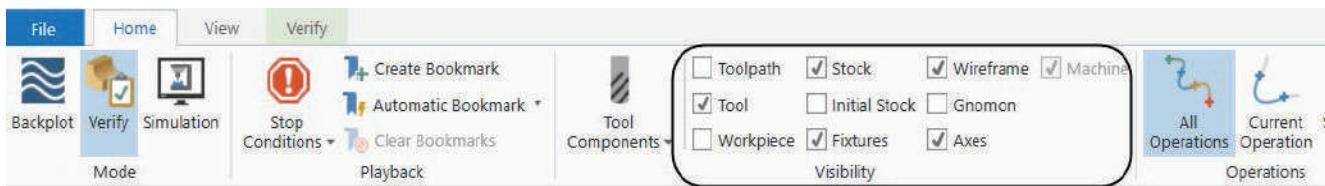
- From the Toolpaths Manager, select the Verify selected operations icon as shown.



NOTE: Mastercam launches a new window that allows you to check the part using Verify.

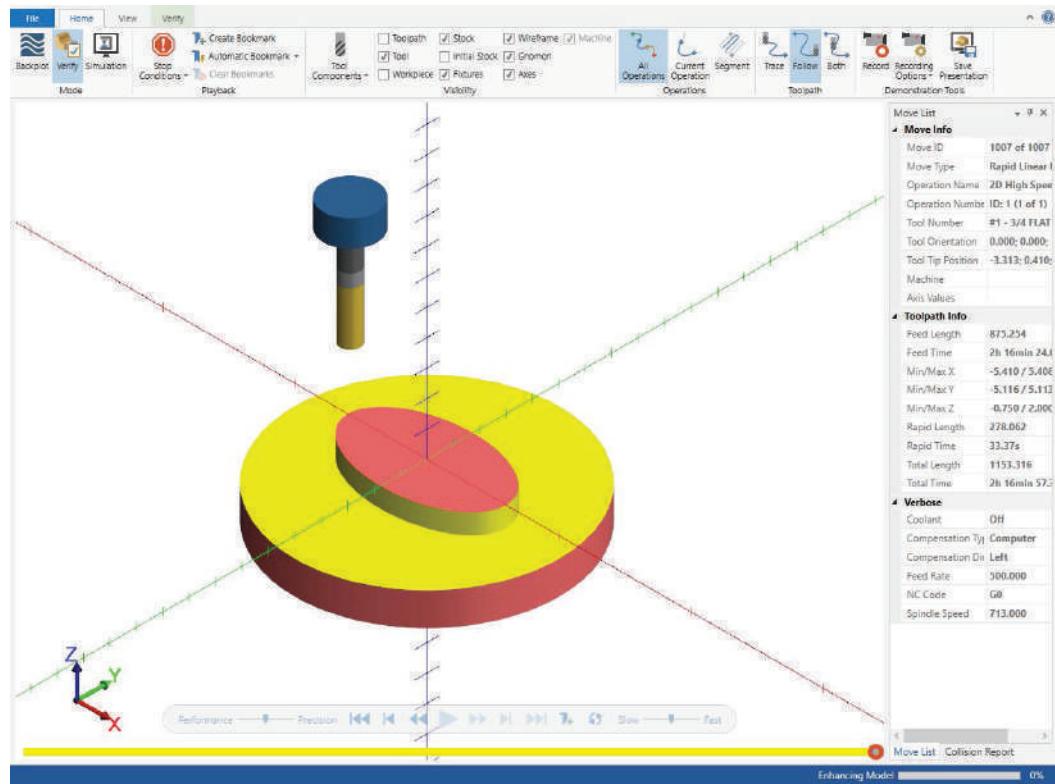
- In Mastercam Simulator, change the settings as shown in [Figure: 11.0.1](#).

Figure: 11.0.1

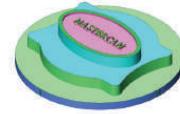


- Select the Play button to run Verify.
- The part should appear as shown in [Figure: 11.0.2](#).

Figure: 11.0.2



- To go back to the Mastercam window, minimize the Mastercam Simulator window as shown.



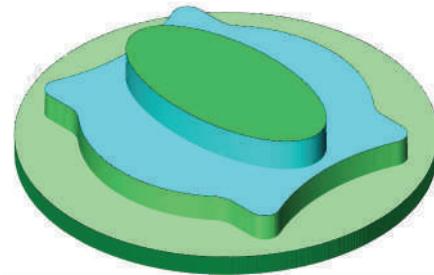
STEP 12: 2D HIGH SPEED DYNAMIC MILL

In this step you will machine the outside profile using **2D HS Dynamic Mill** toolpath which machines pockets, material that other toolpaths left behind, and standing bosses or cores using the entire flute length.

The toolpath supports many powerful entry methods, including a custom entry method. Entry methods and micro lifts support custom feeds and speeds to optimize and generate safe tool motion.

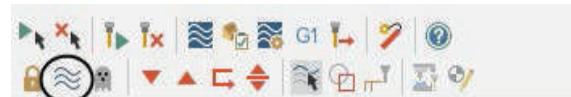
The toolpath depends on the **Machining strategy** that you choose in the **Chain Options**. If the strategy chosen is **From outside**, the toolpath starts at the outmost chain and works its way in taking on the final shape of the part as it approaches the final pass. You can also machine pockets in which case the strategy selected is **Stay inside** which keeps the tool inside the machining regions.

Toolpath Preview:



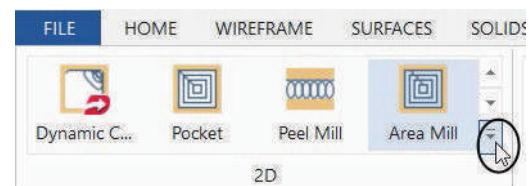
12.1 Chain Selection

- ♦ To remove the toolpath display, press **Alt + T** or click on the **Toggle display on selected operations** in the **Toolpaths Manager**.

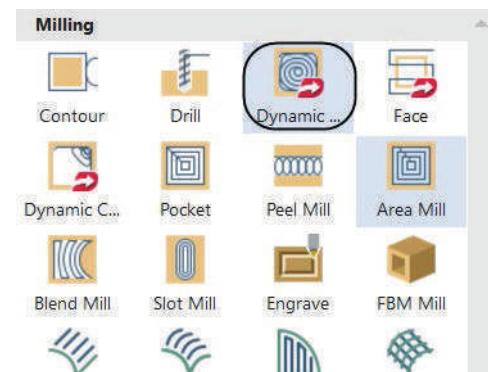


TOOLPATHS

- ♦ In the **2D** group, click on the **Expand gallery** arrow as shown.

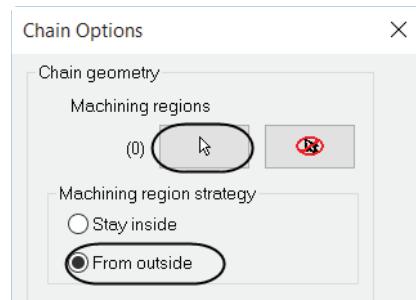


- ♦ Select the **Dynamic Mill** icon as shown.



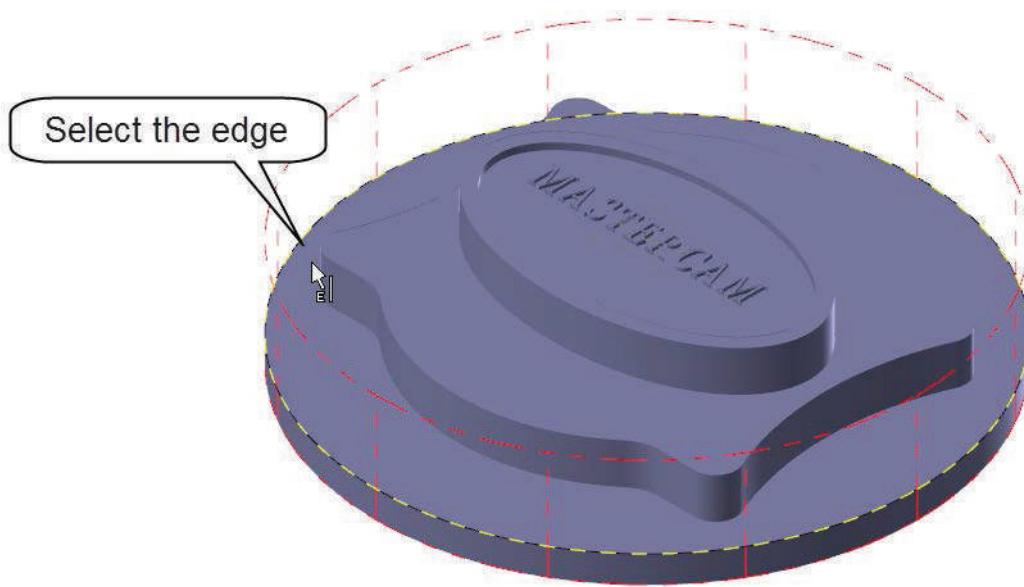


- In the **Chain Options** dialog box, **Machining regions**, enable **From outside** and click on the **Select machining chains** button as shown.



- Leave the default settings in the **Chaining** dialog box.
- Right mouse click in the graphics window and select **Isometric** view.
- [Select 2D HST machining chain 1]: Select the first chain as shown in [Figure: 12.1.1](#).

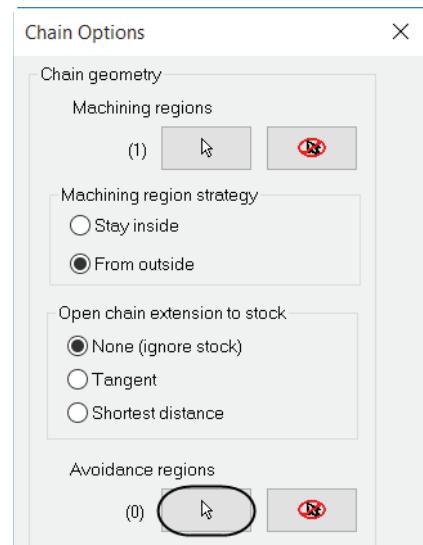
Figure: 12.1.1



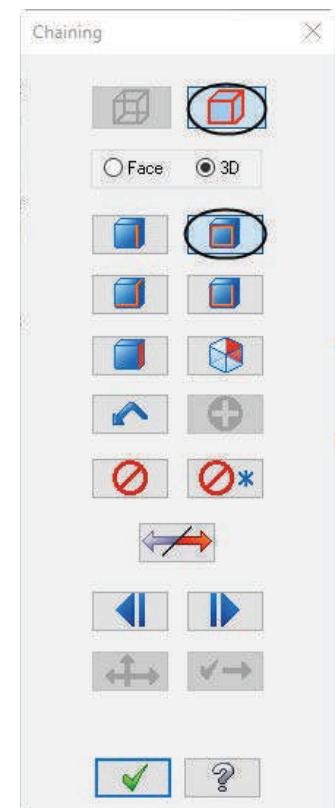
- Select the **OK** button to exit the **Pick Reference Face** dialog box.
- Select the **OK** button to exit the **Chaining** dialog box.



- ♦ In the **Chain Options** panel, **Avoidance regions**, click on the **Select avoidance chains** button as shown.



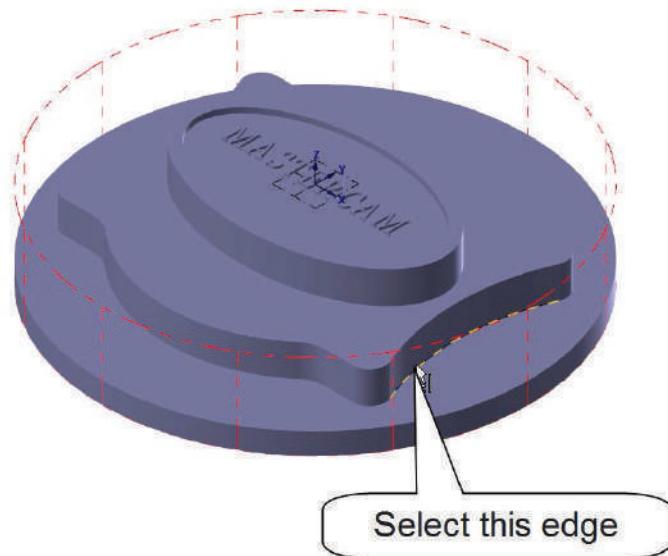
- ♦ Enable **Solids** in the **Chaining** dialog box if needed.
- ♦ Enable only the **Loop** button as shown.



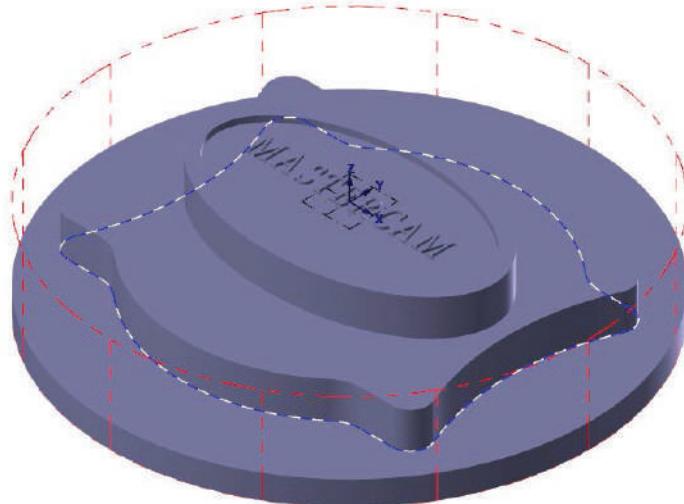


- ♦ [Solid Toolpath Chain: Select faces, edges and/or loops]: Select the edge as shown in [Figure: 12.1.2.](#)

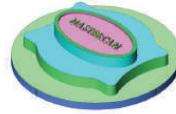
Figure: 12.1.2



- ♦ The chain should be selected as shown.



- ♦ Select the **OK** button to exit the **Pick Reference Face** dialog box.
- ♦ Select the **OK** button to exit the **Chaining** dialog box.
- ♦ Select the **OK** button to exit the **Chain Options** dialog box.

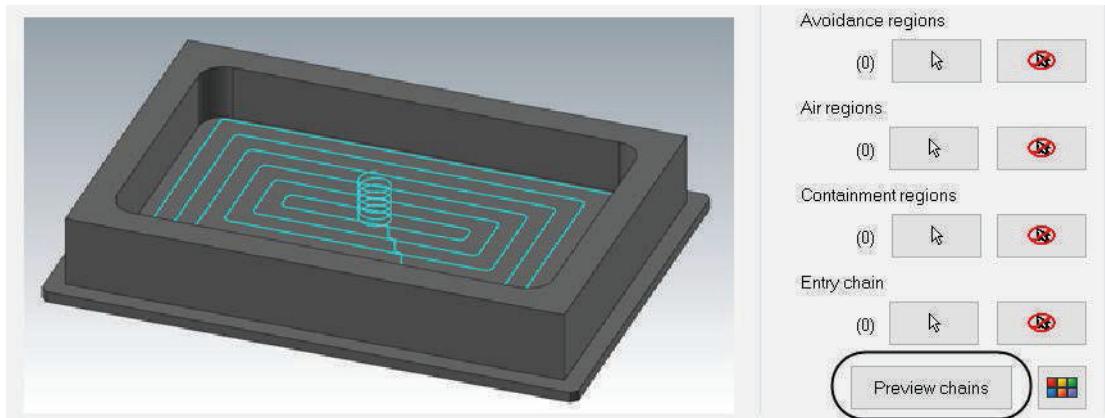


- In the Toolpath Type page, the **Dynamic Mill** should already be selected as shown.



12.2 Preview Chains

- Select the **Preview chains** button as shown.



- See page 239 to review the procedure.
- The **Preview chains** should look as shown.



- Press **Esc** key to return to the toolpath parameters.
- Click on the **Preview chains** button again to clear the **Preview chains** display.

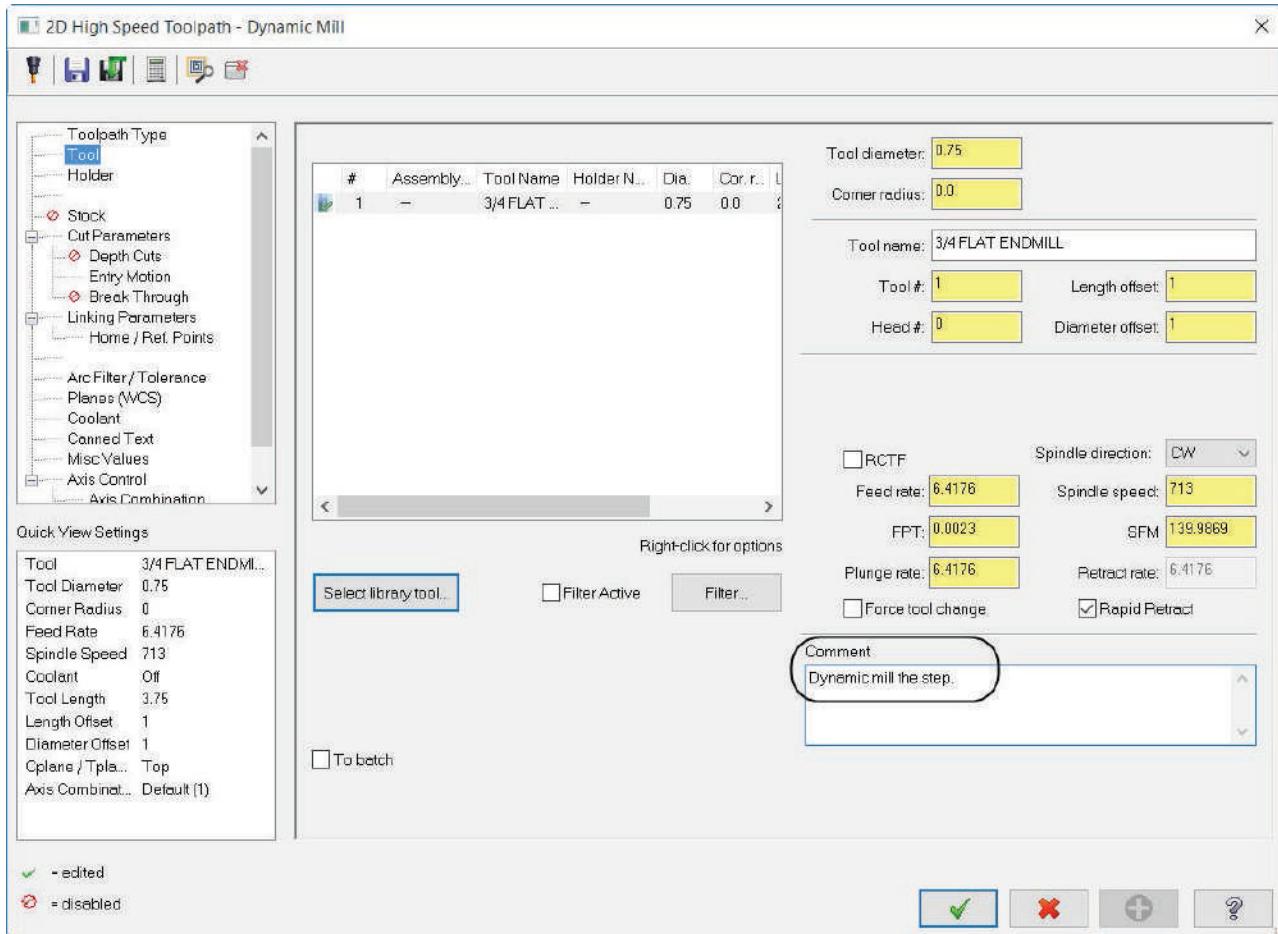


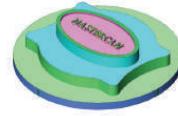


12.3 Select the existing 3/4" Flat Endmill from the list and set the Tool Parameters

- From the Tree View list, click on **Tool** and make all the necessary changes as shown in [Figure: 12.3.1](#).

Figure: 12.3.1

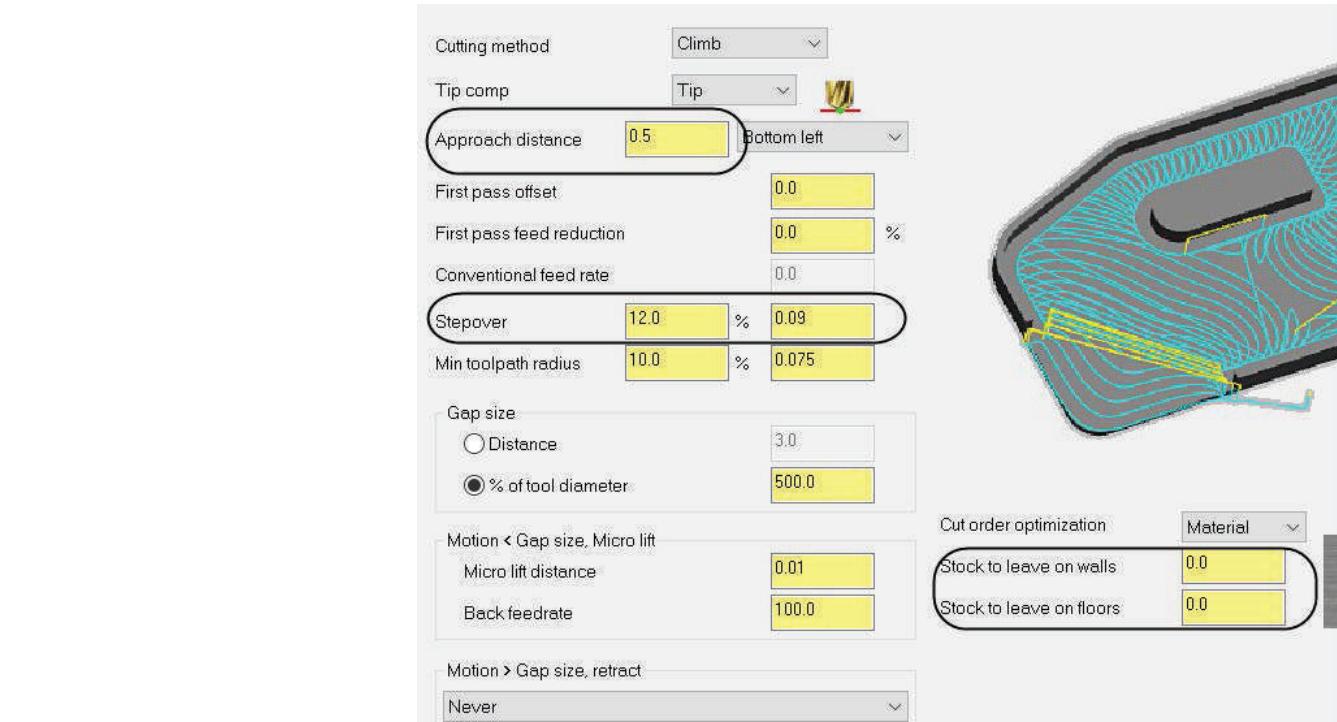




12.4 Set the Cut Parameters

- From the Tree View list, select **Cut Parameters**. Input a **Stepover** value of **12%** as shown in [Figure: 12.4.1](#).

Figure: 12.4.1



Approach distance is available only when open pocket machining is selected. This adds the specified absolute distance to the beginning of the toolpath's first cut.

First pass offset offsets out the machining region with a user defined distance for the tool to safely engage the material from the outside.

First pass feed reduction allows you to slow the feed for the first pass on machining region material approached from the outside.

Stepover sets the distance between cutting passes in the X and Y axes.

Toolpath radius reduces sharp corner motion between cut passes.

Micro lift distance enters the distance the tool lifts off the part on the back moves. Microlifts are slight lifts that help clear chips and minimize excessive tool heating.

Back feedrate controls the speed of the backfeed movement of the tool.

Motion > Gap size, retract controls retracts in the toolpath when making a non-cutting move within an area where the tool can be kept down or microlifted.

Cut order optimization defines the cut order Mastercam applies to different cutting passes in the dynamic mill toolpath.



12.5 Disable the Depth cuts parameters

- From the **Tree View list**, select the **Depth Cuts** and make sure it is disabled as shown.



12.6 Set the Entry Motion

- From the **Tree View list**, select **Entry Motion**.
- Input a **Z clearance** of **0.05** and a **Plunge angle** of **2.0** degrees.
- Enable **Entry feeds / speeds** and set a **Ramp feed rate** of **10.0** inches per minute, a **Ramp spindle speed** of **4000 RPM** and **Dwell before cut spindle speed** of **3.0** seconds as shown in [Figure: 12.6.1](#).

Figure: 12.6.1



Entry method defines the entry point and cutting strategy used to create the toolpath.

Z clearance adds an extra height used in the ramping motion down from a top profile. It ensures that the tool has fully slowed down from rapid speeds before touching the material.

Plunge angle sets the angle of descent for the entry move and determines the pitch.

Rapid feed rate overrides the feed rate set on the **Tool** page and uses the specified feed rate for entry ramps into the material.

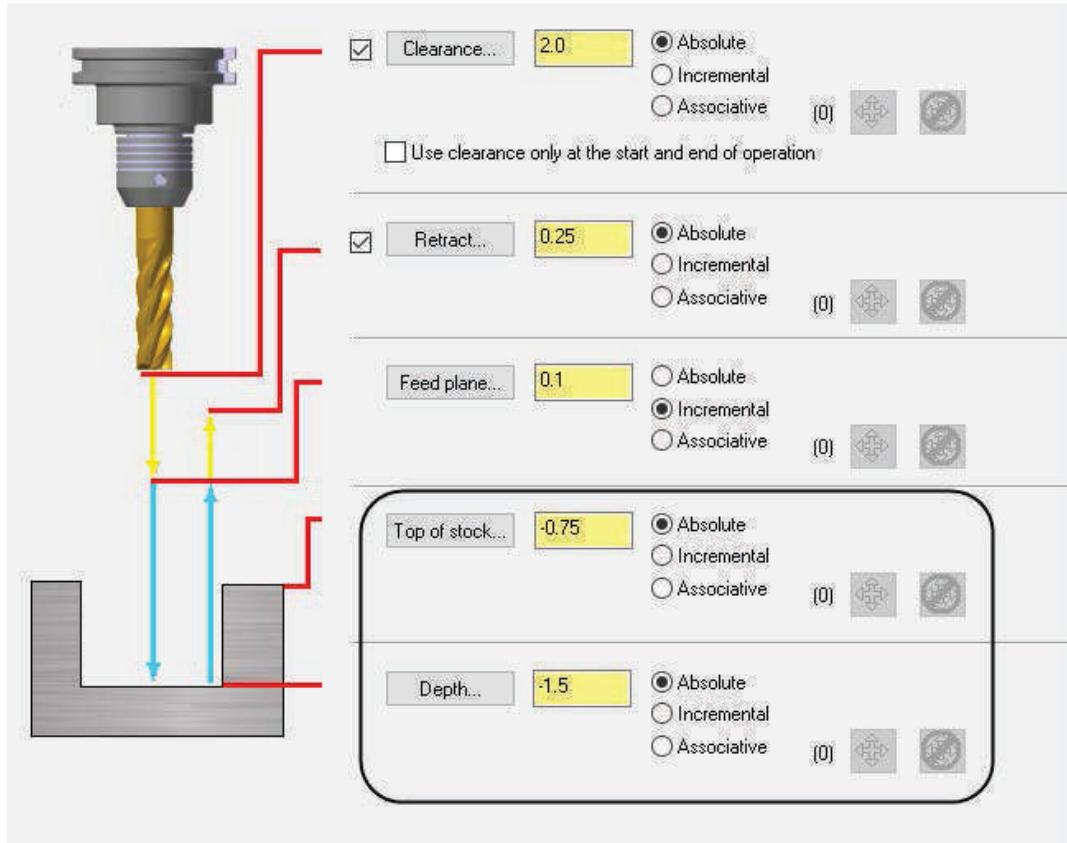
Ramp spindle speed overrides the spindle speed set in the **Tool** page and uses the specified spindle speed for entry ramps into the material.



12.7 Set the Linking Parameters

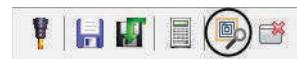
- ♦ Select **Linking Parameters** and input the **Depth** and **Top of stock** as shown in [Figure: 12.7.1](#).

Figure: 12.7.1



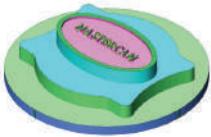
12.8 Preview the Toolpath

- ♦ To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

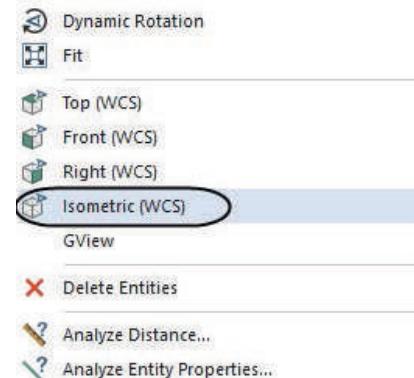


- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.

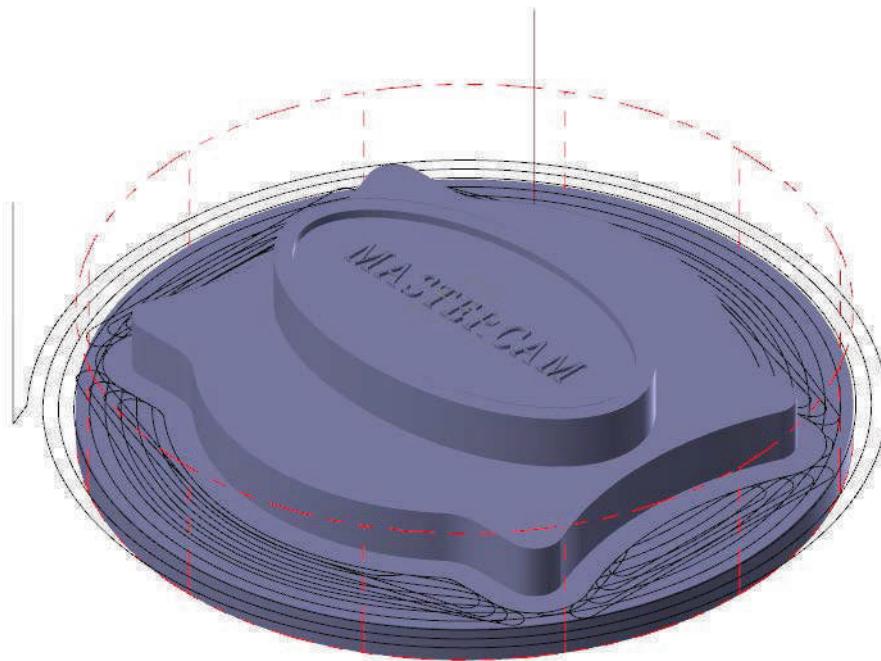




- To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- The toolpath should look as shown.

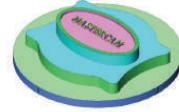


- Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

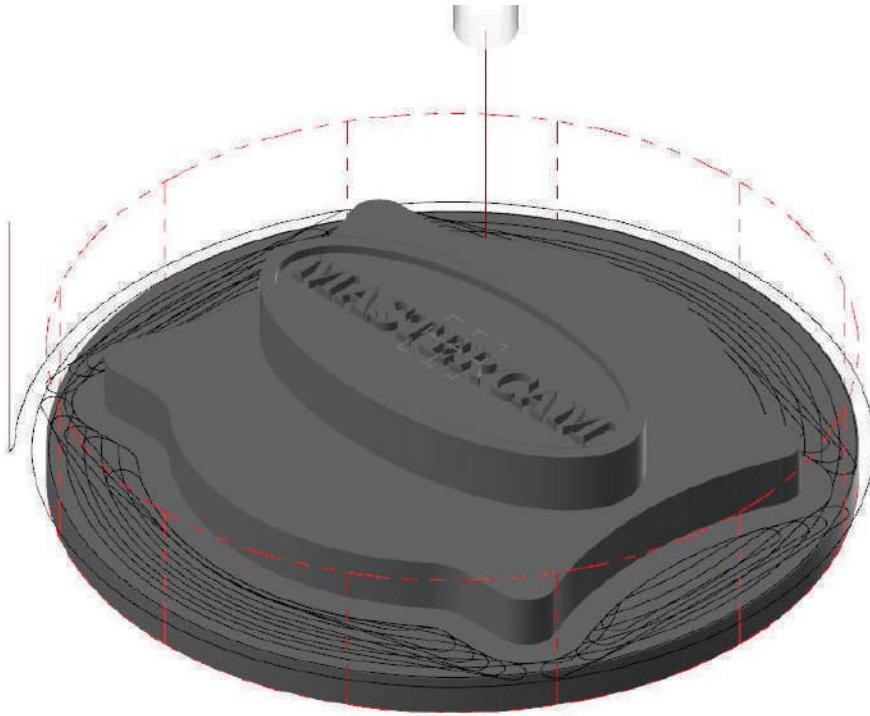
- Select the **OK** button to exit the **2D High Speed Toolpath - Dynamic Mill** toolpath parameters.





12.9 Backplot the toolpath

- To **Backplot** the toolpath, see [page 145](#) to review this procedure. Ensure that only the **Dynamic Mill** toolpath is selected in the **Toolpaths Manager**.

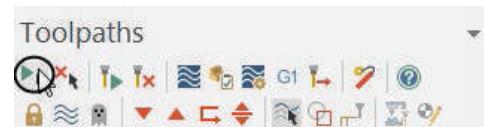


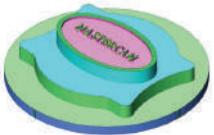
- Select the **OK** button to exit **Backplot**.



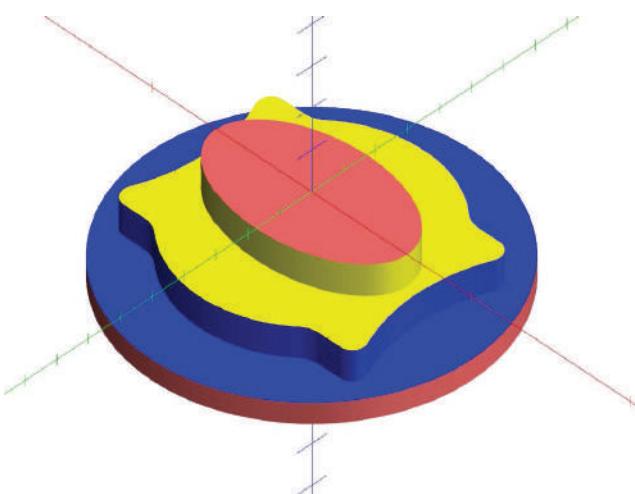
12.10 Simulate the toolpaths using Verify

- To select all operations, in the **Toolpaths Manager**, click on the **Select all operations** icon.





- ♦ To Verify the toolpath, see [page 147](#).

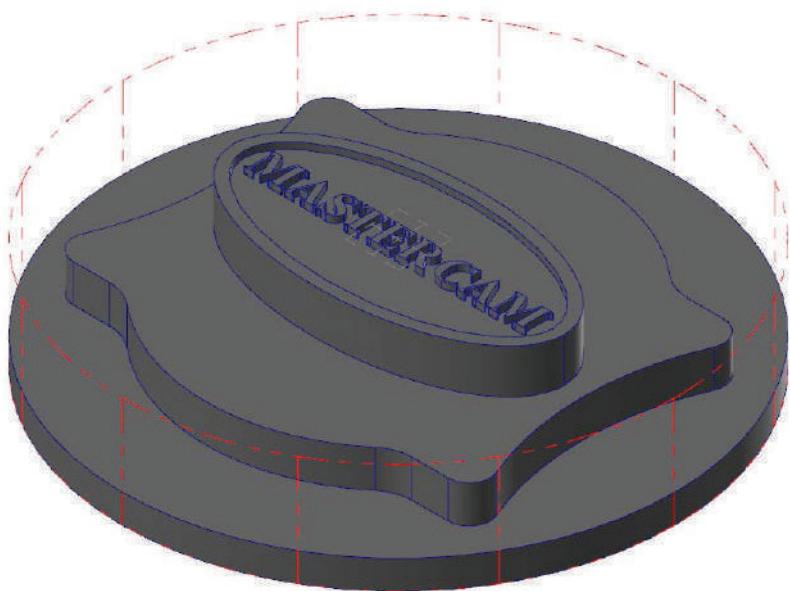


- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.

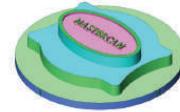


STEP 13: CREATE CURVES FROM A SOLID

Toolpath Preview:



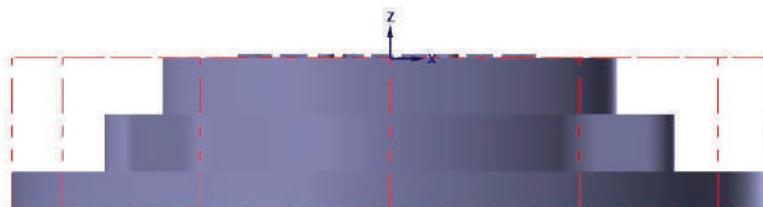
- ♦ Press **Alt + T** to remove the toolpath display.



- Right mouse click in the graphics window and change the graphics view to the **Front**.



- The geometry should look as shown.

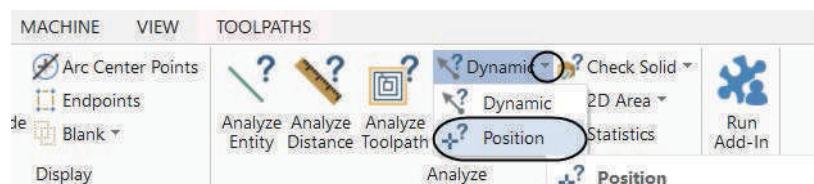


NOTE: The letters are separate solids that are above Z zero. You will have to analyze their current position and use **TRANSFORM/Translate** to move them inside of the pocket.

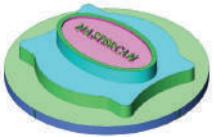
13.1 Analyze the position of the letters and the bottom of the pocket

HOME

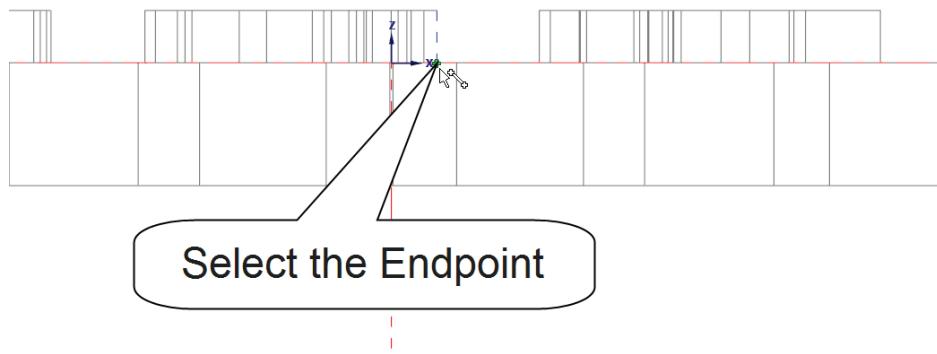
- From the **Analyze** group, click on the drop down arrow besides **Dynamic** and select **Position** as shown.



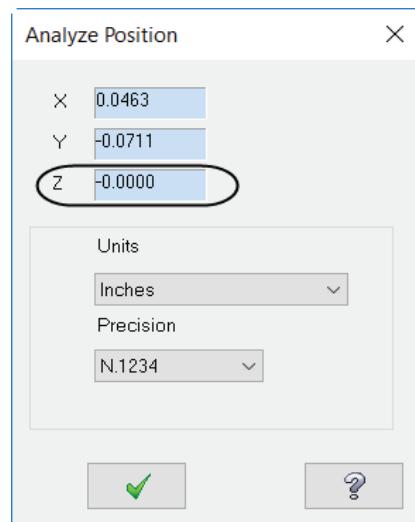
- Press **Alt + S** to unshade the solids.
- Move the cursor close to the **Origin** and scroll up the mouse wheel to zoom in.



- ♦ [Select a position]: Select the Endpoint as shown.



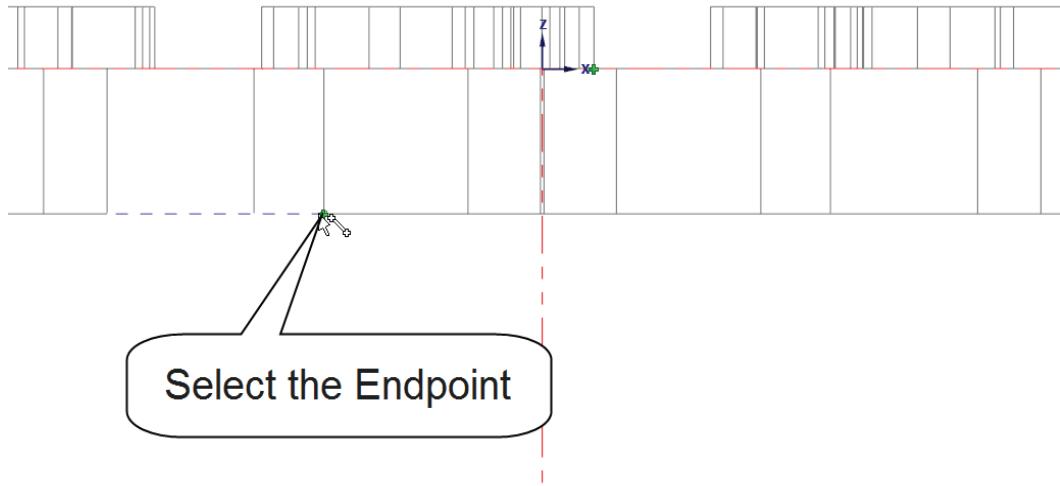
- ♦ The **Analyze Position** dialog box should look as shown.



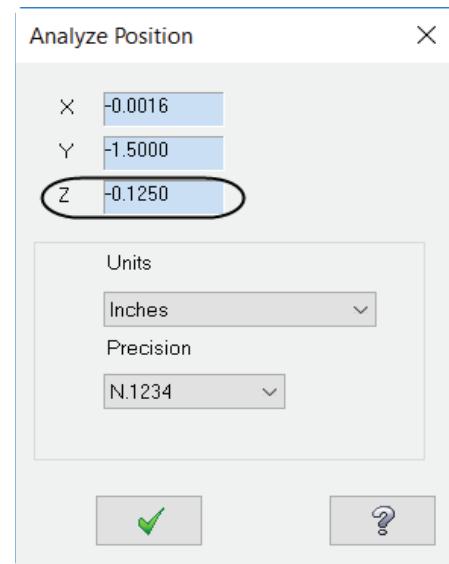
NOTE: Z value is 0.0



- ♦ Select an Endpoint at the bottom of the pocket as shown.



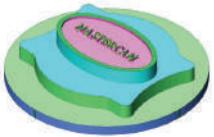
- ♦ The **Z** value in the **Analyze Position** dialog box should be **-0.1250** as shown.



NOTE: You will have to move the letters to **Z -0.125**.

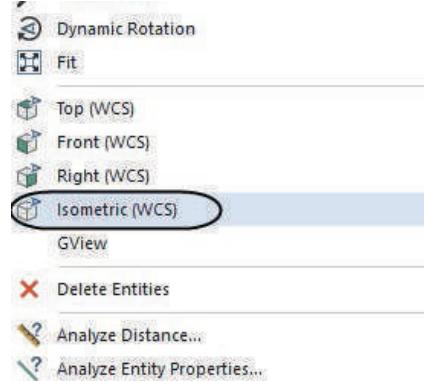
- ♦ Select the **OK** button to exit **Analyze Position**.



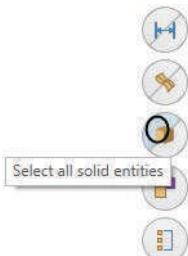


13.2 Move the letters along Z axis

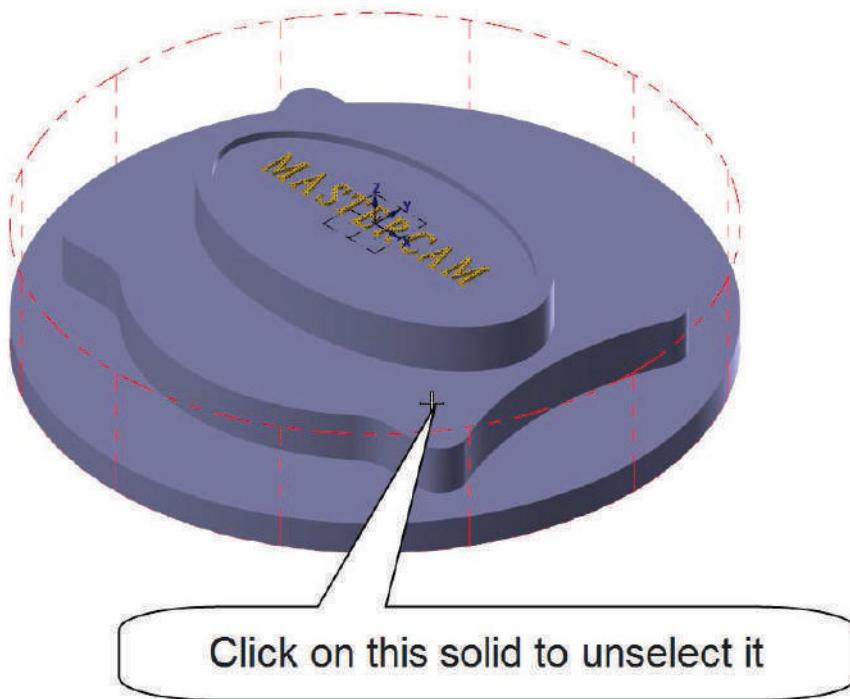
- ♦ Right mouse click in the graphics window and select **Isometric** as shown.



- ♦ Press **Alt + F1** to fit the geometry to the screen.
- ♦ Press **Alt + S** to shade the solid.
- ♦ From the **Quick Mask** toolbar, choose **Select all solid entities** as shown.

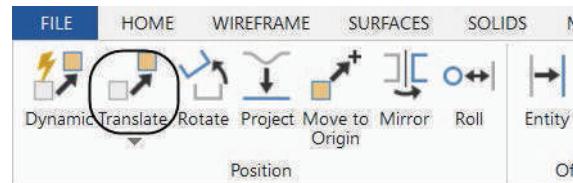


- ♦ Unselect the main body as shown.

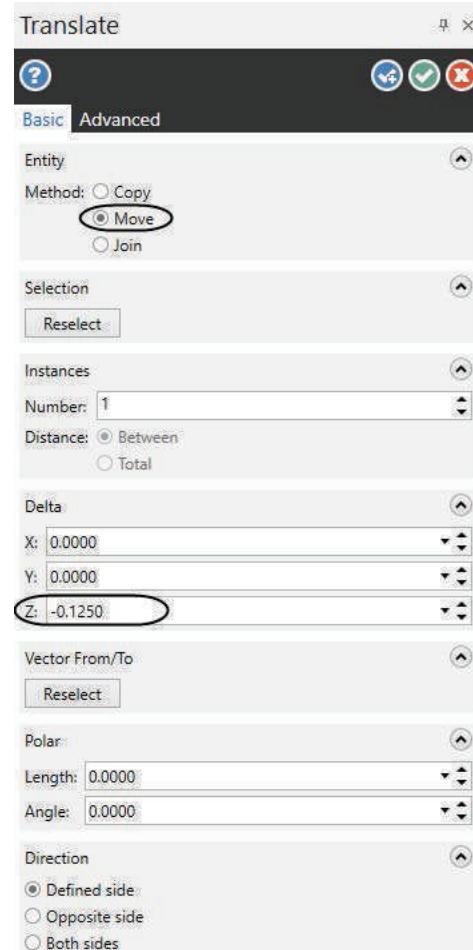


**TRANSFORM**

- From the **Position** group, select **Translate** as shown.



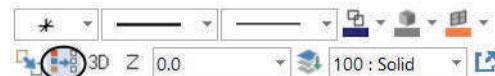
- In the **Translate** panel, enable **Move** and change the **delta Z** value to **-0.125** as shown.

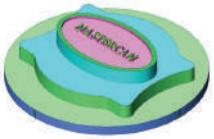


- Select the **OK** button to exit the **Translate** panel.

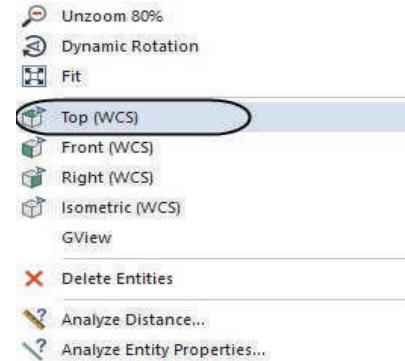


- Right mouse click on the graphics window and select the **Clear Colors** icon to remove the result color.





- Right mouse click again and select the **Top** view from the list to see the part from the top. Press **Alt + S** to unshade the solids.

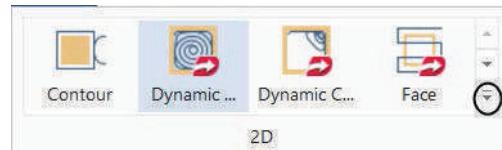


2D HIGH SPEED AREA TO MACHINE THE POCKET WITH ISLANDS

In this step you will use the **2D High Speed Area** with the **Island facing** option to machine the pocket and to leave the inside letters as islands.

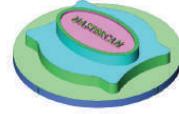
TOOLPATHS

- From the **2D** group, select the **Expand gallery** arrow as shown.



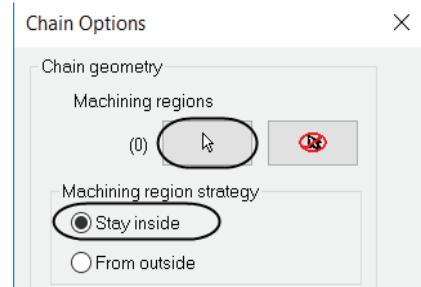
- Select the **Area Mill** icon as shown.





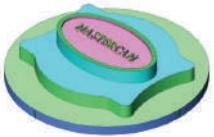
13.3 Select the Geometry

- In the **Chaining Options** dialog box, enable **Stay inside** and click on the **Select machining chain** button as shown.



- In the **Chaining** dialog box, make sure that **Solids** selection is enabled and then enable only the **Face** button as shown.



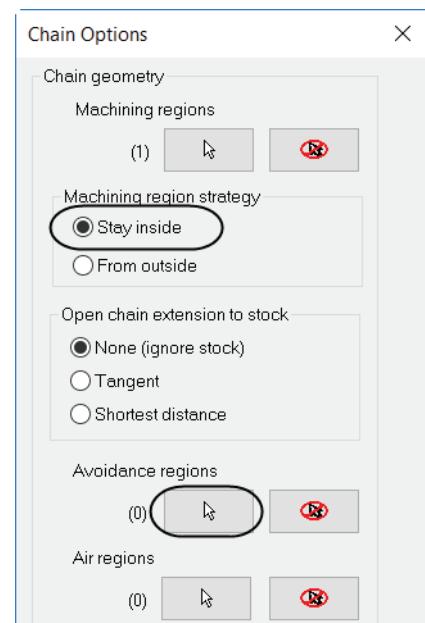


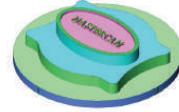
- ♦ [Solid Toolpath Chain: Select faces, edges, and/or loops]: Zoom in using the mouse wheel, and then select the face of the solid as shown in [Figure: 13.3.1](#).

Figure: 13.3.1



- ♦ Select the **OK** button to exit the **Chaining** dialog box.
- ♦ In the **Avoidance regions**, click on the **Select** button as shown.



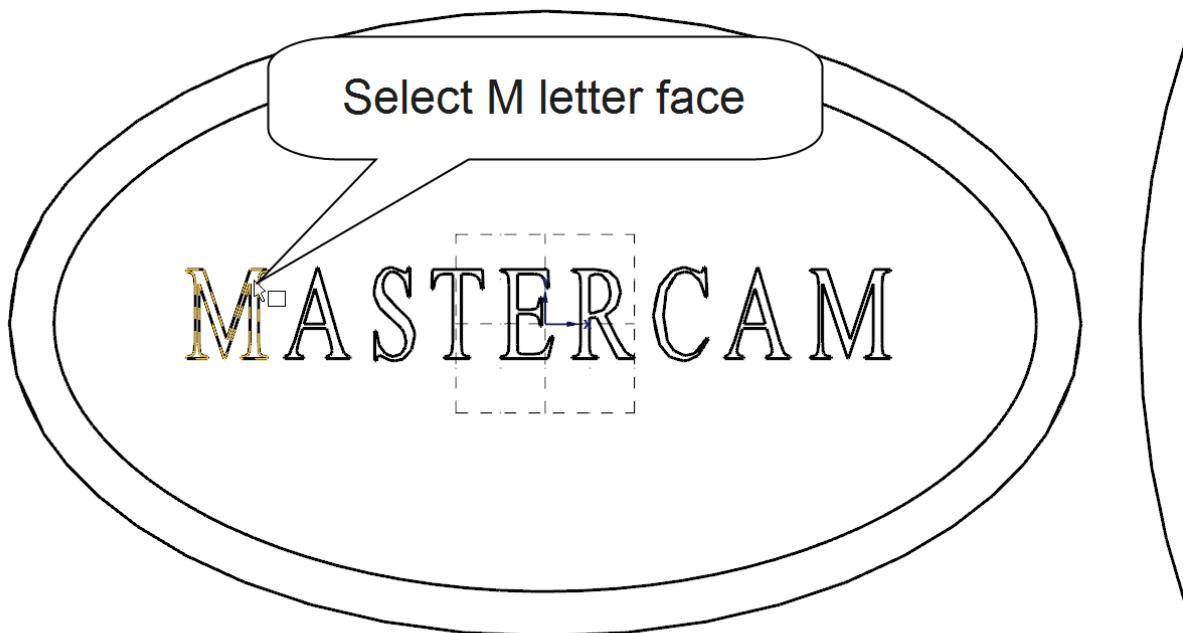
TUTORIAL #3**2D HIGH SPEED AREA TO MACHINE THE POCKET WITH ISLANDS**

- ♦ In the Chaining dialog box, enable only the **Face** button as shown.



- ♦ [Solid Toolpath Chain: Select faces, edges, and/or loops]: Select the letter **M** face as shown in [Figure: 13.3.2](#).

Figure: 13.3.2





- ♦ Continue to select all the other letter faces as shown in [Figure: 13.3.3.](#)

Figure: 13.3.3



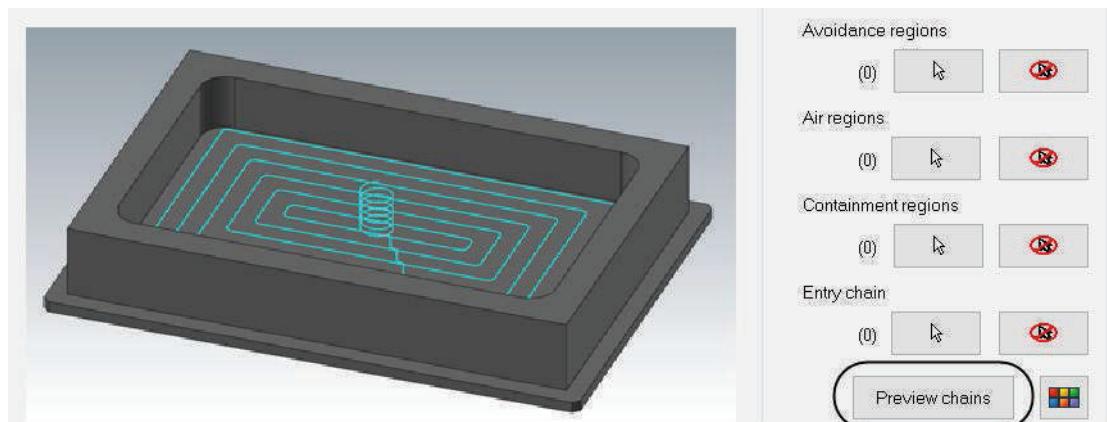
- ♦ Choose the **OK** button to exit the **Chaining** dialog box.
- ♦ Select the **OK** button to exit the **Chain Options** dialog box.
- ♦ On the **Toolpath Type** page, **Area Mill** will be picked.





13.4 Preview Chains

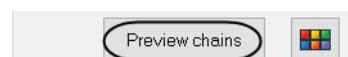
- ♦ Select the **Preview chains** button as shown.

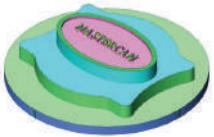


- ♦ See page 239 to review the procedure.
- ♦ The **Preview chains** should look as shown.



- ♦ Press **Esc** key to return to the toolpath parameters.
- ♦ Click on the **Preview chains** button again to clear the **Preview chains** display.





13.5 Select a 1/8" Flat Endmill from the library and set the Tool Parameters

- ♦ Select **Tool** from the **Tree View** list.

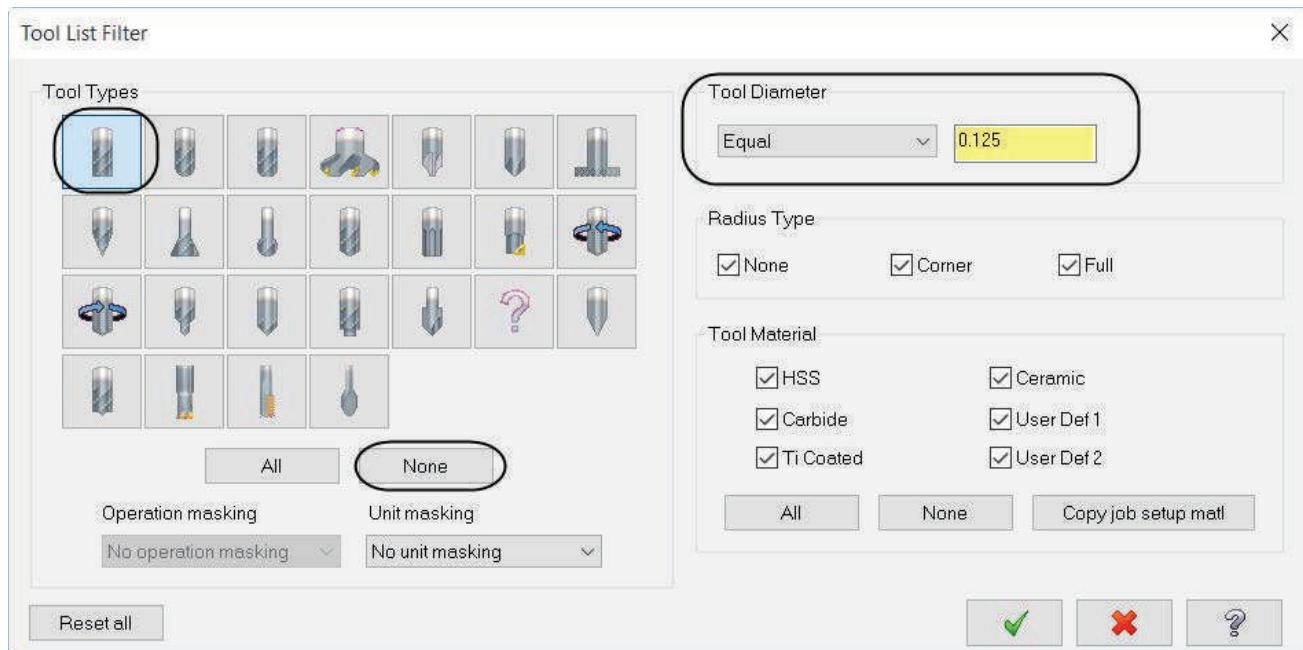
♦ Click on the **Select library tool** button.

- ♦ Select the **Filter** button.



- ♦ Select the **None** button and then under **Tool Types**, choose the **Flat Endmill** icon.
- ♦ Under **Tool Diameter** pick **Equal** and input a value **0.125** as shown in [Figure: 13.5.1](#).

Figure: 13.5.1

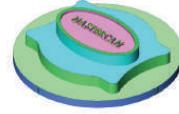


- ♦ Select the **OK** button to exit the **Tool List Filter**.
- ♦ In the **Tool Selection** dialog box, you should only see a **1/8" Flat Endmill**.

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	Type	# Flutes	Ra...
282	-	1/8 FLAT ...	-	0....	0.0	0.375	En...	4	No...

- ♦ Select the **1/8" Flat Endmill** in the **Tool Selection** page and then select the **OK** button to exit.



TUTORIAL #3**2D HIGH SPEED AREA TO MACHINE THE POCKET WITH ISLANDS**

- ♦ Make all the necessary changes as shown in [Figure: 13.5.2.](#)

Figure: 13.5.2

#	Assembly Name	Tool Name	Holder Name	Dia.
1	-	3/4 FLAT ...	-	0.75
2	-	1/8 FLAT ...	-	0....

Tool diameter: 0.125
Corner radius: 0.0
Tool name: 1/8 FLAT ENDMILL
Tool #: 2 Length offset: 2
Head #: 0 Diameter offset: 2

RCTF Spindle direction: CW ▾
Feed rate: 6.160896 Spindle speed: 4278
FPT: 0.0004 SFM 139.9869
Plunge rate: 6.160896 Retract rate: 6.160896
 Force tool change Rapid Retract

Select library tool... Filter Active Filter...
 Tool inspection / change
Force retract every: 0.0 Inches 0.0 Minutes
 To batch

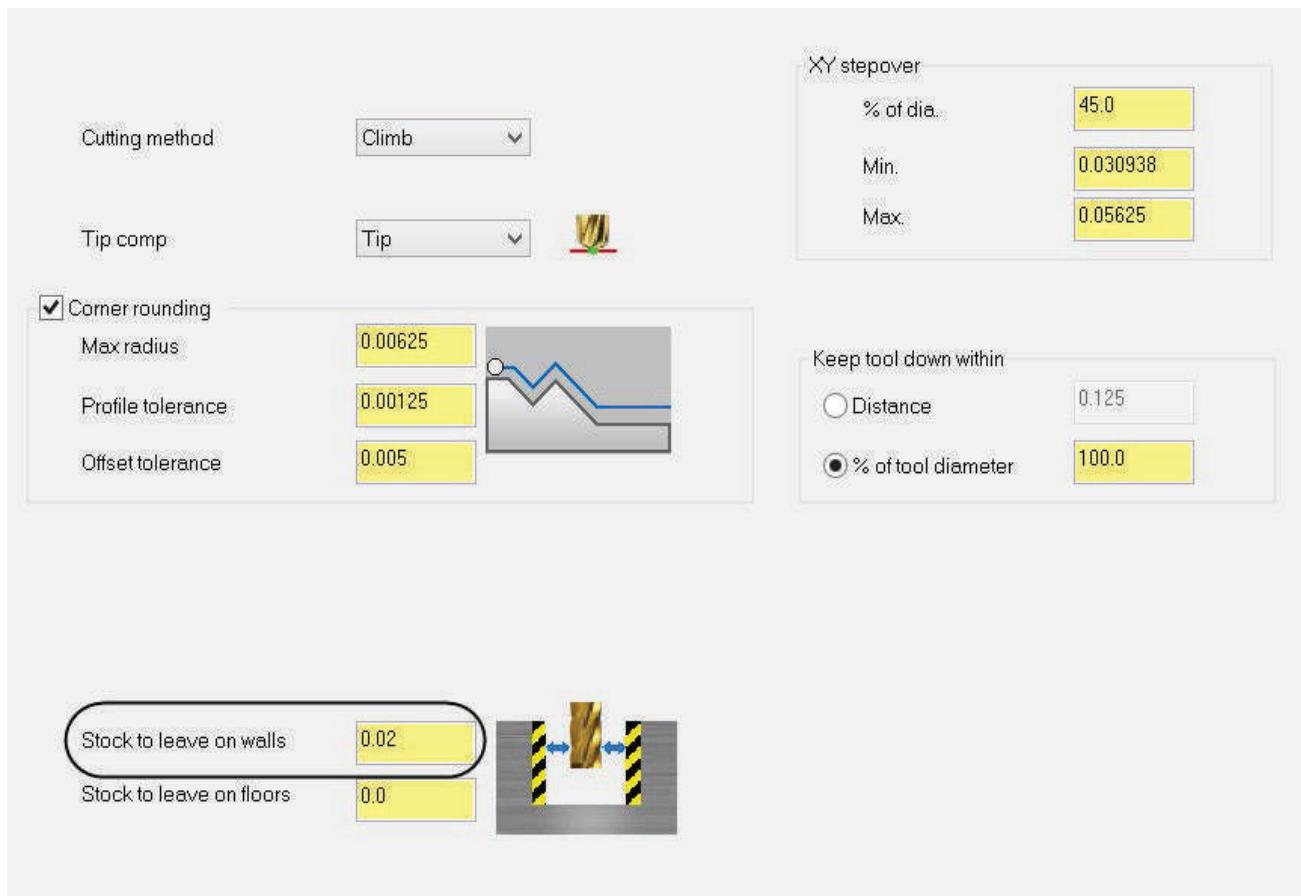
Comment
Machine the pocket leaving the letters as islands.

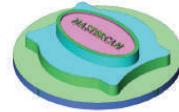


13.6 Set the Cut Parameters

- From the Tree View list, select **Cut Parameters** and ensure the parameters are set as shown in [Figure: 13.6.1](#).

Figure: 13.6.1

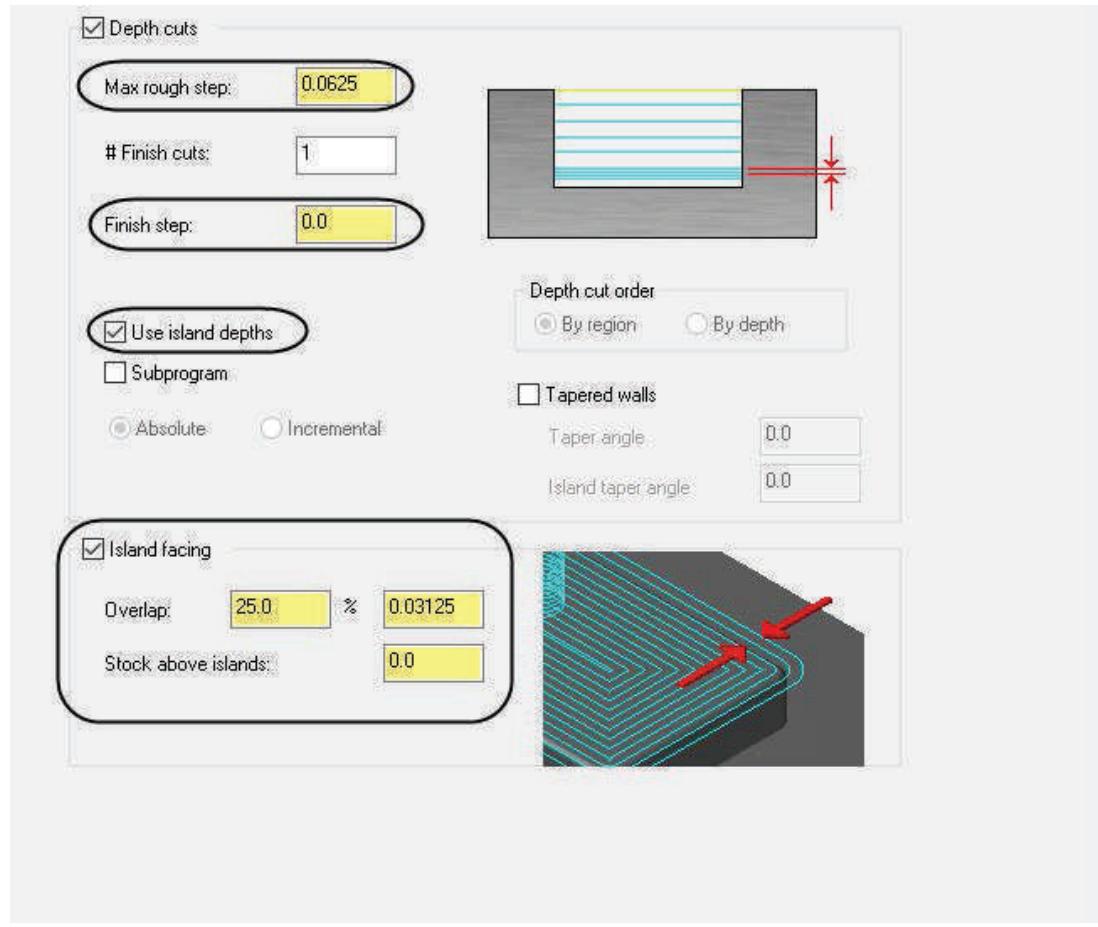




13.7 Set the Depth cuts parameters

- From the Tree View list, select **Depth Cuts**.
- Enable **Depth cuts** and ensure your parameters appear as shown in [Figure: 13.7.1](#).

Figure: 13.7.1



Use island depths creates extra passes over the island until it reaches the depth of the island.

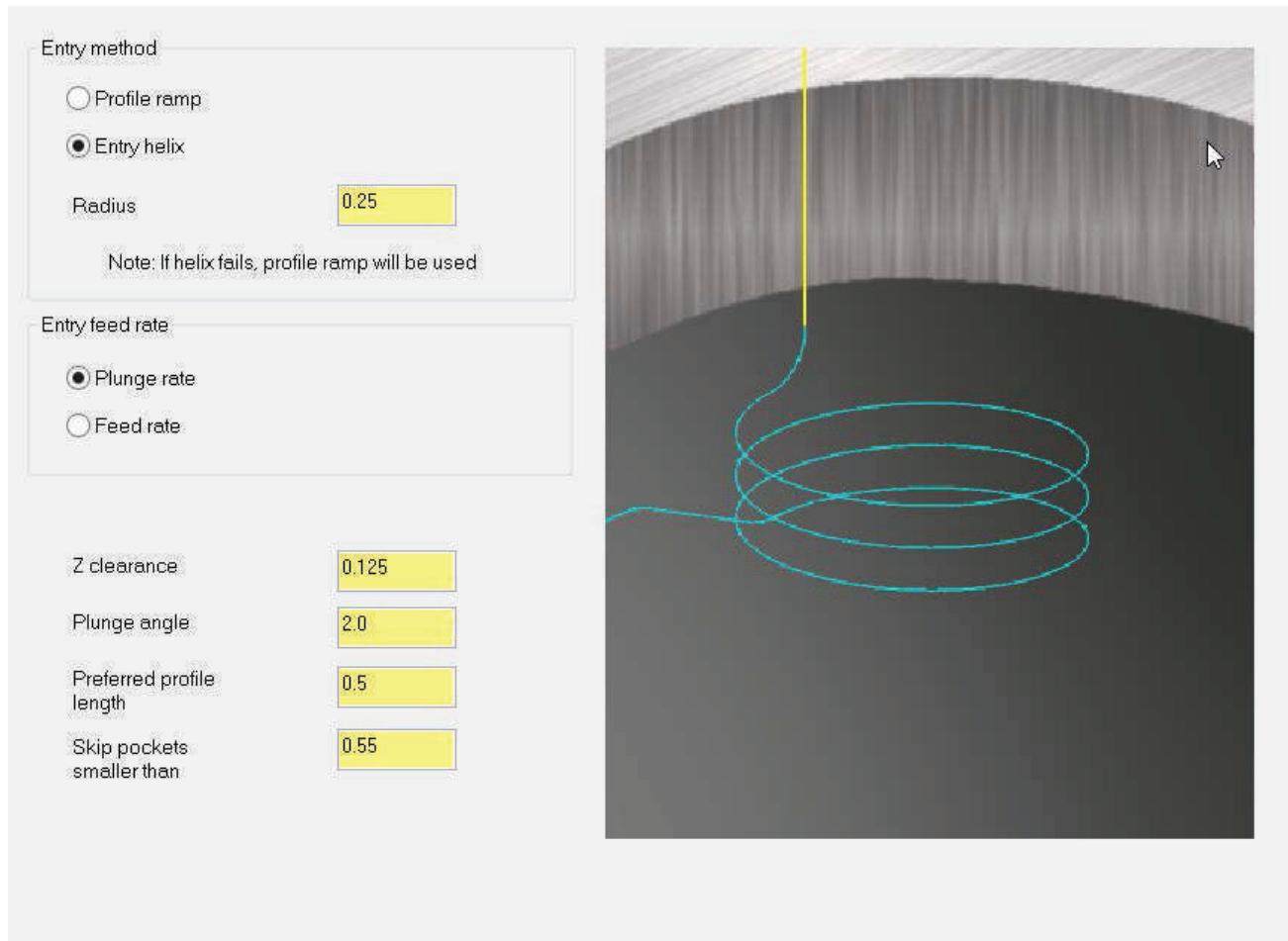
Island facing allows you to face the top of all the islands selected.

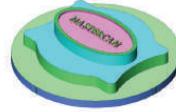


13.8 Set the Transitions

- ♦ Choose **Transitions** from the **Tree View** list. Ensure your settings appear as shown in [Figure: 13.8.1](#).

Figure: 13.8.1

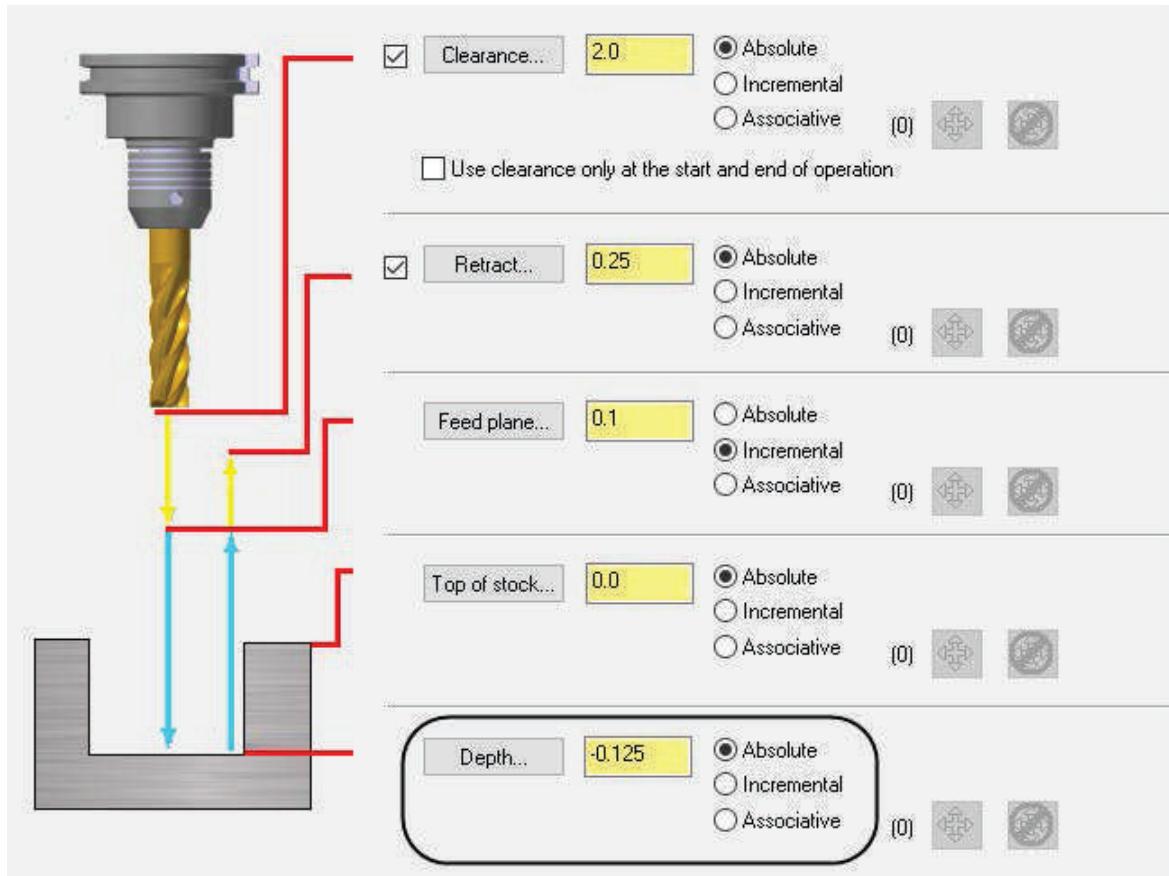




13.9 Set the Linking Parameters

- ♦ Select **Linking Parameters** and ensure your options appear as shown in [Figure: 13.9.1](#).

Figure: 13.9.1



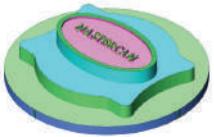
13.10 Preview the Toolpath

- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

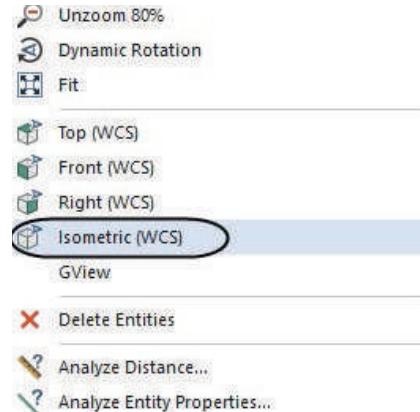


- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.

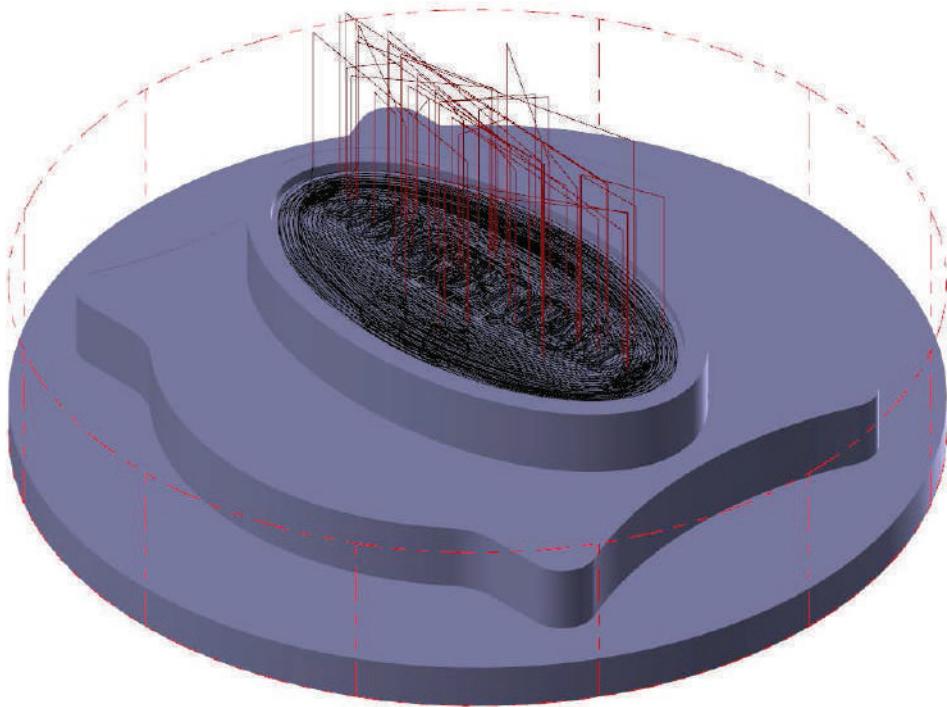




- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.



- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

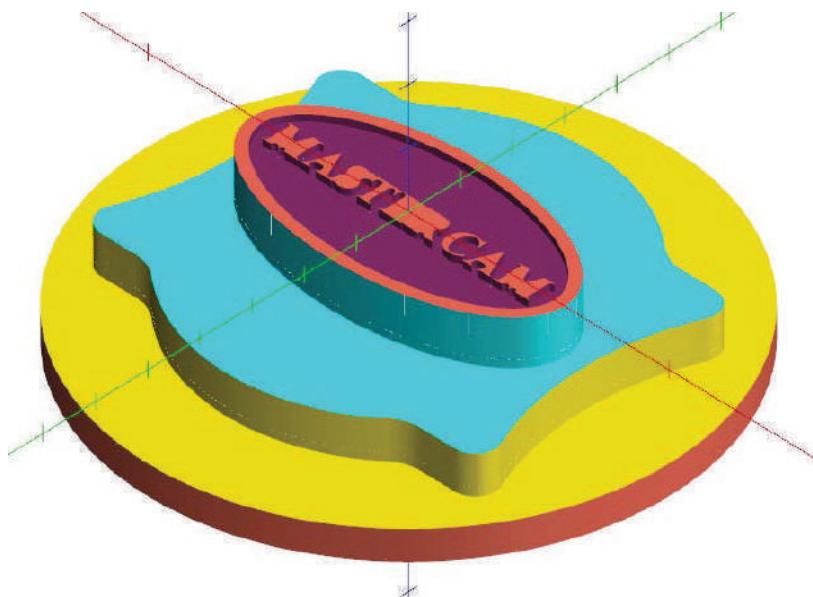
- ♦ Select the **OK** button to exit the **2D High Speed Toolpath - Area Mill** parameters.





13.11 Backplot and Verify the toolpath

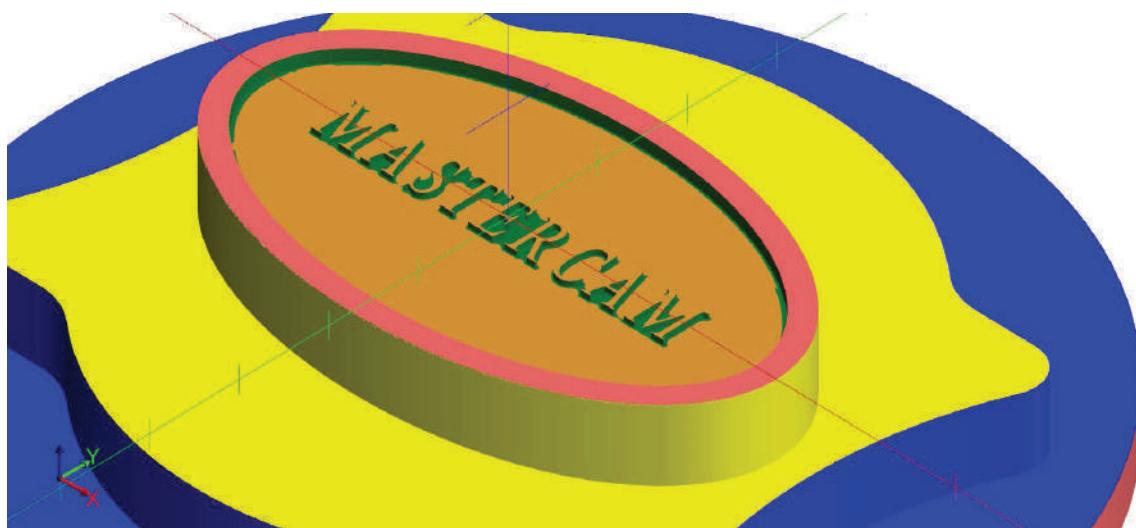
- ♦ To **Backplot** and **Verify** your toolpath, see page 145 to review these procedures.



STEP 14: POCKET REMACHINING

Pocket Remachining is only used with closed chains. It calculates areas where the previous tool could not machine the stock and creates a remachining pocket toolpath to clear the remaining material.

Toolpath Preview:

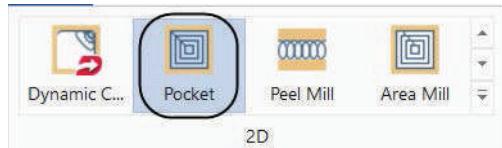




14.1 Select the Geometry

TOOLPATHS

- From the **2D** group, select the **Pocket** icon as shown.



- In the **Chaining** dialog box, make sure that **Solids** selection is enabled and then enable only the **Face** button as shown.





- ♦ [Solid Toolpath Chain: Select faces, edges, and/or loops]: Zoom in using the mouse wheel, and then select the face of the solid as shown in [Figure: 14.1.1](#).

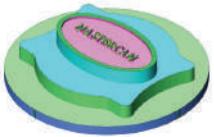


- ♦ [Solid Toolpath Chain: Select faces, edges, and/or loops]: Select all the letter faces as shown in [Figure: 14.1.2](#).



- ♦ Choose the **OK** button to exit the **Chaining** dialog box.





- In the Toolpath Type page, Pocket will be picked.



14.2 Select a 1/32" Flat Endmill from the library and set the Tool Parameters

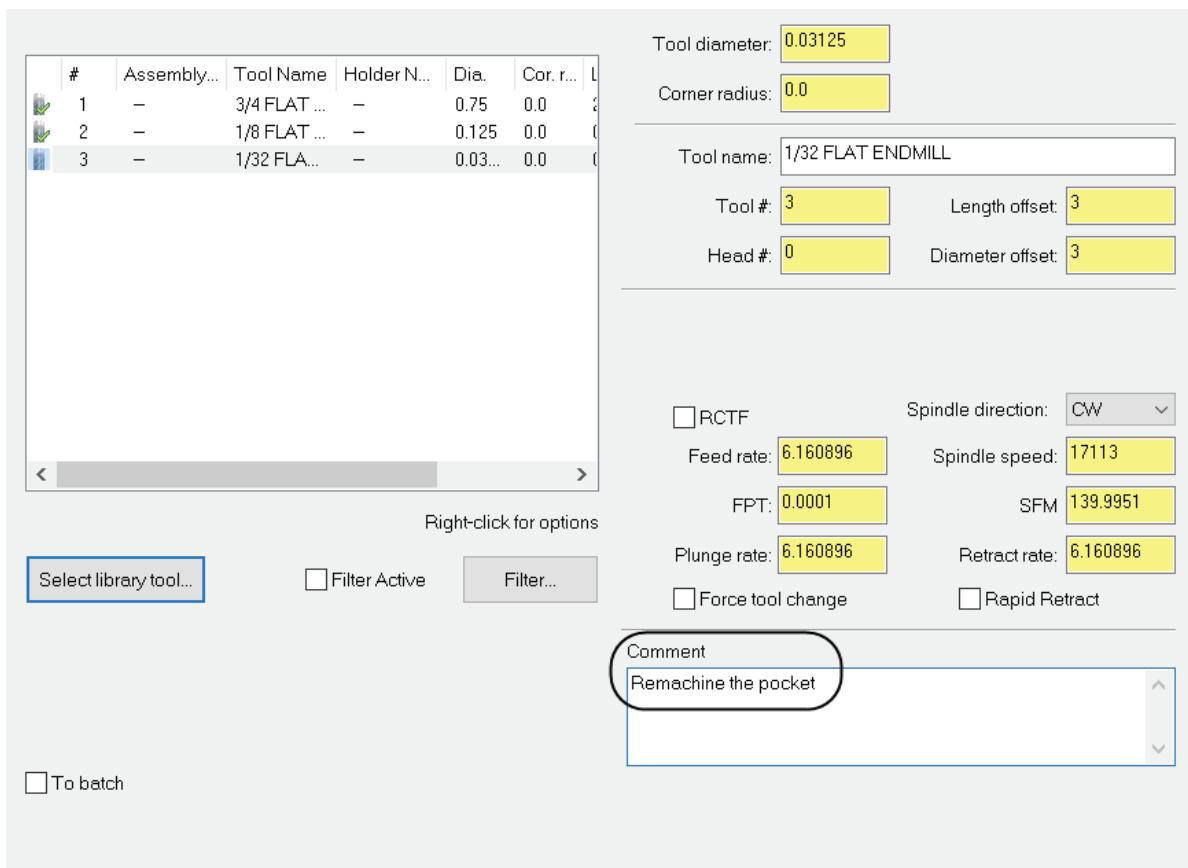
- Select Tool from the Tree View list.

- Click on Select library tool button.
- Select the Filter button.



- Select the None button and then under Tool Types, choose the Flat Endmill Icon.
- Under Tool Diameter, select Equal and input a value of 0.03125. Make all the necessary changes as shown in Figure: 14.2.1.

Figure: 14.2.1





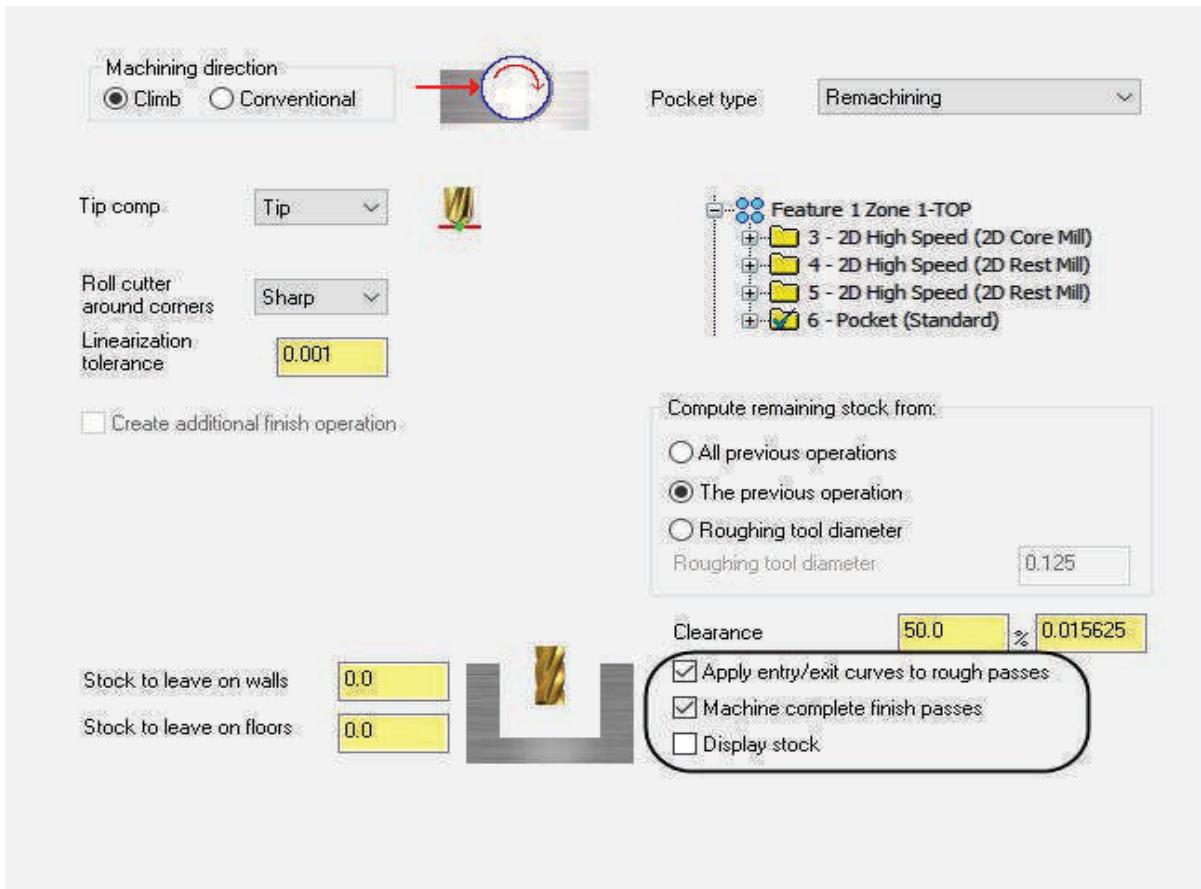
14.3 Set the Cut Parameters

- ♦ Select **Cut Parameters** and change the **Pocket type** to Remachining.

NOTE: The standard pocket removes the material inside of a closed boundary while the remachining pocket removes only the remaining material that a previous toolpath could not clean due to the tool size.

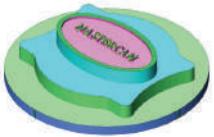
- ♦ Ensure your settings appear as shown in [Figure: 14.3.1](#).

Figure: 14.3.1



Compute remaining stock from The previous operation determines remaining stock for remachining by calculating stock removed during the previous toolpath.

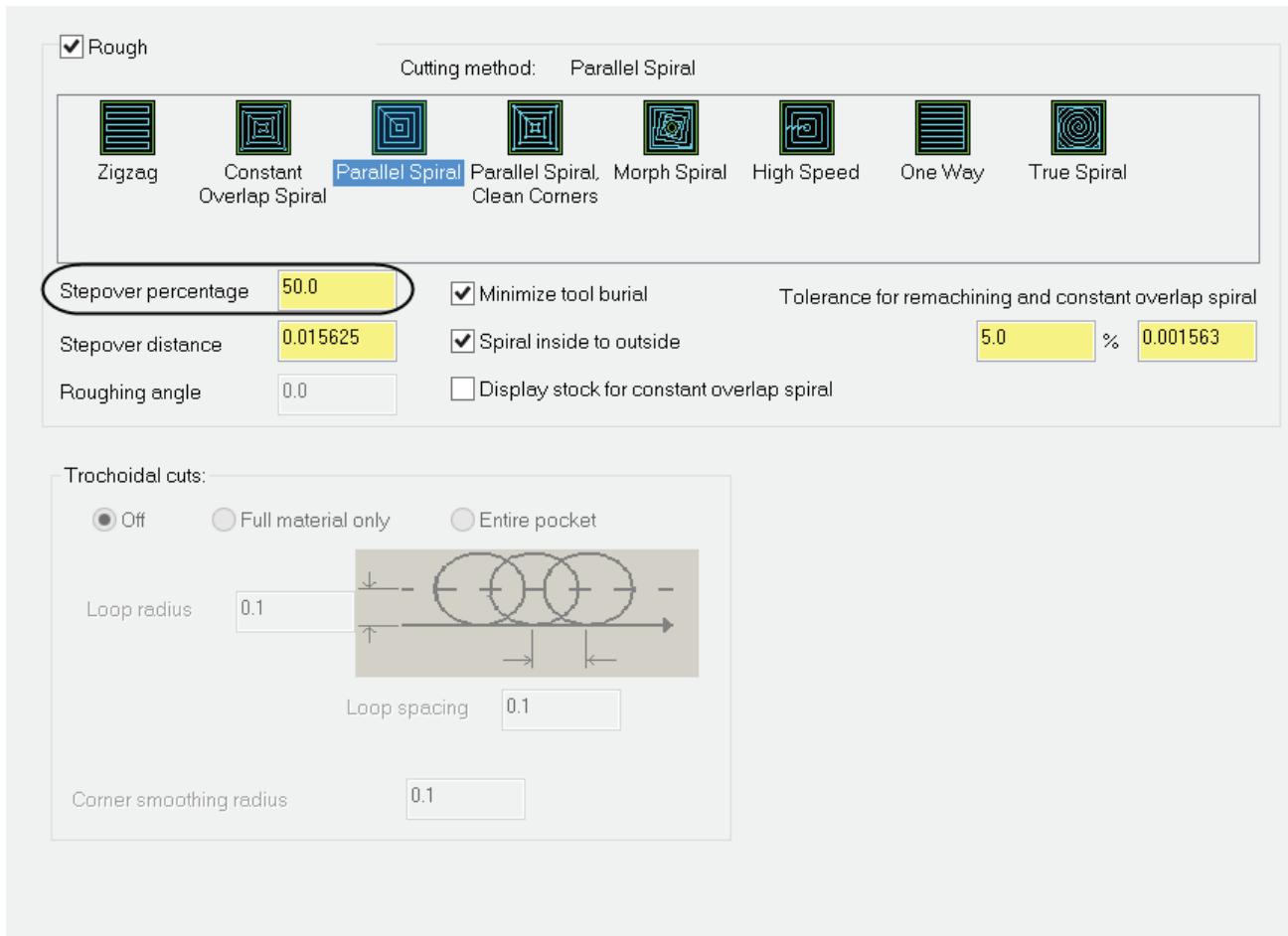
Clearance extends the remachining toolpath at the beginning and end to prevent cusps of material from being left behind.



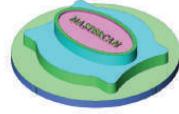
14.4 Set the Roughing Parameters

- Choose **Roughing** and change the **Stepover percentage** to **50.0** as shown in [Figure: 14.4.1](#).

Figure: 14.4.1



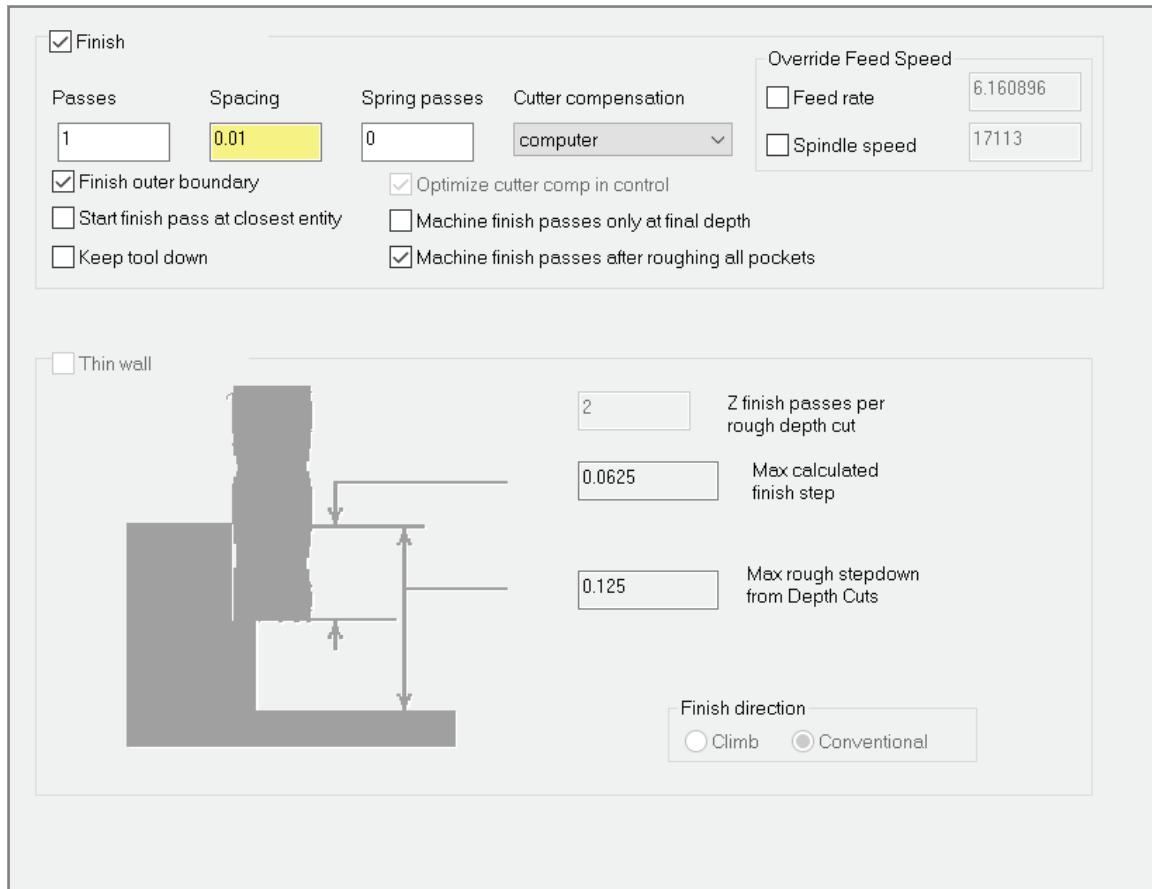
NOTE: The **Cutting method** is defined by the **Pocket type** and cannot be modified for **Remachining**. The **Finishing**, **Lead In/Out** and **Depth Cuts** parameters are the same as the previous toolpath; therefore we do not need to view them.



14.5 Set the Finishing Parameters

- ◆ Choose **Finishing** and ensure that the parameters are as shown in [Figure: 14.5.1](#).

Figure: 14.5.1

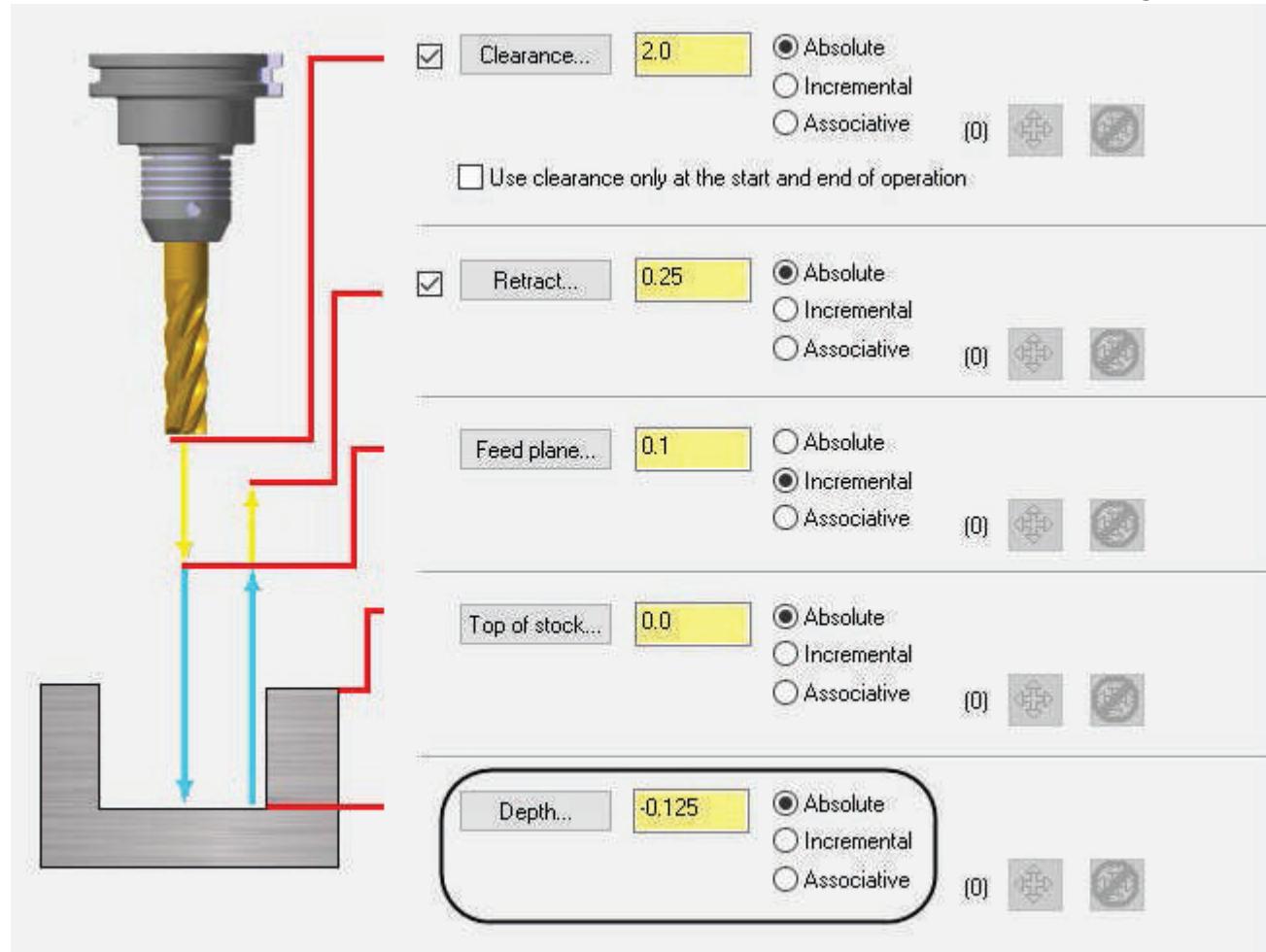




14.6 Set the Linking Parameters

- Choose **Linking Parameters** and set the **Depth** to **-0.125** as shown in [Figure: 14.6.1](#).

Figure: 14.6.1



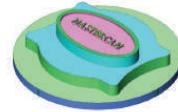
14.7 Preview the Toolpath

- To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

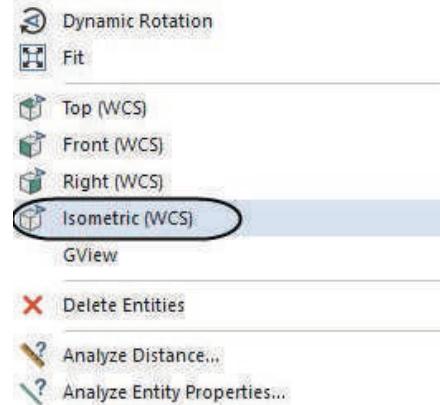


- To hide the dialog box, click on the **Hide dialog** icon as shown.

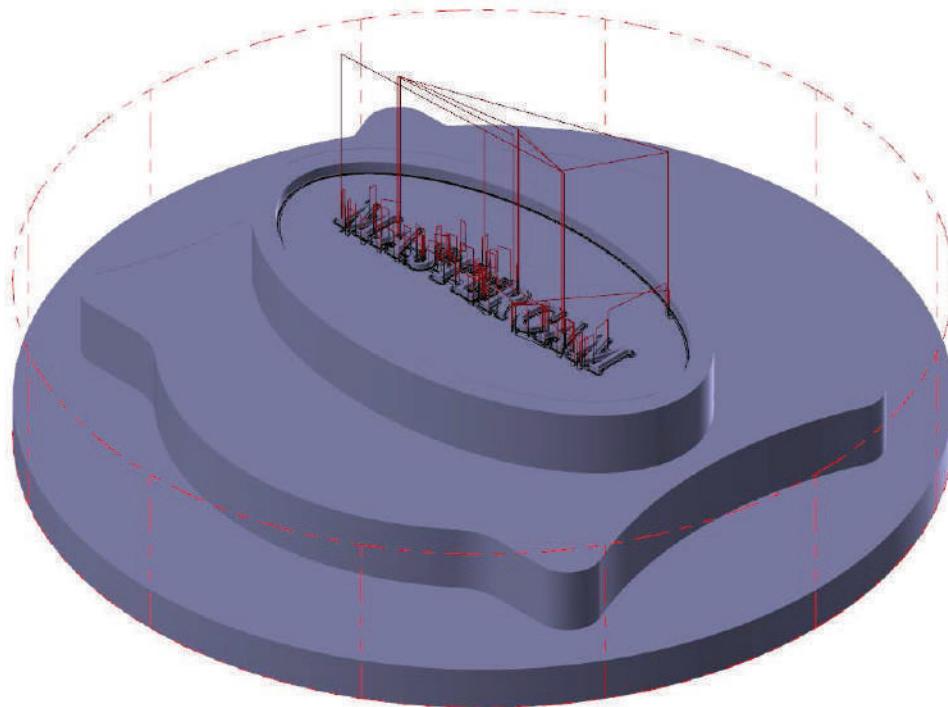




- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.

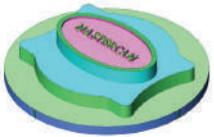


- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

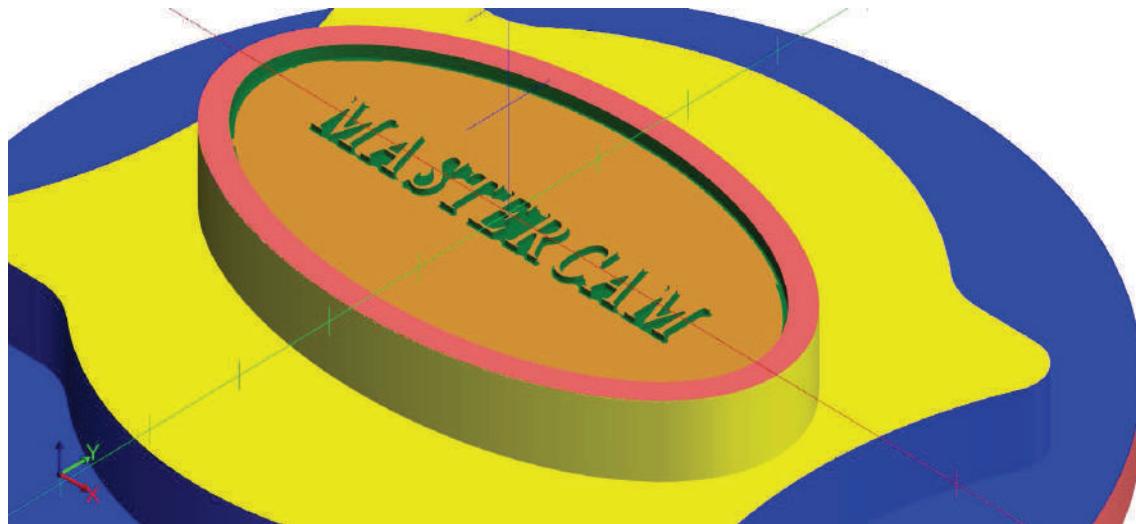
- ♦ Select the **OK** button to exit the **2D Toolpaths - Pocket** toolpath parameters.





14.8 Backplot and Verify the toolpaths

- ♦ To **Backplot** and **Verify** your toolpath, see page 145 to review these procedures.



- ♦ To exit the **Mastercam Simulator**, select the close button as shown.

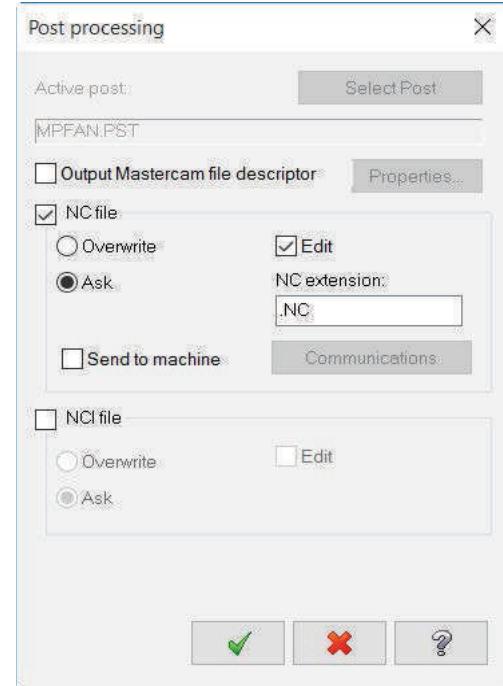




STEP 15: POST THE FILE

- ♦ Ensure all operations are selected. If they are not, use the button **Select all operations** in the **Toolpaths Manager**.
- ♦ Select the **Post selected operations** button from the **Toolpaths Manager**.
- ♦ In the **Post processing** window, make the necessary changes as shown in [Figure: 15.0.1](#).

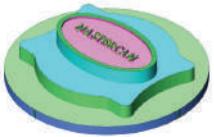
Figure: 15.0.1



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

Edit enabled allows you to automatically launch the default editor.

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



- ♦ A window with **Mastercam Code Expert** will be launched and the NC program will appear as shown in [Figure: 15.0.2](#).

Figure: 15.0.2

The screenshot shows the Mastercam Code Expert interface. The title bar reads "Setup #1.NC - Mastercam 2018 Code Expert". The menu bar includes File, Home, View, Editor, NC Functions, and Utilities. The toolbar contains icons for Insert Block Numbers, Insert Block Skip, Send File, Go To, Remove Block Numbers, Remove Block Skip, Send, Remove Spacing, Remove Comments, Receive, Syncs, Tools, Multi-Stream, and NC Configuration. The main window displays an NC program with lines numbered 1 through 38. The program includes G-codes like G17, G90, G40, G49, G80, G90, G91, G92, G93, G94, G95, G96, G97, G98, G99, M-codes like M100, M110, M120, M130, M140, M150, M160, M170, M180, M190, M200, M210, M220, M230, M240, M250, M260, M270, M280, M290, M300, M310, M320, M330, M340, and T-codes like T1, T2, T3, T4, T5, T6. The code also includes comments such as "(MATERIAL ALUMINUM INCH - 2024)", "(DYNAMIC MILL THE STEPS USING ISLAND FACING.)", and "(DATE=00-MI-YY - 22-02-17 TIME=HH:MM - 14:55)". The status bar at the bottom shows "Ready", "272.43KB", "100%", and zoom controls.

```
Setup #1.NC | Setup #2.NC
1   G17
2   O0004 (SETUP #1)
3   (DATE=00-MI-YY - 22-02-17 TIME=HH:MM - 14:55)
4   (NCX FILE = \\IKISERVER06\BOOGIEDEV\PRODUCTION\BOOKS\2018\TRAINING\TUTORIALS\IT-CAD IMPORT MILL ESSENTIALS TOOLPATHS\MCM FILES\INCH\TUTORIAL TOOLPATH INCH\TUTORIAL_4.INCH.MCAM)
5   (NC FILE = C:\USERS\NINGYUE.MAO\Desktop\SETUP #1.NC)
6   (MATERIAL - ALUMINUM INCH - 2024)
7   ( T1 | 1/2 FLAT ENDMILL | H1 | XY STOCK TO LEAVE - .03 | Z STOCK TO LEAVE - 0. )
8   ( T2 | 3/16 FLAT ENDMILL | H2 | XY STOCK TO LEAVE - .08 | Z STOCK TO LEAVE - 0. )
9   ( T3 | 1/2 SCDTDRILL | H3 )
10  ( T4 | 1/4 DRILL | H4 )
11  ( T5 | 1/2 CHAMFER MILL | H5 )
12  ( T6 | 1/4 CHAMFER MILL | H6 )
13  M100 G20
14  M110 G0 G17 G40 G49 G80 G90
15  ( DYNAMIC MILL THE STEPS USING ISLAND FACING. )
16  M120 T6 M6
17  M130 G0 G90 G54 X-1.6516 Y-1.0003 R0. M12000 M3
18  M140 G43 H1 Z1.
19  M150 Z1
20  M160 G1 Z-.615 F100.
21  M170 X-1.6753 Y-1.7394 E-.6208 F200.
22  M180 X-1.6919 Y-1.7253 Z-.6222
23  M190 X-1.6904 Y-1.7123 E-.6236
24  M200 X-1.7044 Y-1.6958 E-.625
25  M210 G0 X-1.6374 Y-1.5387 I3.4937 J3.0911
26  M220 X-2.3521 Y-.0635 I1.854 J1.4732
27  M230 X-2.314 Y-3.3632 I2.3633 J0.
28  M240 X-2.1037 Y-9.9008 I2.3345 J-.4338
29  M250 X-1.5399 Y1.7388 I2.1421 J-1.0281
30  M260 X0. Y2.3226 I1.5399 J-1.7309
31  M270 X.7273 Y2.2058 I0. J-2.3227
32  M280 X2.2721 Y.4818 I-.7273 J-2.2058
33  M290 X2.3226 Y0. I-2.2721 J-.4818
34  M300 X1.5399 Y-1.7208 I-2.3226 J0.
35  M310 X0. Y-2.3226 I-1.5399 J1.7208
36  M320 X-.7018 Y-2.2226 I0. J2.3226
37  M330 X-1.5404 Y-1.7374 I.695 J2.199
38  M340 X-2.2699 Y-.0638 I1.5549 J1.6736
**
```

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

STEP 16: SAVE THE UPDATED MCAM FILE



REVIEW EXERCISE - STUDENT PRACTICE

IMPORT THE DWG FILE FOR TUTORIAL #3 EXERCISE

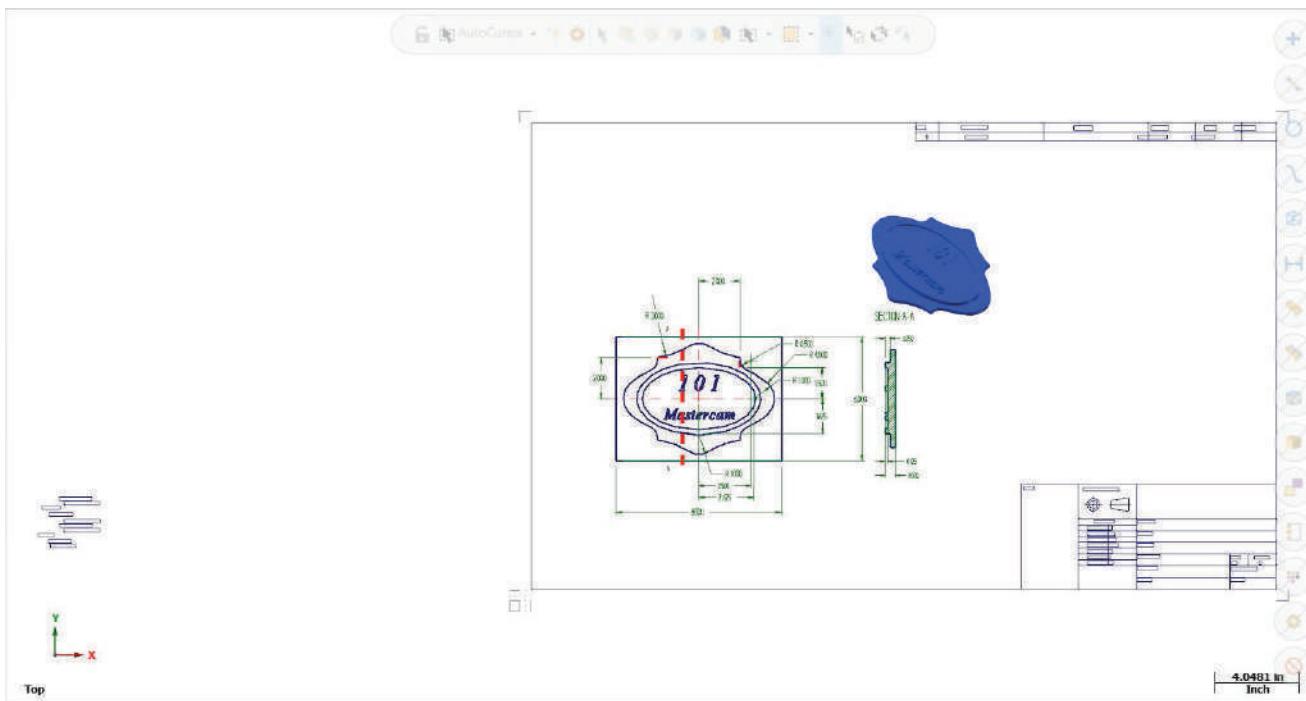
Import the dwg file and manipulate the geometry to prepare it for the toolpath creation.

Download the files from www.emastercam.com/trainingfiles.

Save the file in a preferred location.

Import TUTORIAL_3 EXERCISE.DWG.

- ♦ Use file **Open** and change the file type to AutoCAD (*.dwg; *.dxf...).



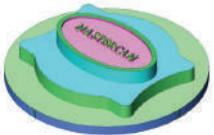
Move the solid geometry to level 100.

- ♦ Create the **Main Level 100** and name it **Geometry**.
- ♦ Use **Select all solid entities** to select the solid.
- ♦ Right mouse click on the graphics window and select the **Change Level** button to move the geometry to **Level 100**.



Make Level 100 the only visible level.

- ♦ In the **Levels Manager**, right mouse click on the **Level 100** and select **All level set off**.



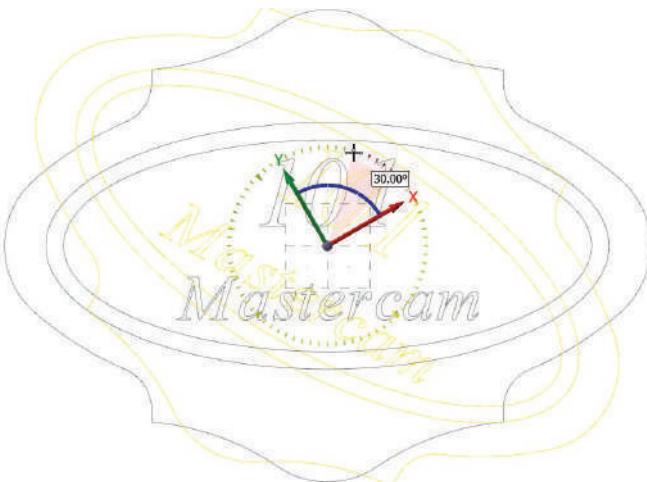
Use WCS to make a new plane and to set as the WCS, Cplane & Tplane.

- ◆ In the Planes Manager, create a new plane and select **From solid face**.
- ◆ Select the face as shown.
- ◆ Accept the first plane.
- ◆ Name the plane "New Plane".
- ◆ Use the equal button to set the **WCS, C**, and **T** planes to the **New Plane**.
- ◆ Select **Top** view to update the plane in the graphics window.

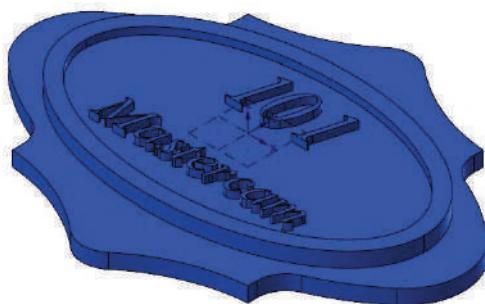


Use Dynamic Transform to rotate the part.

- ◆ Unshade the solid.
- ◆ Select the solid.
- ◆ Move the gnomon to the **Origin**.
- ◆ Leave the mode to move geometry.
- ◆ Select the arc segment between the X and Y axis.
- ◆ Rotate the part in the CCW direction 45 degrees.



- ◆ The geometry should look as shown.



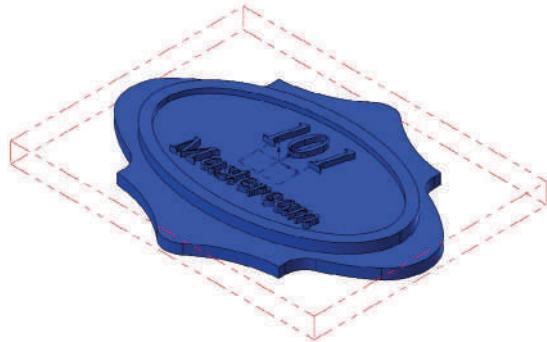


CREATE THE TOOLPATHS FOR TUTORIAL #3 EXERCISE

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

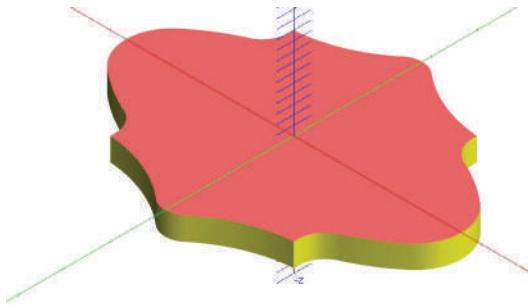
Set the machine properties including the stock setup.

- ◆ Select the **Mill Default**.
- ◆ Use **Bounding box** to determine the size of the stock.
- ◆ Change the sizes in **Bounding Box** to X = 7.5, Y = 5.75.



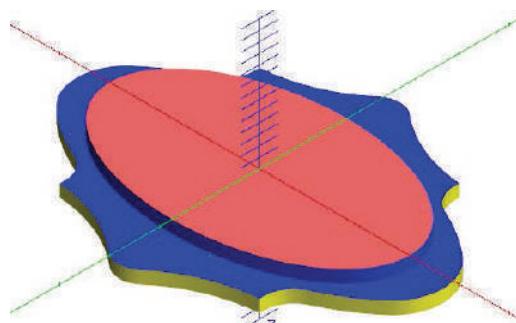
Use 2D HS Dynamic Mill to machine the outside profile.

- ◆ Enable **From outside**.
- ◆ Click on the **Select** button in the **Avoidance regions**.
- ◆ Enable **Solids** selection and **Loops** button only.
- ◆ Select one of the edges at the bottom of the outer profile.
- ◆ Select the **1" Flat Endmill**.
- ◆ Leave the defaults in the **Cut Parameters**.
- ◆ **Stock to leave on walls/floors = 0.0**.
- ◆ Disable **Depth Cuts** if needed.
- ◆ Set your **Entry method**.
- ◆ Enable **Break through** and set a **Breakthrough amount of 0.05**.
- ◆ Set the **Depth = -0.5 (Absolute)**.



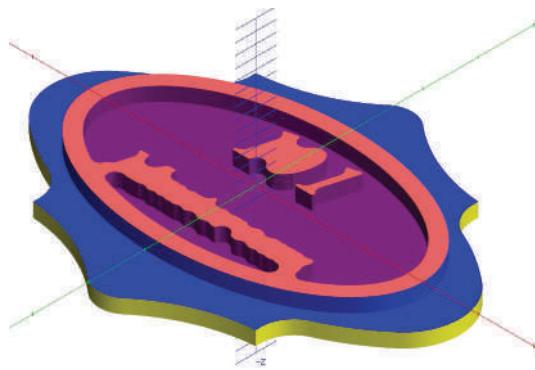
Remove the material at the step using 2D HS Area Mill.

- ◆ In the **Machining regions**, select the outside profile using the same solid selection options.
- ◆ Enable **From outside**.
- ◆ In the **Avoidance regions**, select the elliptical shape as the chain.
- ◆ Use the **1" Flat Endmill**.
- ◆ Enable **Corner Rounding**.
- ◆ **Stock to leave on walls/floors = 0.0**.
- ◆ Use **Depth Cuts** and set **Max rough step = 0.125**.
- ◆ Set the **Entry method**.
- ◆ Disable **Break through** if needed.
- ◆ Set up the **Depth = -0.25 (Absolute)**.



**Pocket out the center using 2D HS Area Mill.**

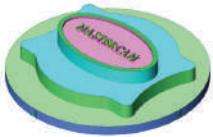
- ◆ In the **Machining regions**, select the inner ellipse.
- ◆ Enable **Stay inside**.
- ◆ In the **Avoidance regions**, enable only **Solid Face** in the **Solid** selection and select the top face of all the letters and figures.
- ◆ Use a **1/4" Flat Endmill**.
- ◆ Enable **Corner Rounding**.
- ◆ **Stock to leave on walls = 0.02**.
- ◆ **Stock to leave on floors = 0.0**.
- ◆ Set the **Entry method**.
- ◆ Use **Depth Cuts** and set **Max rough step = 0.125**.
- ◆ Enable **Use island depths** and **Island facing with Overlap = 25%**.
- ◆ Disable **Break through** if needed.
- ◆ Set the **Depth = -0.125 (Absolute)**.

**Remachine the pocket.**

- ◆ Enable **Loop** option in the **Solid** selection and select the inner ellipse edge.
- ◆ Enable **Face** option in the **Solid** selection and select all the top faces of the letters and figures.
- ◆ Use a **1/16" Flat Endmill**.
- ◆ Change the **Pocket type** to **Remachining**.
- ◆ Enable **The previous operation**.
- ◆ Set the **Stock to leave on walls/floor = 0.0**.
- ◆ Enable **Machine complete finish passes**.
- ◆ Disable **Display stock**.
- ◆ Enable **Depth Cuts** and set a cut depth of **0.0625"**.
- ◆ Set up the **Depth = -0.125 (Absolute)**.



**NOTES:**

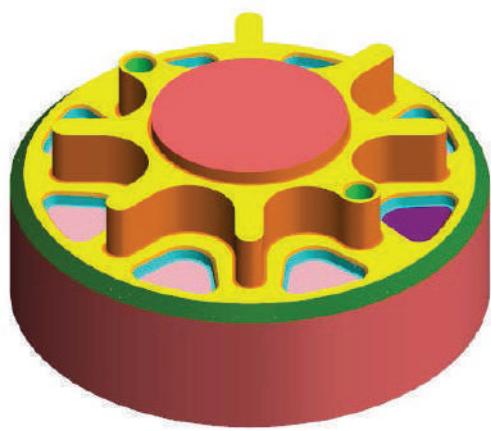


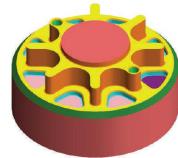
TUTORIAL #3 QUIZ

- ◆ What does Area Mill do?
 - ◆ What does Island Facing do?
 - ◆ What does Pocket Remachining do?



TUTORIAL #4





OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

Import the CAD Model and prepare it to generate Toolpaths:

- ◆ The student will open the Solidworks file in Mastercam.

Create the necessary Toolpaths to machine the part:

- ◆ The student will set up the stock size and the clamping method used. Two setups will be used to machine the part, from the top and then from the bottom.
- ◆ A 2D High Speed Dynamic Mill toolpath will be created to remove the material from the two steps.
- ◆ A Contour toolpath will be created to finish the walls.
- ◆ A 2D High Speed Area Mill toolpath will be created to remove the material inside of one pocket.
- ◆ A Transform-Rotate toolpath will be created to machine the rest of the smaller pocket.
- ◆ A Contour toolpath will be created to finish all the pockets.
- ◆ Two Drill toolpaths will be created to machine the holes.
- ◆ A Contour-Chamfer toolpath will be created to chamfer the edge.
- ◆ A 2D High Speed Dynamic Mill toolpath will be created to remove the material inside of the part from the bottom.
- ◆ A Contour toolpath will be created to finish the walls.

Backplot and Verify the file:

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

Post Process the file to generate the G-code:

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



This tutorial takes approximately two hours to complete.



GEOMETRY CREATION

STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

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

STEP 2: IMPORTING THE SOLIDWORKS FILE GEOMETRY

Mastercam lets you read (import) a variety of CAD file types into the Mastercam database. You can also write (export) Mastercam files to a variety of different file formats.

To import a SolidWorks file into Mastercam, you have to use the Open function and then select SolidWorks files from the type of files list.

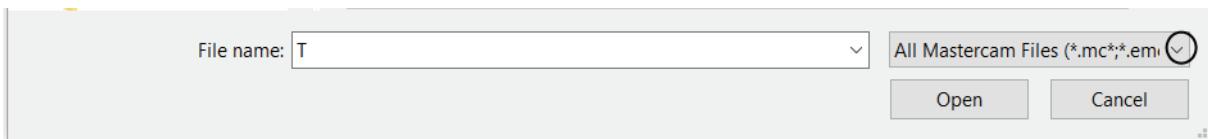
Download the files from www.emastercam.com/trainingfiles.

Save the file at a preferred location.

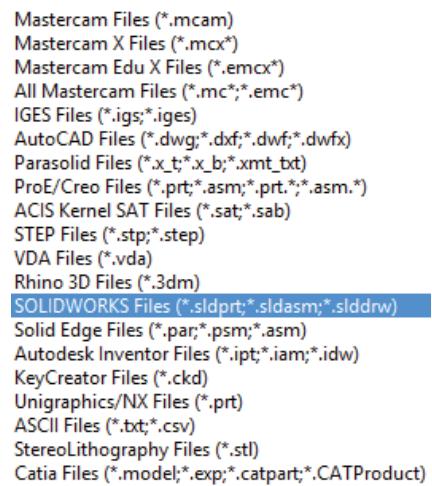
FILE

- ♦ **Open.**

- ♦ In the file name extension, click on the drop down arrow as shown.



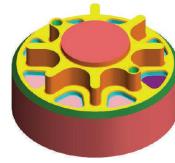
- ♦ From the type of files list, select **SolidWorks Files (*.sldprt;*.sldasm, *.slddrw)** as shown.



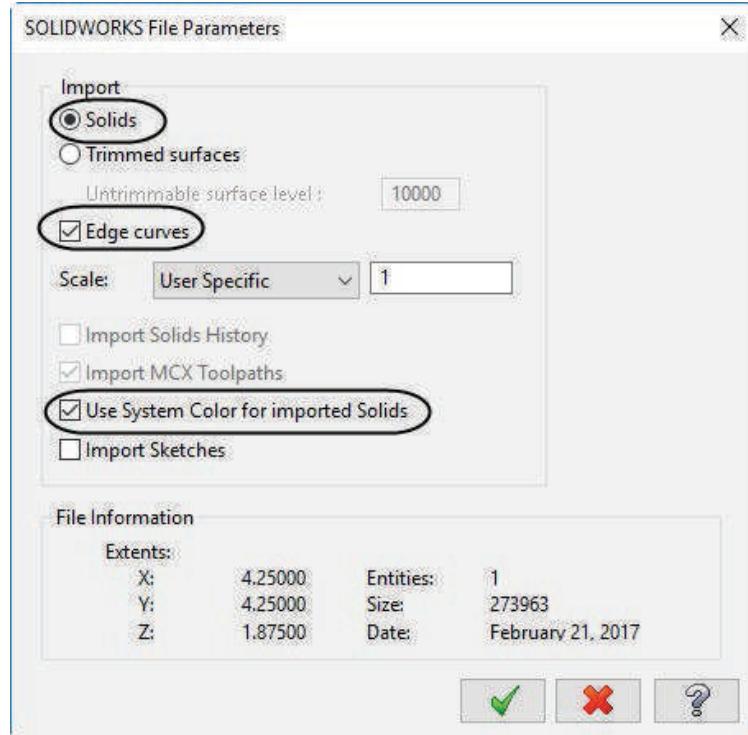
- ♦ Find and select **TUTORIAL #4.sldprt**.

- ♦ Click on the **Options** button.

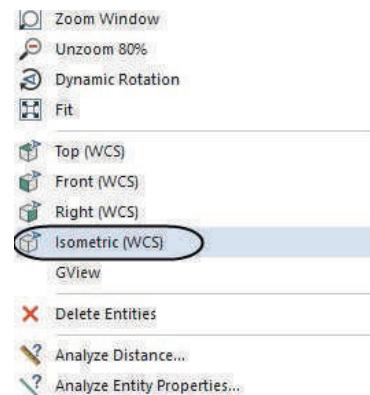




- Leave the **Solids** enabled to import the file as a solid and enable **Edge curves** for Mastercam to automatically create curves at the edges of the solid. To better see the curves, enable also **Use System Color for imported Solids** as shown.



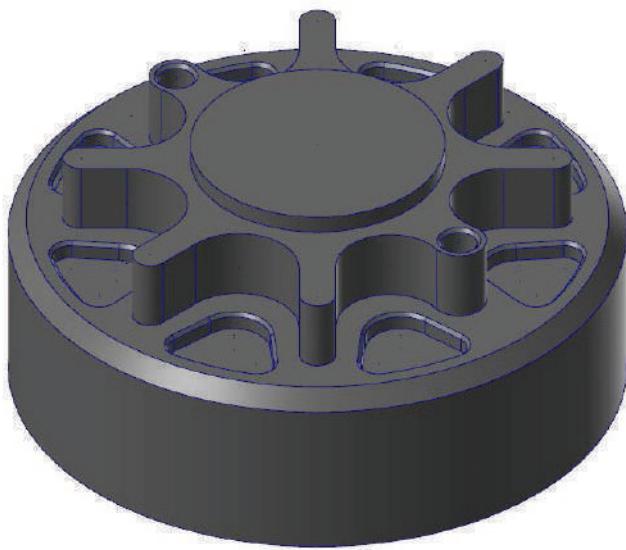
- Select the **OK** button to exit the **SOLIDWORKS File Parameters** dialog box.
- Open** the file.
- Right mouse click in the graphics window and select the **Isometric** view.



- Press **Alt + F1** to fit the geometry in the graphics window.
- Press **Alt + S** to shade the geometry if necessary.



- ♦ The geometry should look as shown.

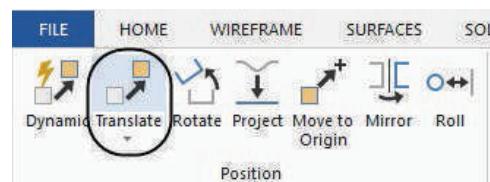


STEP 3: MOVE THE GEOMETRY TO HAVE Z ZERO AT THE TOP OF THE PART

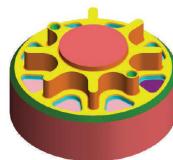
In this step you will use the **Translate** command to move the part.

TRANSFORM

- ♦ From the **Position** group, select **Translate** as shown.

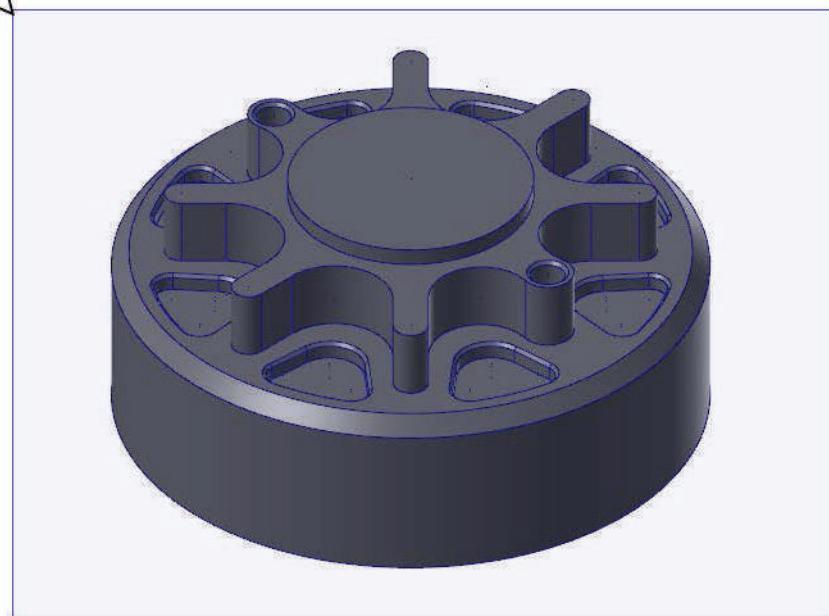


TUTORIAL #4 MOVE THE GEOMETRY TO HAVE Z ZERO AT THE TOP OF THE PART



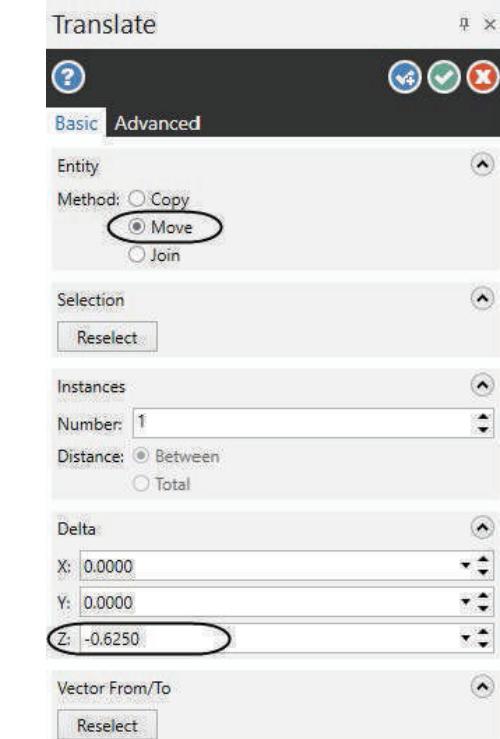
- ♦ [Translate: Select entities to translate]: Make a window around the entire geometry as shown.

Select the upper left corner



Select the lower right corner

- ♦ Click on **End Selection** or press **Enter** to finish the selection.
- ♦ In the **Translate** panel, make sure **Move** is enabled. Enter **-0.625** as **Delta Z** value.

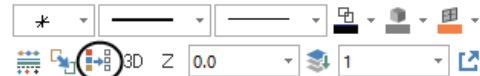




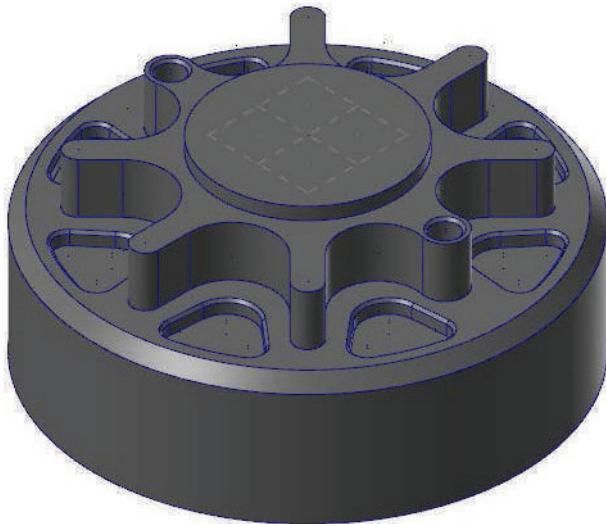
- ♦ Select **OK** to exit the command.



- ♦ Right mouse click in the graphics window and from the **Mini Toolbar**, select the **Clear Colors** icon to reset the colors back to the original colors.



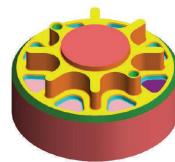
- ♦ Press **Alt + F1** to fit the entire geometry in the graphics window.
- ♦ The geometry should look as shown.



STEP 4: SAVE THE FILE

FILE

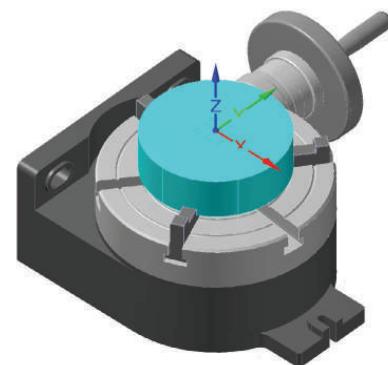
- ♦ **Save As.**
- ♦ File name: "Your Name_4".



TOOLPATH CREATION - SETUP 1

SUGGESTED FIXTURE 1:

NOTE: In order to machine this part, we will have 2 setups and output 2 NC files.

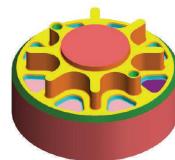




SETUP SHEET 1:

TOOL LIST

	TYPE: Endmill1 Flat DIA OFFSET: 1 HOLDER: Default Holder NUMBER: 1 LENGTH OFFSET: 1 #1 - 0.5000 ENDMILL1 FLAT - 1/2 FLAT ENDMILL	FLUTE LENGTH: 1.0 OVERALL LENGTH: 3.0 CORNER RAD: 0.0 # OF FLUTES: 4
	TYPE: Endmill1 Flat DIA OFFSET: 2 HOLDER: Default Holder NUMBER: 2 LENGTH OFFSET: 2 #2 - 0.1875 ENDMILL1 FLAT - 3/16 FLAT ENDMILL	FLUTE LENGTH: 0.4375 OVERALL LENGTH: 2.375 CORNER RAD: 0.0 # OF FLUTES: 4
	TYPE: Spot Drill DIA OFFSET: 3 HOLDER: Default Holder NUMBER: 3 LENGTH OFFSET: 3 #3 - 0.5000 SPOT DRILL - 1/2 SPOTDRILL	FLUTE LENGTH: 2.0 OVERALL LENGTH: 3.0 CORNER RAD: 0.0 # OF FLUTES: 2
	TYPE: Drill DIA OFFSET: 4 HOLDER: Default Holder NUMBER: 4 LENGTH OFFSET: 4 #4 - 0.2500 DRILL - 1/4 DRILL	FLUTE LENGTH: 2.0 OVERALL LENGTH: 3.0 CORNER RAD: 0.0 # OF FLUTES: 2
	TYPE: Chamfer mill DIA OFFSET: 5 HOLDER: Default Holder NUMBER: 5 LENGTH OFFSET: 5 #5 - 0.5000 CHAMFER MILL - 1/2 CHAMFER MILL	FLUTE LENGTH: 0.75 OVERALL LENGTH: 2.5 CORNER RAD: 0.0 # OF FLUTES: 4
	TYPE: Chamfer mill DIA OFFSET: 6 HOLDER: Default Holder NUMBER: 6 LENGTH OFFSET: 6 #6 - 0.2500 CHAMFER MILL - 1/4 CHAMFER MILL	FLUTE LENGTH: 0.5 OVERALL LENGTH: 2.5 CORNER RAD: 0.0 # OF FLUTES: 4



STEP 5: SELECT THE MACHINE AND SET UP THE STOCK

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

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

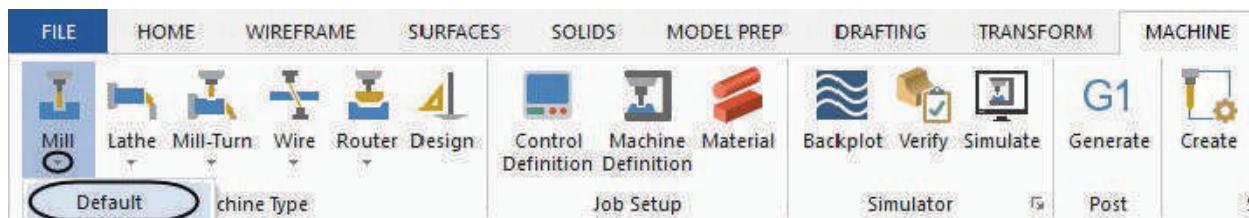
- From the bottom left side of the graphics window, click on the **Toolpaths** tab as shown.



NOTE: Select the **Mill Default** only if there is no **Machine Group** in the **Toolpaths Manager**.

MACHINE

- From the **Machine Type** group, click on the drop down arrow below **Mill** and select the **Default**.



NOTE: Once you select the **Mill Default**, the **ribbon** changes to reflect the toolpaths that could be used with **Mill Default**.

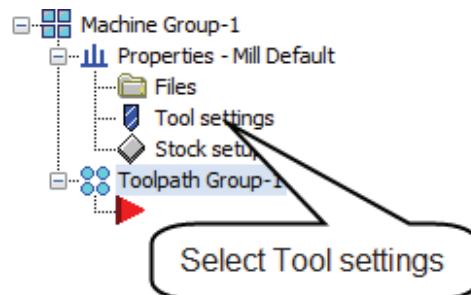
- Select the plus sign in front of **Properties** in the **Toolpaths Manager** to expand the **Toolpaths Group Properties**.



Select the plus sign

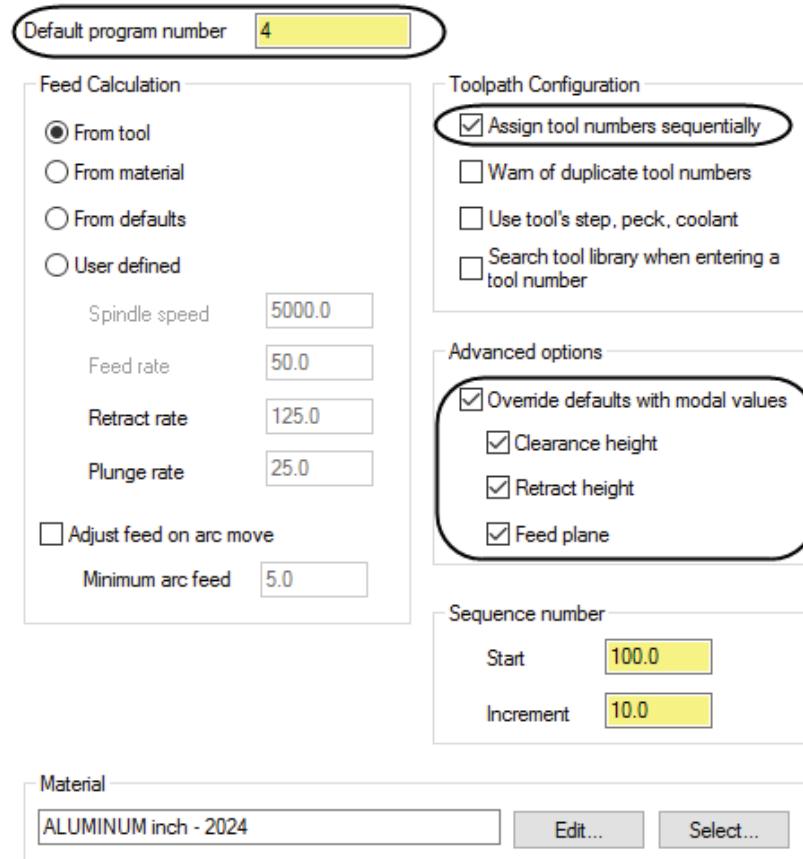


- ♦ Select **Tool settings** to set the tool parameters.



- ♦ Change the parameters to match the screen shot as shown in [Figure: 5.0.1](#).

Figure: 5.0.1



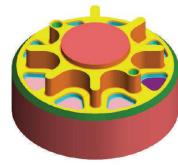
Default program number is used to enter a number if your machine tool requires a number for a program name.

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

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

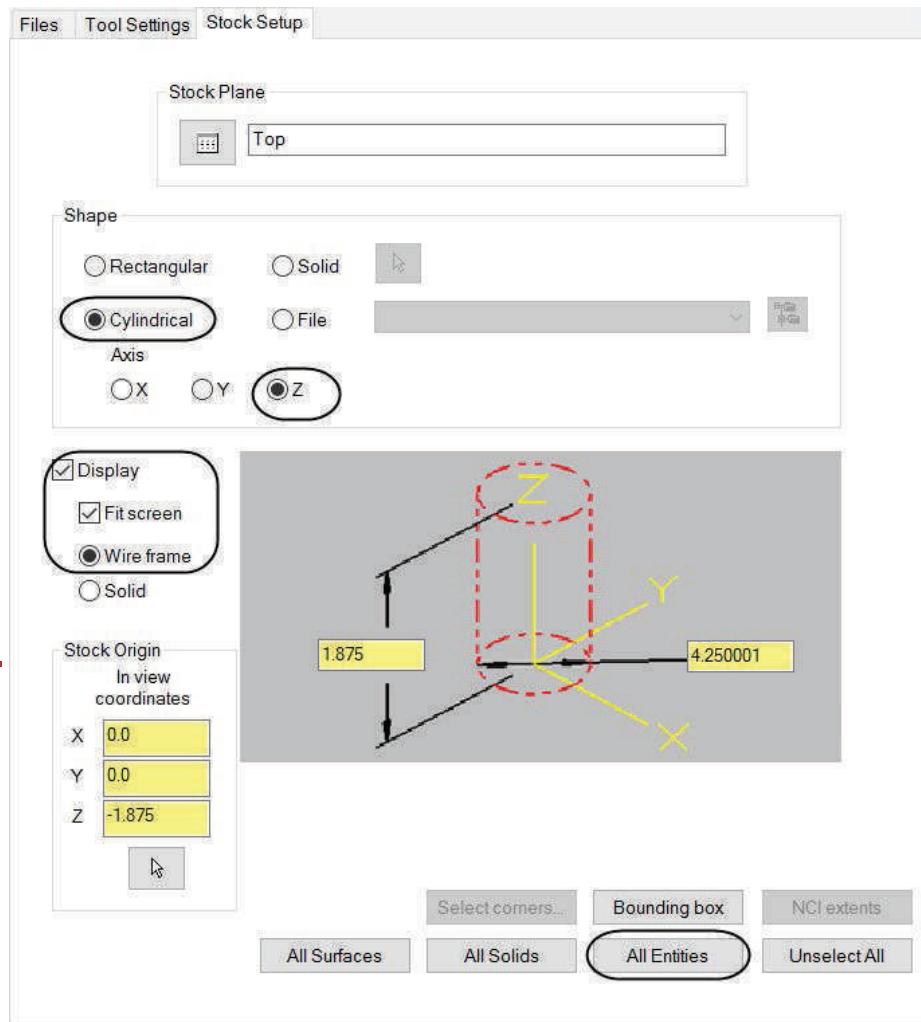
Override defaults with modal values enables the system to keep the values that you enter.

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



- ♦ Select the **Stock Setup** tab to define the stock.
- ♦ Pick the **Cylindrical** shape option and select the **Z** axis.
- ♦ Pick the **All Entities** button to define the stock size as shown in [Figure: 5.0.2](#).

Figure: 5.0.2



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

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

NOTE: The stock model that you create is used when viewing the file or the toolpaths, during backplot, or while verifying toolpaths.

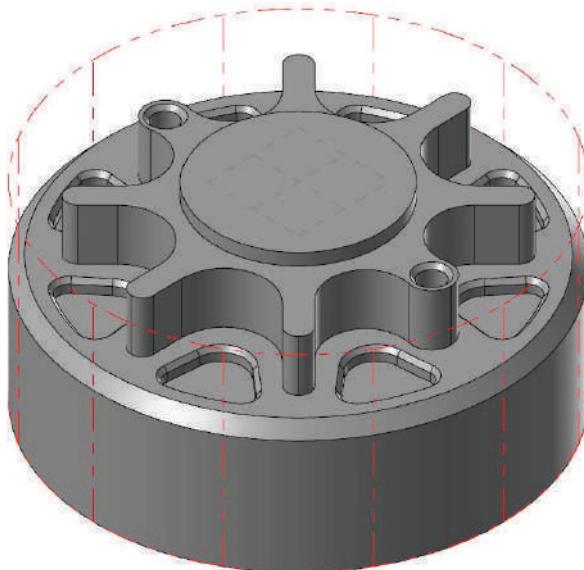
- ♦ Select the **OK** button to exit **Machine Group Properties**.





- The stock model will appear as shown in [Figure: 5.0.3.](#)

Figure: 5.0.3



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

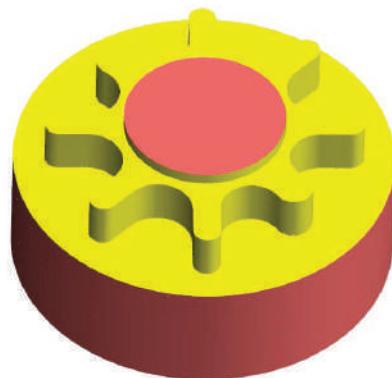
There will not be a facing toolpath because the stock is already to size.

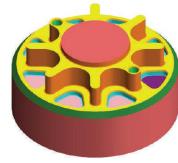
STEP 6: 2D HIGH SPEED DYNAMIC MILL

2D High Speed Dynamic Mill utilizes the entire flute length of cutting tools to produce the smoothest, most efficient tool motion for high speed pocketing. The toolpath supports a custom entry method and many others. Micro lifts further refine the dynamic milling motion and avoid excessive heat build up. Custom feeds and speeds optimize and generate safe tool motion. Dynamic Mill machines pockets using one or more chains to drive the toolpath. The outside chain contains the toolpath; all inside chains are considered islands.

In this step you will machine the three steps of the part using **Island Facing**. You will have to make the wireframe geometry from Level 1 visible.

Toolpath Preview:





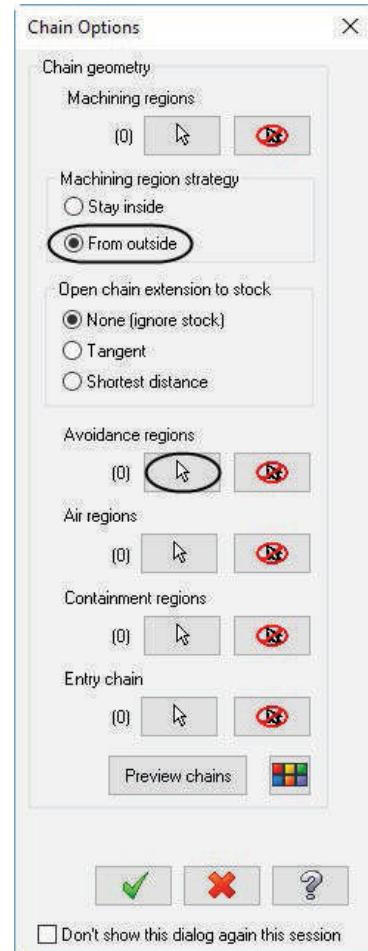
6.1 Chain Selection

TOOLPATHS

- From the **2D** group, select the **Dynamic Mill** icon.



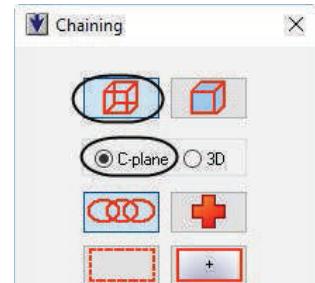
- In the **Chain Options**, select **From outside** as the **Machining region strategy**. Choose **Select avoidance regions** as shown to chain the geometry for the islands.



NOTE: If you do not select a Machining region, the stock is used as the machining region.

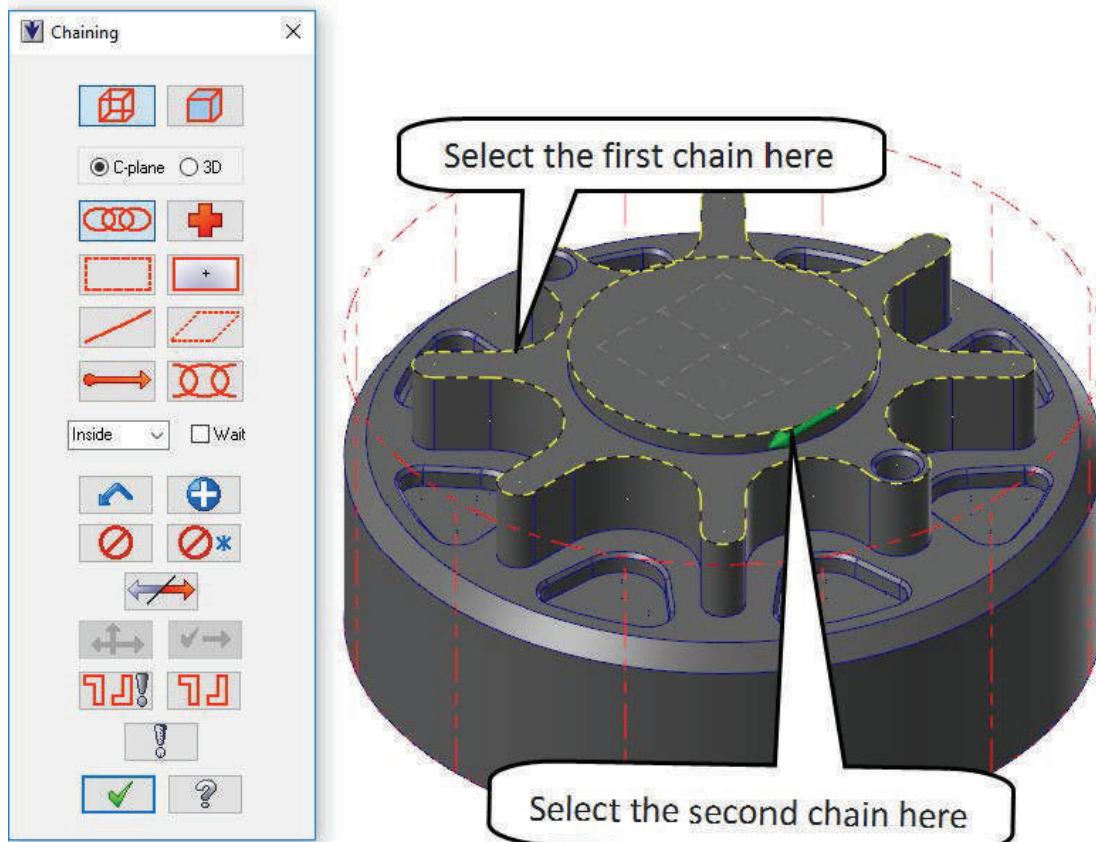


- In the Chaining dialog box, ensure the Chaining dialog box is set to **Wireframe** selection and enable C-plane as shown.



- Pick the chains as shown in [Figure: 6.1.1](#).

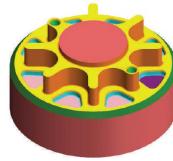
Figure: 6.1.1



- Select the **OK** button to exit the Chaining dialog box.
- Select the **OK** button to exit the **Chain Options** dialog box.
- In the **Toolpath Type** page, **Dynamic Mill** will be already selected as shown in [Figure: 6.1.2](#).

Figure: 6.1.2

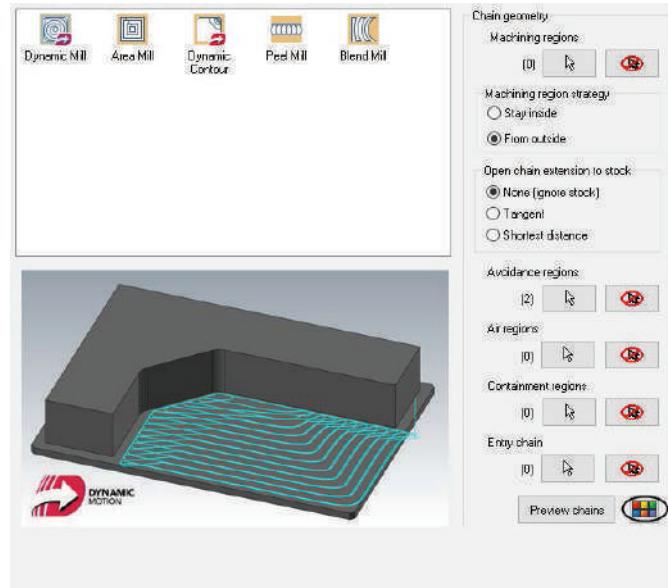




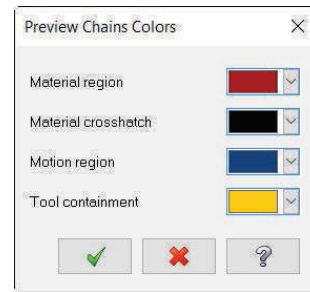
6.2 Preview Chains

The **Preview Chains** function is intended to give the user a quick visual representation of how Mastercam sees the various pieces of geometry that have been selected, how they interact with one another and a general overview of how the toolpath will be calculated with the selections presently made.

- ♦ Click on the **Color** icon to see the legend for **Preview chains** as shown.



- ♦ The **Preview Chains Colors** dialog box should look as shown.



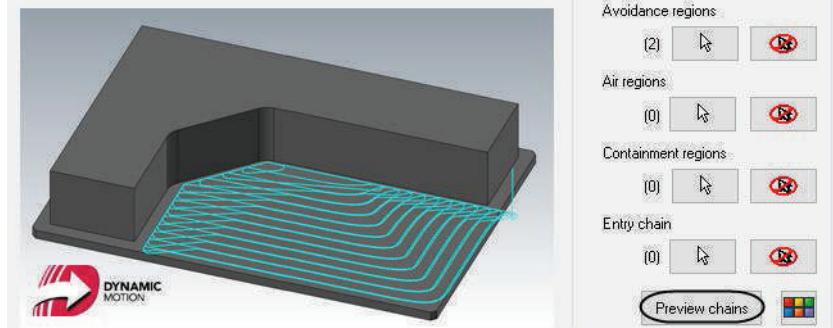
The **Material region** and **Material crosshatch** are the two colors that are used to define the material to be cut. The default colors are red for the background and black for the crosshatch.

The **Motion region** displays the area that Mastercam is making available to the toolpath for motion if it needs it. The color to represent it is dark blue. The primary reason for the display of the entire available (but not necessarily used) Motion region is to help the user visualize how the tool may move near or interact with any adjacent geometry.

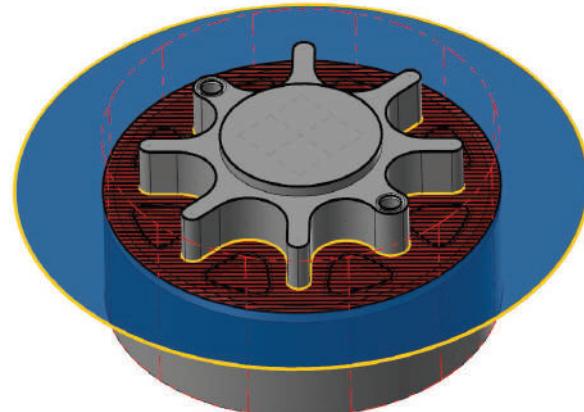
The **Tool containment** is what you have selected as the Containment region in the chain geometry. If you have not selected a containment region, it will default to the outside of the Motion region since that is currently the default area the toolpath is being contained to. The color used to represent the Tool containment is yellow.



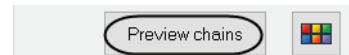
- ♦ Select the **OK** button to exit **Preview Chains Colors**.
- ♦ Select the **Preview chains** button as shown.



- ♦ Select the **Hide dialog** button to see the preview in the graphics window.
- ♦ The **Preview chains** should look as shown.



- ♦ Press **Esc** key to return to the toolpath parameters.
- ♦ Click on the **Preview chains** button again to clear the Preview chains display.

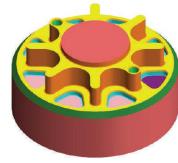


6.3 Select a 1/2" Flat Endmill from the library and set the Tool Parameters

- ♦ Select **Tool** from the **Tree View** list.
- ♦ Click on the **Select library tool** button.
- ♦ Select the **Filter** button.

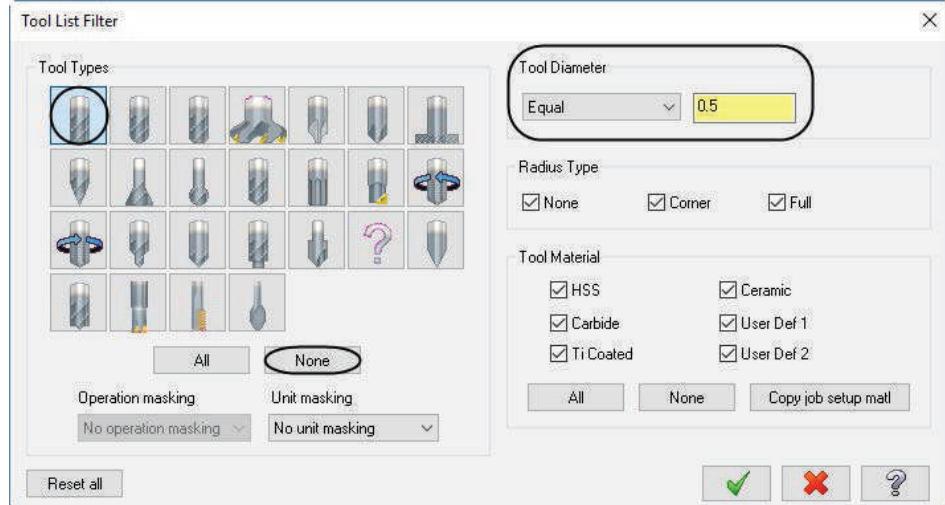
Select library tool...





- ♦ Select the **None** button and then under **Tool Types**, choose the **Flat Endmill** icon.
- ♦ Under **Tool Diameter**, pick **Equal** and input a value of **0.5** as shown in [Figure: 6.3.1](#).

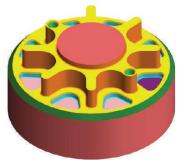
Figure: 6.3.1



- ♦ Select the **OK** button to exit the **Tool List Filter**.
- ♦ In the **Tool Selection** panel you should only see a **1/2" Flat Endmill**.

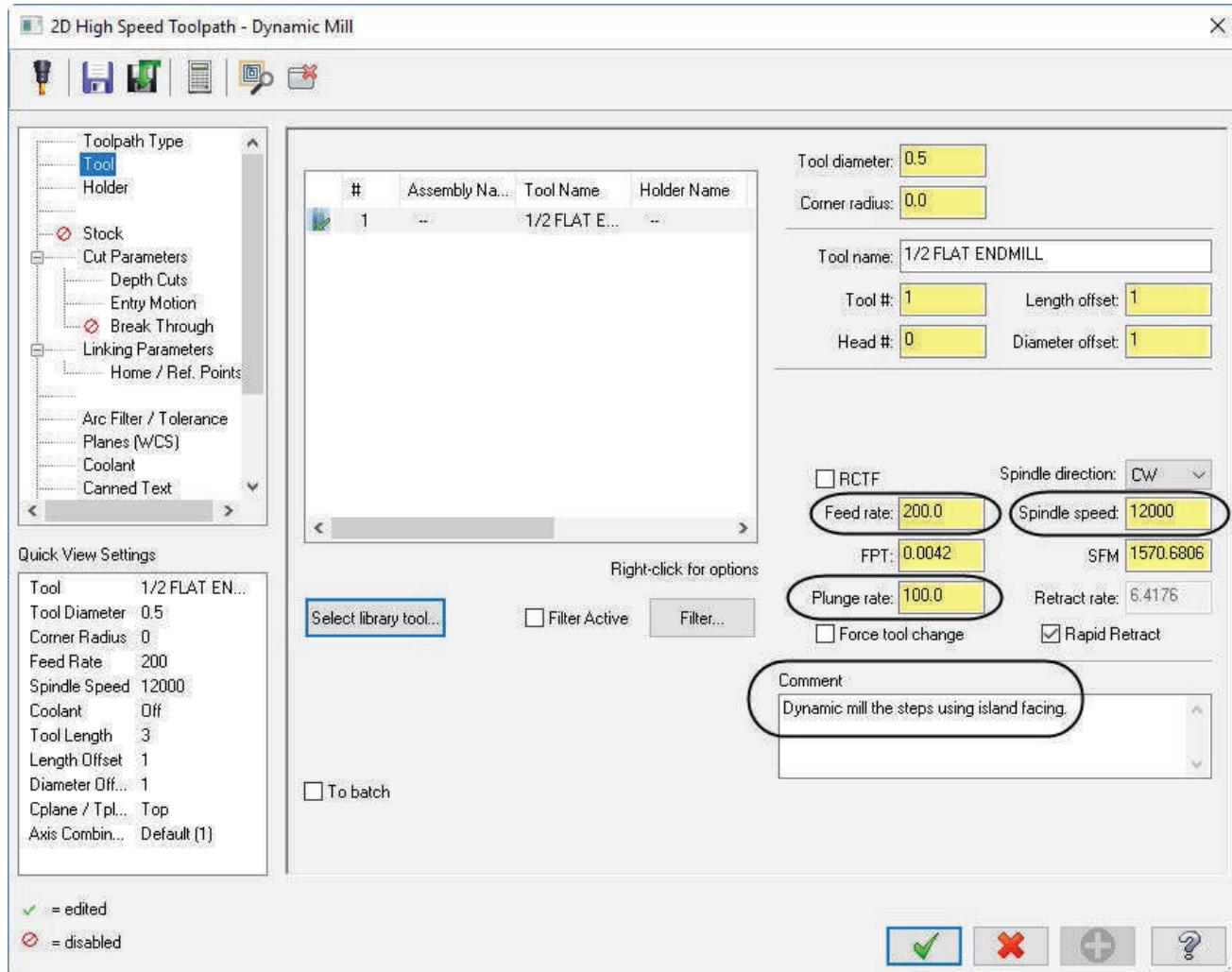
#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	Length	# Flut...	Type	Rad...
290	-	1/2 FLAT ...	-	0.5	0.0	1.0	4	End...	None

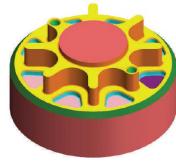
- ♦ Select the **1/2" Flat Endmill** in the **Tool Selection** page and then select the **OK** button to exit.



- ♦ Make all the necessary changes as shown in [Figure: 6.3.2.](#)

Figure: 6.3.2

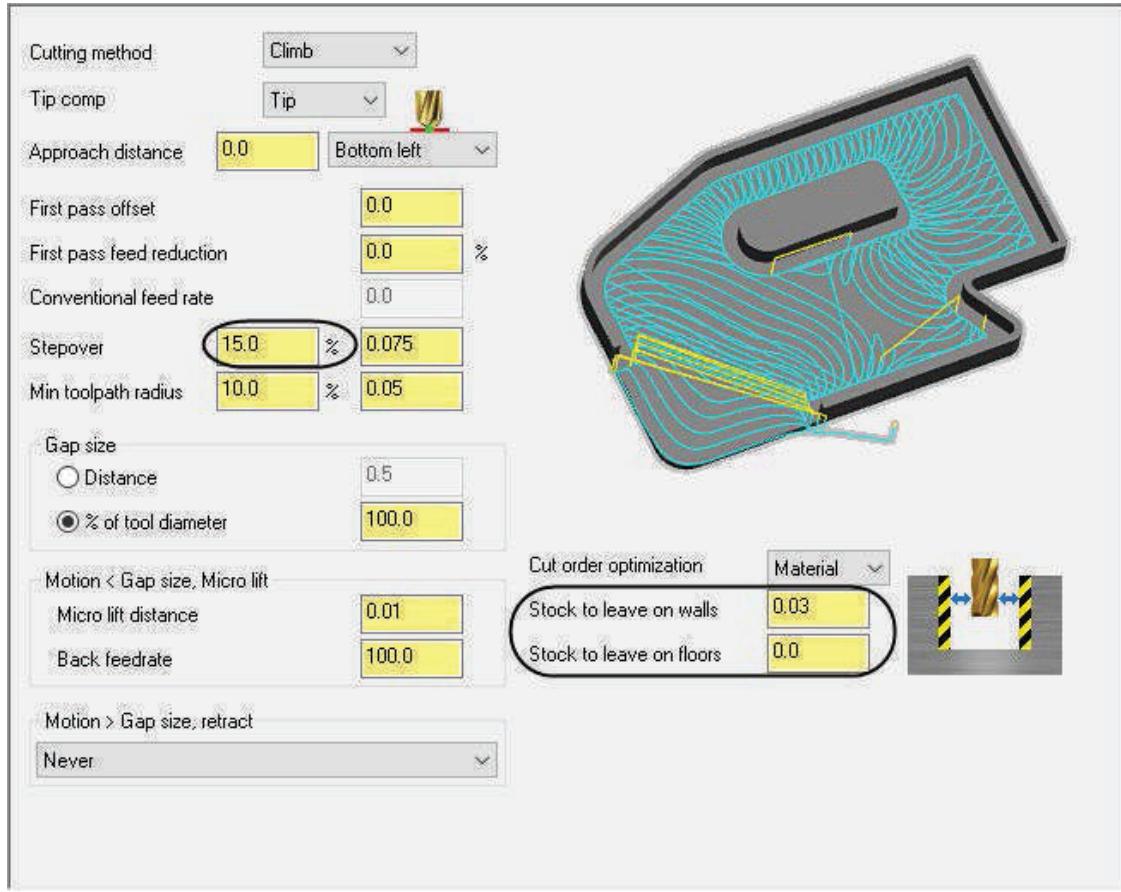




6.4 Set the Cut Parameters

- From the Tree View list, select **Cut Parameters**.
- Change the settings as shown in [Figure: 6.4.1](#).

Figure: 6.4.1



Stepover sets the distance between cutting passes in the X and Y axes.

Toolpath radius reduces sharp corner motion between cut passes.

Micro lift distance enters the distance the tool lifts off the part on the back moves. Microlifts are slight lifts that help clear chips and minimize excessive tool heating.

Back feedrate controls the speed of the backfeed movement of the tool.

Motion > Gap Size, retract controls retracts in the toolpath when making a non-cutting move within an area where the tool can be kept down or microlifted.

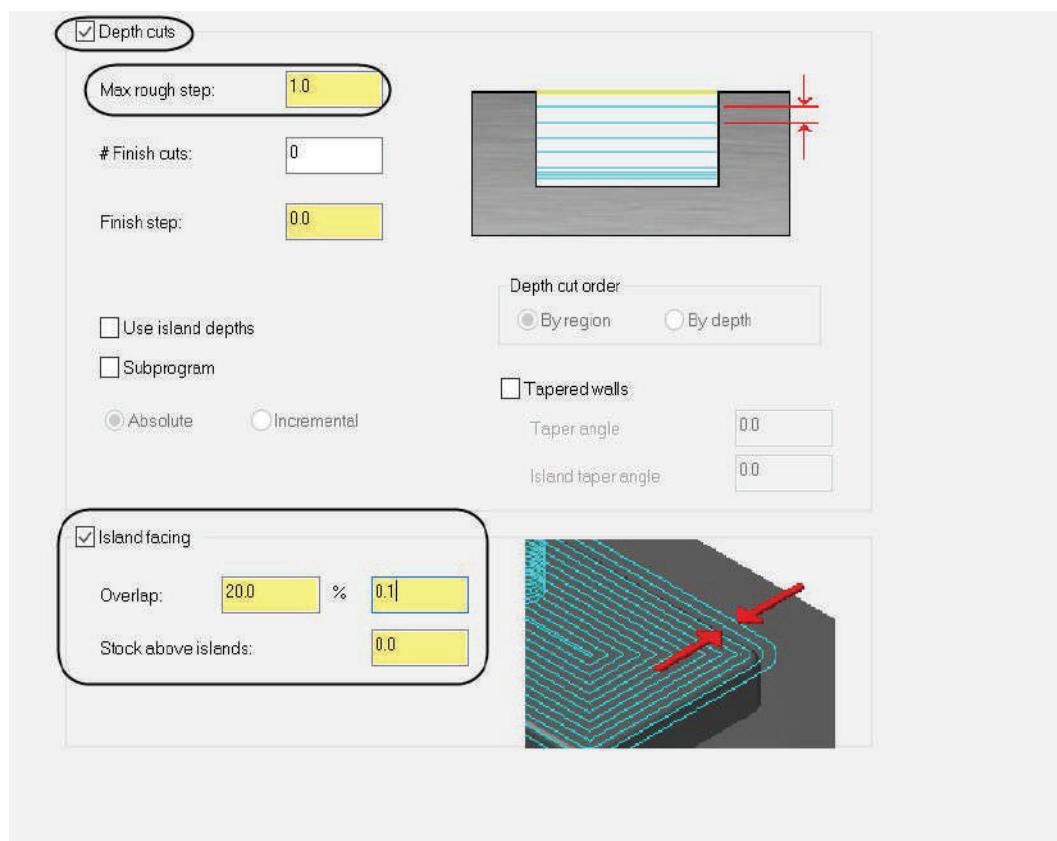
Cut order optimization defines the cut order Mastercam applies to different cutting passes in the dynamic mill toolpath.

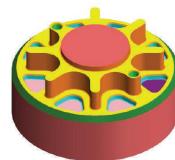
Stock to leave on walls sets the stock left on the walls that has to be removed by another operation.



6.5 Enable Island Facing

- ♦ Select **Depth Cuts** from the **Tree View List**.
- ♦ Enable **Depth cuts** and set **Max rough step** to **1** as shown.
- ♦ Enable **Island facing** and set **Overlap** to **20%** as shown.





6.6 Set the Entry Motion

Entry motion configures an entry method for the dynamic mill toolpath which determines not only how and where the tool enters the part, but the cutting method/machining strategy used by the toolpath.

- From the **Tree view List**, select the **Entry Motion**.
- Change the **Entry method** to **Profile** as shown in [Figure: 6.6.1](#).

Figure: 6.6.1



Entry method set to **Profile** creates a boundary based on the shape of the selected chain and uses the tool to ramp into the part. The slot is cleared by taking lighter cuts in the Z axis until the tool reaches the full depth.

Z clearance adds an extra height used in the ramping motion down from a top profile. It ensures that the tool has fully slowed down from rapid speeds before touching the material.

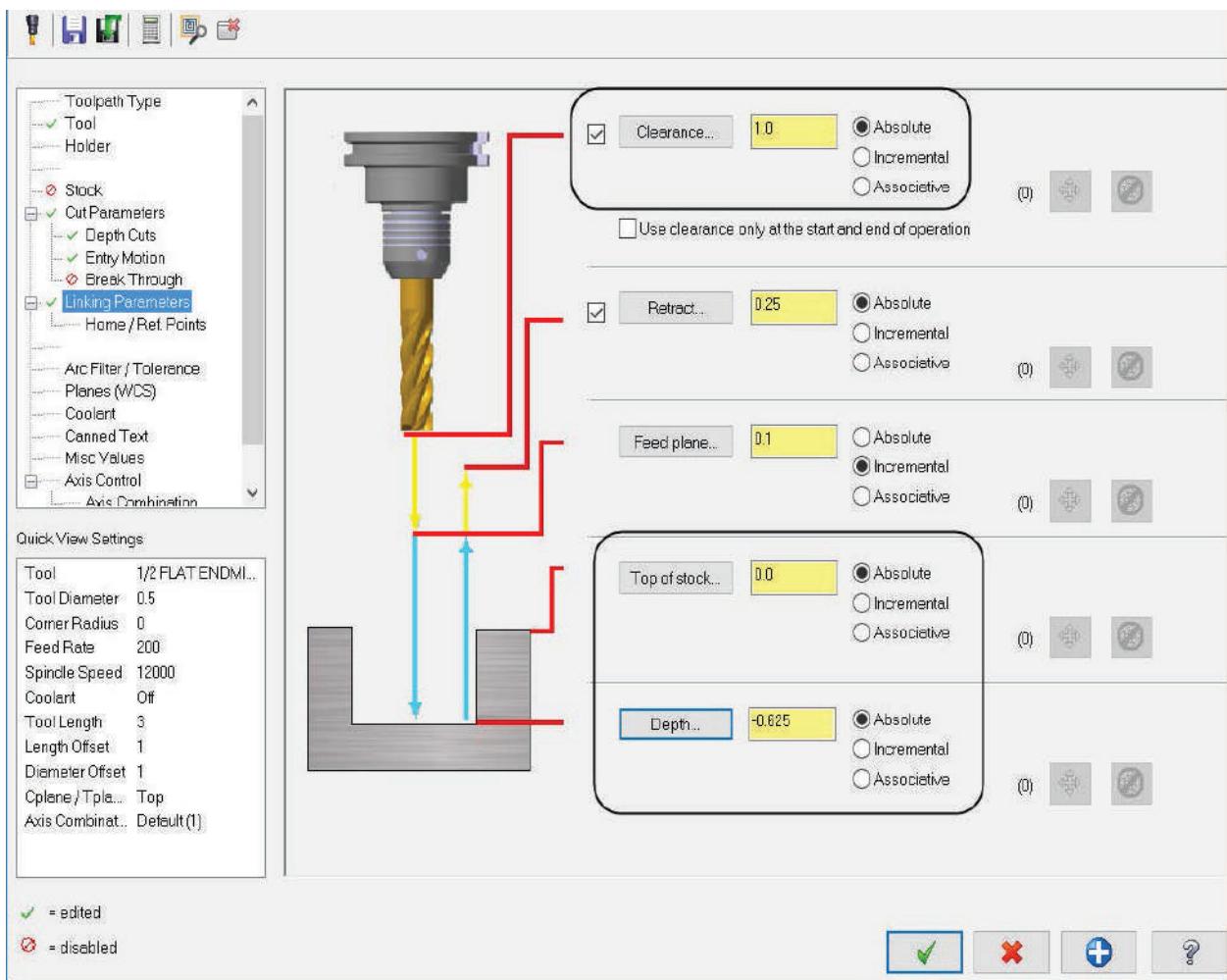
Plunge angle sets the angle of descent for the entry move and determines the pitch.

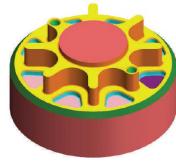


6.7 Set the Linking Parameters

- ♦ Select **Linking Parameters** and enable **Clearance**, change the **Clearance** value to **1.0**. Change the **Top of Stock** value to **0.0, Absolute** and the **Depth** to **-0.625, Absolute** as shown in [Figure: 6.7.1](#).

Figure: 6.7.1



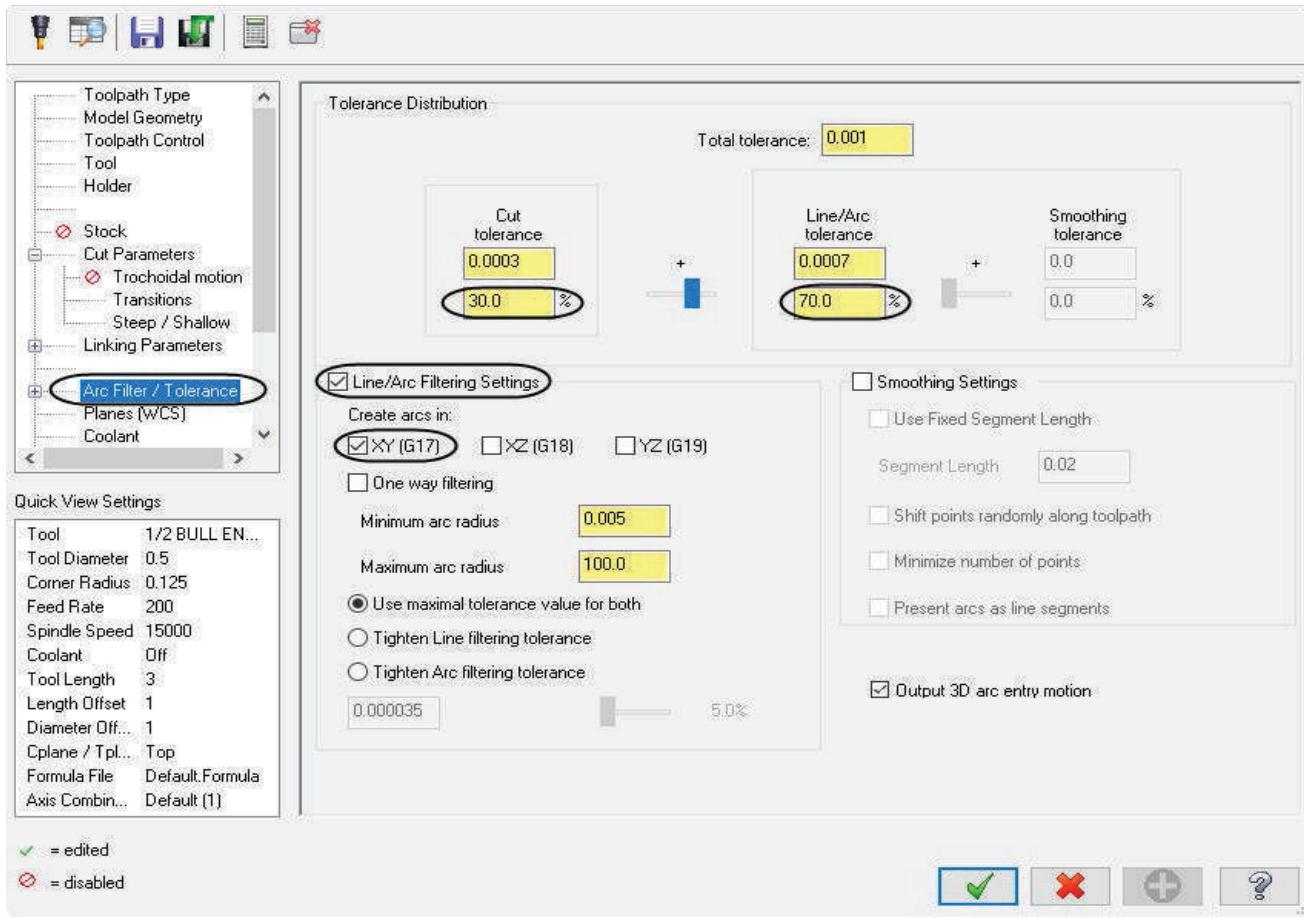


6.8 Set the Arc Filter/Tolerance parameters

Use this page to control the toolpath tolerances. When you activate line and arc filtering, Mastercam replaces very small moves within the defined line/arc filter tolerance with larger ones wherever possible.

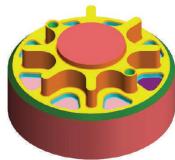
- From the Tree view List, select the **Arc Filter/Tolerance** and enable **Line/Arc Filtering Settings**.
- Set the **Tolerance Distribution** for **Cut tolerance** to **30%** and the **Line/Arc tolerance** to **70%** as shown in [Figure: 6.8.1](#).

Figure: 6.8.1



Line/Arc Filtering Settings replaces very small moves within the defined line/arc filter tolerance with larger ones wherever possible.

Line/arc filtering significantly reduces the number of NC blocks in the post, and may improve the machined precision of the part as well as its surface quality.



6.9 Preview the Toolpath

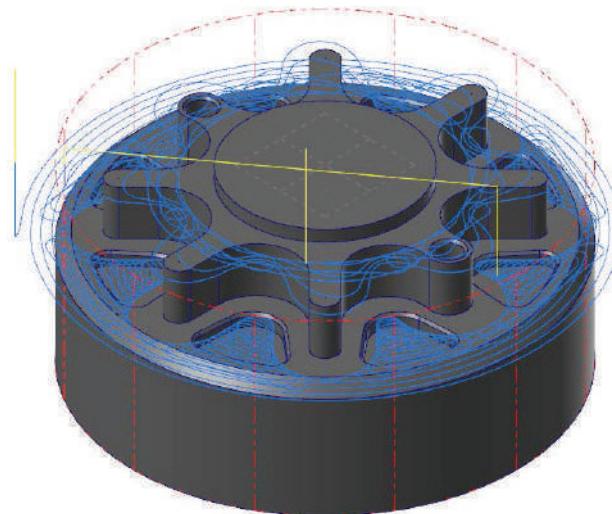
- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.
- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.

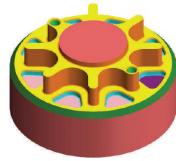


- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to generate the toolpath.





STEP 7: BACKPLOT THE TOOLPATHS

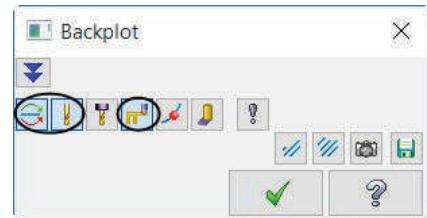
- ♦ Make sure that the toolpaths are selected (signified by the green check mark on the folder icon). If the operation is not selected, choose the **Select all operations** icon.



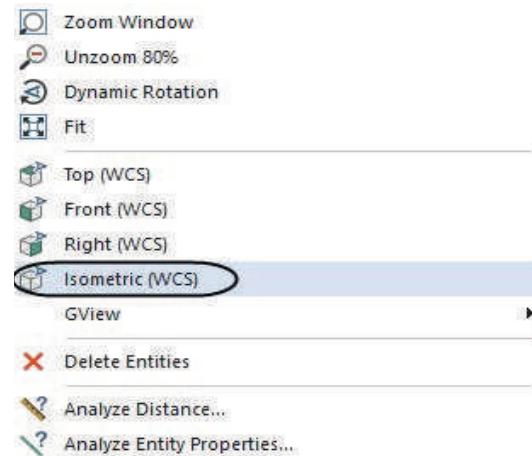
- ♦ Select the **Backplot selected operations** button.



- ♦ In the **Backplot** panel, enable **Display with color codes**, **Display tool** and **Display rapid moves** icons as shown.



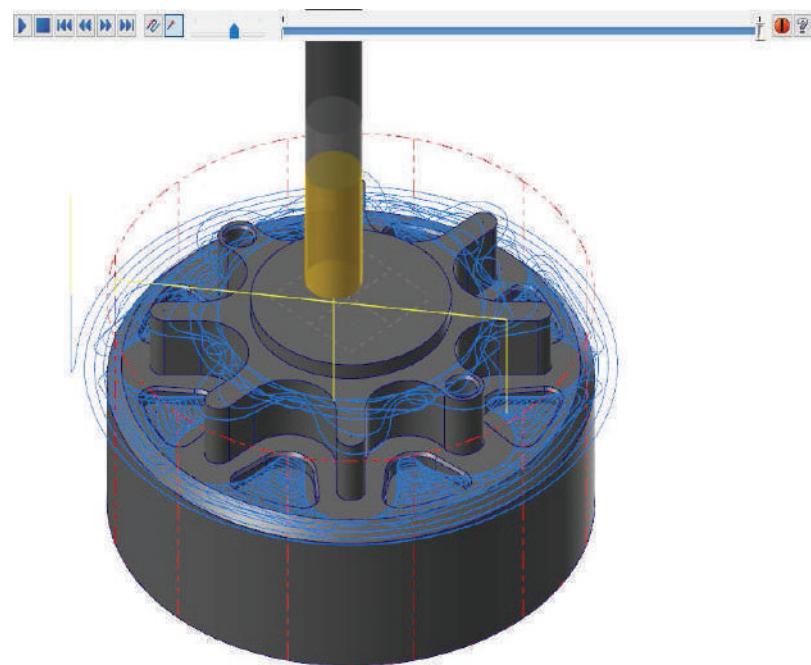
- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ To fit the workpiece to the screen, if needed, right mouse click in the graphics window again and select **Fit**.
- ♦ You can step through the **Backplot** by using the **Step forward** or **Step back** buttons.
- ♦ You can adjust the speed of the backplot.
- ♦ Select the **Play** button to run **Backplot**.



- After Backplot is completed, the toolpath should look as shown.



- Select the OK button to exit the Backplot.



STEP 8: SIMULATE THE TOOLPATH IN VERIFY

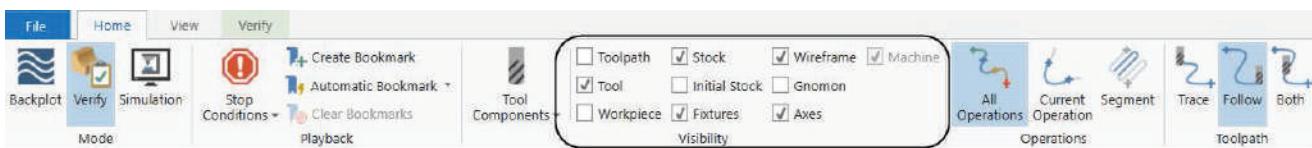
- From the Toolpaths Manager, select the Verify selected operations icon as shown.

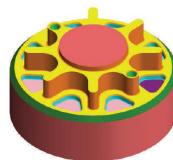


NOTE: Mastercam launches a new window that allows you to check the part using Backplot or Verify.

- From Home tab, change the settings for the Visibility as shown in [Figure: 8.0.1](#).
- In the Visibility group, click on Workpiece twice to display it translucent as shown in [Figure: 8.0.1](#).

Figure: 8.0.1

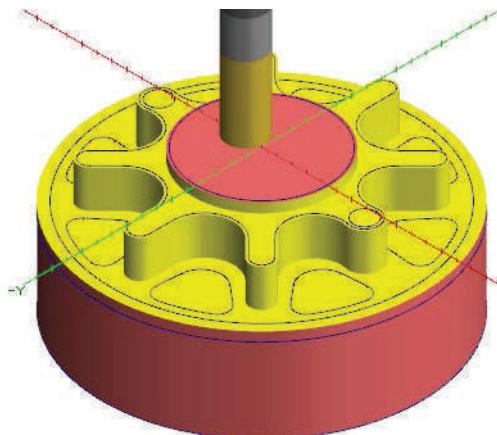




- ♦ Select the **Play** button to run **Verify**.



- ♦ The part will appear as shown.



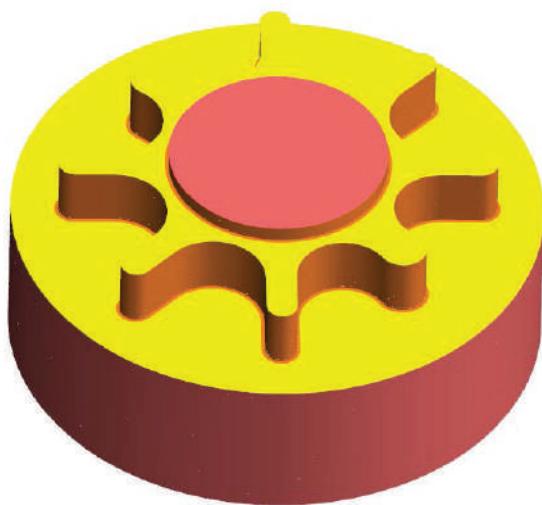
- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.
- ♦ Press **Alt+T** to remove the toolpath display.



STEP 9: FINISH THE WALLS USING CONTOUR TOOLPATH

In this step we will utilize the **Contour** toolpath to finish the walls of the part. You will remove the 0.03" stock from the walls.

Toolpath Preview:

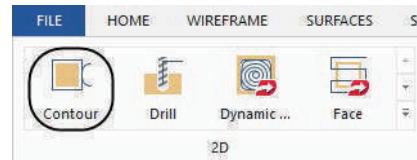




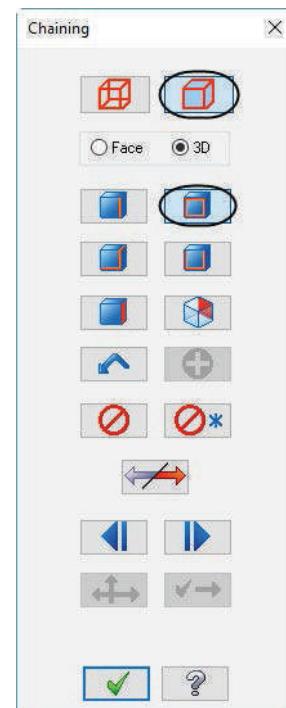
9.1 Chain Selection

TOOLPATHS

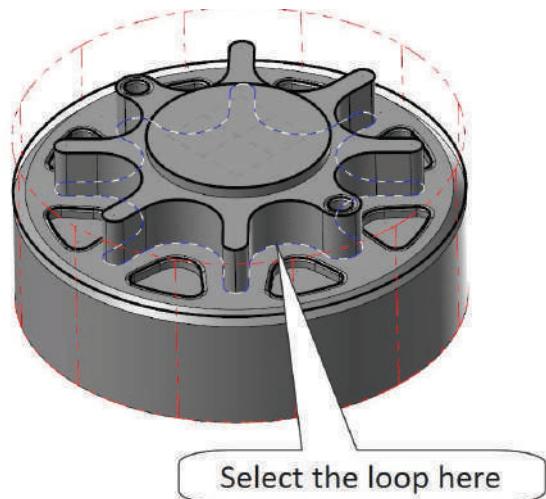
- From the **2D** group, select the **Contour** icon as shown.

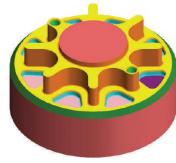


- In the **Chaining** dialog box, select **Solids selection** and enable only **Loop** as shown.



- Select the base of the arms as shown.

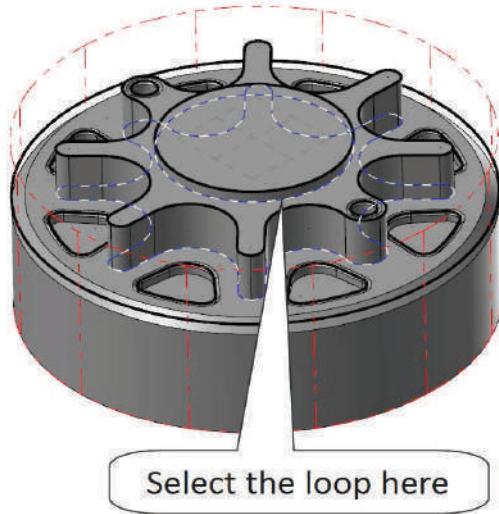




- The **Pick Reference Face** dialog appears, make sure that the bottom edge is selected and click on the **OK** button to continue.



- The arrow should be pointing in the clockwise direction. If it is not, select **Reverse**.
- Select the base of the small cylinder as the final chain.

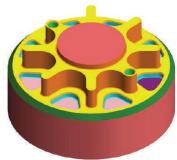


- Ensure the bottom edge is selected and click **OK** to confirm the **Reference Face**.



- The arrow should be pointing in the clockwise direction. If it is not, select **Reverse**.
- Select the **OK** button to exit the **Chaining** dialog box.
- In the **Toolpath Type** page, **Contour** will be selected.

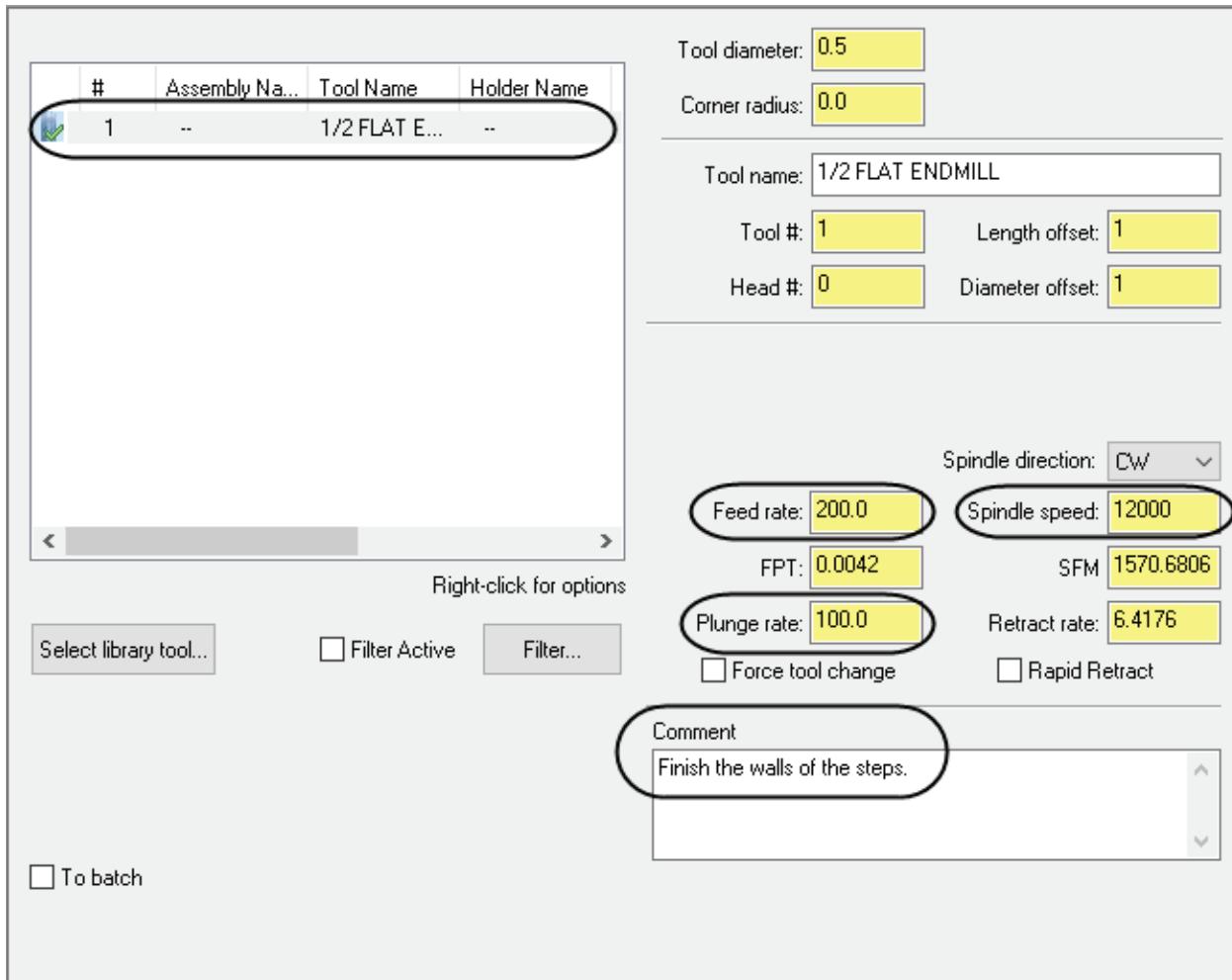


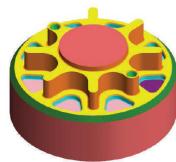


9.2 Select a 1/2" Flat Endmill from Tool list window

- ♦ Select **Tool** from the **Tree View list**.
- ♦ Select the **1/2" Flat Endmill** in the **Tool list** window. Make all the necessary changes as shown in [Figure: 9.2.1](#).

Figure: 9.2.1

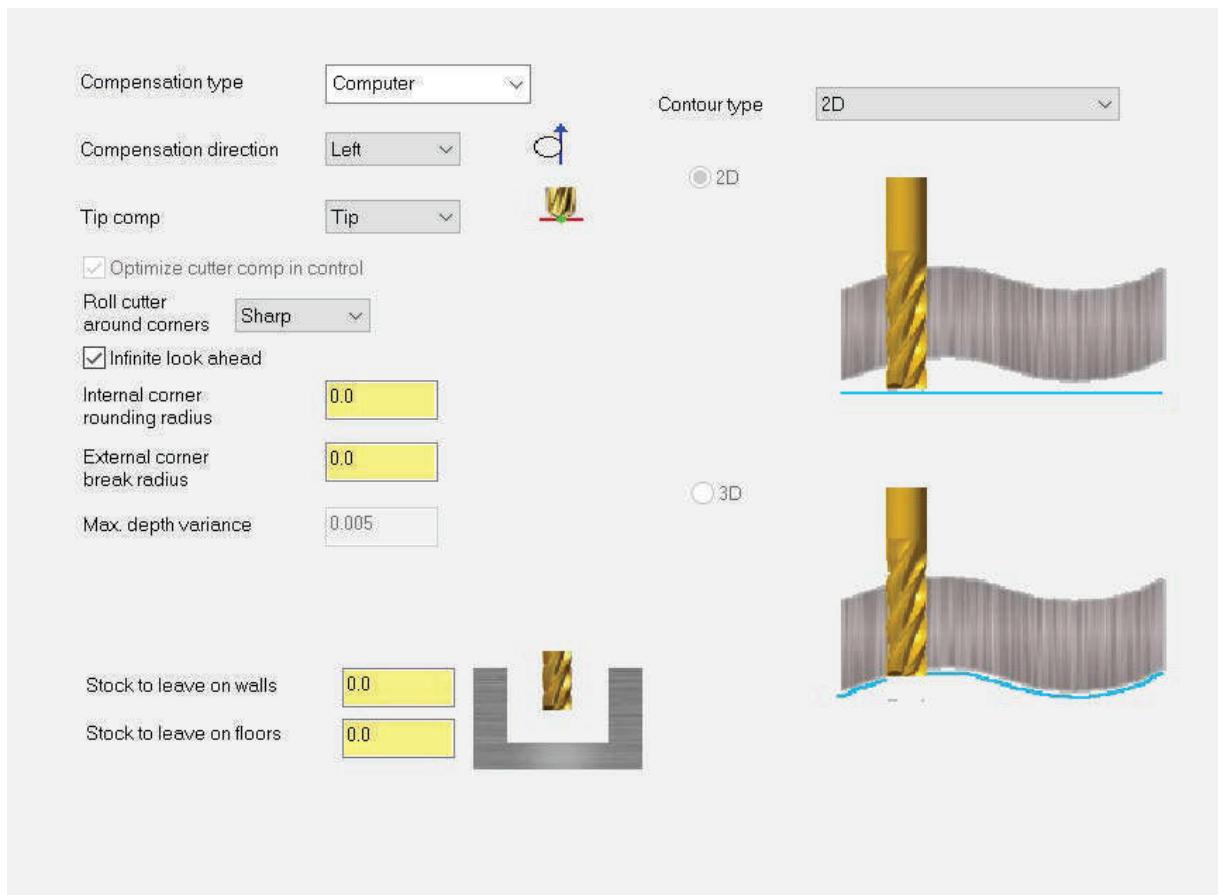




9.3 Set the Cut Parameters

- From the **Tree View** list, select **Cut Parameters** and make sure that the parameters are set as shown in [Figure: 9.3.1](#).

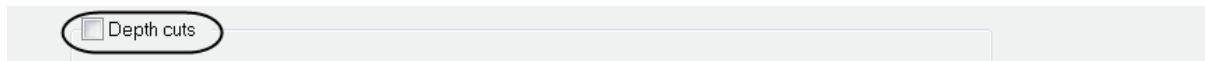
Figure: 9.3.1





9.4 Set the Depth Cuts Parameters

- From the Tree View list, disable the **Depth Cuts** if needed as shown.



9.5 Set the Lead In/Out parameters

- Make sure that the parameters are set as shown in [Figure: 9.5.1](#).

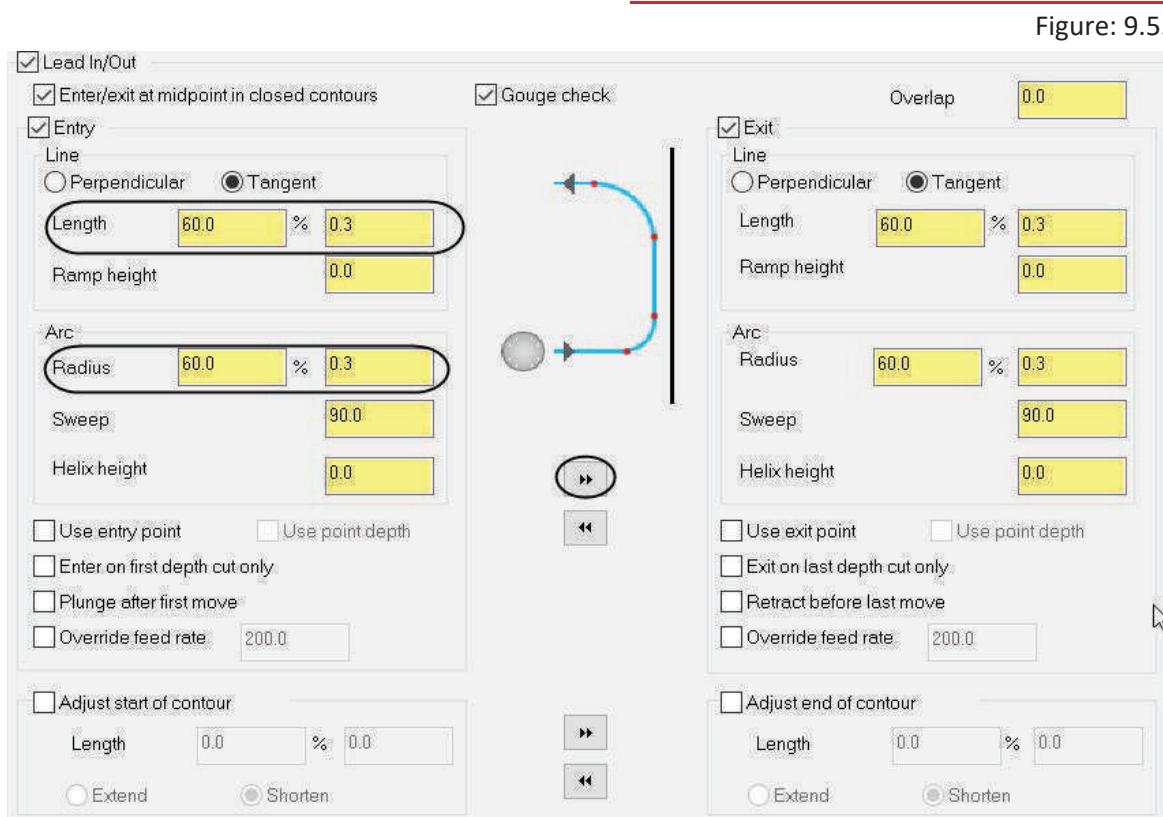
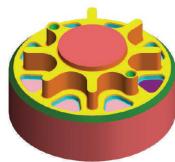


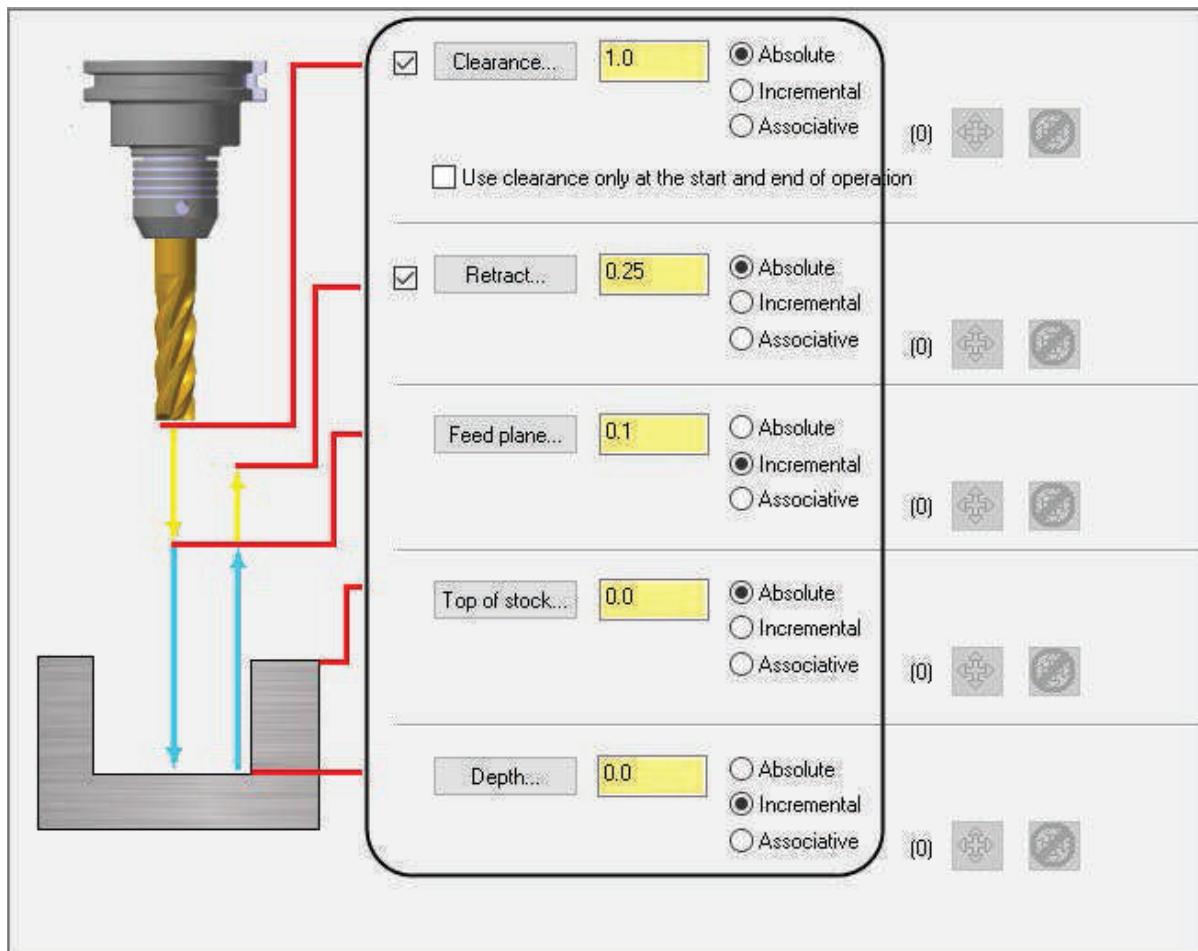
Figure: 9.5.1

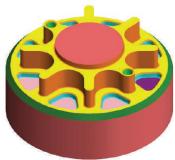


9.6 Set the Linking Parameters

- ♦ Select **Linking Parameters** and enable **Clearance**. Set the **Clearance** value to **1.0**. Set the **Depth** to **0** and **Incremental** as shown in [Figure: 9.6.1](#).

Figure: 9.6.1



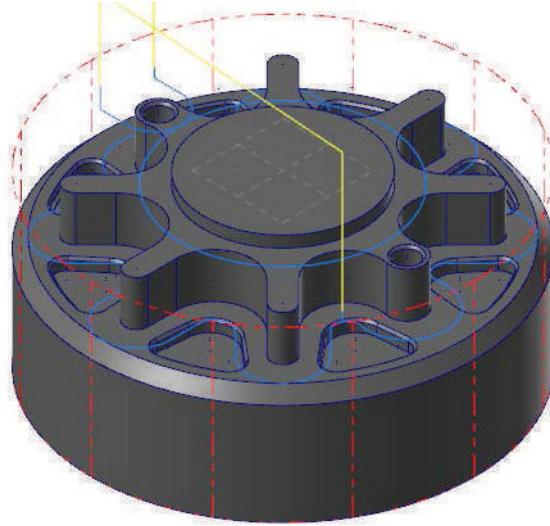


9.7 Preview the Toolpath

- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



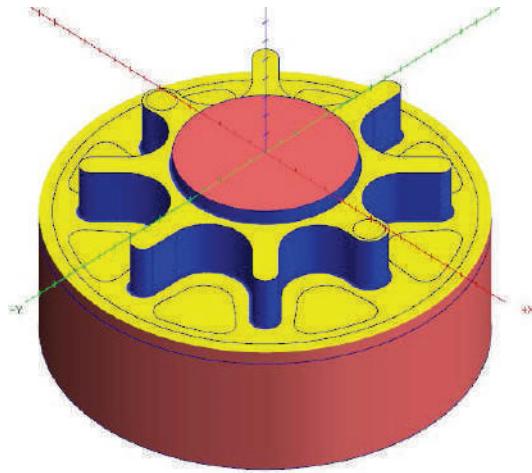
- ♦ See [page 145](#) to review the procedure. The toolpath should look as shown.



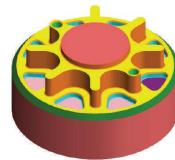
- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to exit the toolpath parameters.
- ♦ **Backplot** and **Verify** your toolpath. See [page 145](#) and [page 147](#) to review these procedures.
- ♦ The part will appear as shown.



- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.
- ♦ Press **Alt + T** to remove the toolpath display if needed.

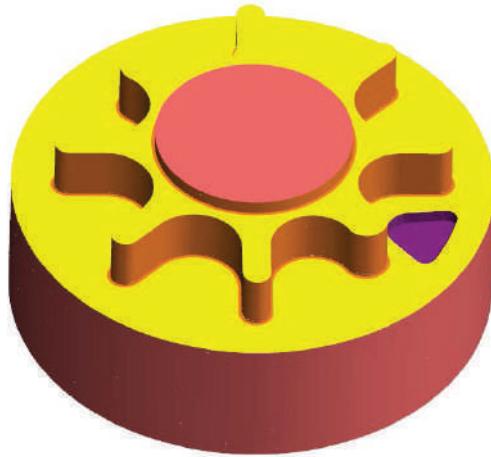


STEP 10: HIGH SPEED AREA MILL TOOLPATH

10.1 2D High Speed Area Mill the pocket

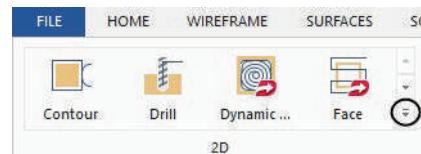
2D High Speed Area Mill toolpath allows you to machine pockets, material that other toolpaths left behind, and standing bosses or cores using a smooth clean motion. Helical entries and tangent stepovers create efficient motion for your machine tools. Cut parameters let you control corner rounding to create the best toolpath, avoiding sharp corners or direction changes. You will machine the pocket keeping the tool inside the machining region by selecting **Stay inside** as your machining region strategy.

Toolpath Preview:



TOOLPATHS

- From the **2D** group, select the **Expand gallery** arrow as shown.

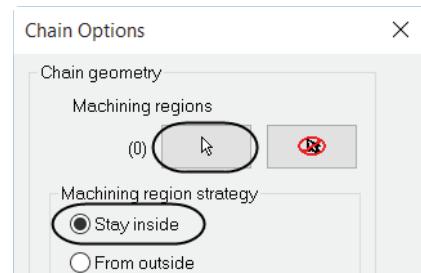


- Select the **Area Mill** icon.

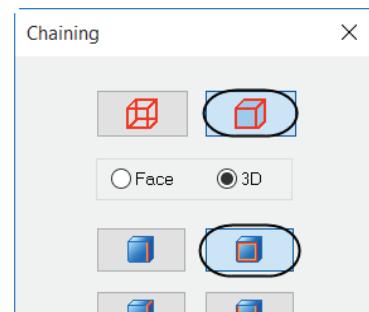




- In the **Chain Options**, make sure that **Stay inside** is enabled and click on the **Select Machining Chains** button in the **Machining regions** as shown.

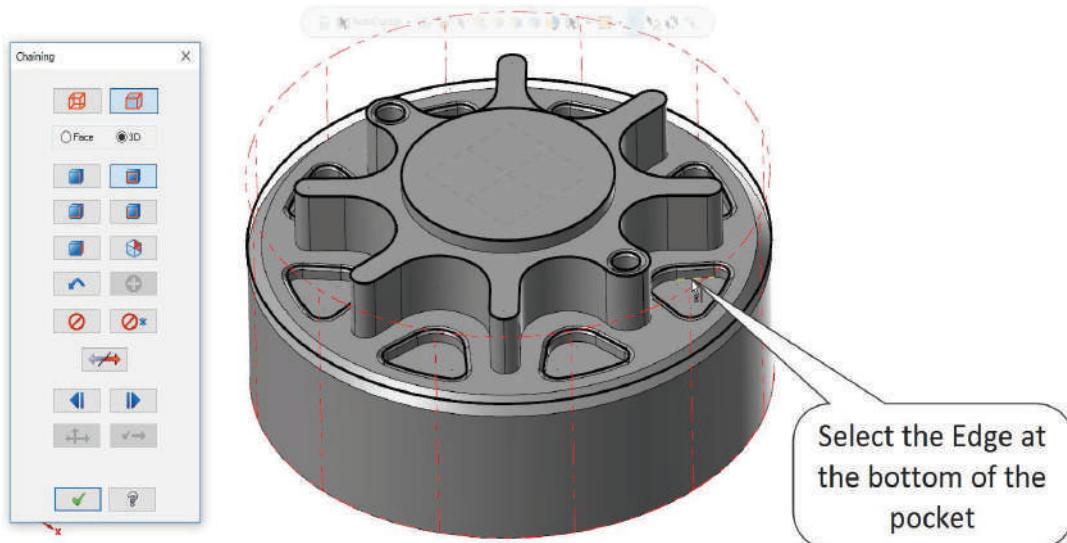


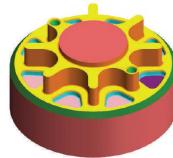
- In the **Chaining** dialog box, switch to **Solids** selection as shown.
- Make sure that only the **Loop** button is enabled as shown.



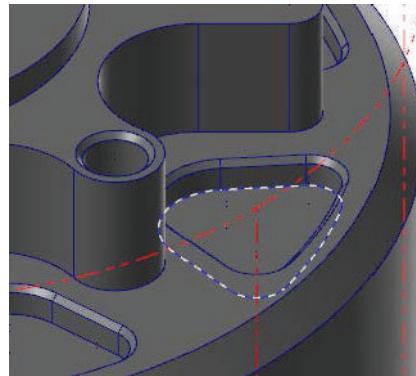
- Select the bottom of the pocket edge as shown in [Figure: 10.1.1](#).

Figure: 10.1.1

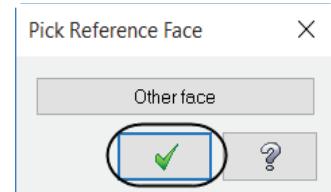




- The pocket bottom loop should be selected as shown.



- The **Pick Reference Face** dialog appears, make sure that the bottom edge is selected and click on the **OK** button to continue.

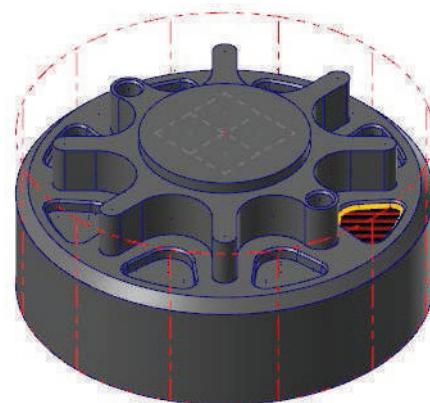


- Select the **OK** button to exit the **Chaining** dialog box.
- Select the **OK** button to exit the **Chain Options** dialog box.
- In the **Toolpath Type** page, **Area Mill** should be already selected as shown.

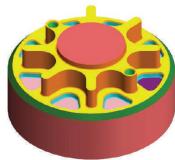


10.2 Preview Chains

- Select the **Preview chains** button as shown.
- See page 318 to review the procedure.
- The **Preview chains** should look as shown.



- Press **Esc** key to return to the toolpath parameters.



- Click on the **Preview chains** button again to clear the Preview chains display.



10.3 Select a 3/16" Flat Endmill from the Library and set the Tool Parameters

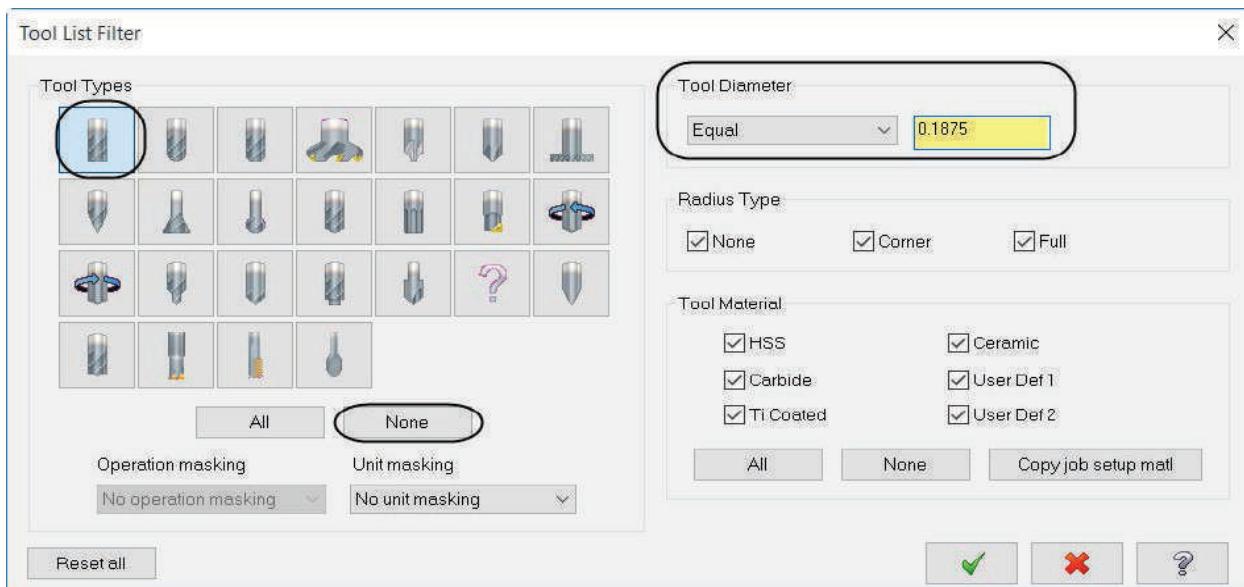
- Select **Tool** from the **Tree View** list.

- Click on the **Select library tool** button.
- Select the **Filter** button.



- Select the **None** button and then under **Tool Types** choose the **Flat Endmill** icon as shown.
- Under **Tool Diameter**, pick **Equal** and input a value **0.1875** as shown in [Figure: 10.3.1](#).

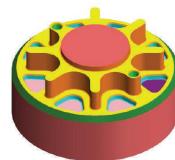
Figure: 10.3.1



- Select the **OK** button to exit the **Tool List Filter**.
- In the **Tool Selection** dialog box, you should only see a **3/16" Flat Endmill**.

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Ra...	Type
284	-	3/16 FLA...	-	0....	0.0	0.4375	4	No...	En...

- Select the **3/16" Flat Endmill** in the **Tool Selection** page and then select the **OK** button to exit.

TUTORIAL #4**HIGH SPEED AREA MILL TOOLPATH**

- ♦ Make all the necessary changes as shown in [Figure: 10.3.2.](#)

Figure: 10.3.2

#	Assembly Na...	Tool Name	Holder Name
1	--	1/2 FLAT E...	--
2	--	3/16 FLAT ...	--

Tool diameter: 0.1875
Corner radius: 0.0
Tool name: 3/16 FLAT ENDMILL
Tool #: 2 Length offset: 2
Head #: 0 Diameter offset: 2

RCTF Spindle direction: Cw
Feed rate: 200.0 Spindle speed: 12000
FPT: 0.0042 SFM: 589.0052
 Plunge rate: 100.0 Retract rate: 6.160896
 Force tool change Rapid Retract

Tool inspection / change
Force retract every: 0.0 Inches 0.0 Minutes
 To batch

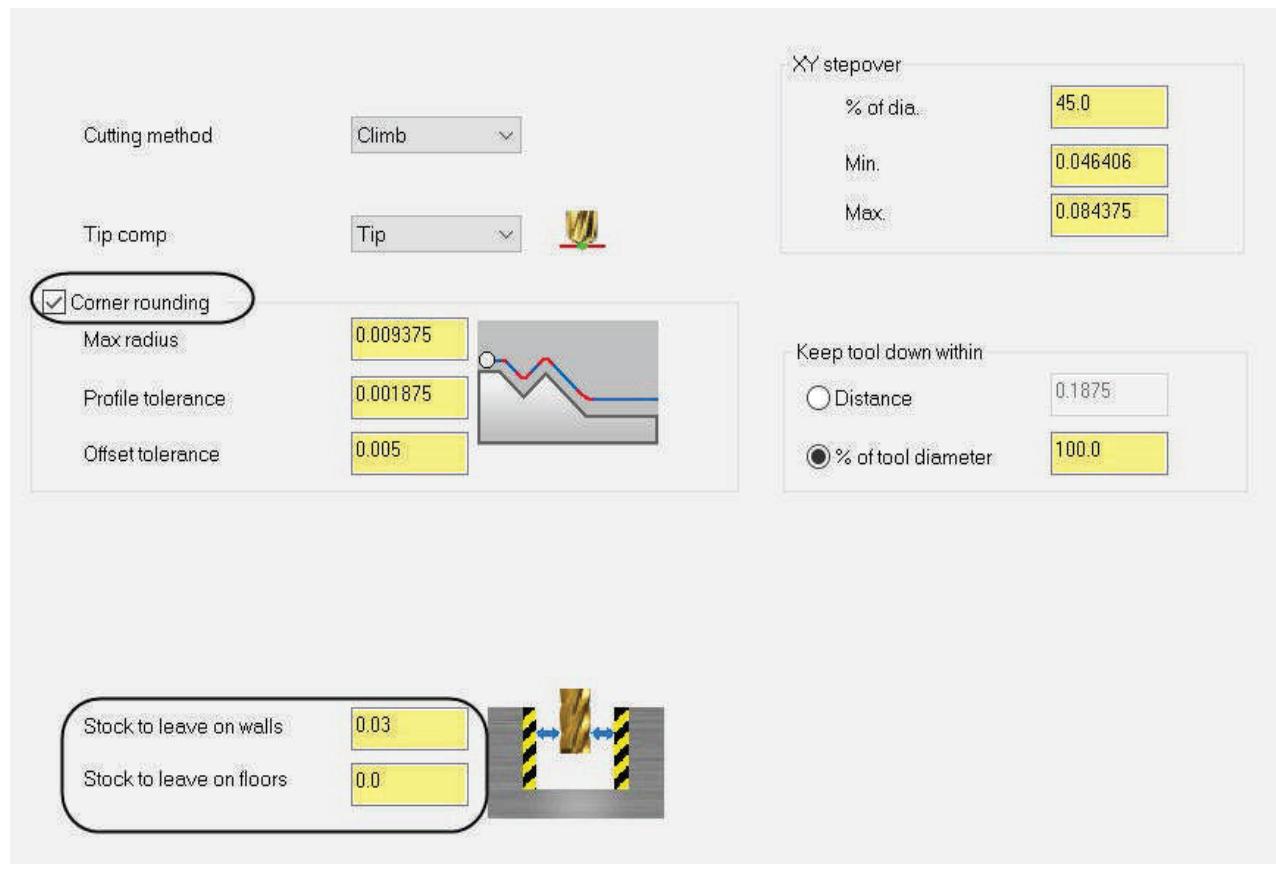
Comment:
Use Area Mill to machine the pocket.



10.4 Set the Cut Parameters

- From the Tree View list, select **Cut Parameters**.
- Set the parameters as shown in [Figure: 10.4.1](#).

Figure: 10.4.1



Cutting method set to **Climb** cuts in one direction with the tool rotating in the opposite direction of the tool motion.

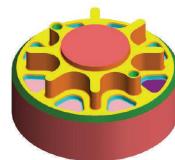
XY stepover sets the distance between cutting passes in the X and Y axes.

% of diameter expresses the maximum XY stepover as a percentage of the tool diameter. The Max. XY stepover field will update automatically when you enter a value in this field. The actual stepover is calculated by Mastercam between the Min. and Max. values.

Corner rounding activates toolpath corner rounding, which replaces sharp corners with arcs for faster and smoother transitions in tool direction.

Profile tolerance represents the maximum distance that the outermost profile of a toolpath created with corner rounding can deviate from the original toolpath.

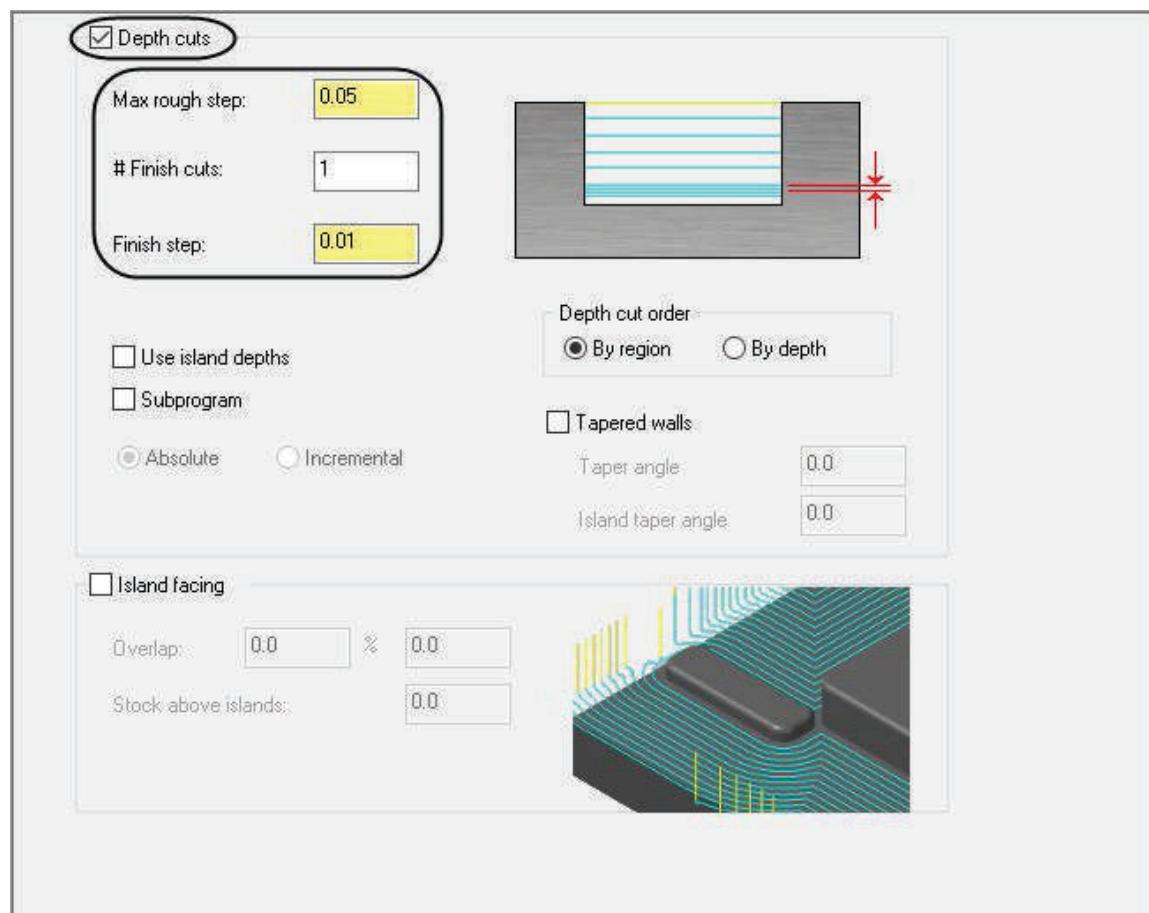
Offset tolerance represents the maximum distance that a profile of a toolpath created with corner rounding can deviate from the original toolpath. This is the same measurement as the profile tolerance but is applied to all the profiles except the outermost one.



10.5 Set the Depth Cuts parameters

- From the Tree View list, select **Depth Cuts** and set the parameters as shown in [Figure: 10.5.1](#).

Figure: 10.5.1





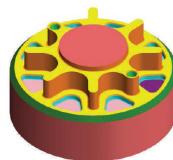
10.6 Set the Transitions

- From the **Tree View** list, select **Transitions** and leave the **Entry method** set to **Entry helix** as shown in [Figure: 10.6.1](#).

Figure: 10.6.1

The screenshot shows the 'Transitions' dialog box in Mastercam. The 'Entry method' section is selected, with 'Entry helix' checked and a radius of 0.1. A note states: 'Note: If helix fails, profile ramp will be used'. The 'Entry feed rate' section shows 'Plunge rate' selected. In the bottom section, 'Z clearance' is set to 0.05, 'Plunge angle' to 2.0, 'Preferred profile length' to 0.5, and 'Skip pockets smaller than' to 0.1. To the right, a 3D model of a part is shown with a yellow line indicating the helical entry path.

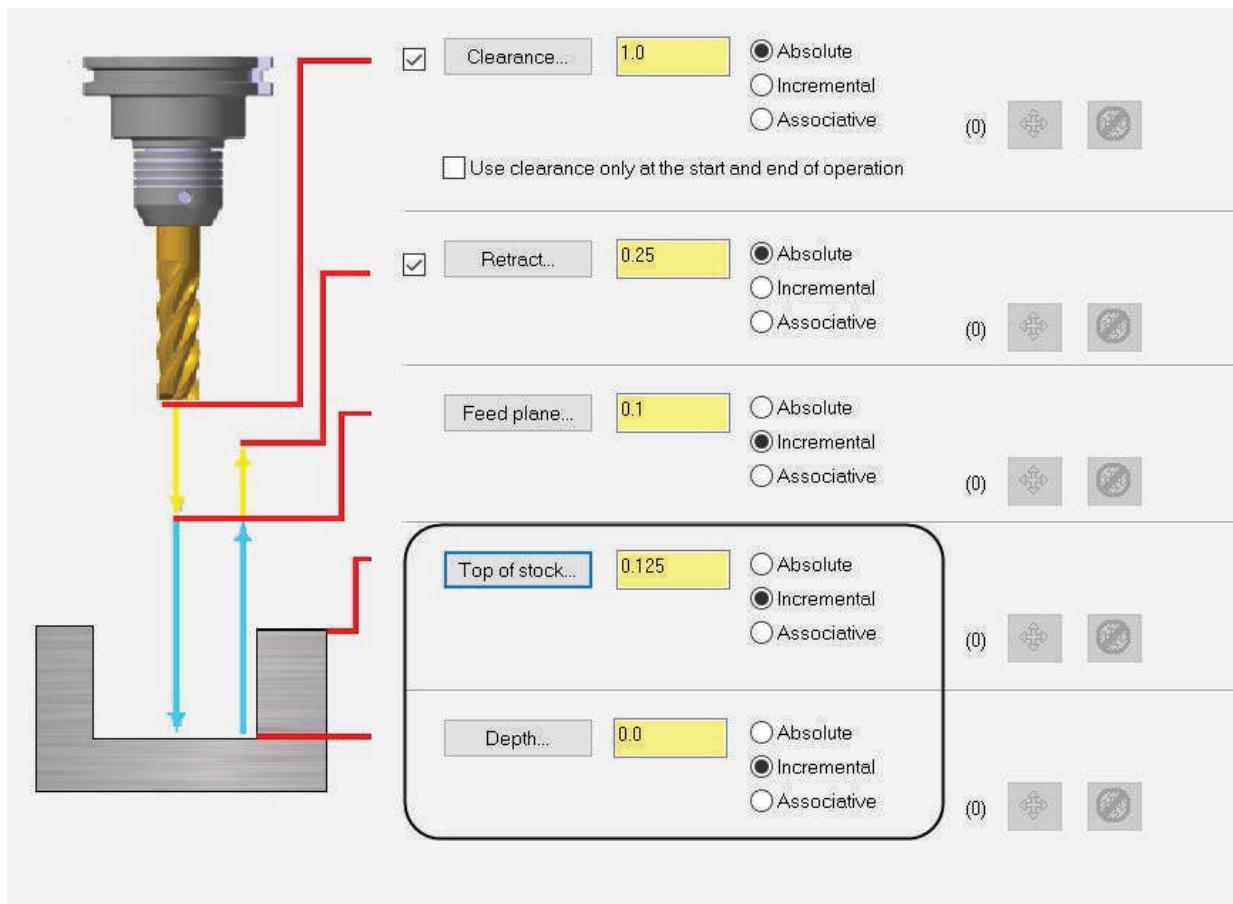
Entry method set to **Helix only** creates a helical entry into the part.



10.7 Set the Linking Parameters

- ♦ Select **Linking Parameters** from the tree view list and set the parameters as shown in [Figure: 10.7.1](#).

Figure: 10.7.1



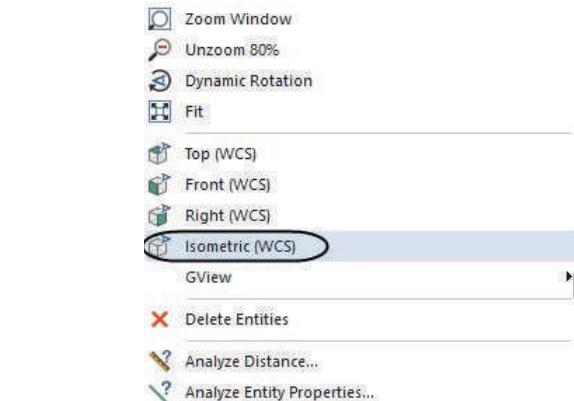


10.8 Preview the Toolpath

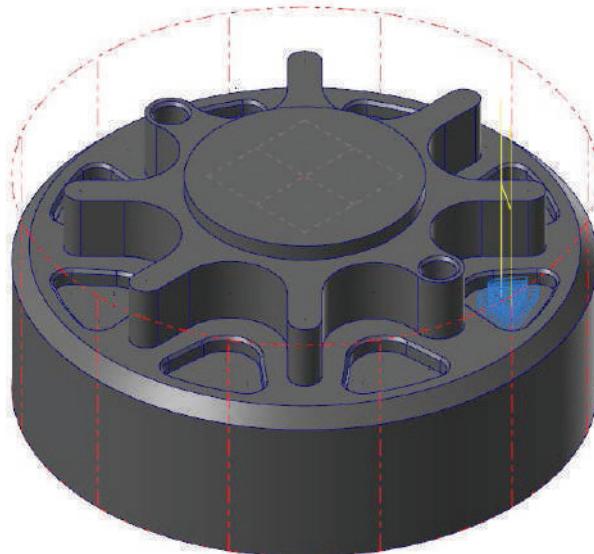
- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.
- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.

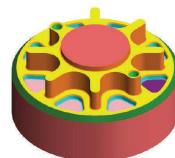


- ♦ Press **Esc** key to exit the preview.

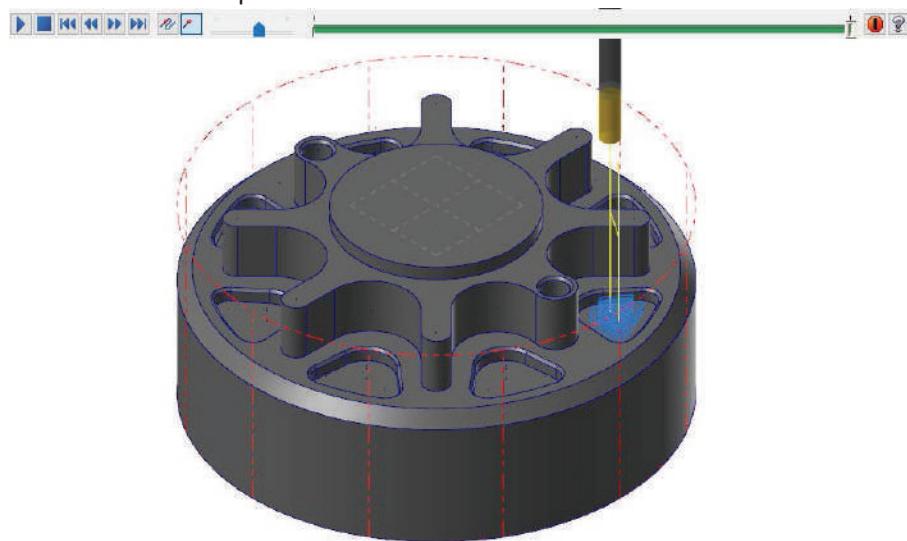
NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to generate the toolpath.

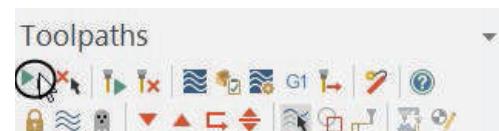




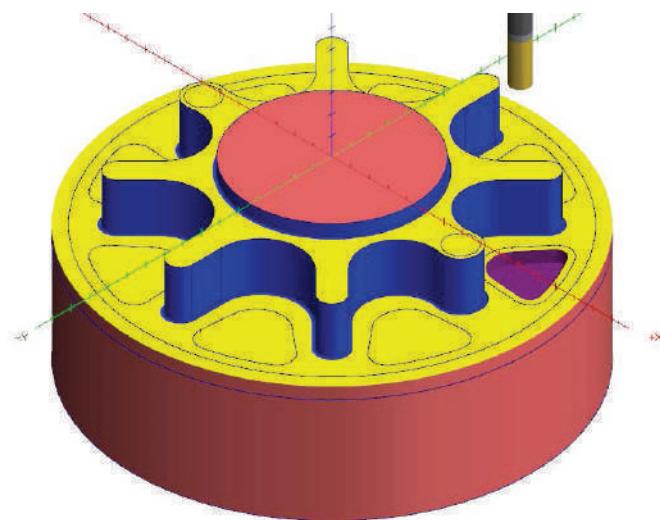
- ♦ To Backplot the toolpath, see [page 145](#) to review this procedure.



- ♦ To Verify the toolpath, see [page 147](#).
- ♦ To select both toolpaths, click on the **Select all operations** icon.



- ♦ The part will appear as shown.



- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.

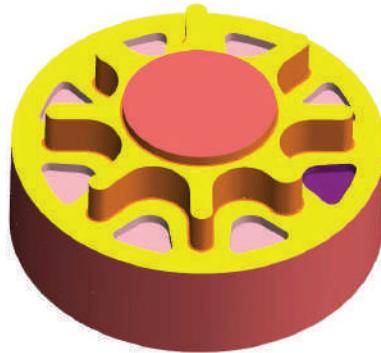




STEP 11: TRANSFORM-ROTATE TOOLPATH

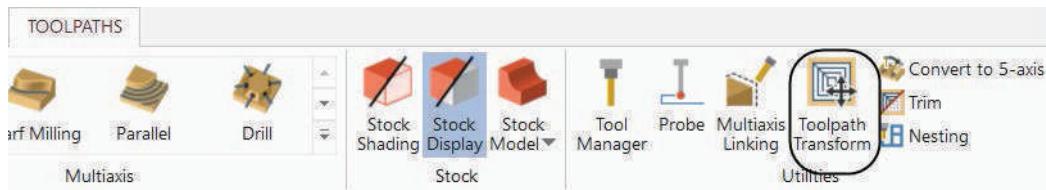
Transform toolpaths are used when you want to run the same toolpath in different locations. You can transform a single toolpath or several at a time. In this step you will machine one of the smaller pockets using 2D HS Area Mill. Then, using the Transform Rotate toolpath, you will generate the toolpaths for the rest of the identical pockets.

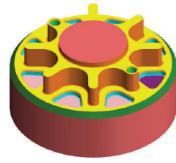
Step Preview:



TOOLPATHS

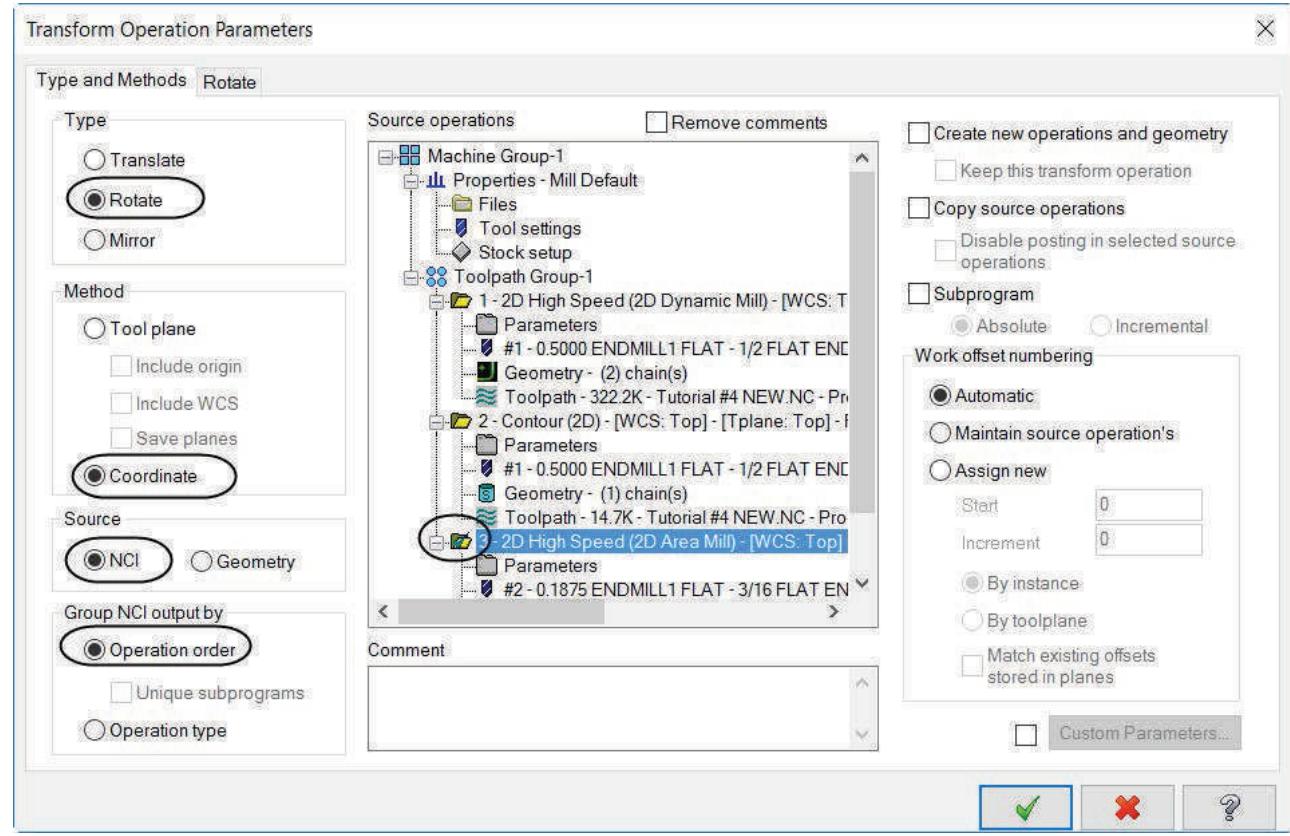
- From the **Utilities** group, select the **Toolpath Transform**.





- For the **Type**, select **Rotate**, and select **Coordinate** for **Method**.
- Select **Operation 3**.
- In the **Group NCI output by** area select **Operation order** as shown in [Figure: 11.0.1](#).

Figure: 11.0.1



Rotate revolves the toolpath about the construction origin or a specified point. Activate the Rotate tab and you can set the rotation point and number of copies.

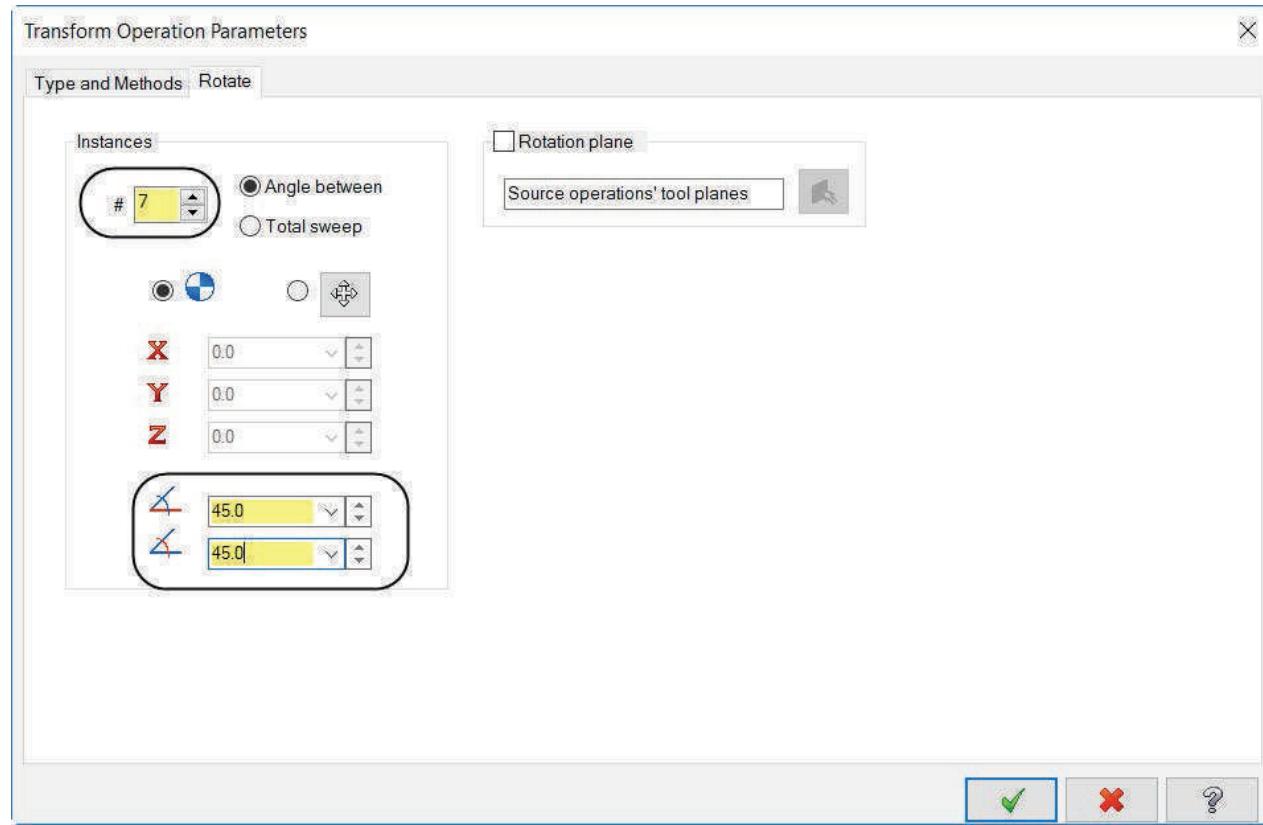
Coordinate creates new coordinate positions for the new toolpaths in the original tool plane.

Operation order sorts the transformed operations by the order they were selected. For example, if we choose the large pocket then small pocket. It will execute them in that order (large pocket, small pocket, large pocket, small pocket, etc.).



- Choose the **Rotate** tab.
- Input the **Instances #7**, a **Start angle** of **45.0** degrees, and a **Rotation angle** of **45.0** degrees as shown in [Figure: 11.0.2](#).

Figure: 11.0.2



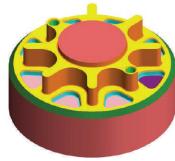
Number of steps is the number of times to rotate the toolpath.

Start angle sets the beginning angle for the rotate toolpath.

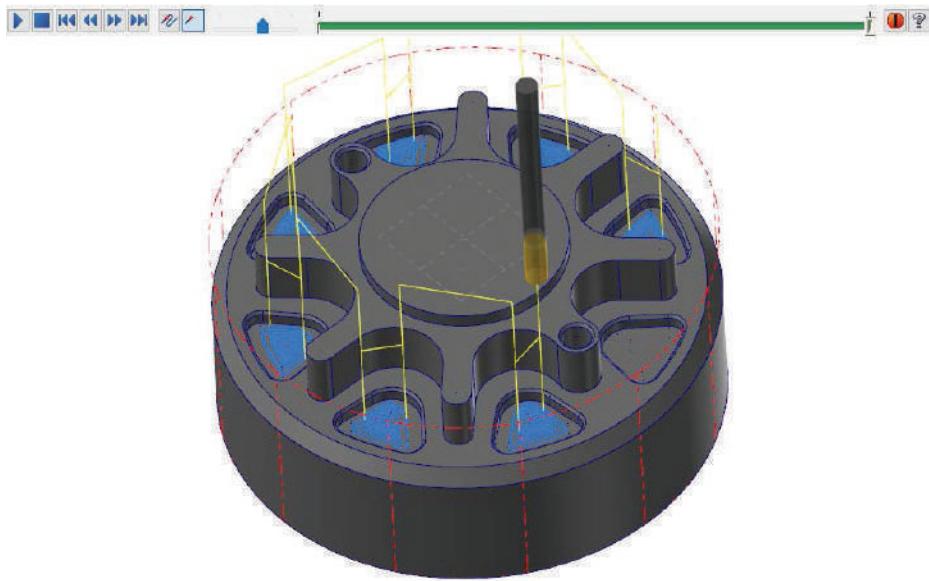
Rotation angle sets the angle of rotation for the transformed toolpath.

- Select the **OK** button to generate the toolpath.

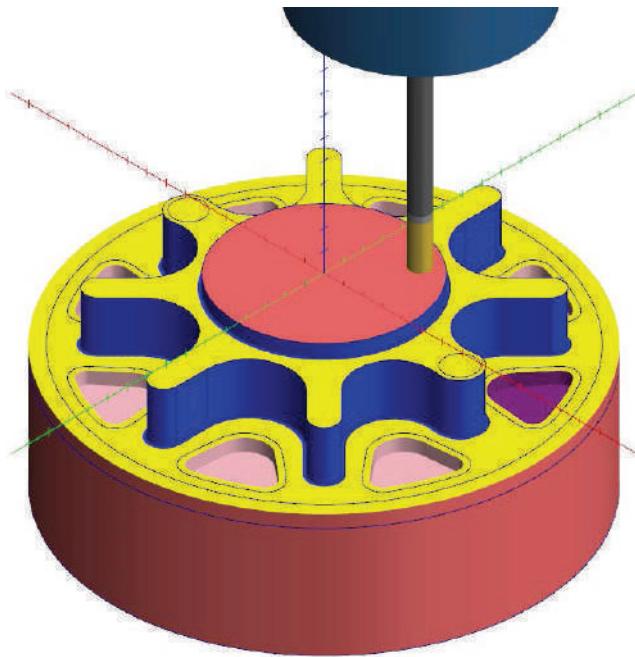




- ♦ Backplot the toolpath as shown on [page 145](#).



- ♦ Select the **OK** button to exit **Backplot**.
- ♦ Select all toolpaths to **Verify** them as shown on [page 147](#).



- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.
- ♦ Press **Alt + S** to shade the solid.
- ♦ Press **Alt + T** to remove the toolpath display.



STEP 12: FINISH THE POCKET WALLS USING CONTOUR TOOLPATH

In this step we will utilize the **Contour** toolpath to finish the walls of the part. You will remove the 0.03" stock from the walls.

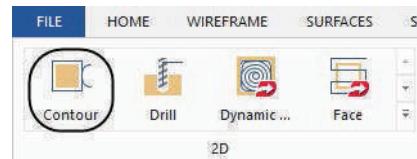
Toolpath Preview:



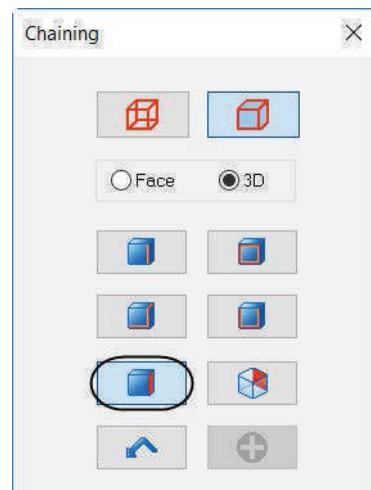
12.1 Chain Selection

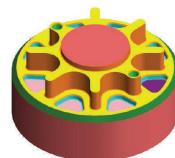
TOOLPATHS

- From the **2D** group, select the **Contour** icon as shown.

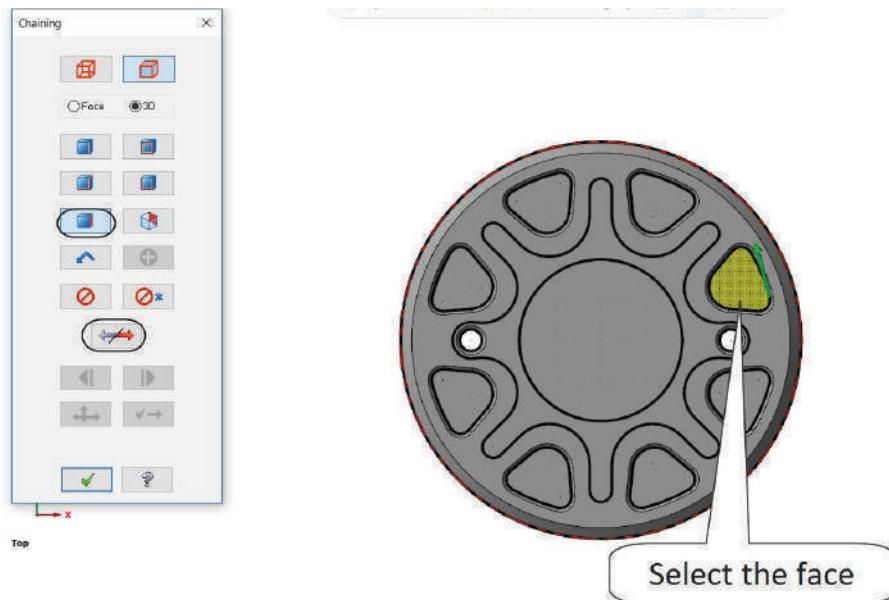


- In the **Chaining** dialog box, select **Solids selection** and enable only **Face** as shown.

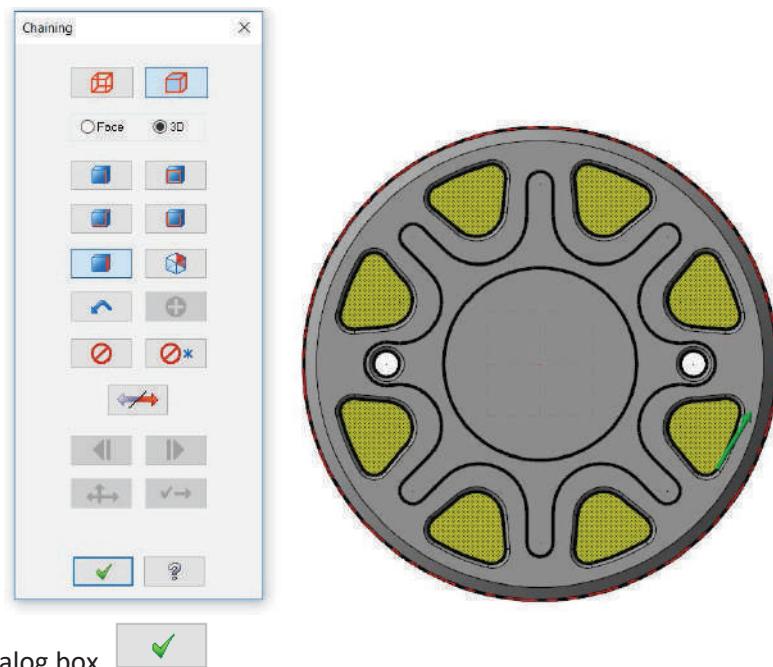




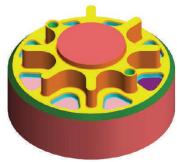
- ♦ Right mouse click in the graphics window and select the **Top** view.
- ♦ Select the bottom pocket as shown.



- ♦ Use Reverse button to change the direction to CCW if needed.
- ♦ Select the rest of the pockets as shown. The arrow should be pointing in the **COUNTERCLOCKWISE** direction. If it is not select **Reverse**.



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



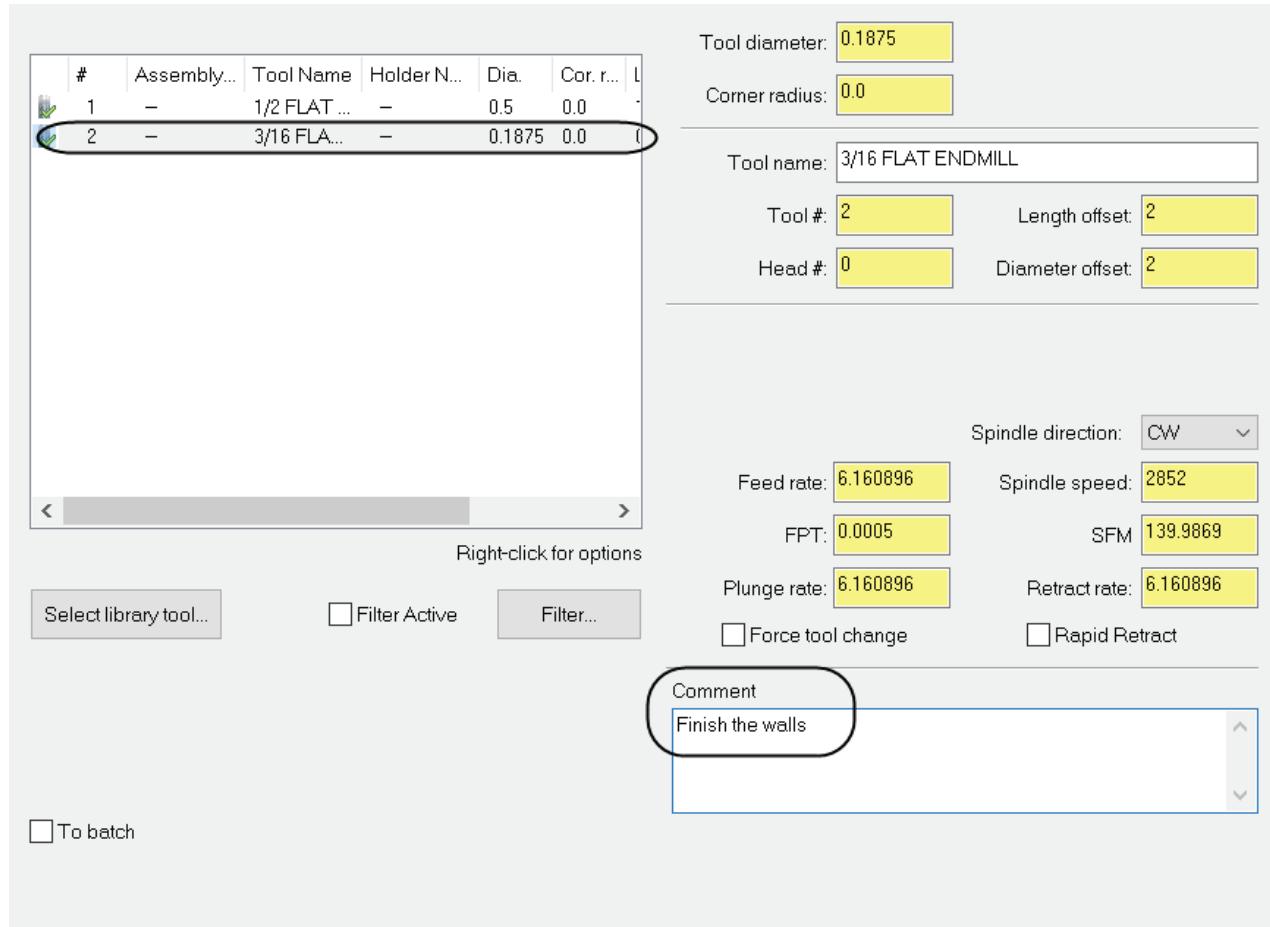
- In the Toolpath Type page, Contour will be selected.

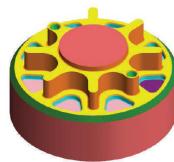


12.2 Select a 3/16" Flat Endmill from Tool list window

- Select **Tool** from the **Tree View list**.
- Select the **3/16" Flat Endmill** in the **Tool list** window. Make all the necessary changes as shown in Figure: [12.2.1](#).

Figure: 12.2.1

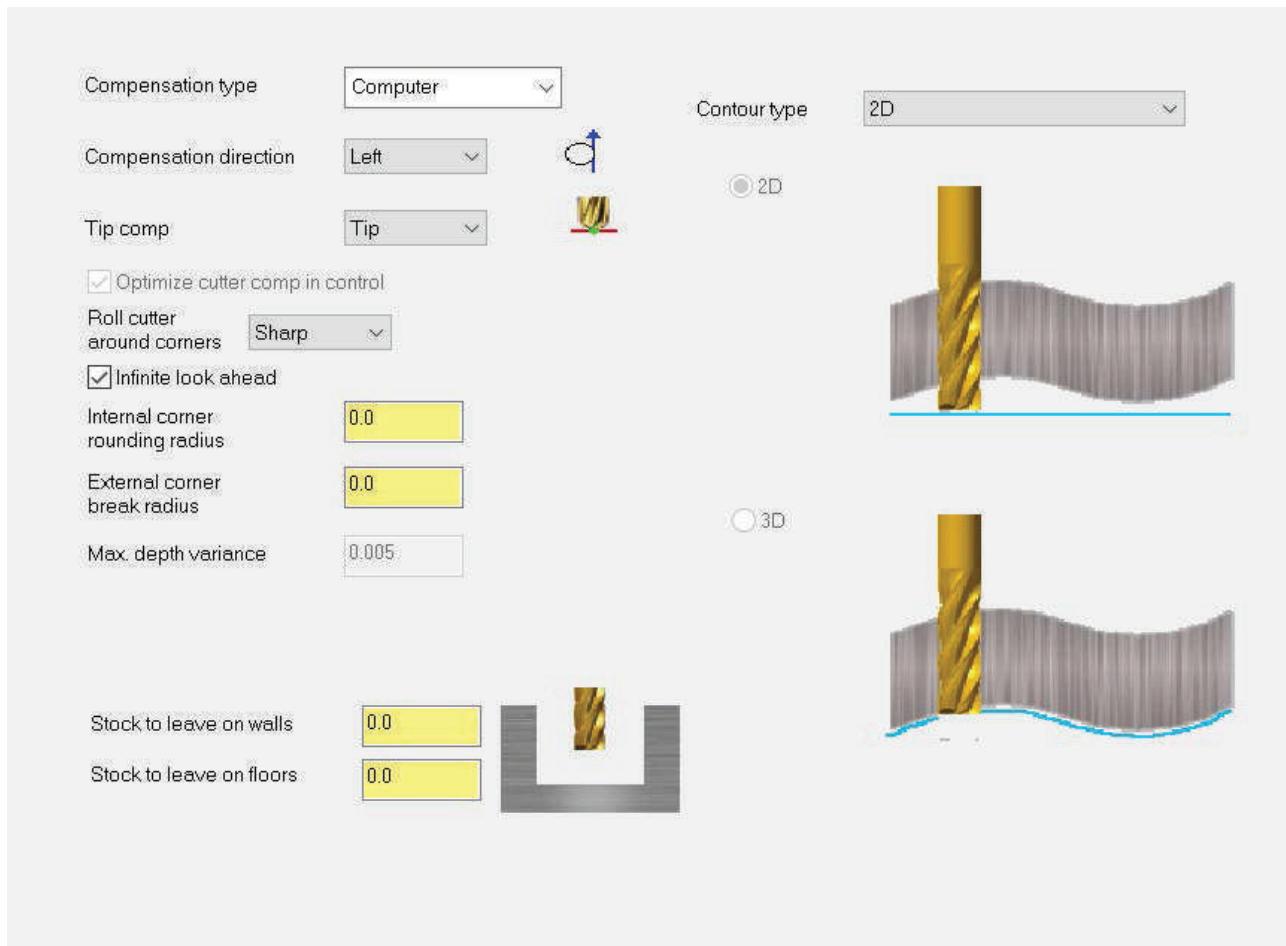


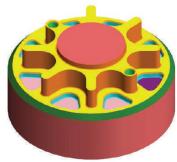


12.3 Set the Cut Parameters

- From the **Tree View** list, select **Cut Parameters** and make sure that the parameters are set as shown in [Figure: 12.3.1](#).

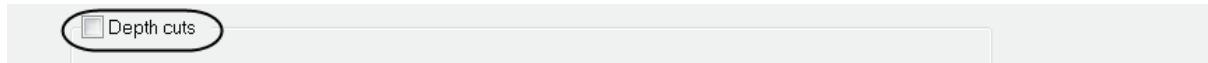
Figure: 12.3.1





12.4 Set the Depth Cuts Parameters

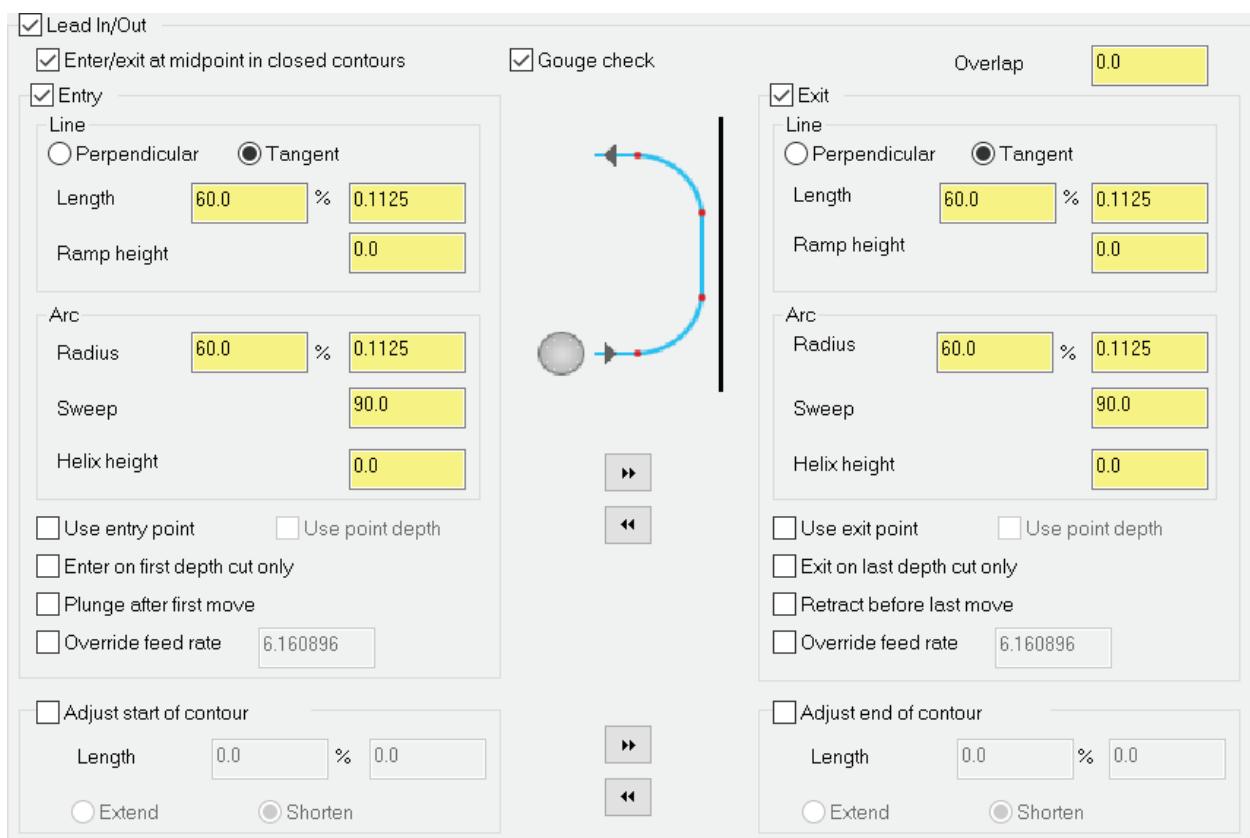
- From the Tree View list, disable the **Depth Cuts** if needed as shown.



12.5 Set the Lead In/Out parameters

- Make sure that the parameters are set as shown in [Figure: 12.5.1](#).

Figure: 12.5.1

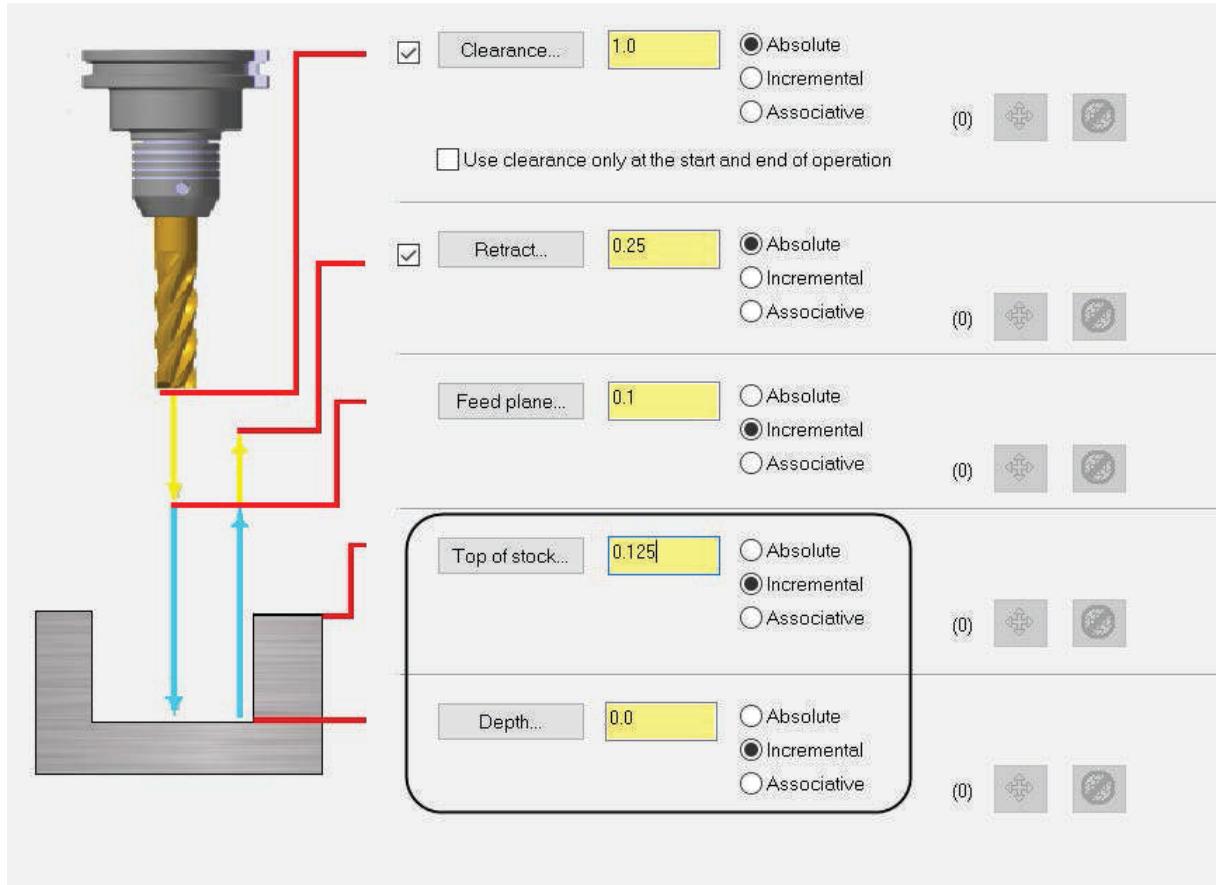




12.6 Set the Linking Parameters

- ♦ Select **Linking Parameters** and enable **Clearance**. Set the **Clearance** value to **1.0**. Set the parameters as shown in [Figure: 12.6.1](#).

Figure: 12.6.1



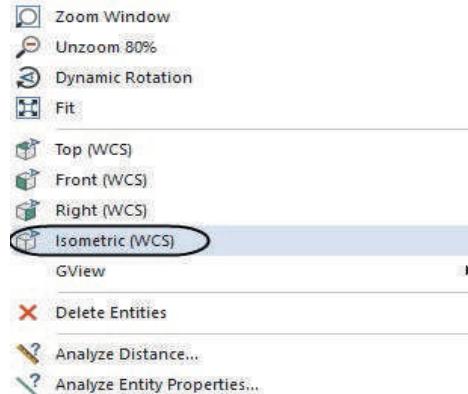


12.7 Preview the Toolpath

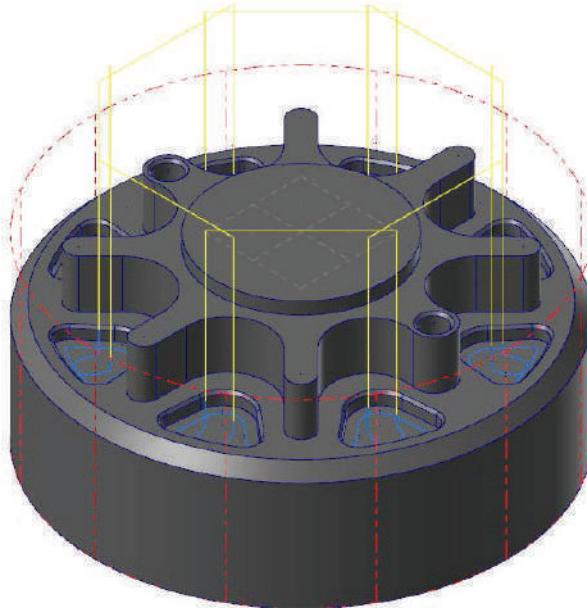
- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.
- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.



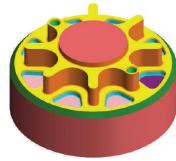
- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

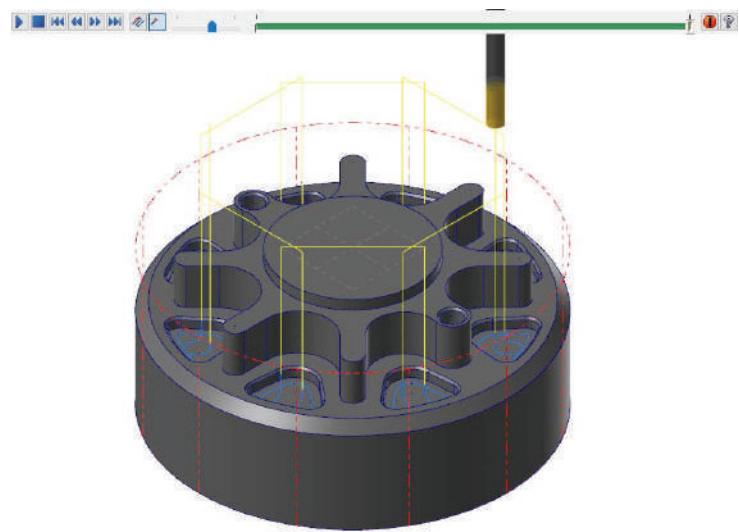
- ♦ Select the **OK** button to generate the toolpath.



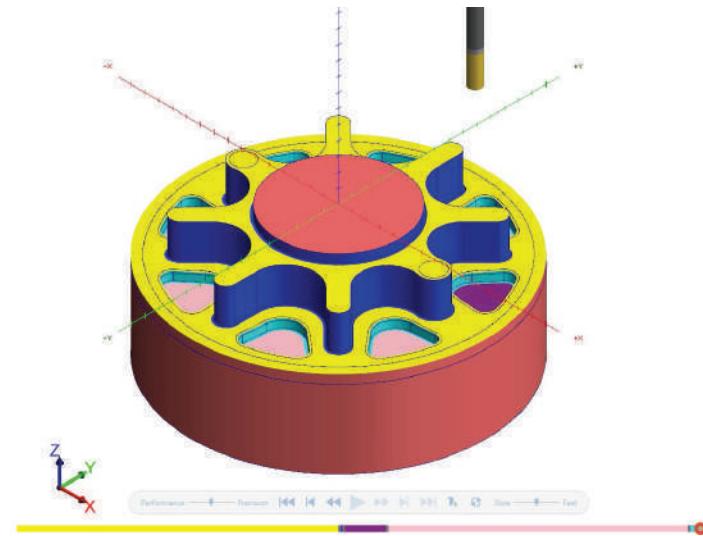
TUTORIAL #4 FINISH THE POCKET WALLS USING CONTOUR TOOLPATH



- ♦ Backplot the toolpath as shown on page 145.



- ♦ Minimize the **Mastercam Simulator** window, select all toolpaths and then **Verify** them as shown on page 147.



- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.





STEP 13: SPOT DRILL THE HOLES

Spot Drilling the holes allows you to start the hole. In this operation, we will use the spot drill to chamfer the hole before drilling it.

Toolpath Preview:

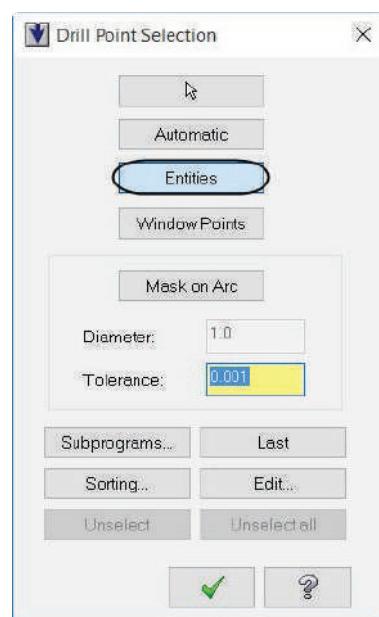


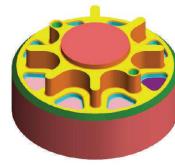
TOOLPATHS

- From the **2D** group, select the **Drill** icon.



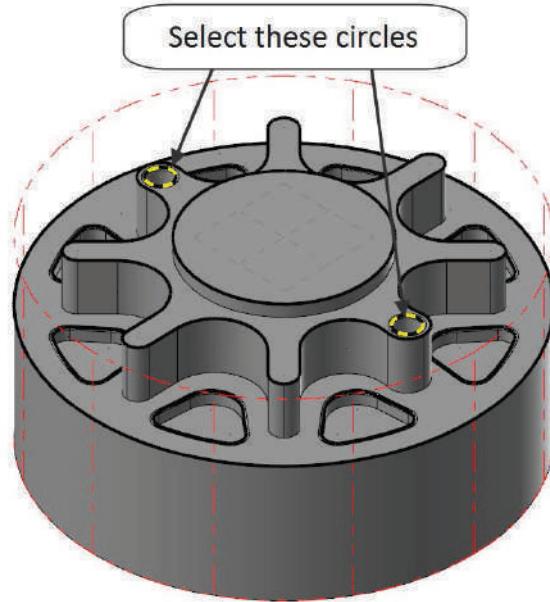
- In the **Drill Point Selection** dialog box, choose the option **Entities**.





- ♦ Select the circles as shown in [Figure: 13.0.1](#).

Figure: 13.0.1



- ♦ Click on the **End Selection** button or press **Enter** to finish the selection.

End Selection

- ♦ Select the **OK** button to exit the **Drill Point Selection** dialog box.
- ♦ In the **Toolpath Type** page, the **Drill** toolpath will be selected.

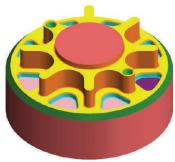


13.1 Select a 1/2" Spot Drill from the Library and set the Tool Parameters

- ♦ Select **Tool** from the **Tree view list**.
- ♦ Click on the **Select library tool** button.
- ♦ To be able to see just the spot drill, select the **Filter** button.

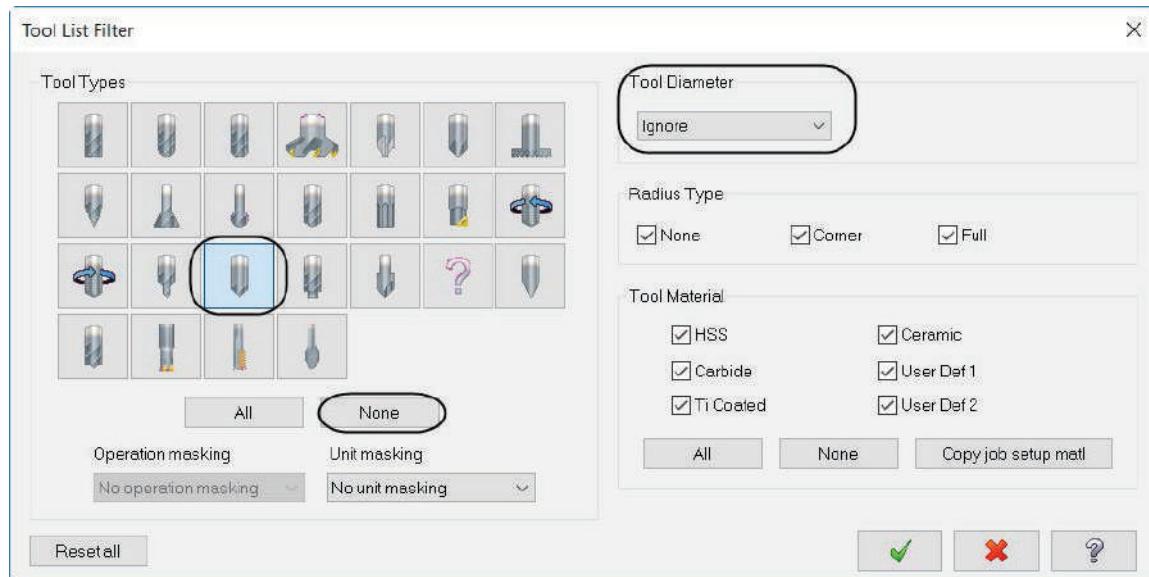
Select library tool...





- Under **Tool Types**, select the **None** button and then choose the **Spot drill** icon as shown in [Figure: 13.1.1](#).

Figure: 13.1.1



- Select **OK** button to exit the **Tool List Filter** dialog box.



- At this point you should only see **Spot Drills**.

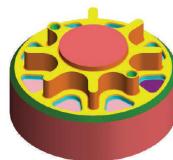
- From that list select the **1/2" Spot Drill**.

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	Type	Ra...	# Flutes
21	--	1/8 SPOTDRILL	--	0.125	0.0	2.0	Sp...	No...	2
22	--	1/4 SPOTDRILL	--	0.25	0.0	2.0	Sp...	No...	2
23	--	3/8 SPOTDRILL	--	0.375	0.0	2.0	Sp...	No...	4
24	--	1/2 SPOTDRILL	--	0.5	0.0	2.0	Sp...	No...	2
25	--	3/4 SPOTDRILL	--	0.75	0.0	2.0	Sp...	No...	4
26	--	1. SPOTDRILL	--	1.0	0.0	2.0	Sp...	No...	4

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

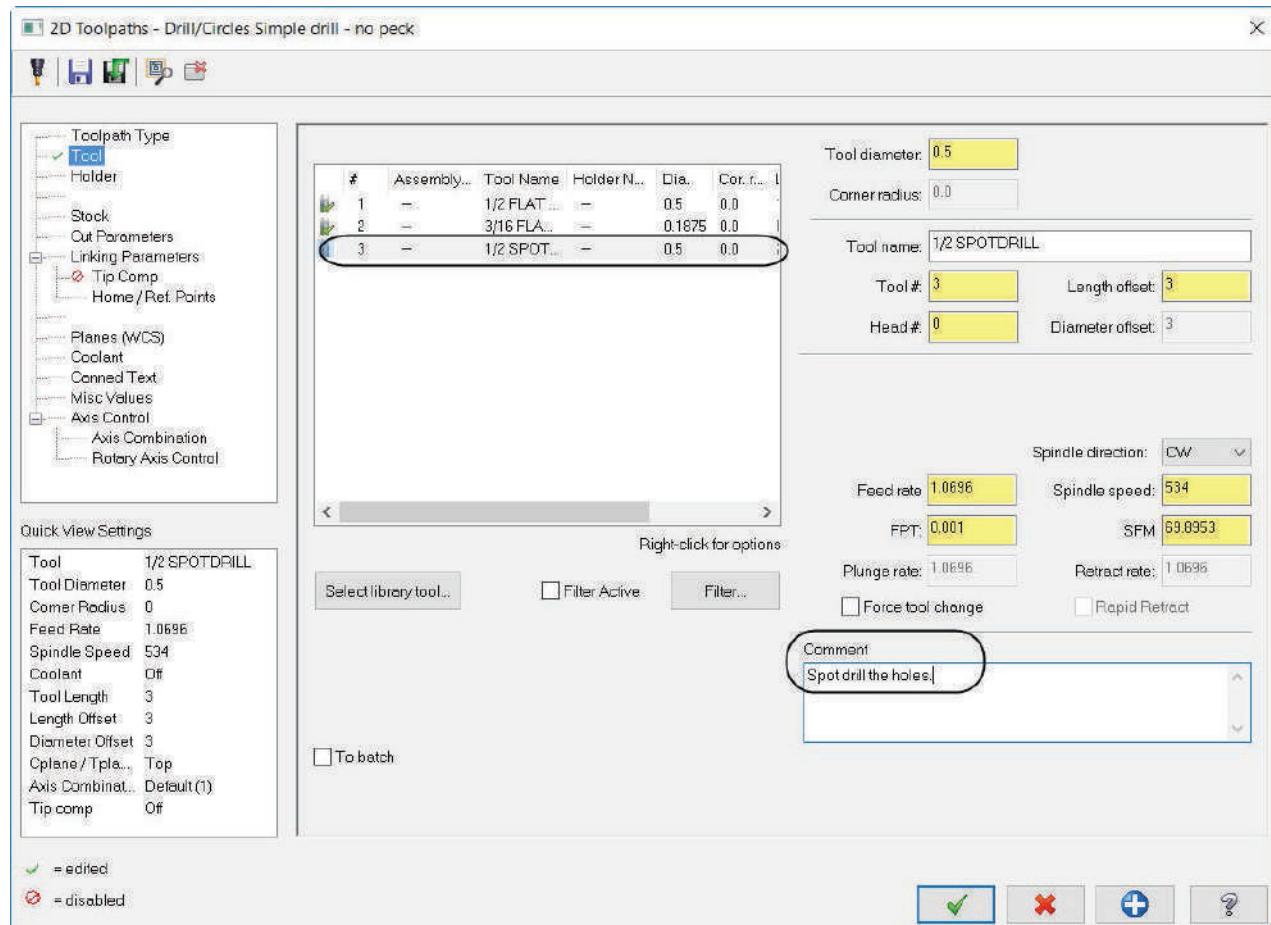


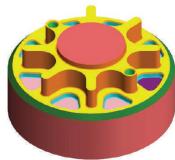
TUTORIAL #4 SPOT DRILL THE HOLES



- Make the necessary changes to the Tool page as shown in [Figure: 13.1.2](#).

Figure: 13.1.2

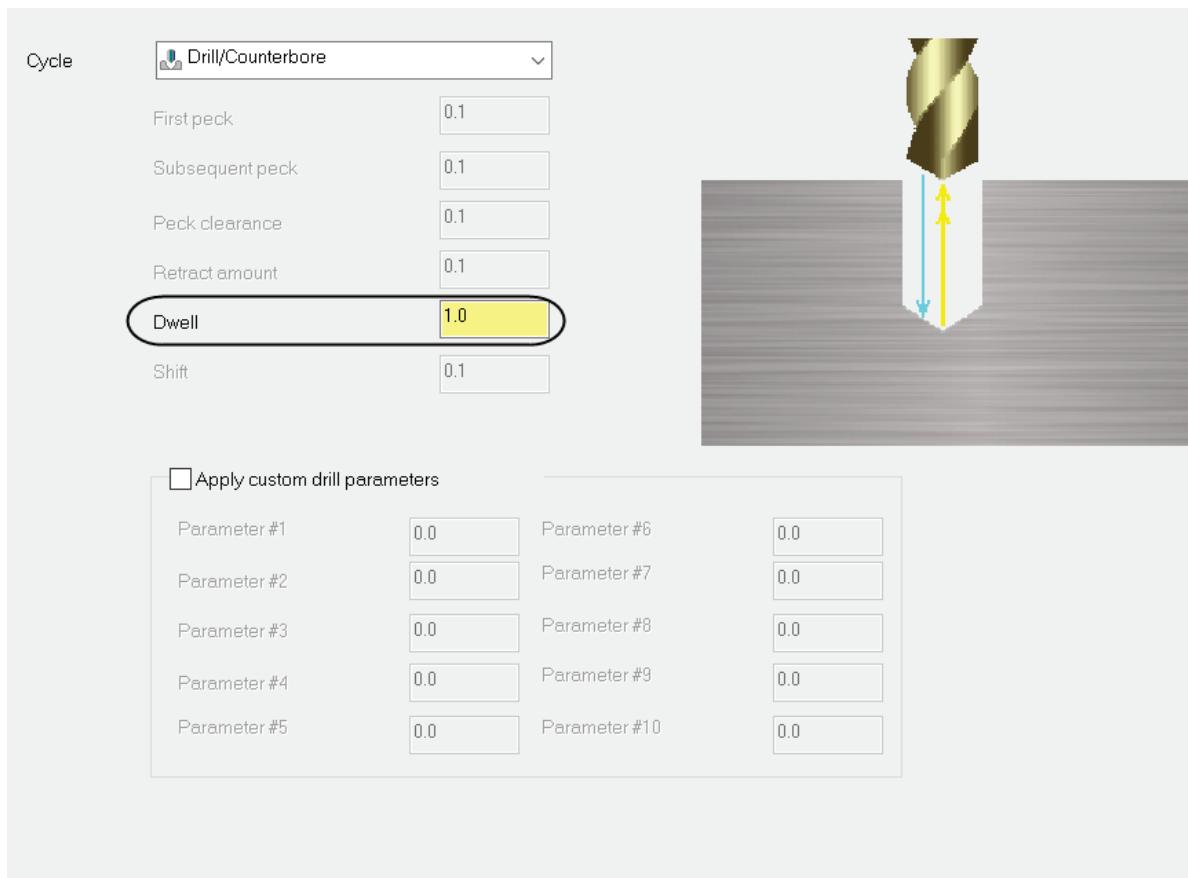




13.2 Set the Cut Parameters

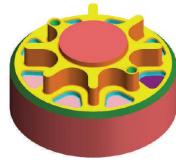
- From the Tree View list, select **Cut Parameters** and make the necessary changes as shown in [Figure: 13.2.1](#).

Figure: 13.2.1



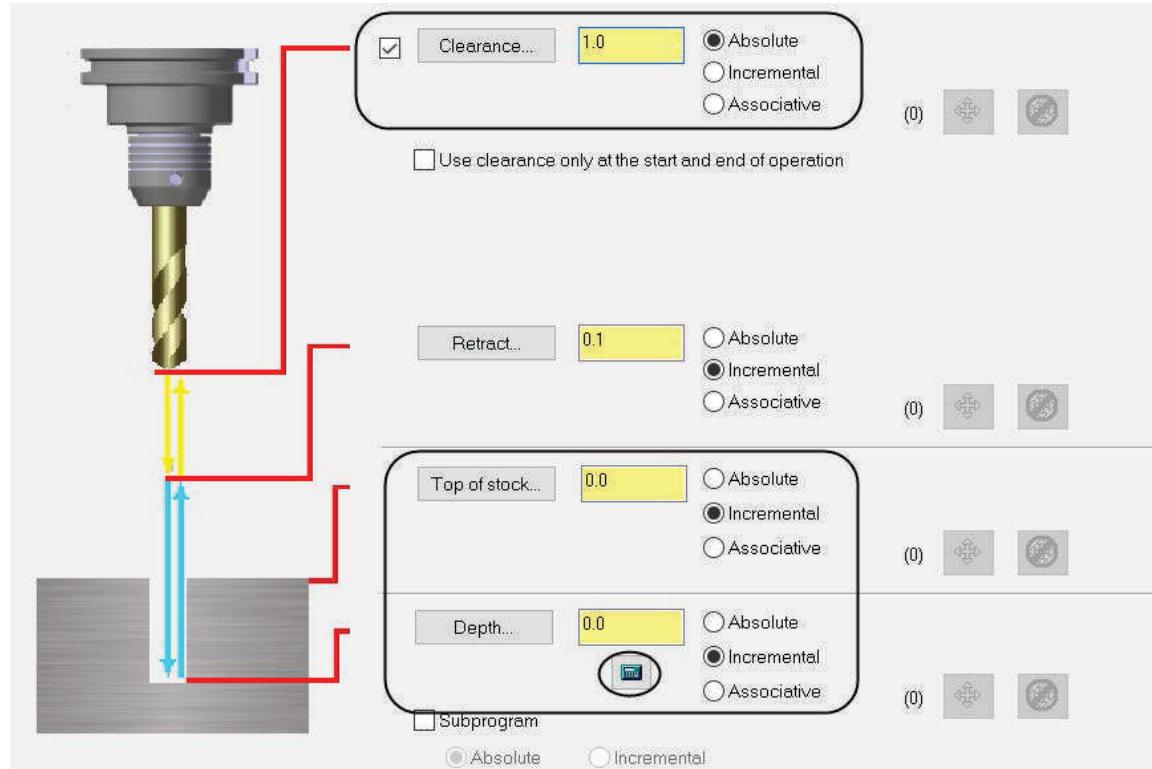
Drill/Counterbore is recommended for drilling holes with depths of less than three times the tool's diameter.

Dwell sets the amount of time in seconds that the tool remains at the bottom of a drilled hole.



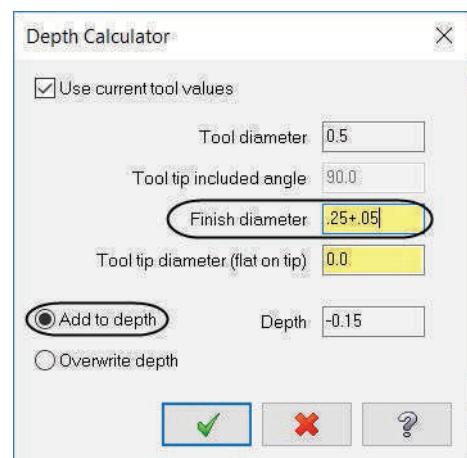
13.3 Set the Linking Parameters

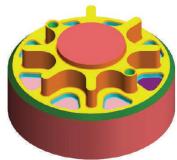
- Choose **Linking Parameters**, ensure **Clearance** is enabled and set to **Absolute** and **1.0** and the **Top of stock** and **Depth** are set to **Incremental** and **0.0**.



- Select the **Calculator** icon.
- To calculate the **Depth** of the spot drill, in the **Finish diameter** area, enter **0.25+0.05** as shown in [Figure: 13.3.1](#).

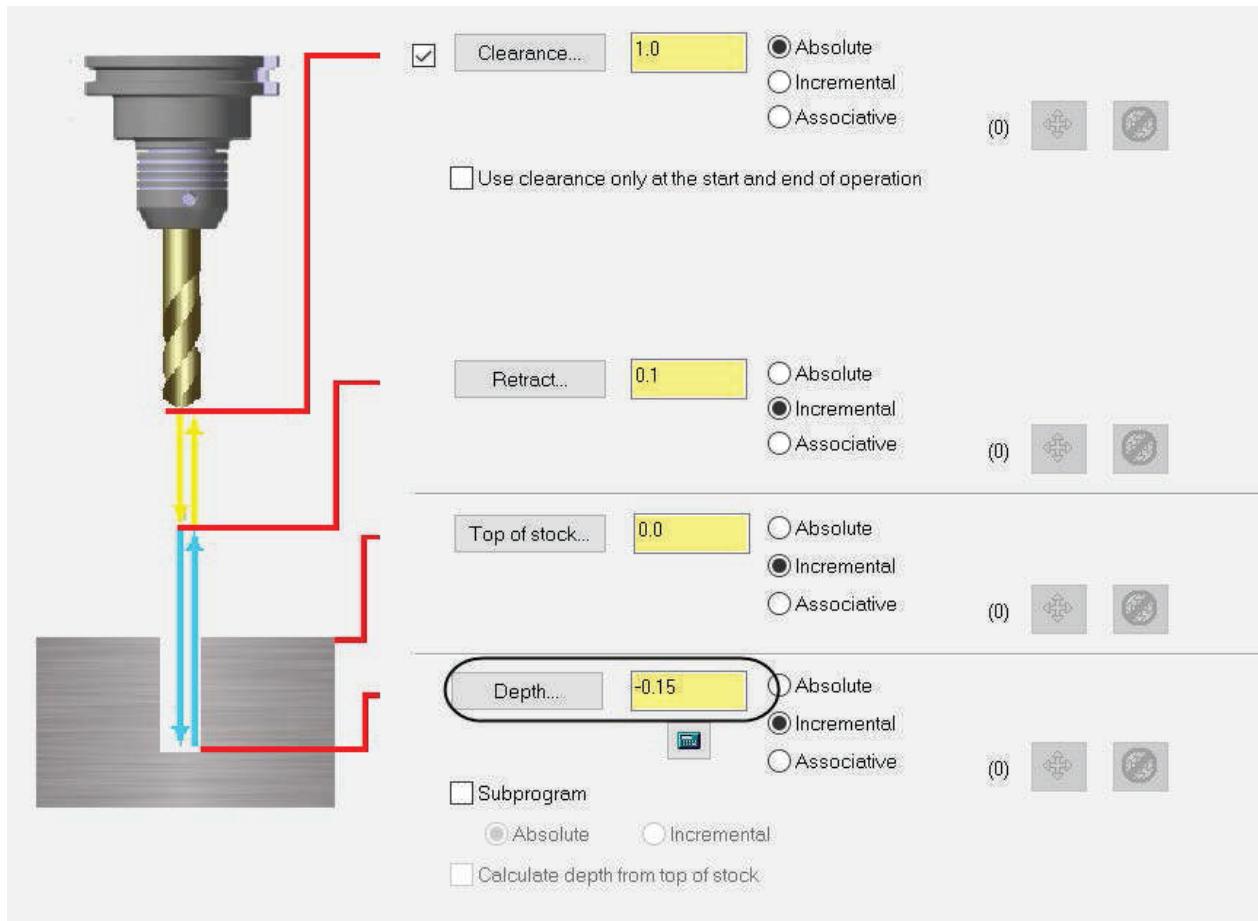
Figure: 13.3.1



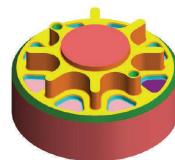


- ♦ Select the **OK** button to exit the **Depth Calculator**. 
- ♦ You will now see the depth we calculated for the spot drilling operation set in the **Depth** field as shown in [Figure: 13.3.2](#).

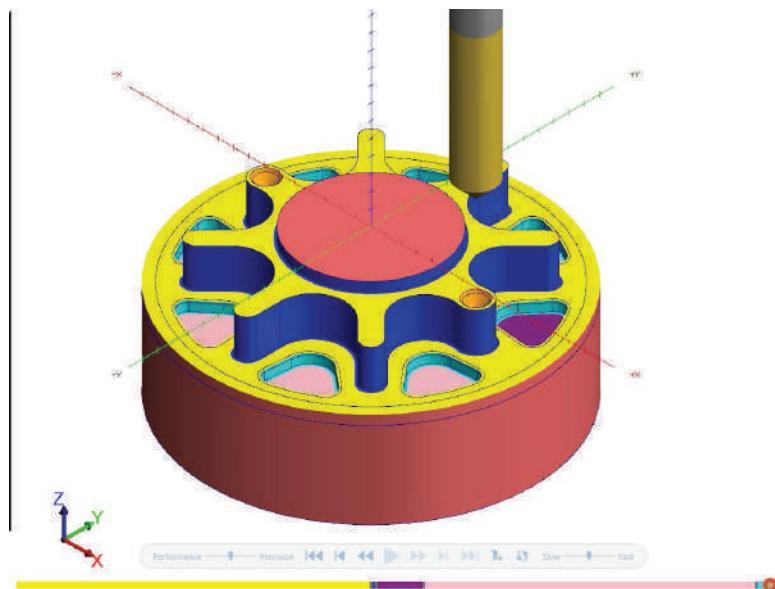
Figure: 13.3.2



- ♦ Select the **OK** button to exit the **2D Toolpaths - Drill/Circles Simple drill - no peck** parameters. 



- ♦ Verify the toolpath as shown on **page 147**.



- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.



STEP 14: DRILL ALL HOLES

In this example, we will drill the 1/4" holes to a specified depth.

Toolpath Preview:



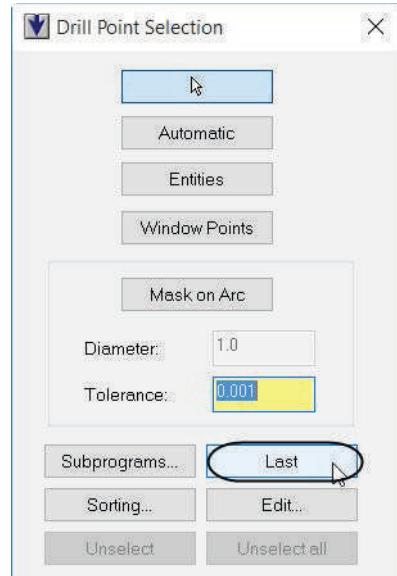
TOOLPATHS

- ♦ From the **2D** group, select **Drill** icon.





- In the **Drill Point Selection** dialog box, choose the option **Last**.



NOTE: This option will automatically select the hole for you based off the selection from the previous drill operation.

- Select the **OK** button in the **Drill Point Selection** panel to accept the 2 drill points.
- In the **Toolpath Type** page, the **Drill** toolpath will be selected.



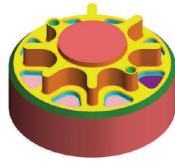
14.1 Select a 1/4" Drill from the Library and set the Tool Parameters

- Select **Tool** from the **Tree View** list.
- Click on the **Select library tool...** button.
- To be able to see just the drill, select the **Filter** button.



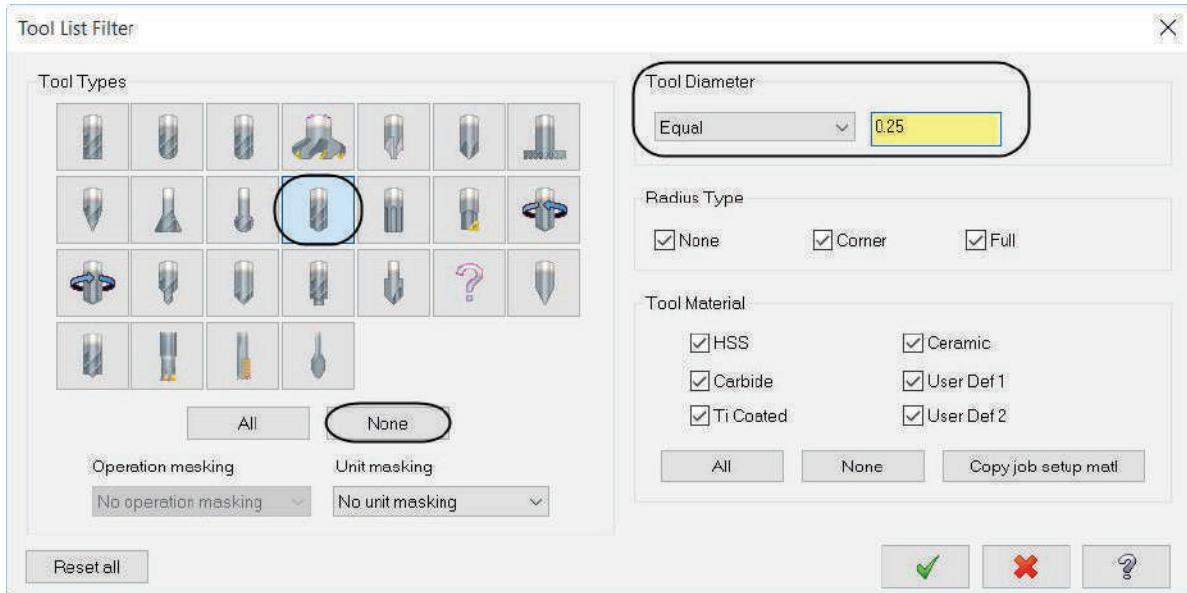
- Under **Tool Types**, select the **None** button and then choose the drill icon.

TUTORIAL #4 DRILL ALL HOLES



- Under Tool Diameter, select Equal and enter 0.25 as shown in [Figure: 14.1.1](#).

Figure: 14.1.1



- Select the OK button to exit the Tool List Filter panel.



- From that list, select the 1/4" Drill.

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type	Ra..
124	--	1/4 DRILL	--	0....	0.0	2.0	2	Drill	No...

- Select the tool in the Tool Selection page and then choose the OK button to exit.





- ♦ Make the necessary changes to the **Tool** page as shown in [Figure: 14.1.2.](#)

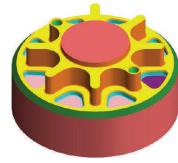
Figure: 14.1.2

#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	L...
1	-	1/2 FLAT ...	-	0.5	0.0	0.0
2	-	3/16 FLA...	-	0.1875	0.0	0.0
3	-	1/2 SPOT...	-	0.5	0.0	0.0
4	-	1/4 DRILL	-	0.25	0.0	0.0

Tool diameter: 0.25
Corner radius: 0.0
Tool name: 1/4 DRILL
Tool #: 4 Length offset: 4
Head #: 0 Diameter offset: 4
Spindle direction: CW
Feed rate: 1.0696 Spindle speed: 534
FPT: 0.001 SFM: 69.8953
Plunge rate: 1.0696 Retract rate: 1.0696
 Force tool change Rapid Retract

Comment:
Drill the holes

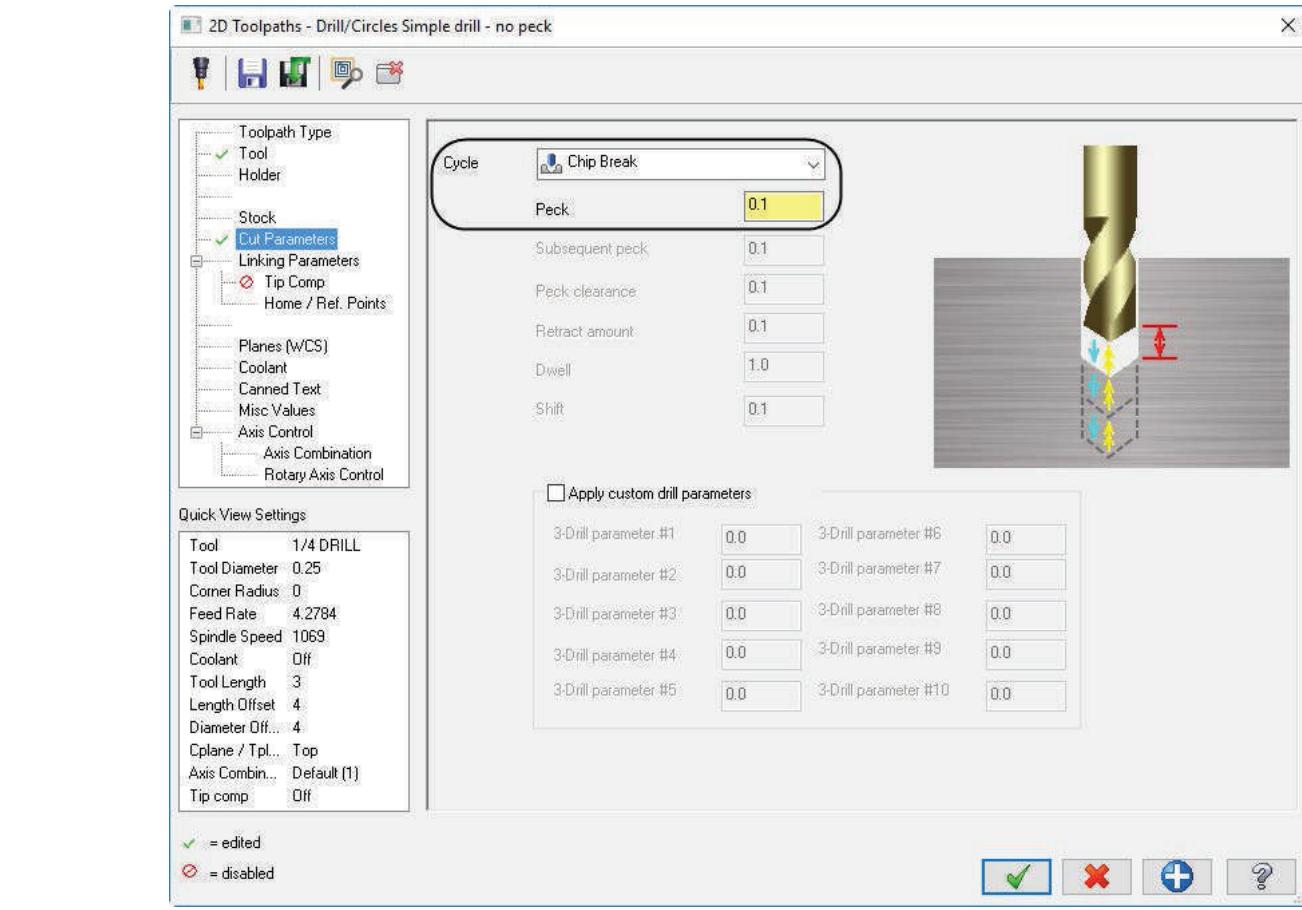
To batch



14.2 Set the Cut Parameters

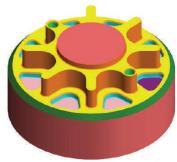
- ♦ Select **Cut Parameters**, change the drill **Cycle** to **Chip Break** and input a **Peck** value of **0.1** as shown in [Figure: 14.2.1](#).

Figure: 14.2.1



Chip Break drills holes with depths of more than three times the tool diameter. The tool retracts partially out of the drilled hole to break material chips.

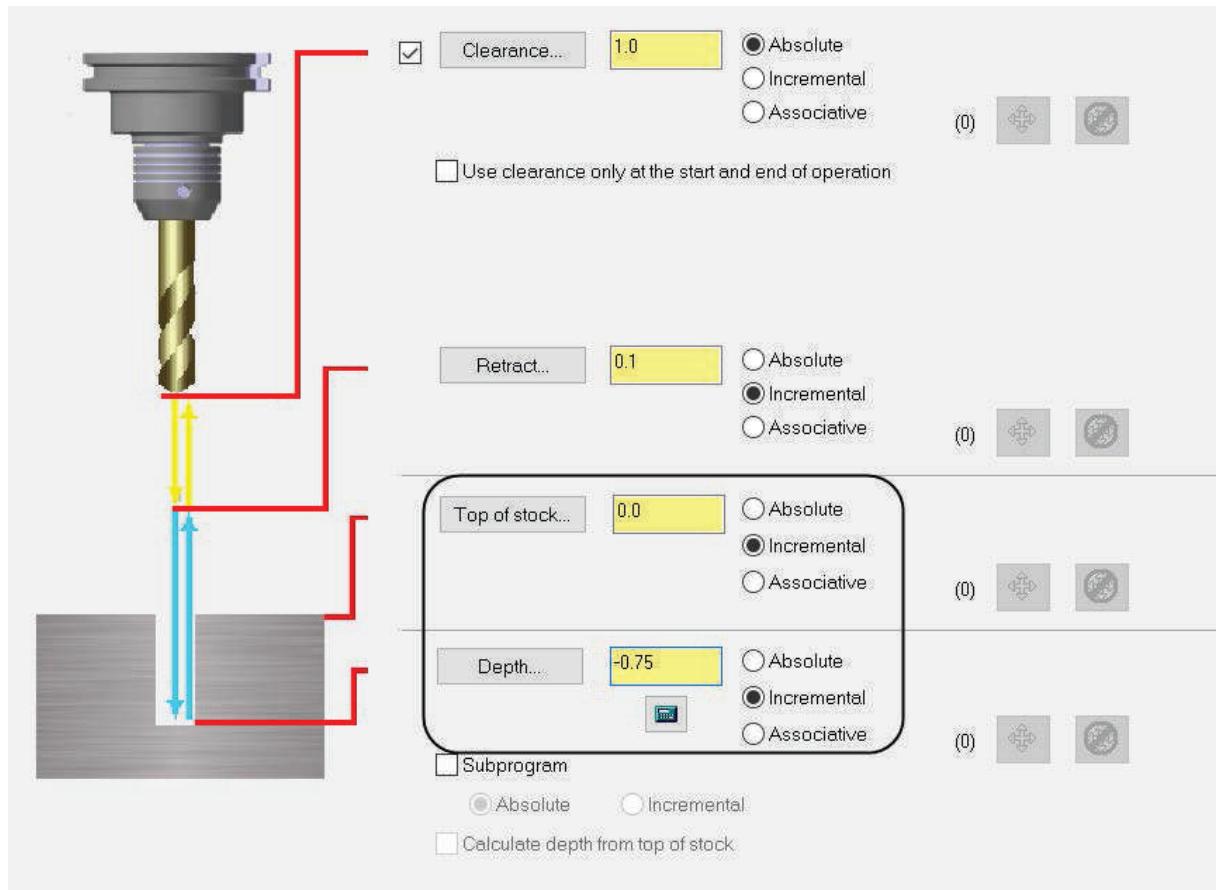
Peck sets the depth for the first peck move which plunges in and out of the material to clear and break chips.

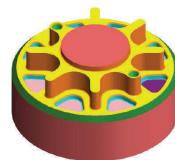


14.3 Set the Linking Parameters

- Choose **Linking Parameters** and input a **Top of Stock** value of **0.0 Incremental** and a **Depth** value of **-0.75 Incremental** as shown in [Figure: 14.3.1](#).

Figure: 14.3.1

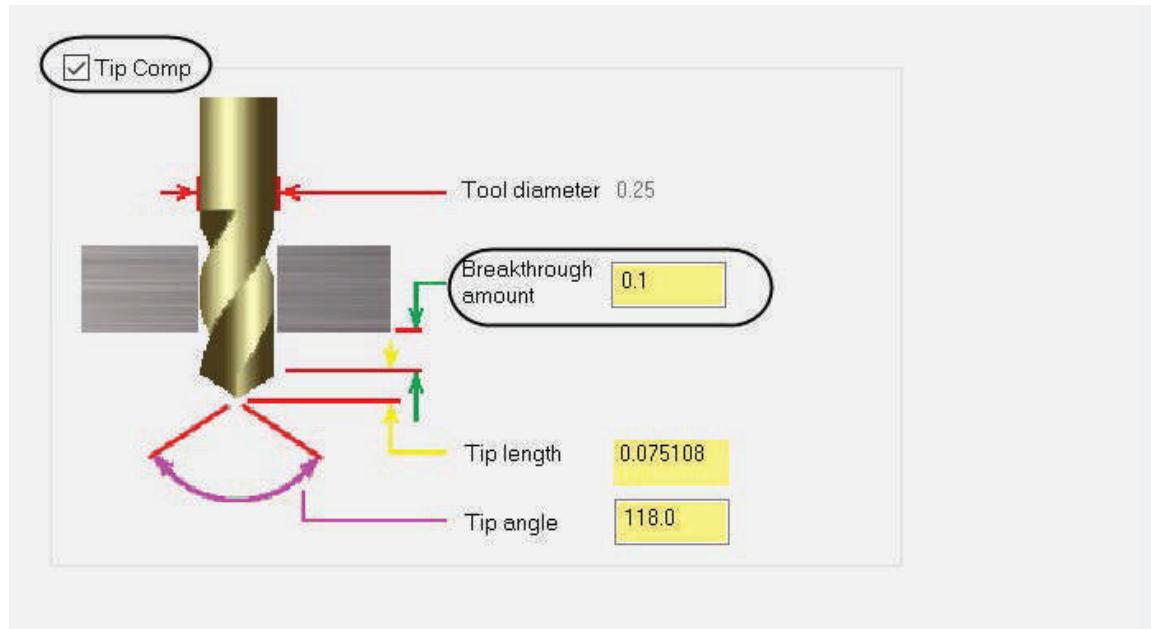




14.4 Set the Tip Compensation page

- ♦ Select **Tip Comp** and enable it.
- ♦ Set the **Breakthrough amount** to **0.1** as shown in [Figure: 14.4.1](#).

Figure: 14.4.1

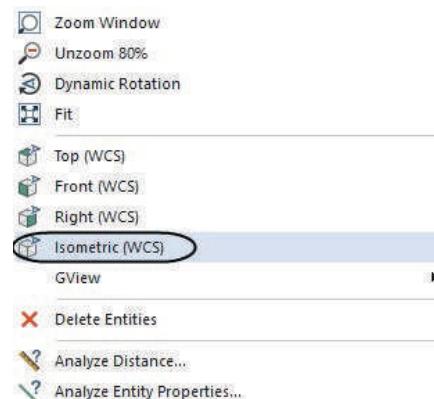


14.5 Preview the Toolpath

- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

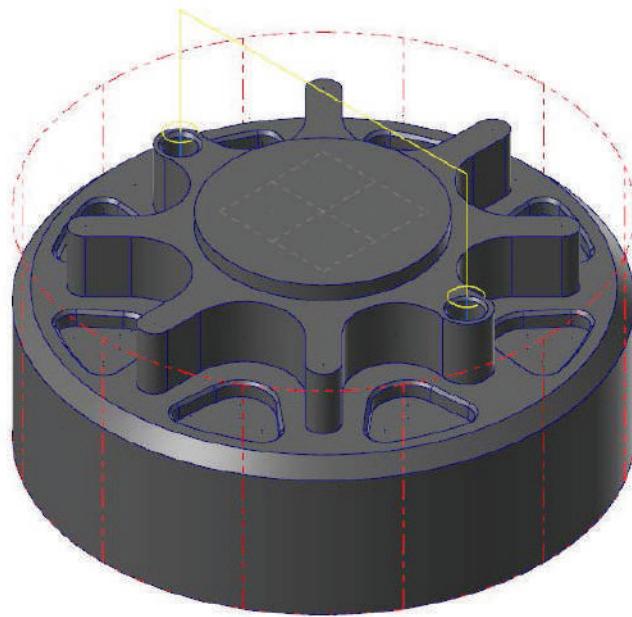


- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.
- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.





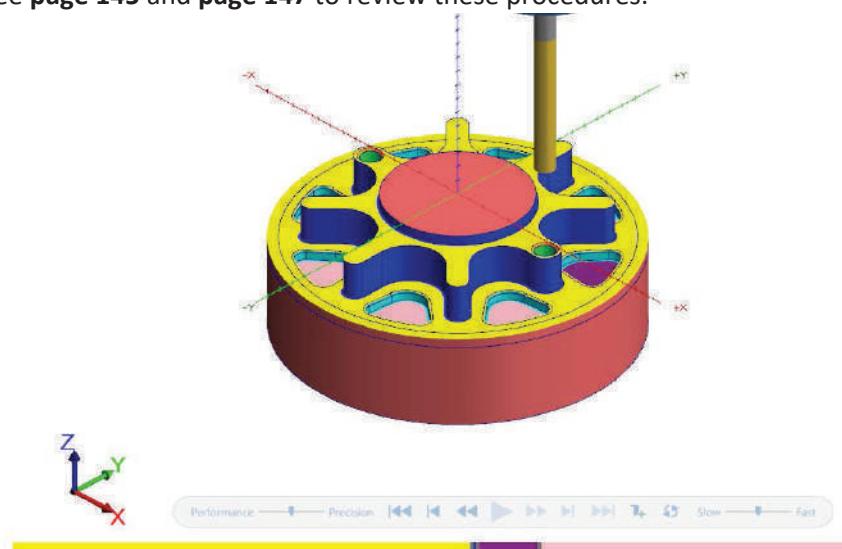
- The toolpath should look as shown.

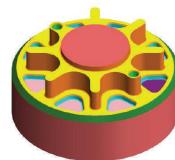


- Press Esc key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- Select the **OK** button to exit the **2D Toolpaths - Drill/Circles Simple drill - no peck** parameters.
- To **Backplot** and **Verify** the toolpaths, see [page 145](#) and [page 147](#) to review these procedures.

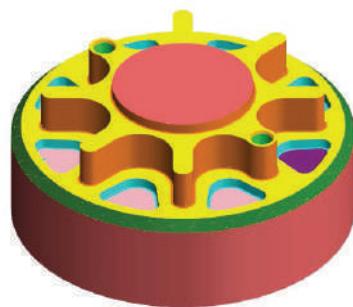




STEP 15: CHAMFER THE OUTSIDE DIAMETER

Contour - Chamfer toolpath automatically cuts a chamfer around a contour using a chamfer mill. The chamfer size you are machining in this step is 0.1" X 45 degrees.

Toolpath Preview:



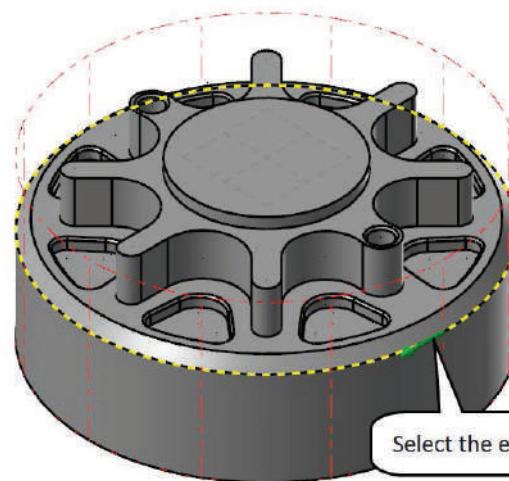
TOOLPATHS

- From the **2D** group, select the **Contour** icon.



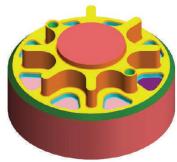
- In the **Chaining** dialog box, make sure that **Wireframe** selection is enabled and leave the defaults. Select the outside circle in **Clockwise** direction as shown in [Figure: 15.0.1](#).

Figure: 15.0.1



NOTE: Use the **Reverse** button to change the direction of the chain if needed.

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



- In the Toolpath Type page, the Contour toolpath will be selected.



15.1 Select a 1/2" Chamfer Mill from the Library and set the Tool Parameters

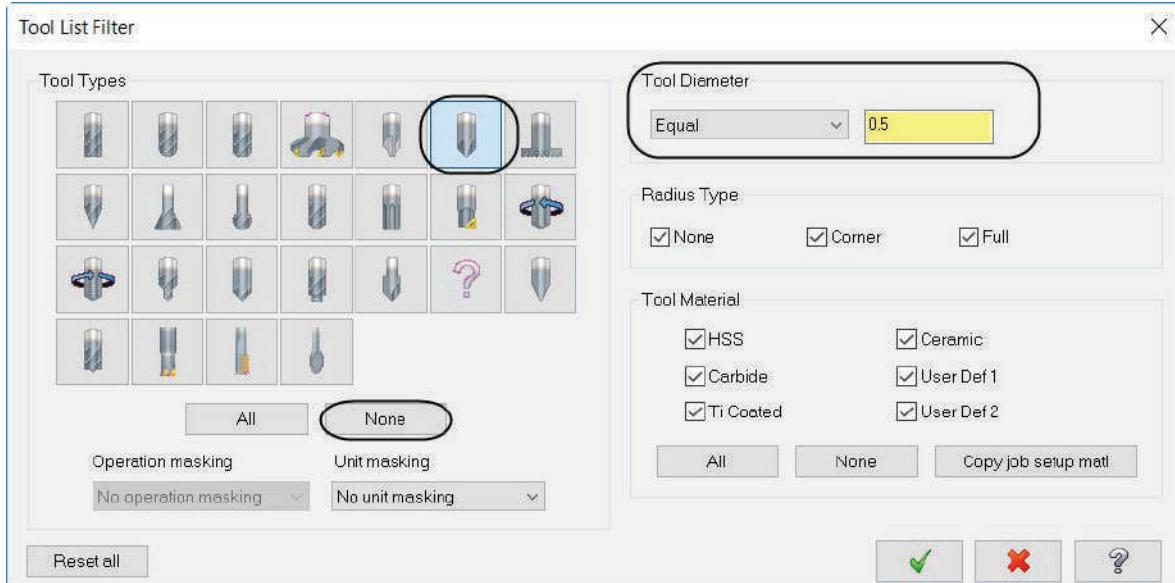
- Select Tool from the Tree View list.

- Click on the Select library tool button.
- To be able to see just the chamfer mill, select the Filter button.



- Under Tool Types, select the None button and then choose the Chamfer Mill icon as shown in [Figure: 15.1.1](#).

Figure: 15.1.1



- Select the OK button to exit the Tool List Filter panel.

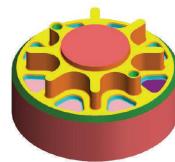
- From that list select the 1/2" Chamfer Mill.

#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	Length	# Flut...	Type	Rad...
319	-	1/2 CHA...	-	0.5-45	0.0	0.75	4	Cha...	None

- Select the tool in the Tool Selection page and then choose the OK button to exit.

TUTORIAL #4

CHAMFER THE OUTSIDE DIAMETER



- Make all the necessary changes as shown in [Figure: 15.1.2.](#)

Figure: 15.1.2

Toolpath Type

Tool

Cut Parameters

- Depth Cuts
- Lead In/Out
- Break Through
- Multi Passes
- Tabs

Linking Parameters

- Home / Ref. Points

Arc Filter/Tolerance

- Planes (WCS)
- Coolant
- Canned Text
- Misc Values
- Axis Control

Quick View Settings

Tool	1/2 CHAMFER...
Tool Diameter	0.5
Corner Radius	0
Feed Rate	6.1609
Spindle Speed	2852
Coolant	Off
Tool Length	2.5
Length Offset	5
Diameter Offset	5
Cplane/Tpla...	Top
Axis Combinat...	Default(1)

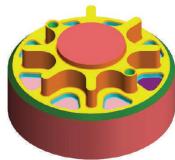
Right-click for options

Select library tool... Filter Active

To batch

Comment:
Chamfer the outside diameter.

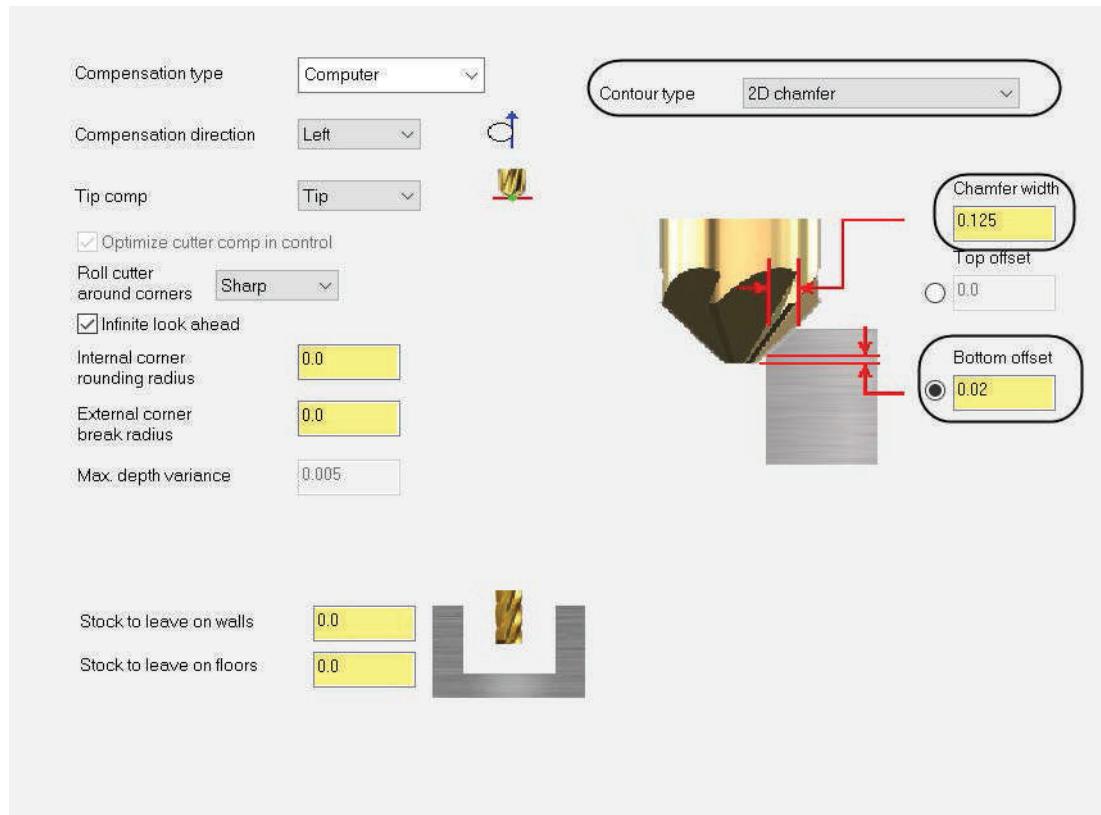
✓ = edited
✗ = disabled



15.2 Set the Cut Parameters

- ♦ Select the **Cut Parameters** page and change the **Contour type** to **2D chamfer**.
- ♦ Input a **Chamfer Width** of **0.125** and a **Bottom offset** of **0.02** as shown in [Figure: 15.2.1](#).

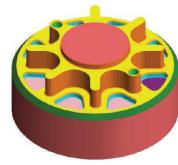
Figure: 15.2.1



2D chamfer cuts chamfers around a contour.

Chamfer Width sets the chamfer width. Mastercam measures the width from the chained geometry adjusted by the cut depths defined on the linking parameters page.

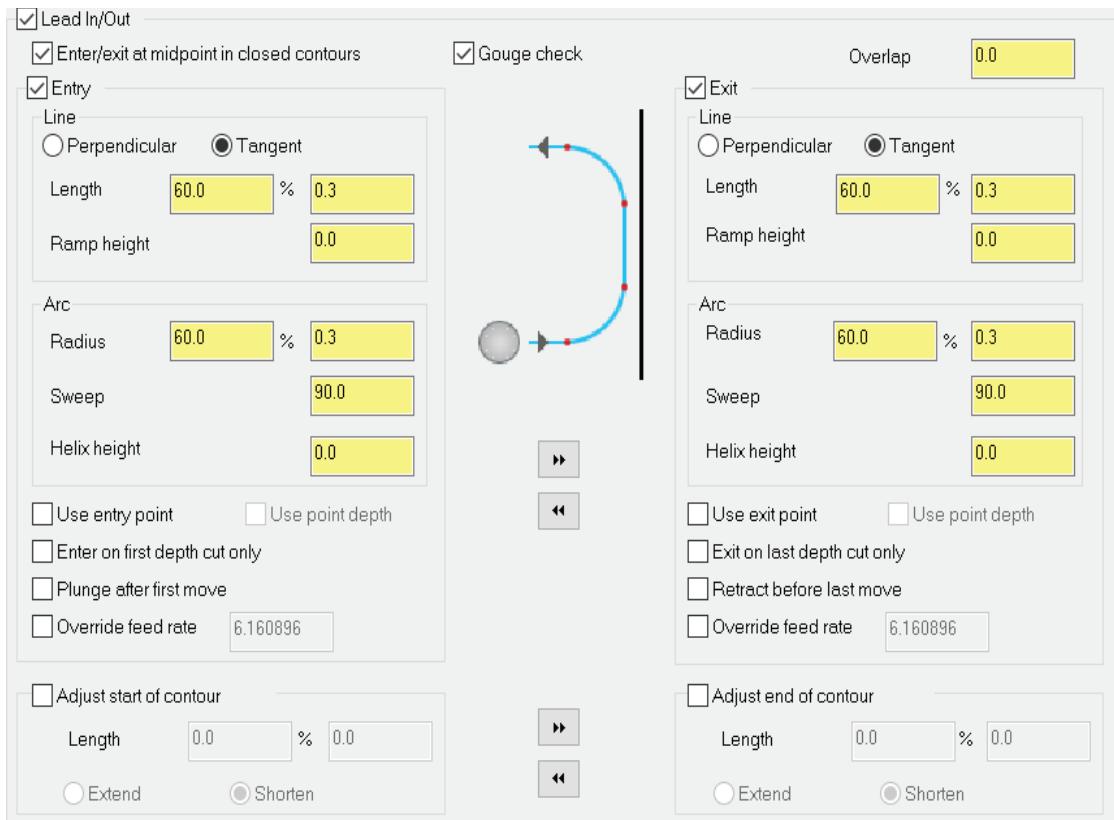
Bottom offset is an amount to ensure that the tip of the tool clears the bottom of the chamfer.



15.3 Set the Lead In/Out Parameters

- Choose the option **Lead In/Out** and input an **Overlap** value.
- Make any other necessary changes as shown in [Figure: 15.3.1](#).

Figure: 15.3.1



Lead In/Out allows you to create either entry moves, exit moves, or both. **Lead In/Out** moves can include both lines and arcs.

Enter/exit at midpoint in closed contours starts and ends a toolpath with closed chains at the midpoint of the first chained entity.

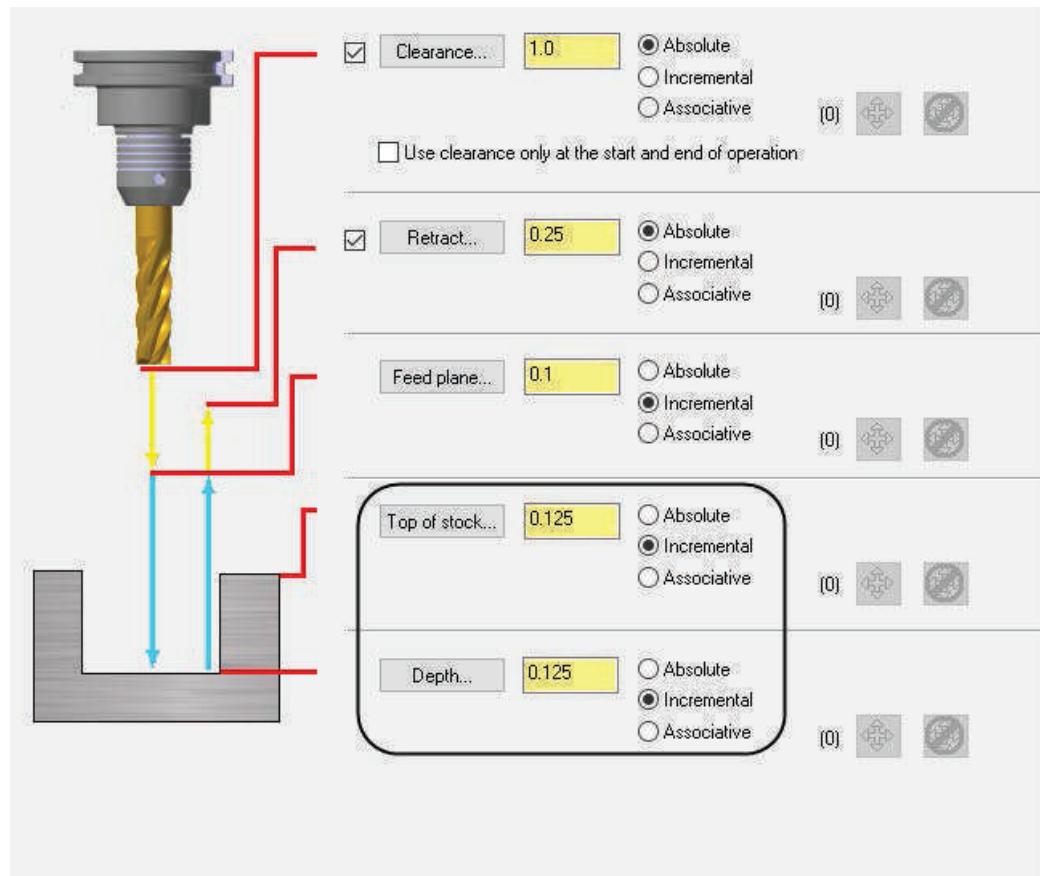
Gouge check ensures that the entry/exit moves do not gouge the part. If the entry/exit moves cause a gouge, they are removed from the toolpath.



15.4 Set the Linking Parameters

- Select the **Linking Parameters** from the **Tree View list**. Set the **Top of stock** to **0.125 Incremental** and the **Depth** to **0.125 Incremental** as shown in [Figure: 15.4.1](#).

Figure: 15.4.1



NOTE: The depth of the chamfer is based on the width and tip offset set in the **Cut Parameters** page. Because we chained the bottom of the chamfer we had to add the chamfer height as incremental value to both **Top of the stock** and **Depth** for Mastercam to make the proper calculation.

15.5 Preview the Toolpath

- To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

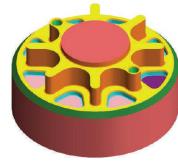


- To hide the dialog box, click on the **Hide dialog** icon as shown.

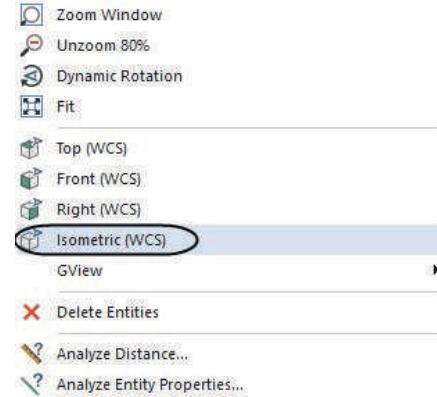


TUTORIAL #4

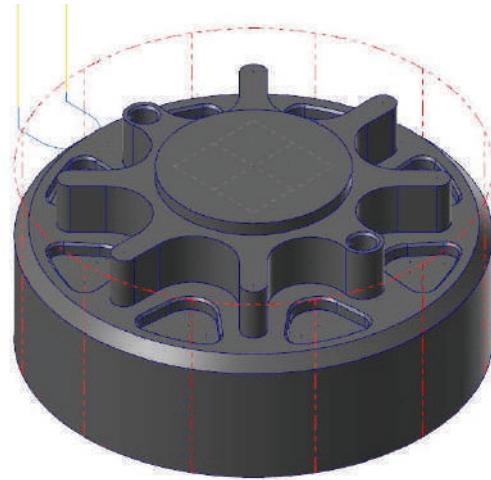
CHAMFER THE OUTSIDE DIAMETER



- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.

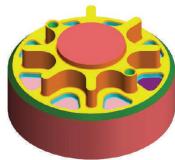


- ♦ Press **Esc** key to exit the preview.

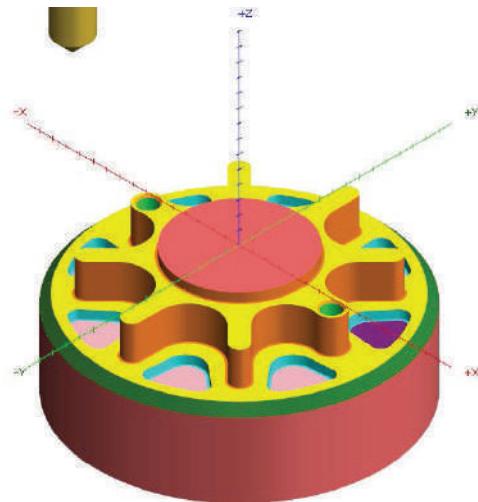
NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to generate the toolpath.





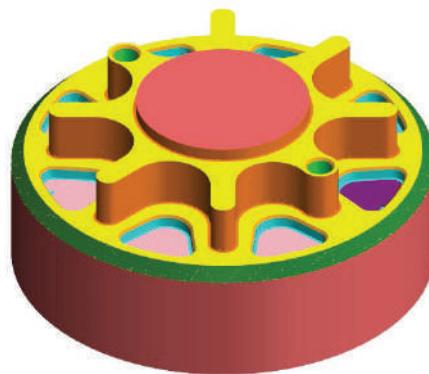
- ♦ To Backplot and Verify your toolpath, see page 145 and page 147 to review these procedures.



STEP 16: CHAMFER THE POCKETS

Contour - Chamfer toolpath automatically cuts a chamfer around a contour using a chamfer mill. The chamfer size you are machining in this step is 0.025" X 45 degrees.

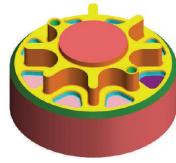
Toolpath Preview:



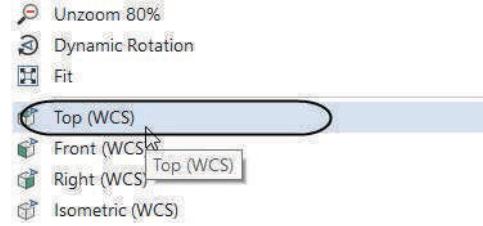
TOOLPATHS

- ♦ From the **2D** group, select the **Contour** icon.



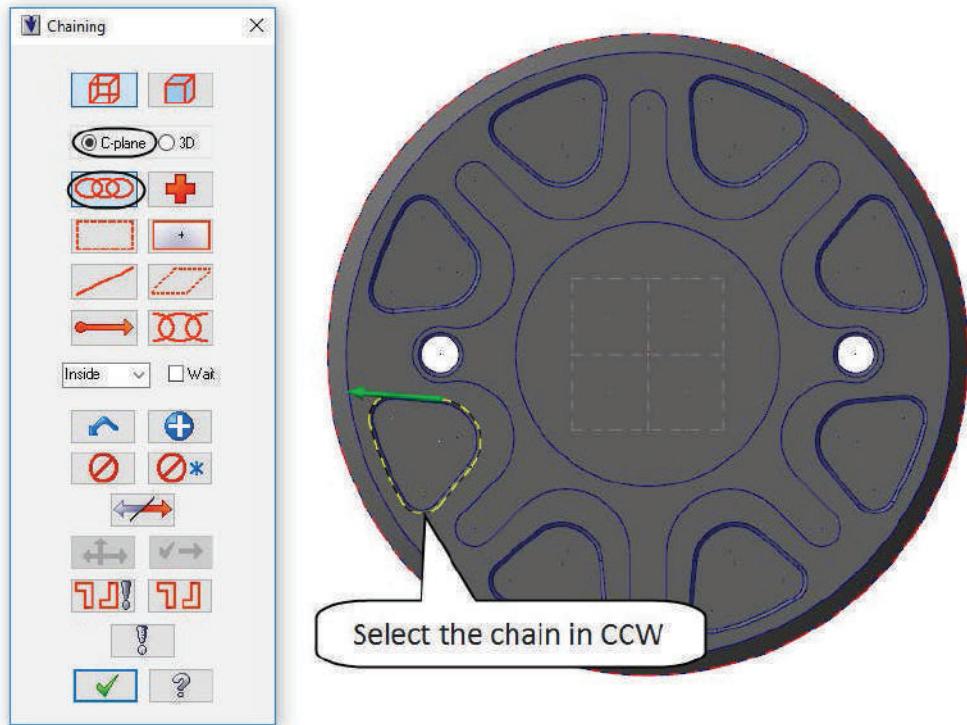


- Right mouse click in the graphic view and change the graphic view to **Top**.

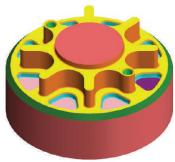


- [Select Contour chain 1]: In the **Chaining** dialog box, make sure that **Wireframe** selection and **C-plane** are enabled and leave the defaults as shown in [Figure: 16.0.1](#). Select the top of the pocket chamfer in the **Clockwise** direction as shown.

Figure: 16.0.1



NOTE: Use the **Reverse** button to change the direction of the chain if needed. 



- ♦ [Select Contour chain 2]: To select the rest of the pocket chamfers, select the **Chain Features** button in the **Chaining** dialog box.

NOTE: All the top chains of the chamfers should be selected.

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



- ♦ In the **Toolpath Type** page, the **Contour** toolpath will be selected.



16.1 Select a 1/4" Chamfer Mill from the Library and set the Tool Parameters

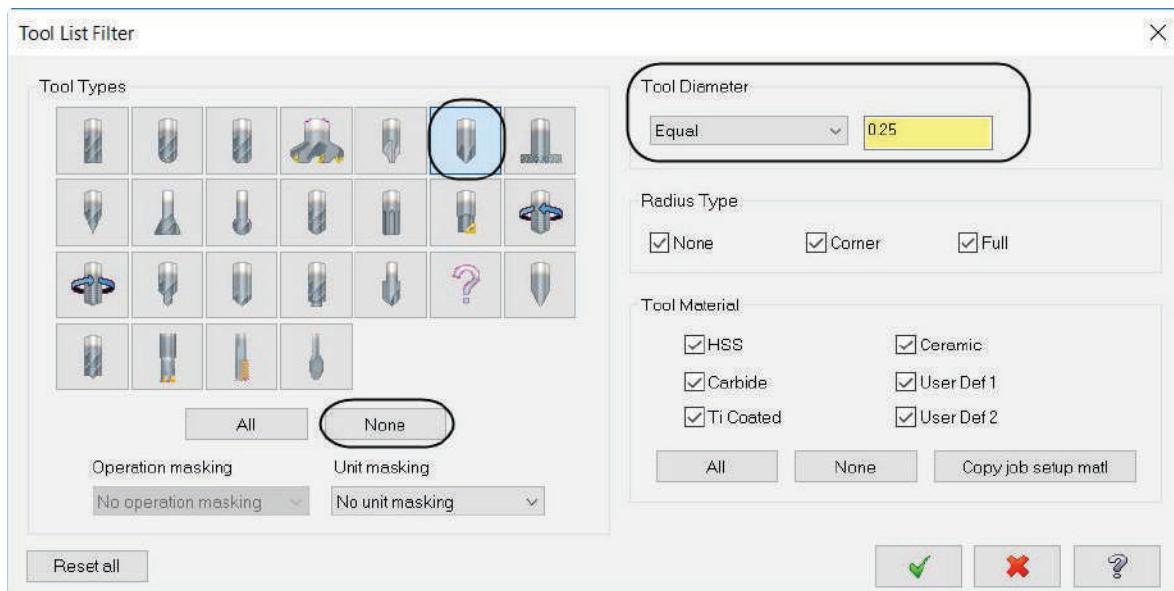
- ♦ Select **Tool** from the **Tree View** list.

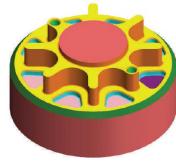
- ♦ Click on the **Select library tool...** button.
- ♦ To be able to see just the chamfer mill, select the **Filter** button.



- ♦ Under **Tool Types**, select the **None** button and then choose the **Chamfer Mill** icon as shown in [Figure: 16.1.1](#).

Figure: 16.1.1





- ♦ Select the **OK** button to exit the **Tool List Filter** panel.
- ♦ Select the tool in the **Tool Selection** page and then choose the **OK** button to exit.

#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	Length	# Flut...	Type	Rad...
318	-	1/4 CHAMFER MILL	-	0.25-45	0.0	0.5	4	Cha...	None

- ♦ Make all the necessary changes as shown in [Figure: 16.1.2](#).

Figure: 16.1.2

#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	Length	# Flut...	Type	Rad...
1	-	1/2 FLAT ...	-	0.5	0.0				
2	-	3/16 FLA...	-	0.1875	0.0				
3	-	1/2 SPOT...	-	0.5	0.0				
4	-	1/4 DRILL	-	0.25	0.0				
5	-	1/2 CHA...	-	0.5-45	0.0				
6	-	1/4 CHA...	-	0.25-...	0.0				

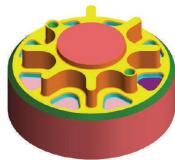
Tool diameter: 0.25
 Corner radius: 0.0
 Tool name: 1/4 CHAMFER MILL
 Tool #: 6 Length offset: 6
 Head #: 0 Diameter offset: 6

Spindle direction: CW
 Feed rate: 8.5568 Spindle speed: 1069
 FPT: 0.002 SFM: 69.9607
 Plunge rate: 8.5568 Retract rate: 8.5568
 Force tool change Rapid Retract

Comment
 Chamfer the pockets.

To batch

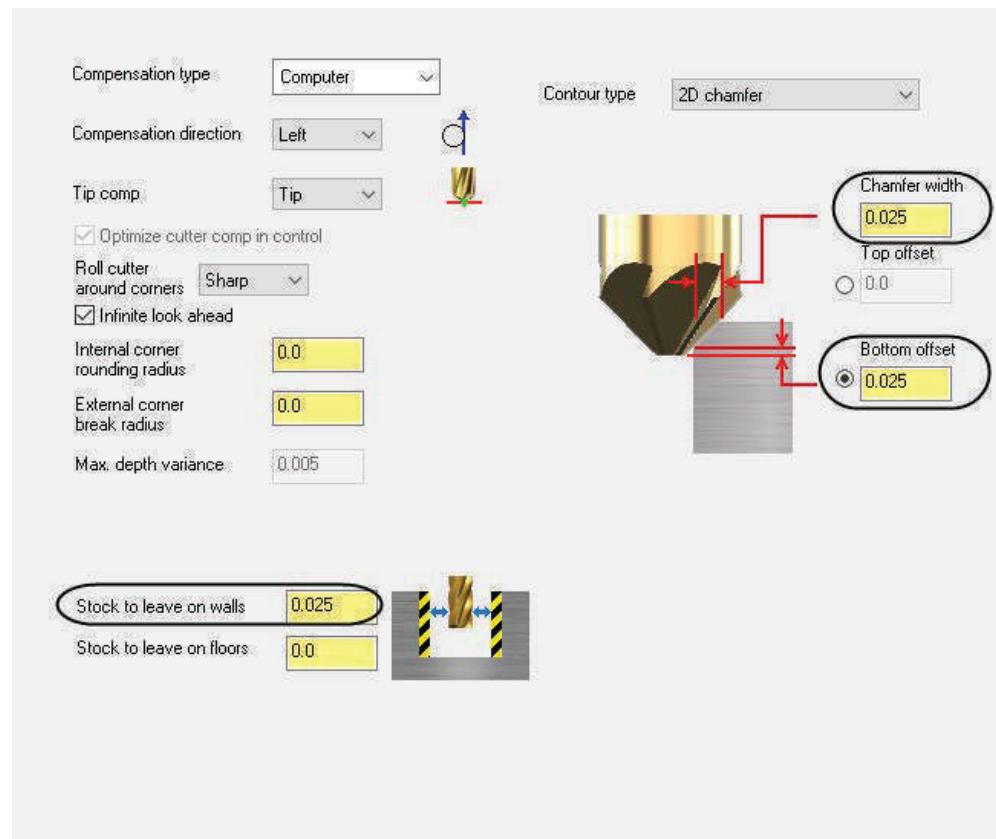
Select library tool... Filter Active



16.2 Set the Cut Parameters

- ♦ Select the **Cut Parameters** page and change the **Contour type** to **2D chamfer**.
- ♦ Input a **Width** of **0.025** and a **Tip offset** of **0.025** as shown in [Figure: 16.2.1](#).
- ♦ Because we selected the outside of the chamfer chains, to move the contact point to the geometry from where Mastercam calculates the chamfer depth, in the **Stock to leave on walls**, add the chamfer size as shown in [Figure: 16.2.1](#).

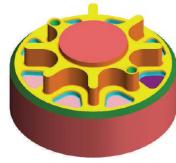
Figure: 16.2.1



2D chamfer cuts chamfers around a contour.

Width sets the chamfer width. Mastercam measures the width from the chained geometry adjusted by the cut depths defined on the linking parameters page. You have to select the geometry without the chamfer for Mastercam to make the proper calculation.

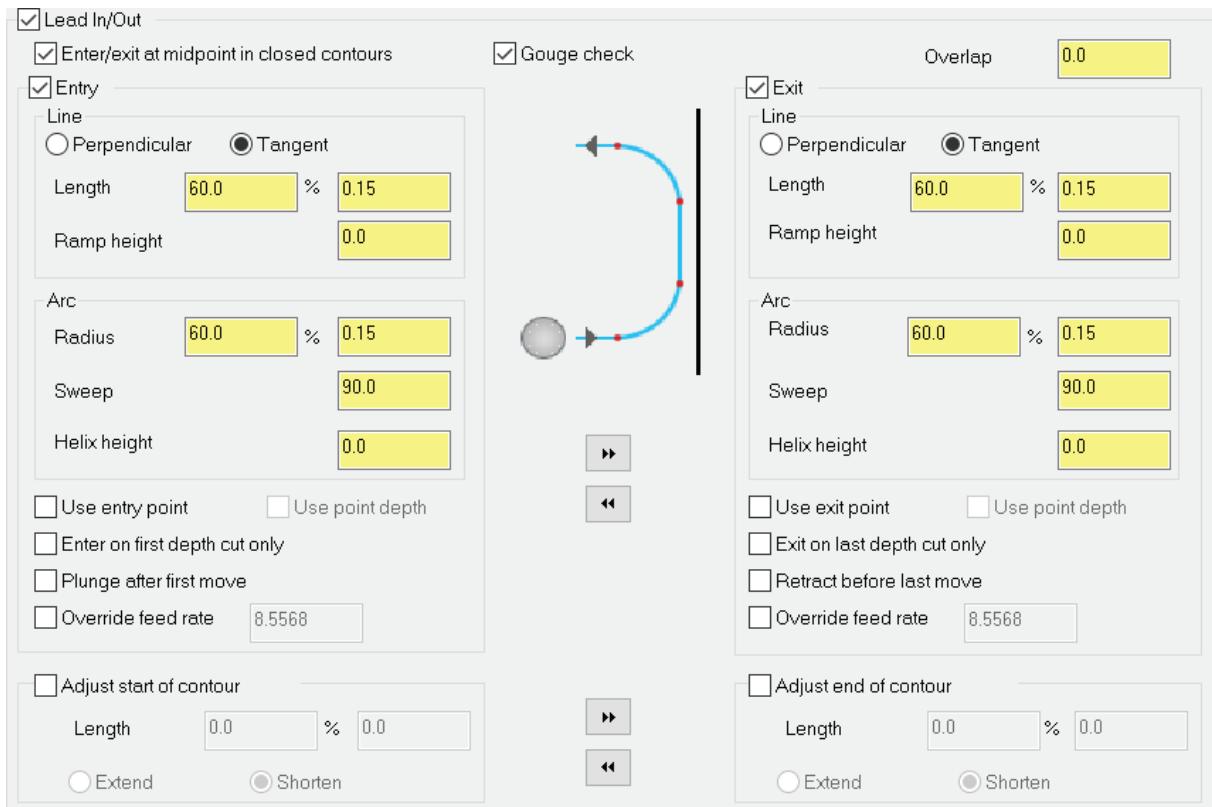
Bottom offset is an amount to ensure that the tip of the tool clears the bottom of the chamfer.



16.3 Set the Lead In/Out Parameters

- ◆ Choose the option **Lead In/Out** and input an **Overlap** value of **0.0**.
- ◆ Make any other necessary changes as shown in [Figure: 16.3.1](#).

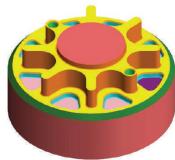
Figure: 16.3.1



Lead In/Out allows you to create either entry moves, exit moves, or both. **Lead In/Out** moves can include both lines and arcs.

Enter/exit at midpoint in closed contours starts and ends a toolpath with closed chains at the midpoint of the first chained entity.

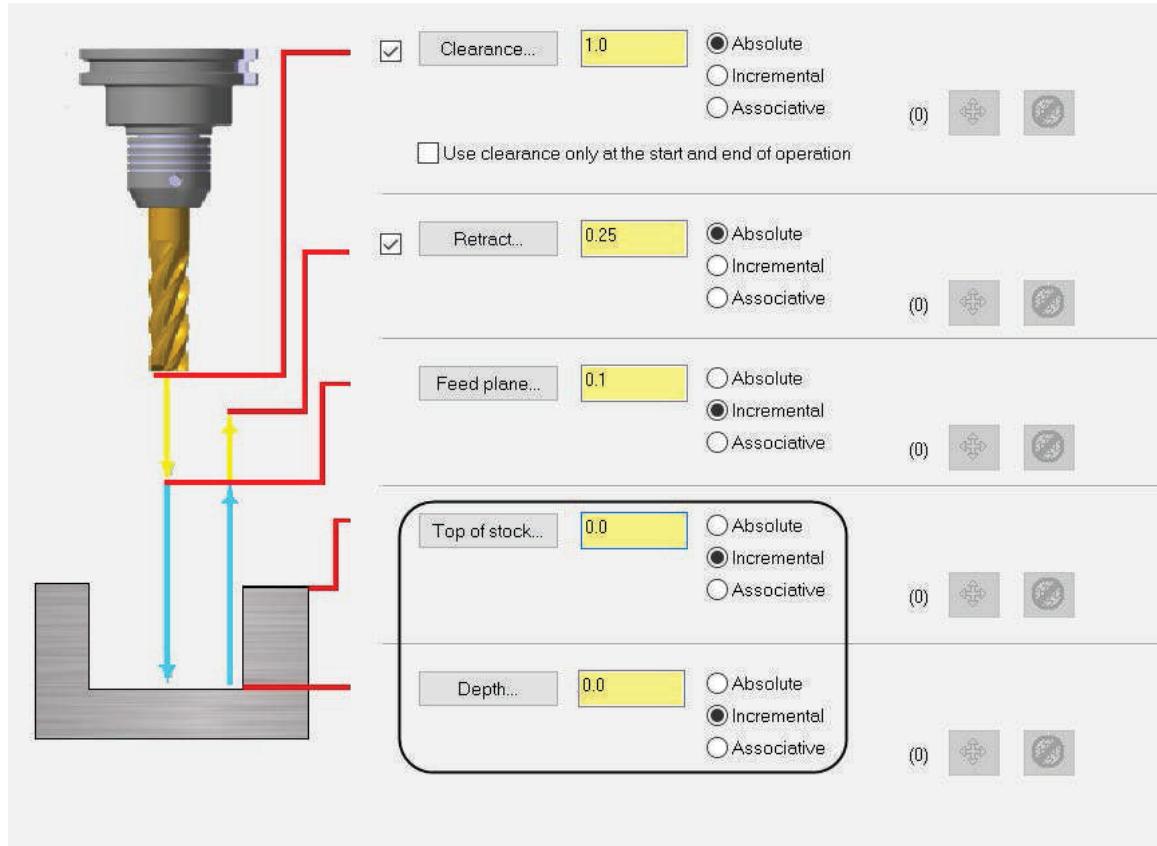
Gouge check ensures that the entry/exit moves do not gouge the part. If the entry/exit moves cause a gouge, they are removed from the toolpath.



16.4 Set the Linking Parameters

- ♦ Select the **Linking Parameters** from the **Tree View list**. Set the **Top of stock** to **0.0 Incremental** and the **Depth** to **0.0 Incremental** as shown in [Figure: 16.4.1](#).

Figure: 16.4.1



NOTE: The depth of the chamfer is based on the width and tip offset set in the **Cut Parameters** page. This is why we set the depth here to zero.

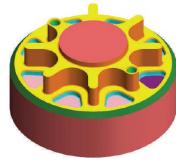
16.5 Preview the Toolpath

- ♦ To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.

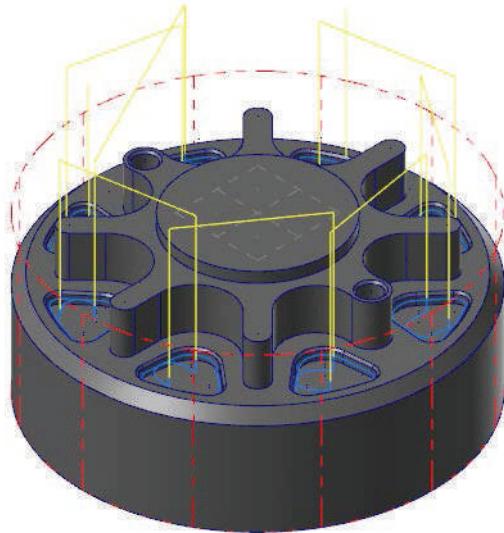




- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.



- ♦ Press **Esc** key to exit the preview.

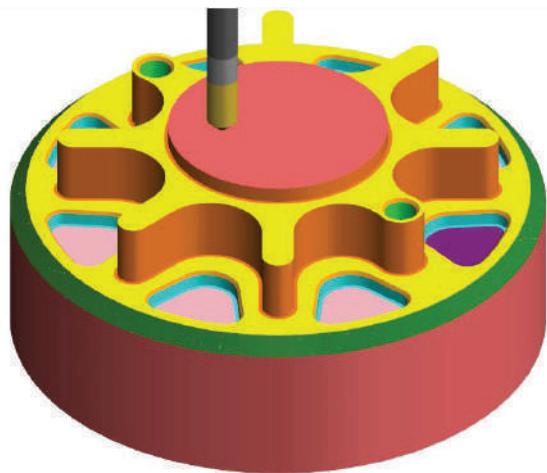
NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to generate the toolpath.



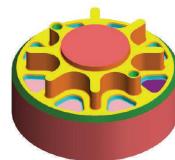


- ♦ To Backplot and Verify the toolpaths, see [page 145](#) and [page 147](#) to review these procedures.



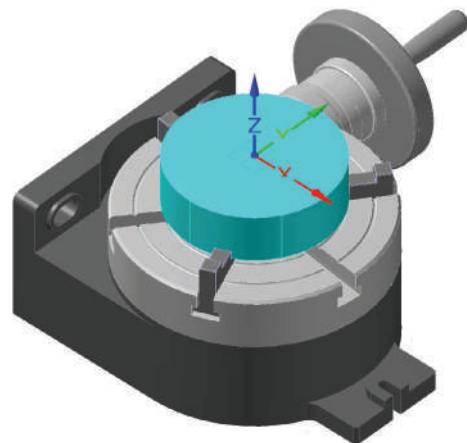
- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.





TOOLPATH CREATION - SETUP 2

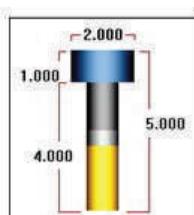
SUGGESTED FIXTURE 2:



NOTE: The part is now flipped over and we will machine the part from the bottom.

SETUP SHEET 2:

TOOL LIST



TYPE: Endmill1 Flat	FLUTE LENGTH: 2.0
DIA OFFSET: 7	OVERALL LENGTH: 4.0
HOLDER: Default Holder	CORNER RAD: 0.0
NUMBER: 7	# OF FLUTES: 4
LENGTH OFFSET: 7	
#7 - 1.0000 ENDMILL1 FLAT - 1 INCH FLAT ENDMILL	

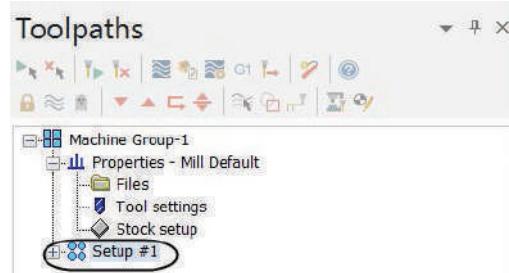


STEP 17: CREATING AND RENAMING TOOLPATH GROUPS

To machine the part in two different setups, we will need to have two separate programs. To be able to post process separately the operations of each setup, we will create them under different toolpath groups with different NC names.

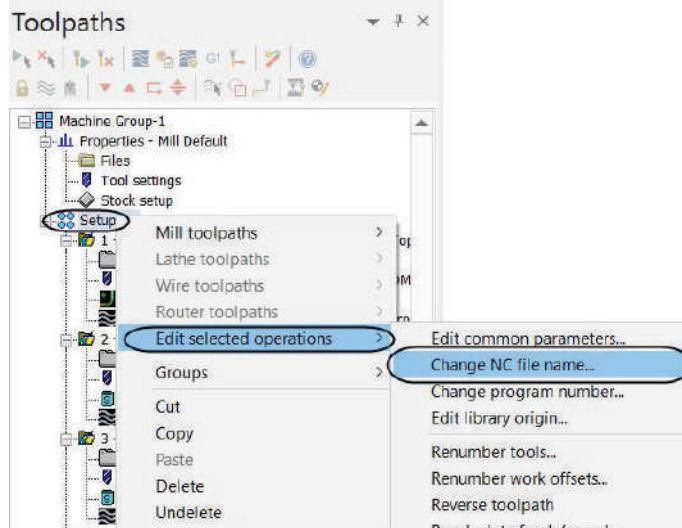
17.1 Rename the current Toolpath Group - 1 and the NC file

- ♦ Click on the **Toolpath Group - 1** to highlight and then click again on it and rename it "**Setup #1**."

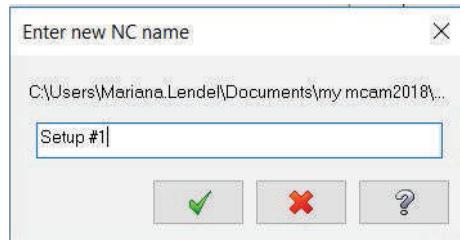


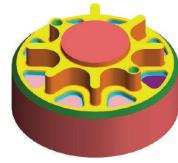
- ♦ Right mouse click on the toolpath group and select **Edit selected operations** and then select **Change NC file name** as shown in [Figure: 17.1.1](#).

Figure: 17.1.1



- ♦ Enter the new NC name: **Setup #1**.

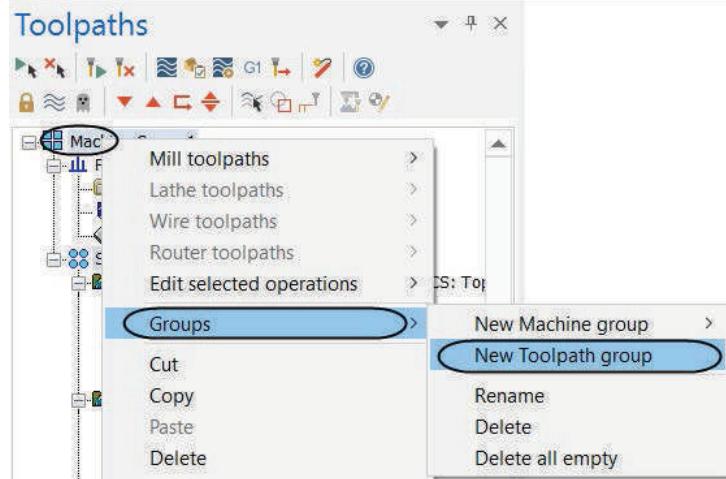




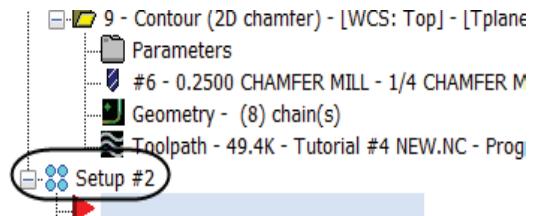
- ♦ Select the **OK** button to accept the new NC name. 

17.2 Create a New Toolpath Group

- ♦ Right mouse click on the **Machine Group-1** and select **Groups** and then the **New Toolpath group**.



- ♦ Rename the toolpath group "**Setup #2**" as shown.



- ♦ Make sure that the **Insert arrow** is below the **Setup #2**, otherwise click on the **Move the insert arrow down an item** icon until the arrow is below the **Setup #2** group.



NOTE: The next operation is going to be generated at the insert arrow location.



STEP 18: SET THE WCS TO BOTTOM

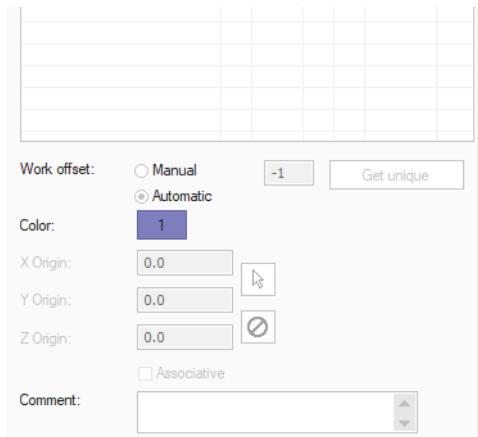
Work coordinate system (WCS) is the active coordinate system in use by Mastercam at any given time. The **WCS** contains the orientation of the **X**, **Y**, **Z** axes plus the location of the zero point (the origin). This tells Mastercam how your part is positioned or oriented in the machine.

Construction plane (Cplane) is the plane in which the geometry is created.

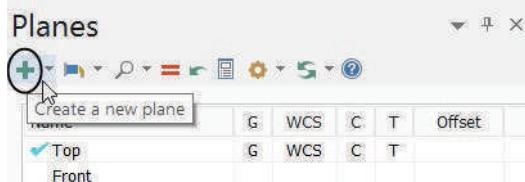
Tool plane (Tplane) is the plane normal to Z or to the vertical tool axis in which the tool moves. When creating a toolpath, both **Cplane** and **Tplane** should be set to the same plane. If the **Tplane** is different then the **WCS**, the post will produce a rotary motion code. By setting the **Cplane**, **Tplane** and **WCS** to one plane, no rotary move will be generated in the code, which is what you want when machining parts with multiple setups.

In this step you are going to create a copy of the **Bottom** plane. This allows you to set a new origin for the plane and set **Z0** to the top of the flipped part.

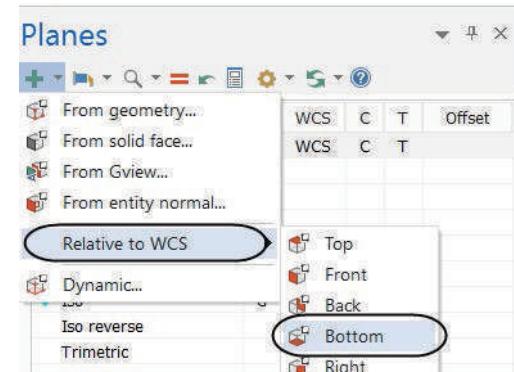
- ♦ Select **Planes** tab located at the bottom left corner.

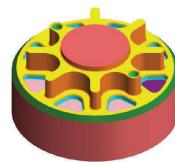


- ♦ To create a new plane based on an existing plane, click on the + sign as shown.

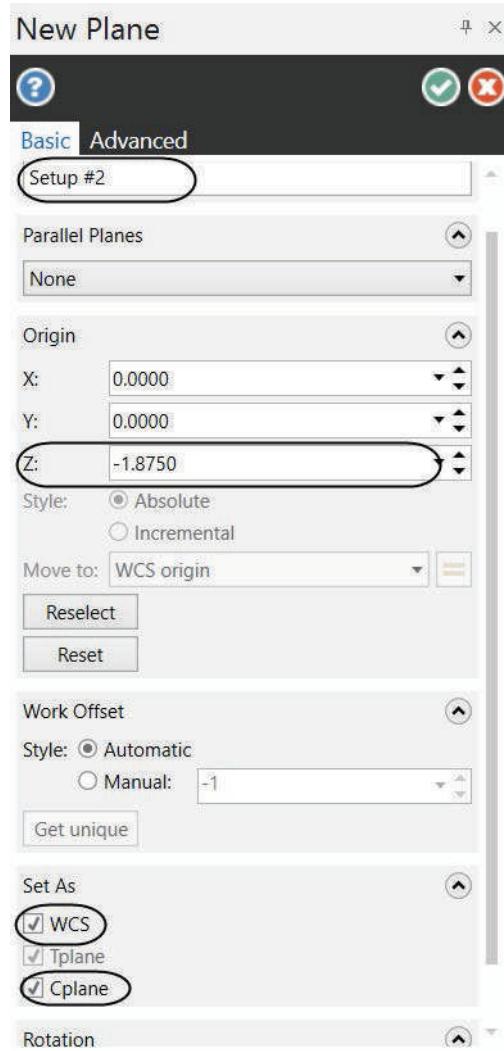


- ♦ To create a copy of the **Bottom** plane, select **Relative to WCS** and select **Bottom** as shown.





- Enter the **Name** and change the **Origin Z** to **-1.875** and enable **Set As WCS** and **Cplane** as shown.



- Select the **OK** button to exit the **New Plane** panel.
- Right mouse click in the graphics window and select the **Isometric** view.

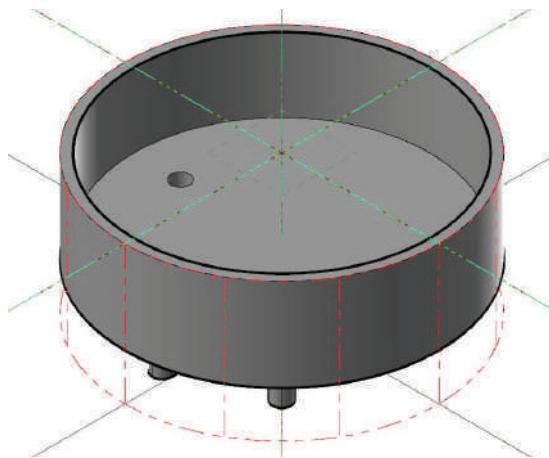




- ♦ Press **F9** on your keyboard to display the coordinate axes.

NOTE: The dark blue axes are the original axes and the light blue axes are the current axes.

- ♦ Your part will appear as shown up to this point.
- ♦ Press **F9** to remove the axes display.
- ♦ Press **Alt + S** to unshade the model.

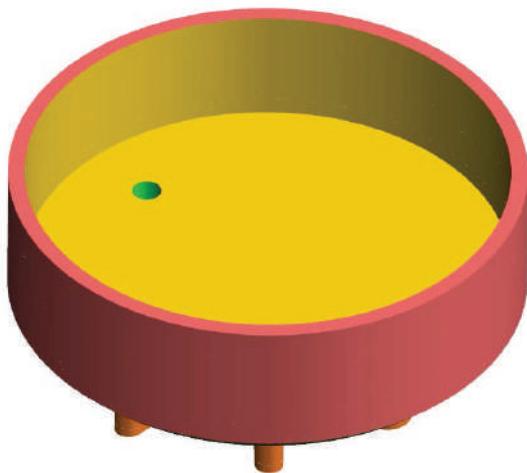


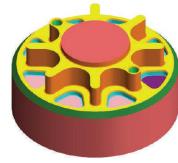
- ♦ Hide the **Plane** panel and open the **Toolpaths** panel.

STEP 19: 2D HS DYNAMIC MILL

In this step we will utilize the 2D High Speed Dynamic Mill toolpath to remove the material in the middle of the part with the part now flipped over.

Toolpath Preview:





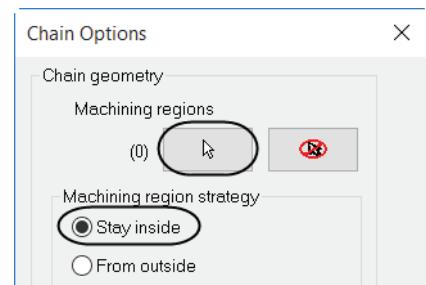
19.1 Chain Selection

TOOLPATHS

- From the **2D** group, select the **Dynamic Mill**.

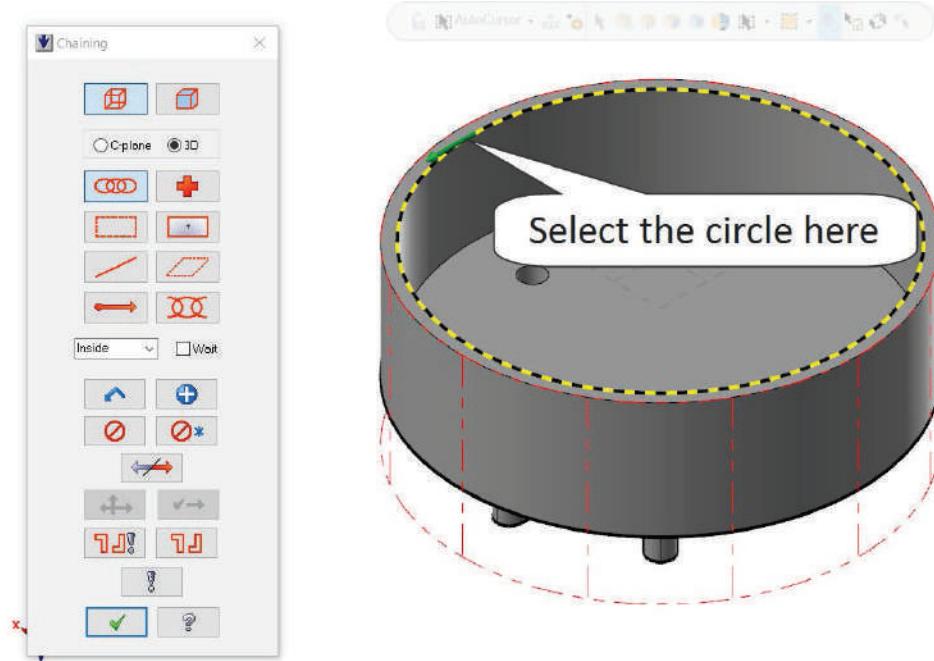


- From the **Chain Options**, make sure that the **Stay inside** is enabled and then click on the **Select machining chains** button as shown.



- Leave the default settings in the **Chaining** dialog box and pick the inner circle as shown in [Figure: 19.1.1](#).

Figure: 19.1.1



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



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



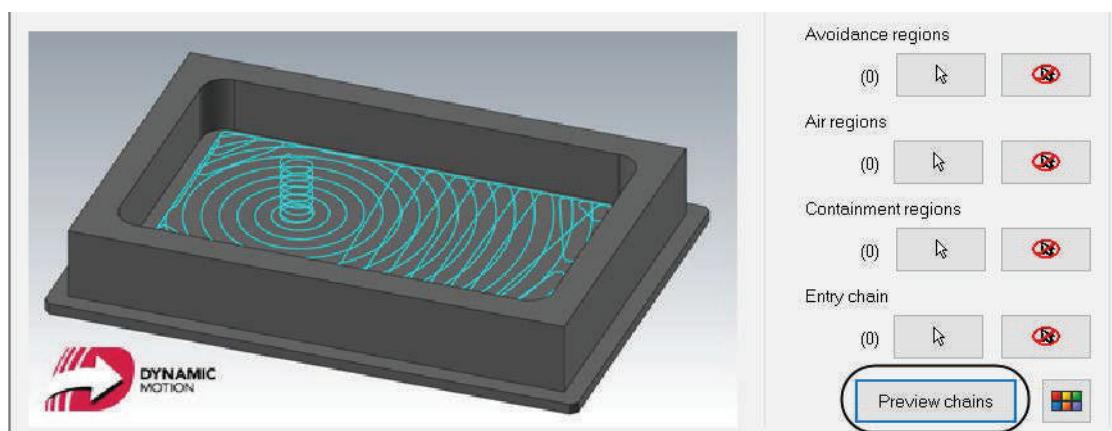


- In the Toolpath Type page, Dynamic Mill will be selected.

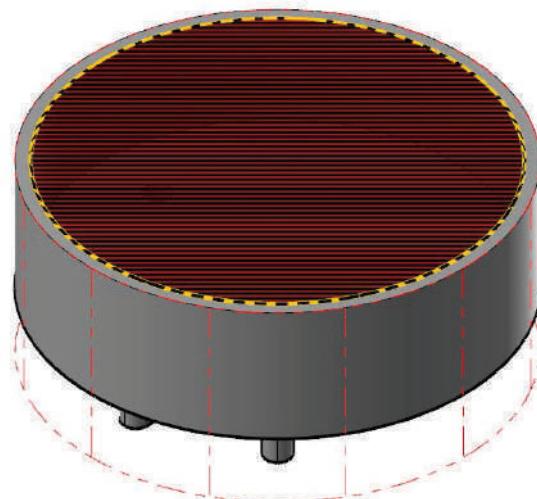


19.2 Preview Chains

- Select the **Preview chains** button as shown.

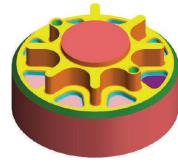


- See page 318 to review the procedure.
- The **Preview chains** should look as shown.



- Press **Esc** key to return to the toolpath parameters.
- Click on the **Preview chains** button again to clear the Preview chains display.





19.3 Select a 1.0" Flat Endmill from the Library and set the Tool parameters

- ♦ Select **Tool** from the **Tree View** list.

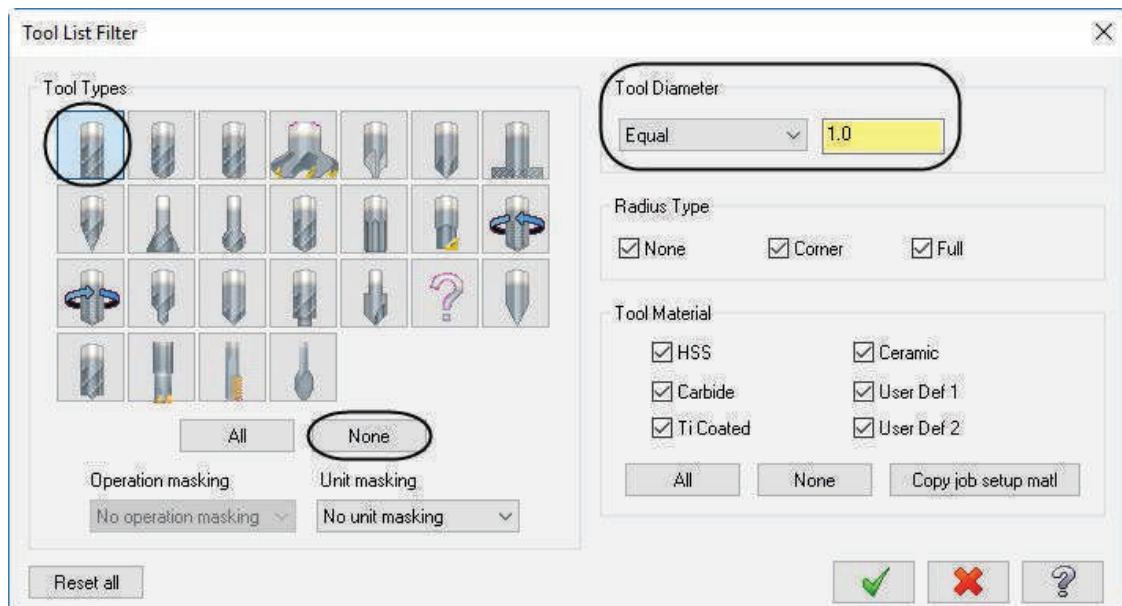
♦ Click on the **Select library tool** button.

- ♦ Select the **Filter** button as shown.



- ♦ Select the **None** button and then under **Tool Types** choose the **Flat Endmill** icon.
- ♦ Under tool diameter, pick **Equal** and input a value **1.0** as shown in [Figure: 19.3.1](#).

Figure: 19.3.1



- ♦ Select the **OK** button to exit the **Tool List Filter**.
- ♦ In the **Tool Selection** panel you should only see **Flat Endmills equal to 1.0"**.

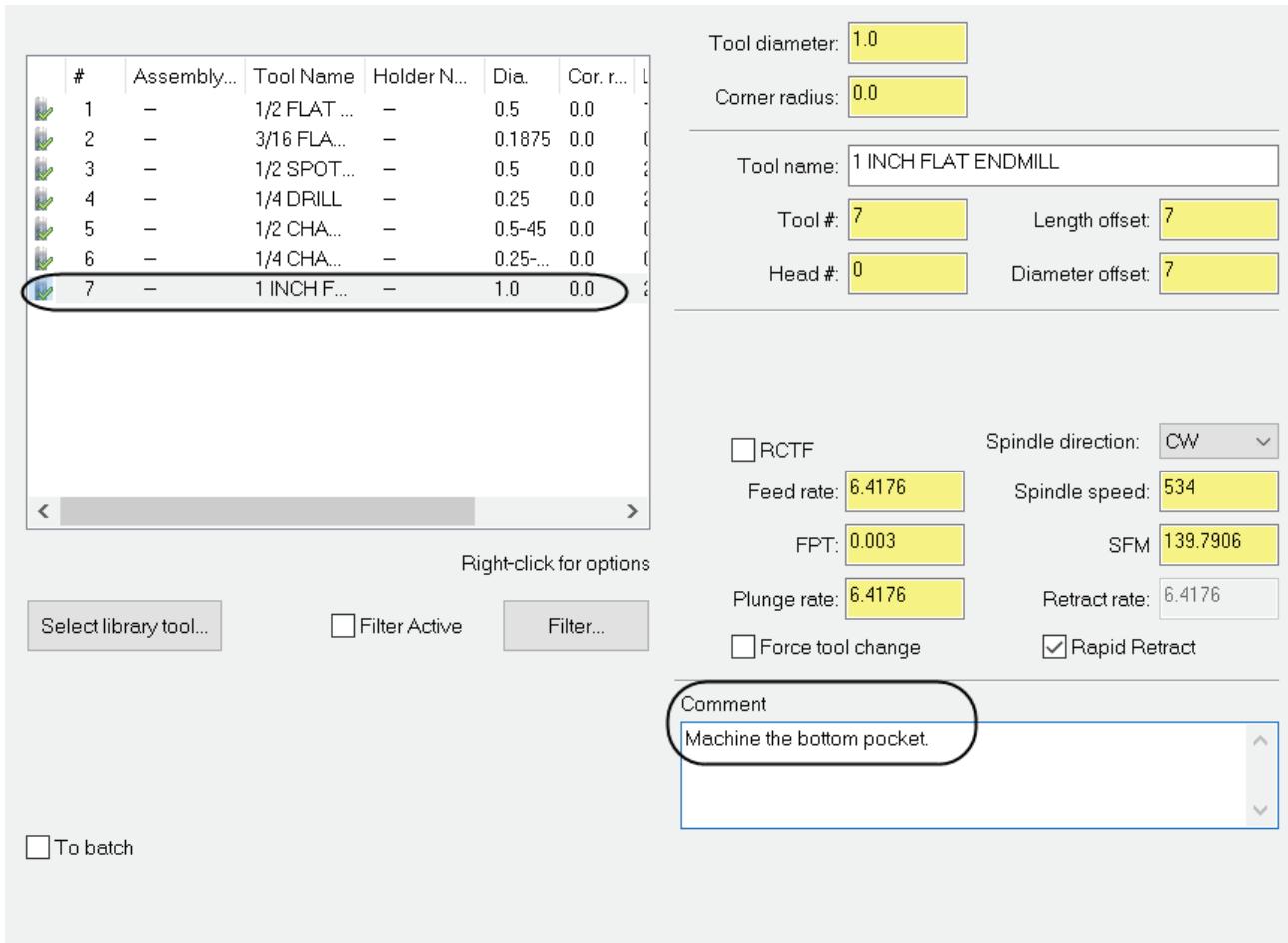
#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	Length	# Flut...	Type	Rad...
297	-	1 INCH FLAT ENDMILL	-	1.0	0.0	2.0	4	End...	None

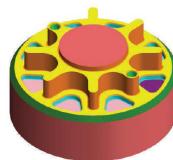
- ♦ Select the **1.0" Flat Endmill** in the **Tool Selection** page.
- ♦ Select the **OK** button to exit.



- Make all the necessary changes as shown in [Figure: 19.3.2.](#)

Figure: 19.3.2



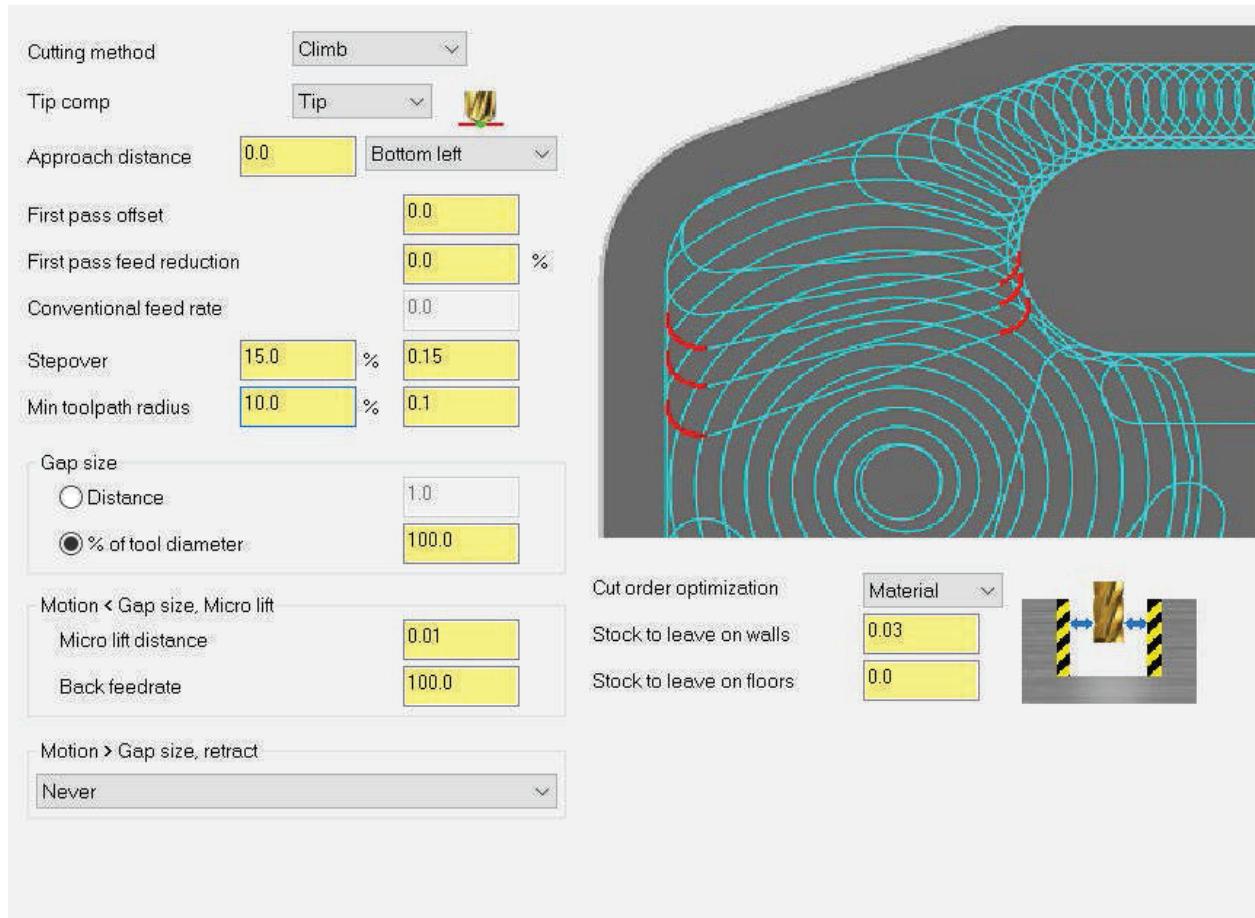


19.4 Set the Cut Parameters

- From the Tree View list, select **Cut Parameters**. Make sure that the parameters are set as shown

Figure: 19.4.1.

Figure: 19.4.1

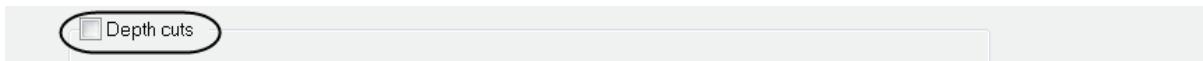


NOTE: For more information on these settings, see [page 322](#).



19.5 Set the Depth Cuts Parameters

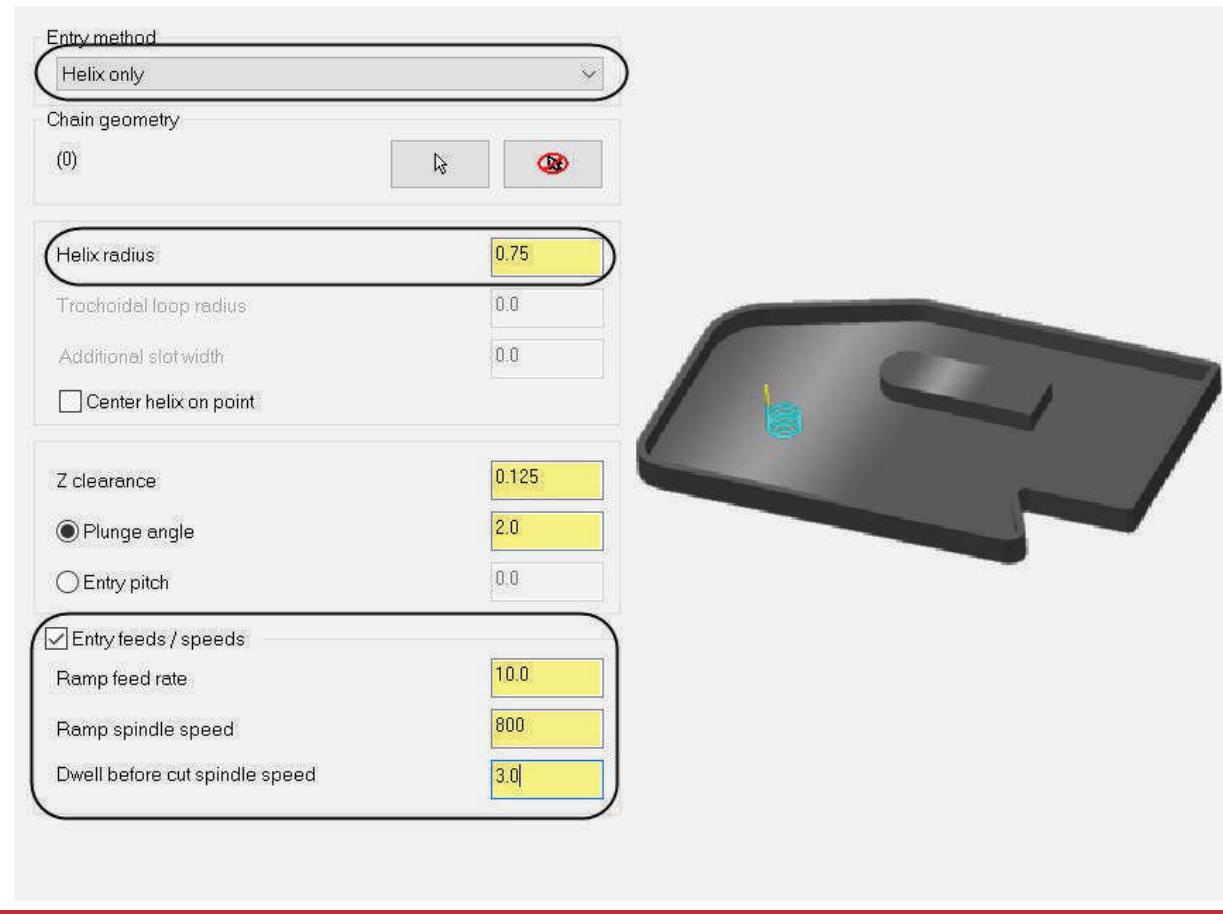
- From the Tree View list, disable the **Depth Cuts** if needed as shown.



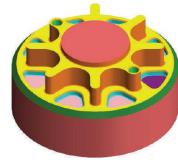
19.6 Set the Entry Motion

- Set the **Entry method** to **Helix only**.
- Set the **Helix radius** to **0.75** and the rest of the parameters as shown in [Figure: 19.6.1](#).
- Enable **Entry feeds / speeds** and set a **Ramp feed rate** of **10.0** Inches per minute, a **Ramp spindle speed** of **800 RPM** and **Dwell before cut spindle speed** of **3.0** seconds as shown in [Figure: 19.6.1](#).

Figure: 19.6.1



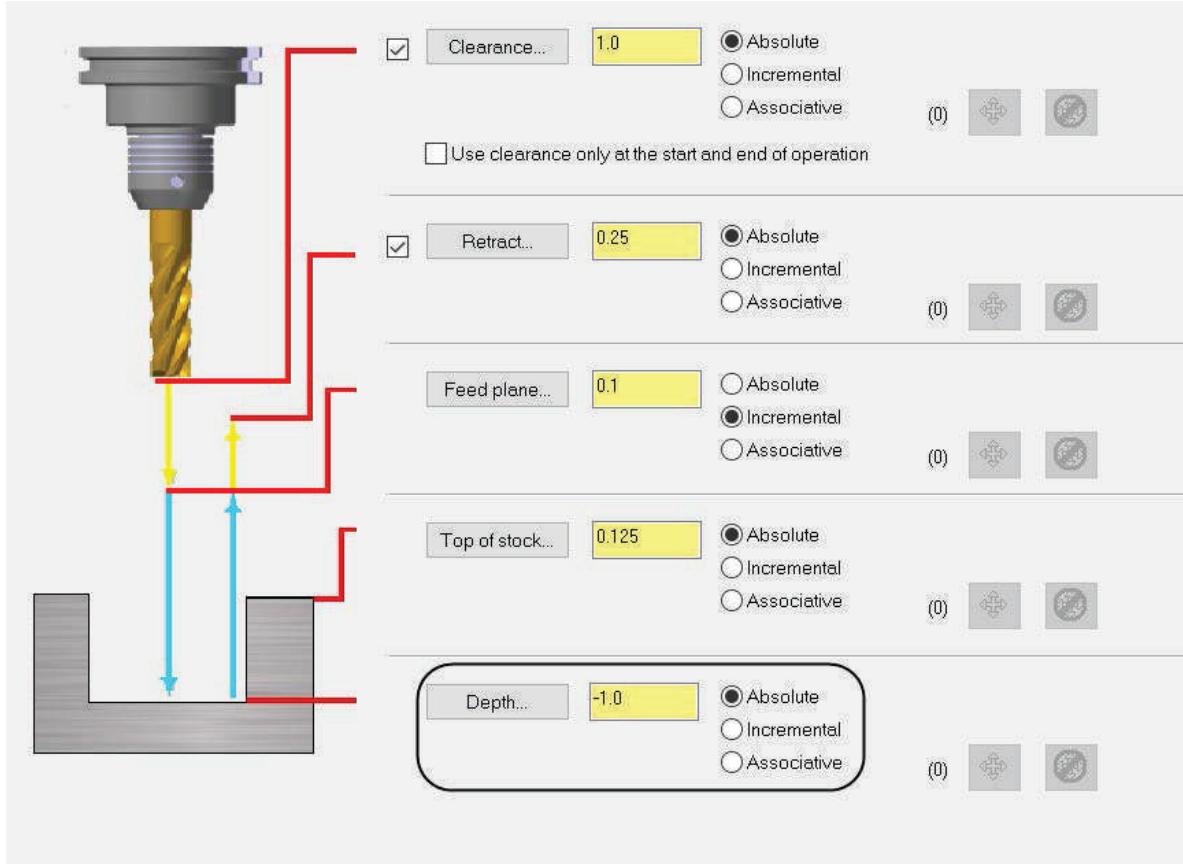
Entry method set to **Helix only** creates a helical entry into the part.



19.7 Set the Linking Parameters

- ♦ Select **Linking Parameters** and input the **Depth** of **-1.0** as shown in [Figure: 19.7.1](#).
- ♦ Change any other parameters to match what is shown in [Figure: 19.7.1](#) if necessary.

Figure: 19.7.1



19.8 Preview the Toolpath

- ♦ To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

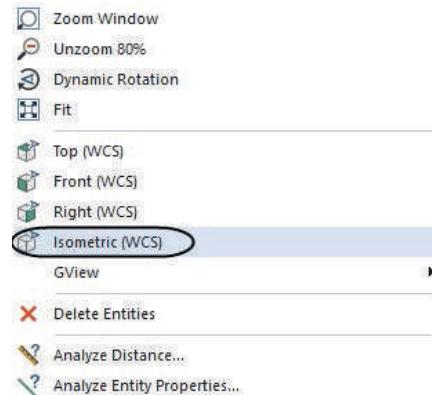


- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.

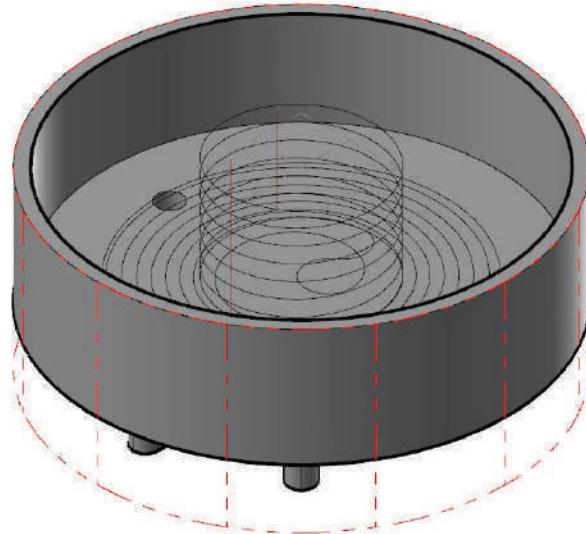




- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



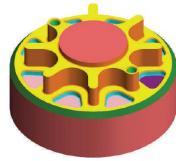
- ♦ The toolpath should look as shown.



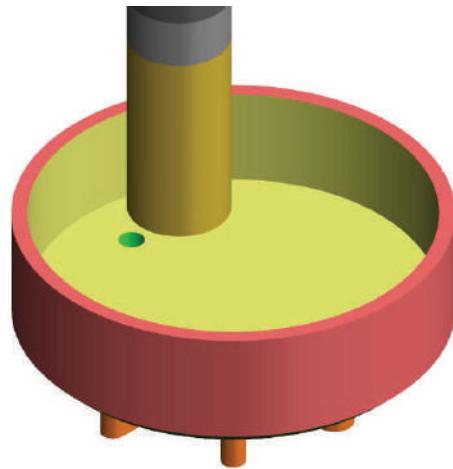
- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Once complete, pick the **OK** button to generate the toolpath. 
- ♦ Select the **Toolpaths** tab in the bottom left of the toolpaths manager.
- ♦ To **Backplot** and **Verify** your toolpath, see [page 145](#) and [page 147](#) to review these procedures.



- Once complete, the part will appear as shown.

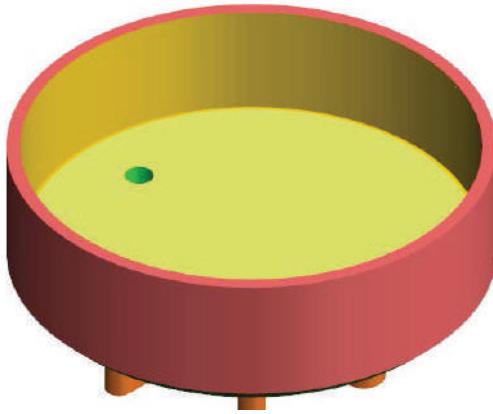


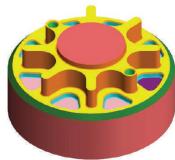
NOTE: The High Speed toolpaths do not have a finish wall option inside of their parameters. You need to finish the walls using a Contour toolpath.

STEP 20: FINISH THE POCKET WALL USING CONTOUR TOOLPATH

In this step we will utilize the Contour toolpath to finish the middle of the part with the part now flipped over. You will remove the 0.03" stock from the walls.

Toolpath Preview:

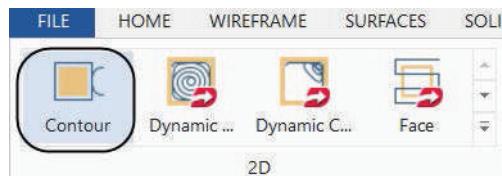




20.1 Chain Selection

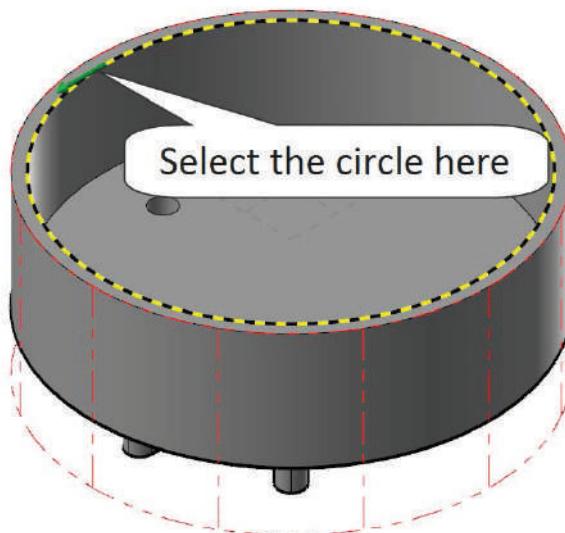
TOOLPATHS

- From the **2D** group, select the **Contour** icon as shown.



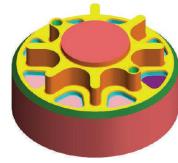
- Leave the default settings in the **Chaining** dialog box and pick the inner circle as shown in [Figure: 20.1.1](#).

Figure: 20.1.1



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

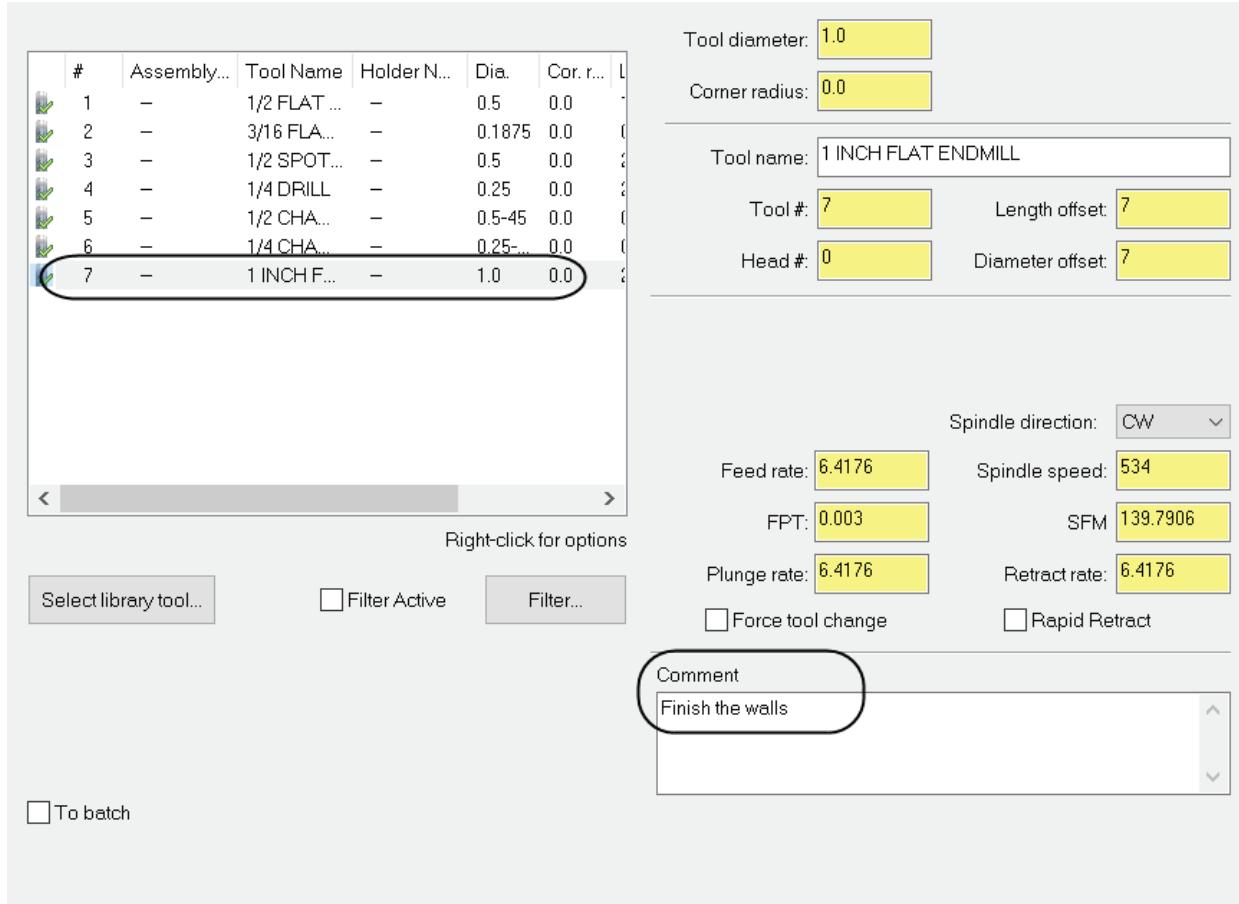




20.2 Select a 1.0" Flat Endmill from Tool list window

- ♦ Select **Tool** from the **Tree View list**.
- ♦ Select the **1.0" Flat Endmill** in the **Tool list** window. Make all the necessary changes as shown in [Figure: 20.2.1](#).

Figure: 20.2.1

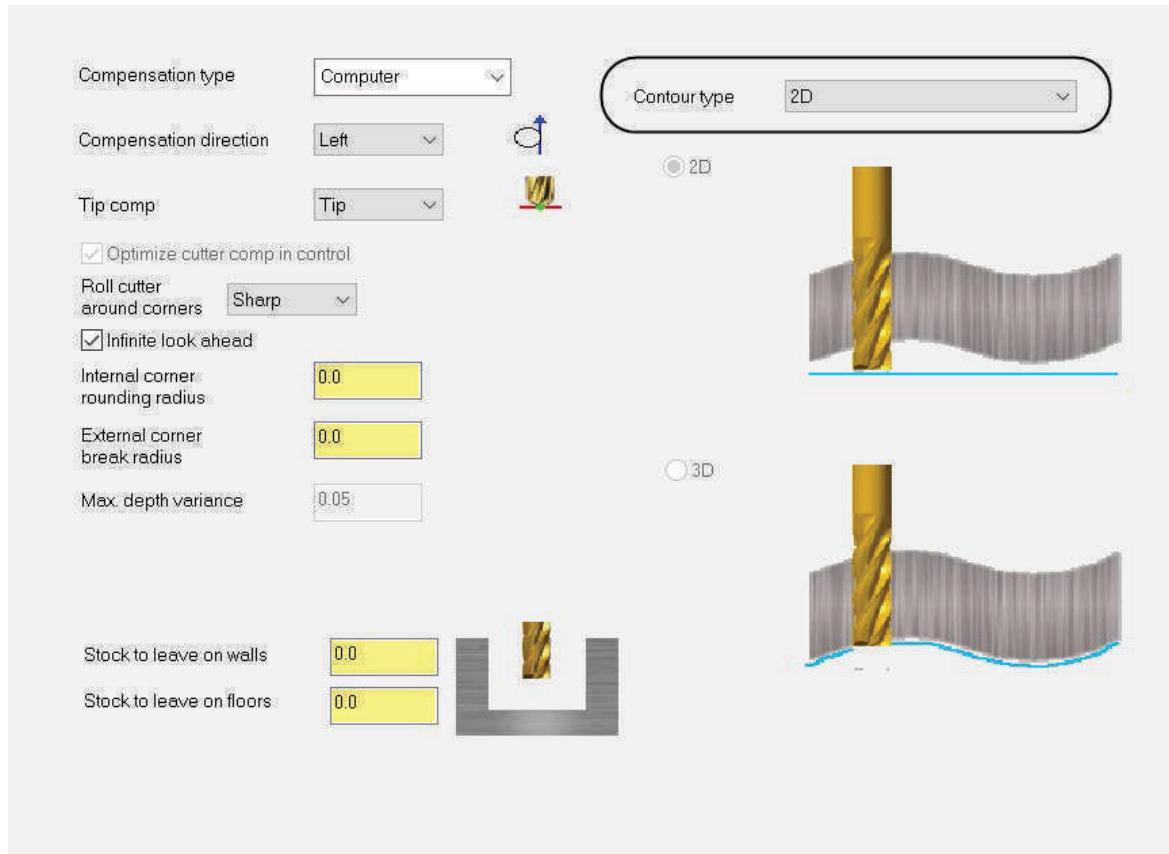


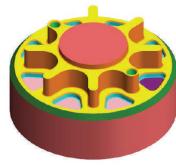


20.3 Set the Cut Parameters

- From the Tree View list, select **Cut Parameters** and make sure that the parameters are set as shown in Figure: [20.3.1](#).

Figure: 20.3.1





20.4 Set the Depth Cuts Parameters

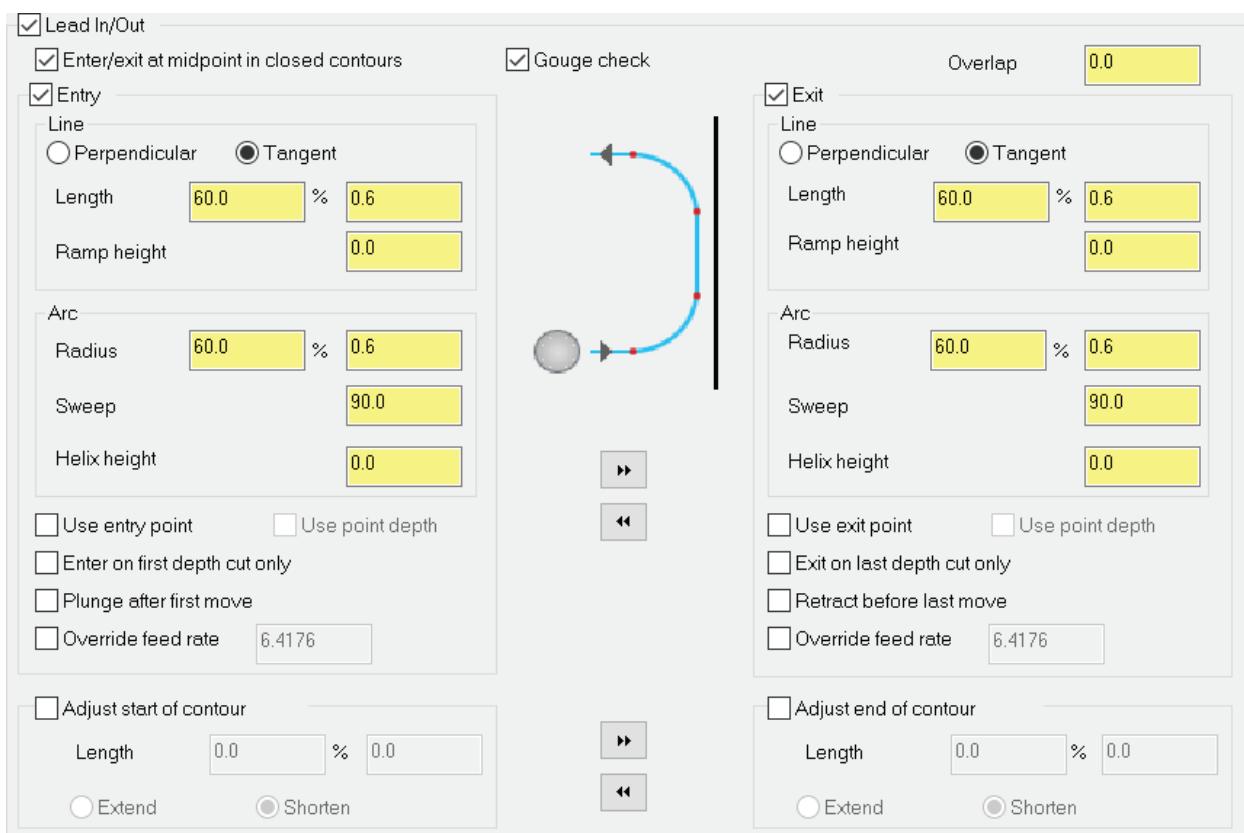
- From the Tree View list, disable the **Depth Cuts** if needed as shown.



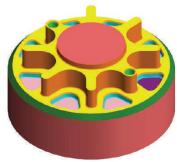
20.5 Set the Lead In/Out parameters

- Make sure that the parameters are set as shown in [Figure: 20.5.1](#).

Figure: 20.5.1



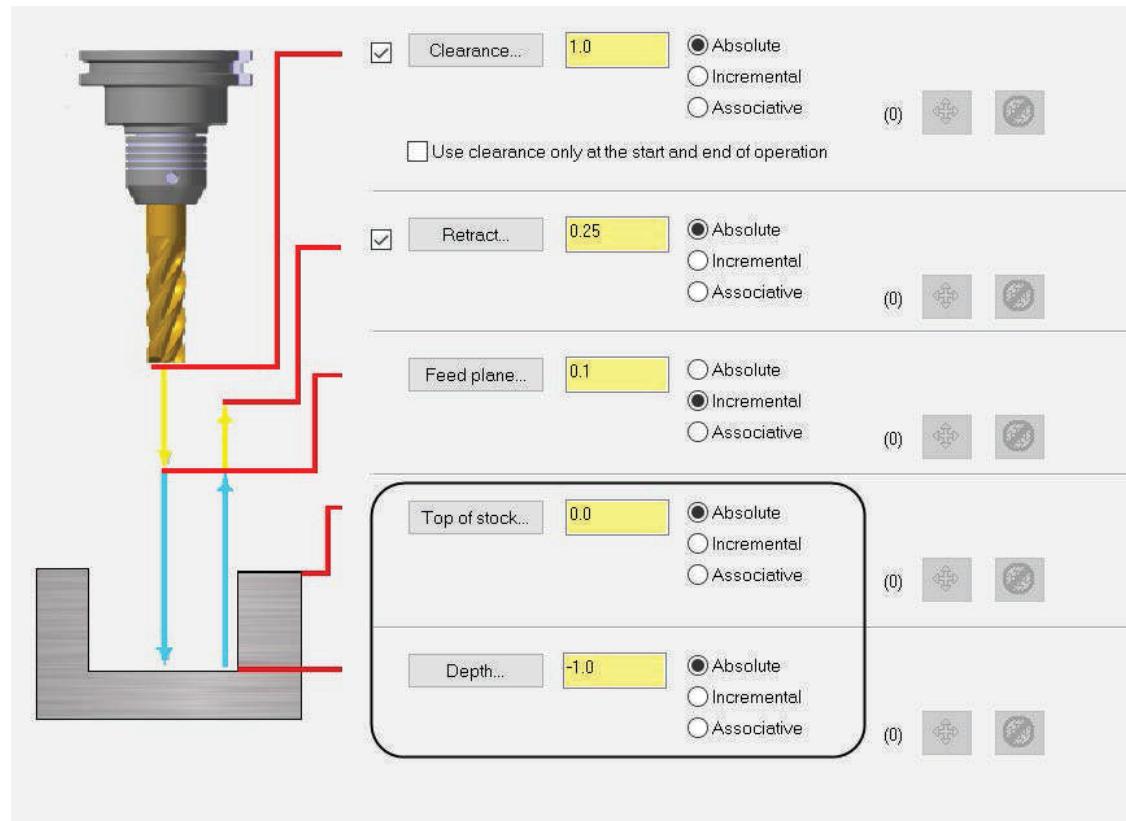
NOTE: For more information on these settings, see [page 335](#).

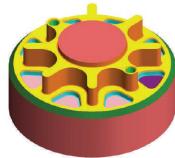


20.6 Set the Linking Parameters

- ♦ Select **Linking Parameters** and input the **Depth** of **-1.0** as shown in [Figure: 20.6.1](#).

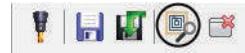
Figure: 20.6.1



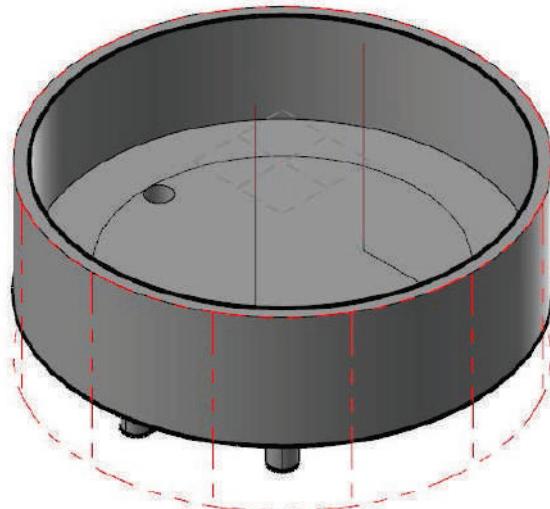


20.7 Preview the Toolpath

- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



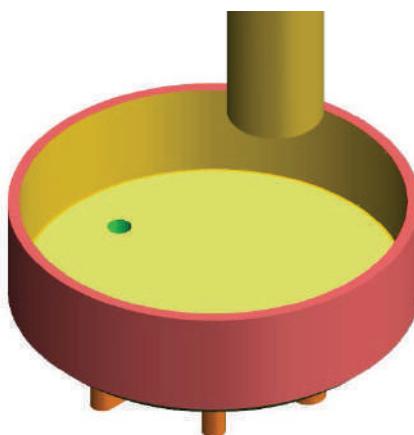
- ♦ See [page 159](#) to review the procedure.
- ♦ The toolpath should look as shown.



- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to exit the toolpath parameters. 
- ♦ **Backplot** and **Verify** your toolpath. See [page 56](#) and [page 58](#) to review these procedures.
- ♦ The part will appear as shown.



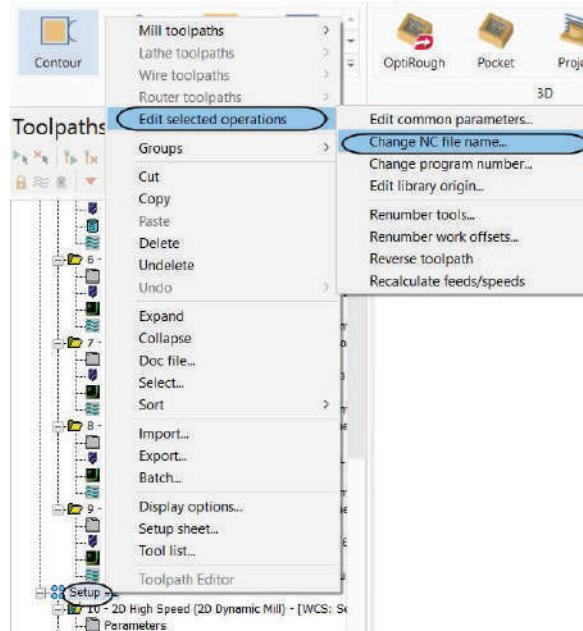
- ♦ To exit the Mastercam Simulator, click on the **Close** icon. 



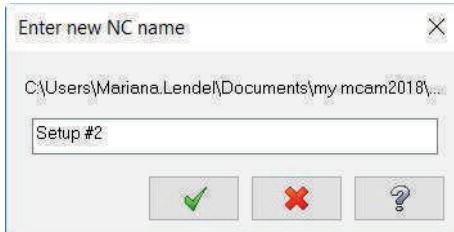
STEP 21: RENAME THE NC FILE

The **2D High Speed Dynamic Mill** operation in Setup #2 kept the NC name from Setup #1. We need to rename this operation so that it will create 2 separate programs.

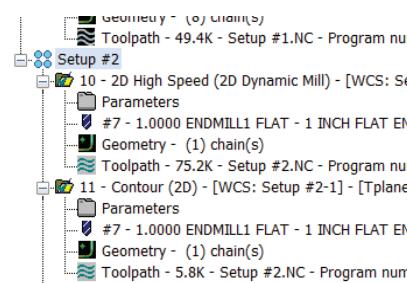
- ♦ Click on the **Setup #2** group to select only operation #10 and operation #11.
- ♦ Right click on **Setup #2**, choose the option **Edit selected operations** and then pick **Change NC file name**.

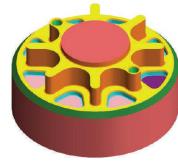


- ♦ When the **Enter new NC name** panel appears enter **Setup #2**.



- ♦ Select the **OK** button to apply the changed **NC name** to operation **#10** and **#11**.
- ♦ As a result, you should see **Setup #2.NC** in the last item of text for operation **#10** and **#11**.

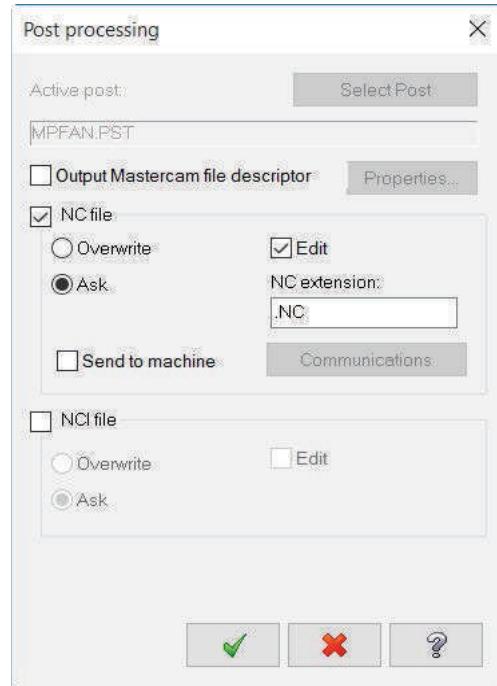




STEP 22: POST THE FILE

- ◆ Ensure all operations are selected. If they are not, use the **Select all operations** button in the **Toolpaths Manager**.
- ◆ Select the **Post selected operations** button from the **Toolpaths Manager**.
- ◆ In the **Post processing** window, make the necessary changes as shown in [Figure: 22.0.1](#).

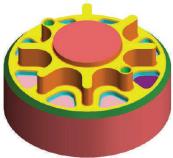
Figure: 22.0.1



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

Edit enabled allows you to automatically launch the default editor.

- ◆ Select the **OK** button to continue.
- ◆ Save Setup #1 NC file.
- ◆ Save Setup #2 NC file.



- ♦ A window with **Mastercam Code Expert** will be launched and the NC program will appear as shown in [Figure: 22.0.2](#).

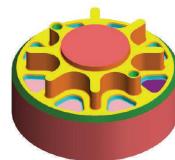
Figure: 22.0.2

The screenshot shows the Mastercam Code Expert interface. The title bar reads "Setup #1.NC - Mastercam 2018 Code Expert". The menu bar includes File, Home, View, and NC Functions. The toolbar has icons for Insert Block Number, Insert Block Skip, Send File, Remove Block Number, Remove Block Skip, Remove Comments, Go To, Remove Spaces, Syncs, Tools, Mult-Stream, and Utilities. The main area displays an NC program (Setup #1.NC) with various G-code commands. The status bar at the bottom shows "Ready", "27243KB 100%", and zoom controls.

```
1
2 O0004 (SETUP #1)
3 (DATE=00-MAY-17 - 22-02-17 TIME=H:M:S - 14:55)
4 (NCX FILE - \\112SERVER06\EOGGEV\PRODUCTION\BOOKS\2015\TRAINING TUTORIALS\IT CAD IMPORT MILL ESSENTIALS TOOLPATHS\NCX FILES\INCH\TUTORIAL TOOLPATH INCH\TUTORIAL_4.INCH.MCAM)
5 (NC FILE - C:\USERS\WINGYUE.HAO\Desktop\SETUP #1.NC)
6 (MATERIAL - ALUMINUM INCH - 2024)
7 ( T1 | 1/8 FLAT ENDMILL | H1 | XY STOCK TO LEAVE = .03 | Z STOCK TO LEAVE = 0. )
8 ( T2 | 3/16 FLAT ENDMILL | H2 | XY STOCK TO LEAVE = .08 | Z STOCK TO LEAVE = 0. )
9 ( T3 | 1/8 SPOTDRILL | H3 )
10 ( T4 | 1/8 DRILL | H4 )
11 ( T5 | 1/2 CHAMFER MILL | H5 )
12 ( T6 | 1/4 CHAMFER MILL | H6 )
13 M100 G20
14 M110 G0 G17 G40 G49 G90 G90
15 ( DYNAMIC MILL THE STEPS USING ISLAND FACING. )
16 N120 T1 M9
17 N130 G0 G90 G54 X-1.6516 Y-1.0000 R0, S12000 M9
18 N140 G43 H1 Z1
19 N150 Z1
20 M140 G1 Z-.615 F100,
21 M210 X-1.6763 Y-1.7294 Z-.6208 F200,
22 M180 X-1.6819 Y-1.7253 Z-.6222
23 M190 X-1.6904 Y-1.7123 Z-.6236
24 M200 X-1.7044 Y-1.6958 Z-.625
25 M210 G2 X-1.6374 Y-1.5387 I3,4937 J3,0911
26 M220 X-2.3527 Y-1.0635 I1,.854 J1,4752
27 M230 X-2.311 V,.3692 I2,.3693 J0
28 M240 X-2.1037 Y,.9908 I2,.3365 J-.4338
29 M250 X-1.5399 Y1,.7388 I2,1421 J-1.0281
30 M260 X0 Y2,.3226 I1,.5399 J-1.7309
31 M270 X,.1273 Y2,2058 I0, J-2,3227
32 M280 X2,.2721 Y,.4818 I-,7273 J-2,2058
33 M290 X2,.3226 Y0, I-2,2721 J-,4818
34 M300 X1,.5599 Y-1,.7208 I-2,3226 J0,
35 M310 X0, Y-2,3226 I-1,.5599 J1,.7208
36 M320 X-1.7018 Y-2,2124 I0, J2,3226
37 M330 X-1.5404 Y-1.7374 I,.695 J2,199
38 M340 X-2.2699 Y-1.0638 I1,.5549 J1,.6736
--
```

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

STEP 23: SAVE THE UPDATED MCAM FILE



REVIEW EXERCISE - STUDENT PRACTICE

IMPORT THE SOLIDWORKS FILE FOR TUTORIAL #4 EXERCISE

Import the dwg file and manipulate the geometry to prepare it for the toolpath creation.

Download the files from www.emastercam.com/trainingfiles.

Save the file in a preferred location.

Import TUTORIAL_4 EXERCISE.SLDPRT.

- ♦ Use file **Open** and change the file type to **SOLIDWORKS Files (*.sldprt; *.sldasm; *.slddrw)**.





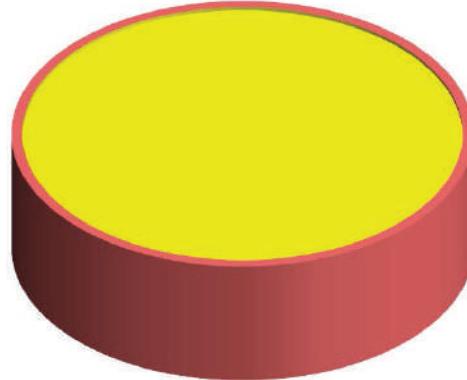
CREATE THE TOOLPATHS FOR TUTORIAL #4 EXERCISE

Create the Toolpaths for Tutorial #4 Exercise as per the instructions below.

Set the machine properties including the stock setup.

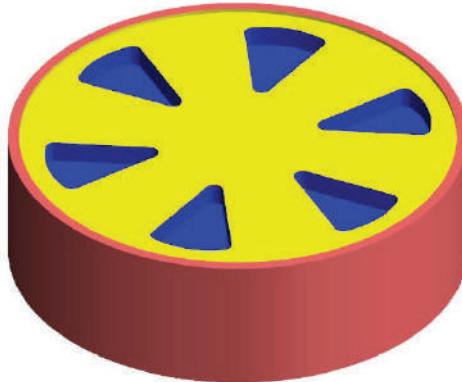
Area Mill the center of the part.

- ◆ Use a **1/2" Flat Endmill**.
- ◆ **Stock to leave on the walls/floors = 0.0**.
- ◆ Disable **Depth Cuts**.
- ◆ Set the **Transitions to Entry helix**.
- ◆ Set the **Depth** according to the part.



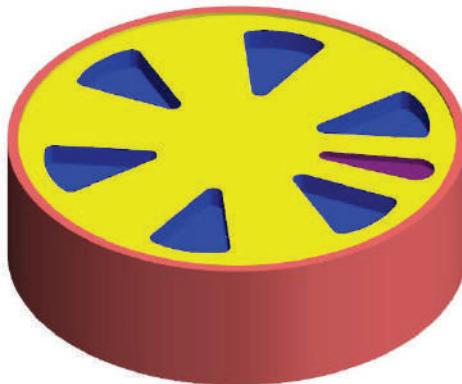
Dynamic Mill the large pockets.

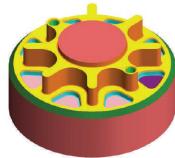
- ◆ Chain 6 larger pockets and enable **Stay inside**.
- ◆ Use a **3/16" Flat Endmill**.
- ◆ **Stock to leave on the walls = 0.03**.
- ◆ **Stock to leave on the floors = 0.0**.
- ◆ Set the **Entry Motion to Profile**.
- ◆ Set the **Top of Stock** and **Depth** according to the part.



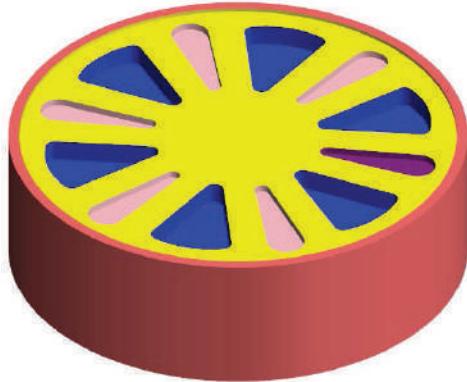
Area Mill one of the small pockets.

- ◆ Select one of the smaller pockets.
- ◆ Use a **3/16" Flat Endmill**.
- ◆ **Stock to leave on the walls = 0.03**.
- ◆ **Stock to leave on the floors = 0.0**.
- ◆ Enable **Depth Cuts**.
- ◆ Set **Max rough step = 0.05**, **# Finish cuts = 1** and **Finish step = 0.01**.
- ◆ Set the **Transitions to Entry helix**.
- ◆ Set the **Top of Stock** and **Depth** according to the part.

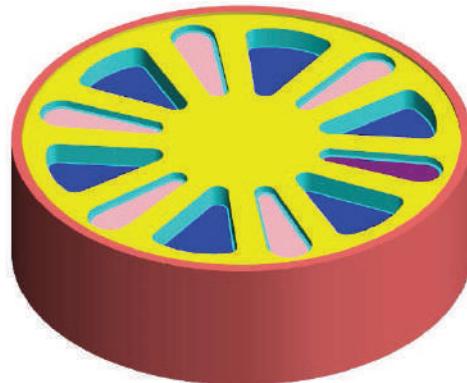


**Transform Toolpaths.**

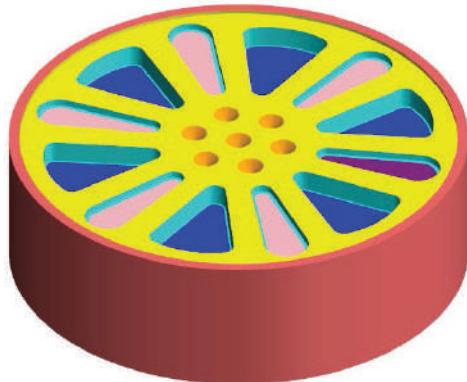
- ◆ Choose **Rotate** and select **Operation #3**.
- ◆ Select **Coordinate**, **NCI** and **Operation order**.
- ◆ Select the **Rotate** tab.
- ◆ Input the number of steps # = 5.
- ◆ **Start angle = 60.0.**
- ◆ **Rotation angle = 60.0.**

**Use a pocket toolpath to finish the walls.**

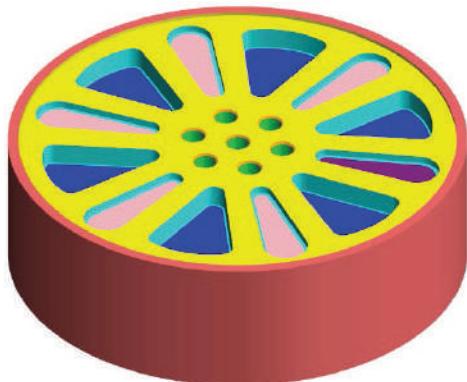
- ◆ Select the bottom of the pockets using **Solid** selections **Loop** and **Face**.
- ◆ Use the existing **3/16" Flat Endmill**.
- ◆ **Stock to leave on walls/floors = 0.0.**
- ◆ Disable **Roughing**.
- ◆ Turn **Off** the **Entry motion**.
- ◆ Set **Finishing Spacing** to **0.03**.
- ◆ In the **Lead In/Out** set the **Length** and the **Radius** to **60%** of the tool diameter.
- ◆ **Depth to Incremental** and **0.0**.

**Spot Drill the holes.**

- ◆ Use a **1/2" Spot Drill**.
- ◆ Set the **Cycle** and **Dwell**.
- ◆ Set the **Top of Stock** and **Depth** using the depth calculator.

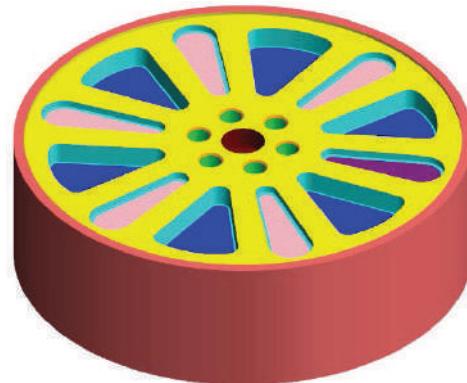
**Drill the holes.**

- ◆ Use a **1/4" Drill**.
- ◆ Set the **Cycle** to **Chip Break** and input your increments.
- ◆ Set the **Top of Stock** and **Depth** according to part.

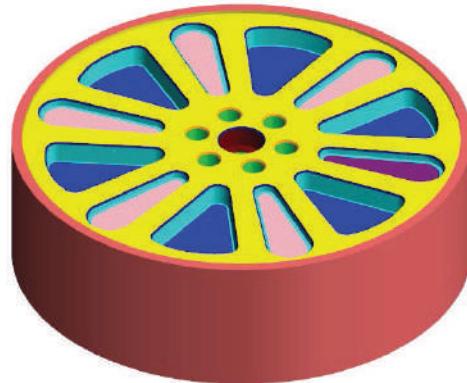


**Circle Mill the Center Hole.**

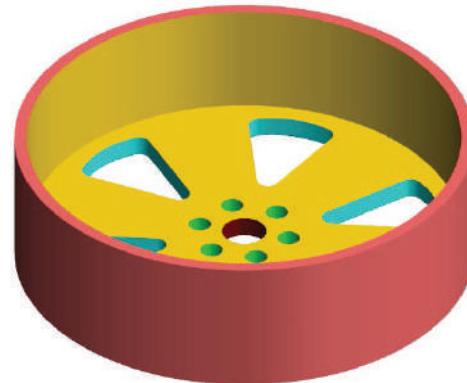
- ◆ Use the **3/16" Flat Endmill**.
- ◆ **Stock to leave = 0.0**.
- ◆ Disable **Roughing**.
- ◆ Enable **Finishing**.
- ◆ Enable **Finish** and set **1 Finish** at a **Spacing of 0.03**.
- ◆ Enable **Final depth** and **Keep tool down**.
- ◆ Set the depth to the appropriate depth.

**Chamfer the sharp edges using Contour toolpaths.**

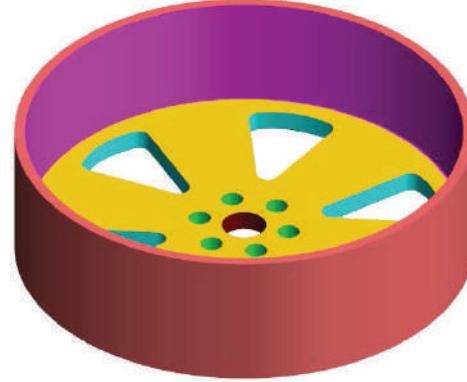
- ◆ Use **C-plane** mode selection and select all the tops of the pockets and the center hole.
- ◆ Select all the chains in CCW direction.
- ◆ Use a **1/4" Chamfer Mill**.
- ◆ **Contour type** set to **2D Chamfer; Width = 0.025**; **Tip offset = 0.02**.
- ◆ In the **Lead In/Out** set the **Length** and the **Radius** to **60%** of the tool diameter.
- ◆ **Top of stock** and **Depth** set to **Increment** and **0.025** (size of the chamfer).

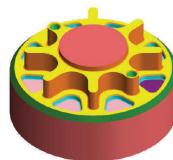
**Flip the part over - Setup #2.****Create a new plane Relative to WCS Bottom, set to appropriate depth and set as WCS.**

- ◆ Use **Dynamic Mill** and enable **Stay inside** to remove the material starting from the center.
- ◆ Use the **1-1/2" Flat Endmill**.
- ◆ **Stock to leave on walls = 0.03**.
- ◆ Disable **Depth Cuts**.
- ◆ Set the **Entry Motion**.
- ◆ Set the **Depth** according to the part.

**Use a contour toolpath to finish the walls.**

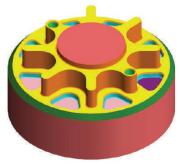
- ◆ Select the inner circle in CCW direction.
- ◆ Use the existing **1-1/2" Flat Endmill**.
- ◆ Set **Contour type** to **2D**.
- ◆ **Stock to leave on walls/floors = 0.0**
- ◆ Disable **Depth cuts**.
- ◆ In the **Lead In/Out** set the **Length** and the **Radius** to **60%** of the tool diameter.
- ◆ Set **Top of stock** and **Depth** according to part.



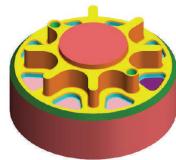
TUTORIAL #4**CREATE THE TOOLPATHS FOR TUTORIAL #4 EXERCISE**

- ◆ Your part should appear as shown once complete.





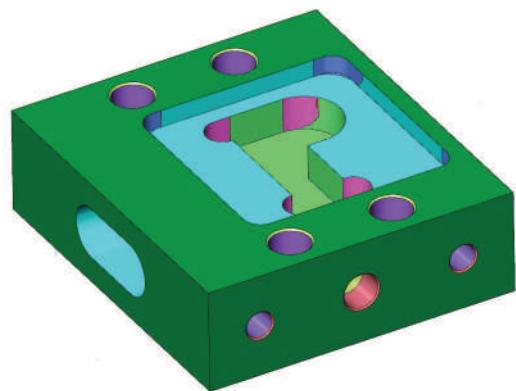
NOTES:

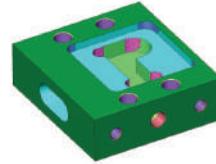


TUTORIAL #4 QUIZ

- ◆ What does a Circle Mill toolpath do?
 - ◆ What does "Dwell before cut spindle speed" do?
 - ◆ What does a Transform toolpath operation do?

TUTORIAL #5



**OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:****Import the CAD Model and prepare it to generate Toolpaths:**

- ◆ The student will open the Solidworks file in Mastercam.

Create the necessary Toolpaths to machine the part:

- ◆ The student will set up the stock size to be used and the clamping method used. Three setups will be used to machine the part from the top and then from the bottom.
- ◆ A 2D High Speed Area Mill toolpath will be created to remove the material inside of the step.
- ◆ Two 2D High Speed Area Mill toolpaths will be created to remove the material inside the pockets.
- ◆ Drill toolpaths will be created to machine the three holes in the front view.
- ◆ A Slot Mill toolpath will be created to remove the material inside of the slot from the left side view.

Backplot and Verify the file:

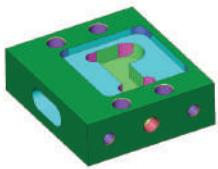
- ◆ The Backplot will be used to simulate a step-by-step process of the tool's movements.
- ◆ The Verify will be used to watch a tool machine the part out of a solid model.

Post Process the file to generate the G-code:

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



This tutorial takes approximately two hours to complete.



GEOMETRY CREATION

STEP 1: SETTING UP THE GRAPHIC USER INTERFACE

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

STEP 2: IMPORTING THE SOLIDWORKS FILE GEOMETRY

Mastercam lets you read (import) a variety of CAD file types into the Mastercam database. You can also write (export) Mastercam files to a variety of different file formats.

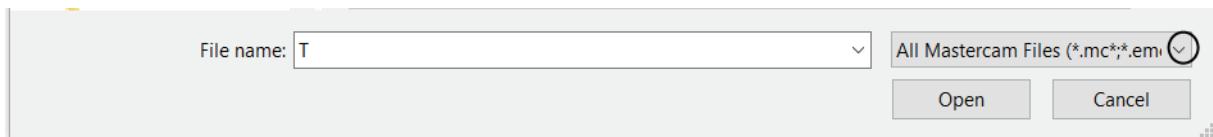
To import a SolidWorks file into Mastercam, you have to use the Open function and then select SolidWorks files from the type of files list.

Download the files from www.emastercam.com/trainingfiles.

Save the file at a preferred location.

FILE

- ♦ **Open.**
- ♦ In the file name extension, click on the drop down arrow as shown.

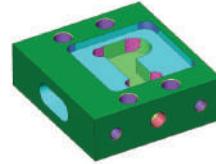


- ♦ From the type of files list, select **SolidWorks Files (*.sldprt;*.sldasm)** as shown.

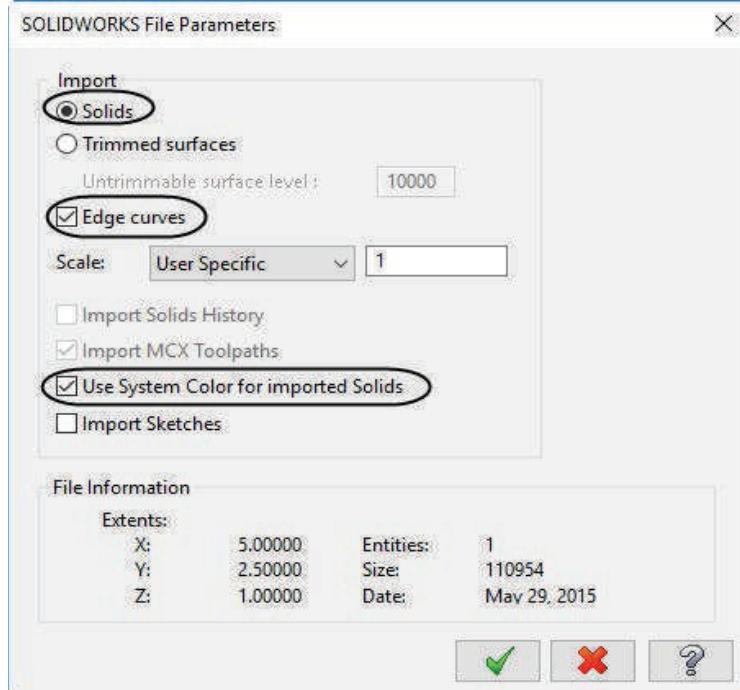


- ♦ Find and select **TUTORIAL #5.sldprt**.
- ♦ Click on the **Options** button.

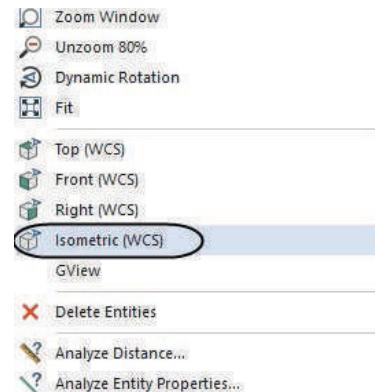




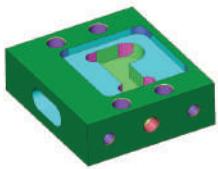
- Leave the **Solids** enabled to import the file as a solid and enable **Edge curves** for Mastercam to automatically create curves at the edges of the solid. To better see the curves, enable also **Use System Color for imported Solids** as shown.



- Select the **OK** button to exit the **SOLIDWORKS File Parameters** dialog box.
- Open** the file.
- Right mouse click in the graphics window and select the **Isometric** view.



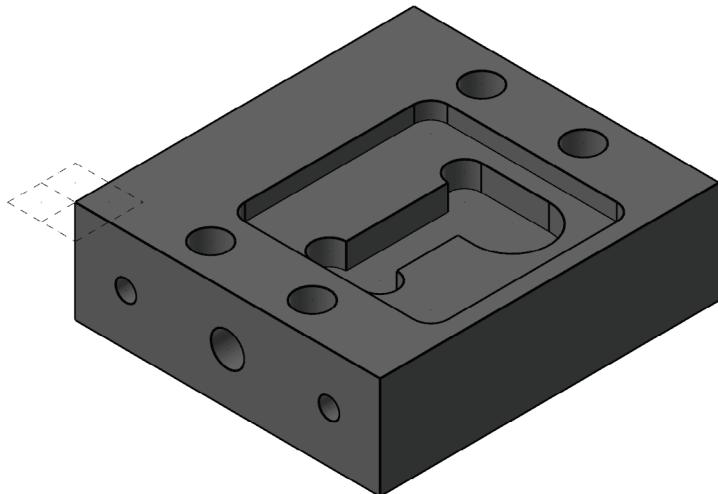
- Press **Alt + F1** to fit the geometry in the graphics window.
- Press **Alt + S** to shade the geometry if necessary.



SAVE THE FILE

TUTORIAL #5

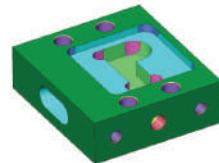
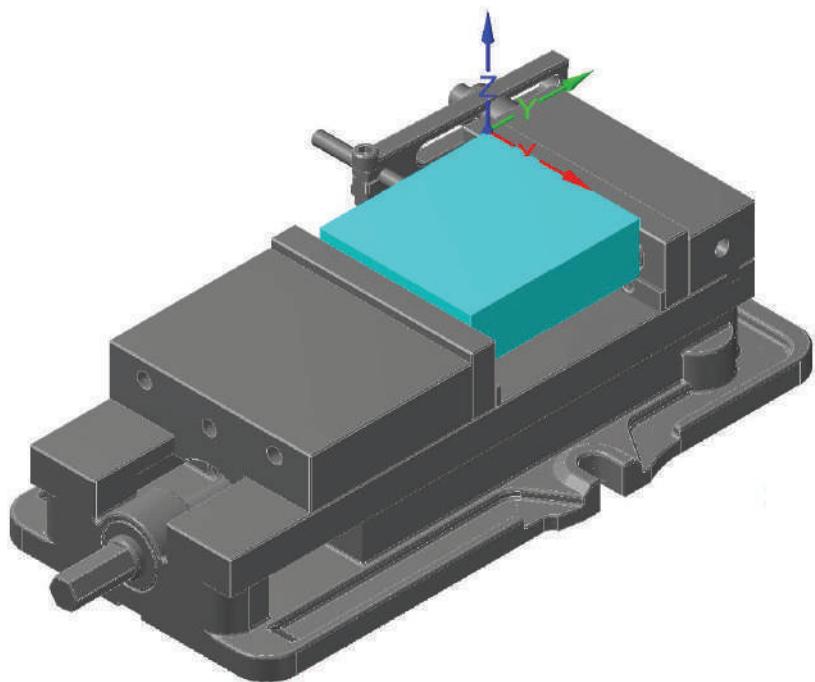
- ♦ The geometry should look as shown.



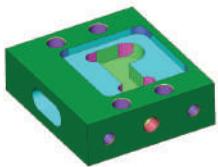
STEP 3: SAVE THE FILE

FILE

- ♦ Save As.
- ♦ File name: "Your Name_5".

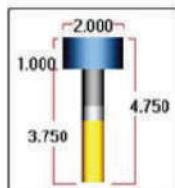
**TOOLPATH CREATION - SETUP 1****SUGGESTED FIXTURE:**

NOTE: In order to machine this part we will have 3 setups and output 3 NC files. To view the second setup see [page 493](#) and to view the third setup see [page 535](#).

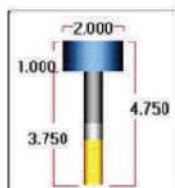


SETUP SHEET:

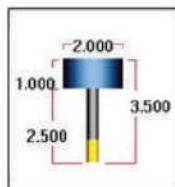
TOOL LIST



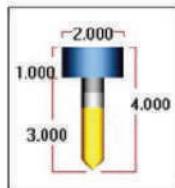
TYPE: Endmill1 Flat **FLUTE LENGTH:** 2.0
DIA: OFFSET: 1 **OVERALL LENGTH:** 3.75
HOLDER: DEFAULT HOLDER **CORNER RAD:** 0.0
NUMBER: 1 **# OF FLUTES:** 4
LENGTH OFFSET: 1
#1 - 0.7500 ENDMILL1 FLAT - 3/4 FLAT ENDMILL



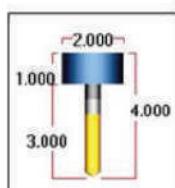
TYPE: Endmill1 Flat **FLUTE LENGTH:** 1.5
DIA: OFFSET: 2 **OVERALL LENGTH:** 3.75
HOLDER: DEFAULT HOLDER **CORNER RAD:** 0.0
NUMBER: 2 **# OF FLUTES:** 4
LENGTH OFFSET: 2
#2 - 0.6250 ENDMILL1 FLAT - 5/8 FLAT ENDMILL



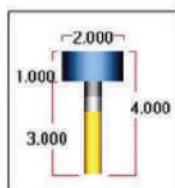
TYPE: Endmill1 Flat **FLUTE LENGTH:** 0.75
DIA: OFFSET: 3 **OVERALL LENGTH:** 2.5
HOLDER: DEFAULT HOLDER **CORNER RAD:** 0.0
NUMBER: 3 **# OF FLUTES:** 4
LENGTH OFFSET: 3
#3 - 0.3750 ENDMILL1 FLAT - 3/8 FLAT ENDMILL



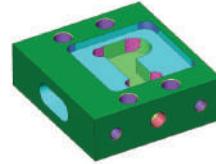
TYPE: Spot Drill **FLUTE LENGTH:** 2.0
DIA: OFFSET: 4 **OVERALL LENGTH:** 3.0
HOLDER: DEFAULT HOLDER **CORNER RAD:** 0.0
NUMBER: 4 **# OF FLUTES:** 4
LENGTH OFFSET: 4
#4 - 0.7500 SPOT DRILL - 3/4 SPOTDRILL



TYPE: Drill **FLUTE LENGTH:** 2.0
DIA: OFFSET: 5 **OVERALL LENGTH:** 3.0
HOLDER: DEFAULT HOLDER **CORNER RAD:** 0.0
NUMBER: 5 **# OF FLUTES:** 2
LENGTH OFFSET: 5
#5 - 0.5156 DRILL - 33/64 DRILL



TYPE: Tap RH **FLUTE LENGTH:** 2.0
DIA: OFFSET: 6 **OVERALL LENGTH:** 3.0
HOLDER: DEFAULT HOLDER **CORNER RAD:** 0.0
NUMBER: 6 **# OF FLUTES:** 1
LENGTH OFFSET: 6
#6 - 0.5625 X 18.00 TAP RH - 9/16-18 TAPRH



STEP 4: SELECT THE MACHINE AND SET UP THE STOCK

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

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

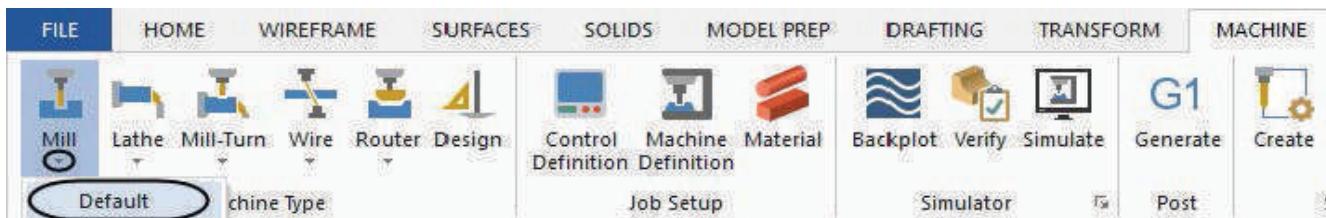
- From the bottom left side of the graphics window, click on the **Toolpaths** tab as shown.



NOTE: Select the **Mill Default** only if there is no **Machine Group** in the **Toolpaths Manager**.

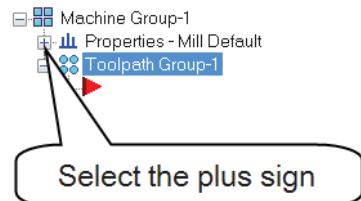
MACHINE

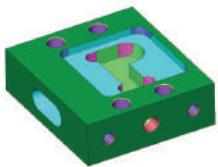
- From the **Machine Type** group, click on the drop down arrow below **Mill** and select the **Default**.



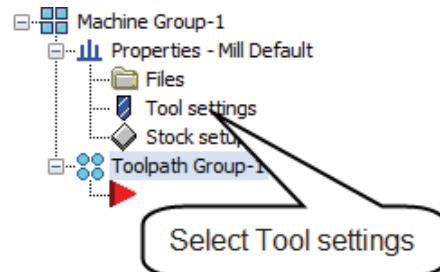
NOTE: Once you select the **Mill Default** the ribbon bar changes to reflect the toolpaths that could be used with **Mill Default**.

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



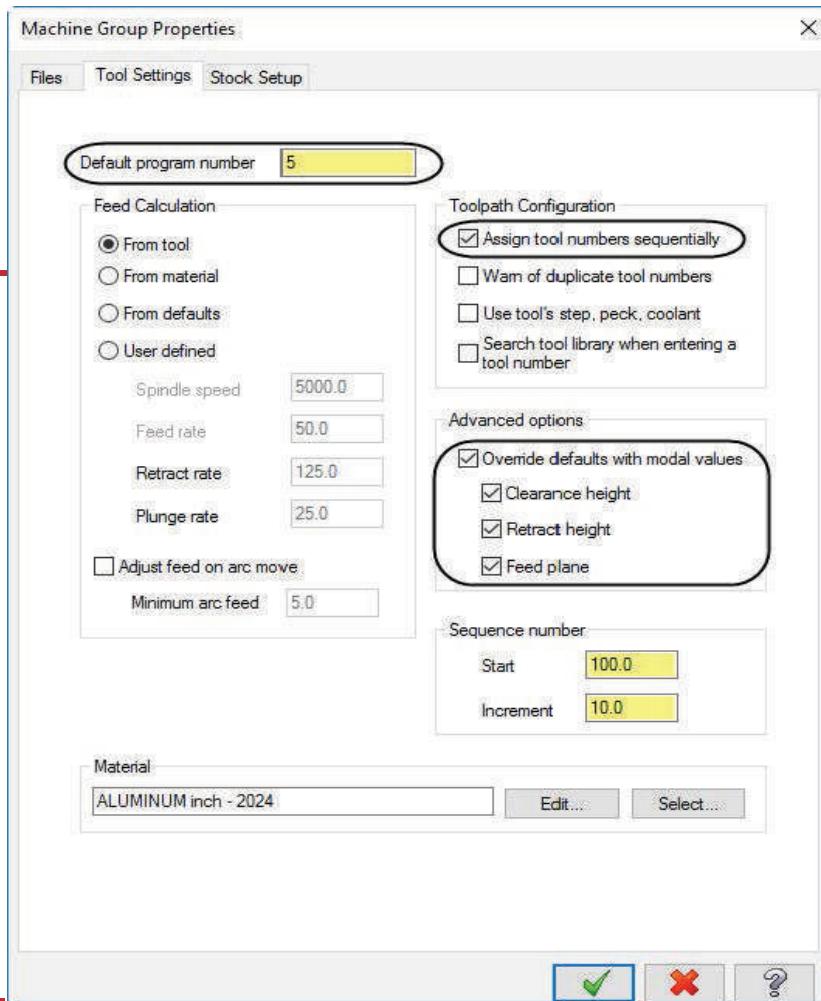


- ♦ Select **Tool settings** to set the tool parameters.



- ♦ Change the parameters to match [Figure: 4.0.1](#).

Figure: 4.0.1



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

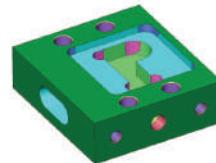
Assign tool numbers sequentially allows you to overwrite the tool number from the library with the next available tool number. (First operation tool number 1; Second operation tool number 2, etc.)

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

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

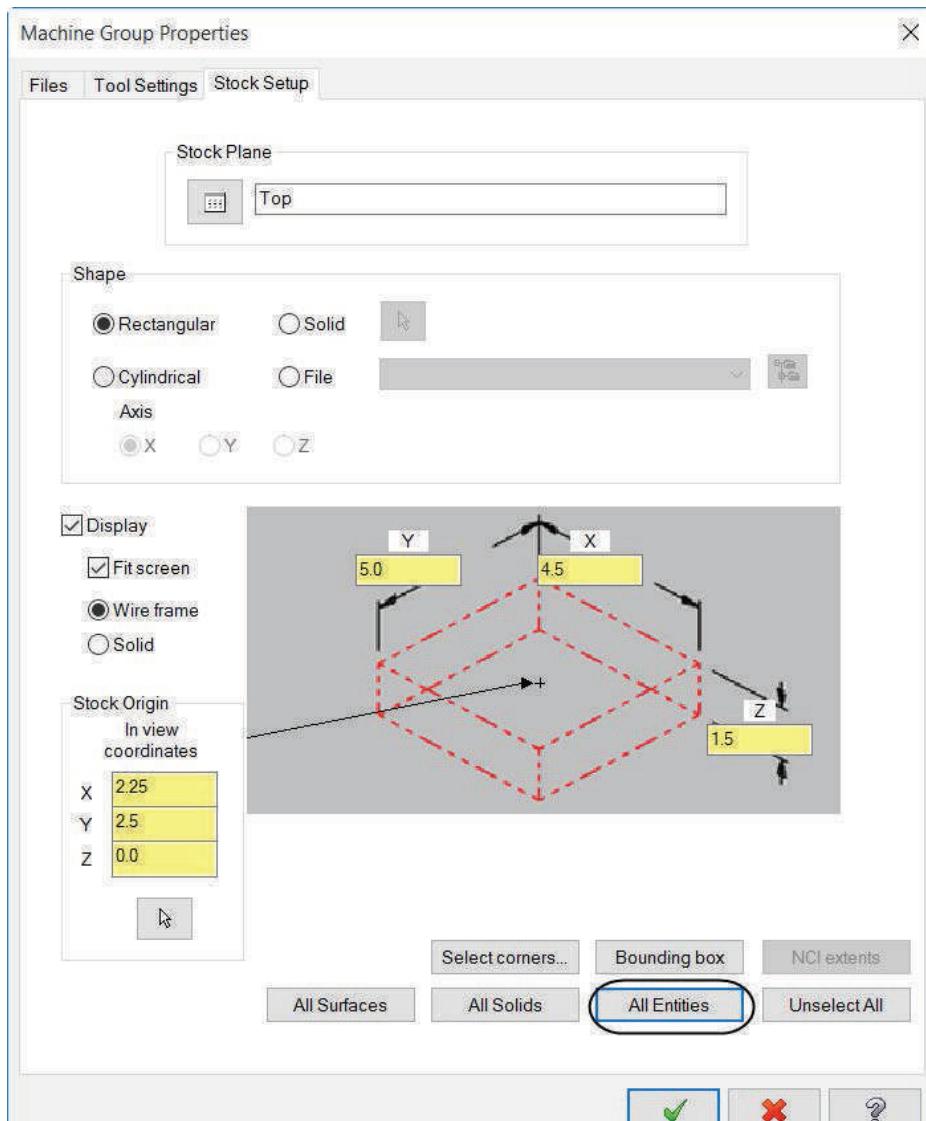
Feed Calculation set **From tool** uses feed rate, plunge rate, retract rate and spindle speed from the tool definition.

TUTORIAL #5 SELECT THE MACHINE AND SET UP THE STOCK



- ♦ Select the **Stock Setup** tab to define the stock.
- ♦ Pick the **Rectangular** shape option.
- ♦ Pick the **All Entities** button to define the stock size as shown in [Figure: 4.0.2](#).

Figure: 4.0.2



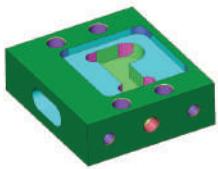
The **Stock Origin** values adjust the positioning of the stock, ensuring that you have equal amount of extra stock around the finished part.

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

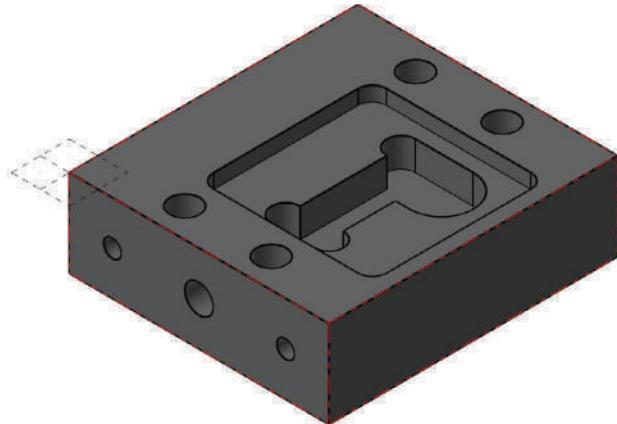
NOTE: The stock model that you create is used when viewing the file or the toolpaths, during backplot, or while verifying toolpaths. In the graphics, the plus shows you where the stock origin is. The default position is the middle of the stock.

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

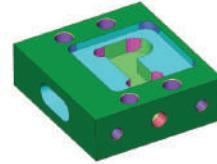




- ♦ The stock model will appear as shown.



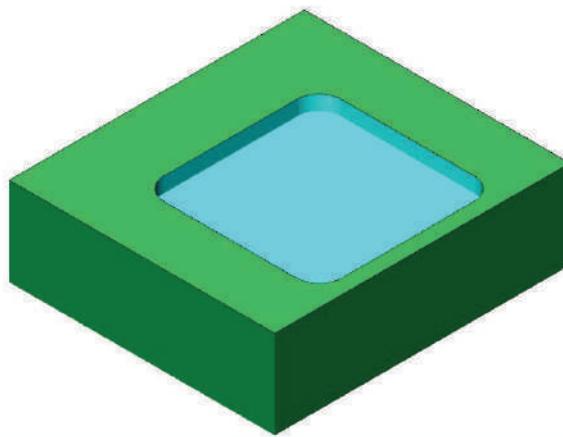
NOTE: You may not be able to see the stock very clearly due to the fact that the stock is the same size as the part. The stock is not geometry and cannot be selected.
There will not be a facing toolpath because the stock is already to size.



STEP 5: 2D HIGH SPEED AREA MILL

2D High Speed Area Mill allows you to machine pockets, material that other toolpaths left behind, and standing bosses or cores. The toolpath depends on the **Machining strategy** that you choose in the **Chain Options**. If the strategy chosen is **From outside**, the toolpath starts at the outmost chain and works its way in taking on the final shape of the part as it approaches the final pass. You can also machine pockets in which case the strategy selected is **Start inside** which keeps the tool inside the machining regions. Helical entries and tangent stepovers create efficient motion for your machine. Cut parameters let you control smoothing to create the best toolpath, avoiding sharp corners or direction changes.

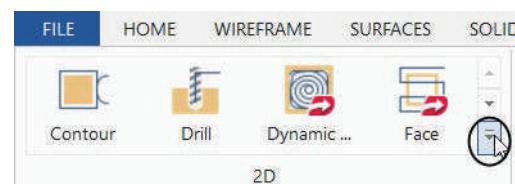
Toolpath Preview:

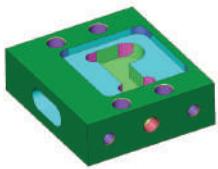


5.1 Chain Selection

TOOLPATHS

- From the **2D** group, click on the **Expand gallery** arrow as shown.

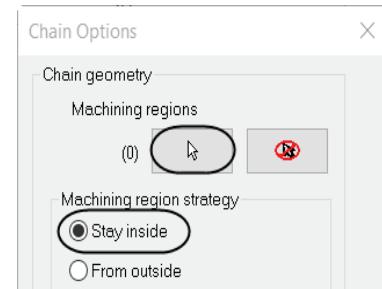


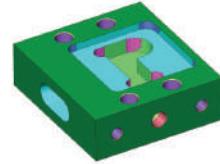


- ♦ Select the **Area Mill** icon as shown.



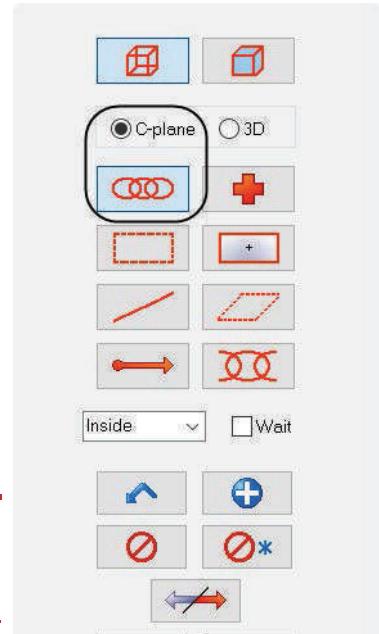
- ♦ From the **Chain Options**, click on the **Select machining chains** button.





- When the chaining dialog box appears select **C-plane** as shown.
- Leave the chaining method set to **Chain** as shown in [Figure: 5.1.1](#).

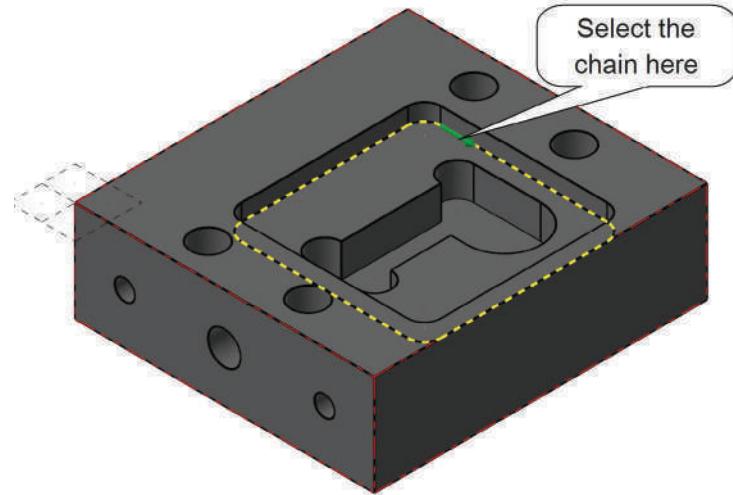
Figure: 5.1.1



C-plane chains only the entities that are parallel to the current construction plane and at the same Z depth as the first entity you chain.

- Select the bottom of the pocket as shown in [Figure: 5.1.2](#).

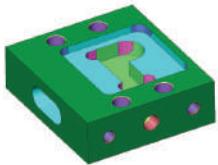
Figure: 5.1.2



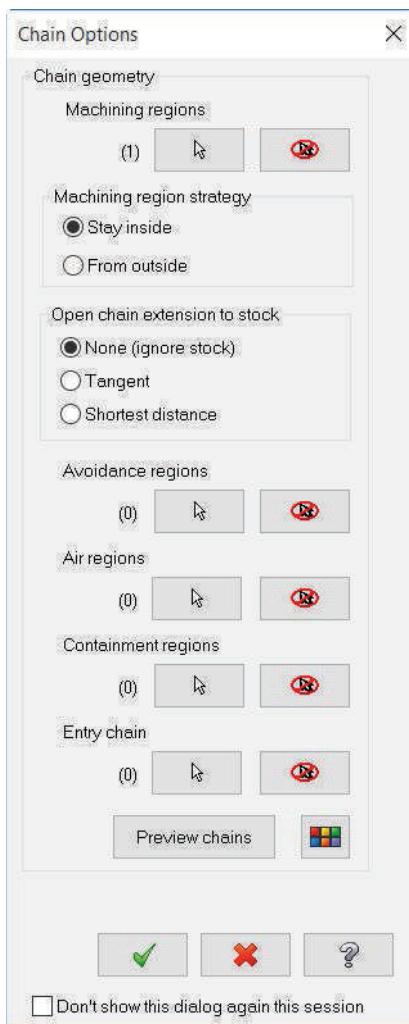
NOTE: If lines cannot be selected, "Edge curves" was probably not enabled when importing on [page 426](#).

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

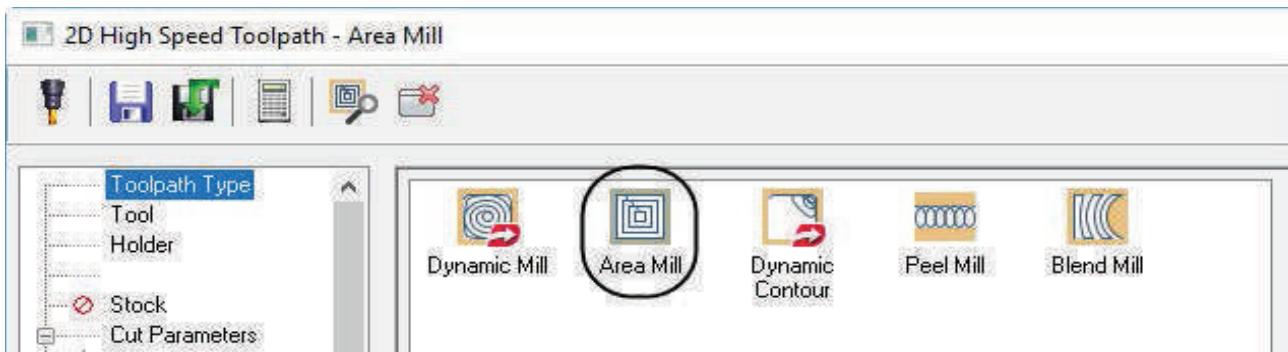


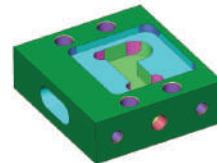


- Now the **Chain Options** dialog box should look as shown.



- Select the **OK** button to exit the **Chain Options** dialog box.
- In the **Toolpath Type** page, make sure that **Area Mill** is selected.

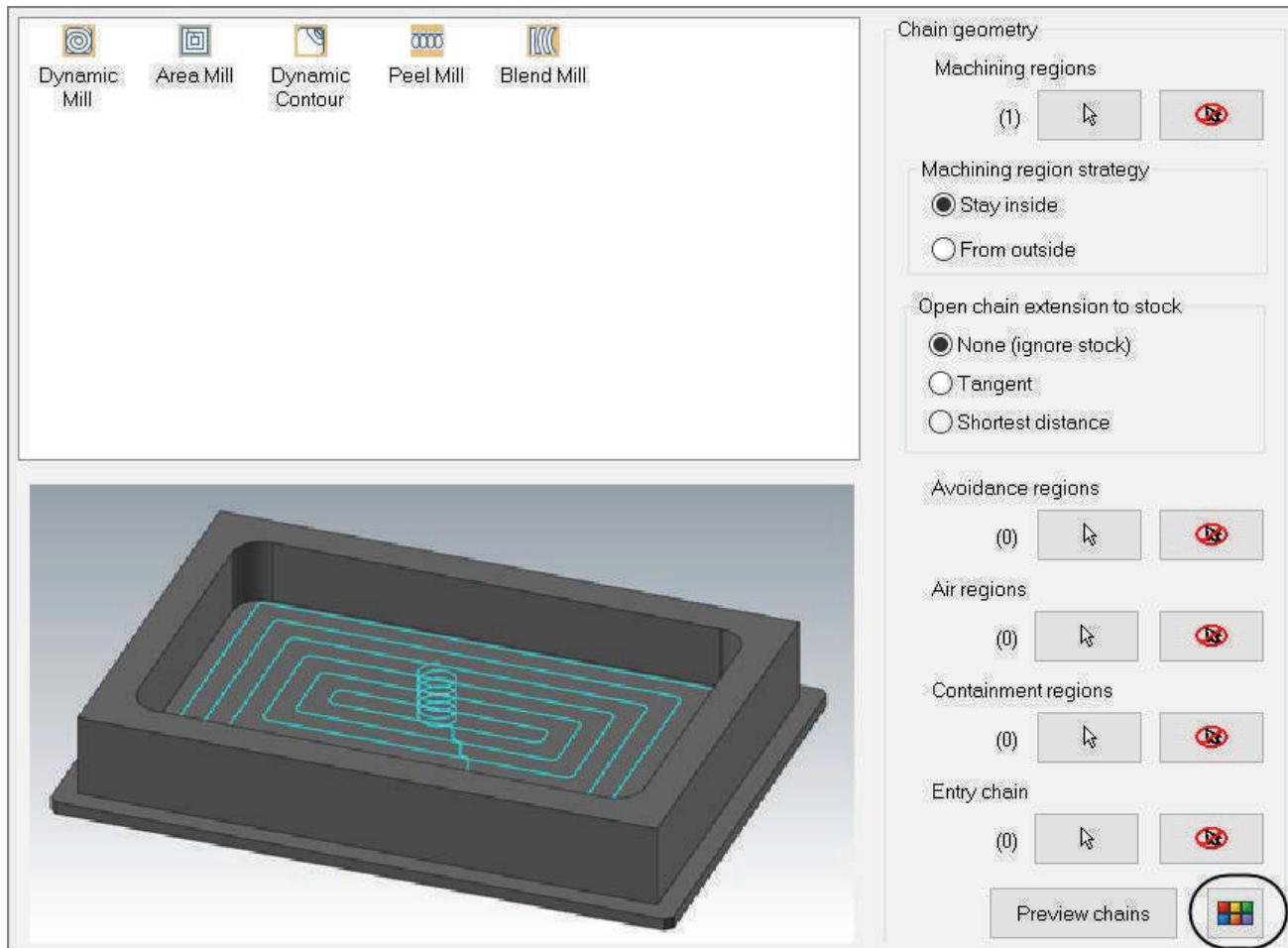


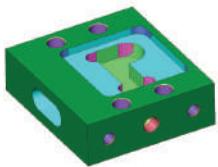


5.2 Preview Chains

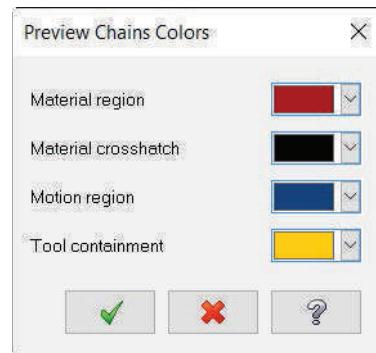
The **Preview Chains** function is intended to give the user a quick visual representation of how Mastercam sees the various pieces of geometry that have been selected, how they interact with one another and a general overview of how the toolpath will be calculated with the selections presently made.

- ♦ Click on the **Color** icon to see the legend for **Preview chains** as shown.





- The Preview Chains Colors dialog box should look as shown.

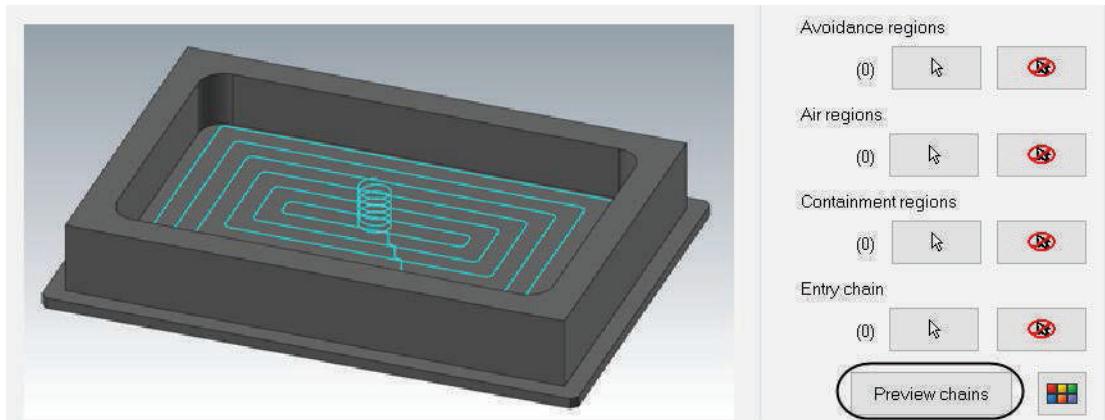


The **Material region** and **Material crosshatch** are the two colors that are used to define the material to be cut. The default colors are red for the background and black for the crosshatch.

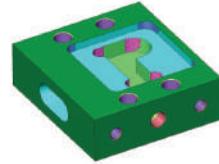
The **Motion region** displays the area that Mastercam is making available to the toolpath for motion if it needs it. The color to represent it is dark blue. The primary reason for the display of the entire available (but not necessarily used) Motion region is to help the user visualize how the tool may move near or interact with any adjacent geometry.

The **Tool containment** is what you have selected as the Containment region in the chain geometry. If you have not selected a containment region, it will default to the outside of the Motion region since that is currently the default area the toolpath is being contained to. The color used to represent the Tool containment is yellow.

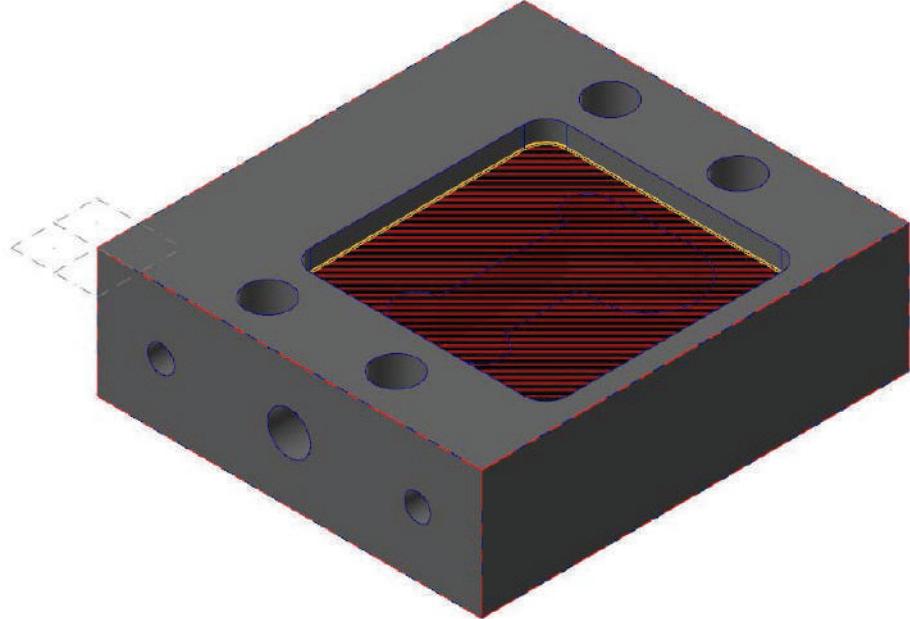
- Select the **OK** button to exit **Preview Chains Colors**.
- Select the **Preview chains** button as shown.



- Select the **Hide dialog** button to see the preview in the graphics window.



- ◆ The **Preview chains** should look as shown.

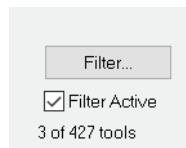


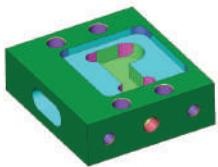
- ◆ Press the **Esc** key to return to the toolpath parameters.
- ◆ Click on the **Preview chains** button again to clear the Preview chains display.



5.3 Select a 3/4" Flat endmill from the library and set the Tool parameters

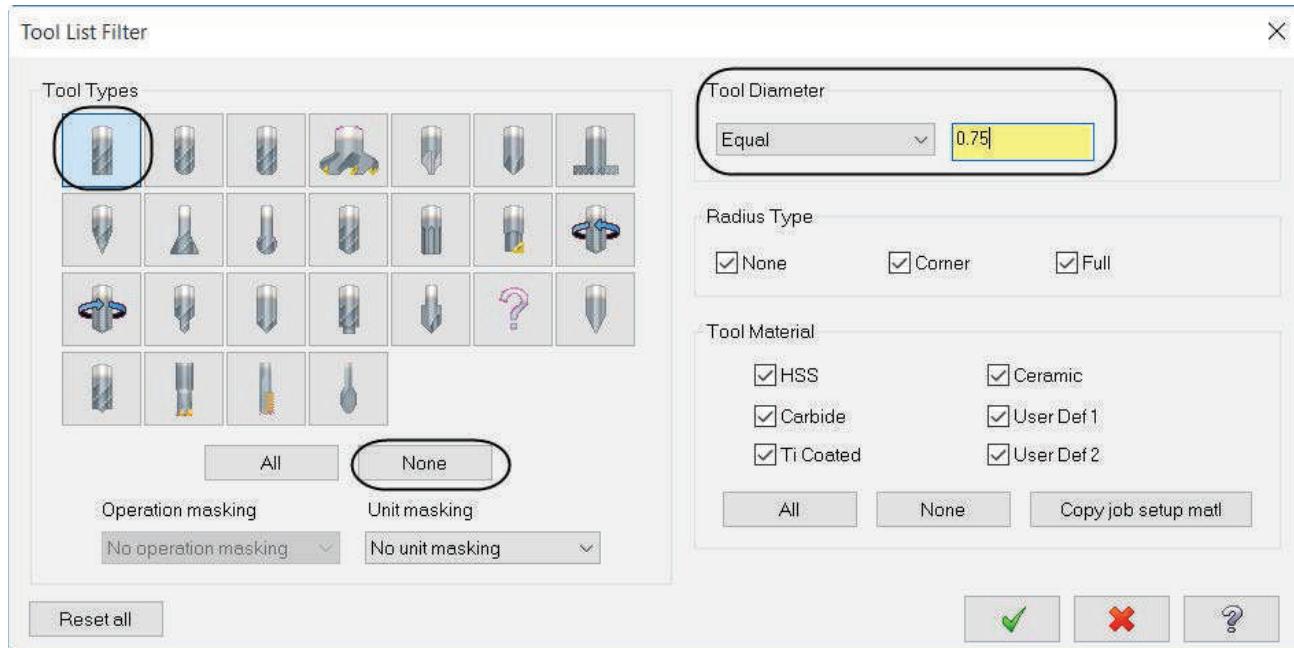
- ◆ Select **Tool** from the **Tree View** list.
- ◆ Click on the **Select library tool** button. Select library tool...
- ◆ Select the **Filter** button as shown.





- ♦ Select the **None** button and then under **Tool Types** choose the **Flat Endmill** icon.
- ♦ Under tool diameter pick **Equal** and input a value of **0.75** as shown in [Figure: 5.3.1](#).

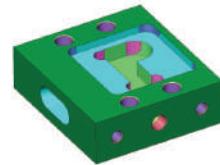
Figure: 5.3.1



- ♦ Select the **OK** button to exit the **Tool List Filter**.
- ♦ In the **Tool Selection** dialog box you should only see a **3/4" Flat Endmill**.

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	Type	# Flutes	Ra...
294	-	3/4 FLAT ...	-	0.75	0.0	2.0	En...	4	No...

- ♦ Select the **3/4" Flat Endmill** in the **Tool Selection** page and then select the **OK** button to exit.



- ♦ Make all of the necessary changes as shown in [Figure: 5.3.2.](#)

Figure: 5.3.2

#	Assembly Name	Tool Name	Holder Name	Dia.
1	-	3/4 FLAT ...	-	0.75

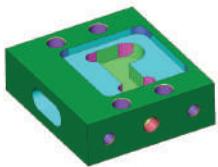
Tool diameter:	0.75		
Corner radius:	0.0		
Tool name:	3/4 FLAT ENDMILL		
Tool #:	1	Length offset:	1
Head #:	0	Diameter offset:	1

<input type="checkbox"/> RCTF	Spindle direction: CW
Feed rate: 6.4176	Spindle speed: 713
FPT: 0.0023	SFM 139.9869
Plunge rate: 6.4176	Retract rate: 6.4176
<input type="checkbox"/> Force tool change	<input checked="" type="checkbox"/> Rapid Retract

Comment

Use Area Mill to machine the pocket.

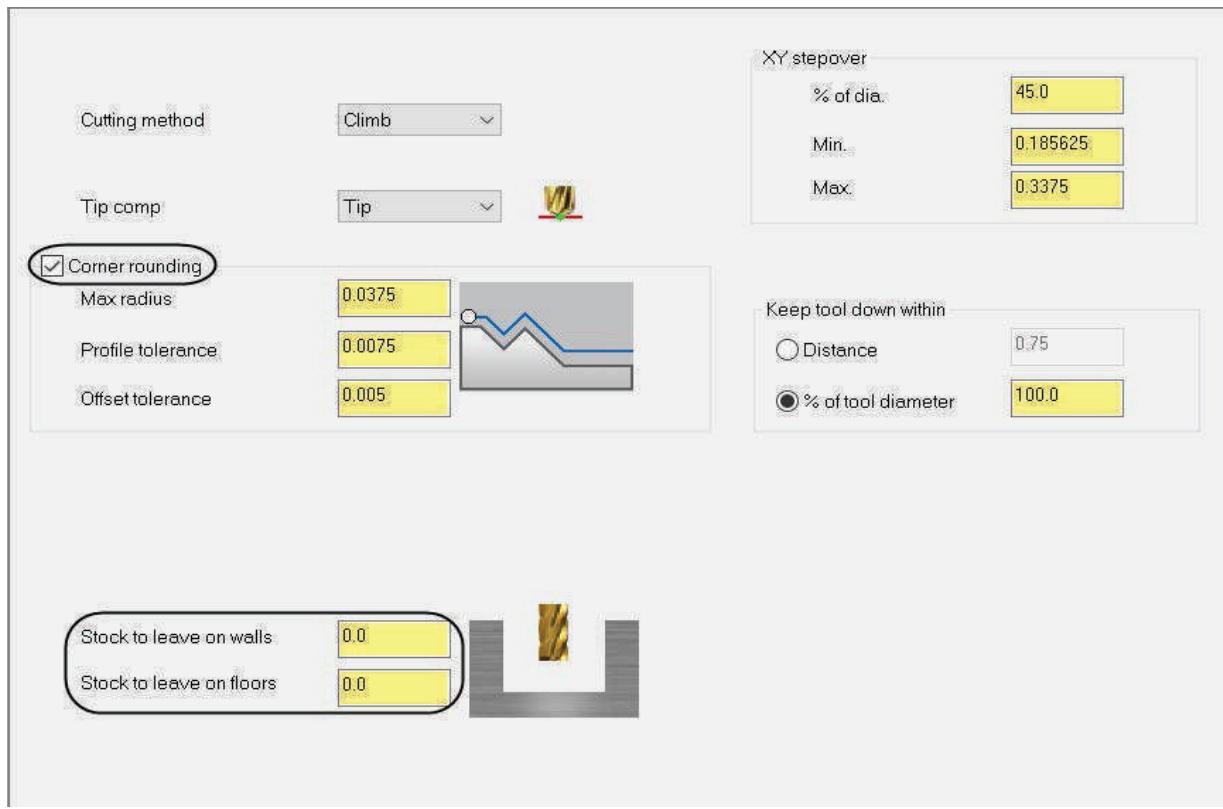
Select library tool...	<input type="checkbox"/> Filter Active	Filter...
<input type="checkbox"/> Tool inspection / change		
Force retract every	<input checked="" type="radio"/> 0.0 <input type="radio"/> 0.0	Inches
		Minutes
<input type="checkbox"/> To batch		



5.4 Set the Cut Parameters

- From the Tree View list, select **Cut Parameters**.
- Enable **Corner rounding** and ensure the settings appear as shown in [Figure: 5.4.1](#).

Figure: 5.4.1

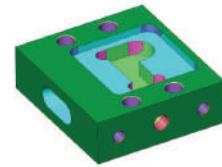


Corner rounding replaces sharp corners with arcs for faster and smoother transitions in tool direction. For more information on the parameters select the **Help** button  and then the **Field definitions** tab.

Max radius inputs the radius of the largest arc that you will allow Mastercam to insert to replace a corner. Larger arcs will result in a smoother toolpath but with a greater deviation from the part corner.

Profile tolerance represents the maximum distance that the outermost profile of a toolpath with corner rounding can deviate from the original toolpath.

Offset tolerance represents the maximum distance that a profile of a toolpath created with corner rounding can deviate from the original toolpath.



5.5 Set the Depth Cuts Parameters

- From the **Tree View** list, select **Depth Cuts** and make sure it is disabled as shown in [Figure: 5.5.1](#).

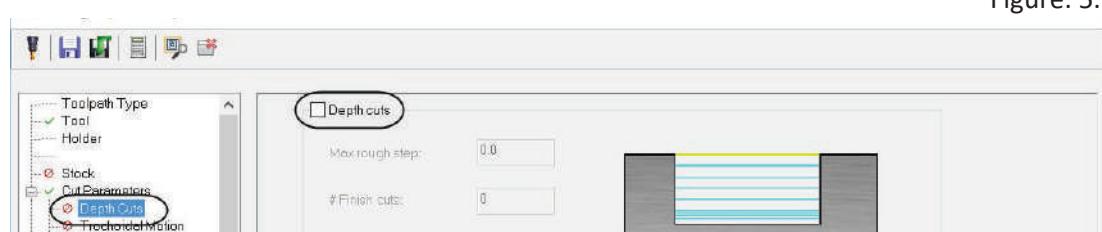


Figure: 5.5.1

5.6 Set the Transitions

- From the **Tree View** list, select **Transitions**.
- Enable **Entry helix**, set the **Radius** to **0.500** and ensure the parameters are the same as shown in [Figure: 5.6.1](#).

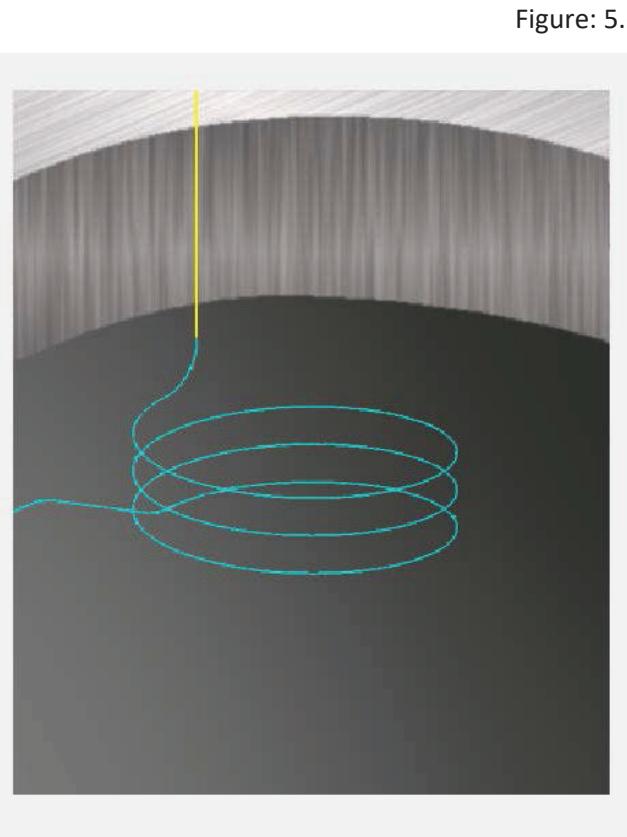
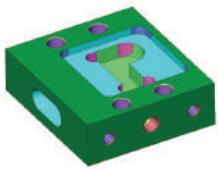


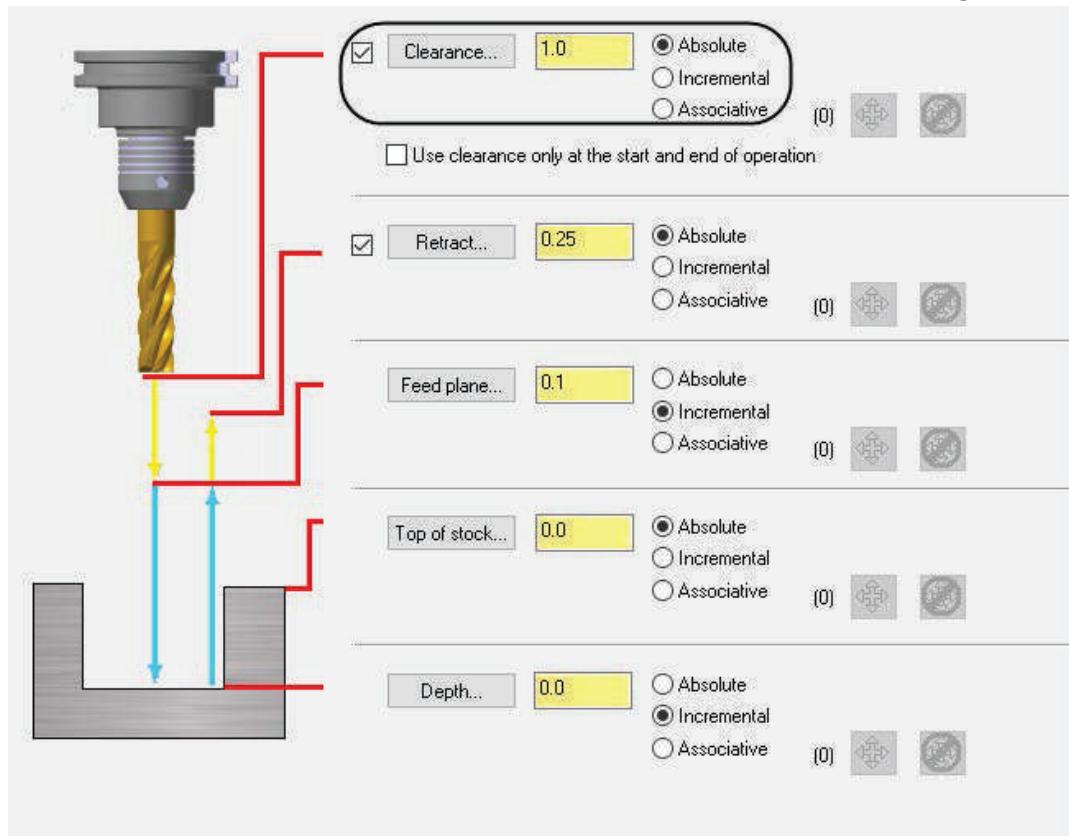
Figure: 5.6.1



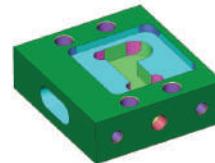
5.7 Set the Linking Parameters

- ♦ Select **Linking Parameters**, enable **Clearance** and input a value of **1.0**.
- ♦ You will notice the depth has been input based on the geometry we selected as shown in [Figure: 5.7.1](#).

Figure: 5.7.1



NOTE: The **Depth** set to **Incremental** and **zero** is relative to the location of the chained geometry which was selected at the bottom of the pocket.

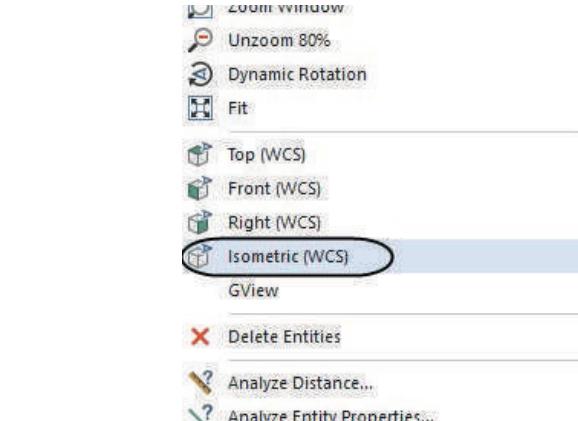


5.8 Preview the Toolpath

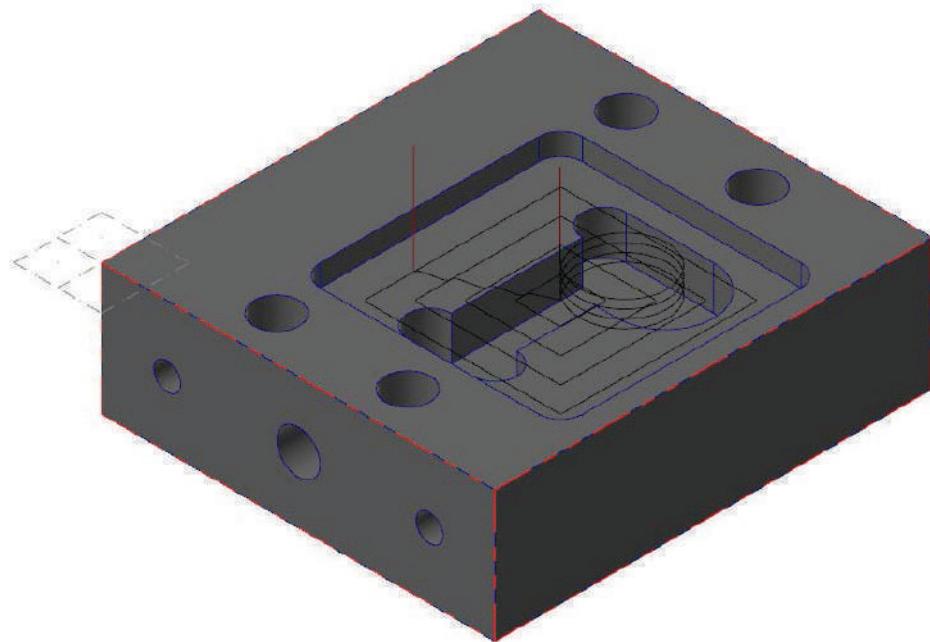
- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.
- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.

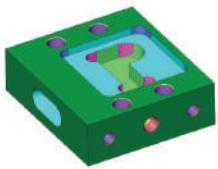


- ♦ Press the **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Once complete, select the **OK** button to generate the toolpath.

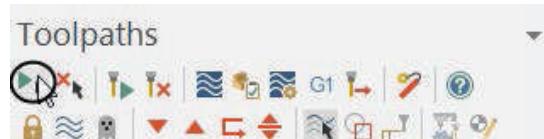




STEP 6: BACKPLOT THE TOOLPATHS

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 additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates and the cycle time. It also shows any collisions between the workpiece and the tool.

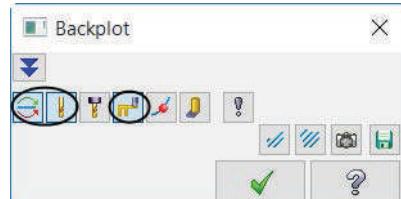
- ♦ Make sure that the toolpaths are selected (signified by the green check mark on the folder). If the operation is not selected choose the **Select all operations** icon.



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



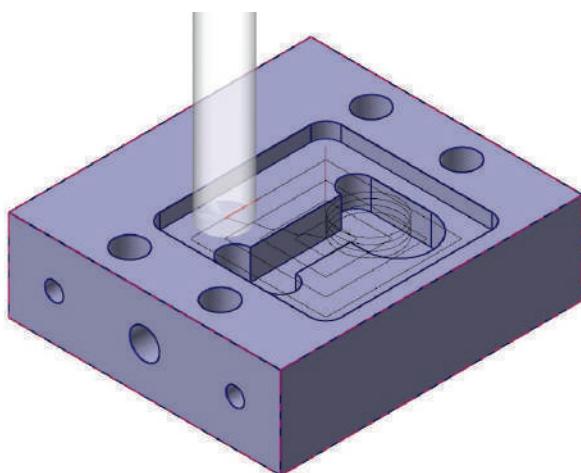
- ♦ In the **Backplot** dialog box, enable **Display with color codes**, **Display tool** and **Display rapid moves** icons as shown.



- ♦ Select the **Play** button to run **Backplot**.

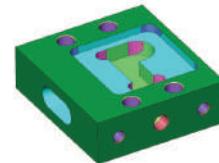


- ♦ After **Backplot** is completed, the toolpath should look as shown.



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





STEP 7: SIMULATE THE TOOLPATH IN VERIFY

Verify Mode shows the path the tools take to cut the part with material removal. This display lets you spot errors in the program before you machine the part. As you verify toolpaths, Mastercam displays additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates and the cycle time. It also shows any collisions between the workpiece and the tool.

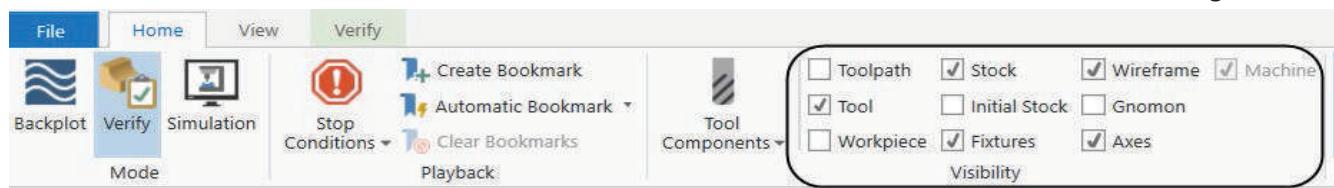
- From the **Toolpaths Manager**, select the **Verify selected operations** icon as shown.



NOTE: Mastercam launches a new window that allows you to check the part using **Verify**.

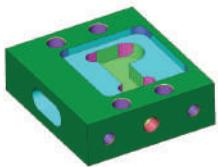
- Disable **Workpiece** in the **Visibility** group as shown in [Figure: 7.0.1](#).

Figure: 7.0.1

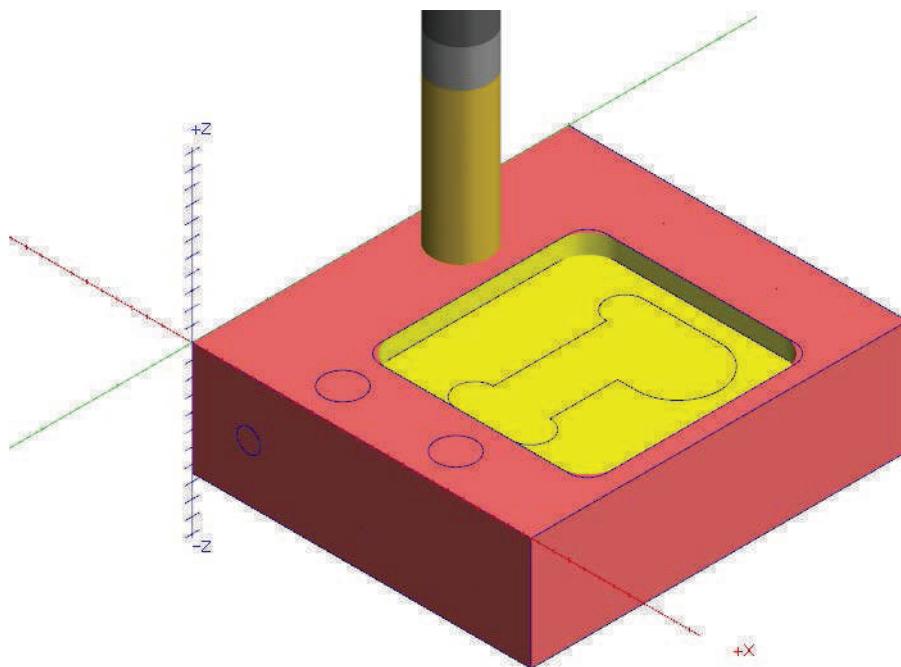


- Select the **Play** button to run **Verify**.





- The part will appear as shown.

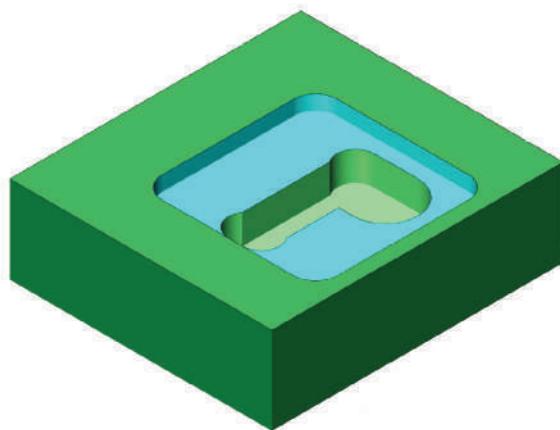


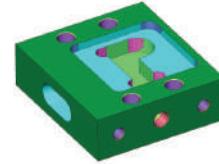
- To go back to the **Mastercam** window, minimize the **Mastercam Simulator** window as shown.

STEP 8: 2D HIGH SPEED AREA MILL

In this step you will learn how to copy a toolpath and reselect geometry. The main advantage of copying a toolpath is the parameters for the 1st toolpath remain intact for the second toolpath.

Toolpath Preview:



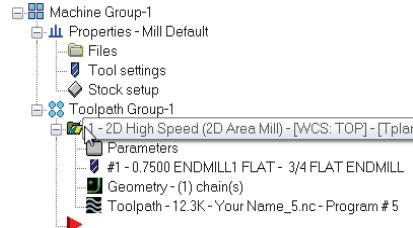


- To remove the toolpath display, from the Toolpaths Manager, click on the **Toggle display on selected operations** button or press **Alt + T**.

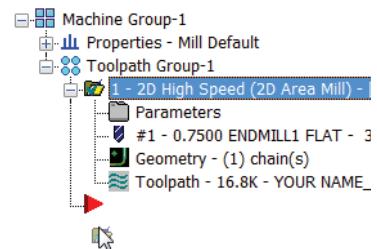


8.1 Copy the Previous Toolpath

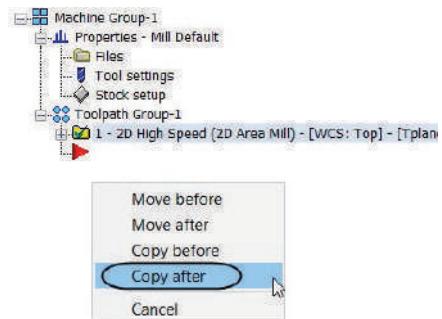
- Select Operation #1.



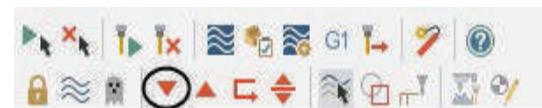
- Right click, hold the right mouse button down, and drag the operation to a point below it as shown.

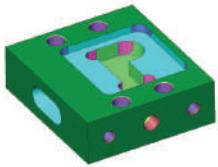


- Release the right mouse button and select the option **Copy after**.



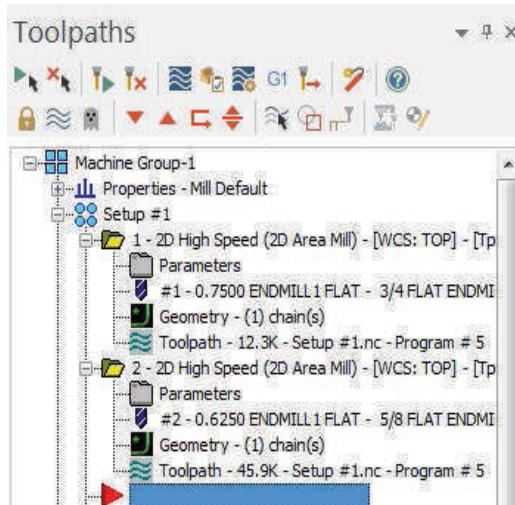
- Select the **Move insert arrow down one item** button to move the insert arrow down.





- The **Insert Arrow** should appear at the bottom of the list as shown in [Figure: 8.1.1](#).

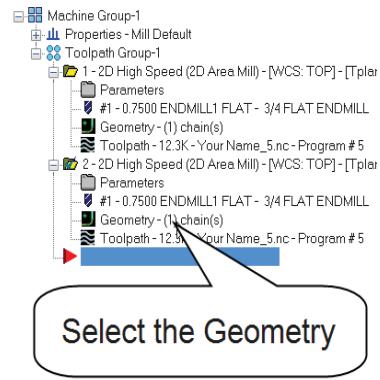
Figure: 8.1.1



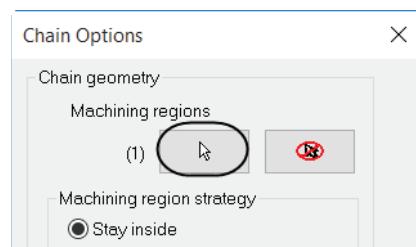
The **Insert Arrow** controls where the new operation will be inserted.

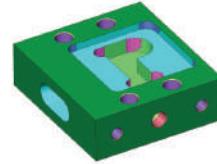
8.2 Re-Chain the Geometry

- In Operation #2 pick the **Geometry** as shown.

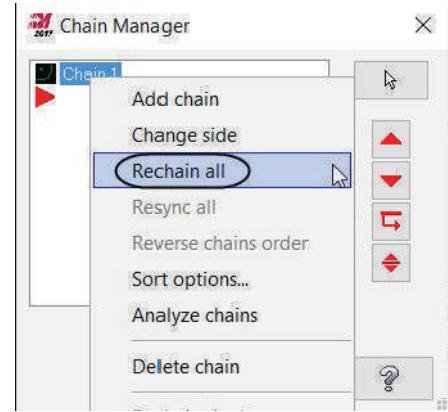


- Click on the **Select** button as shown.

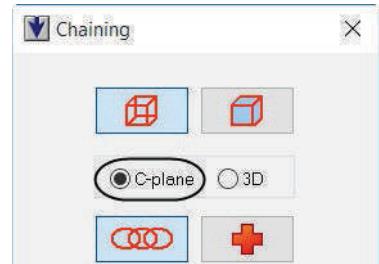




- ♦ When the **Chain Manager** appears, select **Chain 1**.
- ♦ Right click and pick the **Rechain all** option as shown.

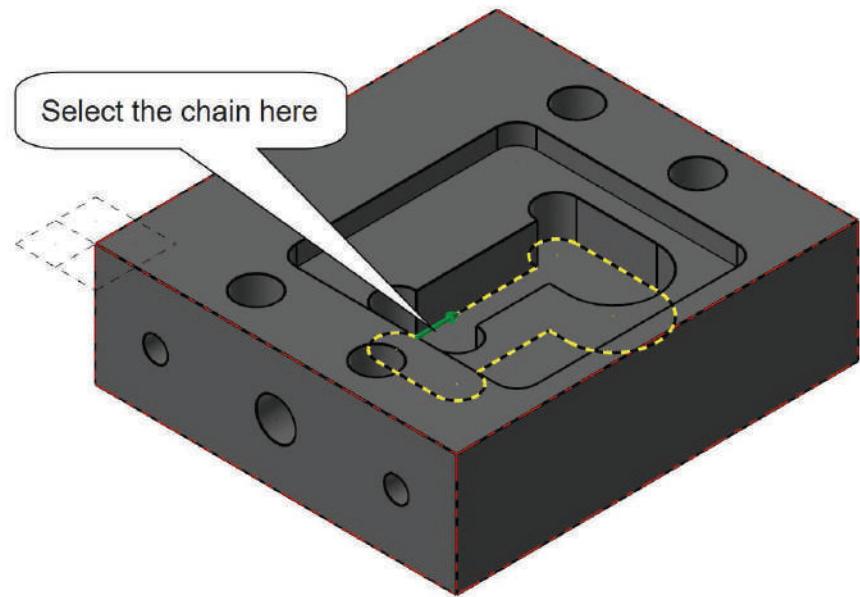


- ♦ When the **Chaining** dialog box appears select **C-plane**.



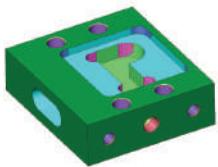
- ♦ Select the bottom of the pocket as shown in [Figure: 8.2.1](#).

Figure: 8.2.1



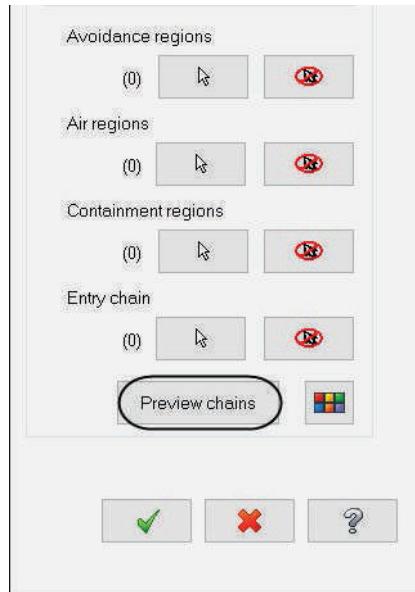
- ♦ Once the geometry has been selected choose the **OK** button to exit the **Chaining** dialog box. 

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

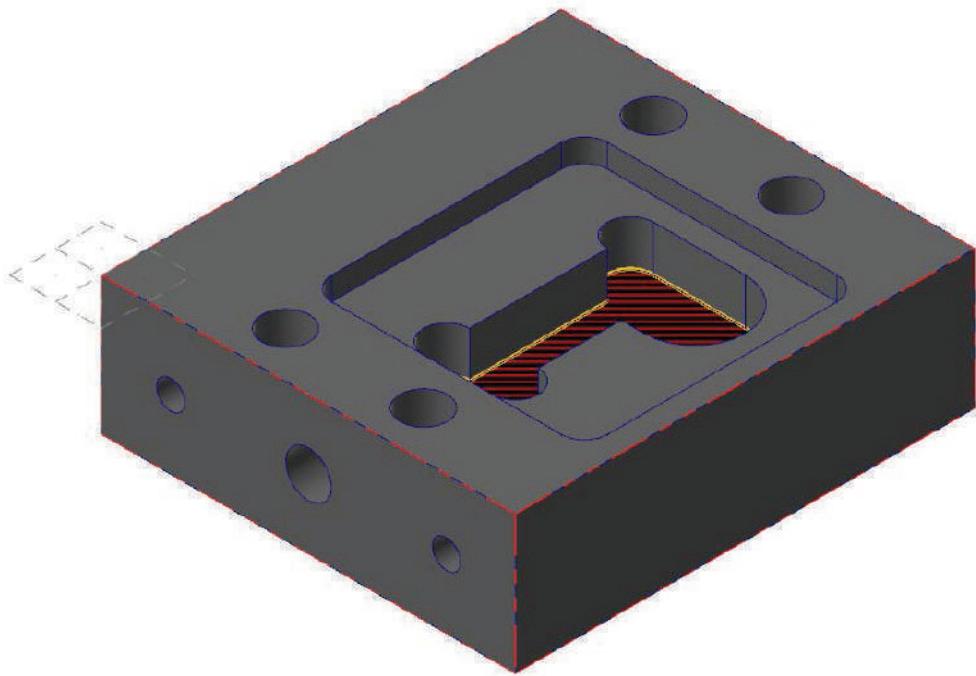


8.3 Preview Chains

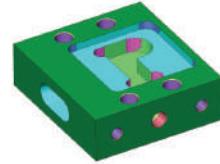
- In the **Chain Options** select the **Preview chains** button as shown.



- The **Preview chains** should look as shown.

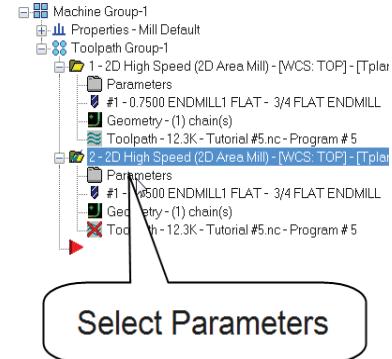


- Select the **OK** button to exit the **Chain Options** dialog box.
- Press the **Esc** key to return to the toolpath parameters.
- Click on the **Preview chains** button again to clear the **Preview chains** display.



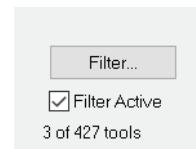
8.4 Select a 5/8" Flat Endmill from the Library and set the Tool Parameters

- ◆ Choose **Parameters** under Operation #2.



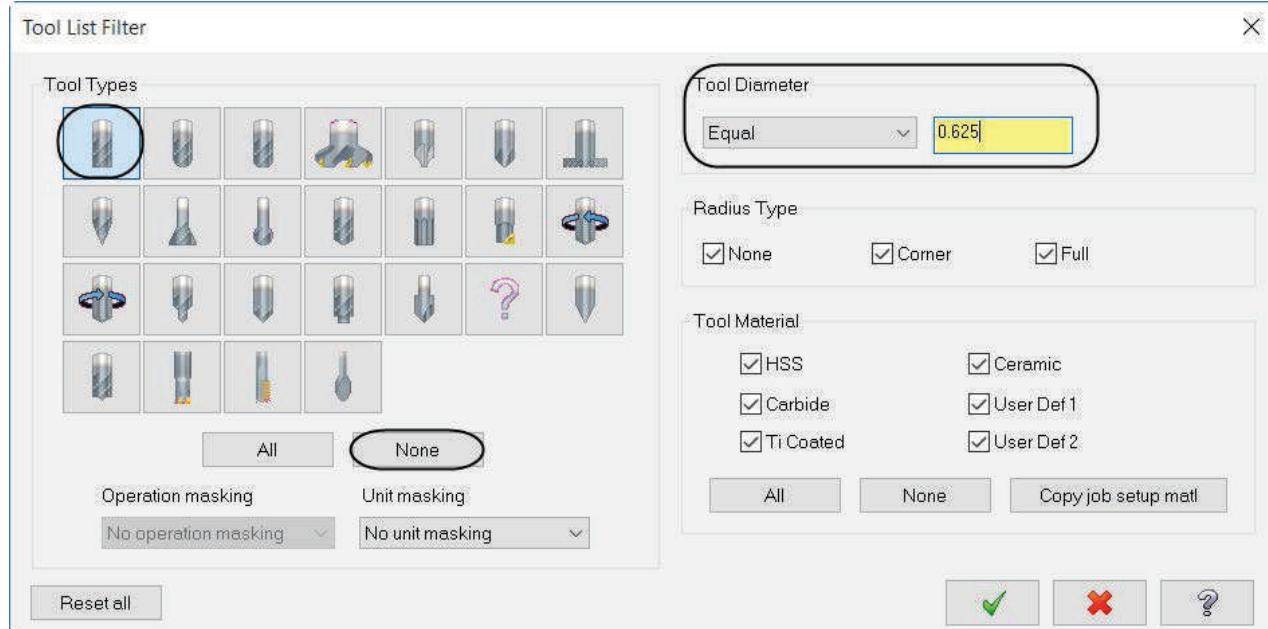
- ◆ Select **Tool** from the **Tree View** list.

- ◆ Click on the **Select library tool...** button.
- ◆ Select the **Filter** button as shown.

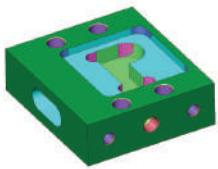


- ◆ Select the **None** button and then under **Tool Types** choose the **Flat Endmill** icon.
- ◆ Under **Tool Diameter**, pick **Equal** and input a value of **0.625** as shown in [Figure: 8.4.1](#).

Figure: 8.4.1



- ◆ Select the **OK** button to exit the **Tool List Filter**.

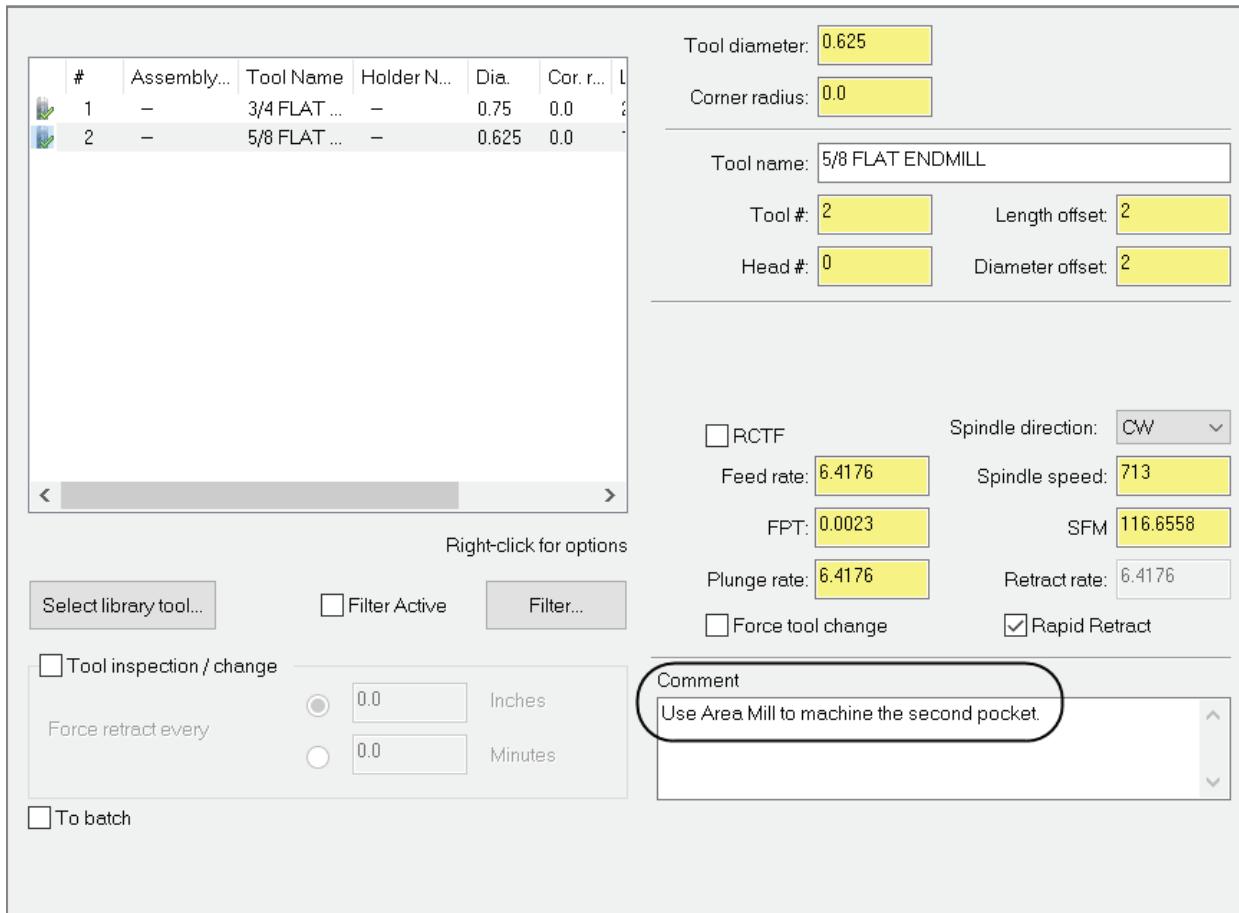


- In the Tool Selection dialog box you should only see a **5/8" Flat Endmill**.

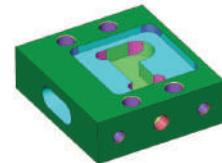
#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type	Ra...
292	-	5/8 FLAT ...	-	0....	0.0	1.5	4	En...	No...

- Select the **5/8" Flat Endmill** in the Tool Selection page and then select the **OK** button to exit.
- Make all the necessary changes as shown in [Figure: 8.4.2](#).

Figure: 8.4.2



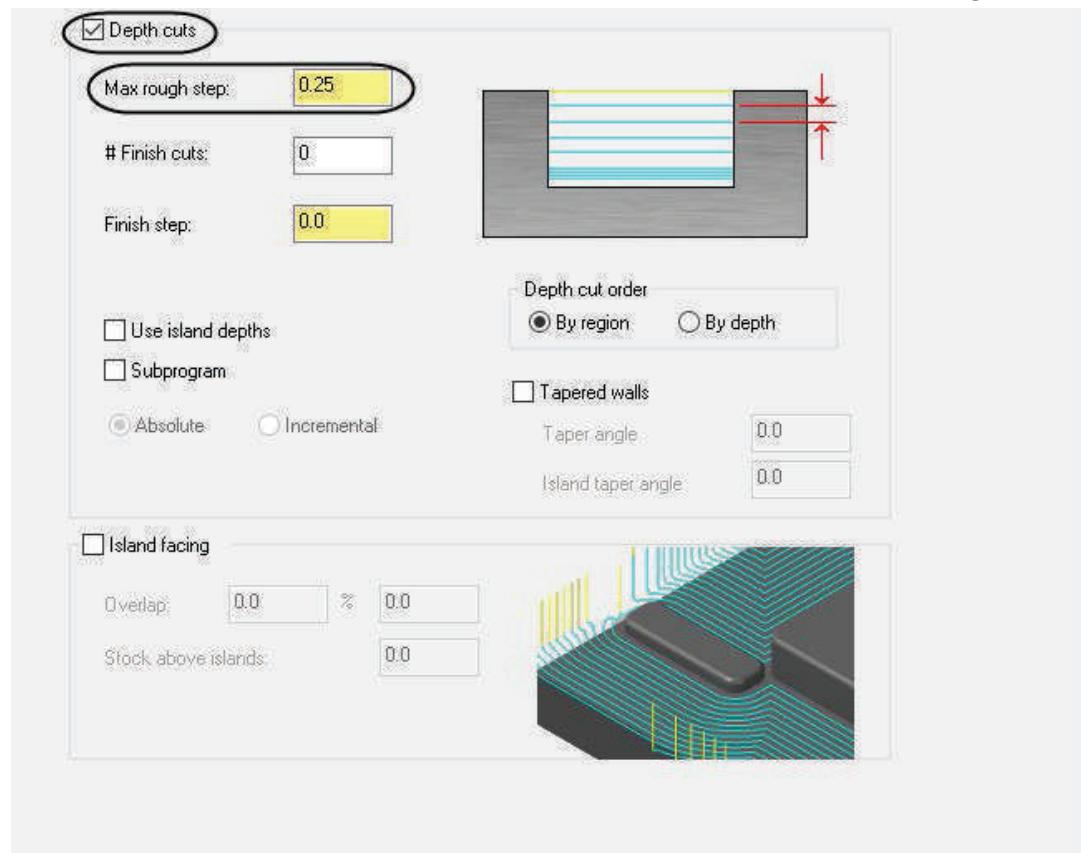
NOTE: Since this toolpath has been copied all the parameters remain the same. Therefore the only parameters shown are the ones we will be changing.

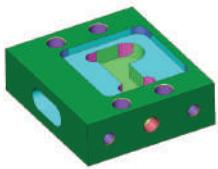


8.5 Set the Depth Cuts parameters

- From the Tree View list, select the **Depth Cuts** and enable **Depth Cuts**.
- Input a **Max rough step** of **0.25** as shown in [Figure: 8.5.1](#).

Figure: 8.5.1

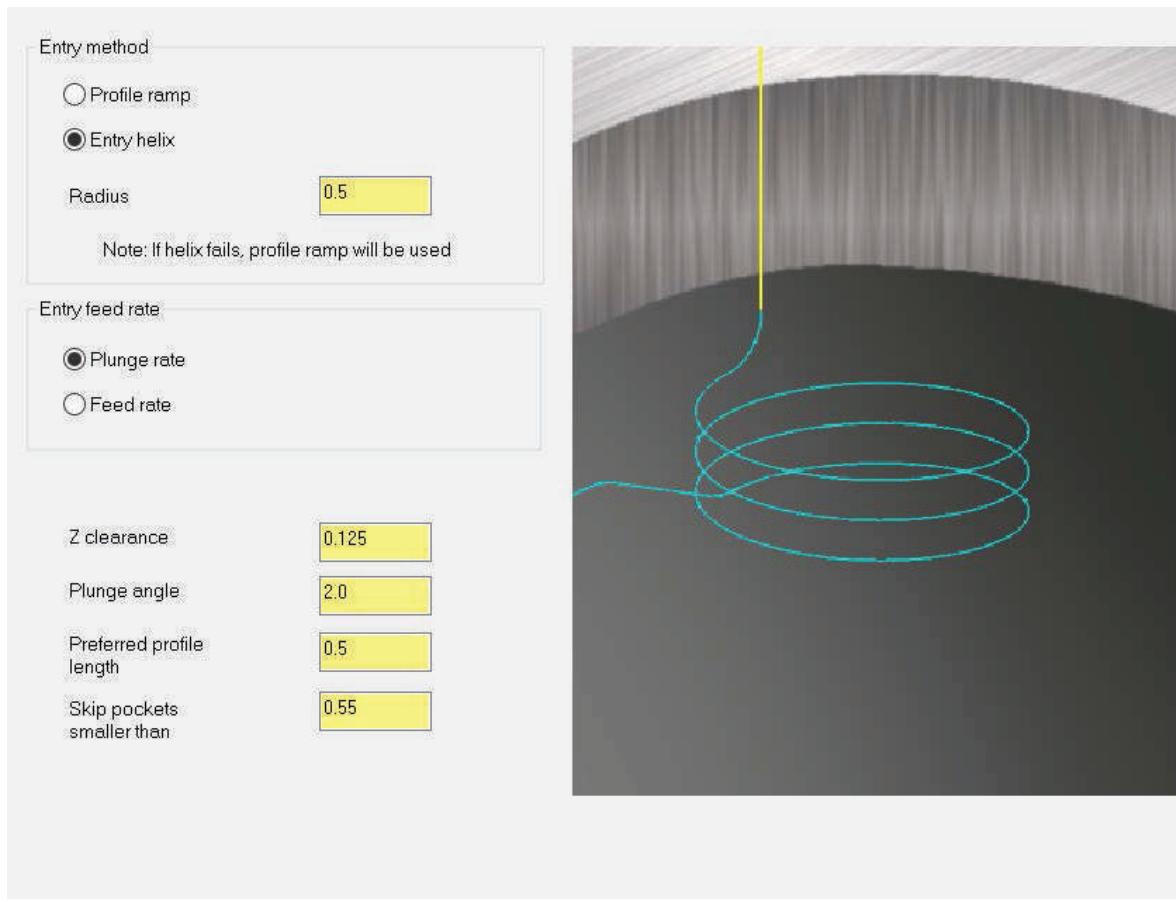




8.6 Set the Transitions

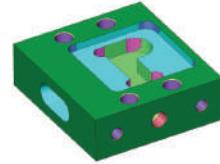
- From the Tree View list, select **Transitions**. Enable **Entry helix** and enter a **Radius** of **0.5** as shown in [Figure: 8.6.1](#).

Figure: 8.6.1



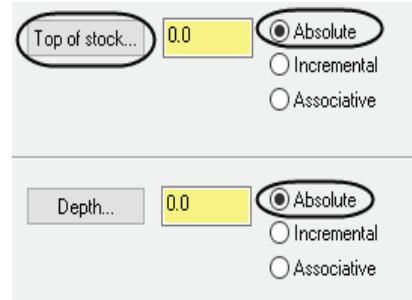
Profile Ramp creates a ramp motion to descend the tool.

Preferred profile length enters a minimum size for the profile in order for a ramp to be created.



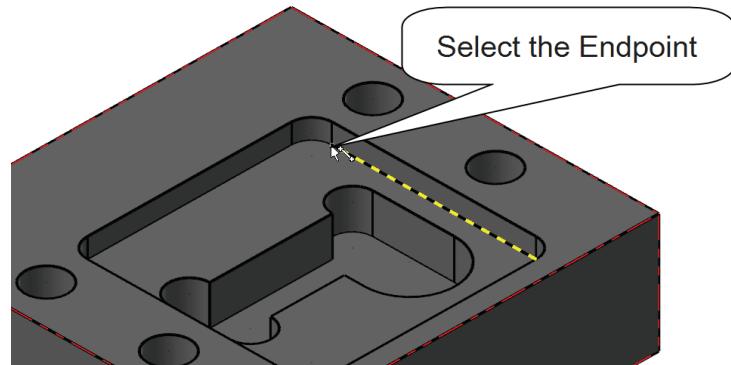
8.7 Set the Linking Parameters

- ♦ Select **Linking Parameters** from the **Tree View** list.
- ♦ Set the **Top of Stock** and the **Depth** to **Absolute**.
- ♦ Select the **Top of Stock** button (this will bring you back to the graphics screen).



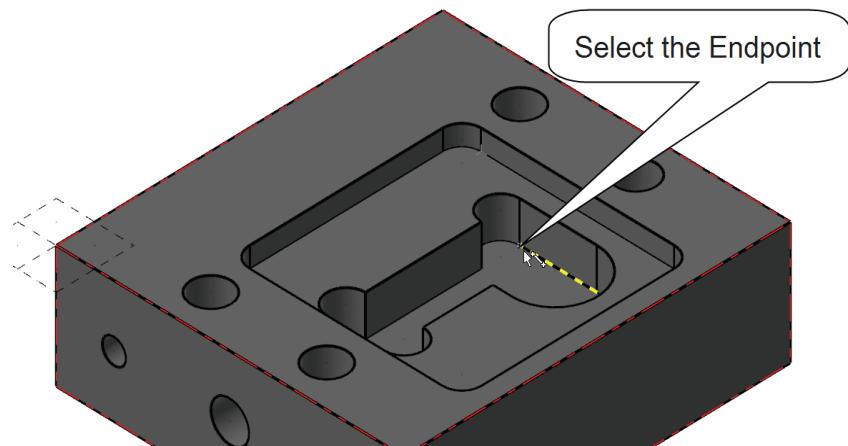
- ♦ Pick the line endpoint as shown in [Figure: 8.7.1](#).

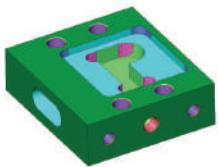
Figure: 8.7.1



- ♦ Choose the **Depth** button (this will bring you back to the graphics screen).
- ♦ Pick the line endpoint as shown in [Figure: 8.7.2](#).

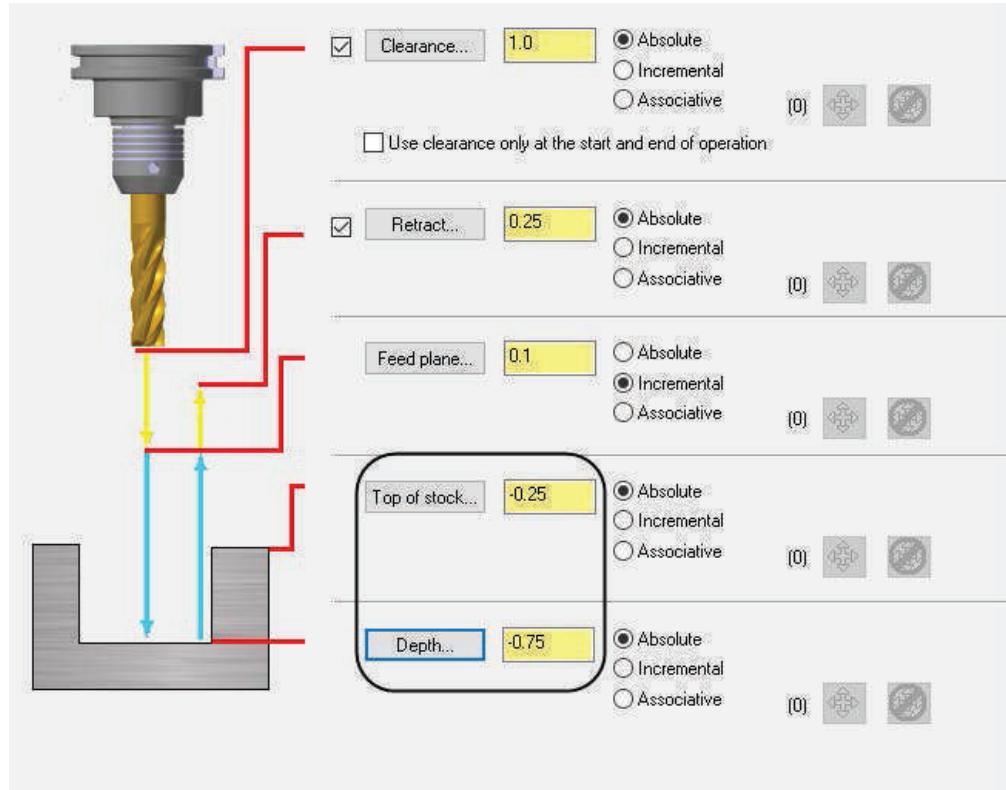
Figure: 8.7.2

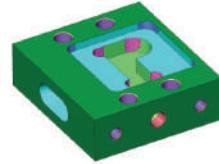




- Top of Stock will be set to **-0.25**, and the Depth will be set to **-0.75** as shown in [Figure: 8.7.3.](#)

Figure: 8.7.3



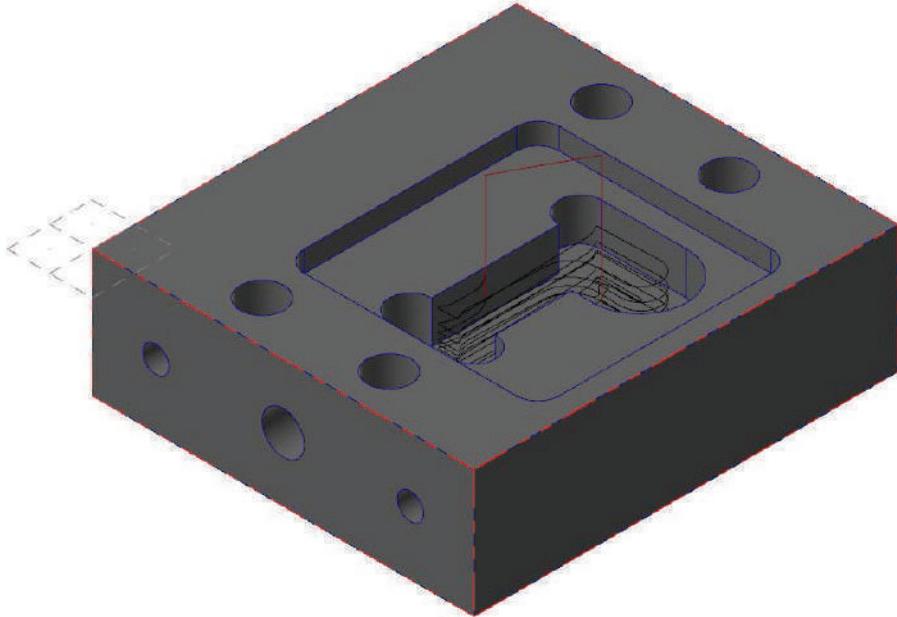


8.8 Preview the Toolpath

- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- ♦ The toolpath should look as shown.

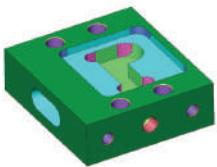


- ♦ Press the **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

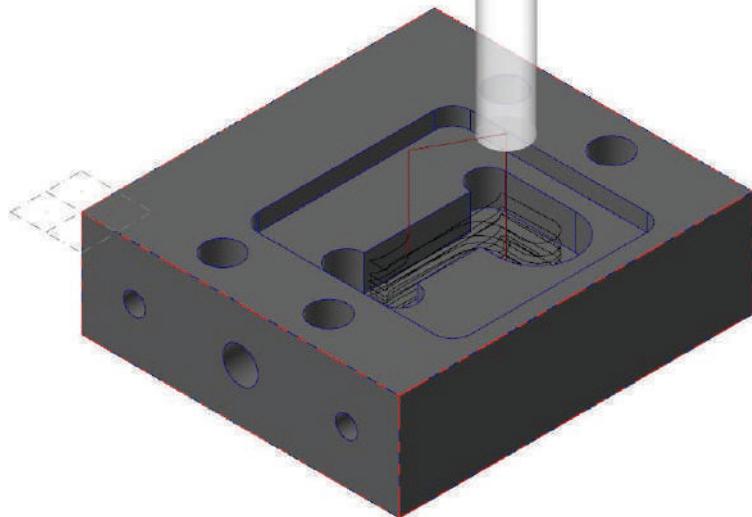
- ♦ Select the **OK** button to generate the **Area Mill** toolpath.
- ♦ Choose to **Regenerate all dirty operations**.





8.9 Backplot the toolpath

- Once the operation has been regenerated, **Backplot** the toolpath. See [page 447](#) to review these procedures.

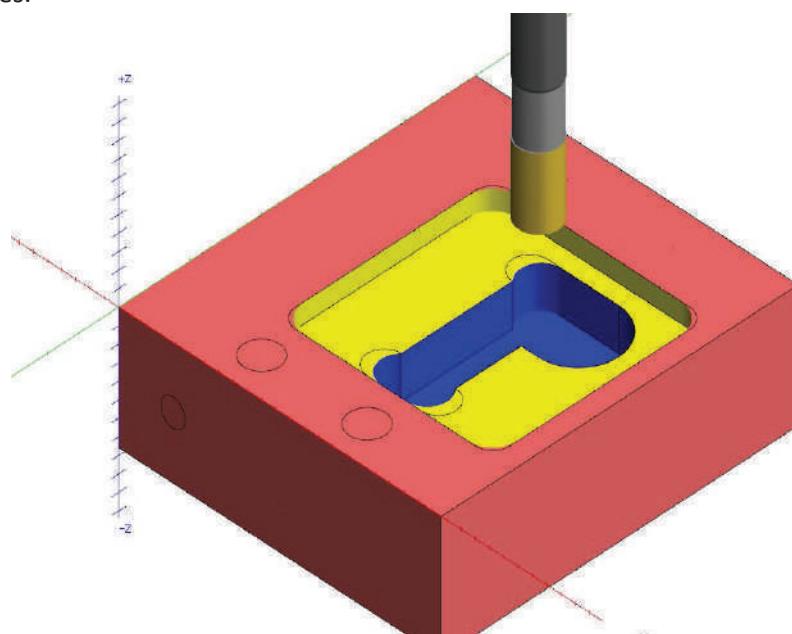


- Select the **OK** button to exit **Backplot**.



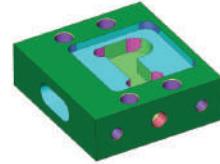
8.10 Verify the toolpaths

- To verify all toolpaths, from the **Toolpaths Manager**, choose the **Select all operations** icon.
- See [page 448](#) to review these procedures.



- To go back to the **Mastercam** window, minimize the **Mastercam Simulator** window as shown.

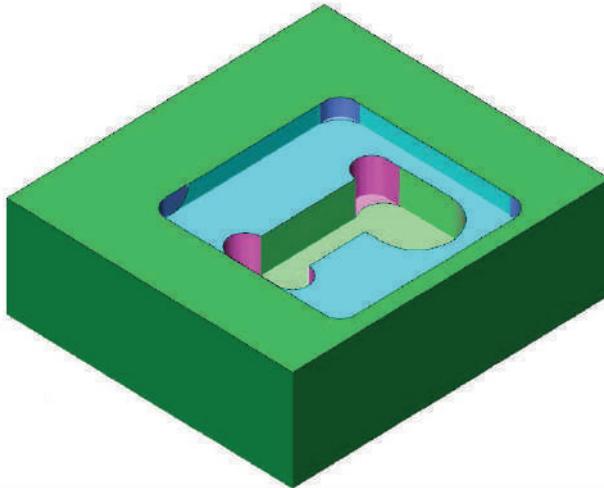




STEP 9: AREA MILL TOOLPATH TO REMACHINE THE MATERIAL LEFT BEHIND

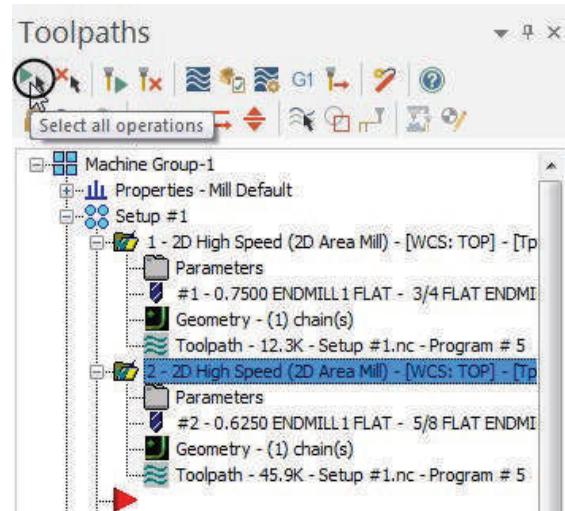
Using the **Area Mill** toolpath to machine the material left behind by the previous toolpaths.

Toolpath Preview:

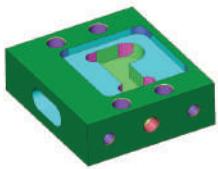


9.1 Copy the previous Toolpaths

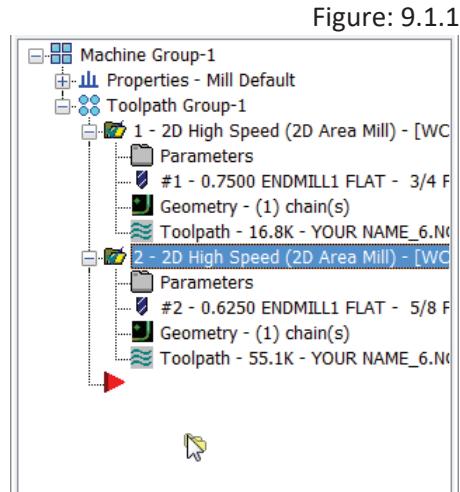
- ♦ Pick the **Select all operations** button.



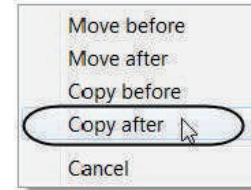
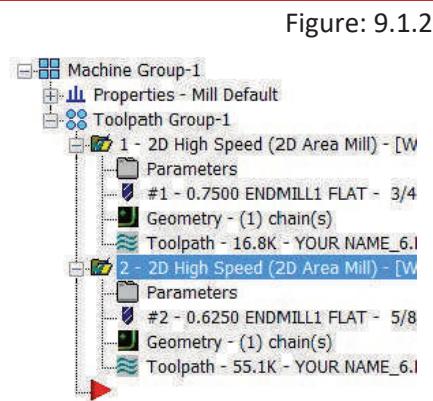
NOTE: Both toolpaths should be selected as shown.



- ♦ Right click, hold the right mouse button down and drag the operation to a point below it as shown in [Figure: 9.1.1](#).



- ♦ Release the right mouse button and select the option **Copy after** as shown in [Figure: 9.1.2](#).

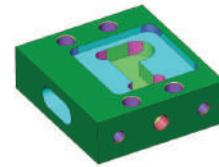


- ♦ Pick the button twice to move the insert arrow down as shown.

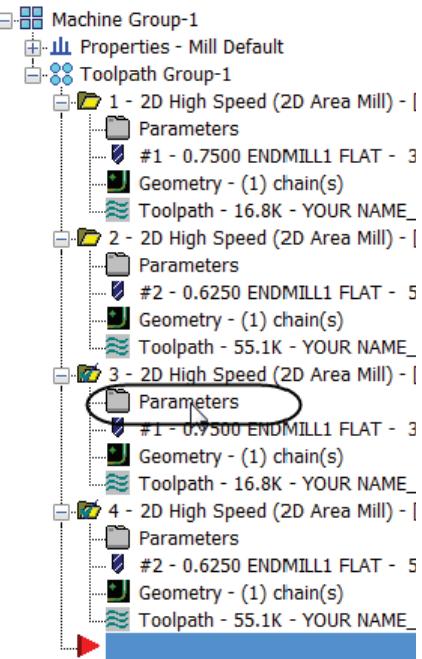


TUTORIAL #5

AREA MILL TOOLPATH TO REMACHINE THE MATERIAL LEFT BEHIND

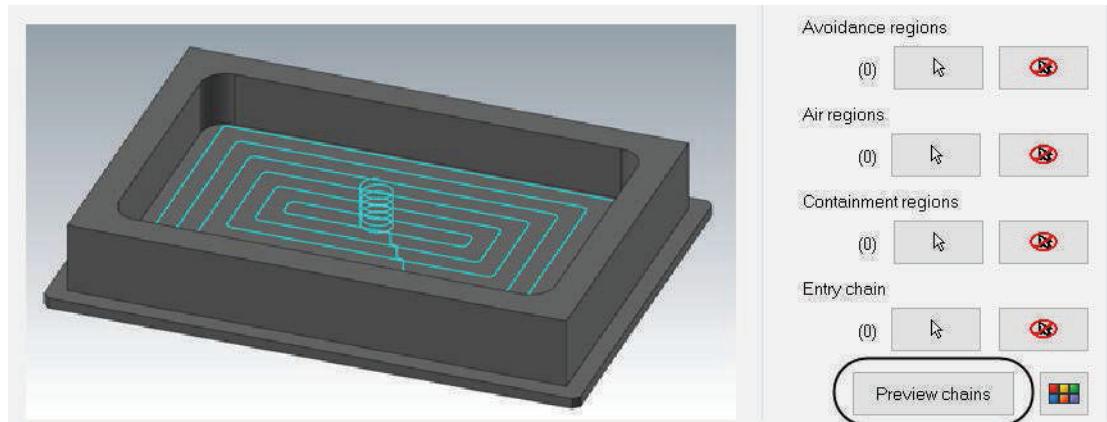


- ◆ Choose **Parameters** under operation #3.

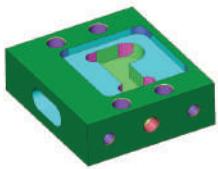


9.2 Preview Chains

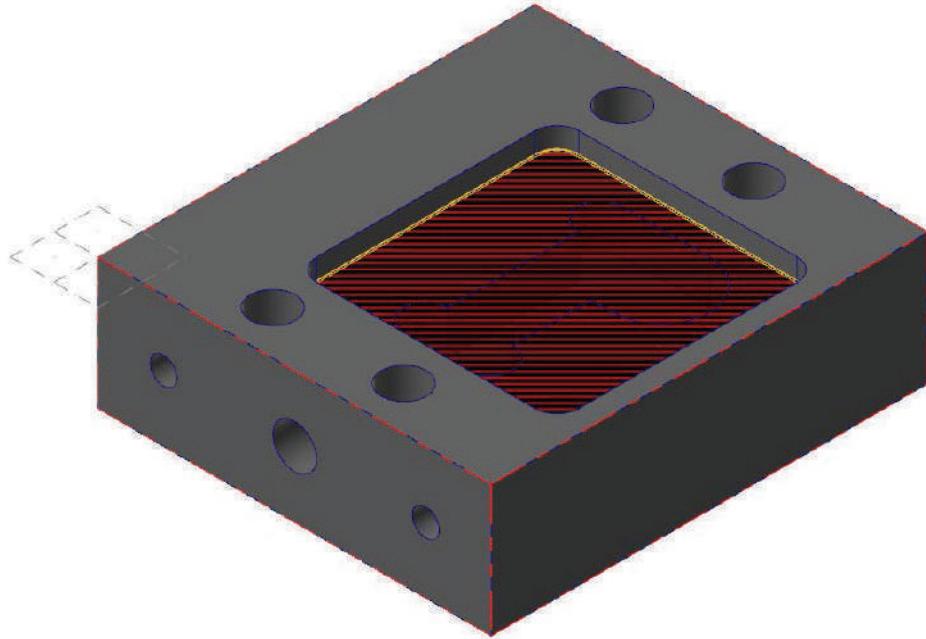
- ◆ In the **Toolpath Type**, select the **Preview chains** button as shown.



- ◆ See page 438 to review the procedure.



- The Preview chains should look as shown.

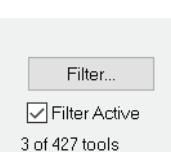


- Press the Esc key to return to the toolpath parameters.
- Click on the Preview chains button again to clear the Preview chains display.



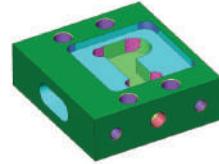
9.3 Select a 3/8" Flat Endmill from the Library and set the Tool parameters

- Select Tool from the Tree View list.
- Click on the Select library tool button.
- Select the Filter button.



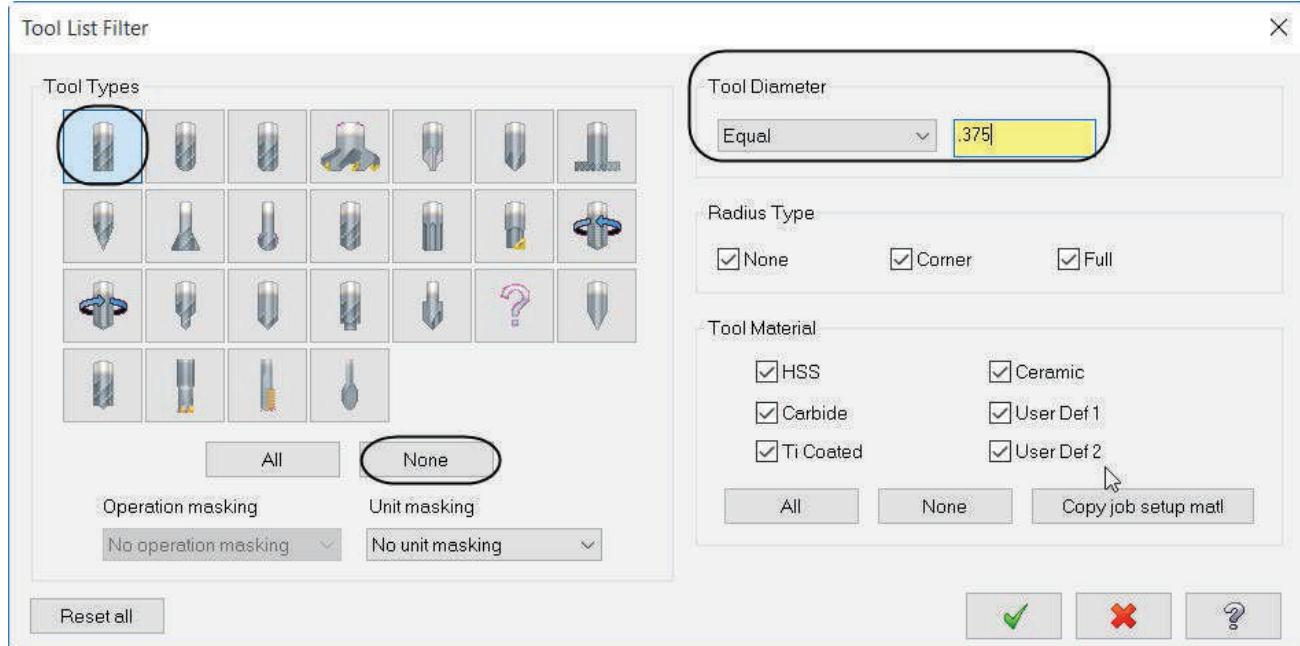
TUTORIAL #5

AREA MILL TOOLPATH TO REMACHINE THE MATERIAL LEFT BEHIND



- ♦ Select the **None** button and then under **Tool Types** choose the **Flat Endmill** icon.
- ♦ Under tool diameter pick **Equal** and input a value of **0.375** as shown in [Figure: 9.3.1](#).

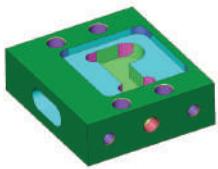
Figure: 9.3.1



- ♦ Select the **OK** button to exit the **Tool List Filter**.
- ♦ In the **Tool Selection** dialog box you should only see a **3/8" Flat Endmill**.

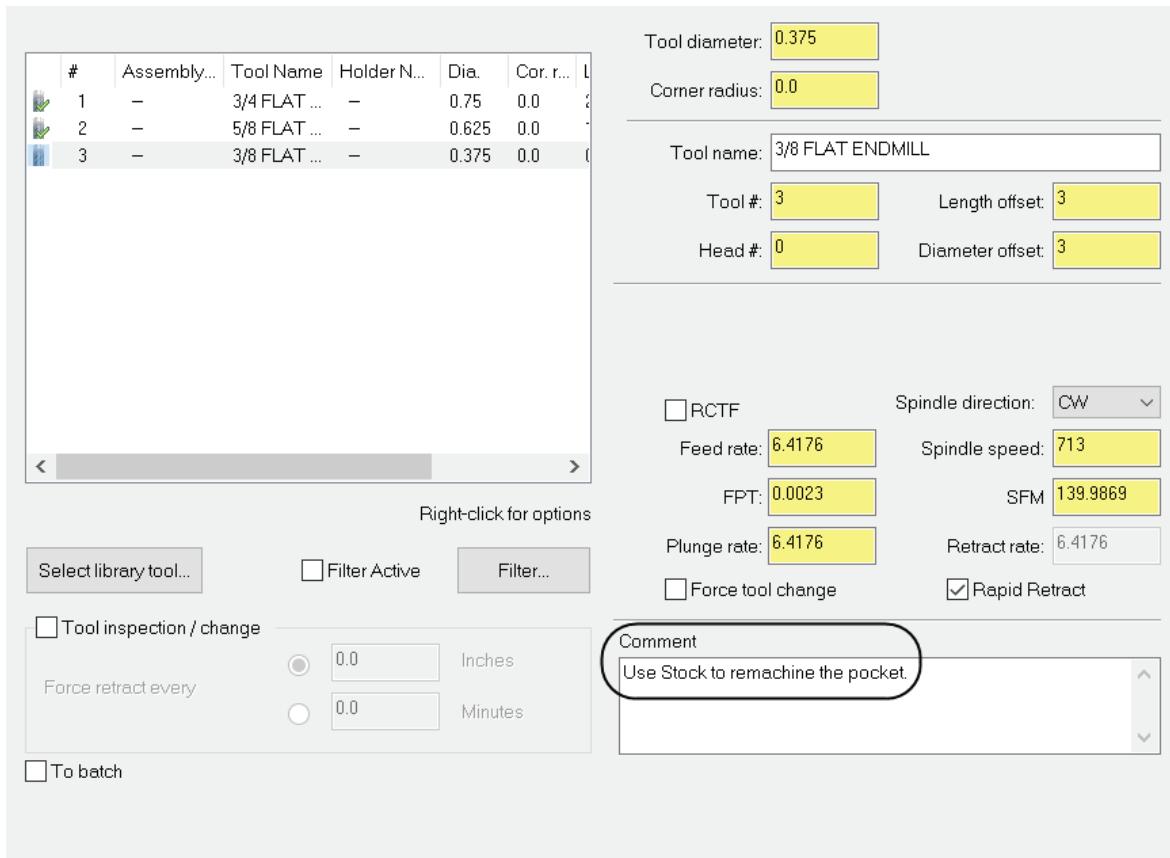
#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Ra...	Type
287	-	3/8 FLAT ...	-	0...	0.0	0.75	4	No...	En...

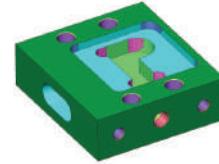
- ♦ Select the **3/8" Flat Endmill** in the **Tool Selection** page and then select the **OK** button to exit.



- ♦ Make the necessary changes as shown in [Figure: 9.3.2.](#)

Figure: 9.3.2

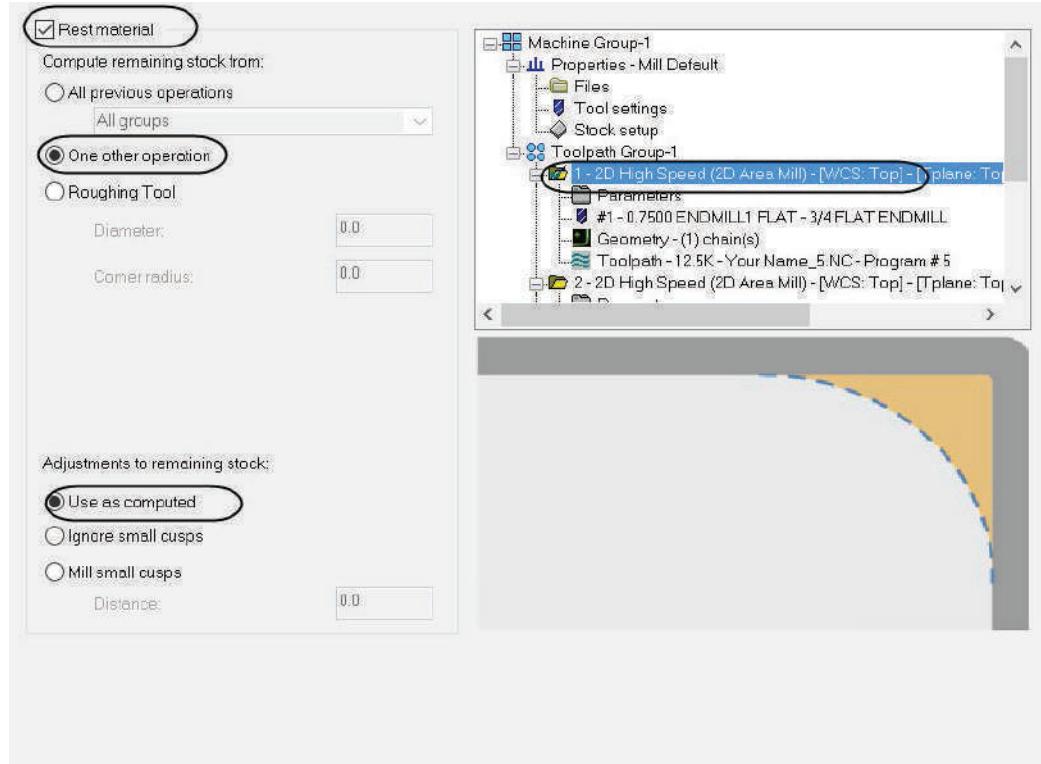




9.4 Set the Rest Material

- From the Tree View list, select **Stock**. Enable **Rest Material** and set the parameters the same as shown in [Figure: 9.4.1](#).

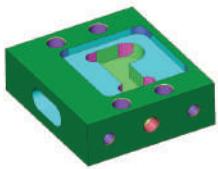
Figure: 9.4.1



The **Rest material** page allows you to set how Mastercam calculates the remaining stock.

One other operation calculates the remaining stock from one source operation.

NOTE: Since this toolpath has been copied all the parameters remain the same. Therefore we do not have to view all of the parameters.

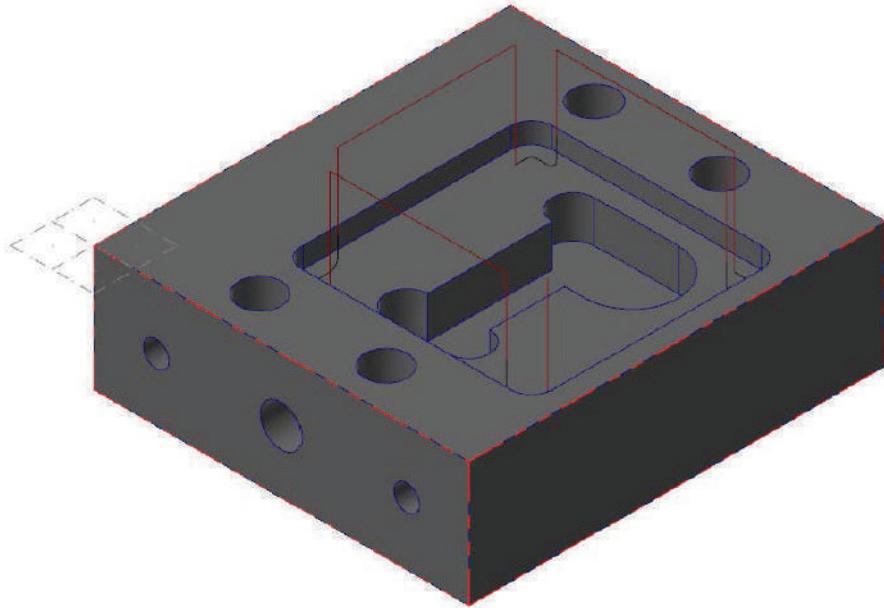


9.5 Preview the Toolpath

- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



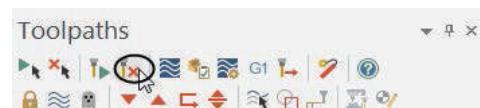
- ♦ See [page 446](#) to review the procedure.
- ♦ The toolpath should look as shown.

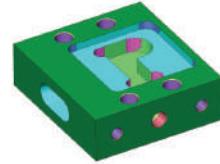


- ♦ Press the **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

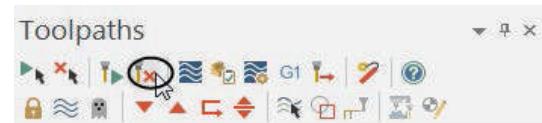
- ♦ Choose the **OK** button to generate the toolpath.
- ♦ Regenerate the dirty operations.





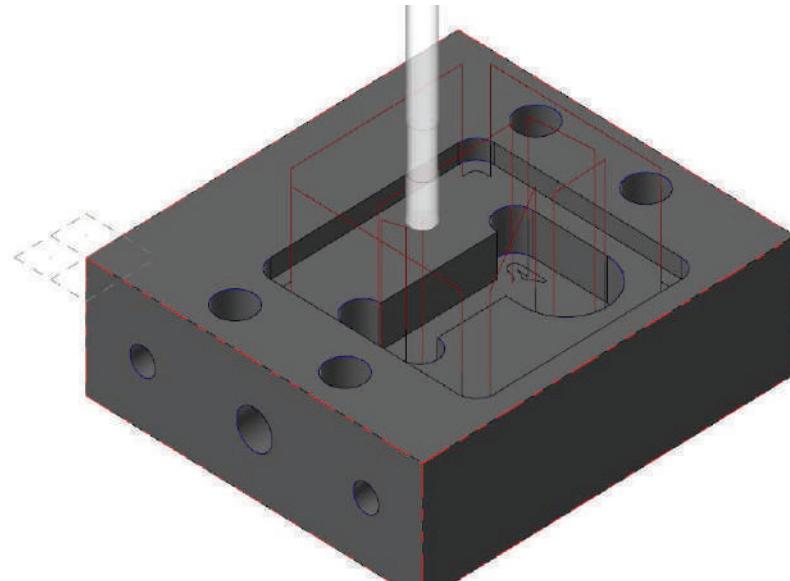
9.6 Repeat the same steps for operation #4

- ♦ Select **Parameters** under operation #4 in the **Tree View list**.
- ♦ In the **Tool** page, select the same tool used in operation no. 3 and input a comment accordingly.
- ♦ Select **Stock** from the **Tree View list**. Similar to the previous **substep 9.4 on page 468**, enable **Rest Material**; check **One other operation** for **Compute remaining stock from** and then select operation no. 2. Check **Use as computed**.
- ♦ Leave the rest parameters as they are and click on the **OK** button to generate the toolpath.
- ♦ Regenerate the dirty operations.



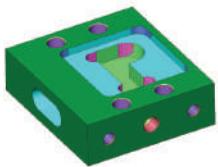
9.7 Backplot and Verify the toolpaths

- ♦ To **Backplot** both toolpaths, see **page 447**.
- ♦ To select both operations, hold down the **Ctrl** key.
- ♦ The toolpaths should look as shown.

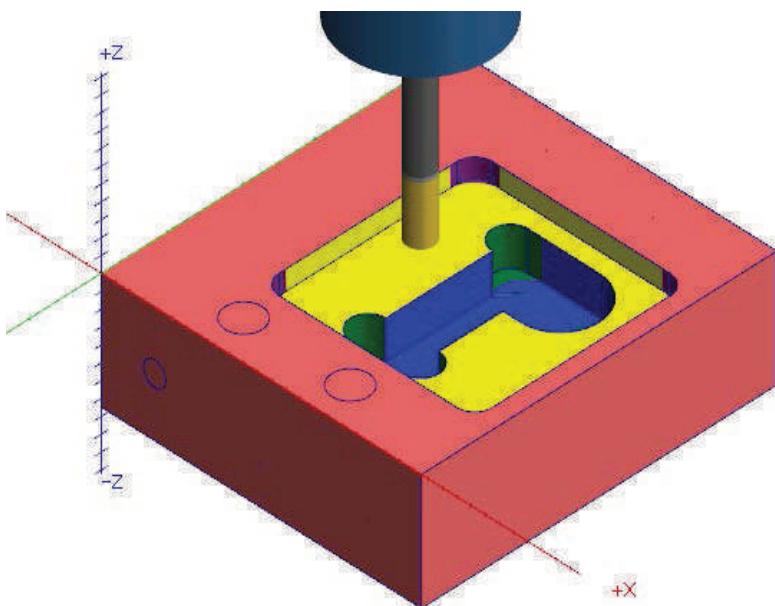


- ♦ Select the **OK** button to exit **Backplot**.

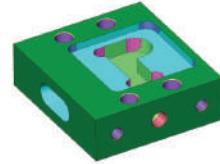
- ♦ To **Verify** make sure that all toolpaths are selected by choosing the **Select all operations** icon.



- ♦ See page 448 for more information.



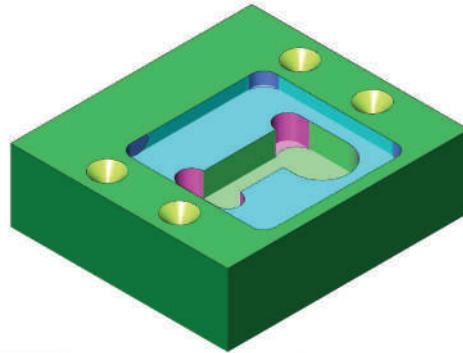
- ♦ To go back to the **Mastercam** window, minimize the **Mastercam Simulator** window as shown.



STEP 10: SPOT DRILL THE HOLE

Spot Drilling the holes allows you to start the hole. In this operation we will use the spot drill to chamfer the hole before drilling it.

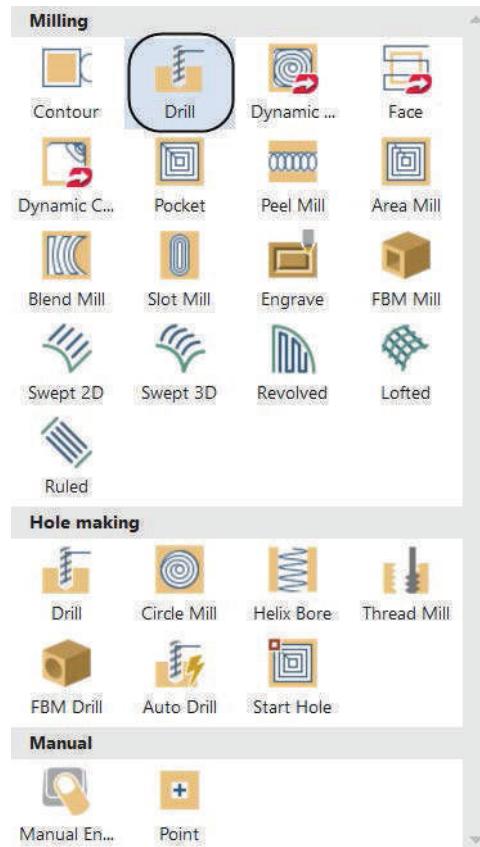
Toolpath Preview:

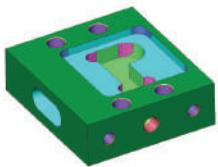


- ◆ Press **Alt + T** to remove the toolpaths display.

TOOLPATHS

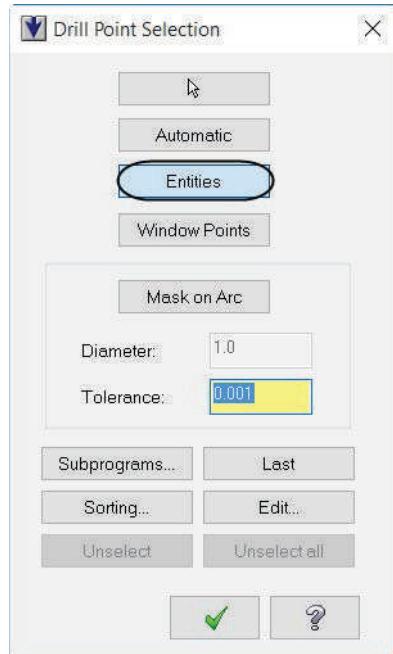
- ◆ In the **2D** group, select the **Expand gallery** arrow and click on the **Drill** icon.





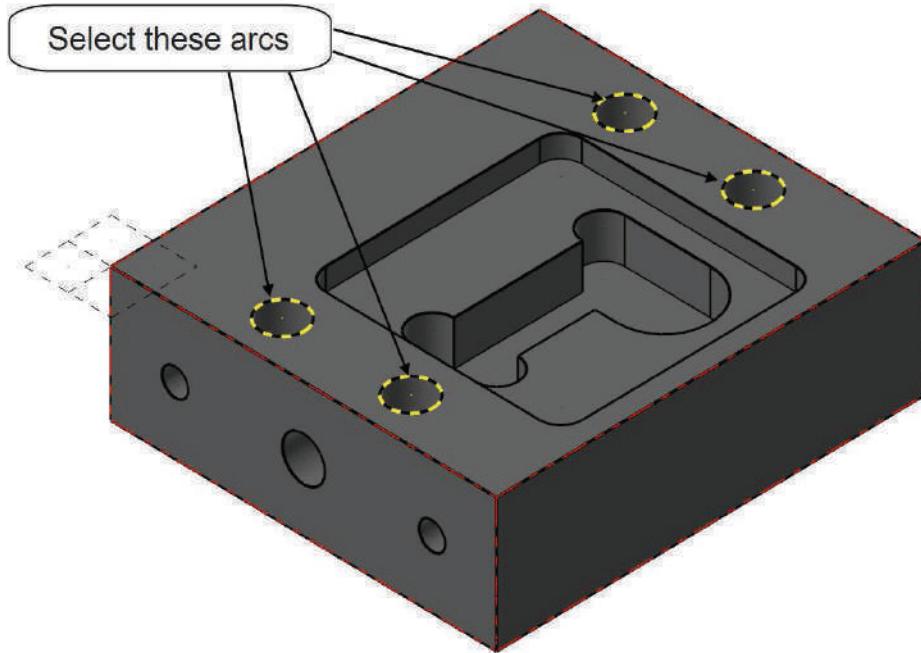
- In the Drill Point Selection dialog box choose the option Entities button as shown in [Figure: 10.0.1](#).

Figure: 10.0.1

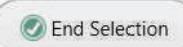


- Select the circles as shown in [Figure: 10.0.2](#).

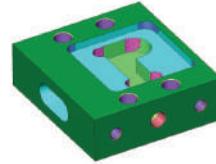
Figure: 10.0.2



- Click on the End Selection button or press Enter to finish the selection.



TUTORIAL #5 SPOT DRILL THE HOLE

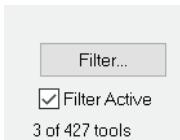


- ♦ Select the **OK** button in the **Drill Point Selection** dialog box once you have selected the four arcs.
- ♦ In the **Toolpath Type** page, the **Drill** toolpath will be selected.



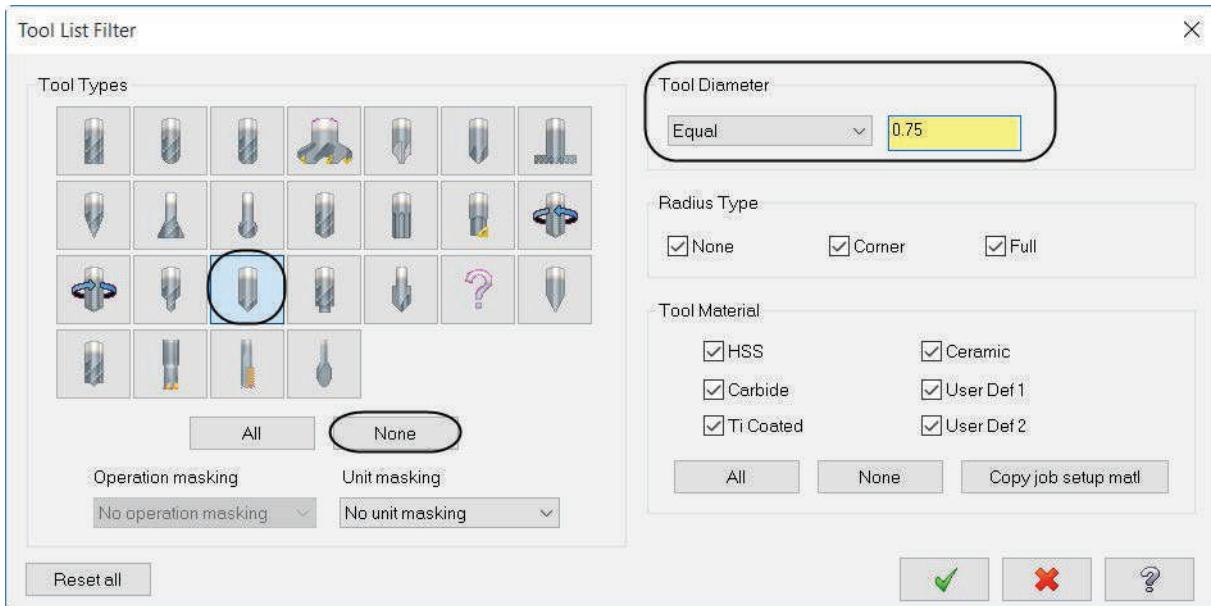
10.1 Select a 3/4" Spot Drill from the Library and set the Tool Parameters

- ♦ Select **Tool** from the **Tree View** list.
- ♦ Click on the **Select library tool...** button.
- ♦ To be able to see just the spot drill select the **Filter** button.



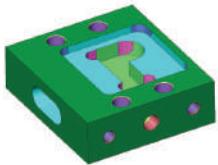
- ♦ Under **Tool Types** select the **None** button and then choose the **Spot drill** icon.
- ♦ Ensure the **Diameter** is set to **Equal 0.75** as shown in [Figure: 10.1.1](#).

Figure: 10.1.1



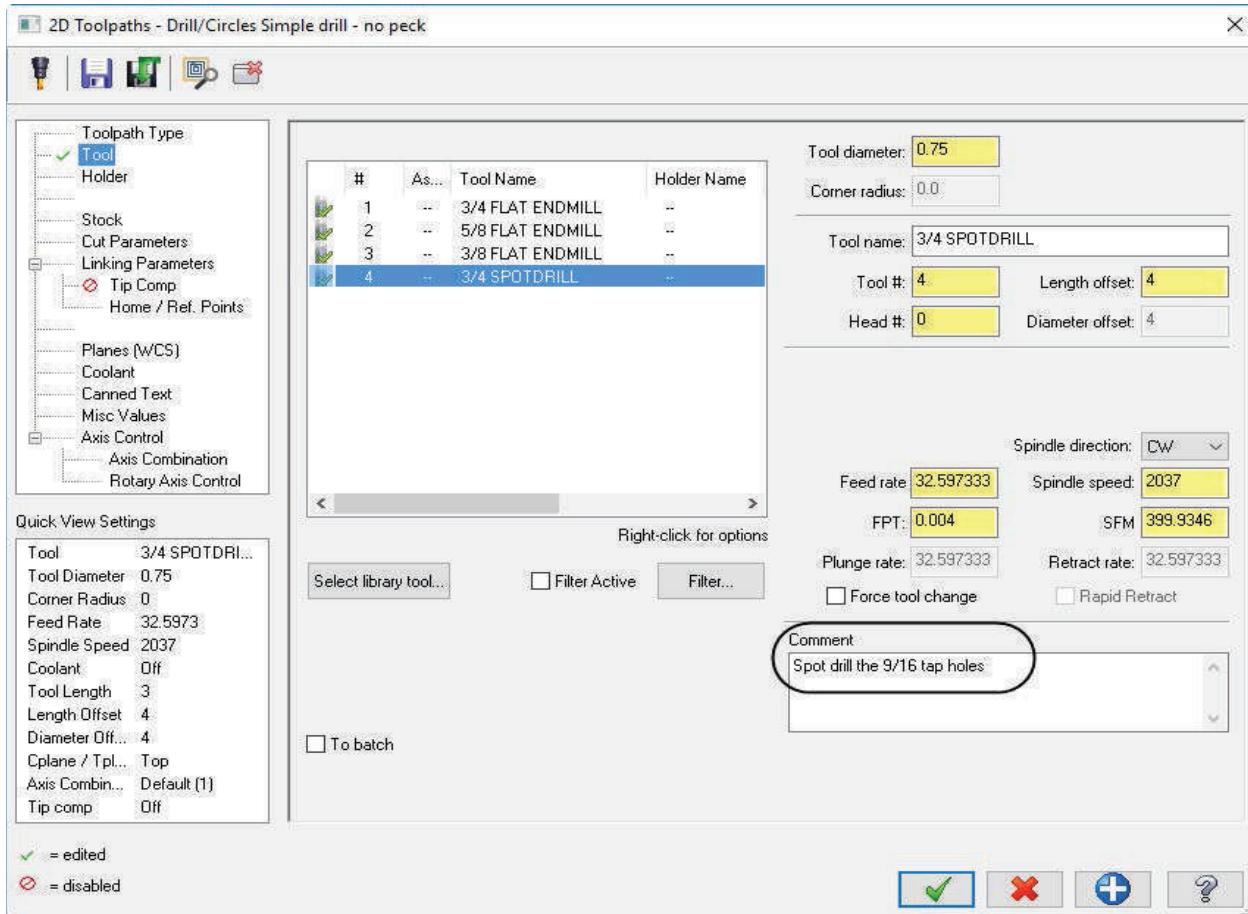
- ♦ Select the **OK** button to exit the **Tool List Filter** dialog box.
- ♦ Select the **3/4" Spot Drill**.

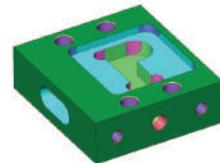
#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	Type	Ra...	# Flutes
25	-	3/4 SPOT...	-	0.75	0.0	2.0	Sp...	No...	4



- ♦ Select the **OK** button to exit.
- ♦ Make the necessary changes to the **Tool** page as shown in [Figure: 10.1.2.](#)

Figure: 10.1.2

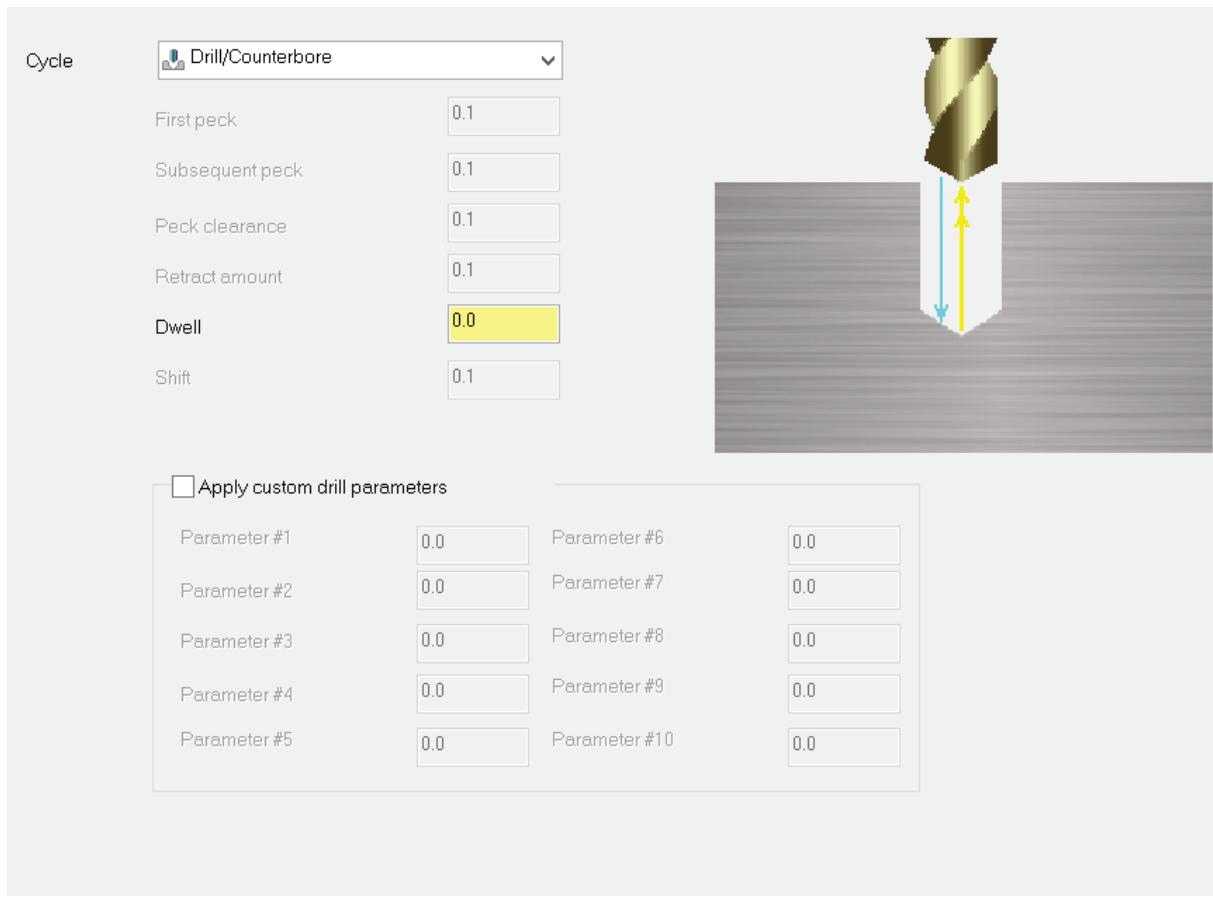




10.2 Set the Cut Parameters

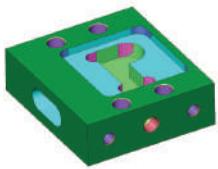
- ♦ Select **Cut Parameters** and make the necessary changes as shown in [Figure: 10.2.1](#).

Figure: 10.2.1



Drill/Counterbore is recommended for drilling holes with depths of less than three times the tool's diameter.

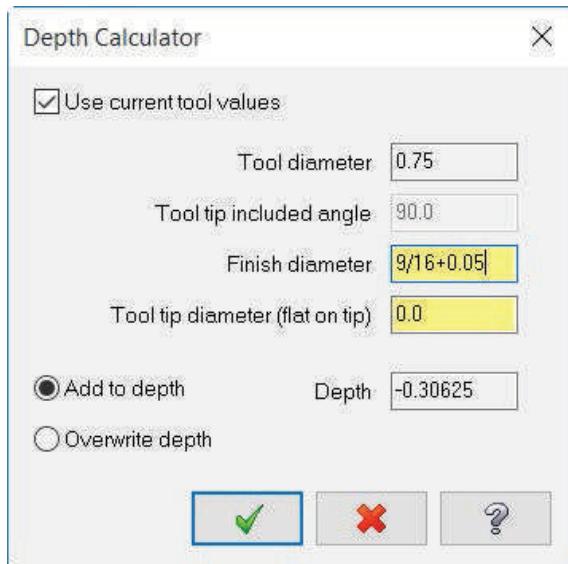
Dwell sets the amount of time in seconds that the tool remains at the bottom of a drilled hole.



10.3 Set the Linking Parameters

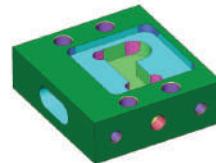
- ◆ Choose **Linking Parameters**, ensure **Clearance** is enabled and set the **Top of stock** and the **Depth** to **Absolute** and **0**.
- ◆ To input the depth select the **Calculator** icon.
- ◆ Input the following equation in the **Finish diameter** area: **9/16 + 0.05** (diameter of the finish hole + 2 X the chamfer size) as shown in [Figure: 10.3.1](#) and hit **Enter** to calculate the **Depth**.

Figure: 10.3.1



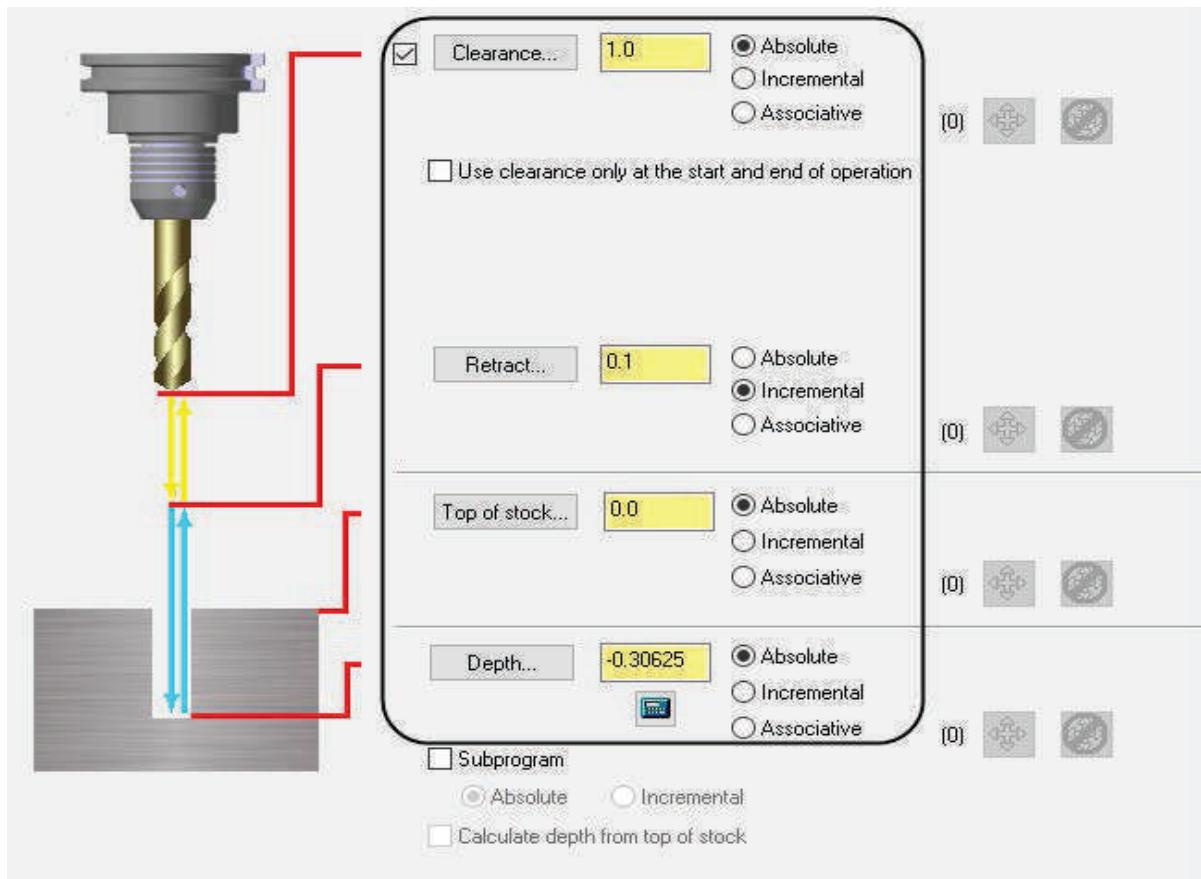
- ◆ Select the **OK** button to exit the **Depth Calculator**.

TUTORIAL #5 SPOT DRILL THE HOLE



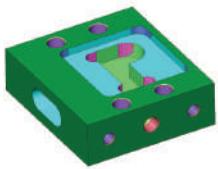
- ♦ You will now see the depth we calculated for the spot drilling operation set in the **Depth** field as shown in [Figure: 10.3.2](#).
- ♦ This will chamfer the hole for the tapping operation.
- ♦ Ensure your settings appear as shown.

Figure: 10.3.2



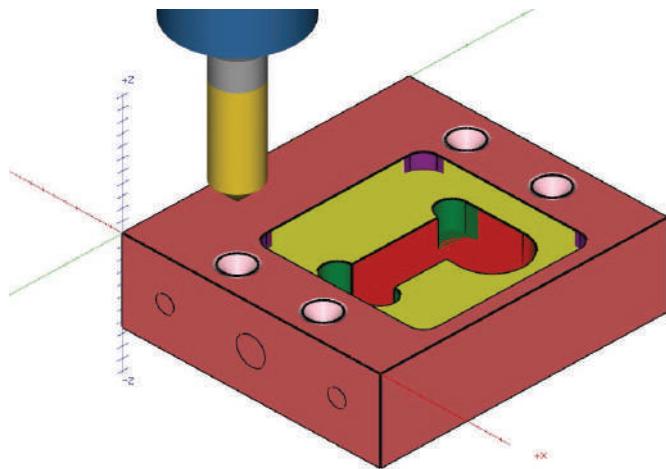
- ♦ Select the **OK** button to exit the **2D Toolpaths - Drill/Circles Simple drill - no peck** parameters.



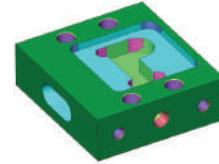


10.4 Backplot and Verify the toolpaths

- ♦ To **Backplot** and **Verify** your toolpaths, see [page 447](#) and [page 448](#).
- ♦ To verify all of the toolpaths, from the **Toolpaths Manager**, choose the **Select all operations** icon.
- ♦ The part should look as shown after verifying.



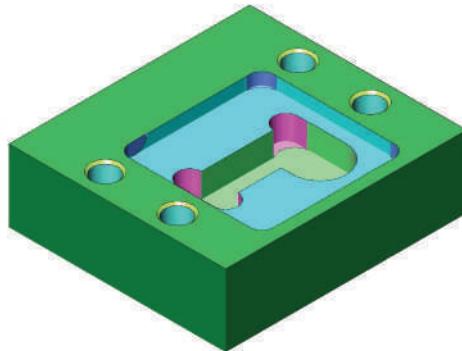
- ♦ To go back to the **Mastercam** window, minimize the **Mastercam Simulator** window as shown.



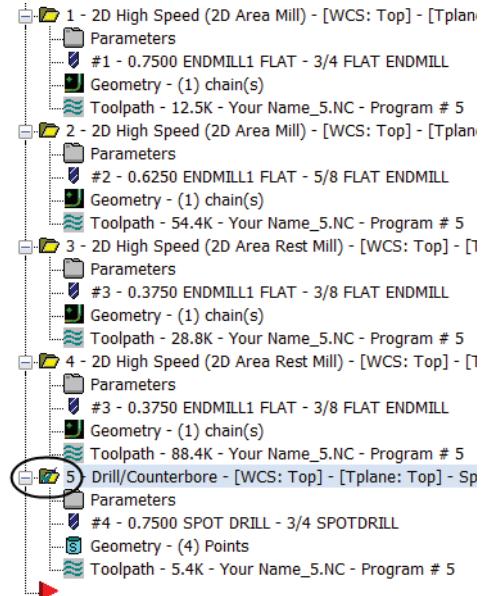
STEP 11: DRILL THE HOLES

In this example we will drill the holes through the part.

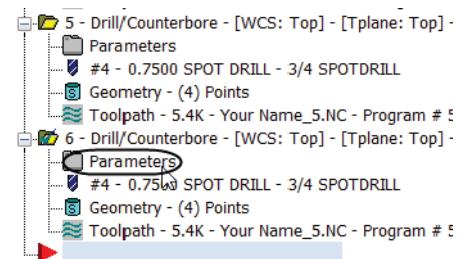
Toolpath Preview:

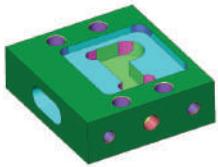


- ♦ In the **Toolpaths Manager**, select only the drilling operation as shown.



- ♦ As shown in previous steps, copy the drilling operation and move the insert arrow to the bottom.
- ♦ Select the **Parameters** in the second drilling operation as shown.





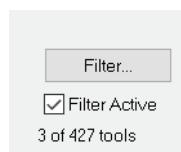
- In the Toolpath Type page, the Drill toolpath will be selected.



11.1 Select a 33/64" Drill from the Library and set the Tool Parameters

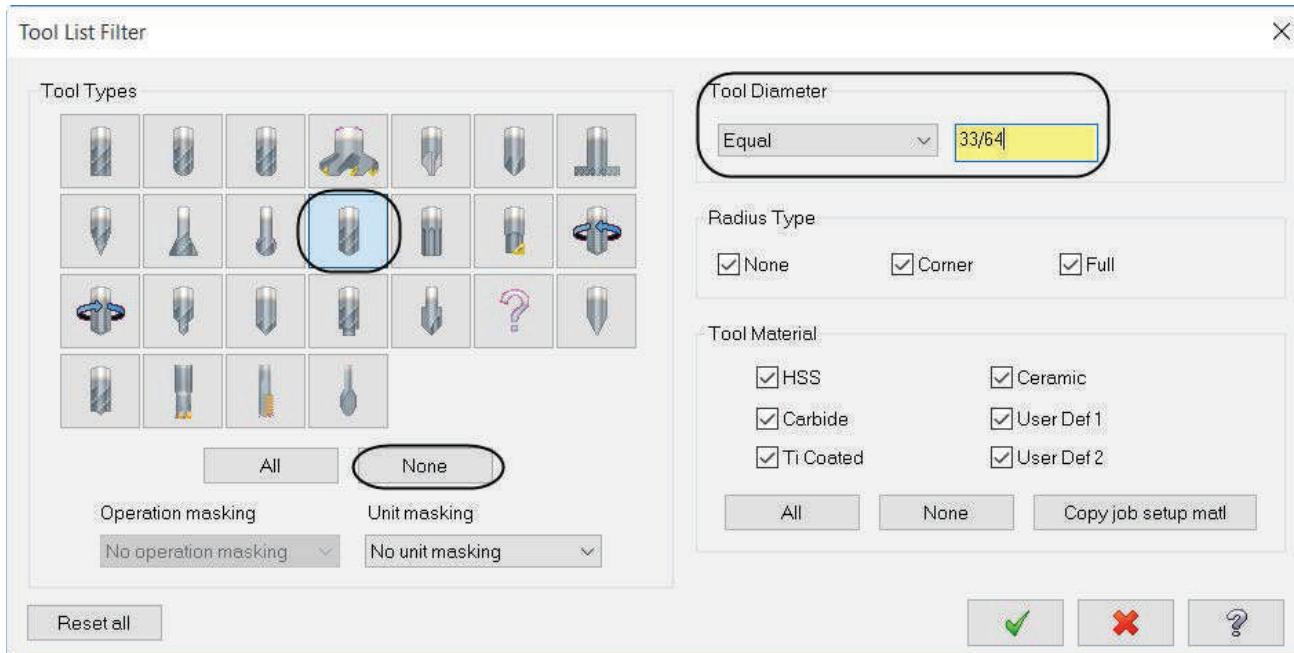
- Select Tool from the Tree View list.

- Click on the Select library tool button.
- To be able to see just the drill, select the Filter button.



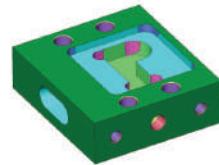
- Under Tool Types select the None button and then choose the Drill icon as shown in [Figure: 11.1.1](#).
- Under Tool Diameter select Equal and enter the value 33/64 as shown in [Figure: 11.1.1](#).

Figure: 11.1.1



- Select the OK button to exit the Tool List Filter dialog box.
- At this point you should see only 33/64" drills.

TUTORIAL #5 DRILL THE HOLES



- From that list select the **33/64" Drill** as shown in [Figure: 11.1.2](#).

Figure: 11.1.2

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Type	Ra...
163	-	33/64 DRI...	-	0....	0.0	2.0	2	Drill	No...

- Select the tool in the **Tool Selection** page and then choose the **OK** button to exit.
- Make the necessary changes to the **Tool** page as shown in [Figure: 11.1.3](#).

Figure: 11.1.3

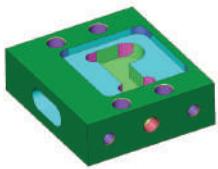
The screenshot shows the Mastercam software interface. On the left, a list of tools is displayed in a table:

#	Assembly Name	Tool Name	Holder Name	Dia.
1	-	3/4 FLAT ...	-	0.75
2	-	5/8 FLAT ...	-	0....
3	-	3/8 FLAT ...	-	0....
4	-	3/4 SPOT...	-	0.75
5	-	33/64 DRI...	-	0....

The tool selected is row 5, "33/64 DRI...". On the right, the "Tool" page settings are shown:

- Tool diameter: 0.515625
- Corner radius: 0.0
- Tool name: 33/64 DRILL
- Tool #: 5
- Length offset: 5
- Head #: 0
- Diameter offset: 5
- Spindle direction: CW
- Feed rate: 4.231727
- Spindle speed: 518
- FPT: 0.0041
- SFM: 69.9198
- Plunge rate: 4.231727
- Retract rate: 4.231727
- Force tool change
- Rapid Retract
- Comment: Dril the through holes.

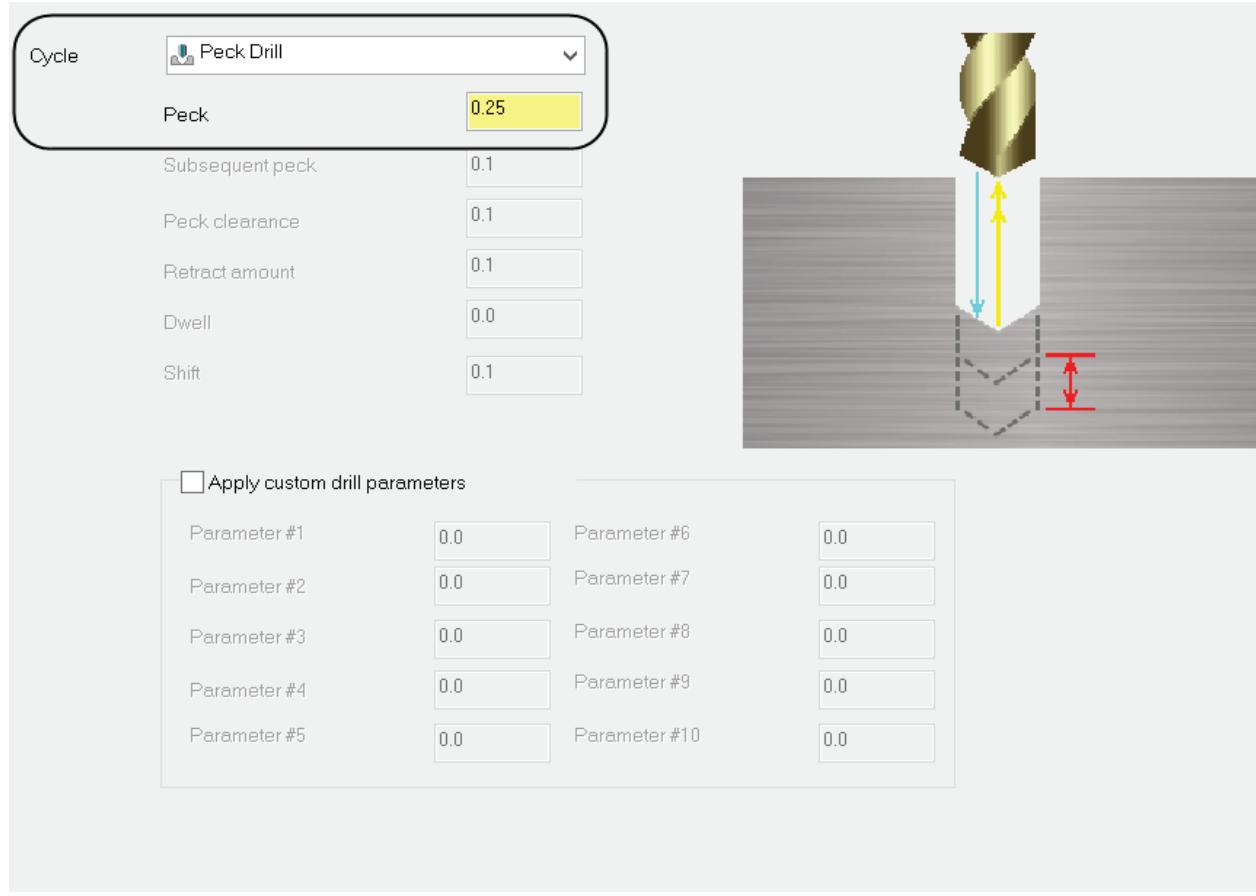
At the bottom left, there is a checkbox labeled "To batch".



11.2 Set the Cut Parameters

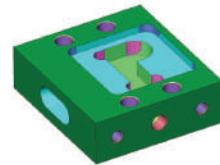
- ♦ Select **Cut Parameters**, change the drill **Cycle** to **Peck Drill**, and input a **Peck** value of **0.25** as shown in [Figure: 11.2.1](#).

Figure: 11.2.1



Peck Drill is recommended for drilling holes with depths of more than three times the tool's diameter. The drill retracts fully out of the drilled hole to remove material.

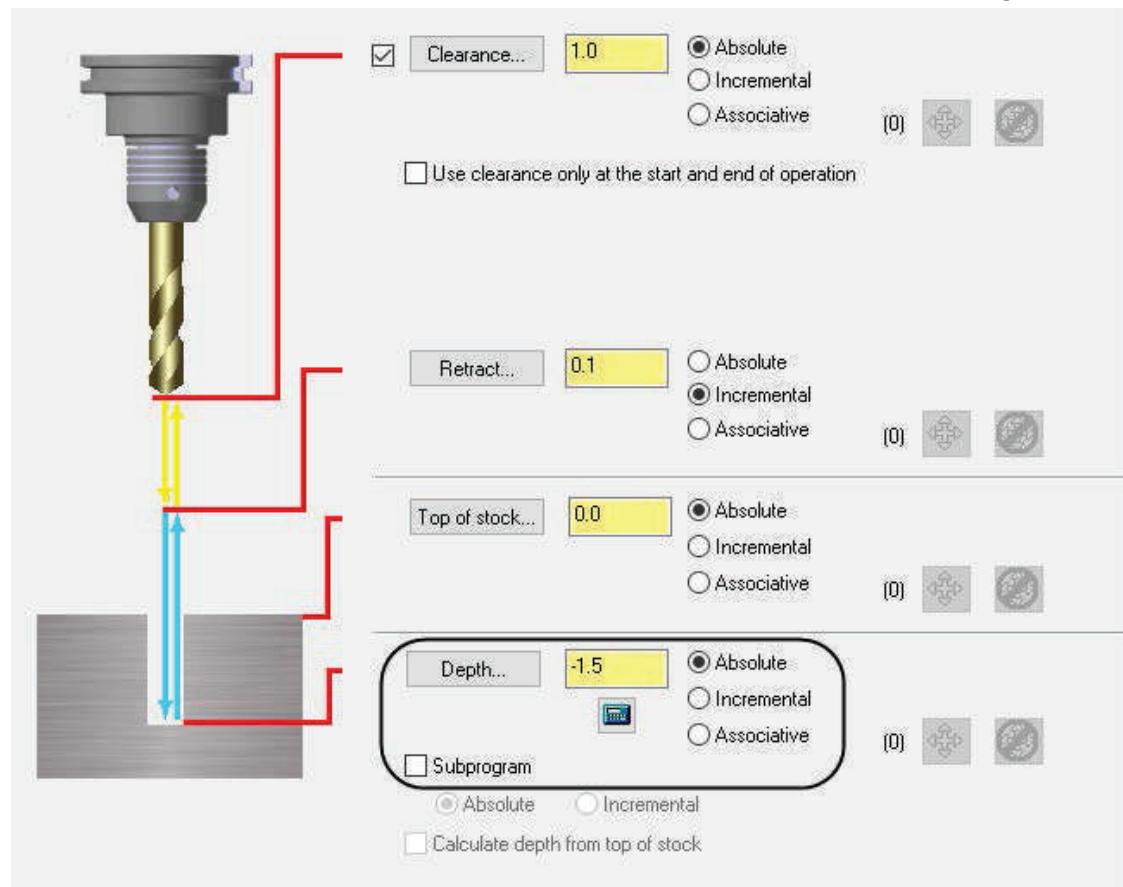
Peck sets the depth for the peck movement.

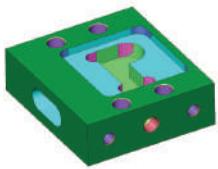


11.3 Set the Linking Parameters

- Choose **Linking Parameters** and input a **Depth** value of **-1.5** as shown in [Figure: 11.3.1](#).

Figure: 11.3.1

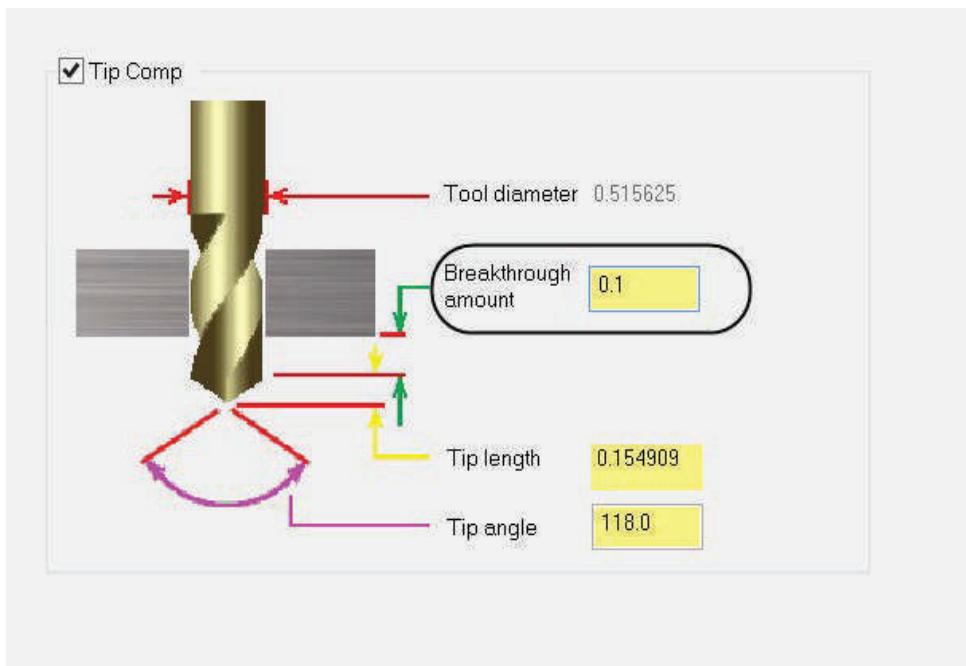




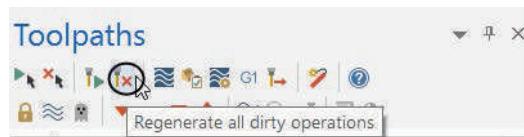
11.4 Set the Tip Comp Parameters

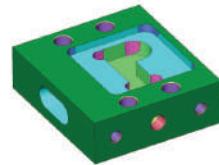
- ♦ Pick **Tip Comp** and enable this option. Input a **Breakthrough amount** of **0.1** as shown in [Figure: 11.4.1](#).

Figure: 11.4.1



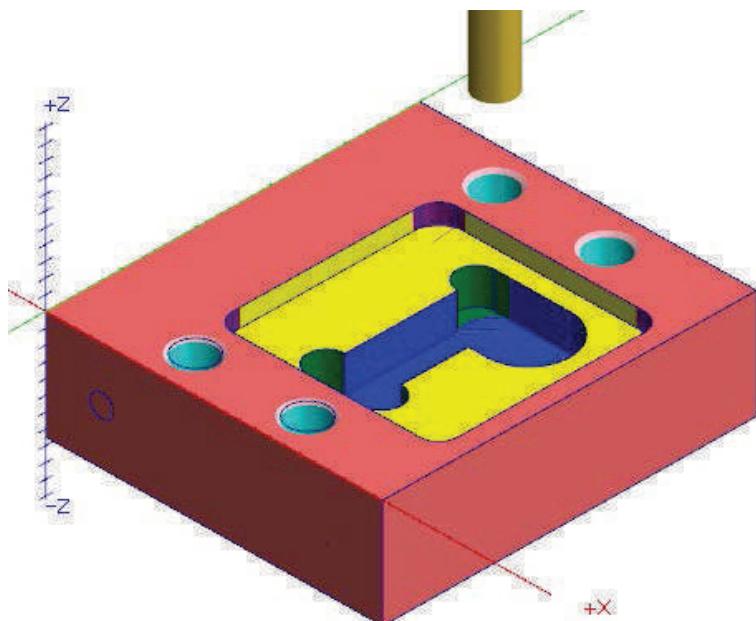
- ♦ Select the **OK** button to exit the **Drill/Circles Simple drill - full retract** parameters.
- ♦ Select the **Regenerate All Dirty Operations** icon in the **Toolpaths Manager**.



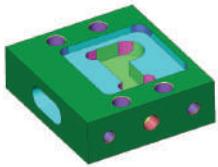


11.5 Backplot and Verify

- ♦ To **Backplot** and **Verify** the toolpaths, see **page 447** and **page 448**.



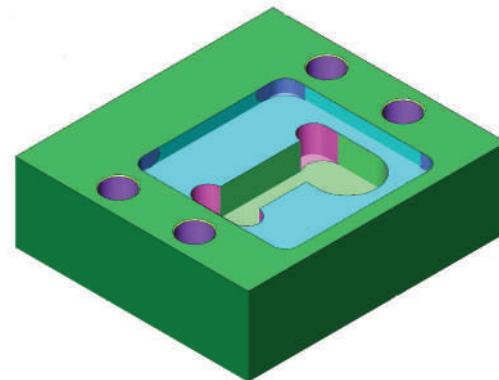
- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.



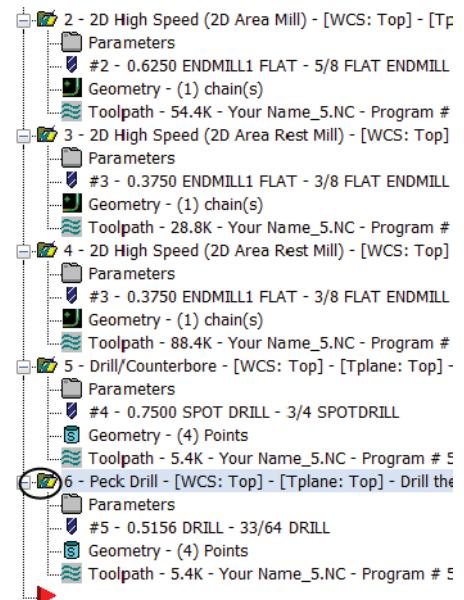
STEP 12: TAP THE HOLES

Tap cycle taps right or left internal threaded holes.

Toolpath Preview:



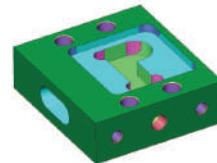
- ♦ In the **Toolpaths Manager**, select only the last drilling operation as shown.



- ♦ As shown in previous steps, copy the drilling operation and move the insert arrow below the last operation.
- ♦ Select the **Parameters** in the third drilling operation as shown.



TUTORIAL #5 TAP THE HOLES



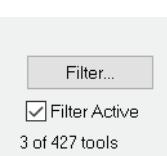
- In the Toolpath Type page, the Drill toolpath will be selected.



12.1 Select a 9/16 - 18 RH Tap from the Library and set the Tool Parameters

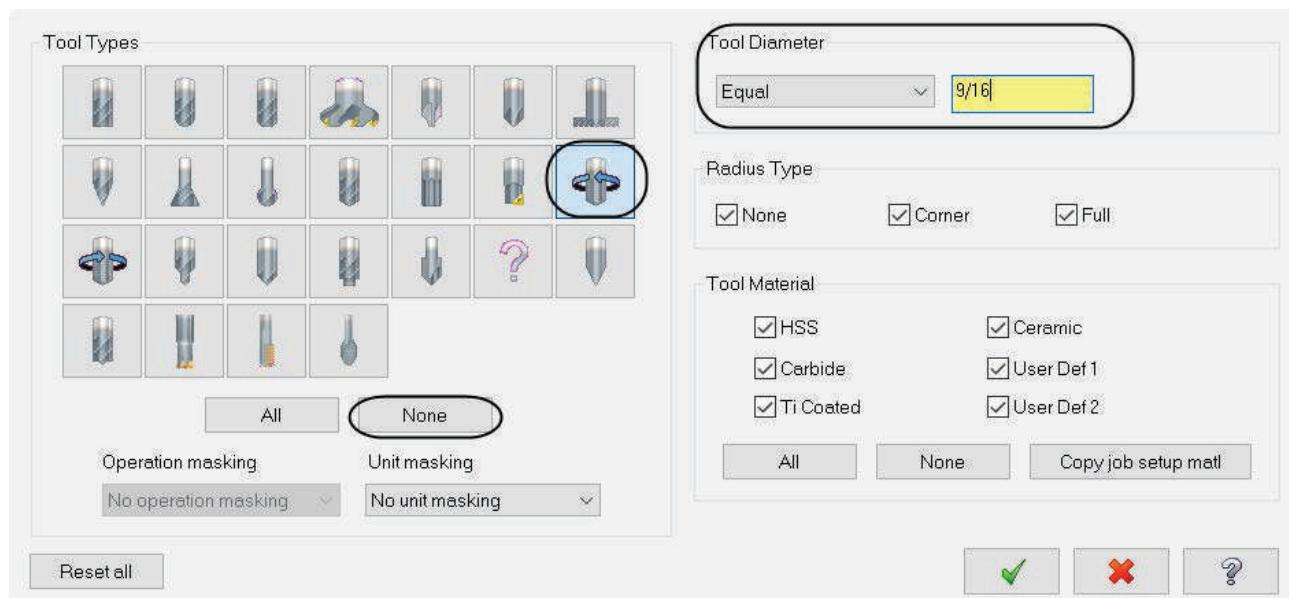
- Select Tool from the Tree View list.
- Click on the Select library tool... button.
- To be able to see just the RH Tap select the Filter button.

Select library tool...

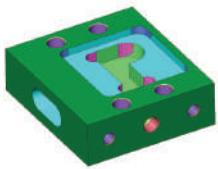


- Under Tool Types, select the None button and then choose the Tap RH icon. Under Tool Diameter, select Equal and enter the value 9/16 as shown in Figure: 12.1.1.

Figure: 12.1.1



- Select the OK button to exit the Tool List Filter dialog box.
- At this point you should see a list full of taps.



- From the list, select the **9/16 - 18 Tap RH** as shown.

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Ra...	Type
229	-	9/16-12 T...	-	0...	0.0	2.0	1	No...	Ta...
230	-	9/16-18 T...	-	0...	0.0	2.0	1	No...	Ta...

- Choose the **OK** button to exit.
- Make the necessary changes in the **Tool** page as shown in [Figure: 12.1.2](#).

Figure: 12.1.2

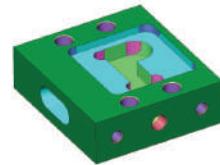
The screenshot shows the Mastercam software interface. On the left, there is a list of tools in a library:

#	Assembly Name	Tool Name	Holder Name	Dia.
1	-	3/4 FLAT ...	-	0.75
2	-	5/8 FLAT ...	-	0....
3	-	3/8 FLAT ...	-	0....
4	-	3/4 SPOT...	-	0.75
5	-	33/64 DRI...	-	0....
6	-	9/16-18 T...	-	0....

On the right, the tool settings are displayed:

- Tool diameter: 0.5625
- Corner radius: 0.0
- Tool name: 9/16-18 TAPRH
- Tool #: 6 Length offset: 6
- Head #: 0 Diameter offset: 6
- Spindle direction: CW
- Feed rate: 26.409877 Spindle speed: 475
- FPT: 0.0556 SFM: 69.9444
- Plunge rate: 26.409877 Retract rate: 26.409877
- Force tool change Rapid Retract
- Comment: Tap the 4 holes.

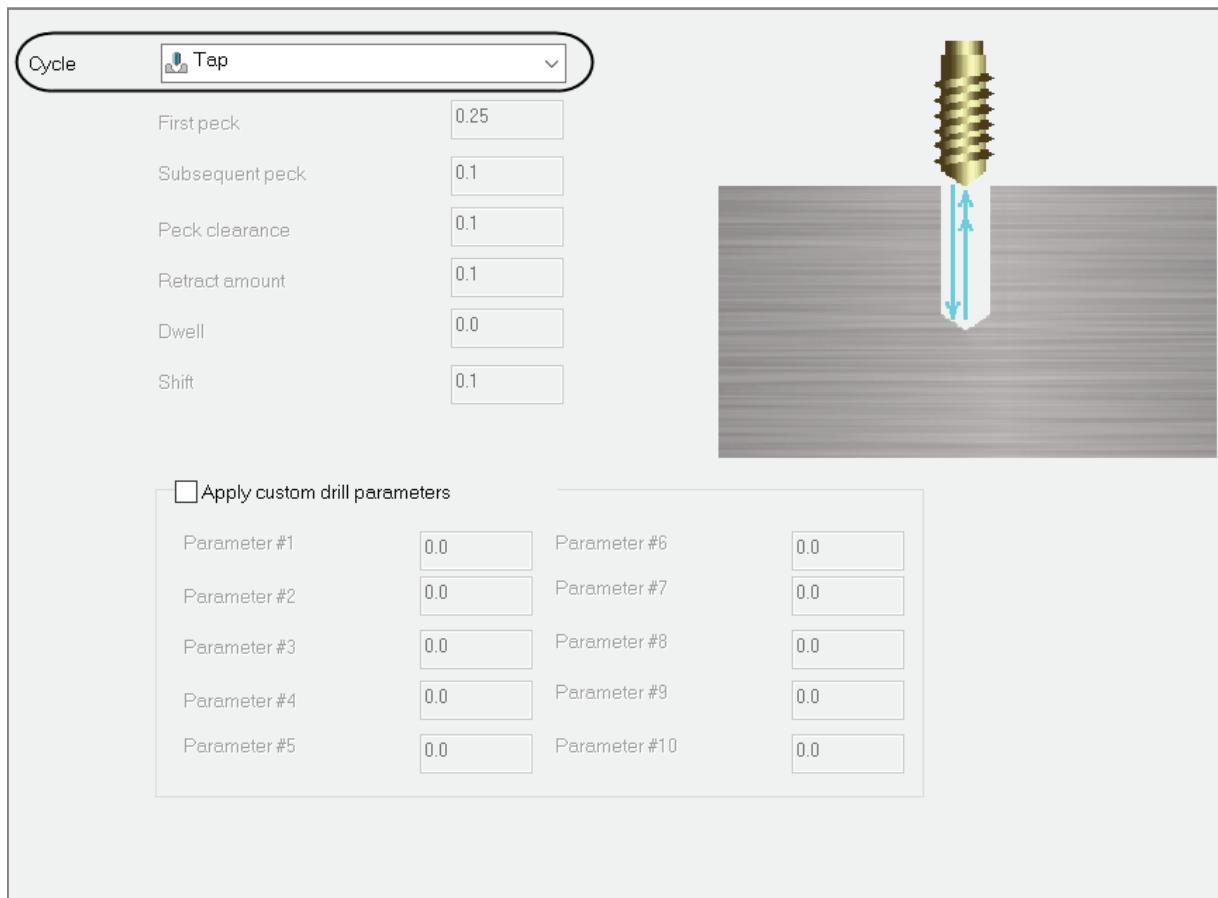
At the bottom left, there is a checkbox labeled "To batch".

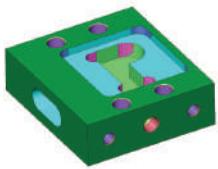


12.2 Set the Cut Parameters

- ♦ Select **Cut Parameters** and change the drill **Cycle to Tap** as shown in [Figure: 12.2.1.](#)

Figure: 12.2.1

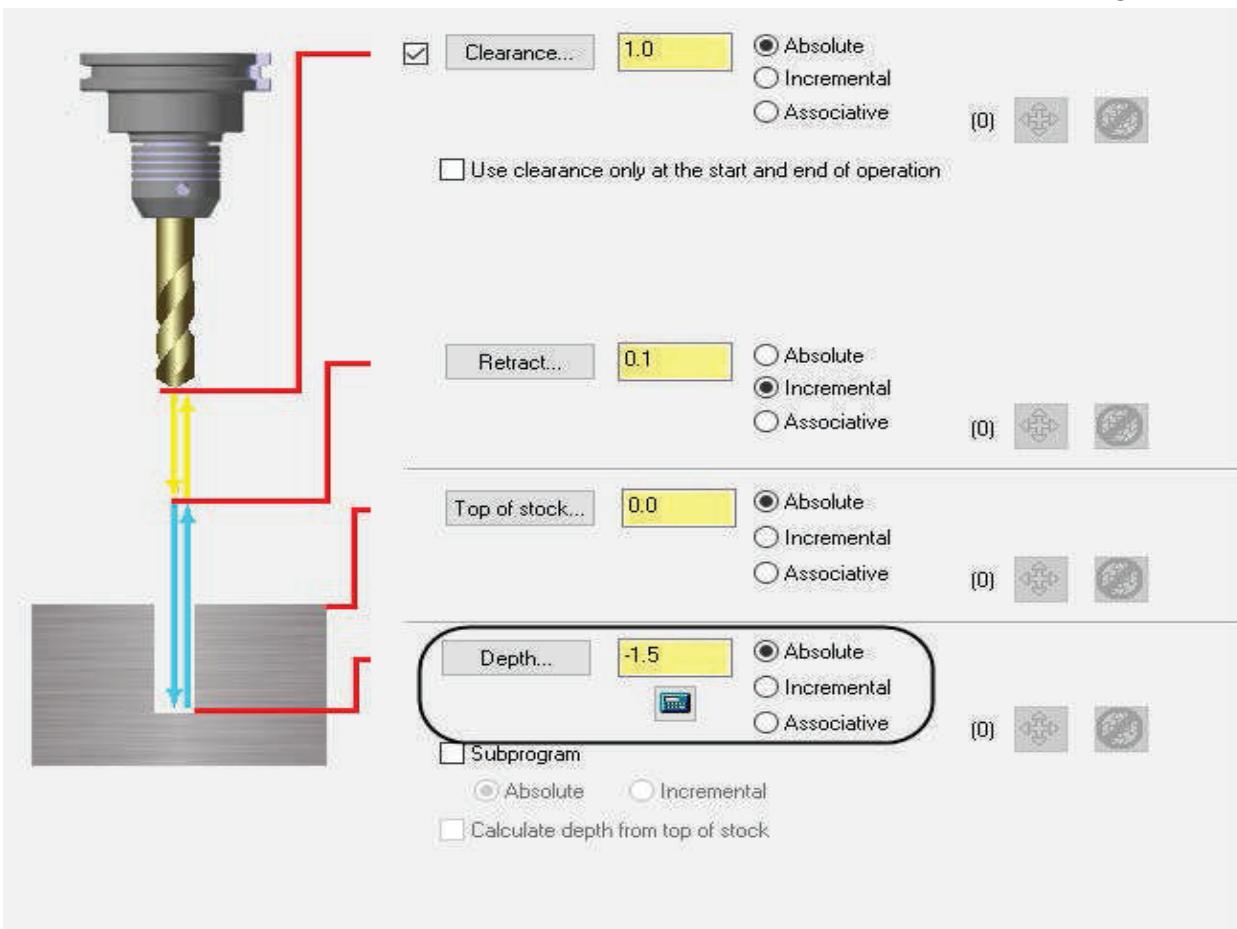




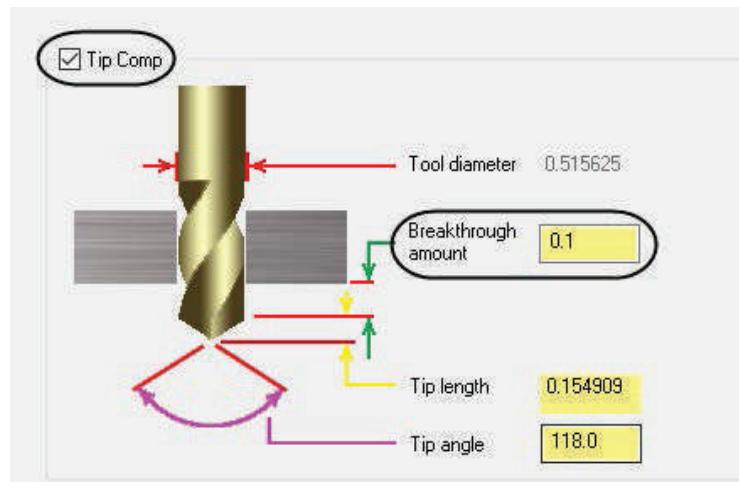
12.3 Set the Linking Parameters

- Choose **Linking Parameters** and input a **Depth** value of **-1.5** as shown in [Figure: 12.3.1](#).

Figure: 12.3.1

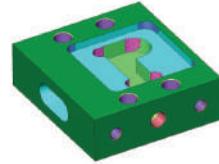


- Enable **Tip Compensation**.

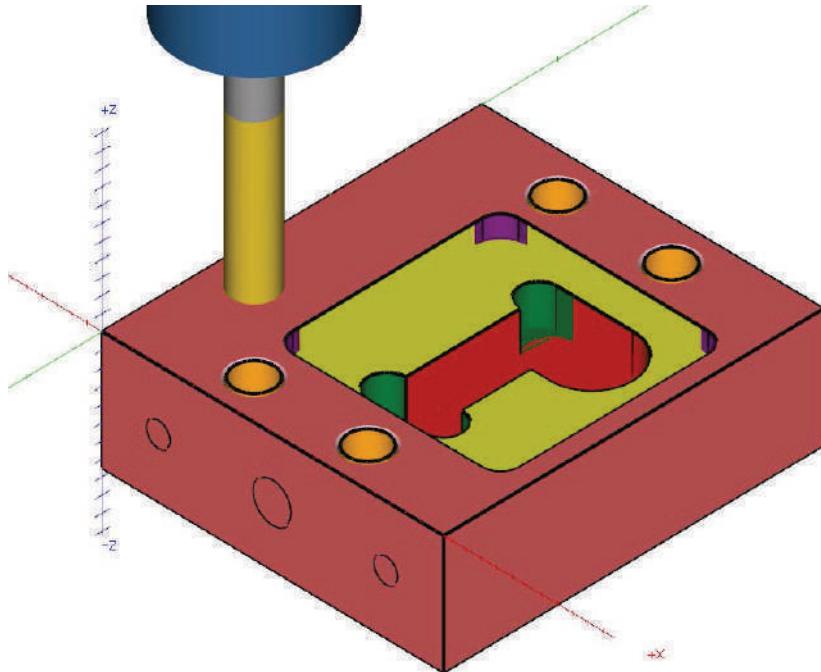


TUTORIAL #5

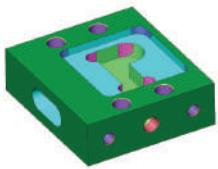
TAP THE HOLES



- ♦ Select the **OK** button to exit the **2D Toolpaths-Drill/Circles Tapping - feed in, reverse spindle - feed out** parameters.
- ♦ Select the **Regenerate All Dirty Operations** icon in the **Toolpaths Manager**.
- ♦ To **Backplot** and **Verify** the toolpaths, see [page 447](#) and [page 448](#).
- ♦ To make sure that all toolpaths are selected, choose the **Select all operations** icon.
- ♦ The part will appear as shown.

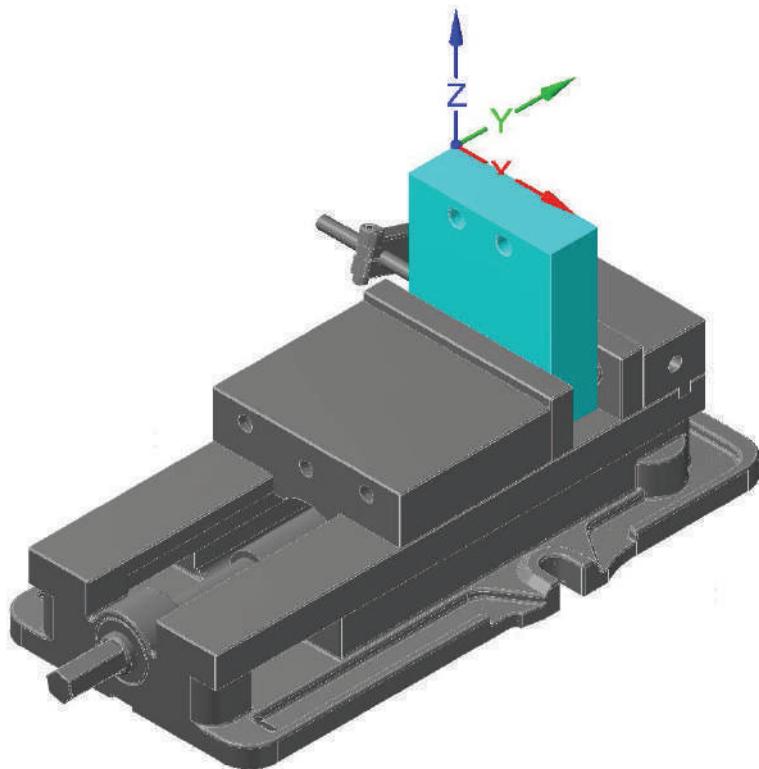


- ♦ To go back to the **Mastercam** window, minimize the **Mastercam Simulator** window as shown.

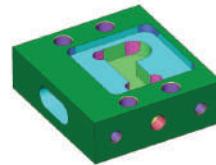


TOOLPATH CREATION - SETUP 2

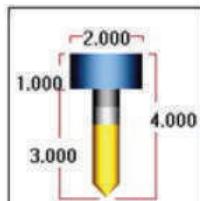
SUGGESTED FIXTURE:



NOTE: The part is now flipped over and we will machine the part from the **Front**.

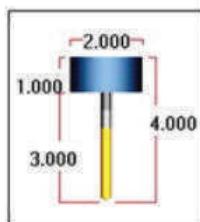


SETUP SHEET:

TOOL LIST

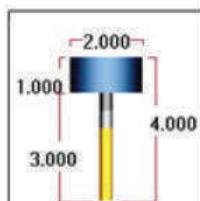
TYPE: Spot Drill
DIA OFFSET: 4
HOLDER: DEFAULT HOLDER
NUMBER: 4
LENGTH OFFSET: 4
#4 - 0.7500 SPOT DRILL - 3/4 SPOTDRILL

FLUTE LENGTH: 2.0
OVERALL LENGTH: 3.0
CORNER RAD: 0.0
OF FLUTES: 4



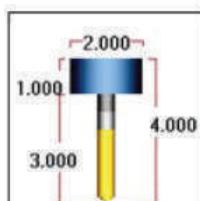
TYPE: Drill
DIA OFFSET: 7
HOLDER: DEFAULT HOLDER
NUMBER: 7
LENGTH OFFSET: 7
#7 - 0.3125 DRILL - 5/16 DRILL

FLUTE LENGTH: 2.0
OVERALL LENGTH: 3.0
CORNER RAD: 0.0
OF FLUTES: 2



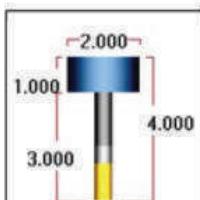
TYPE: Tap RH
DIA OFFSET: 8
HOLDER: DEFAULT HOLDER
NUMBER: 8
LENGTH OFFSET: 8
#8 - 0.3750 X 16.00 TAP RH - 3/8-16 TAPRH

FLUTE LENGTH: 2.0
OVERALL LENGTH: 3.0
CORNER RAD: 0.0
OF FLUTES: 1



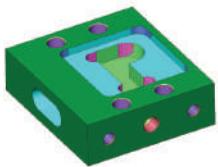
TYPE: Drill
DIA OFFSET: 9
HOLDER: DEFAULT HOLDER
NUMBER: 9
LENGTH OFFSET: 9
#9 - 0.5000 DRILL - 1/2 DRILL

FLUTE LENGTH: 2.0
OVERALL LENGTH: 3.0
CORNER RAD: 0.0
OF FLUTES: 2



TYPE: Endmill1 Flat
DIA OFFSET: 10
HOLDER: DEFAULT HOLDER
NUMBER: 10
LENGTH OFFSET: 10
#10 - 0.5000 ENDMILL1 FLAT - 1/2 FLAT ENDMILL

FLUTE LENGTH: 1.0
OVERALL LENGTH: 3.0
CORNER RAD: 0.0
OF FLUTES: 4



STEP 13: CREATING AND RENAMING TOOLPATH GROUPS

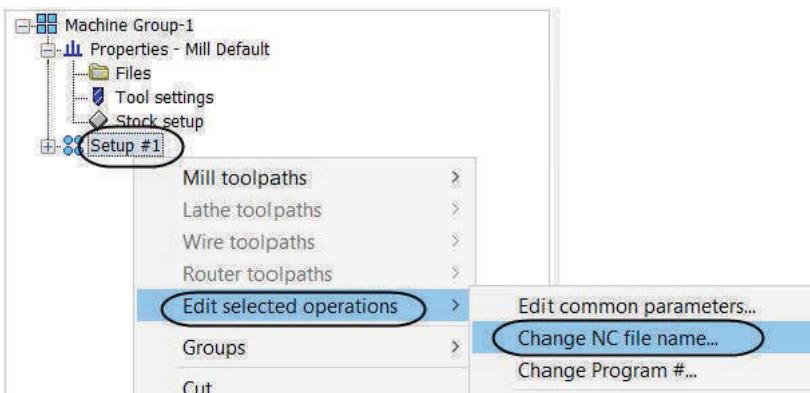
To machine the part in different setups, we will need to have separate programs. To be able to post separate programs for each setup, we will create them under different toolpath groups with different NC names.

13.1 Rename the Current Toolpath Group - 1 and the NC File

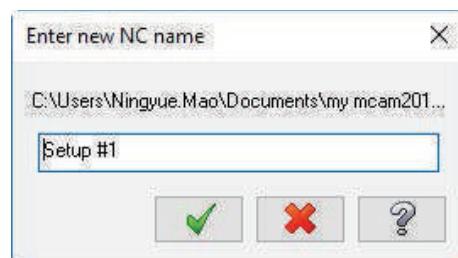
- ♦ Click on the Toolpath Group - 1 to highlight it and then click again to rename it "Setup #1."



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

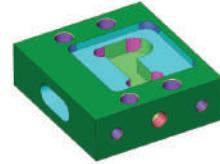


- ♦ Enter the new NC name: "Setup #1."



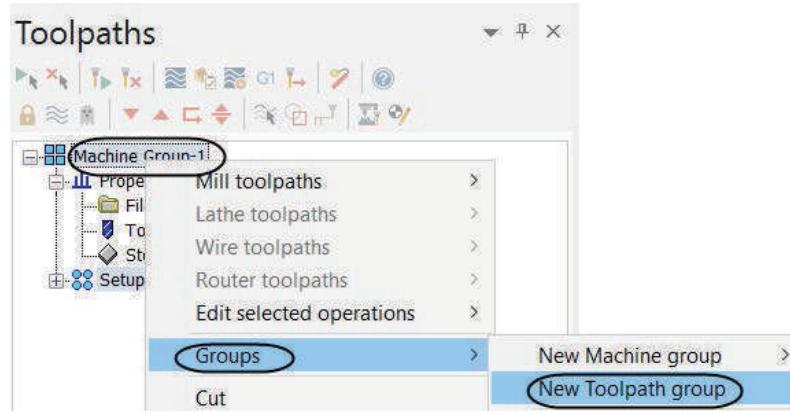
- ♦ Select the **OK** button to accept the new NC name.





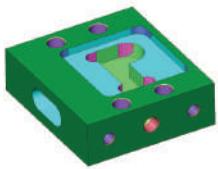
13.2 Create a new Toolpath Group.

- ♦ Right mouse click on the **Machine Group 1**.
- ♦ From the list, select **Groups** and then **New Toolpath group** as shown.



- ♦ Double click on the new **Toolpath Group 1** and rename it "**Setup #2 - Front**."



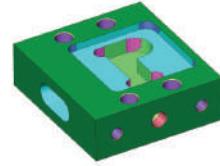


STEP 14: SET WCS TO FRONT

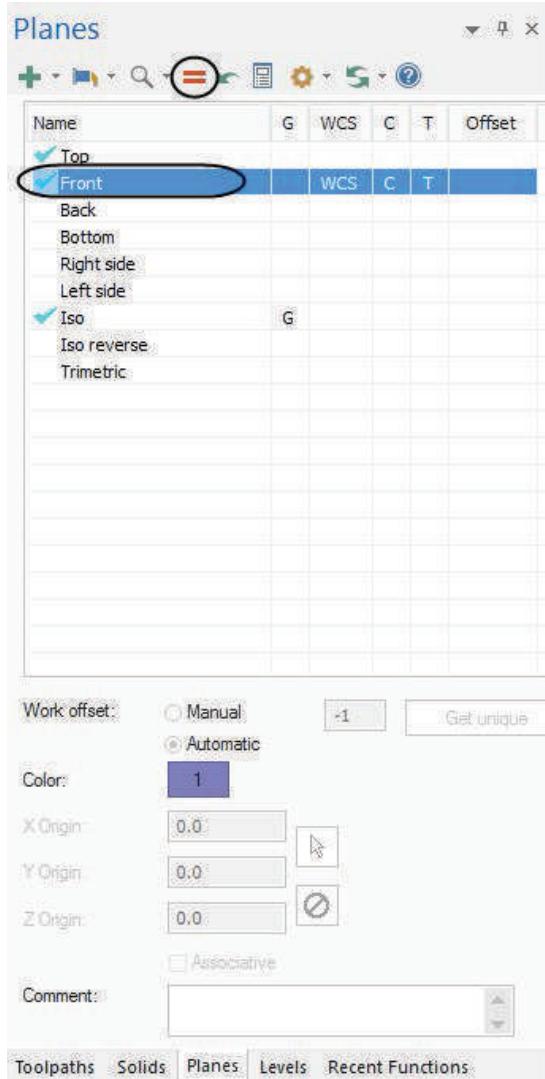
Work coordinate system (WCS) is the active coordinate system in use by Mastercam at any given time. The **WCS** contains the orientation of the X-Y-Z axes plus the location of the zero point (the Origin). This tells Mastercam how your part is positioned or oriented in the machine.

- Select the **Planes** tab located at the bottom left corner of the screen.

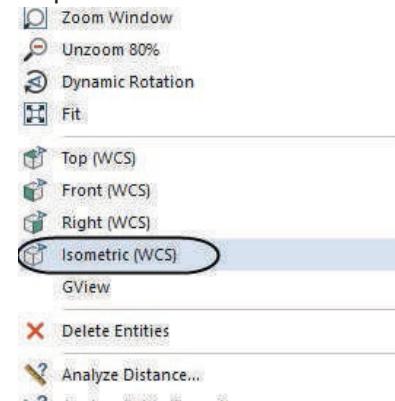


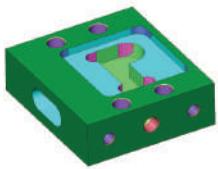


- ♦ Select the **Front** plane and click on the equal sign to set the **WCS**, **Construction Plane** and **Tool Plane** to the **Front** as shown.



- ♦ Right mouse click in the graphics window and select the **Isometric** view to see the part in the new orientation.

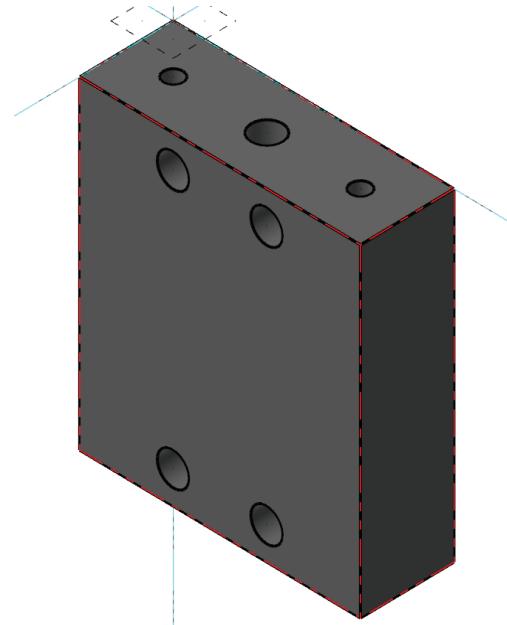




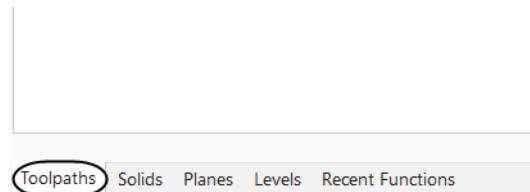
- ♦ Press **F9** on your keyboard to view the coordinate axes.

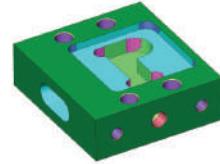
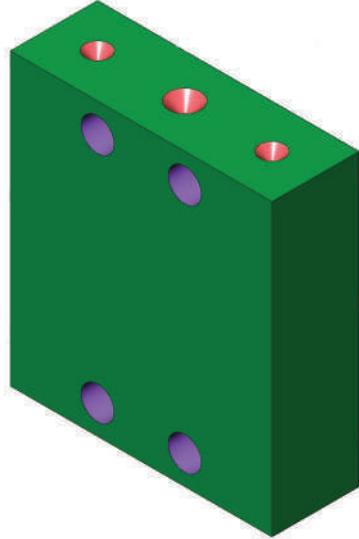
NOTE: The color of the coordinate axes remains the same because it is the same origin.

- ♦ Press **Alt + F1** to fit the part to screen.
- ♦ Your part should look as shown.



- ♦ Press **F9** again to remove the axes display.
- ♦ To open the **Toolpaths Manager**, select the **Toolpaths** tab from the lower left corner of the screen.

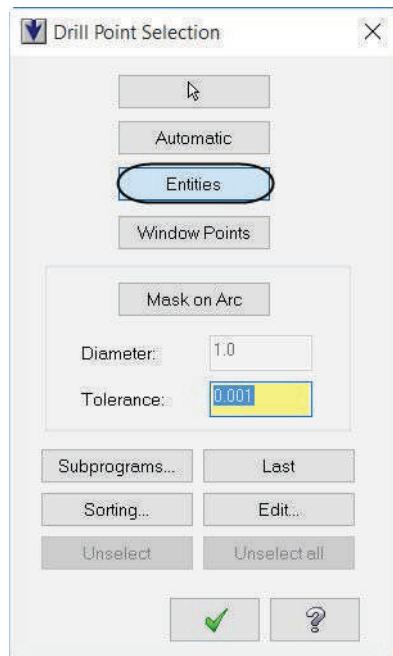


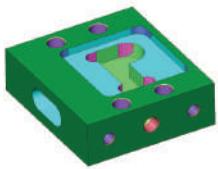
**STEP 15: SPOT DRILL ALL 3 HOLES***Toolpath Preview:***TOOLPATHS**

- From the **2D** group, select the **Drill** icon as shown.



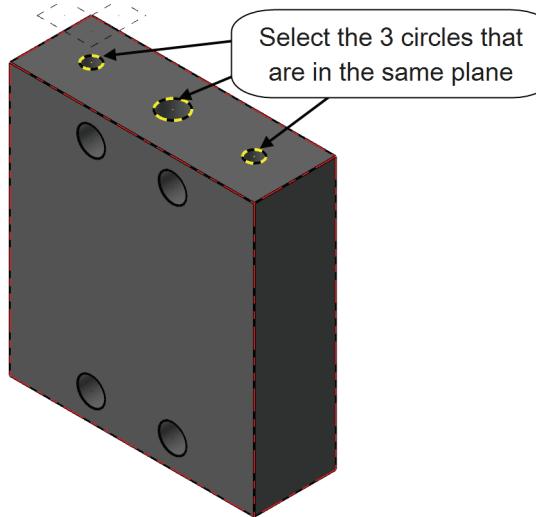
- In the **Drill Point Selection** dialog box choose the option **Entities** as shown.



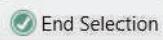


- ♦ Select the 3 circles as shown in [Figure: 15.0.1](#).

Figure: 15.0.1



- ♦ Click on the **End Selection** button or press **Enter** to finish the selection.



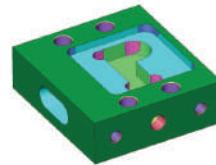
- ♦ Select the **OK** button in the **Drill Point Selection** dialog box once you have picked the arcs.



- ♦ In the **Toolpath Type** page, the **Drill** toolpath will be selected.



TUTORIAL #5 SPOT DRILL ALL 3 HOLES



- From the Tree View list, select Tool.
- Select the 3/4" Spot Drill from the list.
- Make the necessary changes to the Tool page as shown in [Figure: 15.0.2](#).

Figure: 15.0.2

2D Toolpaths - Drill/Circles Simple drill - no peck

Toolpath Type:

- Tool (selected)
- Holder
- Stock
- Cut Parameters
- Linking Parameters
- Tip Comp (disabled)
- Home / Ref. Points
- Planes (WCS)
- Coolant
- Canned Text
- Misc Values
- Axis Control
 - Axis Combination
 - Rotary Axis Control

Quick View Settings:

Tool	3/4 SPOTDRILL
Tool Diameter	0.75
Corner Radius	0
Feed Rate	32.59733
Spindle Speed	2037
Coolant	Off
Tool Length	3
Length Offset	4
Diameter Off...	4
Cplane / Tpl...	Front
Axis Combin...	Default (1)
Tip comp	Off

Tool Selection:

#	Assem...	Tool Name	Holder Name
1	...	3/4 FLAT ENDMILL	...
2	...	5/8 FLAT ENDMILL	...
3	...	3/8 FLAT ENDMILL	...
4	...	3/4 SPOTDRILL	...
5	...	33/64 DRILL	...
6	...	9/16-18 TAPRH	...

Tool Configuration:

- Tool diameter: 0.75
- Corner radius: 0.0
- Tool name: 3/4 SPOTDRILL
- Tool #: 4
- Length offset: 4
- Head #: 0
- Diameter offset: 4
- Spindle direction: CW
- Feed rate: 32.597333
- Spindle speed: 2037
- FPT: 0.004
- SFM: 399.9346
- Plunge rate: 32.597333
- Retract rate: 32.597333
- Force tool change
- Rapid Retract

Comment:

Spot drill the 3 holes in the front plane.

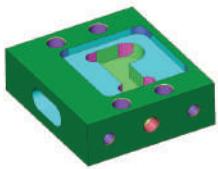
Toolpath Options:

- Select library tool...
- Filter Active
- Filter...
- To batch

Toolpath Status:

- = edited
- = disabled

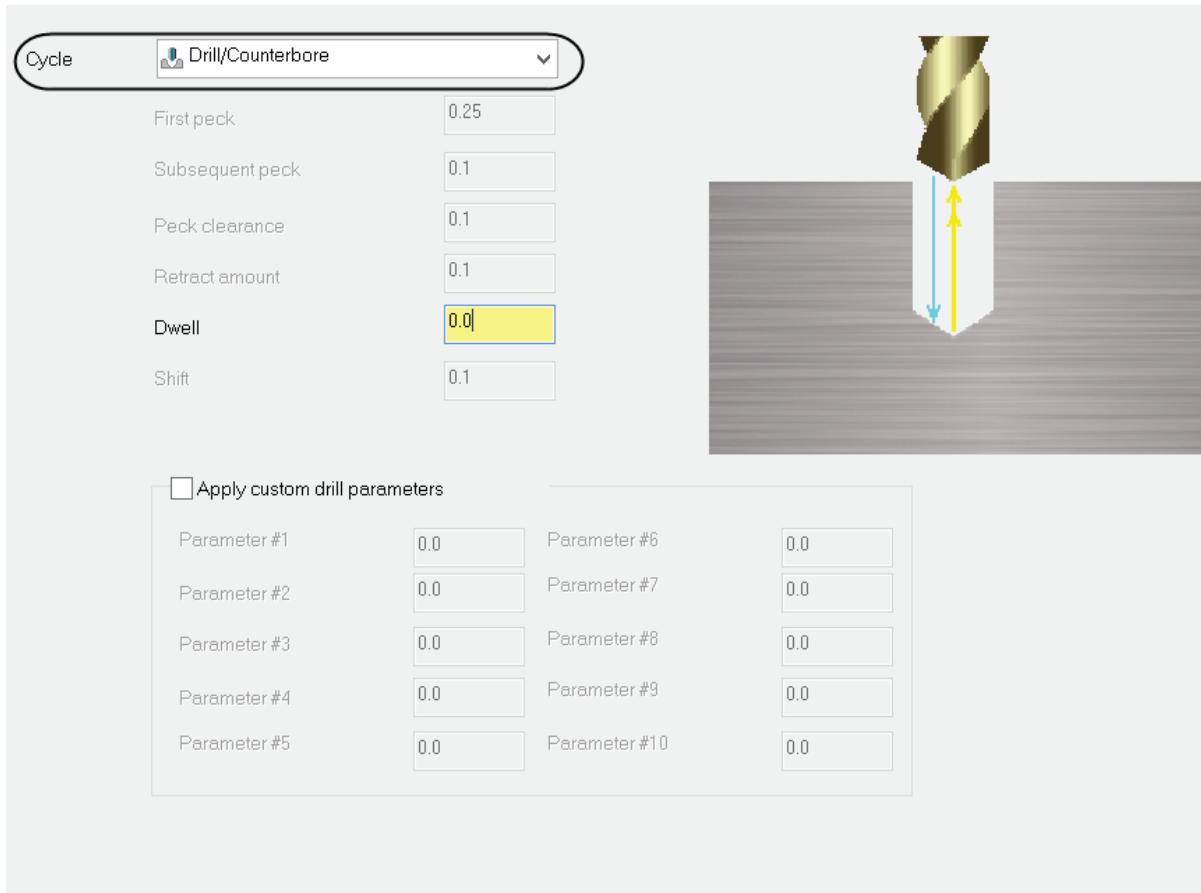
Checkmarks for Save, Cancel, Add, and Help.

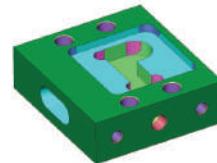


15.1 Set the Cut Parameters

- ♦ Select **Cut Parameters** and change the **Cycle** to **Drill/Counterbore** as shown in [Figure: 15.1.1](#).

Figure: 15.1.1

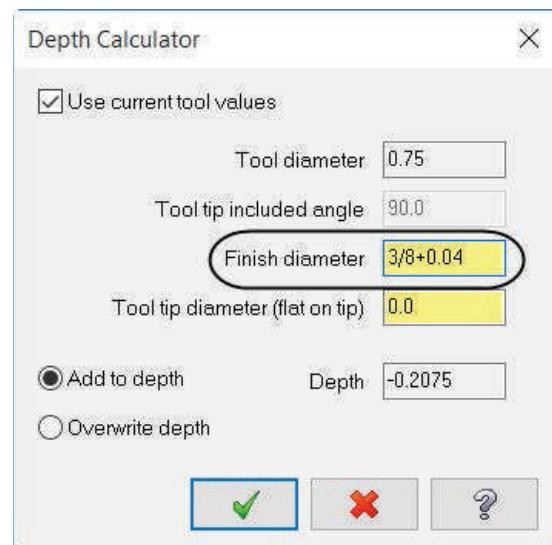




15.2 Set the Linking Parameters

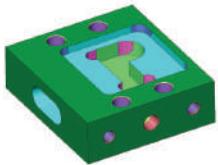
- ◆ Choose **Linking Parameters**, ensure **Clearance** is enabled, and the **Top of stock** and **Depth** are set to **Absolute** and **0.0**.
- ◆ To input the **Depth** select the **Calculator** icon.
- ◆ Input the following equation in the **Finish diameter** area: **$3/8+0.04$** and hit **Enter** to calculate the **Depth** as shown in [Figure: 15.2.1](#).

Figure: 15.2.1



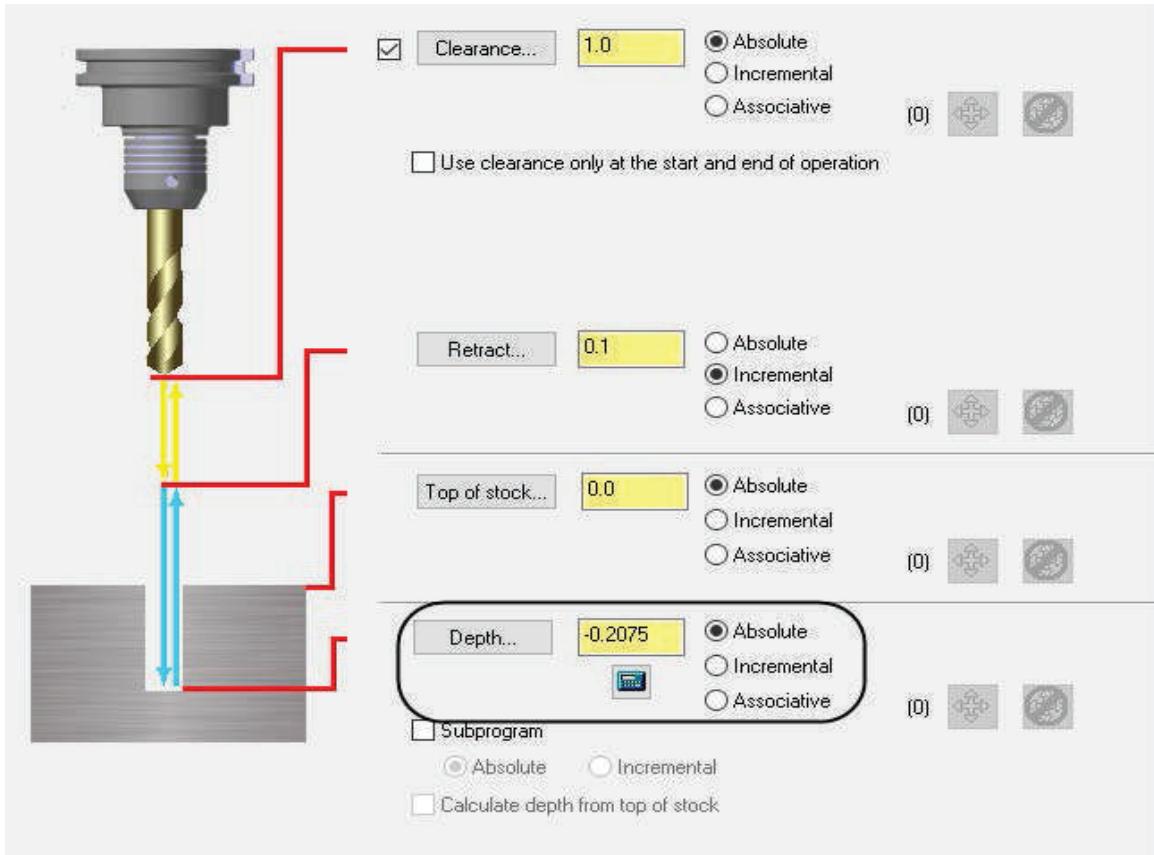
- ◆ Select the **OK** button to exit the **Depth Calculator**.





- You will now see the depth we calculated for the spot drilling operation set in the **Depth** field as shown in [Figure: 15.2.2](#). This will chamfer the holes for the tapping operation.

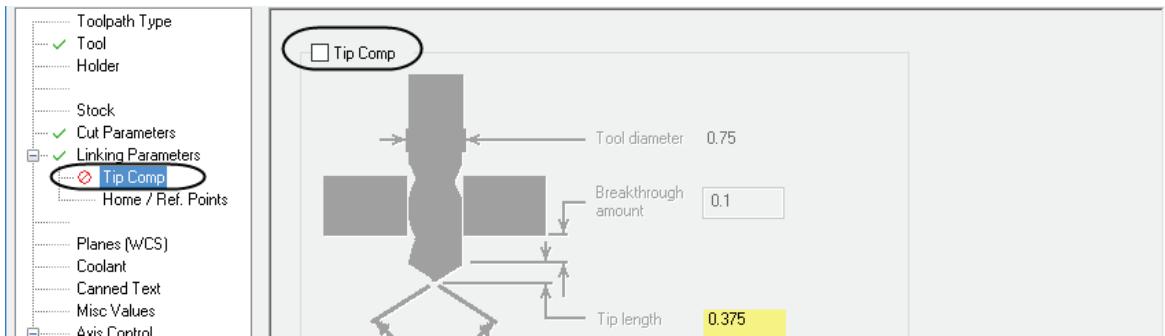
Figure: 15.2.2

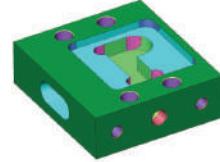


15.3 Set the Tip Comp

- Select **Tip Comp** and disable this option as shown in [Figure: 15.3.1](#). If left enabled, the holes would be drilled much deeper.

Figure: 15.3.1





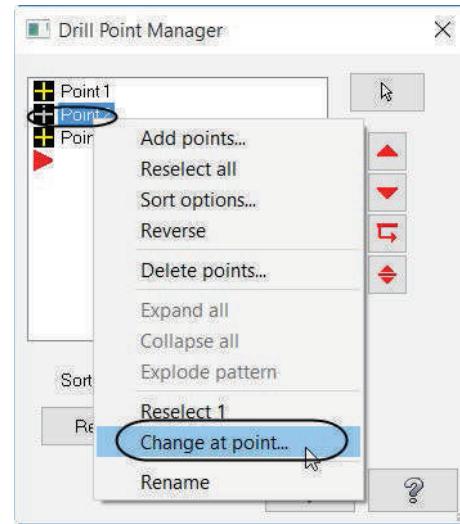
- ♦ Select the **OK** button to exit the **2D Toolpaths - Drill/Circles Simple drill - no peck** parameters and generate the toolpath. 

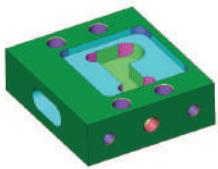
NOTE: All 3 holes are spot drilled to the same depth. The 0.5" diameter hole has to be drilled to a deeper depth.

15.4 Adjust the Depth of the Spot Drill

- ♦ Left click on **Geometry** in **Operation #8**.
- ♦ When the **Drill Point Manager** appears, select the point which represents the **0.5" diameter hole**.
- ♦ Then right click on it and pick the option to **Change at point** as shown in [Figure: 15.4.1](#).

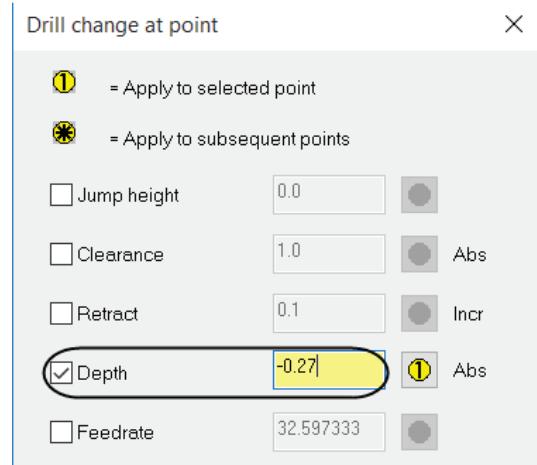
Figure: 15.4.1





- When the **Drill change at point** dialog box appears enable **Depth** and change the depth to **-0.27** as shown in [Figure: 15.4.2](#).

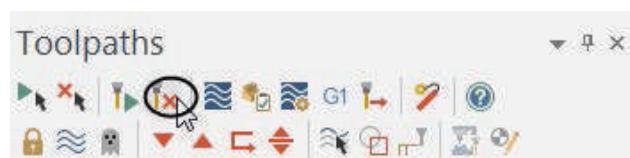
Figure: 15.4.2

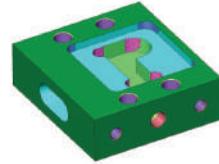


Drill Change At Point allows you to make point-specific changes to a drill toolpath.

Depth changes the hole depth at the selected point. The coordinate you enter here will be output as either an absolute or incremental value, depending on the original settings for the operation.

- Select the **OK** button to apply the changes and exit the **Drill change at point** dialog box.
- Choose the **OK** button to exit the **Drill Point Manager**.
- Pick the **Regenerate all dirty operations** button.

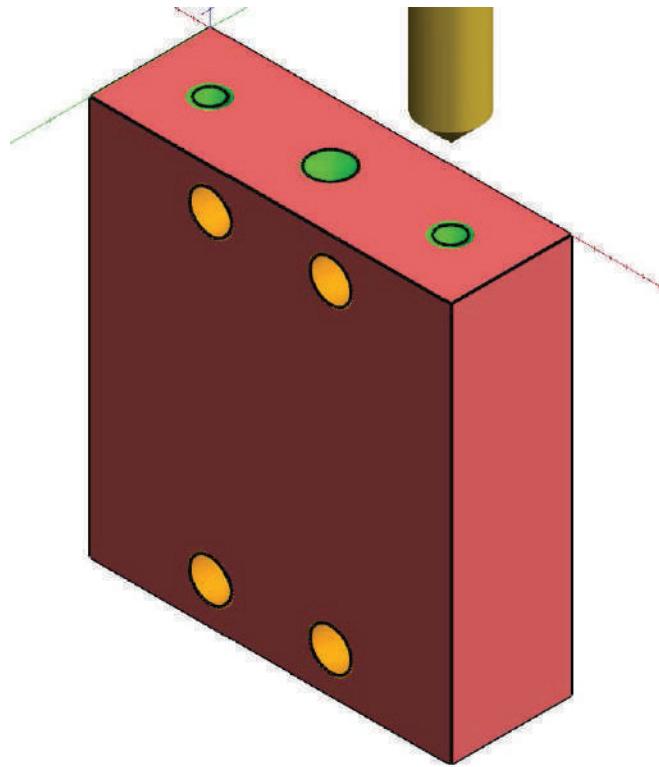




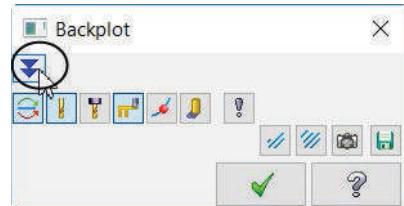
15.5 Backplot and Verify the toolpaths

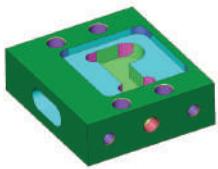
- ♦ To **Backplot** and **Verify** your toolpaths, see [page 447](#) and [page 448](#).

- ♦ To make sure that all toolpaths are selected, choose the **Select all operations** icon.

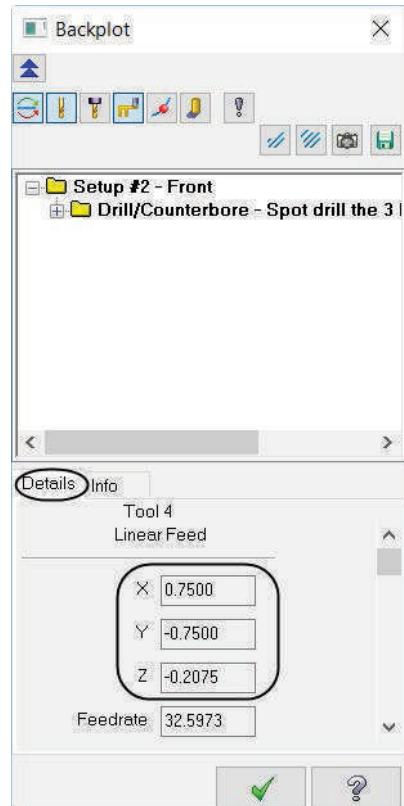


- ♦ To go back to the **Mastercam** window, minimize the **Mastercam Simulator** window as shown.
- ♦ When backplotting the toolpath, you can check the depth of the holes.
- ♦ Select the **Expand or contract this dialog** double arrow to expand the **Backplot** dialog box.

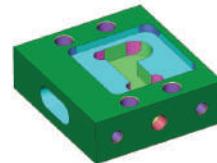




- ♦ Select the **Details** tab as shown below.
- ♦ You can step through the **Backplot** by using the **Step forward** or **Step back** .
- ♦ The coordinate values of the tip of the tool will be displayed with each step.



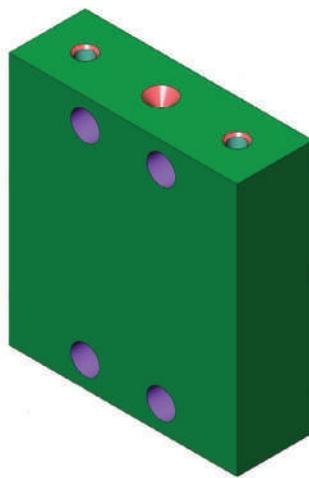
- ♦ Select the **OK** button to exit **Backplot**.



STEP 16: DRILL THE TWO 3/8" TAPPED HOLES

In this example we will drill the holes to a specific depth.

Toolpath Preview:

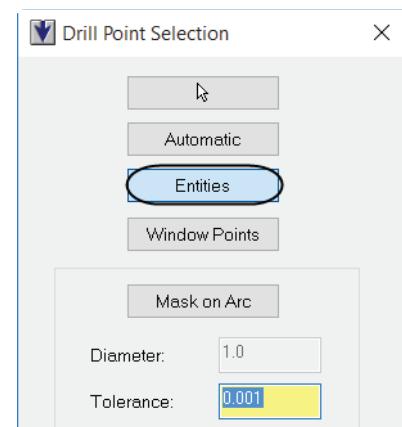


TOOLPATHS

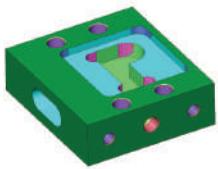
- From the **2D** group, select the **Drill** icon.



- In the **Drill Point Selection** dialog box choose the option **Entities**.

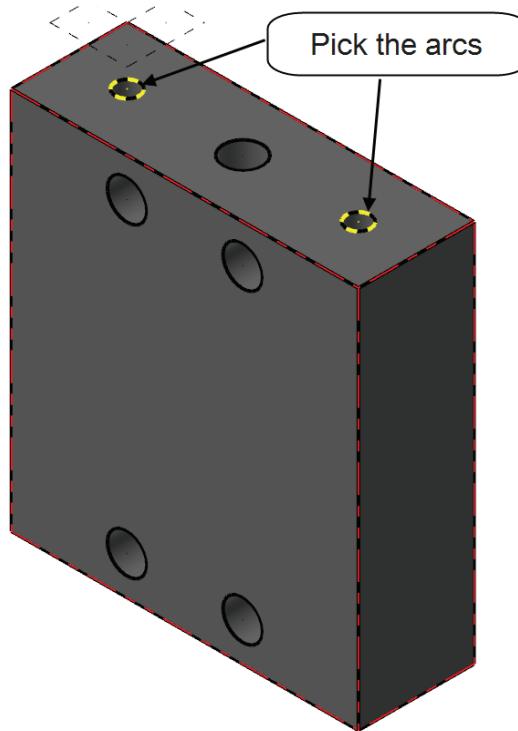


NOTE: This option will let you select the arcs we wish to drill.

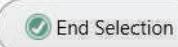


- ♦ Pick the arcs as shown in [Figure: 16.0.1](#).

Figure: 16.0.1



- ♦ Click on the **End Selection** button or press **Enter** to finish the selection.



- ♦ Select the **OK** button in the **Drill Point Selection** dialog box to accept the 2 drill points.

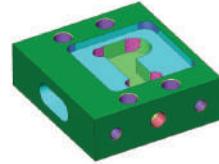


- ♦ In the **Toolpath Type** page, the **Drill** toolpath will be selected.



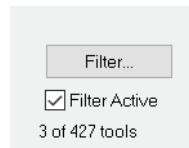
TUTORIAL #5

DRILL THE TWO 3/8" TAPPED HOLES



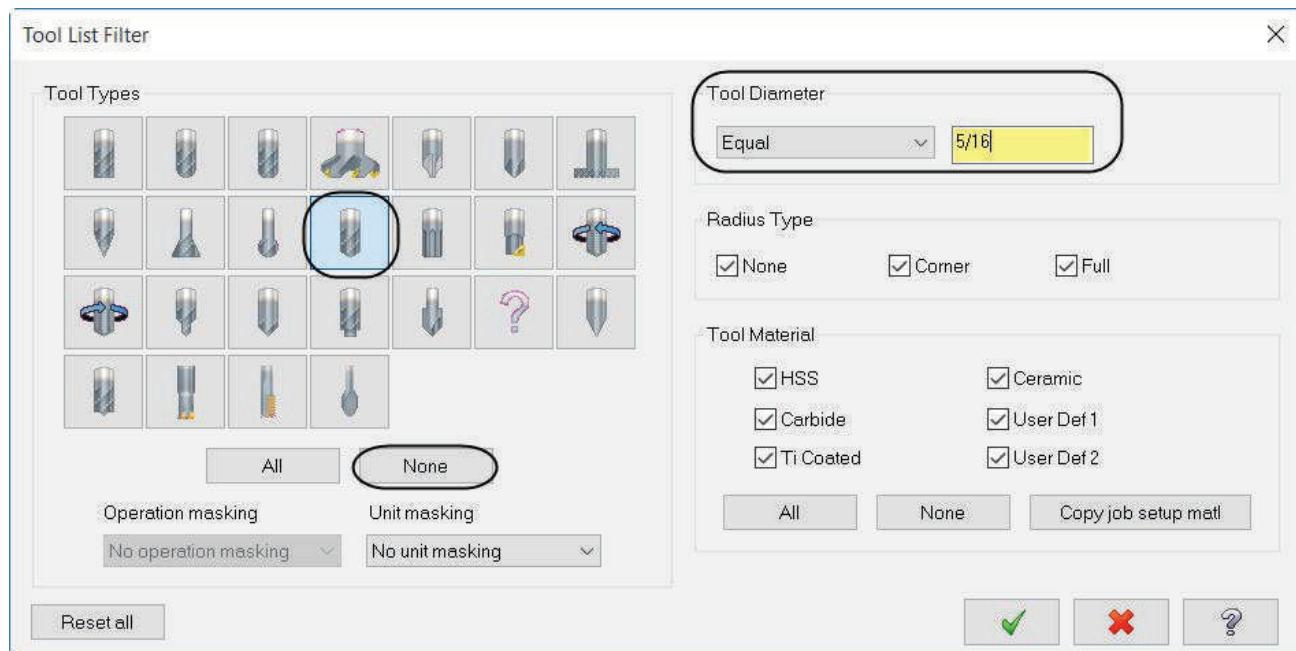
16.1 Select a 5/16" Drill from the Library and set the Tool Parameters

- ♦ Select **Tool** from the **Tree View list**.
- ♦ Click on the **Select library tool** button. Select library tool...
- ♦ To be able to see just the **5/16" Drill**, select the **Filter** button.



- ♦ Under **Tool Types** select the **None** button and then choose the **Drill** icon.
- ♦ Under **Tool Diameter** select **Equal** and enter the value **5/16** as shown in [Figure: 16.1.1](#).

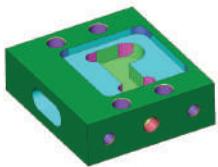
Figure: 16.1.1



- ♦ Select the **OK** button to exit the **Tool List Filter** dialog box. ✓
- ♦ At this point you should see a **5/16" Drill**.
- ♦ Select the **5/16" Drill**.

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	Type	# Flutes	Ra...
137	-	5/16 DRILL	-	0....	0.0	2.0	Drill	2	No...

- ♦ After selecting the tool in the **Tool Selection** page, choose the **OK** button to exit. ✓



- ♦ Make the necessary changes to the **Tool** page as shown in [Figure: 16.1.2.](#)

Figure: 16.1.2

The screenshot shows the Mastercam software interface with the 'Tool' page open. On the left, there is a list of tools with the following data:

#	Assembly Name	Tool Name	Holder Name	Dia.
1	-	3/4 FLAT...	-	0.75
2	-	5/8 FLAT...	-	0....
3	-	3/8 FLAT...	-	0....
4	-	3/4 SPOT...	-	0.75
5	-	33/64 DRI...	-	0....
6	-	9/16-18 T...	-	0....
7	-	5/16 DRILL	-	0....

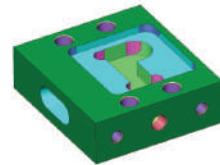
On the right, the tool settings are displayed:

- Tool diameter: 0.3125
- Corner radius: 0.0
- Tool name: 5/16 DRILL
- Tool #: 7
- Length offset: 7
- Head #: 0
- Diameter offset: 7
- Spindle direction: CW
- Feed rate: 4.244173
- Spindle speed: 855
- FPT: 0.0025
- SFM: 69.9444
- Plunge rate: 4.244173
- Retract rate: 4.244173
- Force tool change
- Rapid Retract
- Comment: Drill the two 3/8 tap holes.

At the bottom left, there is a checkbox labeled 'To batch'.

TUTORIAL #5

DRILL THE TWO 3/8" TAPPED HOLES



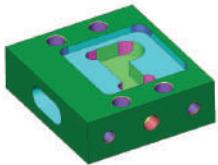
16.2 Set the Cut Parameters

- ♦ Select **Cut Parameters**, change the drill **Cycle** to **Peck Drill** and input a **Peck** value of **0.25** as shown in [Figure: 16.2.1](#).

Figure: 16.2.1

Cycle: Peck Drill
Peck: 0.25
Subsequent peck: 0.1
Peck clearance: 0.1
Retract amount: 0.1
Dwell: 0.0
Shift: 0.1
 Apply custom drill parameters
Parameter #1: 0.0 Parameter #6: 0.0
Parameter #2: 0.0 Parameter #7: 0.0
Parameter #3: 0.0 Parameter #8: 0.0
Parameter #4: 0.0 Parameter #9: 0.0
Parameter #5: 0.0 Parameter #10: 0.0

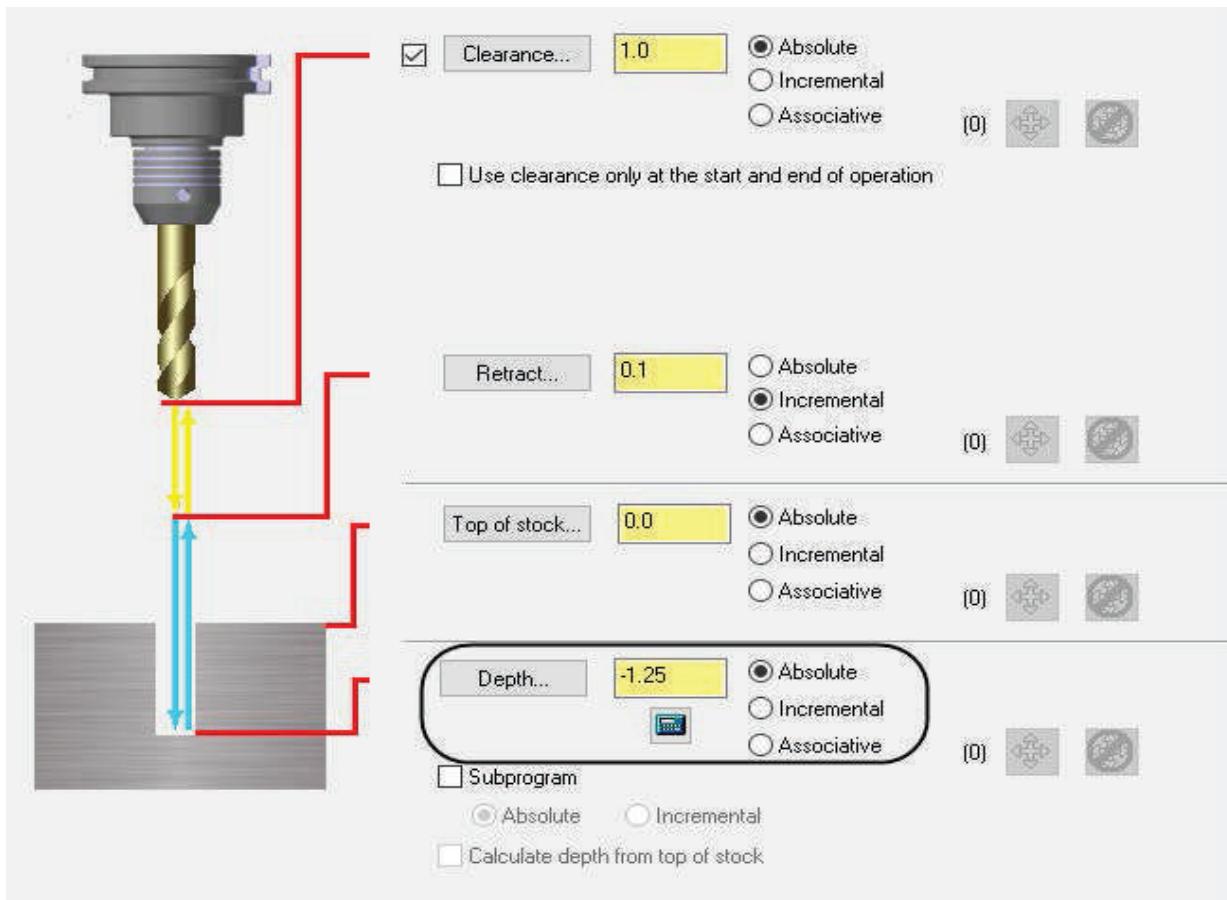
NOTE: For more information regarding the settings found on this page, see [page 483](#).



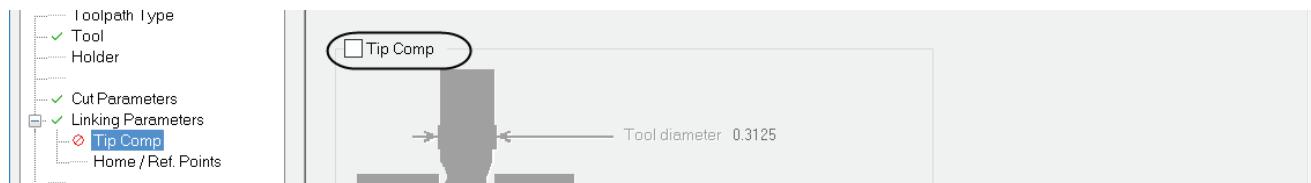
16.3 Set the Linking Parameters

- Choose **Linking Parameters** from the **Tree View** list and input a **Depth** value of **-1.25** as shown in [Figure: 16.3.1](#).

Figure: 16.3.1

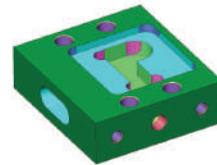


- Pick **Tip Comp** from the **Tree View** list and ensure this option is disabled.



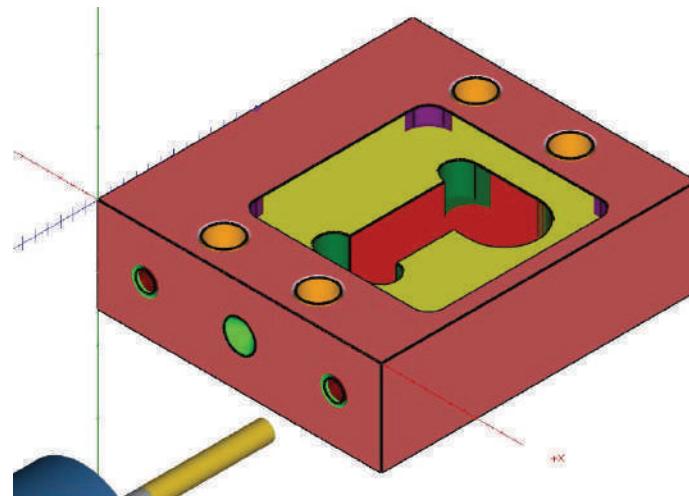
- Select the **OK** button to exit the **2D Toolpaths - Drill/Circles Simple drill - full retract** parameters.





16.4 Backplot and Verify

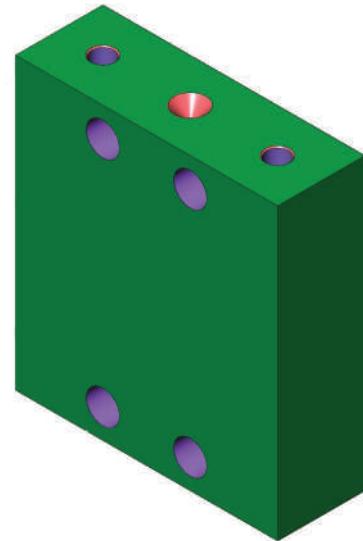
- ♦ To **Backplot** and **Verify** the toolpaths, see **page 447** and **page 448**.
- ♦ Use the mouse wheel to rotate the part as shown.

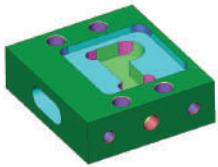


- ♦ To go back to the **Mastercam** window, minimize the **Mastercam Simulator** window as shown.

STEP 17: TAP THE TWO HOLES

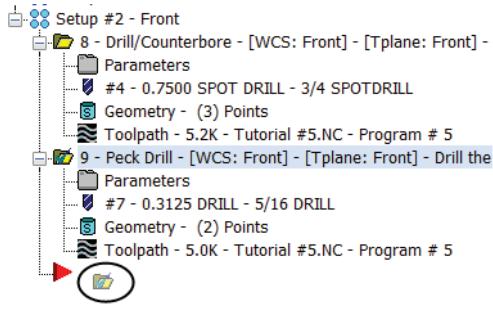
Toolpath Preview:



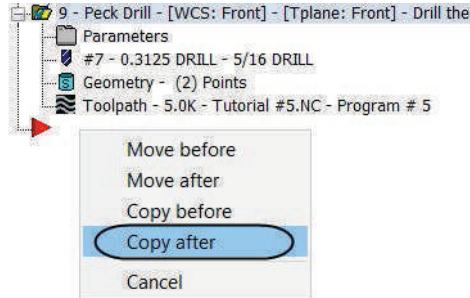


TOOLPATHS

- ♦ Select operation #9.
- ♦ Right click, hold the right mouse button down, and drag the operation to a point below it as shown.



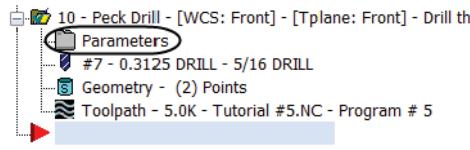
- ♦ Release the right mouse button and select the option **Copy After**.



- ♦ Select the **Move insert arrow down one item** button to move the insert arrow down.



- ♦ Select **Parameters**.



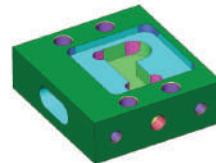
- ♦ In the **Toolpath Type** page, the **Drill** toolpath will be selected.



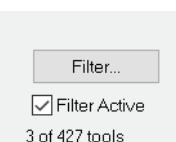
17.1 Select a 3/8 - 16 RH Tap from the Library and set the Tool Parameters

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

TUTORIAL #5 TAP THE TWO HOLES

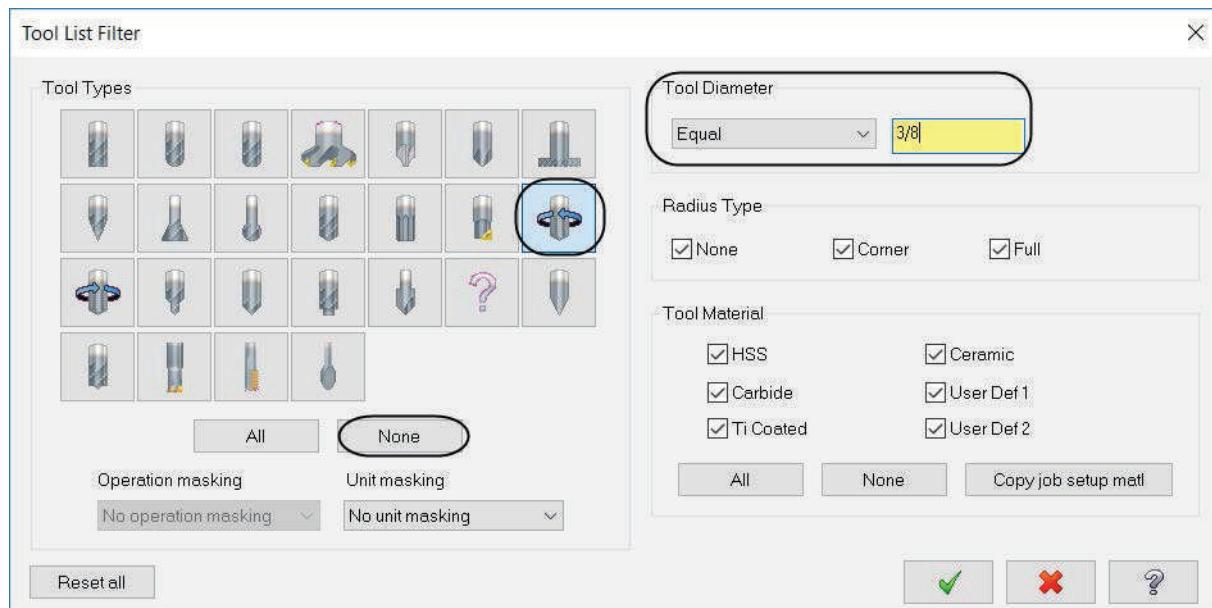


- To be able to see just the RH Tap select the Filter button.

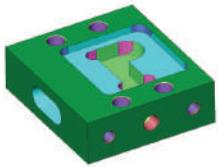


- Under **Tool Types** select the **None** button and then choose the **Tap RH Icon**. Under **Tool Diameter**, make sure the **Equal** option is selected and enter the diameter **3/8** as shown in [Figure: 17.1.1](#).

Figure: 17.1.1



- Select **OK** button to exit the **Tool List Filter** dialog box.



- From the list select the **3/8 - 16 Tap RH** as shown in [Figure: 17.1.2](#).

Figure: 17.1.2

#	Assembly...	Tool Name	Holder Name	Dia.	Cor. r...	Length	# Flut...	Type	Rad....
223	-	3/8-24 TAPRH	-	0.37...	0.0	2.0	1	Tap ...	None
224	-	3/8-16 TAPRH	-	0.37...	0.0	2.0	1	Tap ...	None

- Select the tool in the **Tool Selection** page and then choose the **OK** button to exit.
- Make the necessary changes to the **Tool** page as shown in [Figure: 17.1.3](#).

Figure: 17.1.3

The screenshot shows the Mastercam software interface. On the left, a list of tools is displayed in a table:

#	Assembly Name	Tool Name	Holder Name	Dia.
1	-	3/4 FLAT ...	-	0.75
2	-	5/8 FLAT ...	-	0....
3	-	3/8 FLAT ...	-	0....
4	-	3/4 SPOT...	-	0.75
5	-	33/64 DRI...	-	0....
6	-	9/16-18 T...	-	0....
7	-	5/16 DRILL	-	0....
8	-	3/8-16 TA...	-	0....

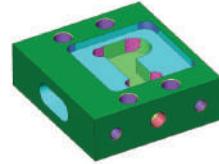
On the right, the **Tool** page settings are shown:

- Tool diameter: 0.375
- Corner radius: 0.0
- Tool name: 3/8-16 TAPRH
- Tool #: 8
- Length offset: 8
- Head #: 0
- Diameter offset: 8
- Spindle direction: CW
- Feed rate: 44.566667
- Spindle speed: 713
- FPT: 0.0625
- SFM: 69.9935
- Plunge rate: 44.566667
- Retract rate: 44.566667
- Force tool change
- Rapid Retract

A comment box at the bottom contains the text: "Comment Tap the two 3/8 holes in the front plane."

At the bottom left, there is a checkbox labeled "To batch".

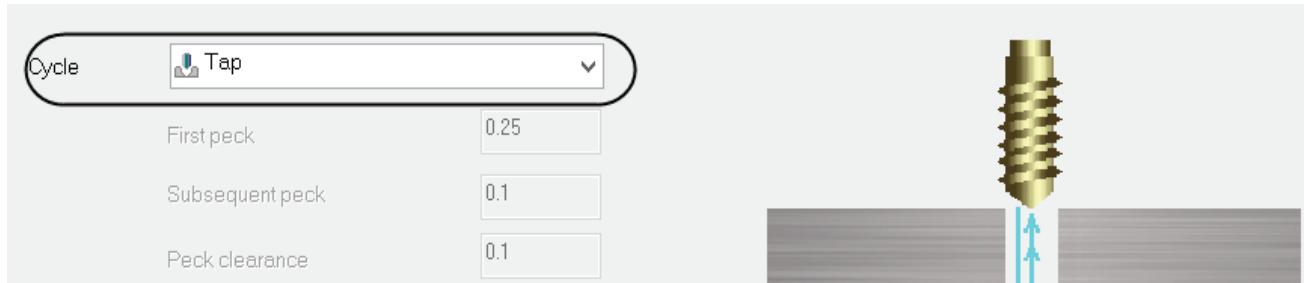
TUTORIAL #5 TAP THE TWO HOLES



17.2 Set the Cut Parameters

- ♦ Select **Cut Parameters** from the **Tree View list** and change the drill **Cycle to Tap** as shown in [Figure: 17.2.1.](#)

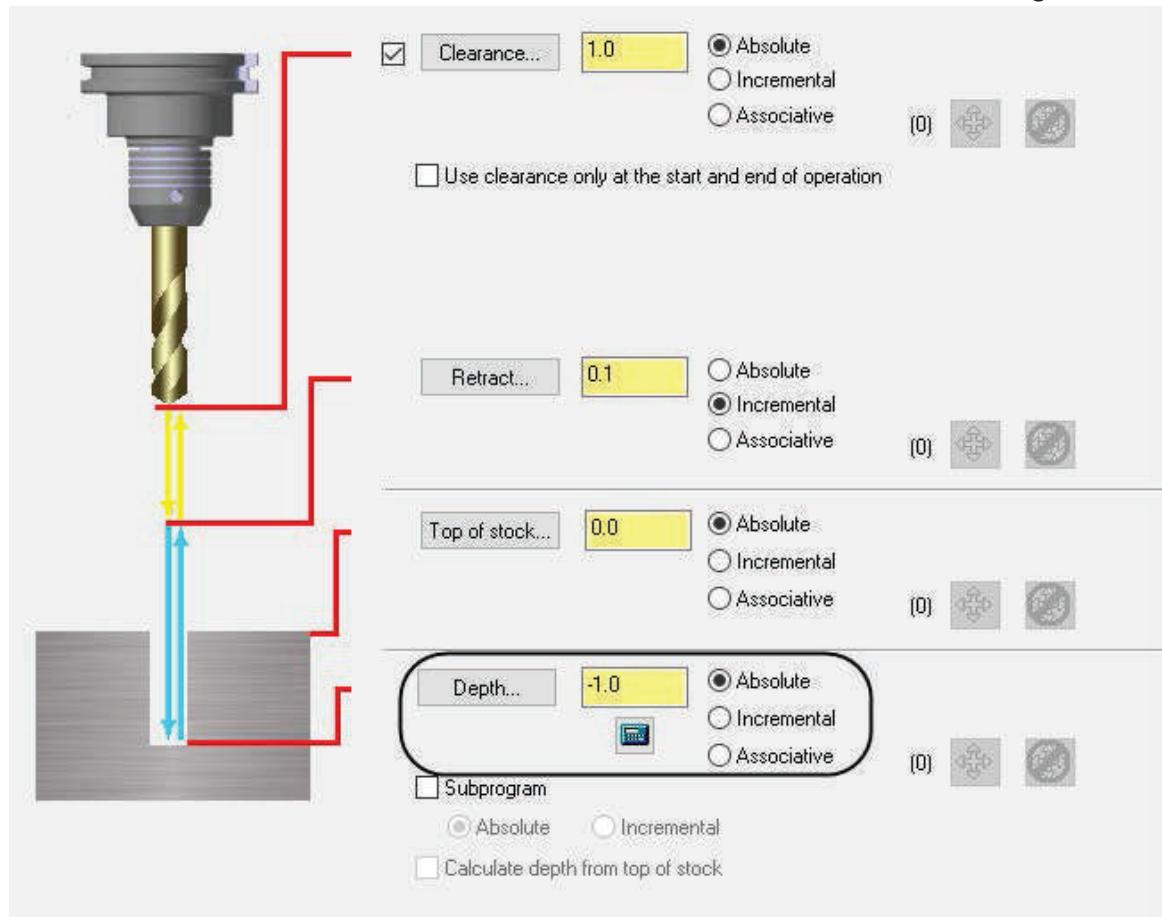
Figure: 17.2.1



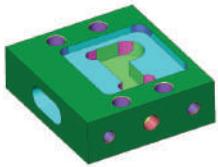
17.3 Set the Linking Parameters

- ♦ Choose **Linking Parameters** and input a **Depth** of **-1.0** as shown in [Figure: 17.3.1.](#)

Figure: 17.3.1



- ♦ Select **Tip Comp** and ensure this option is disabled.
- ♦ The **Depth** set in the **Linking Parameters** page is as deep as we would like the tap to go.



TAP THE TWO HOLES

TUTORIAL #5

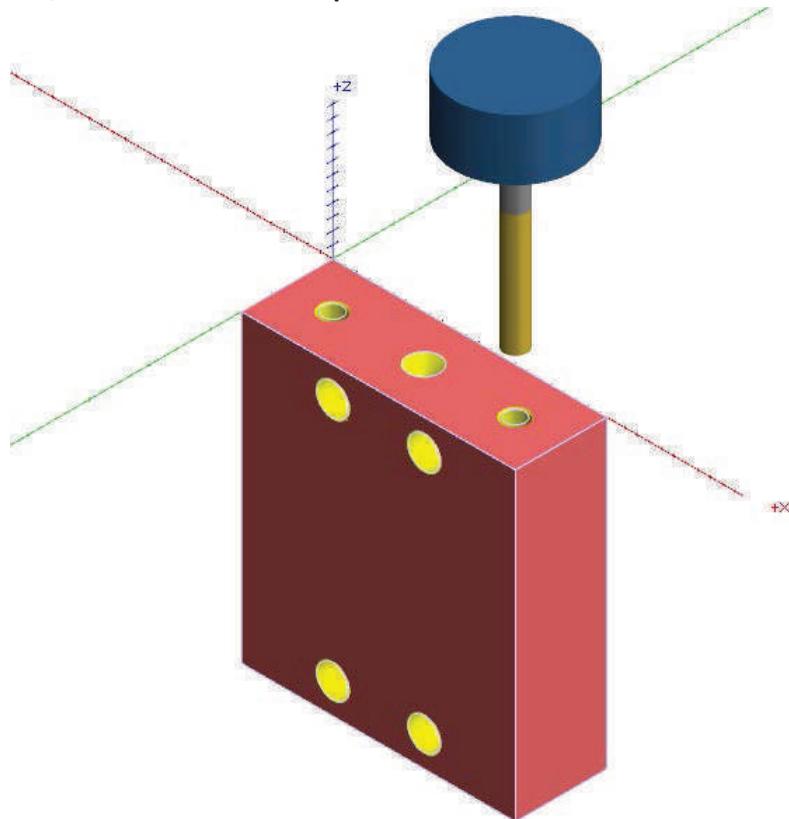
- ♦ Select the **OK** button to exit the **2D Toolpaths - Drill/Circles Tapping - feed in, reverse spindle, feed out** parameters.
- ♦ Select the **Regenerate all dirty operations** icon if necessary.



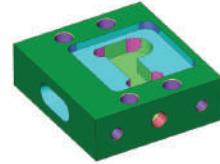
17.4 Backplot and Verify

- ♦ To **Backplot** and **Verify** the toolpaths, see [page 447](#) and [page 448](#).

- ♦ To make sure that all toolpaths are selected, choose the **Select all operations** icon.



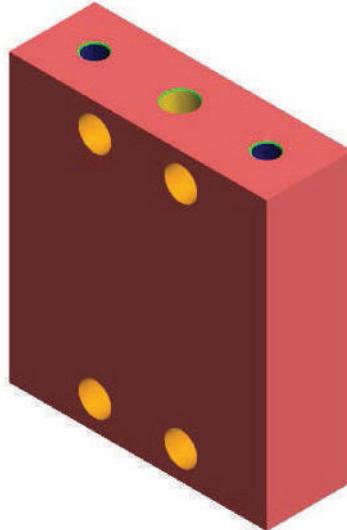
- ♦ To go back to the **Mastercam** window, minimize the **Mastercam Simulator** window as shown.



STEP 18: DRILL THE 1/2" HOLE

You will drill the hole to a specific depth.

Toolpath Preview:



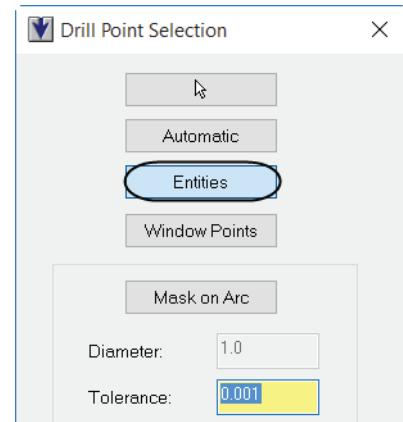
TOOLPATHS

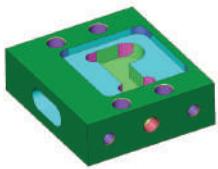
- From the **2D** group, select the **Drill** icon.



- In the **Drill Point Selection** dialog box, choose the **Entities** button as shown in [Figure: 18.0.1](#).

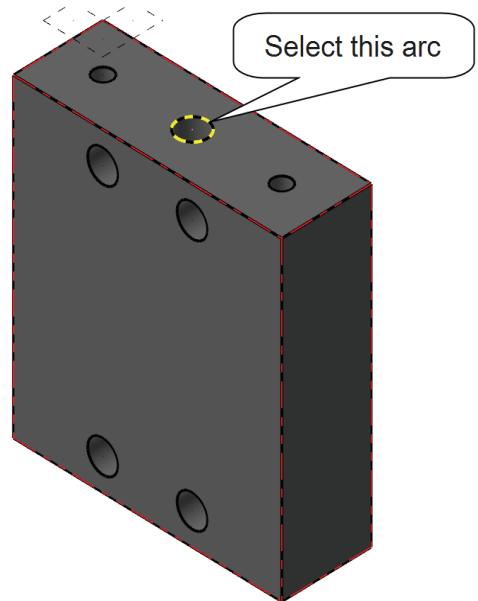
Figure: 18.0.1





- ◆ This option will let you select the arcs we wish to drill. Pick the arc as shown in [Figure: 18.0.2](#).

Figure: 18.0.2



- ◆ Click on the **End Selection** button or press **Enter** to finish the selection.



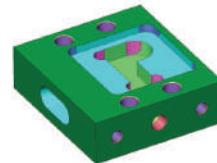
- ◆ Select the **OK** button in the **Drill Point Selection** dialog box to accept the 1 drill point.



- ◆ In the **Toolpath Type** page, the **Drill** toolpath will be selected.

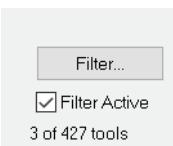


TUTORIAL #5 DRILL THE 1/2" HOLE



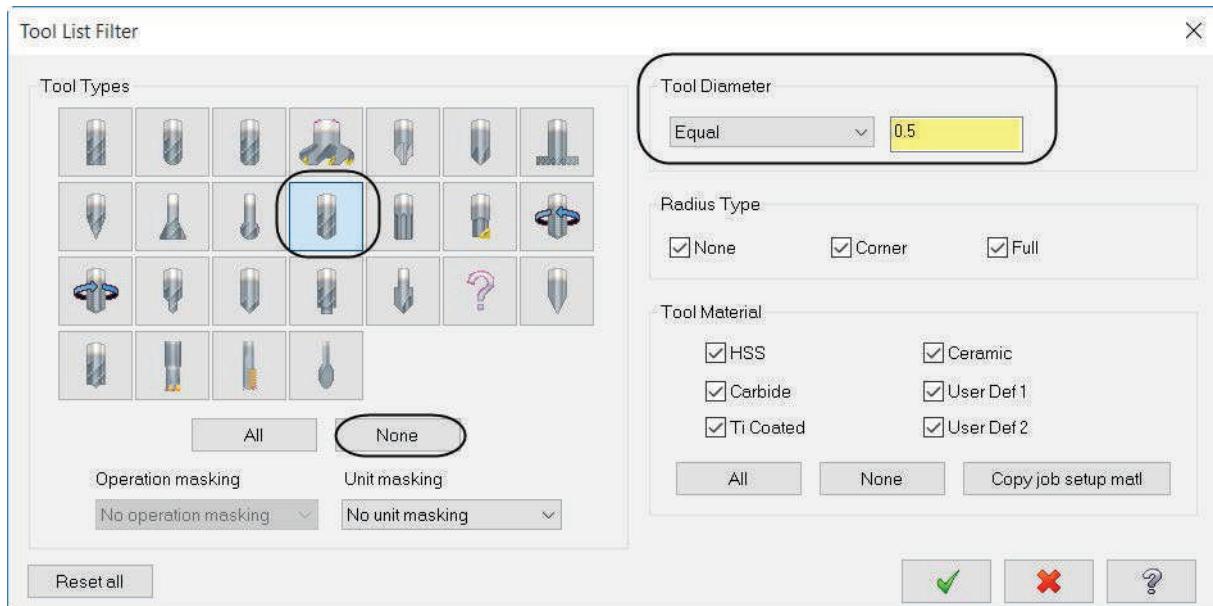
18.1 Select a 1/2" Drill from the Library and set the Tool Parameters

- ♦ Select **Tool** from the **Tree View** list.
- ♦ Click on the **Select library tool...** button.
- ♦ To be able to see just the **1/2" Drill**, select the **Filter** button.



- ♦ Under **Tool Types** select the **None** button and then choose the **Drill** Icon. Under **Tool Diameter** select **Equal** and enter the value **0.5** as shown in [Figure: 18.1.1](#).

Figure: 18.1.1



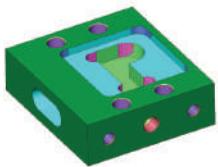
- ♦ Select the **OK** button to exit the **Tool List Filter** dialog box.
- ♦ Select the **1/2"** tool.



#	Asse...	Tool Name	Holder Name	Dia.	Cor. rad.	Length	Type	# Flutes	Ra...
162	--	1/2 DRILL	--	0.5	0.0	2.0	Drill	2	No...

- ♦ Select the tool in the **Tool Selection** page and then choose the **OK** button to exit.





- ♦ Make the necessary changes to the **Tool** page as shown in [Figure: 18.1.2.](#)

Figure: 18.1.2

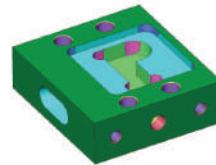
The screenshot shows the Mastercam software interface with the 'Tool' page open. On the left, a list of tools is displayed in a table:

#	Assembly Name	Tool Name	Holder Name	Dia.
1	-	3/4 FLAT ...	-	0.75
2	-	5/8 FLAT ...	-	0....
3	-	3/8 FLAT ...	-	0....
4	-	3/4 SPOT...	-	0.75
5	-	33/64 DRILL...	-	0....
6	-	9/16-18 T...	-	0....
7	-	5/16 DRILL	-	0....
8	-	3/8-16 TA...	-	0....
9	-	1/2 DRILL	-	0.5

The right side of the screen displays various toolpath parameters:

- Tool diameter: 0.5
- Corner radius: 0.0
- Tool name: 1/2 DRILL
- Tool #: 9
- Length offset: 9
- Head #: 0
- Diameter offset: 9
- Spindle direction: CW
- Feed rate: 4.2784
- Spindle speed: 534
- FPT: 0.004
- SFM: 69.8953
- Plunge rate: 4.2784
- Retract rate: 4.2784
- Force tool change
- Rapid Retract
- Comment: Drill the 1/2" hole in the front plane.

At the bottom left, there are checkboxes for 'Select library tool...', 'Filter Active', 'Filter...', and 'To batch'.



18.2 Set the Cut Parameters

- ♦ Select **Cut Parameters**, change the drill **Cycle** to **Chip Break** and input a **Peck** of **0.25** as shown in [Figure: 18.2.1](#).

Figure: 18.2.1

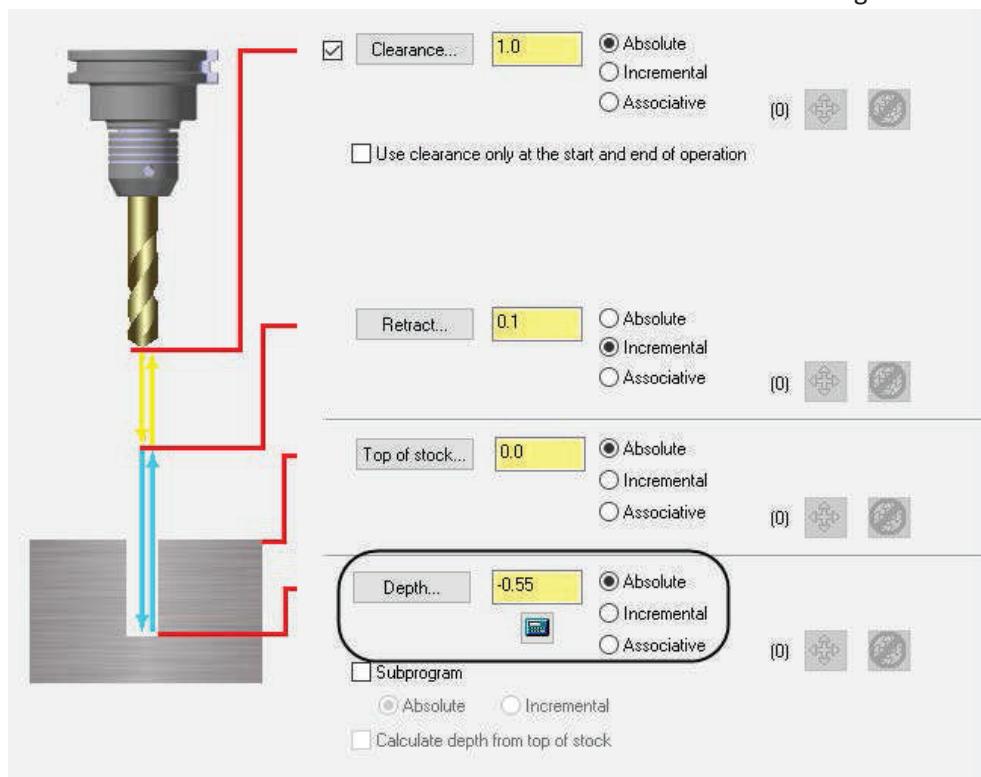


Chip Break retracts partially out of the drilled hole to break the material chips.

18.3 Set the Linking Parameters

- ♦ Choose **Linking Parameters** and input a **Depth** value of **-0.55** as shown in [Figure: 18.3.1](#).

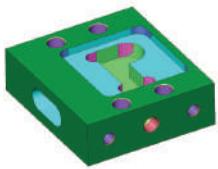
Figure: 18.3.1



- ♦ Select **Tip Comp** and ensure this option is disabled.

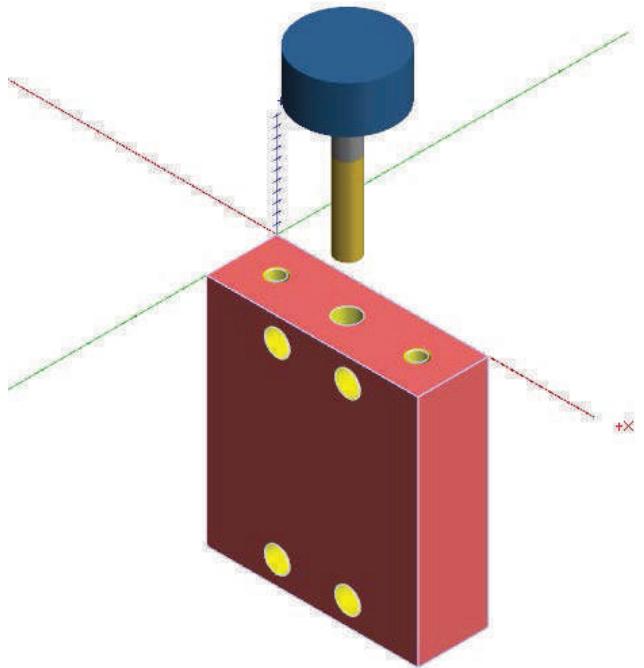
- ♦ Select the **OK** button to exit the parameters.





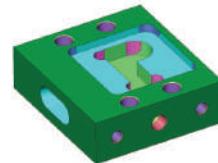
18.4 Backplot and Verify

- ♦ To **Backplot** and **Verify** the toolpaths, see [page 447](#) and [page 448](#).



- ♦ To go back to the **Mastercam** window, minimize the **Mastercam Simulator** window as shown.

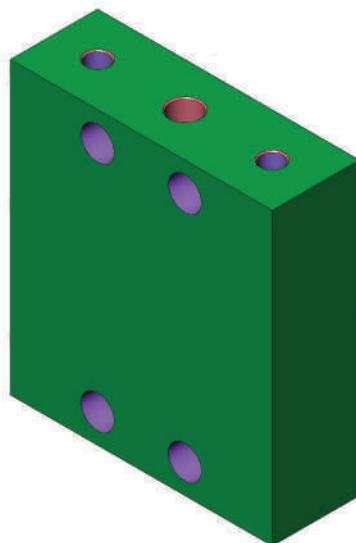




STEP 19: COUNTERBORE THE 1/2" HOLE

In this step you will drill the holes to a specific depth using a 1/2" Flat endmill to give us a hole with a flat bottom.

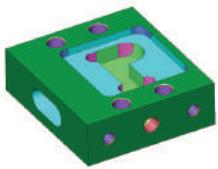
Toolpath Preview:



TOOLPATHS

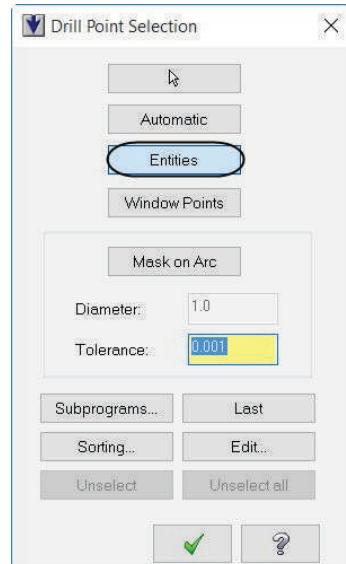
- From the **2D** group, select the **Drill** icon.





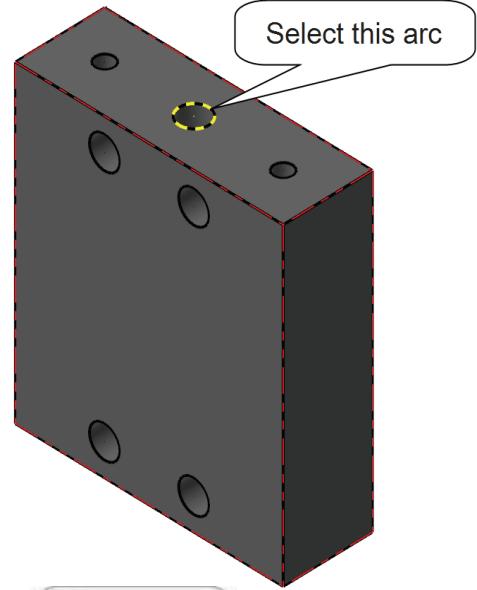
- In the Drill Point Selection dialog box, choose the Entities button as shown in [Figure: 19.0.1](#).

Figure: 19.0.1

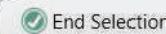


- Select the arc as shown in [Figure: 19.0.2](#).

Figure: 19.0.2

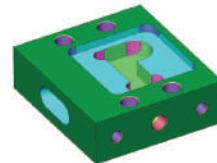


- Click on the End Selection button or press Enter to finish the selection.

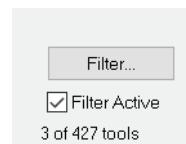


- Select the OK button in the Drill Point Selection dialog box to accept the 1 drill point.
- In the Toolpath Type page, the Drill toolpath will be selected.



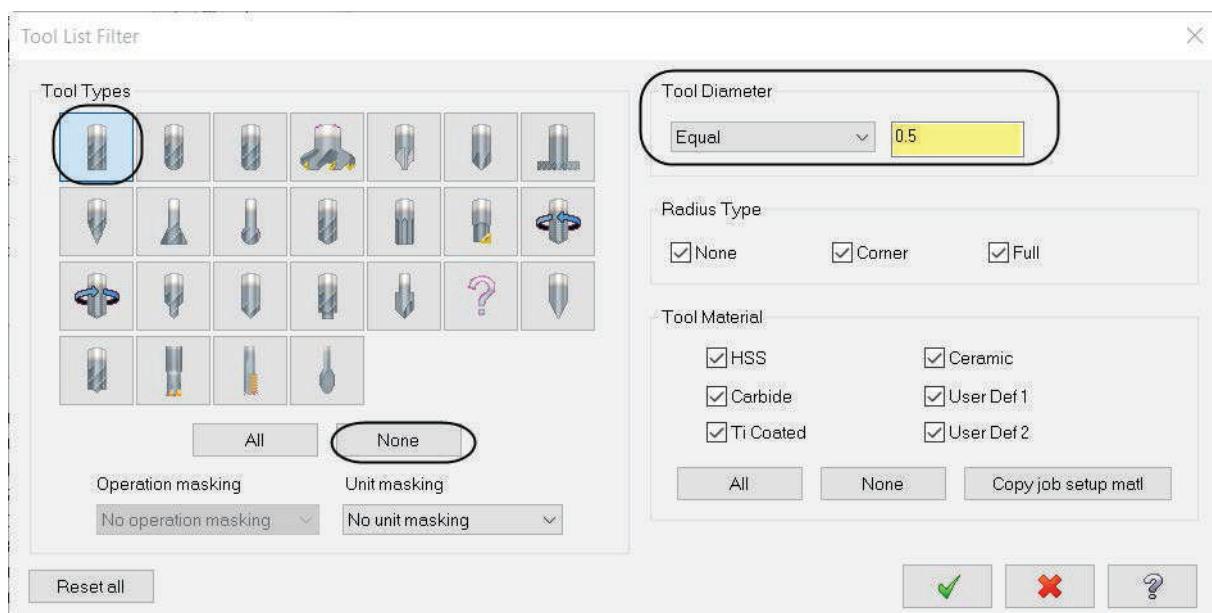
TUTORIAL #5**COUNTERBORE THE 1/2" HOLE****19.1 Select a 1/2" Flat Endmill from the Library and set the Tool Parameters**

- ♦ Select **Tool** from the **Tree View** list.
- ♦ Click on the **Select library tool...** button.
- ♦ To be able to see just the **Flat Endmill**, select the **Filter** button.



- ♦ Under **Tool Types** select the **None** button and then choose the **Flat Endmill** icon. Under **Tool Diameter**, select **Equal** and enter the value **0.5** as shown in [Figure: 19.1.1](#).

Figure: 19.1.1

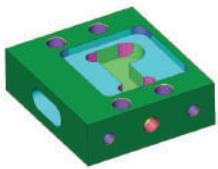


- ♦ Select the **OK** button to exit the **Tool List Filter** dialog box.
- ♦ Select the **1/2" Flat Endmill** as shown in [Figure: 19.1.2](#).

Figure: 19.1.2

#	Assembly...	Tool Name	Holder Name	Dia.	Cor. r...	Length	# Flut...	Type	Rad....
290	-	1/2 FLAT ENDMILL	-	0.5	0.0	1.0	4	End...	None

- ♦ Select the tool in the **Tool Selection** page and then choose the **OK** button to exit.



- ♦ Make the necessary changes to the **Tool** page as shown in [Figure: 19.1.3.](#)

Figure: 19.1.3

The screenshot shows the Mastercam software interface with the 'Tool' page selected. On the left, there is a list of tools in a grid format:

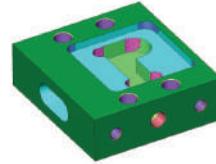
#	Assembly Name	Tool Name	Holder Name	Dia.
1	-	3/4 FLAT ...	-	0.75
2	-	5/8 FLAT ...	-	0....
3	-	3/8 FLAT ...	-	0....
4	-	3/4 SPOT...	-	0.75
5	-	33/64 DRILL...	-	0....
6	-	9/16-18 T...	-	0....
7	-	5/16 DRILL	-	0....
8	-	3/8-16 TA...	-	0....
9	-	1/2 DRILL	-	0.5
10	-	1/2 FLAT ...	-	0.5

On the right side, there are several input fields for toolpath parameters:

- Tool diameter: 0.5
- Corner radius: 0.0
- Tool name: 1/2 FLAT ENDMILL
- Tool #: 10
- Length offset: 10
- Head #: 0
- Diameter offset: 10
- Spindle direction: CW
- Feed rate: 6.4176
- Spindle speed: 1069
- FPT: 0.0015
- SFM: 139.9215
- Plunge rate: 6.4176
- Retract rate: 6.4176
- Force tool change
- Rapid Retract

A comment box at the bottom contains the text: "Counterbore the 1/2" hole in the front plane."

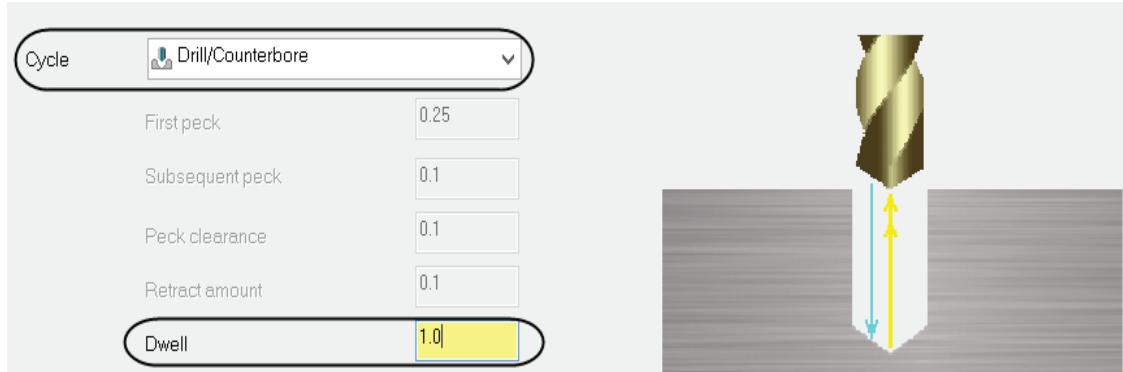
At the bottom left, there are buttons for "Select library tool...", "Filter Active", and "Filter...". There is also a checkbox for "To batch".



19.2 Set the Cut Parameters

- ♦ Select **Cut Parameters**, change the drill **Cycle** to **Drill/Counterbore** and input a **Dwell** of **1.0** second as shown in [Figure: 19.2.1](#).

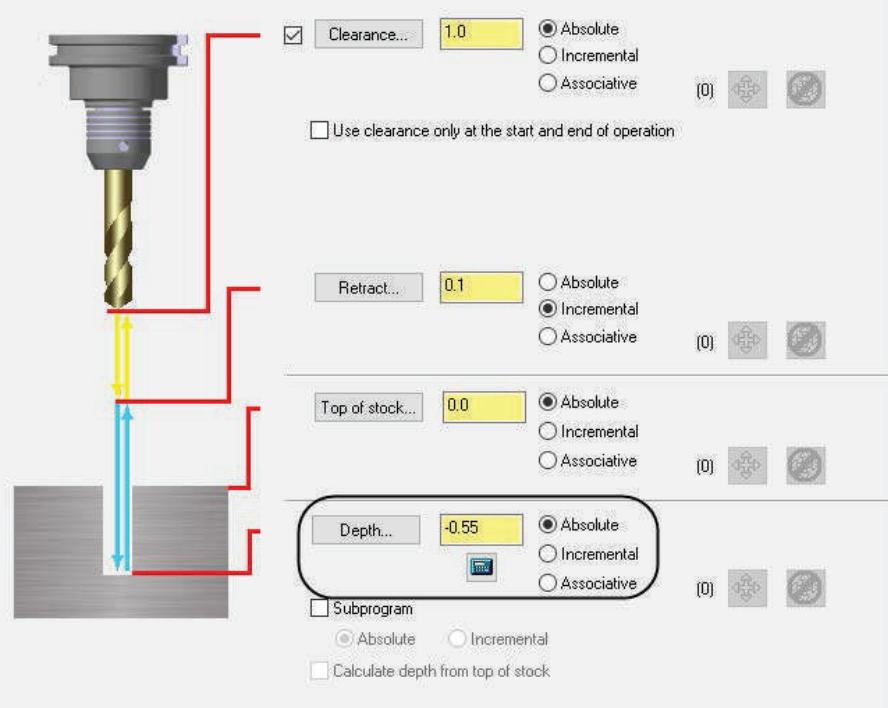
Figure: 19.2.1



19.3 Set the Linking Parameters

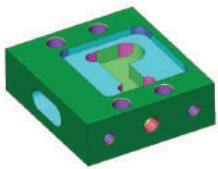
- ♦ Choose **Linking Parameters** and input a **Depth** value of **-0.55** as shown in [Figure: 19.3.1](#).

Figure: 19.3.1



- ♦ Select **Tip Comp** from the **Tree View list** and ensure this page is disabled.
- ♦ Select the **OK** button to exit the **2D Toolpaths - Drill/Circles Simple drill - no peck** parameters and generate the toolpath.

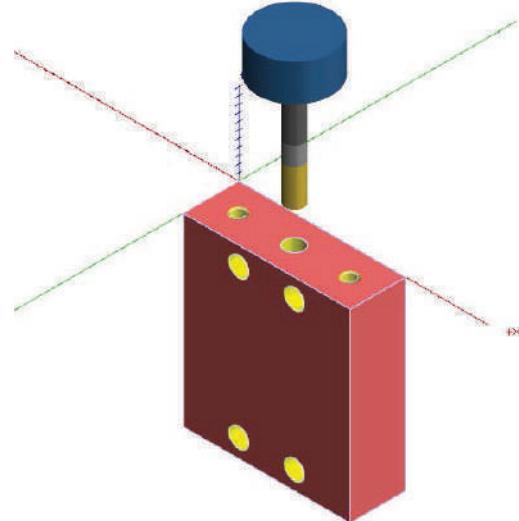




19.4 Backplot and Verify

- ♦ To **Backplot** and **Verify** the toolpaths, see [page 447](#) and [page 448](#).

- ♦ To make sure that all toolpaths are selected, choose the **Select all operations** icon.

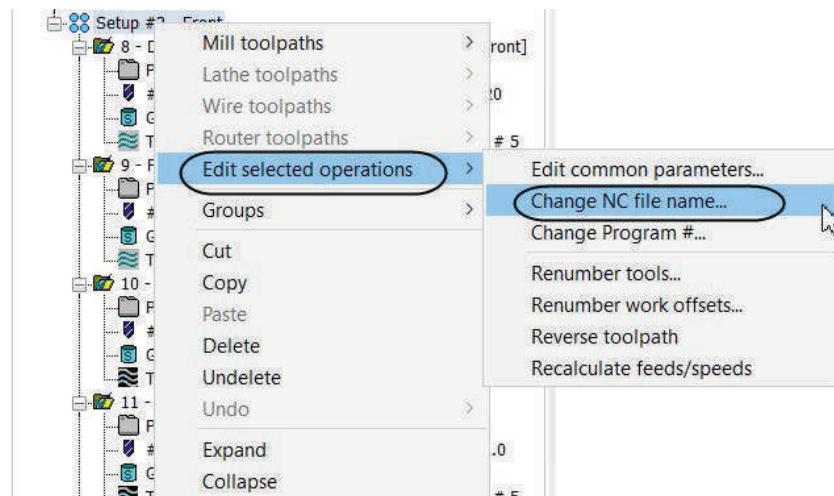


- ♦ To go back to the **Mastercam** window, minimize the **Mastercam Simulator** window as shown.

STEP 20: RENAME NC FILE

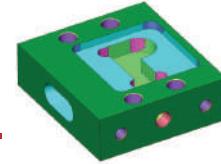
The Drilling and Tapping operations in "Setup #2 - Front" kept the NC name from Setup #1. We need to rename this operation so it will create a separate program for this setup.

- ♦ Select "Setup #2 - Front" (make sure all of the operations in setup #2 are selected).
- ♦ Right click on the group, choose the option **Edit selected operations** and then pick **Change NC file name...**.

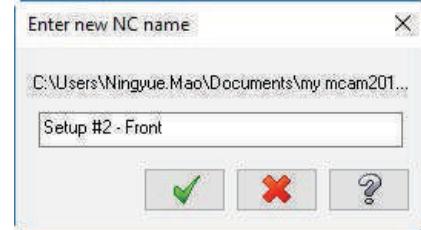


TUTORIAL #5

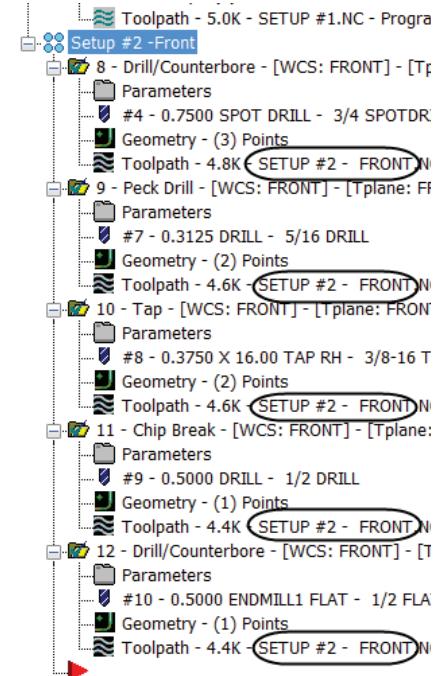
RENAME NC FILE

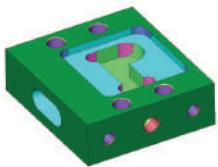


- When the Enter new NC name dialog box appears enter "Setup #2 - Front".



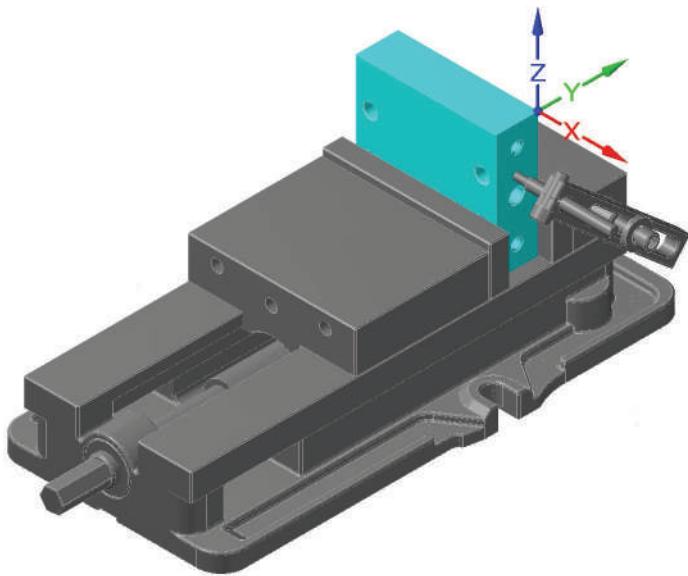
- Select the **OK** button to apply the changed NC name to all of the operations in the second setup.
- You should see is **Setup #2 - Front.NC** in all of the operations in the second setup.





TOOLPATH CREATION - SETUP 3

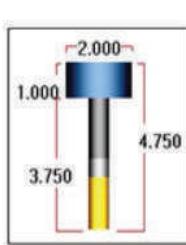
SUGGESTED FIXTURE:



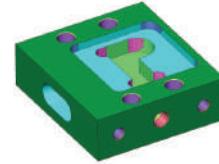
NOTE: The part is now flipped and we will machine the part from the left side.

SETUP SHEET:

TOOL LIST



TYPE: Endmill1 Flat	FLUTE LENGTH: 1.5
DIA OFFSET: 2	OVERALL LENGTH: 3.75
HOLDER: DEFAULT HOLDER	CORNER RAD: 0.0
NUMBER: 2	# OF FLUTES: 4
LENGTH OFFSET: 2	
#2 - 0.6250 ENDMILL1 FLAT - 5/8 FLAT ENDMILL	



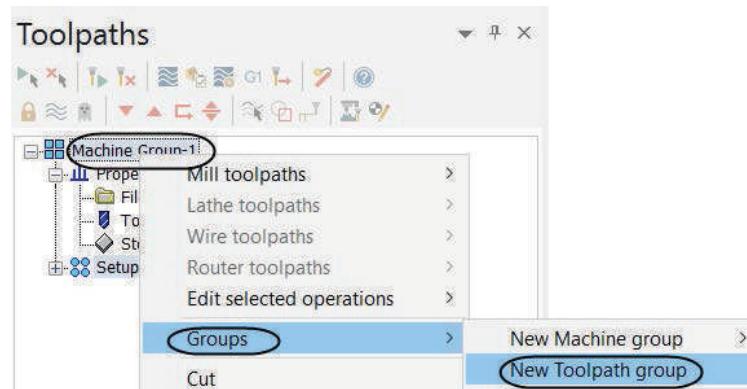
STEP 21: CREATING AND RENAMING TOOLPATH GROUPS

To machine the part in different setups, we will need to have separate programs. To be able to post the operations separate of each setup, we will create them under different toolpath groups with different NC names.

21.1 Create Toolpath Group #3 (Setup #3 - Left)

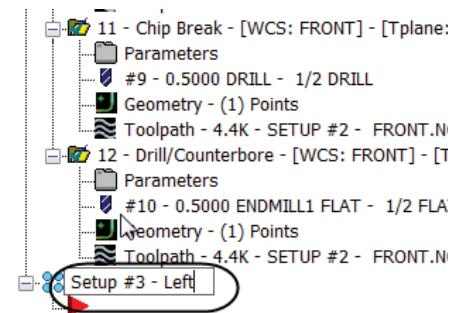
- ♦ Right mouse click on the **Machine Group-1**.
- ♦ Select **Groups** and then **New Toolpath group** as shown in [Figure: 21.1.1](#).

Figure: 21.1.1

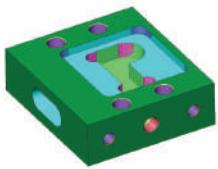


- ♦ Rename the toolpath group "**Setup #3 - Left**" as shown in [Figure: 21.1.2](#).

Figure: 21.1.2



- ♦ Ensure the arrow is below "**Setup #3 - Left**."



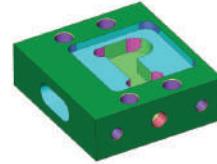
STEP 22: SET THE WCS TO LEFT

The **Work coordinate system (WCS)** is the active coordinate system in use by Mastercam at any given time. The WCS contains the orientation of the X-Y-Z axes plus the location of the zero point (the origin). This tells Mastercam how your part is positioned or orientated in the machine.

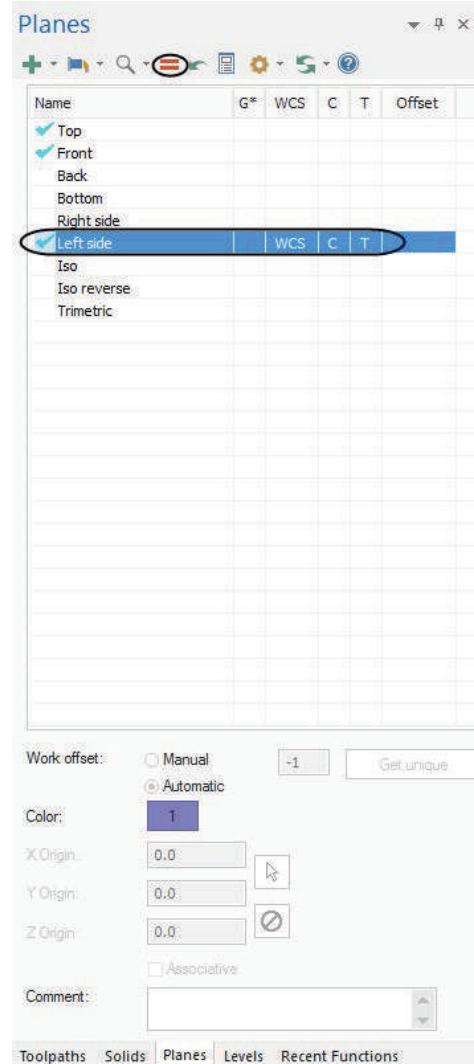
- Select the **Planes** tab located in the bottom left corner.



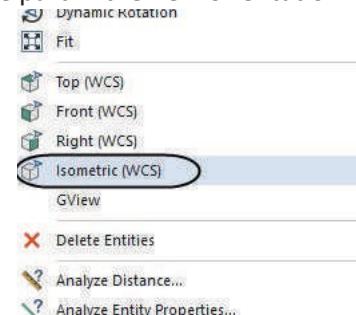
TUTORIAL #5 SET THE WCS TO LEFT



- ♦ Select the **Left side** plane and click on the **equal** sign to set the **WCS**, **Construction Plane** and **Tool Plane** to the **Left side** as shown.

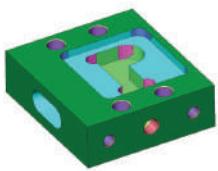


- ♦ Right mouse click in the graphics window and select the **Isometric** view to see the part in the new orientation.



- ♦ Press **F9** on your keyboard to view the coordinate axes.

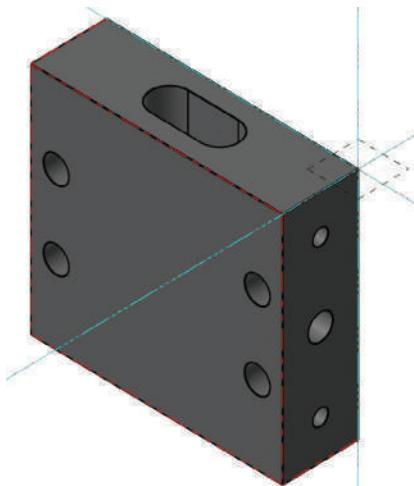
NOTE: The color of the coordinate axes remains the same because it is the same origin.



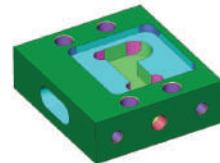
SET THE WCS TO LEFT

TUTORIAL #5

- ◆ Your part will appear as shown up to this point.



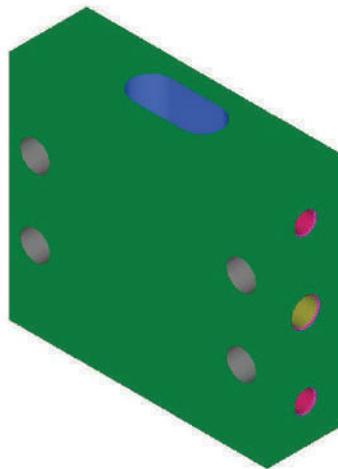
- ◆ Press **F9** to remove the axes display.



STEP 23: MACHINE THE SLOT

Slot Mill efficiently machines obround slots. This toolpath automatically calculates plunge, entry, and exit points appropriate for the slots.

Toolpath Preview:



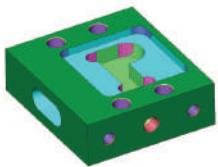
TOOLPATHS

- From the **2D** group, click on the **Expand gallery** arrow and select **Slot Mill**.

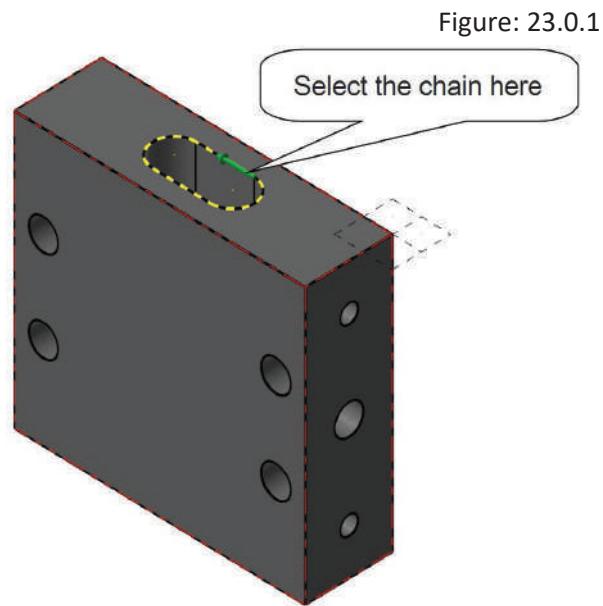


- When the **Chaining** dialog box appears, make sure that the **C-plane** is enabled as shown.





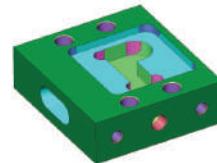
- ♦ Select the slot as shown in [Figure: 23.0.1](#).



- ♦ Select the **OK** button in the chaining dialog box to continue.
- ♦ In the **Toolpath Type** page, the **Slot Mill** toolpath will be selected as shown in [Figure: 23.0.2](#).

Figure: 23.0.2





23.1 Select the 5/8" Flat Endmill from the Tool List

- ♦ Select **Tool** from the **Tree View** list.
- ♦ Pick the **5/8" Flat Endmill** tool from the list.
- ♦ Make the necessary changes to the **Tool** page as shown in [Figure: 23.1.1](#).

Figure: 23.1.1

#	Assembly Name	Tool Name	Holder Name	Dia.
1	-	3/4 FLAT ...	-	0.75
2	-	5/8 FLAT ...	-	0.625
3	-	3/8 FLAT ...	-	0.375
4	-	3/4 SPOT...	-	0.75
5	-	33/64 DRILL	-	0.5156
6	-	9/16-18 T...	-	0.5625
7	-	5/16 DRILL	-	0.3125
8	-	3/8-16 TA...	-	0.375
9	-	1/2 DRILL	-	0.5
10	-	1/2 FLAT ...	-	0.5

Right-click for options

 Filter Active

Tool diameter:

Corner radius:

Tool name:

Tool #: Length offset:

Head #: Diameter offset:

RCTF Spindle direction:

Feed rate: Spindle speed:

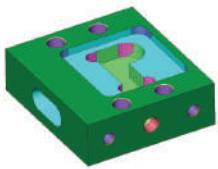
FPT: SFM:

Plunge rate: Retract rate:

Force tool change Rapid Retract

Comment

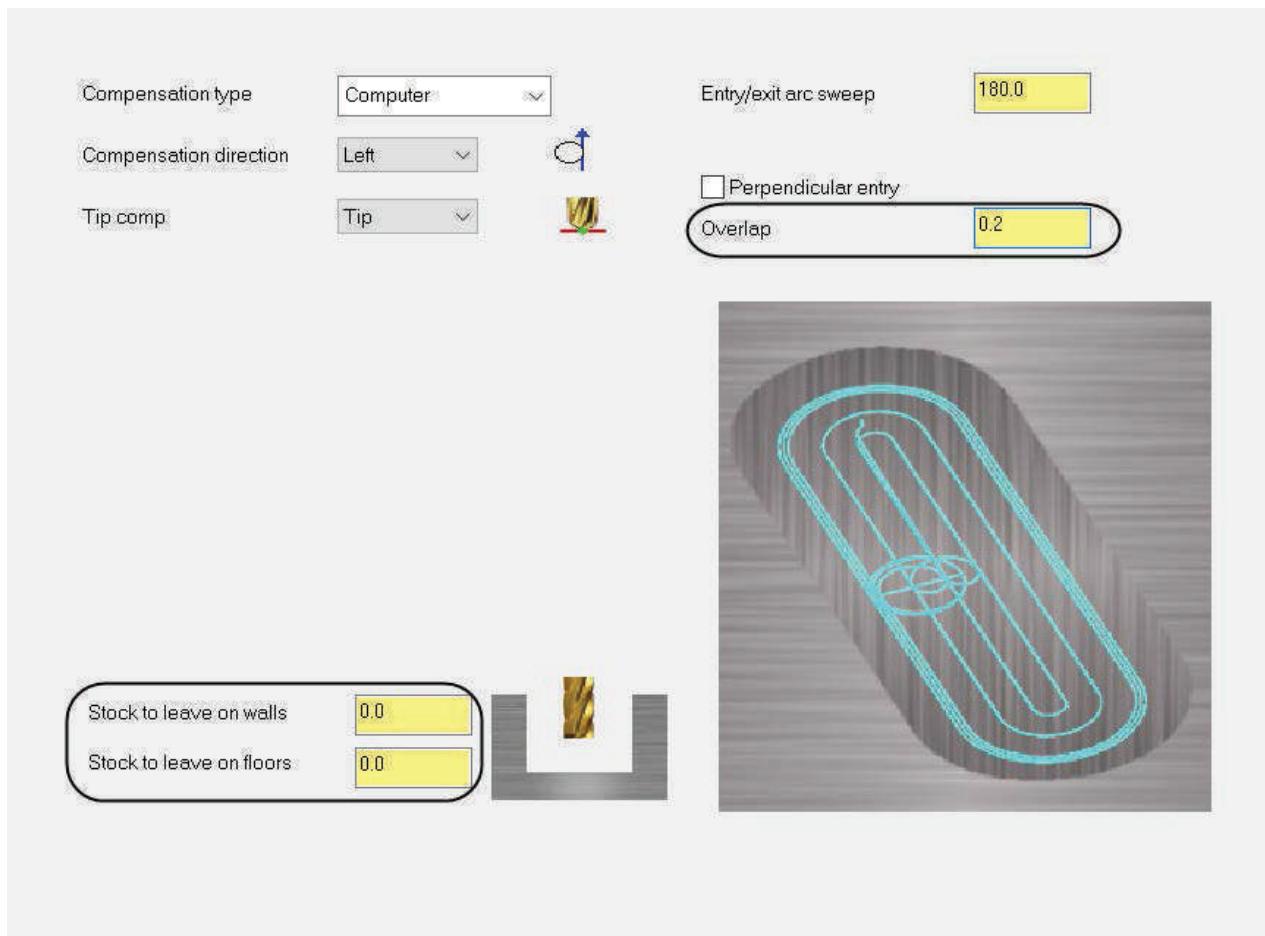
Cut the slot on the left side of the part.



23.2 Set the Cut Parameters

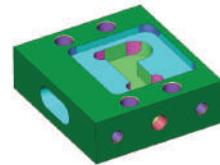
- ♦ Select **Cut Parameters** and add an overlap of **0.2** as shown in [Figure: 23.2.1](#).

Figure: 23.2.1



Entry/exit arc sweep sets the included angle of each entry and exit arc. If this value is set to less than 180 degrees, a line will be created.

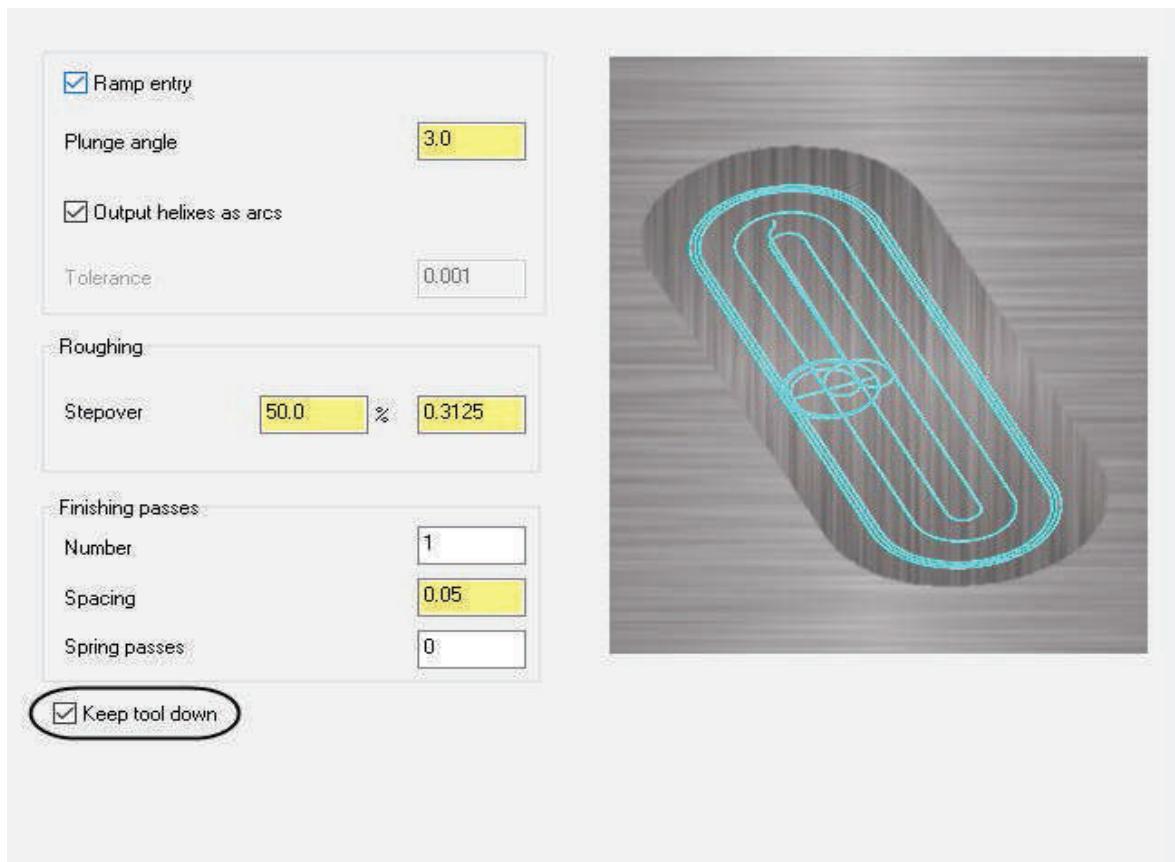
Overlap sets how far the tool goes past the end of the toolpath before exiting for a cleaner finish.

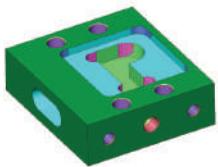


23.3 Set the Rough/Finish Parameters

- ◆ Choose **Rough/Finish** and ensure your settings appear as shown in [Figure: 23.3.1](#).

Figure: 23.3.1



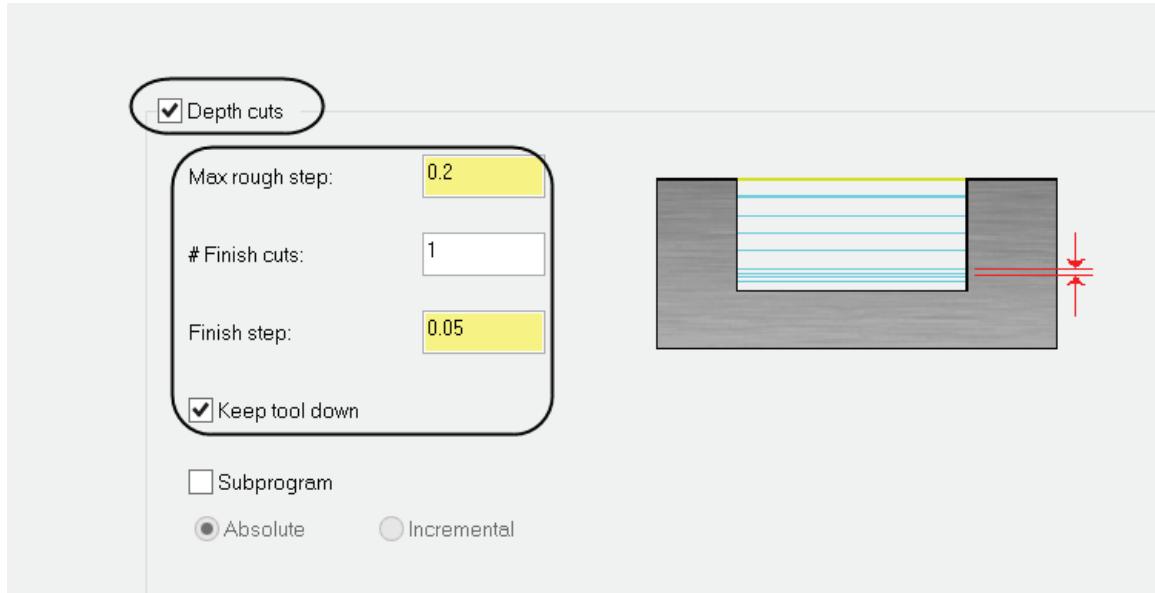


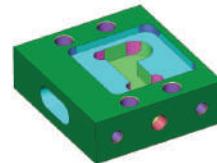
23.4 Set the Depth Cut Parameters

- ♦ Select **Depth Cuts** and input a **Max rough step** of **0.2** as shown in [Figure: 23.4.1](#).
- ♦ Enter a **# of Finish Cuts** value of **1** and a **Finish step** of **0.05**.

NOTE: These settings instruct Mastercam to rough the slot leaving 0.05" and then create one lighter cut removing 0.05" of material for a finish pass.

Figure: 23.4.1

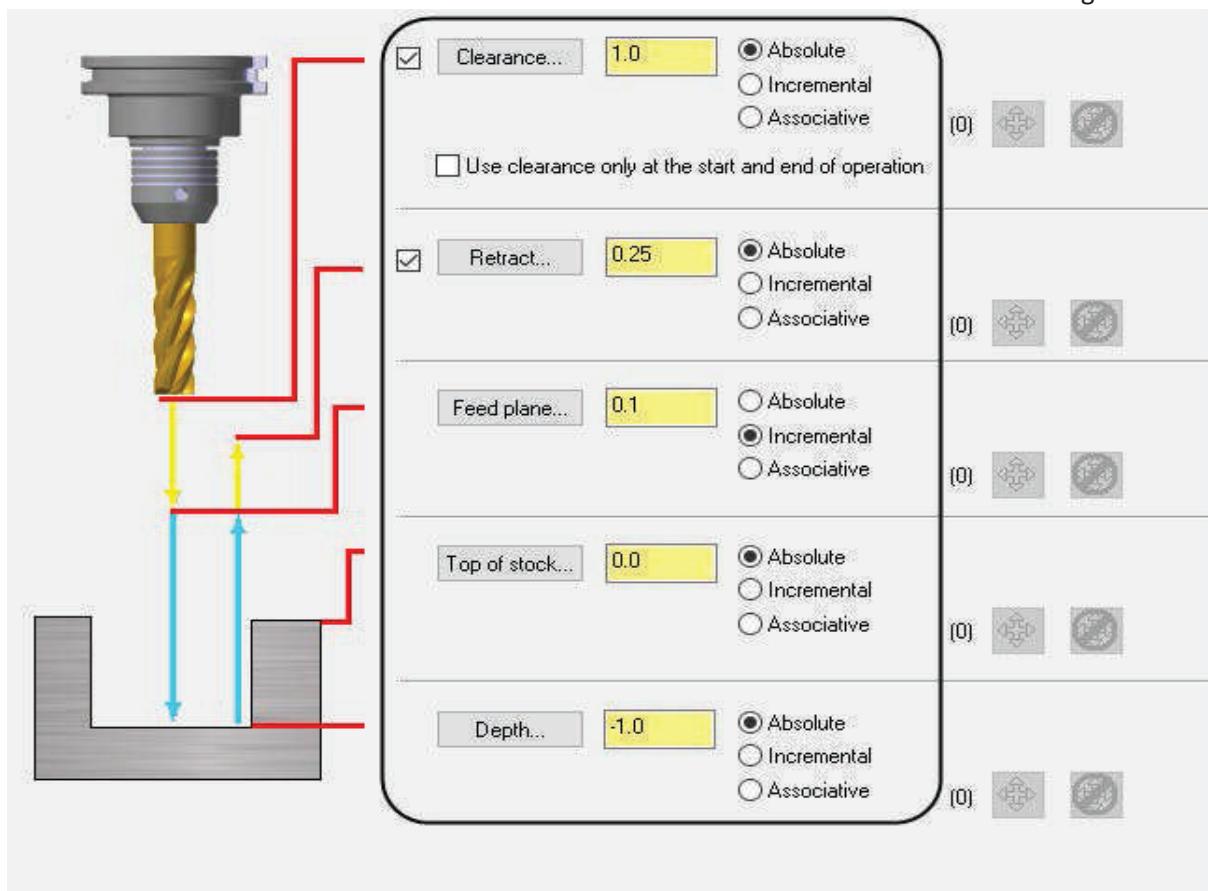




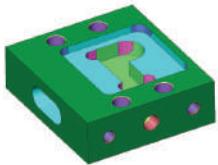
23.5 Set the Linking Parameters

- ♦ Select **Linking Parameters**, enable **Clearance**, set it to **1.0**, and input a **Depth** of **-1.0** as shown in [Figure: 23.5.1](#).

Figure: 23.5.1

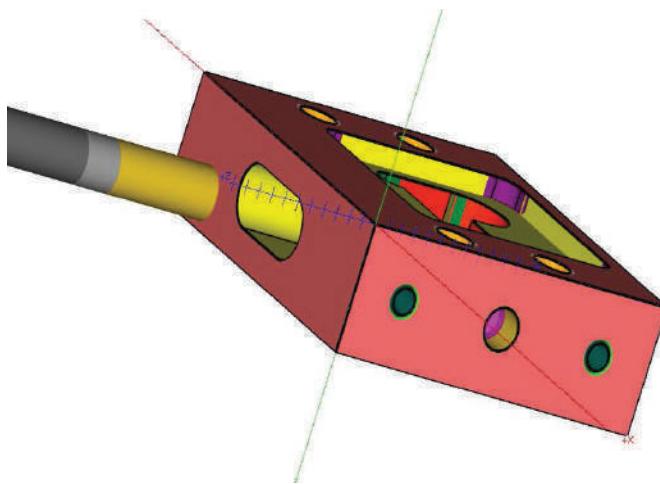


- ♦ Select the **OK** button to exit the **2D Toolpaths - Slot Mill** parameters and generate the toolpath.



23.6 Backplot and Verify

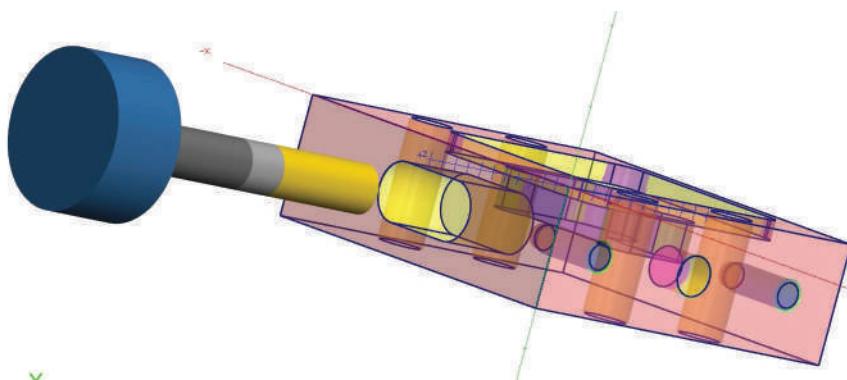
- ♦ Select the **Toolpaths Manager**.
- ♦ To **Backplot** and **Verify** the toolpaths, see [page 447](#) and [page 448](#).
- ♦ Ensure all operations are selected. If they are not use the **Select all operations** icon from the **Toolpaths Manager**.
- ♦ To rotate the part, pick a point in the center of the part with the mouse wheel and slowly move the cursor in the graphics window.



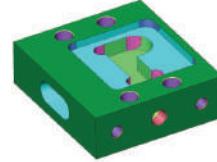
- ♦ Release the mouse wheel when the part is in the desired position.
- ♦ To see the stock translucent, in the **Visibility** area, click on the check mark in front of the stock. It should look as shown.



- ♦ The part should look as shown.



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

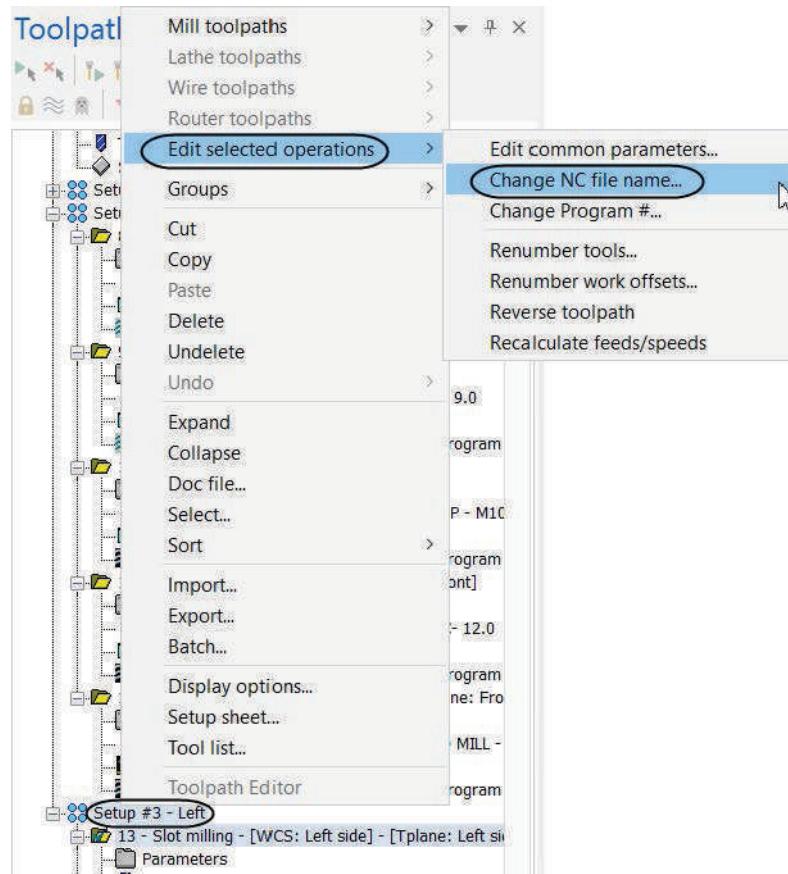


STEP 24: RENAME THE NC FILE

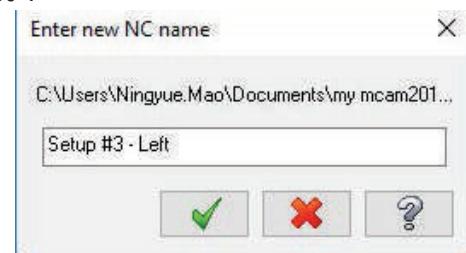
The Slot milling operation in "Setup #3 - Left" kept the NC name from "Setup #2 - Front". We need to rename this operation so it will create a separate program for this setup.

- ♦ Select "Setup #3 - Left", right click on the group (make sure all the operations in setup #3 are selected), choose the option **Edit selected operations** and then pick **Change NC file name** as shown in [Figure: 24.0.1](#).

Figure: 24.0.1

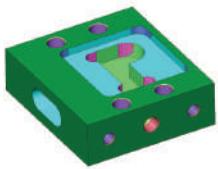


- ♦ When the **Enter new NC name** dialog box appears enter "**Setup #3 - Left**".



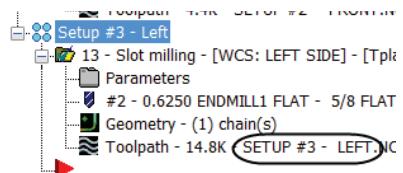
- ♦ Select the **OK** button to apply the changed **NC name** to all the operations in the third setup.





- You should see **Setup #3 - Left.NC** in the last operation in the third setup as shown in [Figure: 24.0.2](#).

Figure: 24.0.2



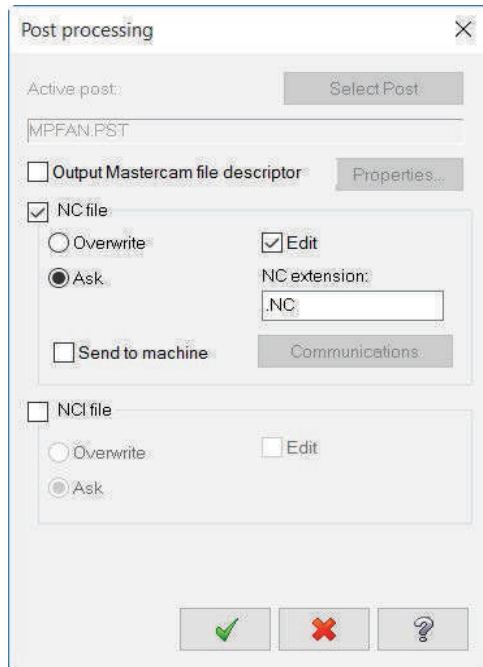
STEP 25: POST THE FILE

- Ensure all operations are selected. If they are not use the button **Select all operations** from the **Toolpaths Manager**.

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

- In the **Post processing** window make the necessary changes as shown in [Figure: 25.0.1](#).

Figure: 25.0.1



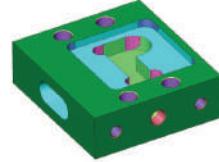
NC File enabled allows you to keep the NC file and to assign the same name as the MCAM file.

Edit enabled allows you to automatically launch the default editor.

- Select the **OK** button to continue.
- Save your file and name it **Setup #1.NC**.
- Save your file and name it **Setup #2 - Front.NC**.
- Save your file and name it **Setup #3 - Left.NC**.

TUTORIAL #5

SAVE THE UPDATED MCAM FILE



- ♦ A window with **Mastercam Code Expert** will be launched and the NC programs will appear as shown in [Figure: 25.0.2](#).

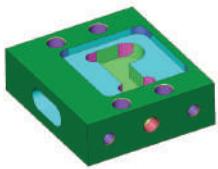
Figure: 25.0.2

The screenshot shows the 'Mastercam Code Expert' application window. The title bar reads 'Setup #3 - Left.NC - Mastercam 2018 Code Expert'. The menu bar includes 'File', 'Home', 'View', 'Editor', 'NC Functions', 'Send File', 'Receive', 'Syncs', 'Tools', 'Multi-Stream', 'NC Configuration', and 'Utilities'. The main area displays an NC program for a left side slot. The code consists of several G-code commands, including G0, G1, G2, G3, G90, G91, G92, G93, G94, G95, G96, G97, G98, G99, M0, M1, M2, M3, M4, M5, M6, M7, M8, M9, M10, M11, M12, M13, M14, M15, M16, M17, M18, M19, M20, M21, M22, M23, M24, M25, M26, M27, M28, M29, M30, M31, M32, M33, M34, M35, M36, M37, M38, M39, M40, M41, M42, M43, M44, M45, and M46. The code is color-coded to highlight different command types. The status bar at the bottom shows 'Ready', '5.76KB', '100%', and zoom controls.

```
Setup #1.NC Setup #2 - Front.NC Setup #3 - Left.NC
1. G0
2. 00005(RETUP #3 - LEFT)
3. (DATE-DD-MM-YY - 31-01-17 TIME=HH:MM - 13:09)
4. (NCX FILE - \THSSERVER06\BOOKDEV\PRODUCTION\BOOKS\2018\TRAINING TUTORIALS\TT MILL ESSENTIALS\MCAM FILES\TUTORIAL TOOLPATH INCH\TUTORIAL #5.MCAM)
5. (NC FILE - C:\USERS\NINGYOU\DESKTOP\T5\SETUP #3 - LEFT.NC)
6. (MATERIAL - ALUMINUM INCH - 2024)
7. ( T2 | 5/8 FLAT ENDMILL | H2 )
8. N100 G20
9. N110 G90 G17 G40 G49 G50 G90
10. ( CUI THE SLOT ON THE LEFT SIDE OF THE PART. )
11. M120 T2 M6
12. N130 G90 G54 X-3.0071 Y-.7603 R0.8655 M3
13. N140 G45 H2 Z1.
14. N150 Z1.
15. N160 G1 ZD. F6.37
16. N170 X-3.0037 Y-.7619 Z-.0002
17. N180 X-3. Y-.7625 Z-.0004
18. N190 X-2. Z-.0528
19. N200 G3 X-1.9875 Y-.75 Z-.0539 I0. J.0125
20. N210 X-2. Y-.7375 Z-.0549 I-.0125 J0.
21. N220 G1 X-3. Z-.1073
22. N230 G3 X-3.0125 Y-.75 Z-.1083 I0. J-.0125
23. N240 X-3. Y-.7825 Z-.1093 I.0125 J0.
24. N250 G1 X-2. Z-.1617
25. N260 G3 X-1.9875 Y-.75 Z-.1628 I0. J.0125
26. N270 X-2. Y-.7375 Z-.1638 I-.0125 J0.
27. N280 G1 X-2.5 Z-.19
28. N290 X-3.
29. N300 G3 X-3.0125 Y-.75 I0. J-.0125
30. N310 X-3. Y-.7625 I.0125 J0.
31. N320 G1 X-2.
32. N330 G3 X-1.9875 Y-.75 I0. J.0125
33. N340 X-2. Y-.7375 I-.0125 J0.
34. N350 G1 X-2.5
35. N360 Y-.75
36. N370 G3 X-2.4688 Y-.7188 I0. J.0312
37. N380 X-2.5 Y-.6875 I-.0312 J0.
38. N390 G1 X-3.
39. N400 G3 X-3.0625 Y-.75 I0. J-.0625
40. N410 X-3. Y-.8125 I.0625 J0.
41. N420 G1 X-2.
42. N430 G3 X-1.9875 Y-.75 I0. J.0625
43. N440 X-1. Y-.6875 I-.0625 J0.
44. N450 G1 X-2.7
45. N460 G3 X-2.7313 Y-.7188 I0. J-.0313
46. N470 X-2.7 Y-.75 I.0313 J0.
```

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

STEP 26: SAVE THE UPDATED MCAM FILE



REVIEW EXERCISE - STUDENT PRACTICE

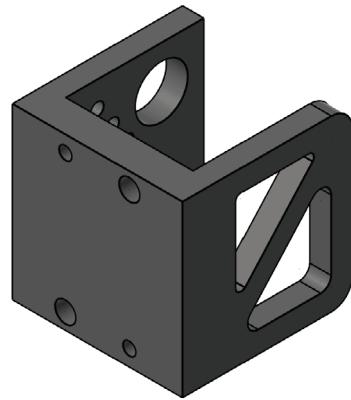
IMPORTING THE GEOMETRY FOR TUTORIAL #5 EXERCISE

Download the file from emastercam.com.

- ◆ Save the file to a known location.

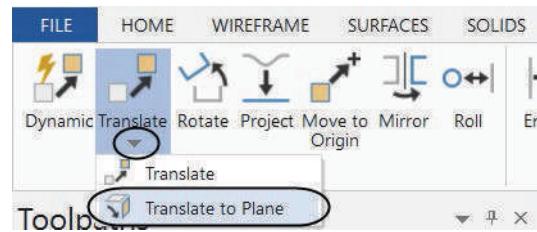
Use File Open.

- ◆ Set the extension to SolidWorks (*.sldprt, *.sldasm).
- ◆ Select Tutorial #5 Exercise.SLDPRT.
- ◆ In Options enable **Solids**, **Edge curves** and **Use System Color for imported Solids**.
- ◆ Open the file.
- ◆ The geometry should look as shown.



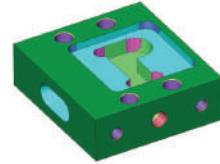
Use TRANSFORM/Translate to Plane to rotate the part for machining.

- ◆ Make a Window around the entire part.
- ◆ **TRANSFORM**.
- ◆ **Translate to plane**.



TUTORIAL #5

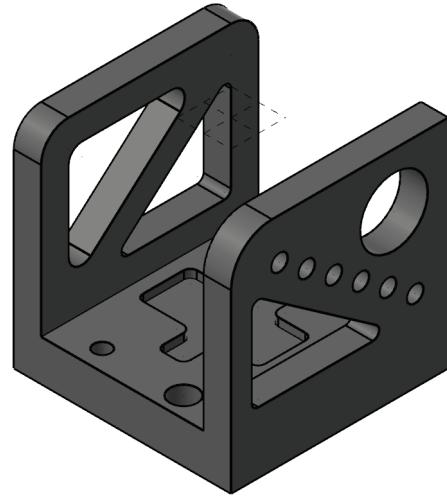
IMPORTING THE GEOMETRY FOR TUTORIAL #5 EXERCISE



- ♦ In the **Translate To Plane** panel, set the parameters to make the **Back** view **Top** view as shown.

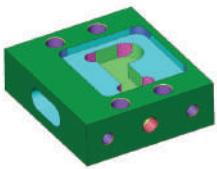


- ♦ Clear colors and the part should look as shown.



Save the file.

- ♦ Save the file as Your Name_5 Exercise.



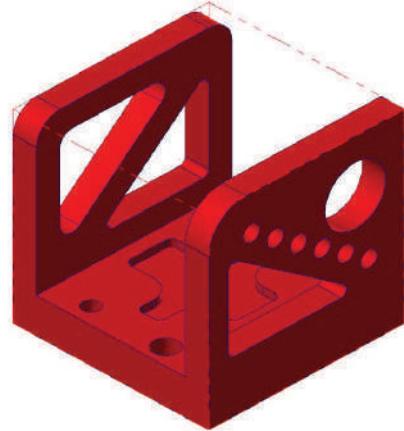
CREATE THE TOOLPATHS FOR TUTORIAL #5 EXERCISE

Create the Toolpaths for the Tutorial #5 Exercise as per the instructions below.

Select the Mill Default.

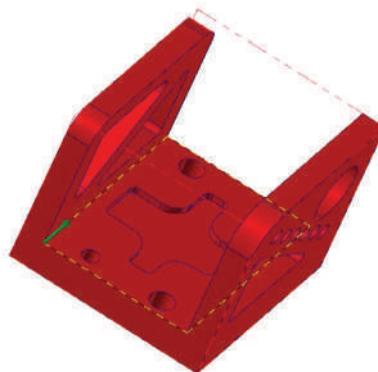
Set the machine properties including the stock setup.

- ♦ Set the **Back** plane as the **WCS, C, Tplane**.
- ♦ The stock will appear as shown.



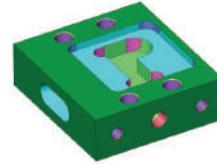
Remove the material in the center of the part using the 2D HS Dynamic Mill.

- ♦ As **Machining regions** select the bottom rectangle (enable **C-plane**).
- ♦ Enable **From outside**.

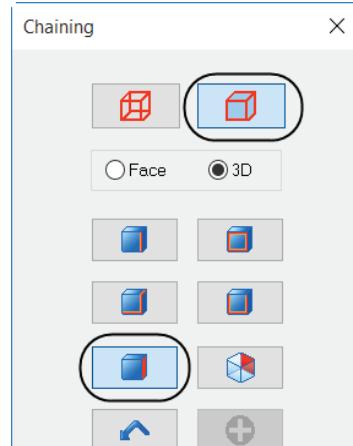


TUTORIAL #5

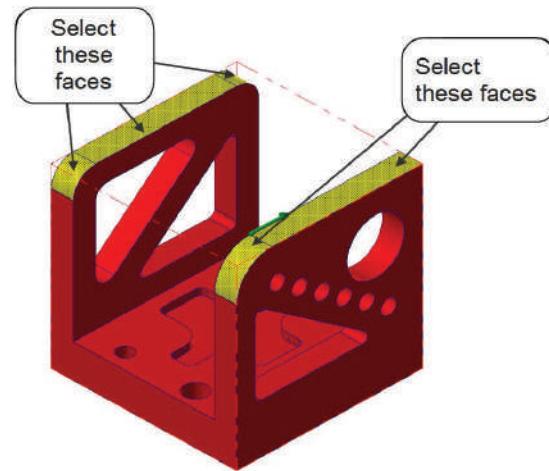
CREATE THE TOOLPATHS FOR TUTORIAL #5 EXERCISE



- In **Avoidance regions**, switch in the **Chaining** dialog to **Solids** selection and enable only the **Face** button as shown.

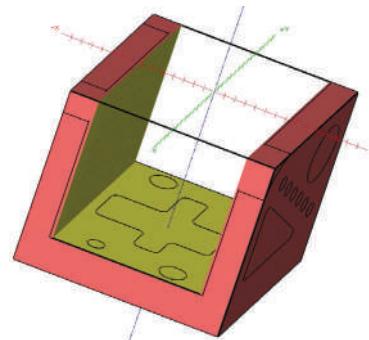


- Select the five faces shown.

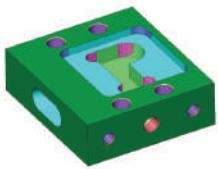


- Use a **1" Flat Endmill**.

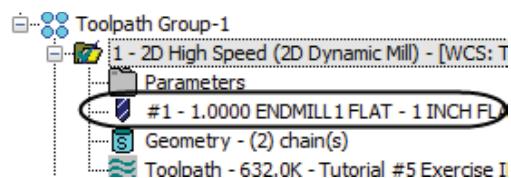
- Ensure the **Cutting Method** is set to **Climb**.
- Approach distance** = 1.0.
- Stepover** = 25%.
- Min toolpath radius** = 10%.
- Gap size % of tool diameter** = 100.
- Stock to leave on walls** = 0.05.
- Stock to leave on floors** = 0.0.
- In **Depth Cuts** set the **Max rough step** = 1.0.
- Choose **Entry method** as **Helix**.
- No **Break Through**.
- Set the depth according to the part.



NOTE: Some of these toolpaths will require you to change the default tool's settings. Refer to page 695 for more information on how to do this.

**Change Tool Length.**

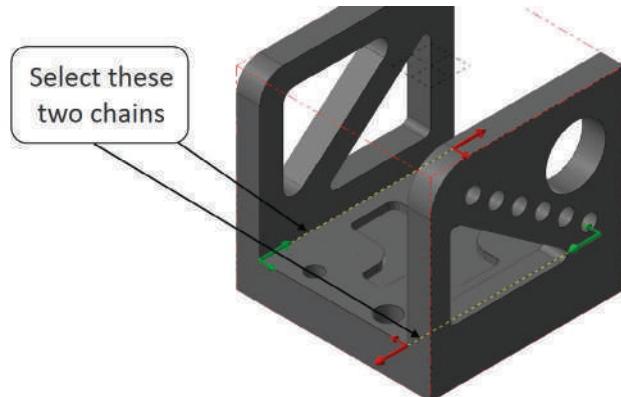
- ♦ Select the tool under Operation #1.
- ♦ **Overall length = 6".**
- ♦ **Cutting length = 4".**
- ♦ Select Finish.

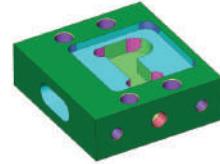
**Use Contour toolpath to finish the vertical faces.**

- ♦ Stay in **Solid** mode in the **Chaining** dialog box and enable only the **Edge** button as shown.

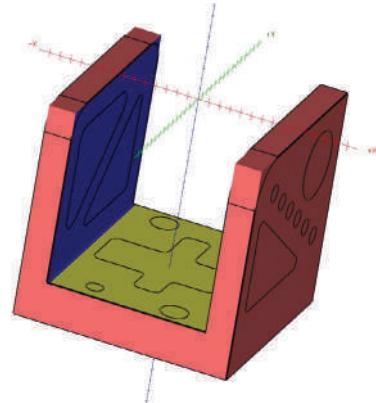


- ♦ Select the two lines in the clockwise direction.



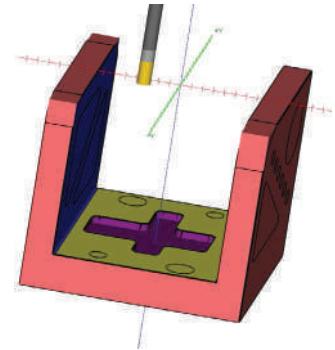


- ◆ Use a **3/4" Flat Endmill**.
- ◆ Increase the tool's **Cutting length** to **3.5"**.
- ◆ **Overall length = 4"**.
- ◆ Set the **Compensation direction** to **Right**.
- ◆ In the **Depth cuts** set the **Max rough step** to **0.375**.
- ◆ In the **Lead In/Out**, disable **Entry** and enable **Adjust start/end of contour**, setting the **Length** to **Extend 100%**.
- ◆ Set the **Linking parameters** according to the part.



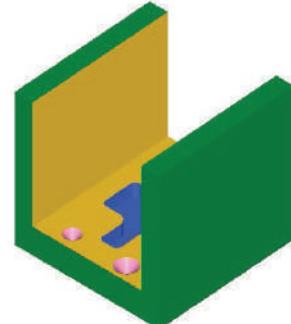
Machine the Pocket.

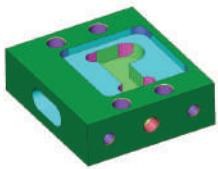
- ◆ Use a **1/4" Flat Endmill**
- ◆ Edit the tool's **Cutting length** to **4"** and the **Overall length** to **4.5"**.
- ◆ Ensure the **Machining direction** is set to **Climb**.
- ◆ **Stock to leave on walls/floors = 0.0**.
- ◆ Enable **Roughing** and select a desired **Cutting method**.
- ◆ Choose an **Entry motion**.
- ◆ Set the **Finishing** parameters.
- ◆ Enable **Lead In/Out**.
- ◆ Disable **Depth Cuts** and **Break Through**.
- ◆ Set the **Top of stock** and the **Depth** according to the part.



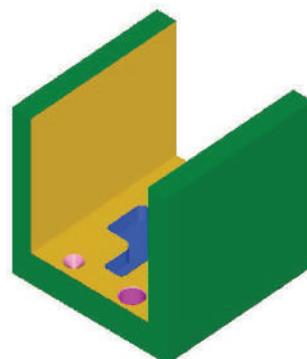
Spot Drill the holes.

- ◆ Use a **3/4" Spot Drill**.
- ◆ Edit the tool **Cutting length** to **3"** and the **Overall length** to **4"**.
- ◆ Set the **Drill cycle** to **Drill/Counterbore**.
- ◆ Set the **Top of stock** according to the material.
- ◆ Set the **Depth** so that it leaves a **0.05" Chamfer** on all of the holes.
- ◆ Ensure that you change the **Depth** for the two larger holes.

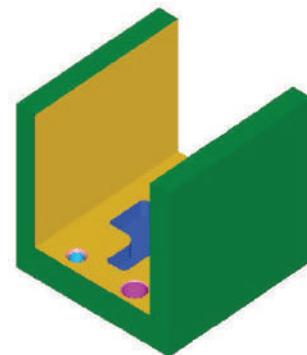


**Drill the 1/2" Holes.**

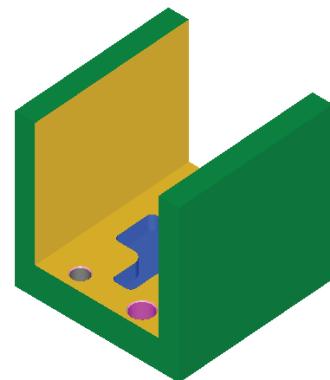
- ◆ Use a **1/2" Drill**.
- ◆ Edit the tool **Cutting length** to **3"** and the **Overall length** to **5"**.
- ◆ Set the **Drill cycle** to **Drill/Counterbore**.
- ◆ Set the **Top of stock** according to the material.
- ◆ Set the **Depth** and include a **Tip Comp**.

**Drill the 3/8 Tapped Holes.**

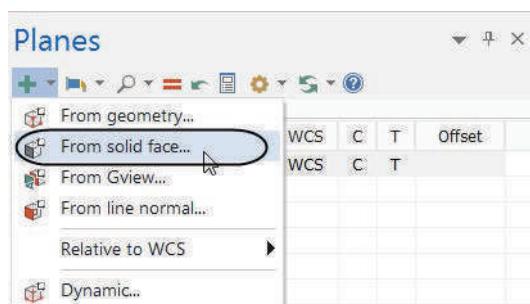
- ◆ Use a **5/16" Drill**.
- ◆ Edit the tool **Cutting length** to **3"** and the **Overall length** to **5"**.
- ◆ Set the **Drill cycle** to **Drill/Counterbore**.
- ◆ Set the **Top of stock** according to the material.
- ◆ Set the **Depth** and include a **Tip Comp**.

**Tap the 3/8 - 16 Holes.**

- ◆ Use a **3/8 - 16 Tap RH**.
- ◆ Edit the tool **Cutting length** to **3"** and the **Overall length** to **5"**.
- ◆ Set the **Drill Cycle** to **Tap**.
- ◆ Set the **Top of stock** according to the material.
- ◆ Set the **Depth** and include a **Tip Comp**.

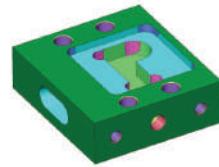
**Set the WCS to a new plane defined by existing geometry.**

- ◆ In the **Planes** tab, click on the **+** (plus sign) to create a new plane.
- ◆ From the list, select **From Solid Face** as shown.



TUTORIAL #5

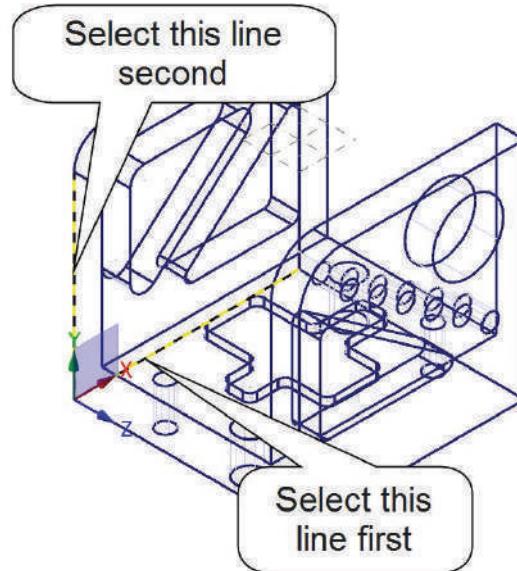
CREATE THE TOOLPATHS FOR TUTORIAL #5 EXERCISE



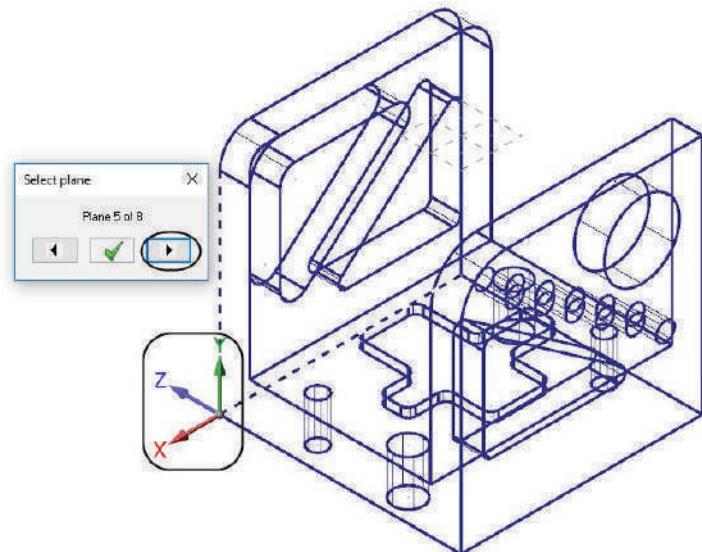
- ♦ Press **Alt + S** to see the solid in an wireframe mode.

NOTE: You will select two lines that can define the plane.

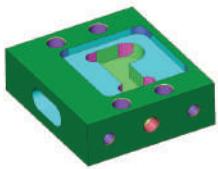
- ♦ [Select an entity]: Select the lines in the order as shown.



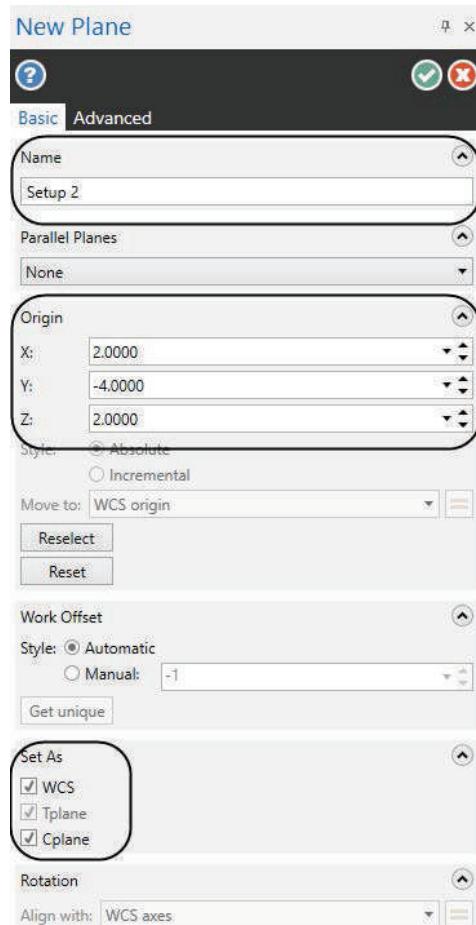
- ♦ To change the axes orientation from the **Select plane**, click on the **Next plane** button until the orientation looks as shown.



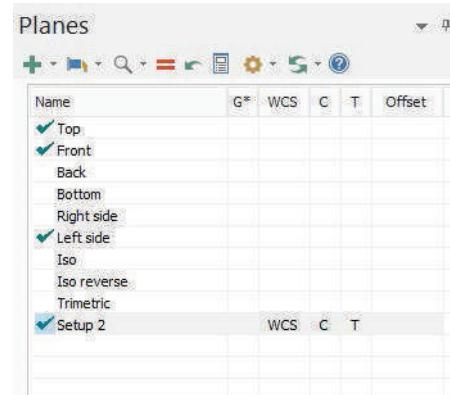
- ♦ From **Select plane** click on the **OK** button to continue.



- In the **New Plane** panel, change the plane name to **Setup 2** and enable **Set as WCS**.



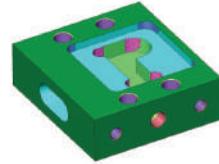
- The **Planes Manager** panel should look as shown.



- Right mouse click in the graphics window and select **Isometric**.
- Press **F9** to see where the axes and the Origin is. The origin should be at the top of the part lower right corner.

TUTORIAL #5

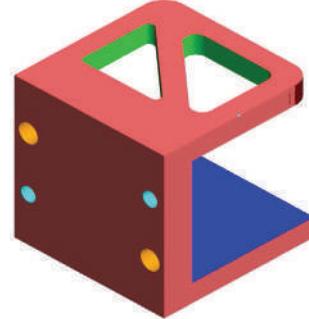
CREATE THE TOOLPATHS FOR TUTORIAL #5 EXERCISE



Create a new Toolpath Group and rename the NC File name for the last toolpath group.

- ◆ **Pocket** the two cut through triangles.
- ◆ Use a **1/4" Flat Endmill**.
- ◆ **Stock to leave on walls = 0.0**.
- ◆ You may wish to use the original tool lengths from this point on as we no longer need to machine very deep pockets.

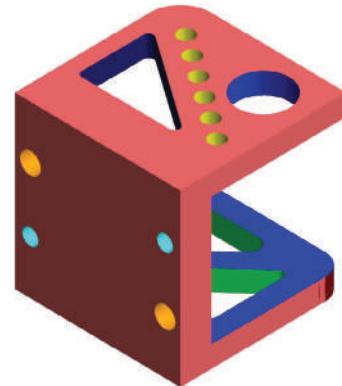
- ◆ Create a **Contour** toolpath around the two fillets.
- ◆ Use the **1/4" Flat Endmill**.
- ◆ Set the **Depth** according to the part.



Set the WCS to Right and input an Origin Z value of 2.0".

Create a new Toolpath Group and rename the NC File name for the last toolpath group.

- ◆ **Pocket** the cut through triangle.
- ◆ Use a **1/4" Flat Endmill**.
- ◆ **Stock to leave on walls/floors = 0.0**.



- ◆ **Drill** the 6 Holes.
- ◆ Use the **5/16" Drill**.
- ◆ Set the **Drill Cycle** to **Drill/Counterbore**.

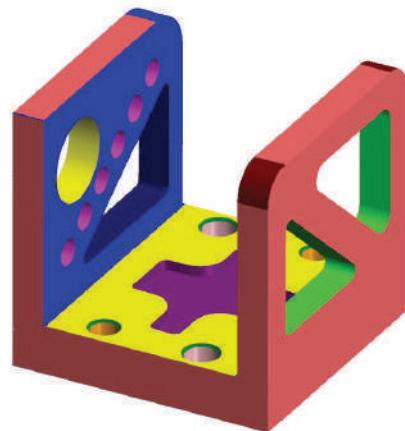
- ◆ **Tap** the 6 Holes.
- ◆ Use the **3/8 - 16 Tap**.
- ◆ Set the **Drill Cycle** to **Tap**.

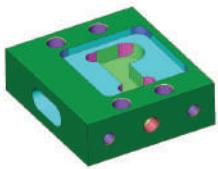
- ◆ **Circle Mill** the **Ø1.25"** Hole.
- ◆ Use a **1/2" Flat Endmill**.
- ◆ **Stock to leave on walls/floors = 0.0**.
- ◆ Enable **Roughing**.

- ◆ Enable **Finishing** and set the **Finish Passes** to **2** with a spacing of **0.02"**.
- ◆ Set the depth according to the part.

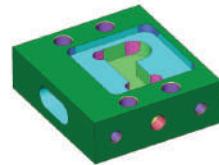
- ◆ Create a **Contour** operation to remove the material around the fillet.
- ◆ Use a **1/2" Flat Endmill**.

Once complete your part will appear as shown.



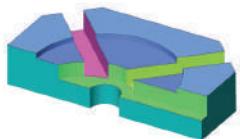


NOTES:



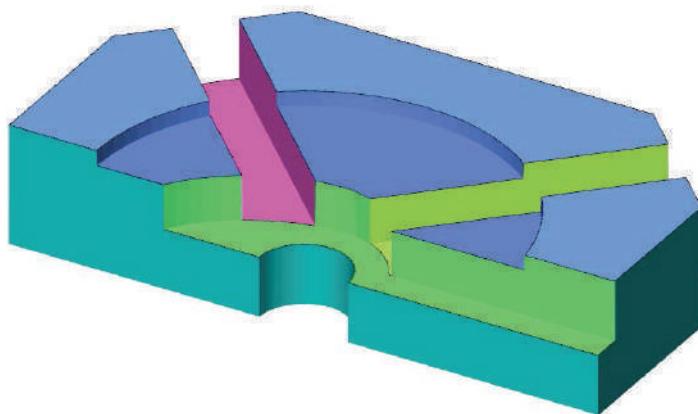
TUTORIAL #5 QUIZ

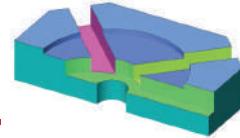
- ◆ What settings do you need to use to remachine a pocket using a High Speed Area Mill Toolpath?
 - ◆ What is the use of the WCS in Mastercam?
 - ◆ After creating a new toolpath group why do you rename the NC file?



TUTORIAL #6

TUTORIAL #6



**OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:****Import the 2D CAD Model and prepare it to generate Toolpaths from:**

- ◆ The student will open the SolidWorks file in Mastercam.
- ◆ The student will move the geometry to set the origin at the top of the part.

Create the necessary Toolpaths to machine the part:

- ◆ The student will set up the stock size to be used and the clamping method used.
- ◆ A 2D High Speed Dynamic Mill toolpath will be created to remove the material outside of the step.
- ◆ A 2D High Speed Blend Mill toolpath will be created to machine the semi arc shape pocket.
- ◆ A 2D High Speed Area Mill toolpath will be created to machine the second step.

Backplot and Verify the file:

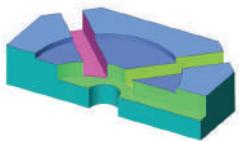
- ◆ The Backplot will be used to simulate a step-by-step process of the tool's movements.
- ◆ The Verify will be used to watch a tool machine the part out of a solid model.

Post Process the file to generate the G-code:

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



This tutorial takes approximately one hour to complete.



GEOMETRY CREATION

STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

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

STEP 2: IMPORTING THE SOLIDWORKS FILE GEOMETRY

Mastercam lets you read (import) a variety of CAD file types into the Mastercam database. You can also write (export) Mastercam files to a variety of different file formats.

To import a SolidWorks file in Mastercam, you have to use the Open function and then select SolidWorks file from the list of file types.

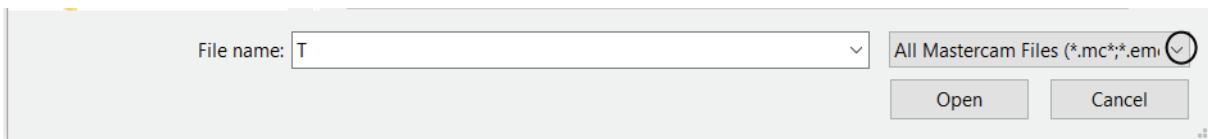
Download the files from www.emastercam.com/trainingfiles.

Save the file at a preferred location.

FILE

- ♦ **Open.**

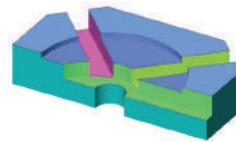
- ♦ In the file name extension click on the drop down arrow as shown.



- ♦ From the list of file types, select **SolidWorks Files (*.sldprt;*.sldasm)** as shown.

Mastercam Files (*.mcam)
Mastercam X Files (*.mcx*)
Mastercam Edu X Files (*.emcx*)
All Mastercam Files (*.mc*;*.emc*)
IGES Files (*.igs;*.iges)
AutoCAD Files (*.dwg;*.dxf;*.dwf;*.dwfx)
Parasolid Files (*.x_t;*.x_b;*.xmt_txt)
ProE/Creo Files (*.prt;*.asm;*.prt.*;*.asm.*)
ACIS Kernel SAT Files (*.sat;*.sab)
STEP Files (*.stp;*.step)
VDA Files (*.vda)
Rhino 3D Files (*.3dm)
SOLIDWORKS Files (*.sldprt;*.sldasm;*.sladdrw)
Solid Edge Files (*.par;*.psm;*.asm)
Autodesk Inventor Files (*.ipt;*.iam;*.idw)

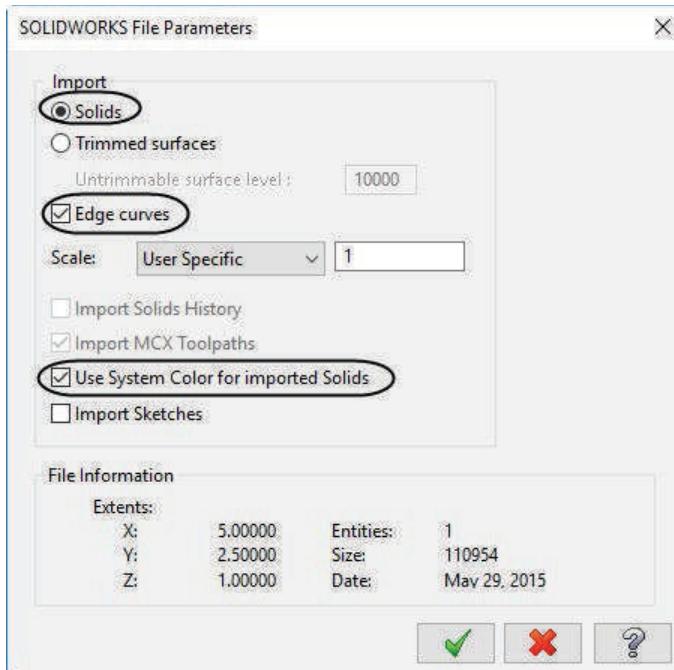
- ♦ Find and select **TUTORIAL #6.sldprt**.



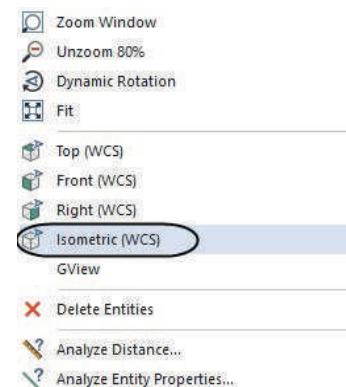
- ♦ Click on the **Options** button.



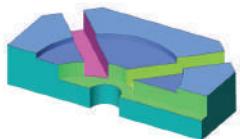
- ♦ Leave the **Solids** enabled to import the file as a solid and enable **Edge curves** for Mastercam to automatically create curves at the edges of the solid. To better see the curves, also enable **Use System Color for imported Solids** as shown.



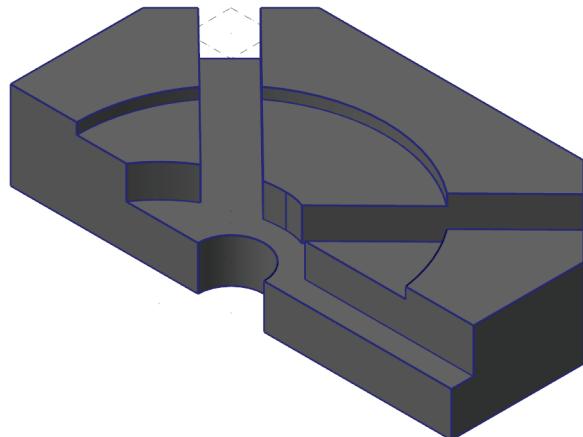
- ♦ Select the **OK** button to exit the **SolidWorks File Parameters** dialog box.
- ♦ **Open** the file.
- ♦ Right mouse click in the graphics window and select the **Isometric** view.



- ♦ Press **Alt + F1** to fit the geometry in the graphics window.



- The geometry should look as shown.



NOTE: Z zero is at the bottom of the part.

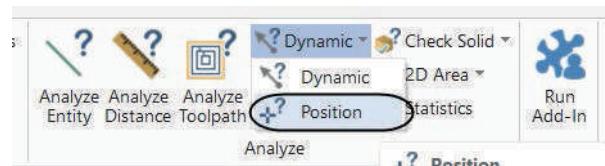
STEP 3: ANALYZE THE TOP OF THE GEOMETRY

In this step, you will use **Analyze Position** to establish the distance you need to translate the part to have Z zero at the top of the part.

Analyze Position allows you to view the XYZ coordinates of a selected position or point entity.

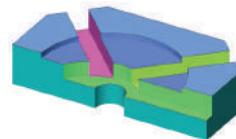
HOME

- From the **Analyze** group, click on the drop down arrow beside **Dynamic** and select **Position** as shown.

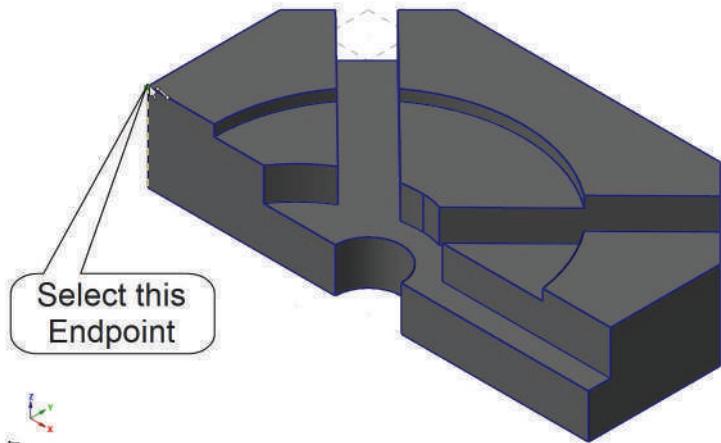


TUTORIAL #6

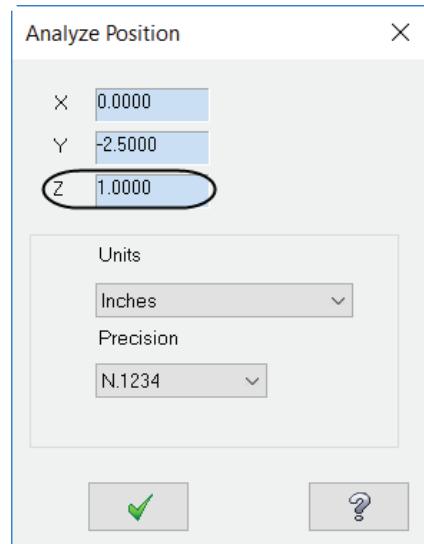
ANALYZE THE TOP OF THE GEOMETRY



- ♦ [Select a position]: Select the **Endpoint** as shown.

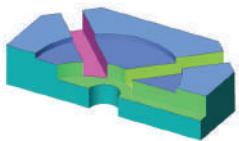


- ♦ In the **Analyze Position** dialog box, the **Z** value is **1.0** as shown.



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

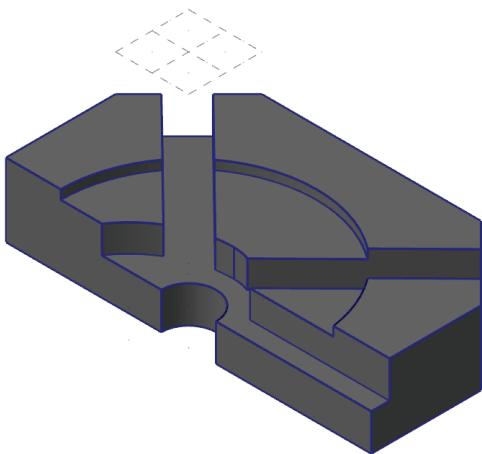




STEP 4: MOVE THE GEOMETRY

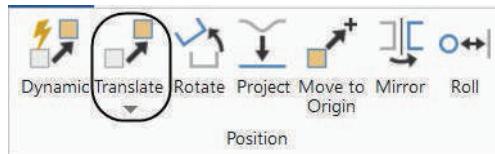
In this step, you will use **Transform Translate** to move the geometry along the Z axis -1.0".

Step Preview:



TRANSFORM

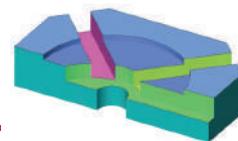
- From the **Position** group, select **Translate**.



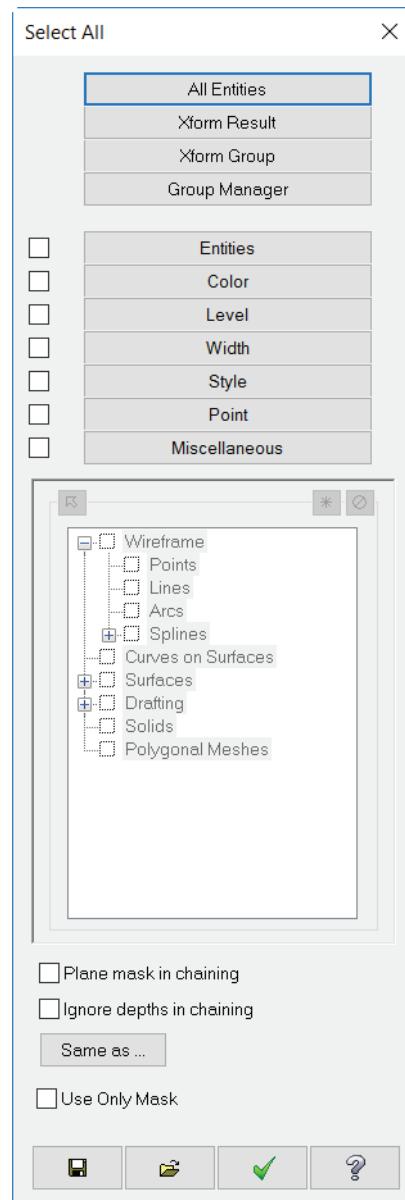
- [Translate: select entities to translate]: From the **Quick Mask** buttons on the right side of the screen, click on the **Select all advanced** icon as shown.



TUTORIAL #6 MOVE THE GEOMETRY



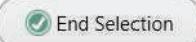
- The Select All dialog box will appear.

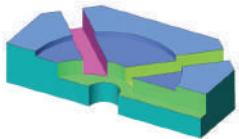


- Leave the default parameters and select the **OK** button to continue.

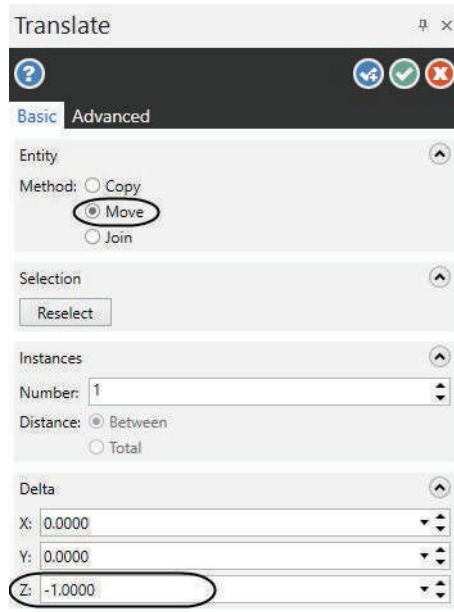


- Click on the **End Selection** button or press **Enter** to finish the selection.

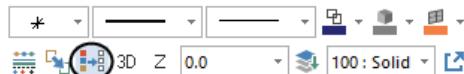




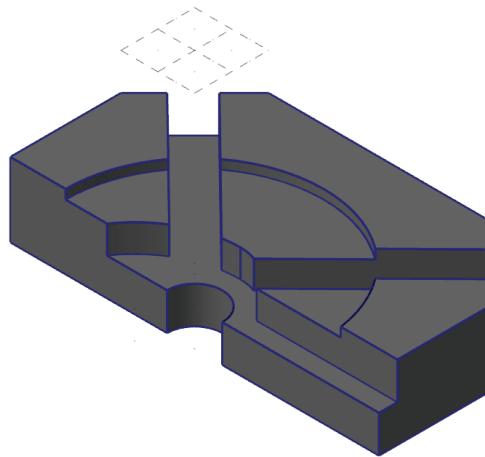
- ♦ In the **Translate** panel, enable **Move** and set the delta **Z** to **-1.0**.



- ♦ Select the **OK** button to finish the command.
- ♦ Right mouse click on the graphics window and select **Clear Colors** as shown to remove the result color.



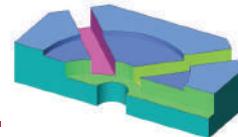
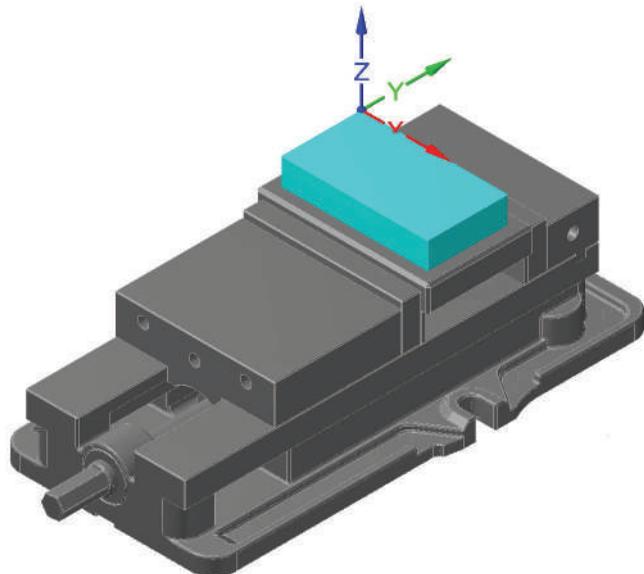
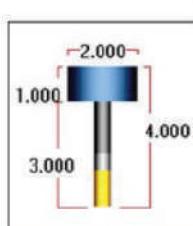
- ♦ The grid showing the origin location should look as shown.



STEP 5: SAVE THE FILE

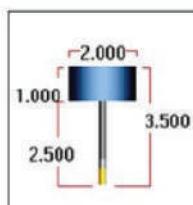
FILE

- ♦ **Save As.**
- ♦ File name: "Your Name_6".

TUTORIAL #6**SUGGESTED FIXTURE:****TOOLPATH CREATION****SUGGESTED FIXTURE:****SETUP SHEET:****TOOL LIST**

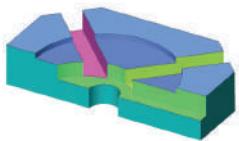
TYPE: Endmill1 Flat
DIA OFFSET:1
HOLDER: DEFAULT HOLDER
NUMBER:1
LENGTH OFFSET:1
#1 - 0.5000 ENDMILL1 FLAT - 1/2 FLAT ENDMILL

FLUTE LENGTH:1.0
OVERALL LENGTH:3.0
CORNER RAD:0.0
OF FLUTES:4



TYPE: Endmill1 Flat
DIA OFFSET:2
HOLDER: DEFAULT HOLDER
NUMBER:2
LENGTH OFFSET:2
#2 - 0.2500 ENDMILL1 FLAT - 1/4 FLAT ENDMILL

FLUTE LENGTH:0.5
OVERALL LENGTH:2.5
CORNER RAD:0.0
OF FLUTES:4



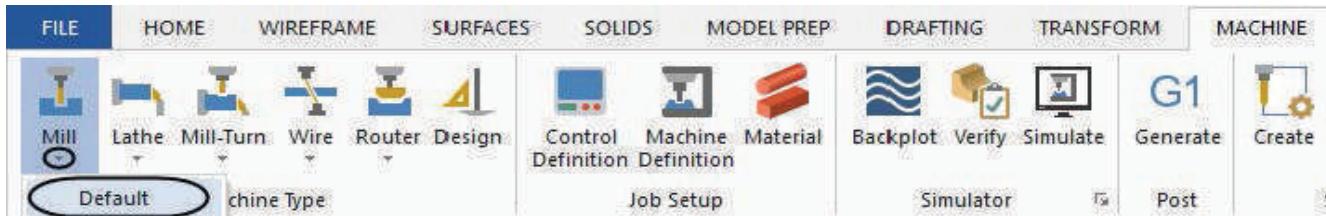
STEP 6: SELECT THE MACHINE AND SET UP THE STOCK

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

NOTE: For the purpose of this tutorial, we will be using the **Default Mill** machine.
Select the **Mill Default** only if there is no **Machine Group** in the **Toolpaths Manager**.

MACHINE

- From the **Machine Type** group, click on the drop down arrow below **Mill** and select the **Default**.

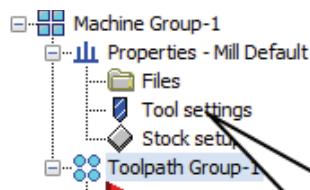


- Select the plus sign in front of **Properties** in the **Toolpaths Manager** to expand the **Toolpaths Group Properties**.

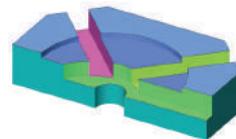


Select the plus sign

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

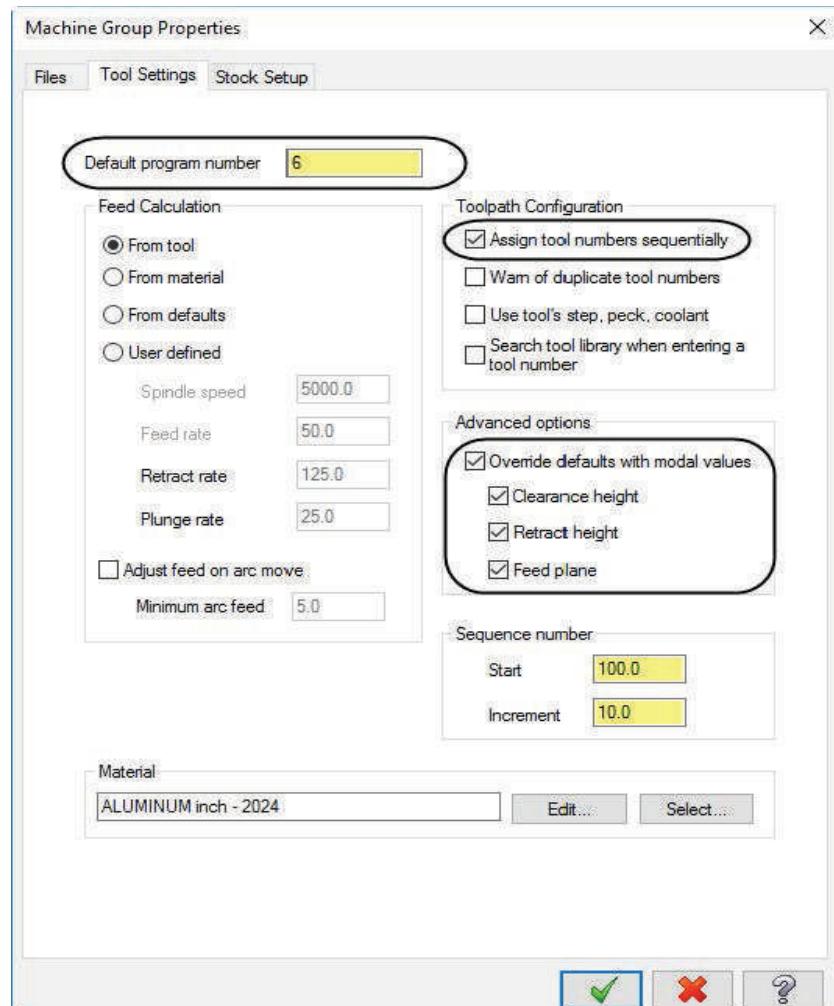


Select Tool settings



- ◆ Change the parameters to match the screen shot as shown in [Figure: 6.0.1](#).

Figure: 6.0.1



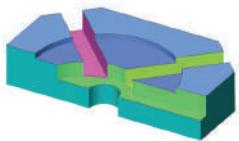
Default program number is used to enter a number if your machine tool requires a number for a program name.

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

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

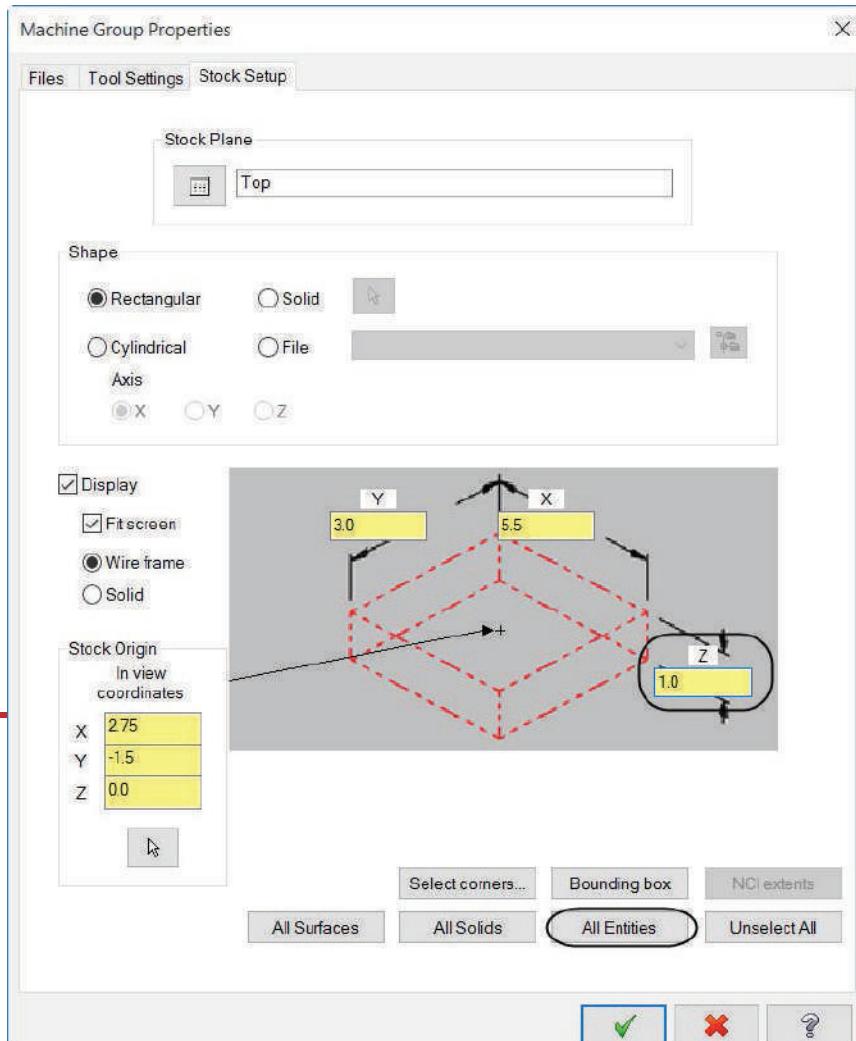
Override defaults with modal values enables the system to keep the values that you enter.

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



- ♦ Select the **Stock Setup** tab to define the stock.
- ♦ Pick the **Rectangular** shape option.
- ♦ Pick the **All Entities** button as shown in [Figure: 6.0.2](#).

Figure: 6.0.2



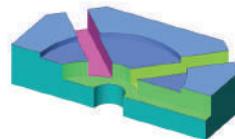
The **Stock Origin** values adjust the positioning of the stock, ensuring that you have an equal amount of extra stock around the finished part.

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

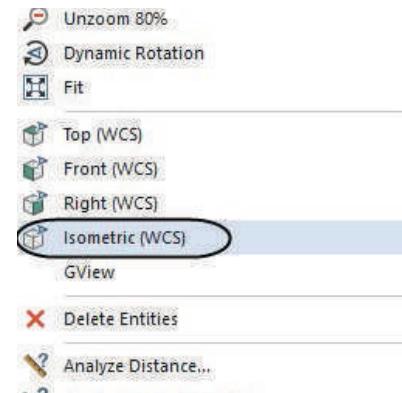
NOTE: The **stock** model that you create is used when viewing the file or the toolpaths, during backplot, or while verifying toolpaths. In the graphics, the plus shows you where the stock origin is. The default position is the middle of the stock.

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

TUTORIAL #6 SELECT THE MACHINE AND SET UP THE STOCK

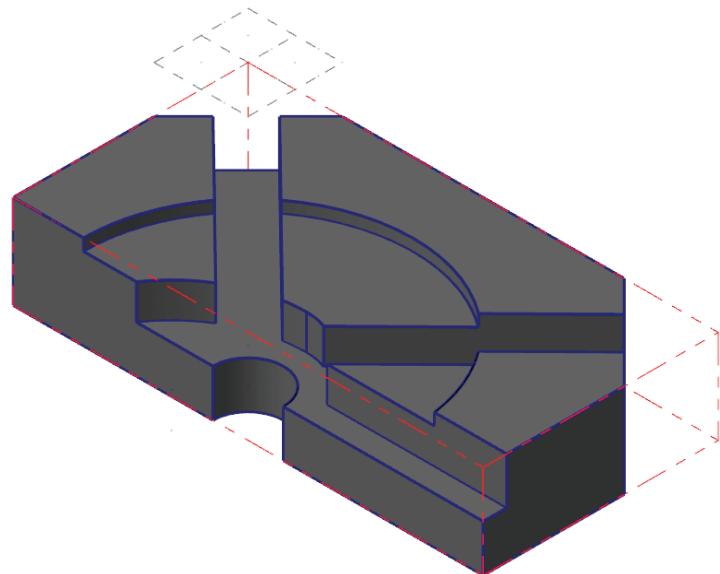


- ♦ Right mouse click in the graphics window and select the **Isometric** view to see the stock.

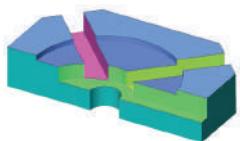


- ♦ Press **Alt + F1** to fit the drawing to the screen.
- ♦ The stock model should appear as shown in [Figure: 6.0.3](#).

Figure: 6.0.3



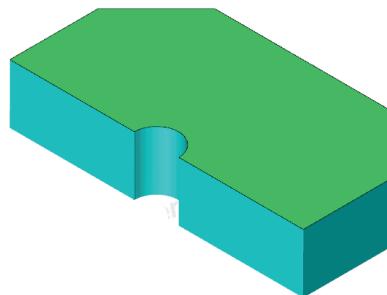
NOTE: The stock is not geometry and cannot be selected.
There will not be a facing toolpath because the stock is already to size.



STEP 7: 2D HIGH SPEED DYNAMIC MILL

2D High Speed Dynamic Mill utilizes the entire flute length of cutting tools to produce the smoothest, most efficient tool motion for high speed pocketing and core milling. The **Dynamic Mill** toolpath machines pockets, material that other toolpaths left behind, and standing bosses or cores. The toolpath depends on the **Machining strategy** that you choose in the **Chain Options**. If the strategy chosen is **From outside**, the toolpath starts at the outmost chain (in our case the stock) and moves freely outside of this area; the inner chain defines the limit of the toolpath. You can also machine pockets, in which case the strategy selected is **Stay inside** which keeps the tool inside the machining regions.

Toolpath Preview:



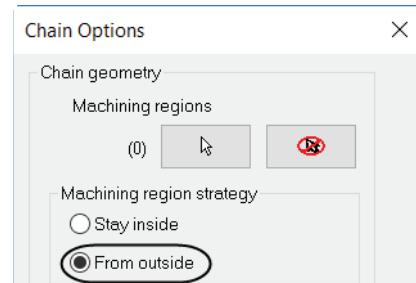
7.1 Chain Selection

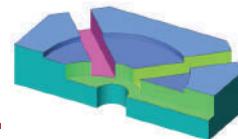
TOOLPATHS

- From the **2D** group, select **Dynamic Mill** as shown.

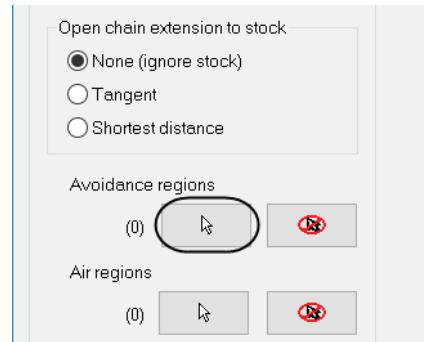


- In the **Chain Options, Machining region strategy**, enable **From outside**.

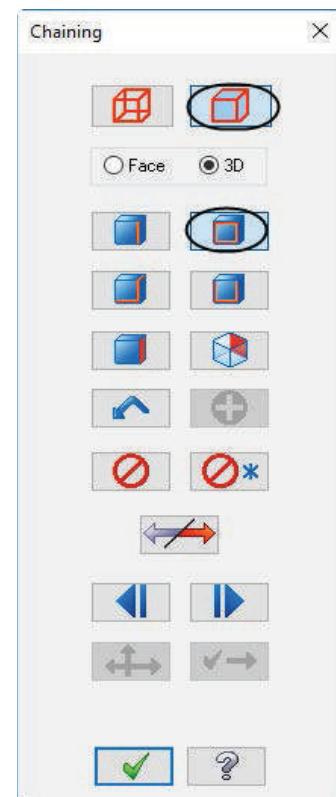


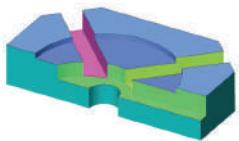


- In the **Chain Options**, **Avoidance regions**, click on the **Select avoidance chains** button as shown.



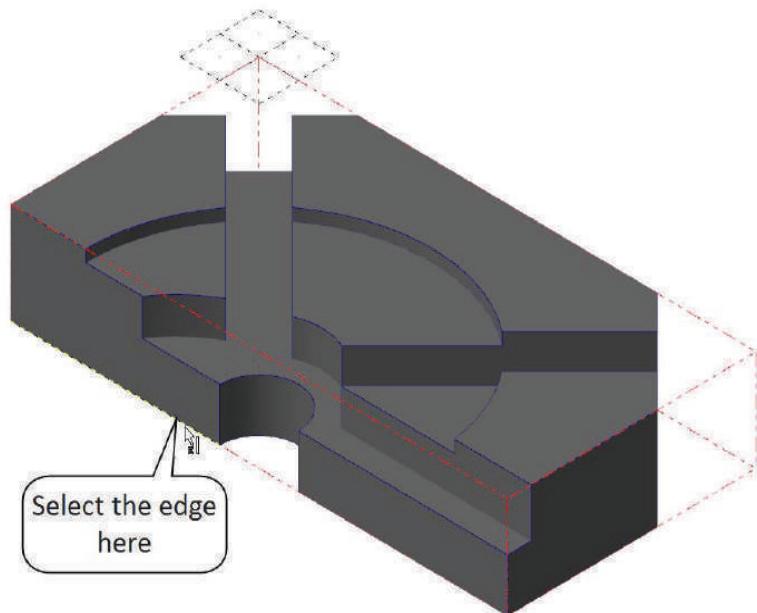
- In the **Chaining** dialog box, enable **Solid selection** and enable only the **Loop** button.



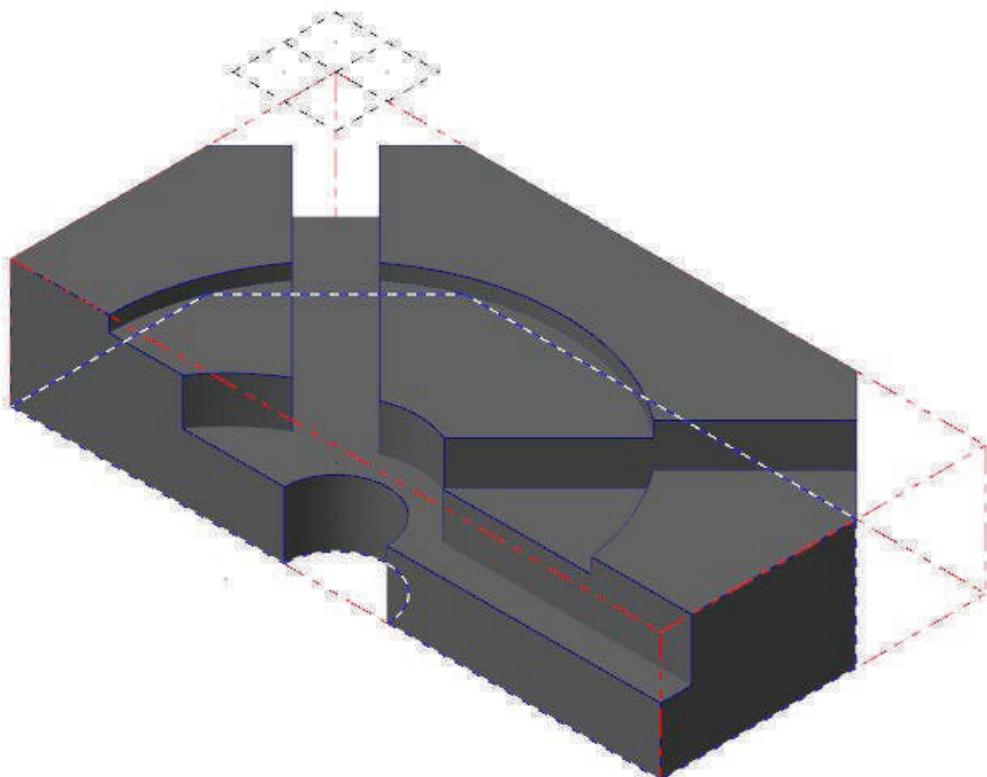


- ♦ [Solid Toolpath Chain: Select faces, edges, and/or loops]: Select the edge of the part as shown in [Figure: 7.1.1](#).

Figure: 7.1.1

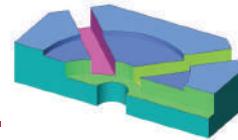


- ♦ The entire bottom of the part will be selected as shown.

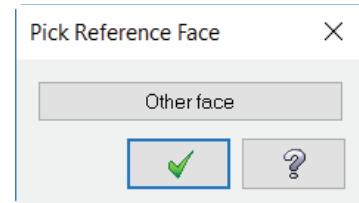


TUTORIAL #6

2D HIGH SPEED DYNAMIC MILL



- The Pick Reference Face dialog will appear allowing you to change the face selected if needed.



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



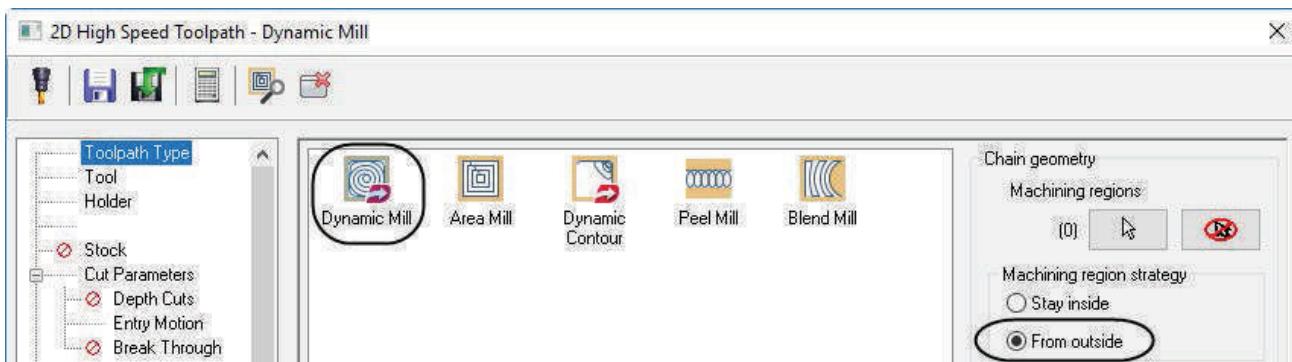
- Select the **OK** button to exit the **Chaining** dialog box.

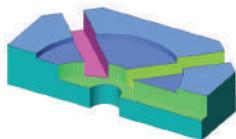


- Select the **OK** button to exit the **Chain Options** dialog box.



- In the **Toolpath Type** page, **Dynamic Mill** with **From outside** should be already selected.

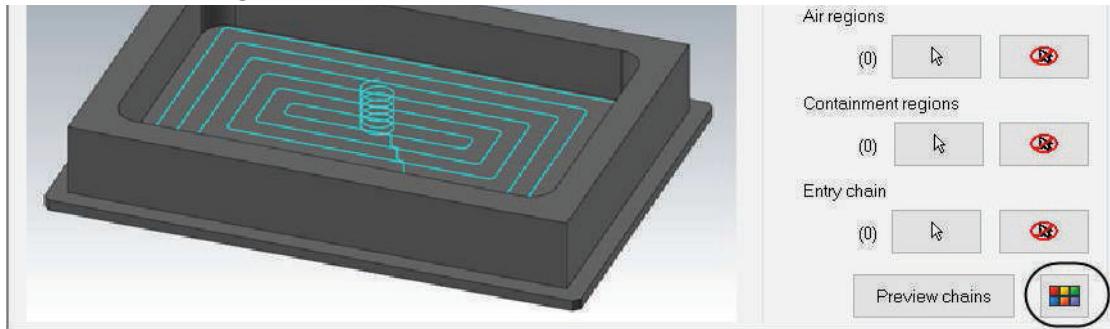




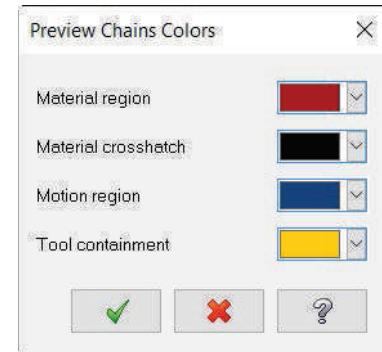
7.2 Preview Chains

The **Preview Chains** function is intended to give the user a quick visual representation of how Mastercam sees the various pieces of geometry that have been selected, how they interact with one another, and a general overview of how the toolpath will be calculated with the selections presently made.

- ♦ Click on the **Color** icon to see the legend for **Preview chains** as shown.



- ♦ The **Preview Chains Colors** dialog box should look as shown.



The **Material region** and **Material crosshatch** are the two colors that are used to define the material to be cut. The default colors are red for the background and black for the crosshatch.

The **Motion region** displays the area that Mastercam is making available to the toolpath for motion if it needs it. The color to represent it is dark blue. The primary reason for the display of the entire available (but not necessarily used) Motion region is to help the user visualize how the tool may move near or interact with any adjacent geometry.

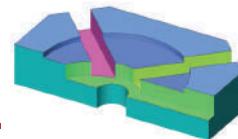
The **Tool containment** is what you have selected as the Containment region in the chain geometry. If you have not selected a containment region, it will default to the outside of the Motion region since that is currently the default area the toolpath is being contained to. The color used to represent the Tool containment is yellow.

- ♦ Select the **OK** button to exit **Preview Chains Colors**.

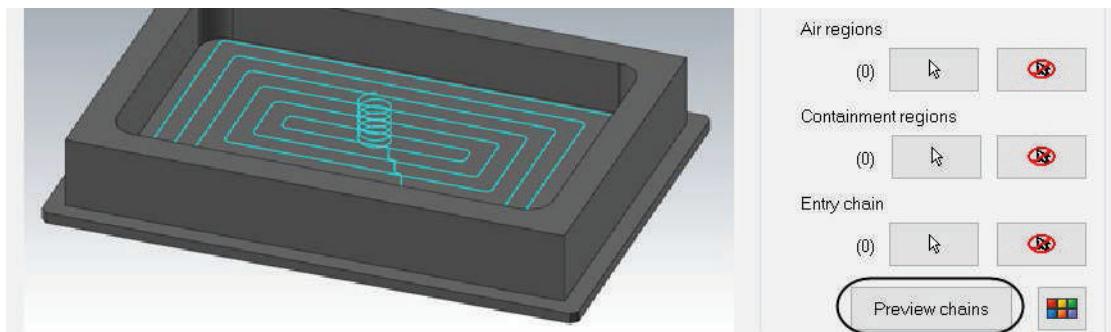


TUTORIAL #6

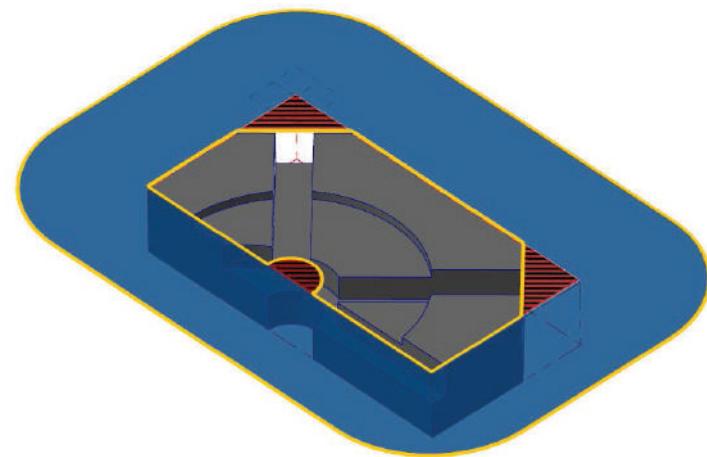
2D HIGH SPEED DYNAMIC MILL



- ♦ Select the **Preview chains** button as shown.

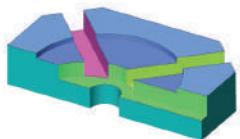


- ♦ Select the **Hide dialog** button to see the preview in the graphics window.
- ♦ The **Preview chains** should look as shown.



- ♦ Press **Esc** key to return to the toolpath parameters.
- ♦ Click on the **Preview chains** button again to clear the **Preview chains** display.





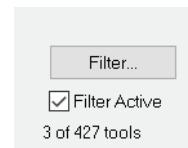
7.3 Select a 1/2" Flat Endmill from the Library and set the Tool Parameters

- ♦ Select **Tool** from the **Tree View** list.

- ♦ Click on the **Select library tool** button.

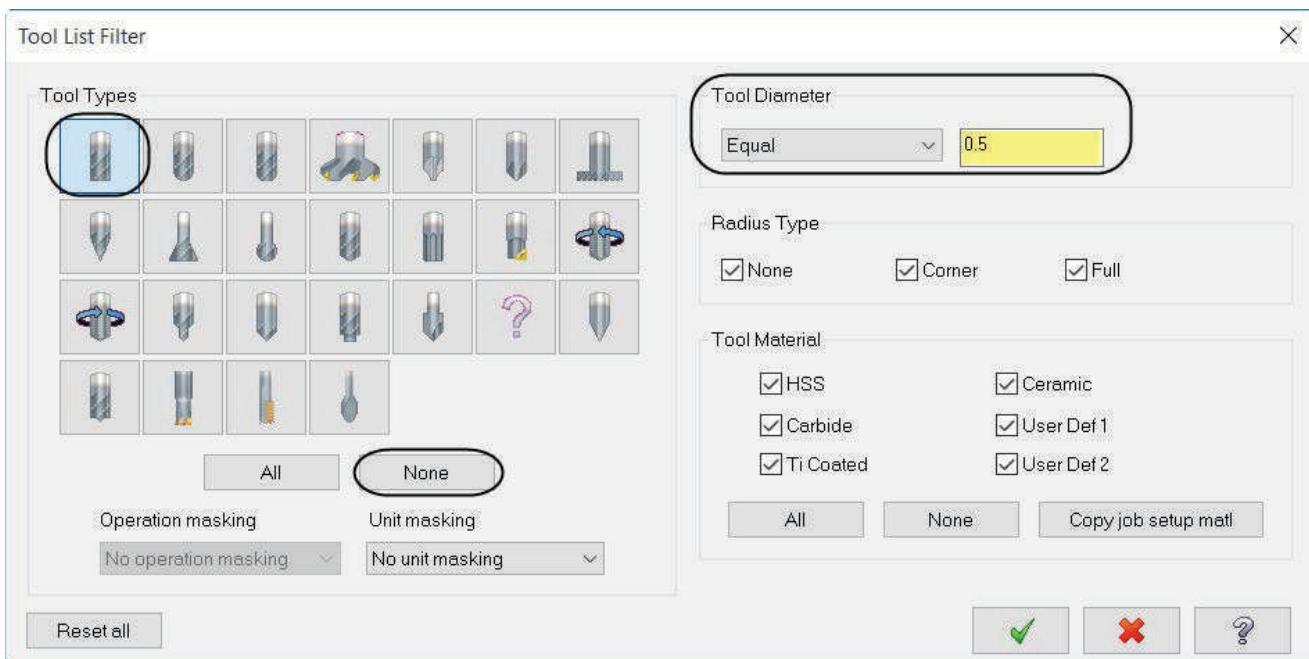
Select library tool...

- ♦ Select the **Filter** button as shown.



- ♦ Select the **None** button and then under **Tool Types**, choose the **Flat Endmill** icon.
- ♦ Under **Tool Diameter**, pick **Equal** and input a value of **0.5** as shown in [Figure: 7.3.1](#).

Figure: 7.3.1

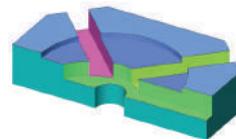


- ♦ Select the **OK** button to exit the **Tool List Filter**.
- ♦ In the **Tool Selection** dialog box, you should only see a **1/2" Flat Endmill**.

#	Assembly...	Tool Name	Holder Name	Dia.	Cor. r...	Length	# Flut...	Type	Rad....
290	-	1/2 FLAT ENDMILL	-	0.5	0.0	1.0	4	End...	None

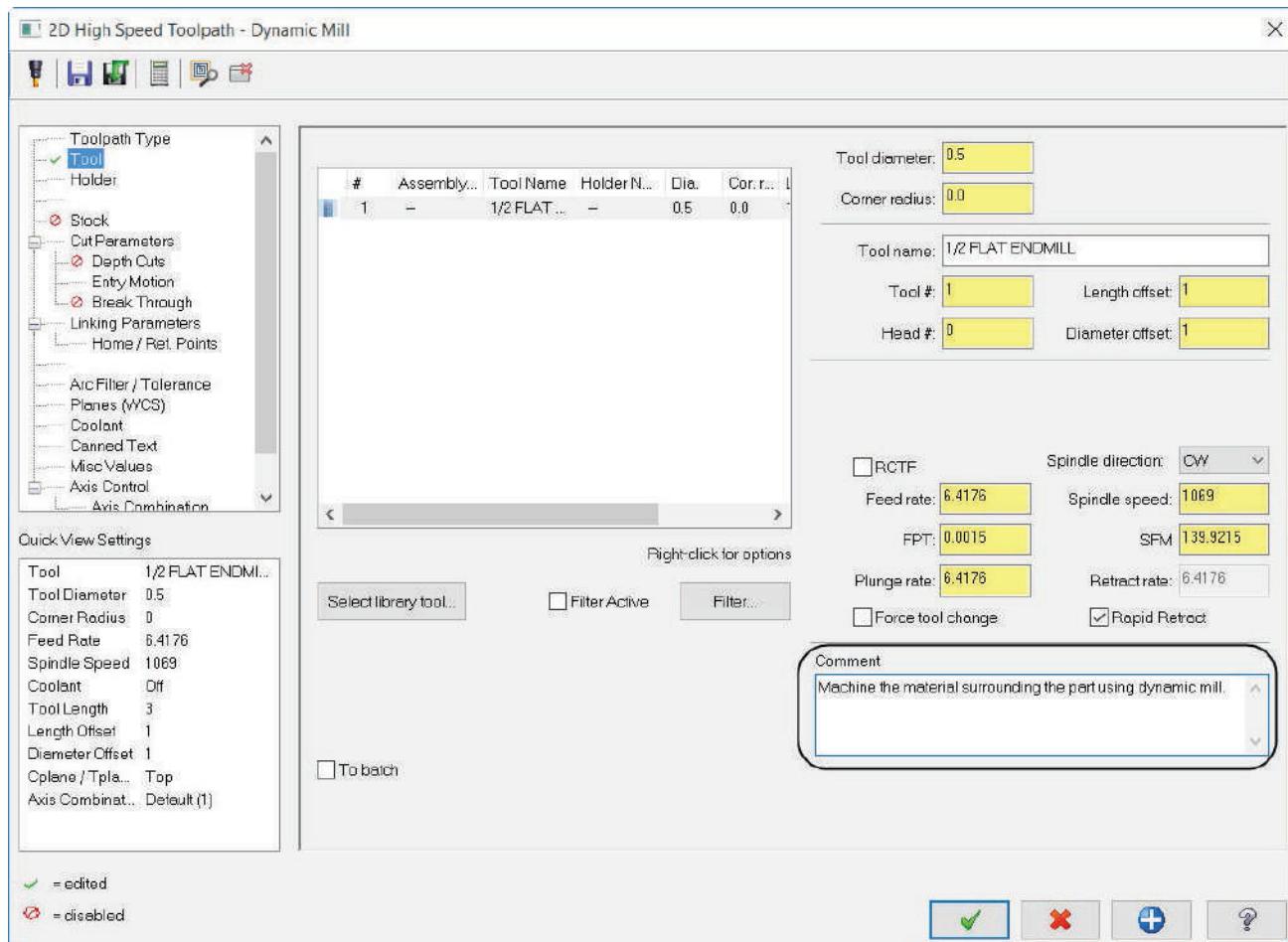
- ♦ Select the **1/2" Flat Endmill** in the **Tool Selection** page and then select the **OK** button to exit.

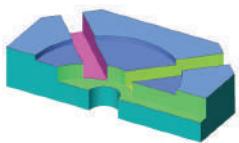
TUTORIAL #6 2D HIGH SPEED DYNAMIC MILL



- Make all the necessary changes as shown in [Figure: 7.3.2](#).

Figure: 7.3.2

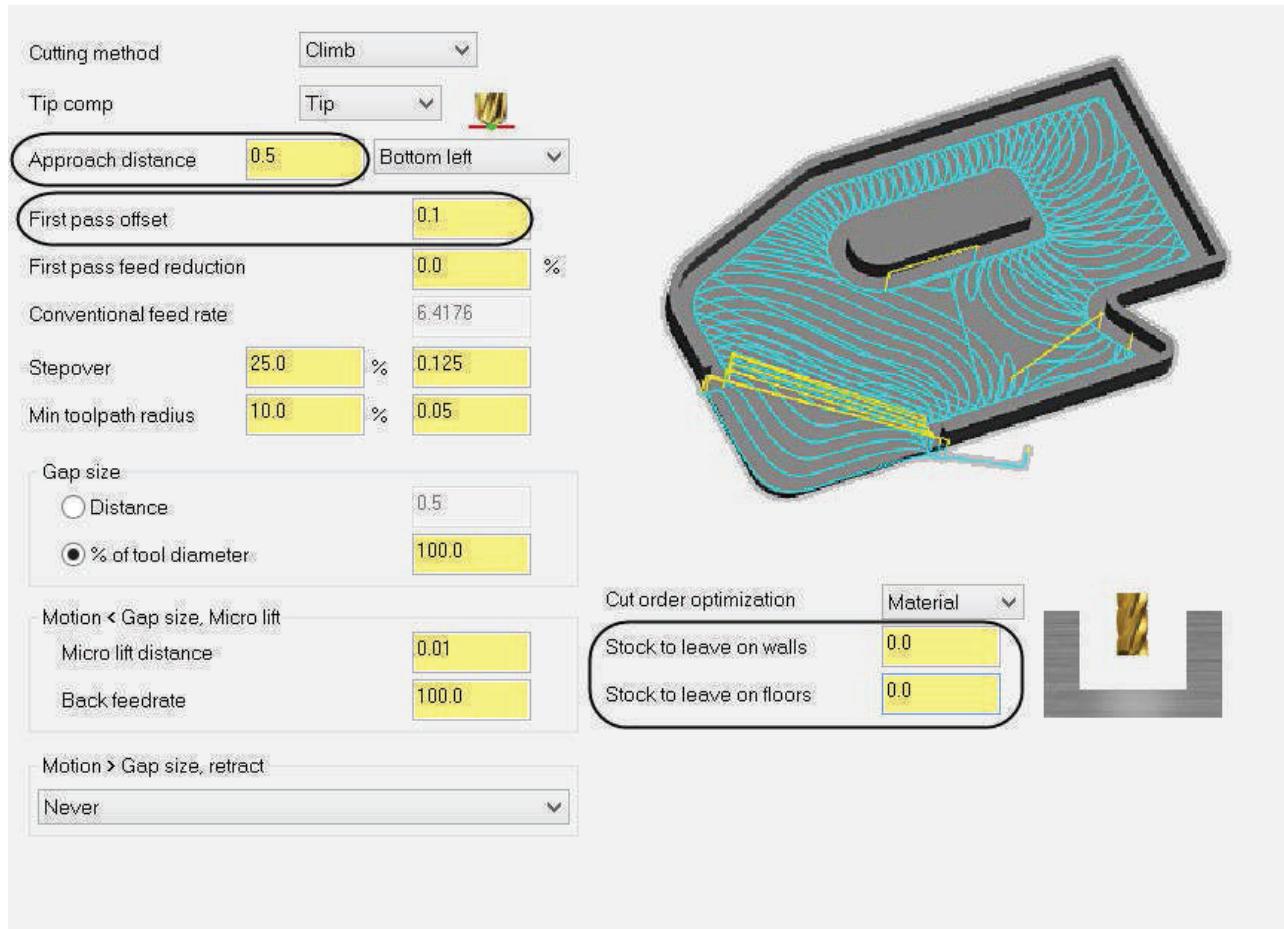


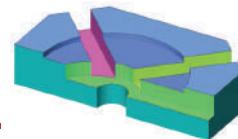


7.4 Set the Cut Parameters

- From the Tree View list, select **Cut Parameters** and ensure the parameters appear the same as shown in [Figure: 7.4.1.](#)

Figure: 7.4.1

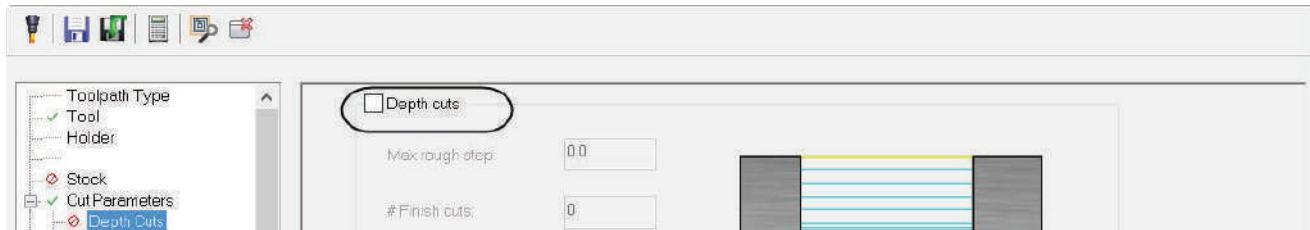




7.5 Set the Depth Cuts Parameters

- From the Tree View list, select the **Depth Cuts** parameters and disable **Depth cuts** as shown in [Figure: 7.5.1](#).

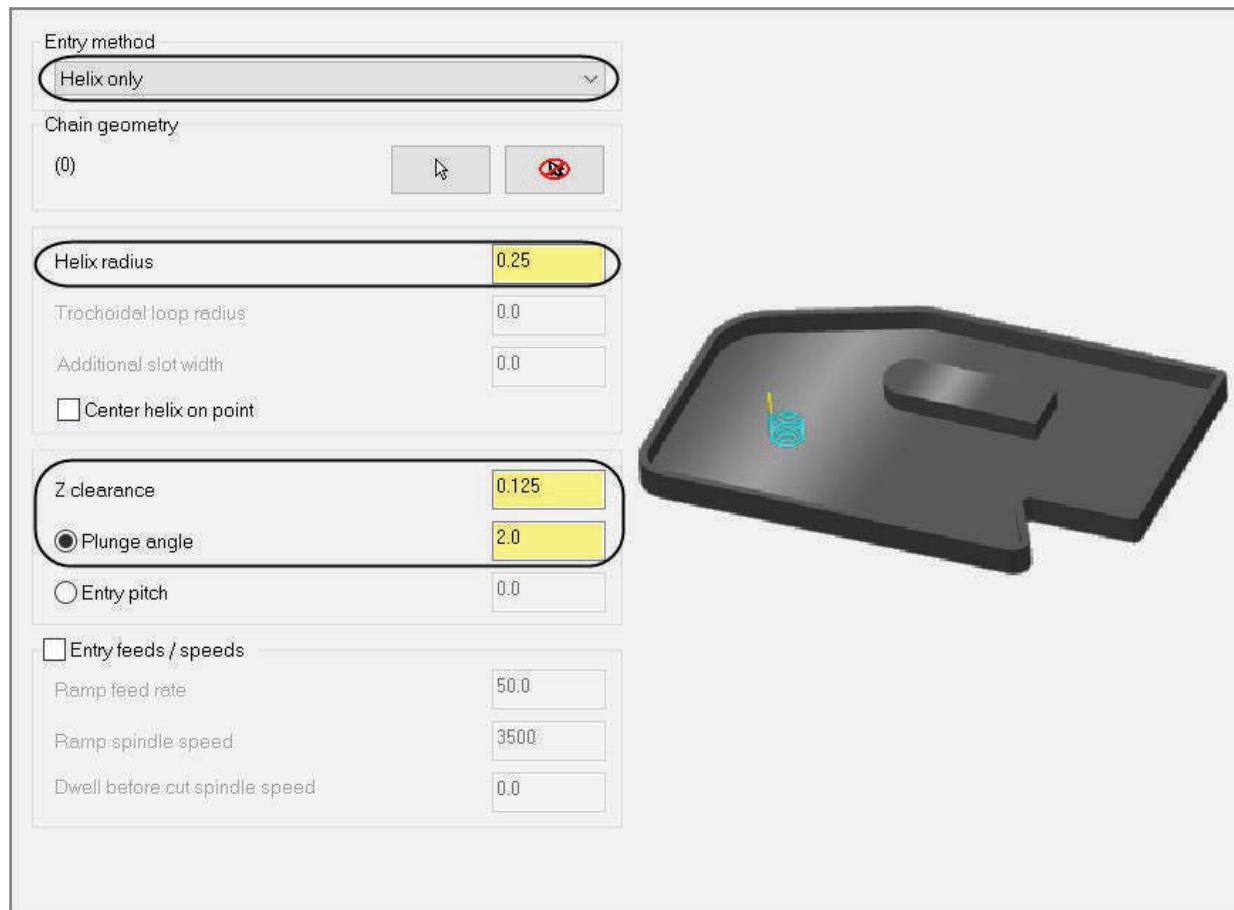
Figure: 7.5.1

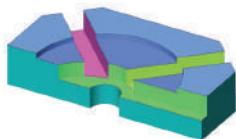


7.6 Set the Entry Motion

- From the Tree View list, select **Entry Motion**.
- Set the **Entry method** to **Helix only**. Make sure that the rest of the parameters are set as shown in [Figure: 7.6.1](#).

Figure: 7.6.1

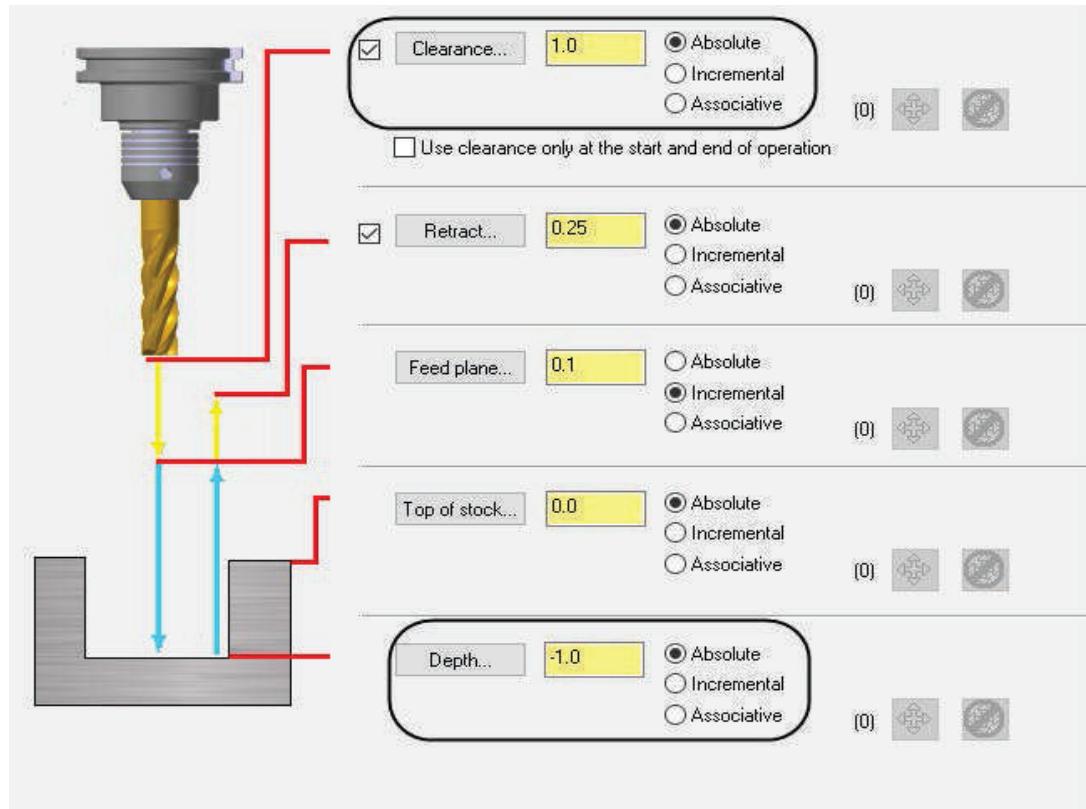




7.7 Set the Linking Parameters

- ♦ Select **Linking Parameters**, enable **Clearance**, input an **Absolute** value of **1.0** and an **Absolute Depth** value of **-1.0** as shown in [Figure: 7.7.1](#).

Figure: 7.7.1



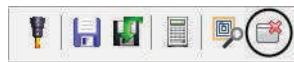
NOTE: You may press **Alt + T** at any time to toggle the toolpath display of the selected toolpaths.

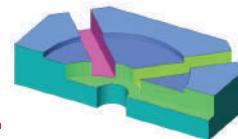
7.8 Preview the Toolpath

- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

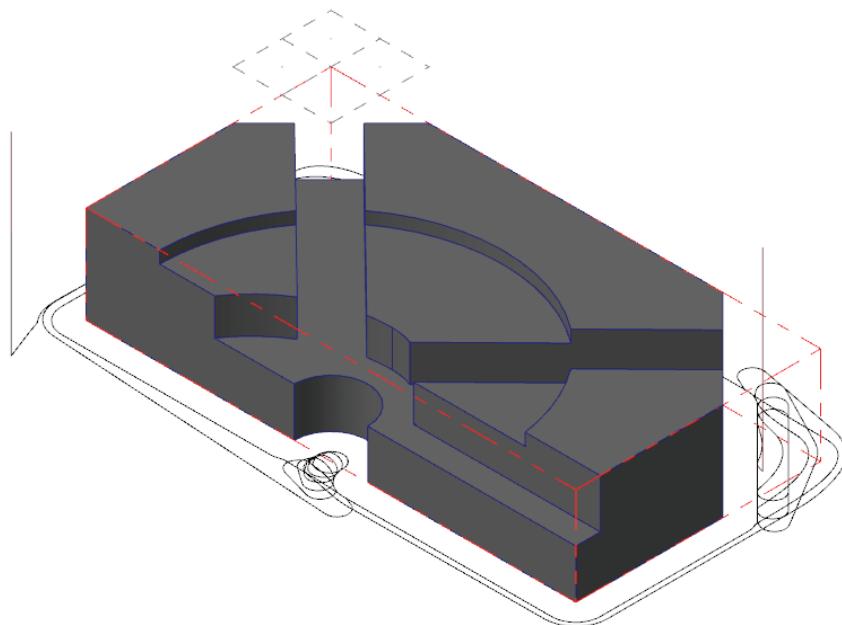


- ♦ Select the **Hide dialog** button to see the preview in the graphics window.





- The toolpath should look as shown.



- Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- Select the **OK** button to generate the **Dynamic Mill** toolpath.



STEP 8: BACKPLOT THE TOOLPATHS

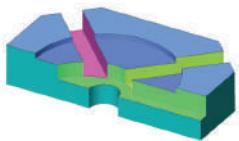
Backplotting shows the path the tools take to cut the part. This display lets you spot errors in the program before you machine the part. As you backplot toolpaths, Mastercam displays additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates, and the cycle time. It also shows any collisions between the workpiece and the tool.

- Make sure that the toolpaths are selected (signified by the green check mark on the folder icon). If the operation is not selected, choose the **Select all operations** icon.

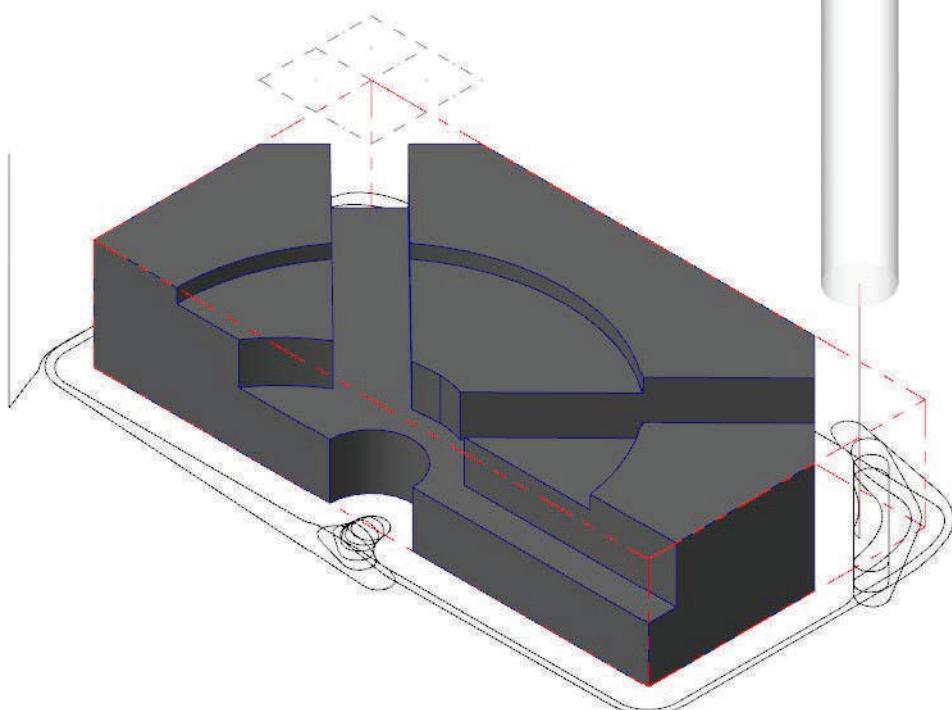


- Select the **Backplot selected operations** button.



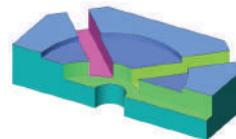


- ♦ Select the **Play** button to run **Backplot**.
- ♦ The toolpath should look as shown.



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





STEP 9: SIMULATE THE TOOLPATH IN VERIFY

Verify Mode shows the path the tools take to cut the part with material removal. This display lets you spot errors in the program before you machine the part. As you verify toolpaths, Mastercam displays additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates, and the cycle time. It also shows any collisions between the workpiece and the tool.

- From the **Toolpaths Manager**, select the **Verify selected operations** icon as shown.

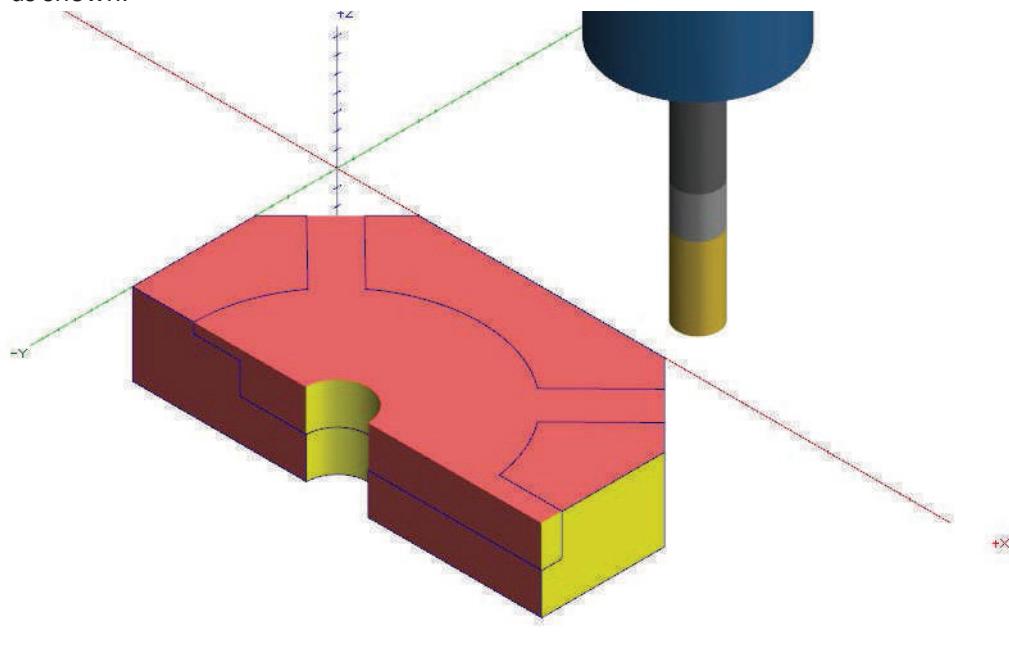


NOTE: Mastercam launches a new window that allows you to check the part using **Backplot** or **Verify**.

- Select the **Play** button to run **Verify**.

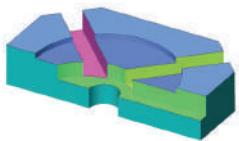


- The part should appear as shown.



- To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.
- Press **Alt + T** to remove the toolpath display if necessary.



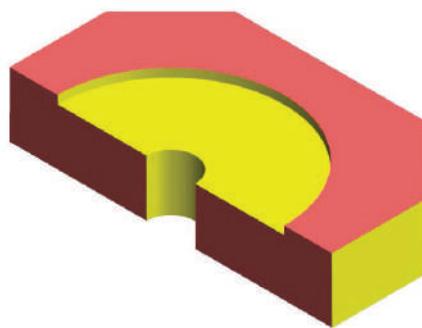


STEP 10: 2D HIGH SPEED BLEND MILL

2D High Speed Blend Mill toolpaths morph smoothly between two open chains. You can create the toolpath along or across the selected chains.

In this step you will use **Transform Project** to project the edge curves needed for the toolpath above the part. You will need to use **Join** and **Trim** commands to fix the chains.

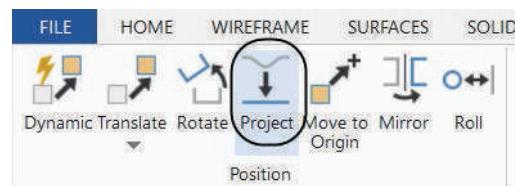
Toolpath Preview:



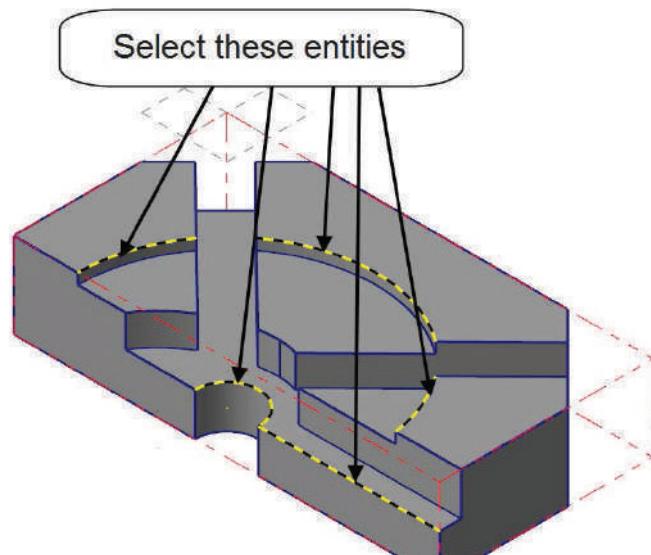
10.1 Create the two chains

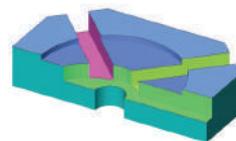
TRANSFROM

- From the **Position** group, select **Project** as shown.

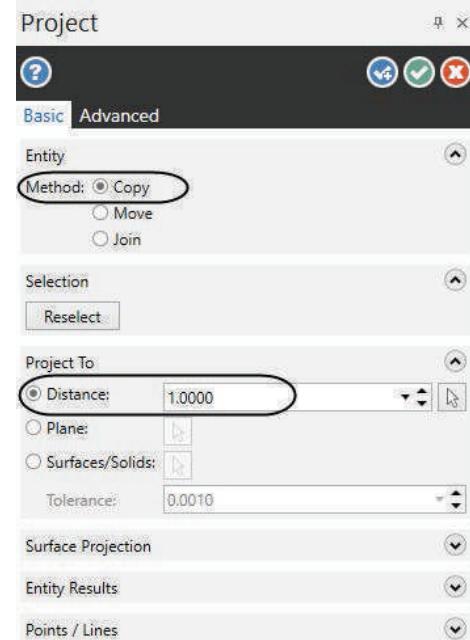


- [Select entities to project]: Select the arcs and the line as shown.

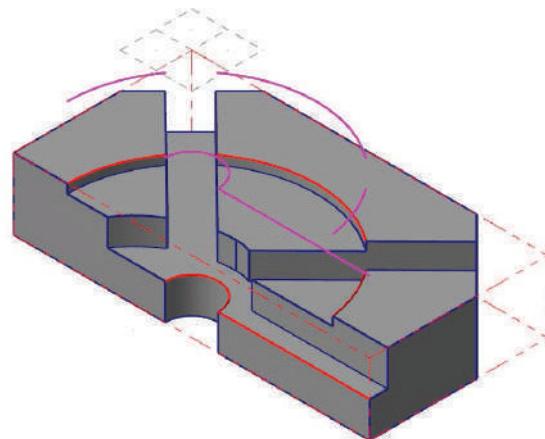


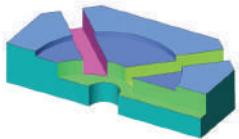


- ♦ Click on the **End Selection** button or press **Enter** to finish the selection.
- ♦ In the **Project** panel, make sure that **Copy** is enabled and change the **Distance** to **1.0** as shown.



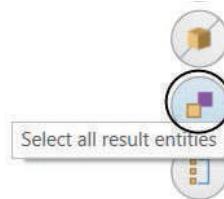
- ♦ Select the **OK** button to exit the command.
- ♦ The geometry should look as shown.





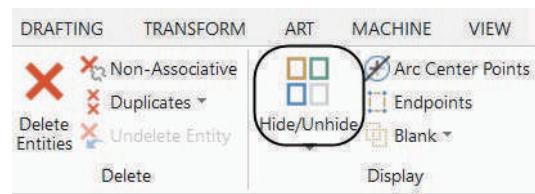
10.2 Use Hide Entity command to see on the screen only the projected entities

- From the **Quick Mask** buttons on the right side of the screen, choose **Select all result entities** as shown.

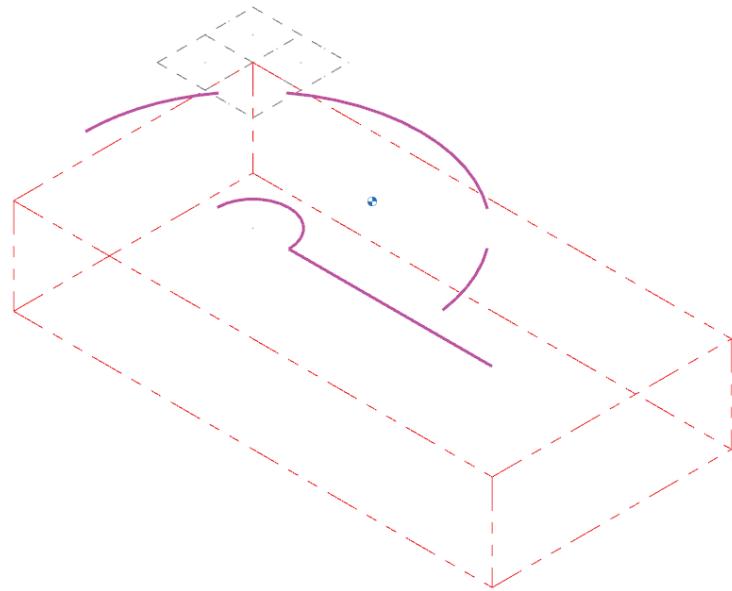


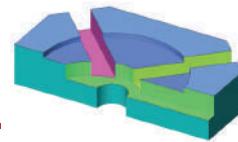
HOME

- From the **Display** group, select **Hide/Unhide**.



- Only the entities selected will be on the screen as shown.

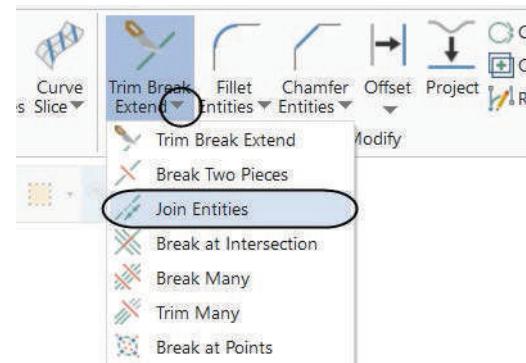




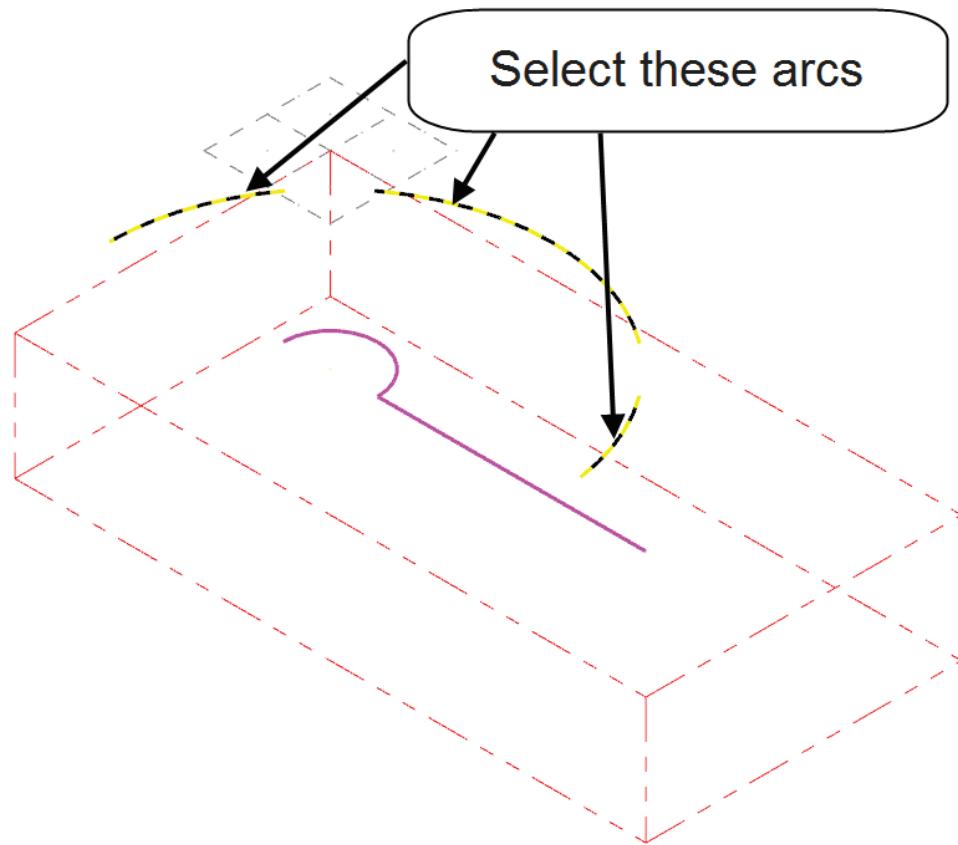
10.3 Join the arc

WIREFRAME

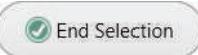
- From the **Modify** group, click on the drop down arrow under **Trim Break Extend** and select **Join Entities**.

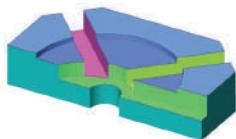


- [Select entities to join]: Select the arcs as shown.

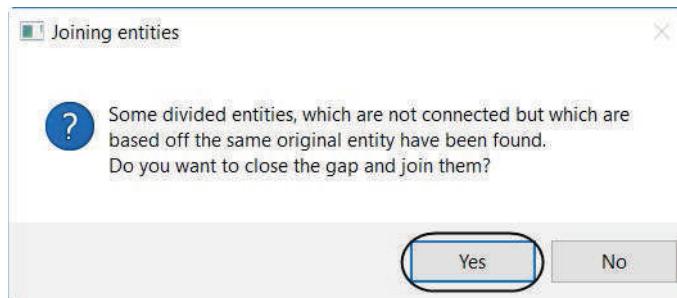


- Click on the **End Selection** button or press **Enter** to finish the selection.

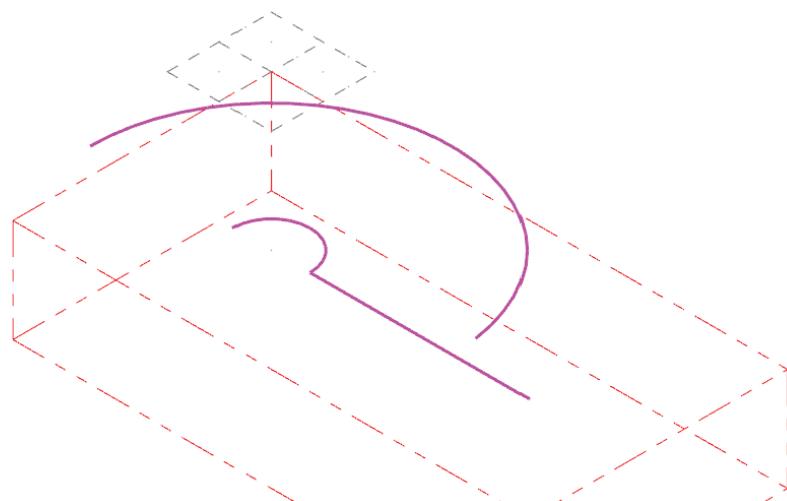




- ◆ This message will appear on the screen.



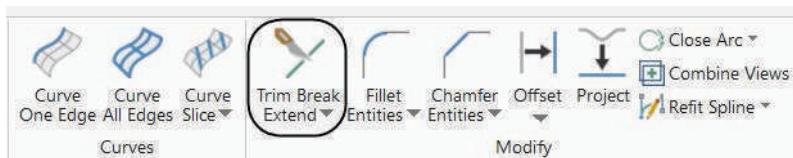
- ◆ Select the **Yes** button to continue.
- ◆ The geometry should look as shown.

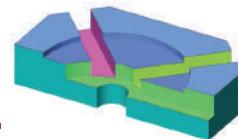


10.4 Trim the geometry

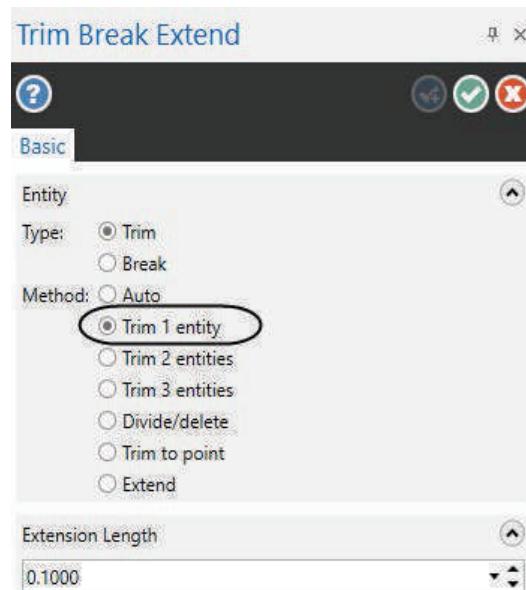
WIREFRAME

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

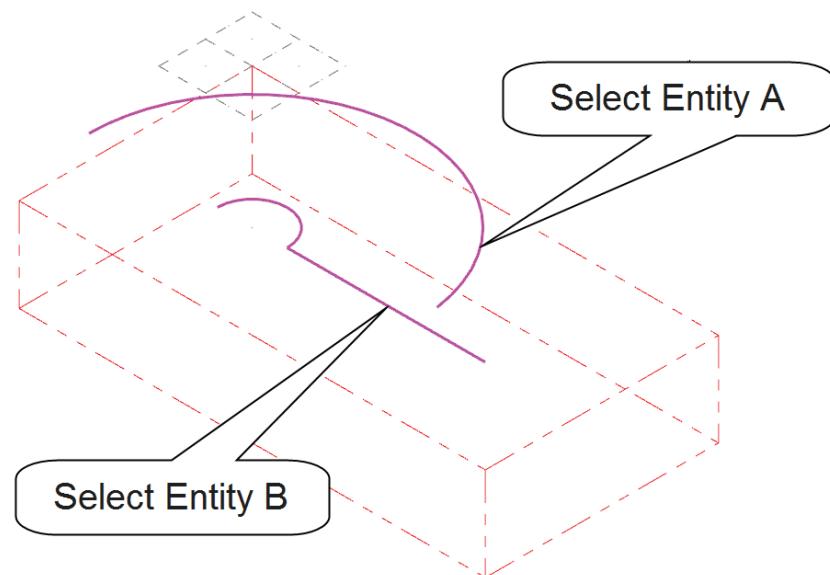




- From the Trim Break Extend panel, select Trim 1 entity as shown.

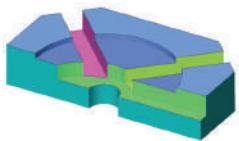


- [Select the entity to trim/extend]: Select Entity A as shown.
- [Select the entity to trim/extend to]: Select Entity B as shown.

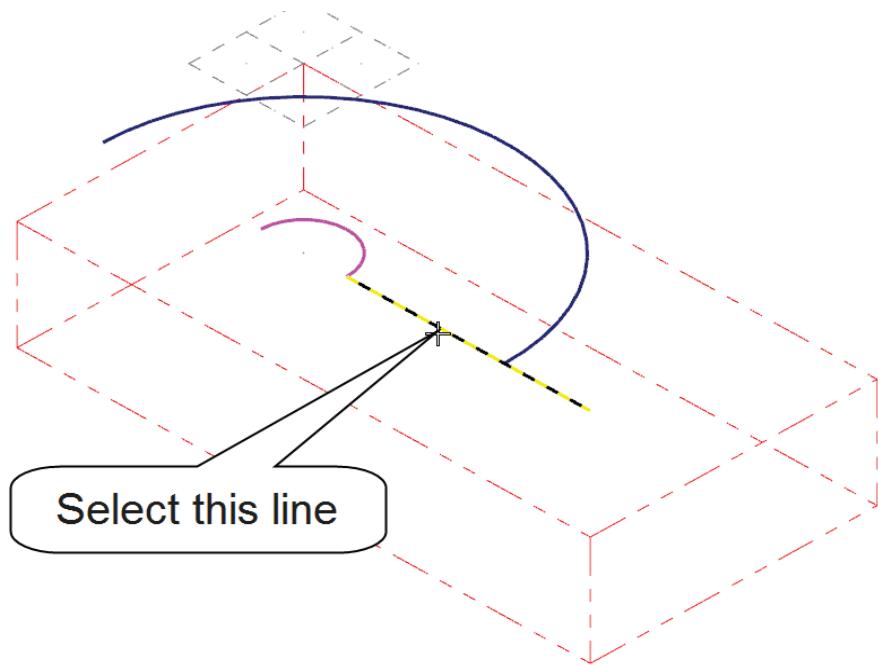


- Select the OK button to exit the command.





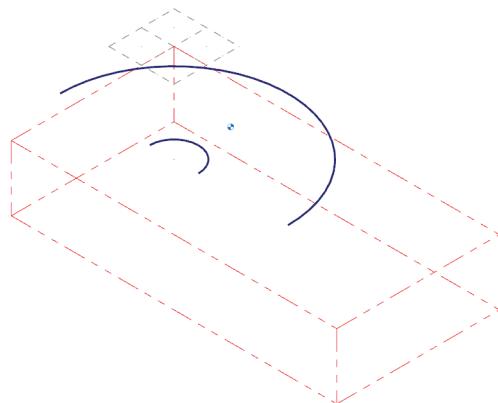
- ♦ Select the line as shown.



- ♦ Press **Delete** from the keyboard.
- ♦ Right mouse click on the graphics window and select **Clear Colors** as shown to remove the result color.

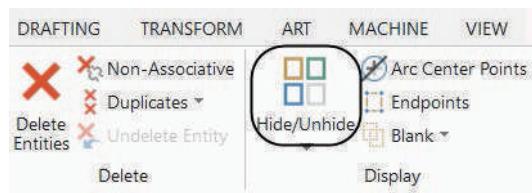


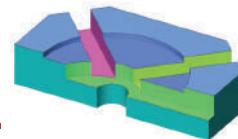
- ♦ The geometry should look as shown.



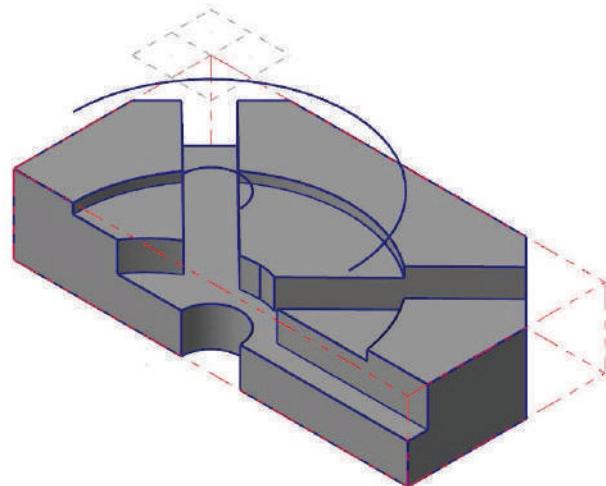
HOME

- ♦ From the **Display** group, select **Hide/Unhide**.





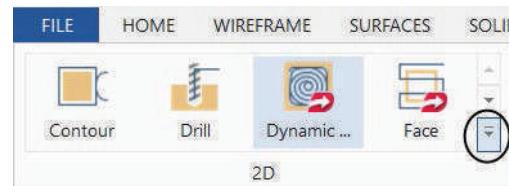
- ♦ The rest of the entities will appear on the screen as shown.



10.5 Chain the Entities

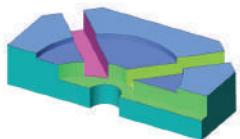
TOOLPATHS

- ♦ From the **2D** group, click on the **Expand gallery** icon.

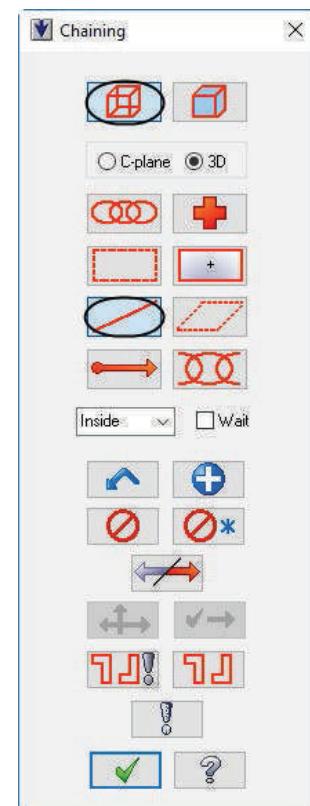


- ♦ Select the **Blend Mill** icon as shown.





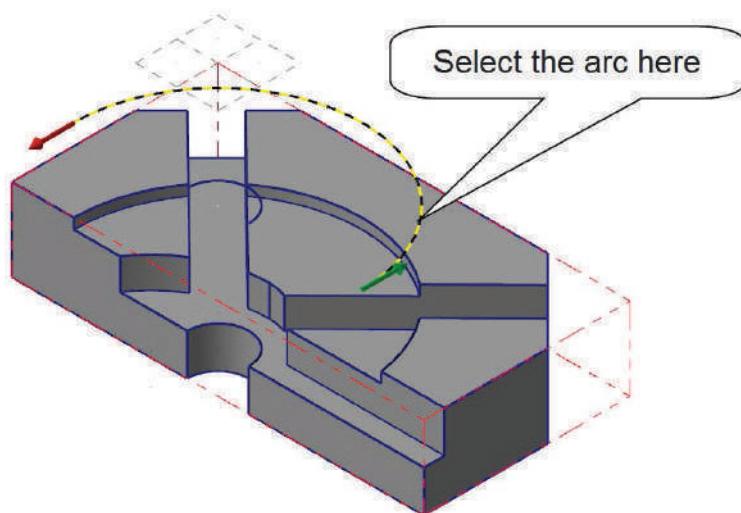
- When the **Chaining** dialog box appears, choose **Wireframe** and select the **Single** button.

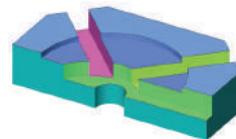


The **Single** button allows you to select one entity (a single line, arc, or spline) in a chain.

- [Blend: define chain 1]: Select the first arc as shown in [Figure: 10.5.1](#).

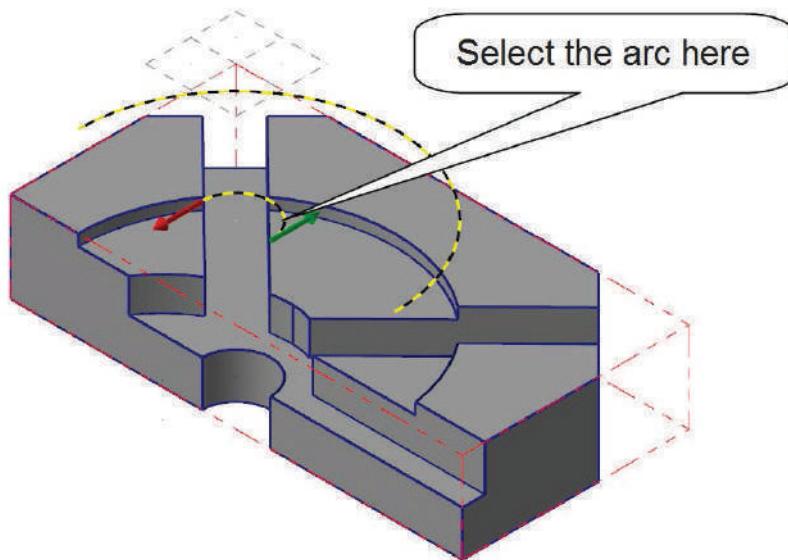
Figure: 10.5.1





- ♦ [Blend: define chain 2]: Select the second arc in the same direction as shown in [Figure: 10.5.2](#).

Figure: 10.5.2



NOTE: Make sure the chain direction is as shown; otherwise, in the **Chaining** dialog box click on the **Reverse**

button.



- ♦ Choose the **OK** button to exit the **Chaining** dialog box.
- ♦ Select **Toolpath Type** from the **Tree View list** and **Blend Mill** should be selected as shown.



Dynamic Mill



Area Mill



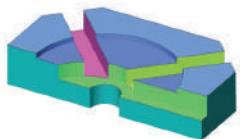
Dynamic Contour



Peel Mill



Blend Mill



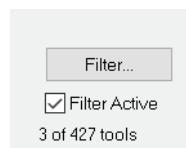
10.6 Select a 3/8" Flat Endmill from the Library and set the Tool Parameters

- ♦ Select **Tool** from the **Tree View** list.

- ♦ Click on the **Select library tool** button.

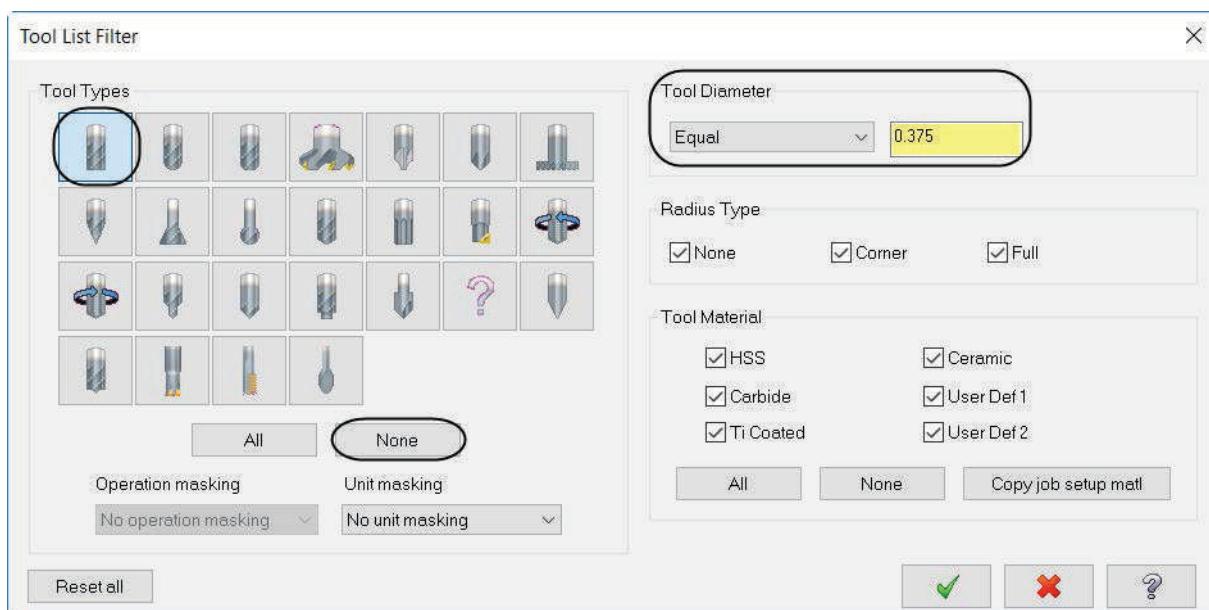
Select library tool...

- ♦ Select the **Filter** button as shown.



- ♦ Select the **None** button and then under **Tool Types**, choose the **Flat Endmill** icon.
- ♦ Under **Tool Diameter**, pick **Equal** and input a value **0.375** as shown in [Figure: 10.6.1](#).

Figure: 10.6.1

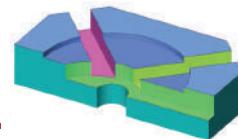


- ♦ Select the **OK** button to exit the **Tool List Filter**.
- ♦ In the **Tool Selection** dialog box, you should only see a **3/8" Flat Endmill**.

#	Assembly Name	Tool Name	Holder Name	Dia.	Cor. rad.	Length	# Flutes	Ra...	Type
287	-	3/8 FLAT ...	-	0...	0.0	0.75	4	No...	En...

- ♦ Select the **3/8" Flat Endmill** in the **Tool Selection** page and then select the **OK** button to exit.

TUTORIAL #6 2D HIGH SPEED BLEND MILL



- ♦ Make the necessary changes as shown in [Figure: 10.6.2](#).

Figure: 10.6.2

The screenshot shows the Mastercam software interface for a 2D High Speed Blend Mill. On the left, a list of tools is displayed:

#	Assembly Name	Tool Name	Holder Name	Dia.
1	-	1/2 FLAT ...	-	0.5
2	-	3/8 FLAT ...	-	0...

On the right, various toolpath parameters are set:

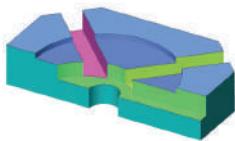
- Tool diameter: 0.375
- Corner radius: 0.0
- Tool name: 3/8 FLAT ENDMILL
- Tool #: 2
- Length offset: 2
- Head #: 0
- Diameter offset: 2

Machine settings include:

- RCTF:
- Spindle direction: CW
- Feed rate: 6.332032
- Spindle speed: 1426
- FPT: 0.0011
- SFM: 139.9869
- Plunge rate: 6.332032
- Retract rate: 6.332032
- Force tool change:
- Rapid Retract:

A comment box contains the text: "Machine the step in a morph transition between the two curves using blend toolpath."

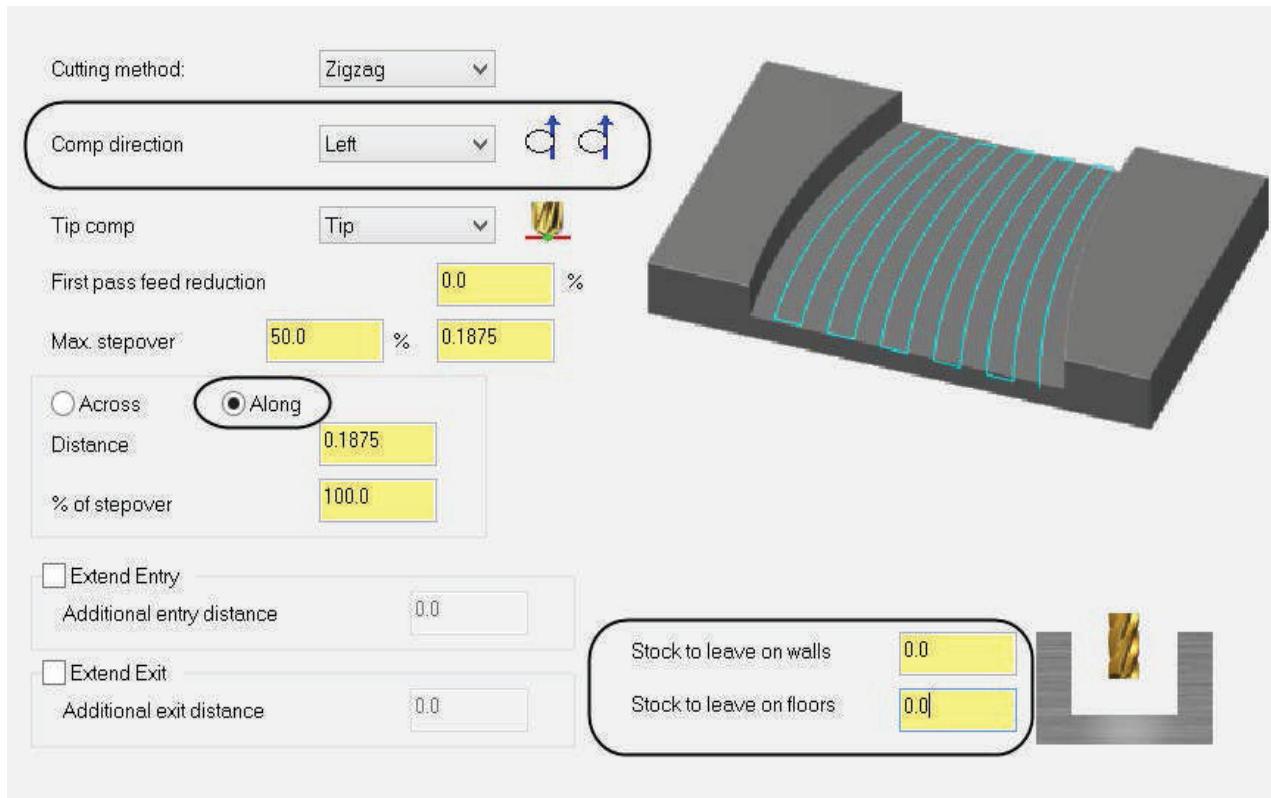
Other interface elements include "Select library tool...", "Filter Active", "Filter...", "Right-click for options", and "To batch".



10.7 Set the Cut Parameters

- ♦ Select **Cut Parameters** from the Tree View list.
- ♦ Set the **Cutting method** to **Zigzag** and **Along** as shown in [Figure: 10.7.1](#). This will morph start the cut along the first chain and then morph it towards the second chain.

Figure: 10.7.1

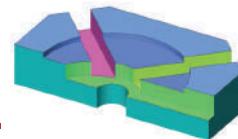


Compensation direction set to **Left** will allow the tool to travel to the left of the selected chains.

Along cuts in the along direction while stepping over in the across direction.

Max stepover sets the distance between adjacent passes.

Distance and **% of stepover** set the spacing between the temporary across moves. These moves are used to generate the final toolpath but are not included in the final toolpath.



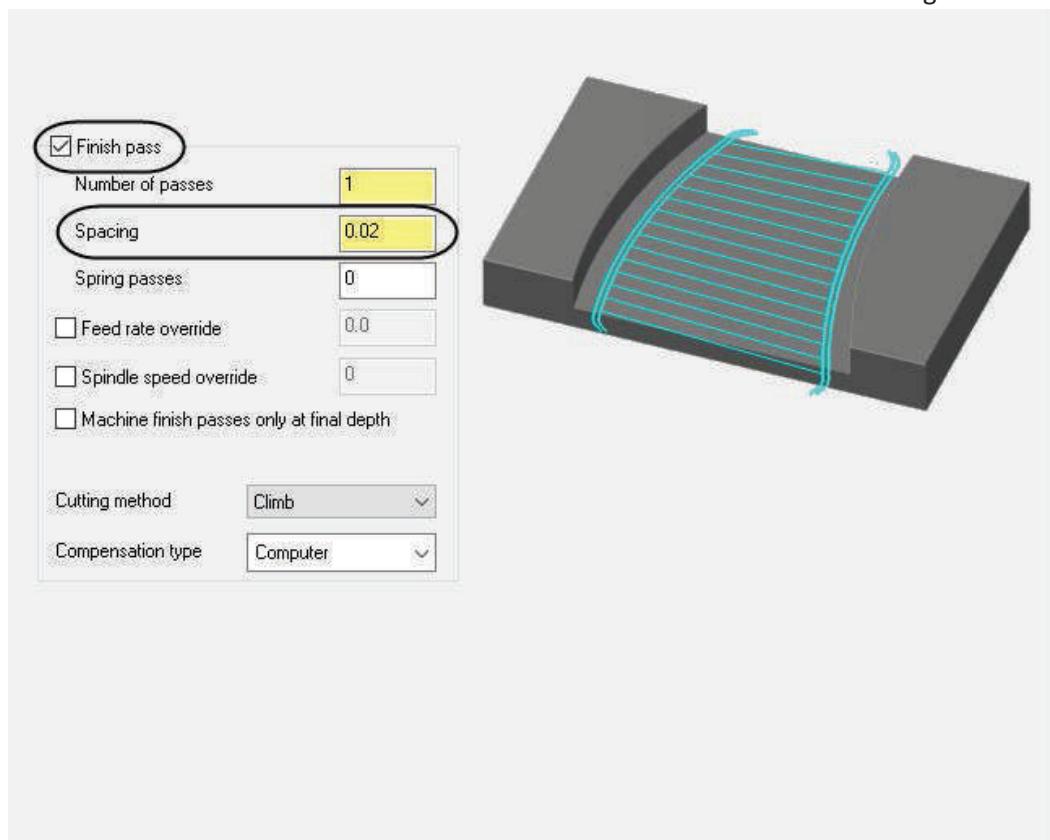
10.8 Set Depth Cuts

- ♦ Ensure **Depth Cuts** is disabled.

10.9 Finish Passes

- ♦ Select **Finish Passes** from the **Tree View** list.
- ♦ Enable **Finish pass** and change the parameters as shown in [Figure: 10.9.1](#).

Figure: 10.9.1

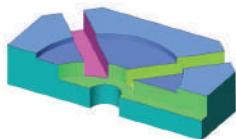


The **Finish pass** page allows you to add finish passes along the selected chains of the toolpath.

Number of passes sets the number of finish passes.

Spacing sets the distance between the finish passes.

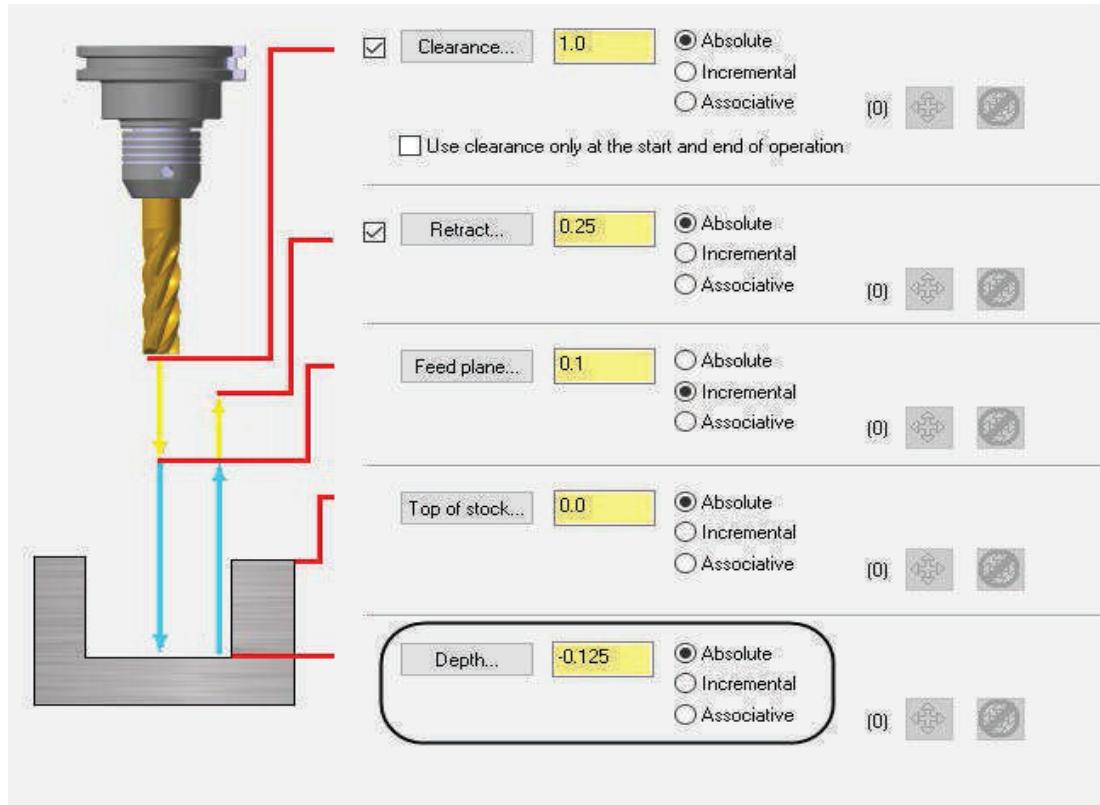
Machine finish passes only at final depth performs the finish passes only at the final cutting depth.



10.10 Set the Linking Parameters

- ♦ Select **Linking Parameters** from the **Tree View** list.
- ♦ Enter a **Depth** of **-0.125** as shown in [Figure: 10.10.1](#).

Figure: 10.10.1



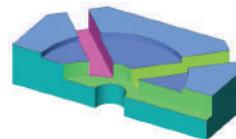
10.11 Preview the Toolpath

- ♦ To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

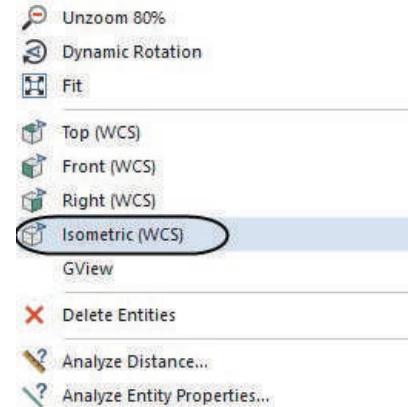


- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.

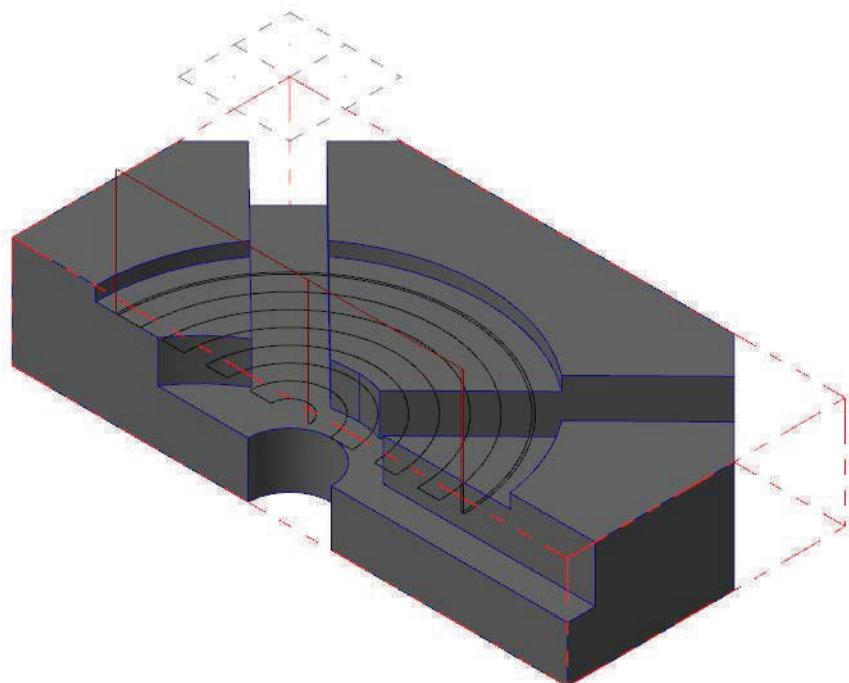




- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



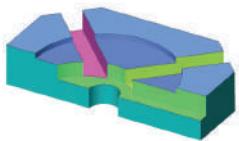
- ♦ The toolpath should look as shown.



- ♦ Press **Esc** key to exit the preview.

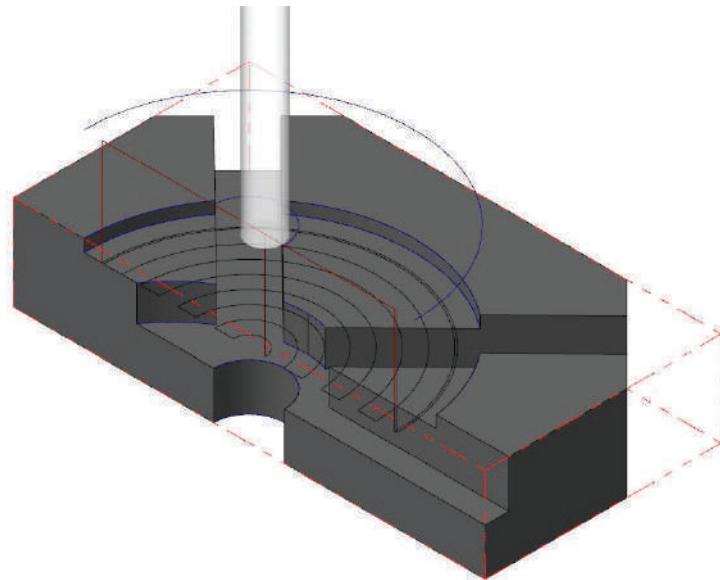
NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to generate the **Blend Mill** toolpath. 



10.12 Backplot the toolpath

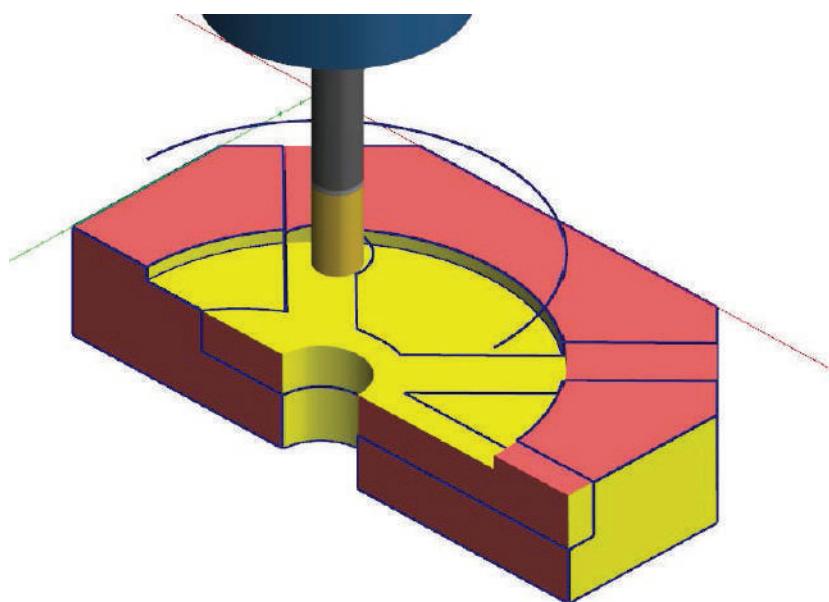
- ♦ To **Backplot** the toolpath, see [page 586](#) for more information.



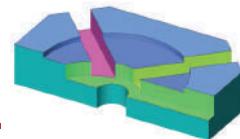
- ♦ Select the **OK** button to exit the **Backplot**.

10.13 Verify the toolpaths

- ♦ To **Verify** the toolpaths, see [page 588](#) for more information.
- ♦ To verify all toolpaths, from the **Toolpaths Manager**, choose the **Select all operations** icon.



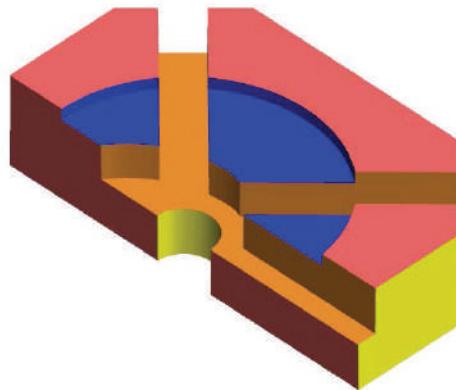
- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.



STEP 11: 2D HIGH SPEED AREA MILL

2D High Speed Area Mill toolpaths machine pockets, material that other toolpaths left behind, and standing bosses or cores. Like **Dynamic Mill**, based on the **Machining strategy** selected, it can generate the free flowing motion needed to machine features such as standing bosses and cores in a single operation.

Toolpath Preview:



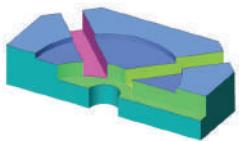
11.1 Chain the Entities

- ◆ Press **Alt + T** to remove the toolpath display from the previous operation.
- ◆ From the **2D** group, click on the **Expand gallery** icon.

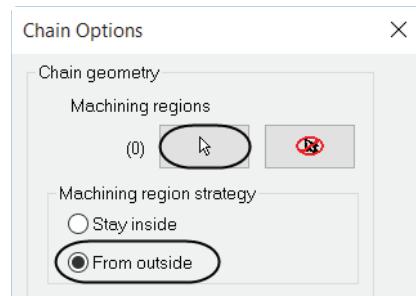


- ◆ From the **Toolpath Gallery**, select **Area Mill** as shown.



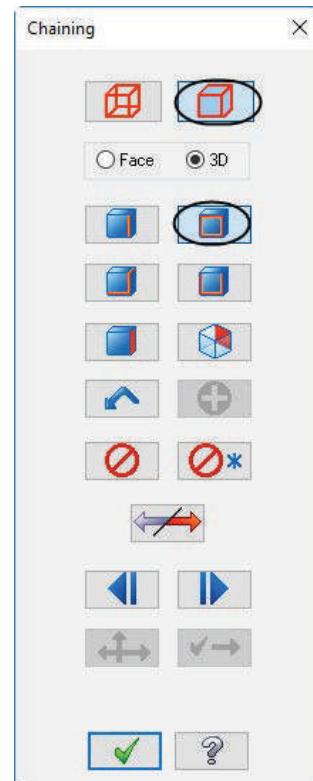


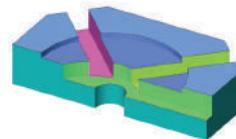
- In the **Chain Options, Machining region strategy**, enable **From outside** and click on the **Select** button under **Machining regions** as shown.



NOTE: The **Machining regions** chain determines where the tool will start to machine from.

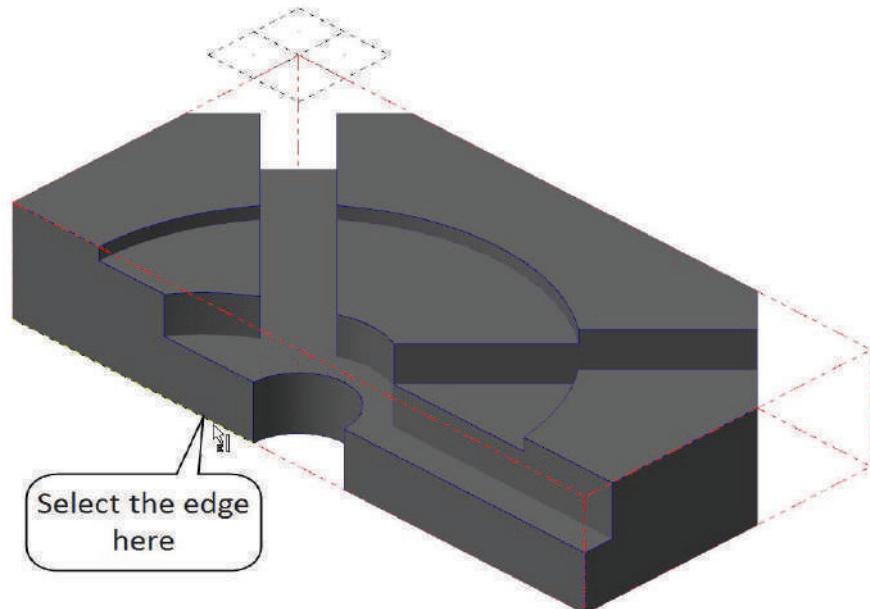
- In the **Chaining** dialog box, enable **Solids** and enable only the **Loop** button.



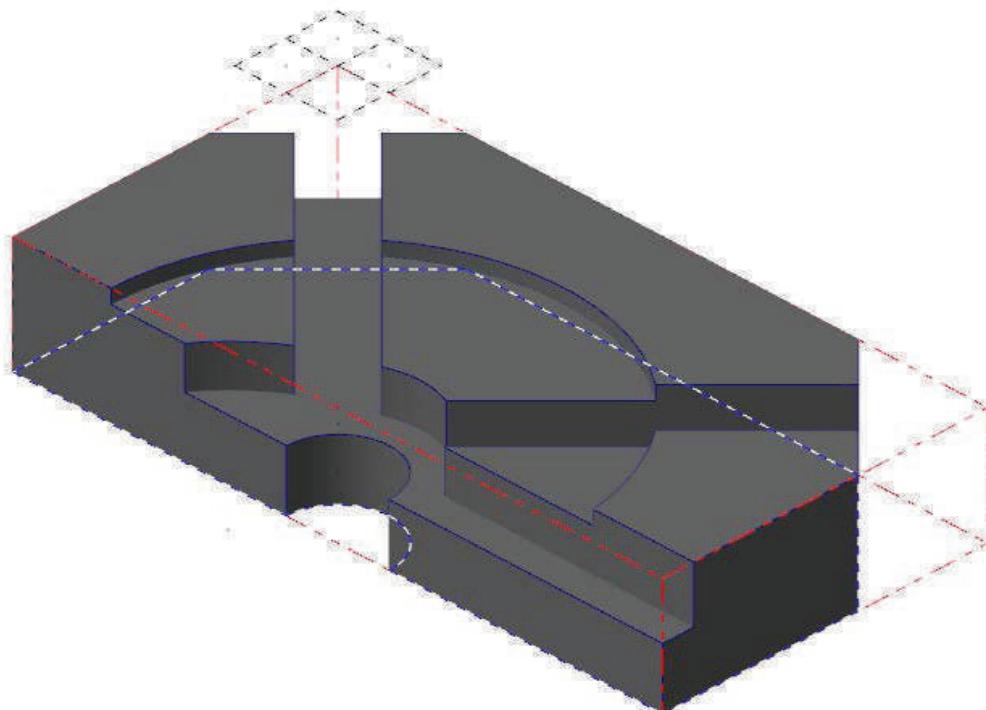


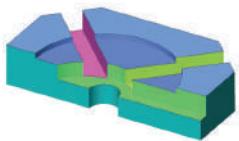
- ♦ [Solid Toolpath Chain: Select faces, edges, and/or loops]: Select the edge of the part as shown in [Figure: 11.1.1](#).

Figure: 11.1.1

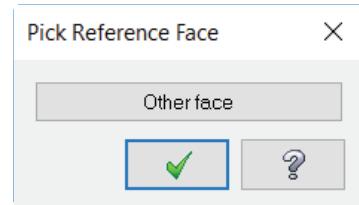


- ♦ The entire bottom of the part will be selected as shown.

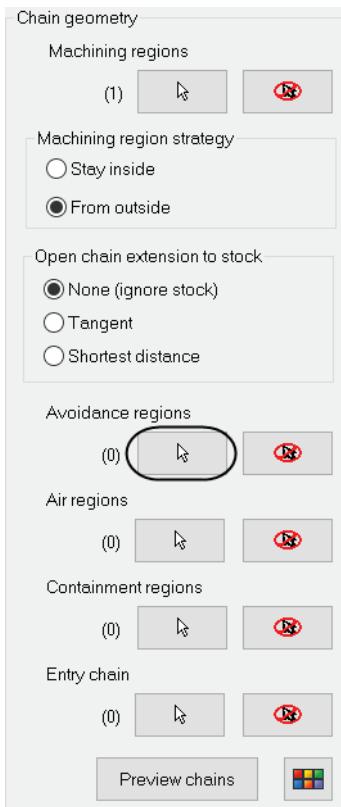


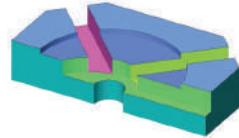


- The Pick Reference Face dialog will appear allowing you to change the face selected if needed.



- Select the **OK** button to continue.
- Select the **OK** button to exit Chaining.
- In the **Chain Options, Avoidance regions**, click on the **Select** button as shown.

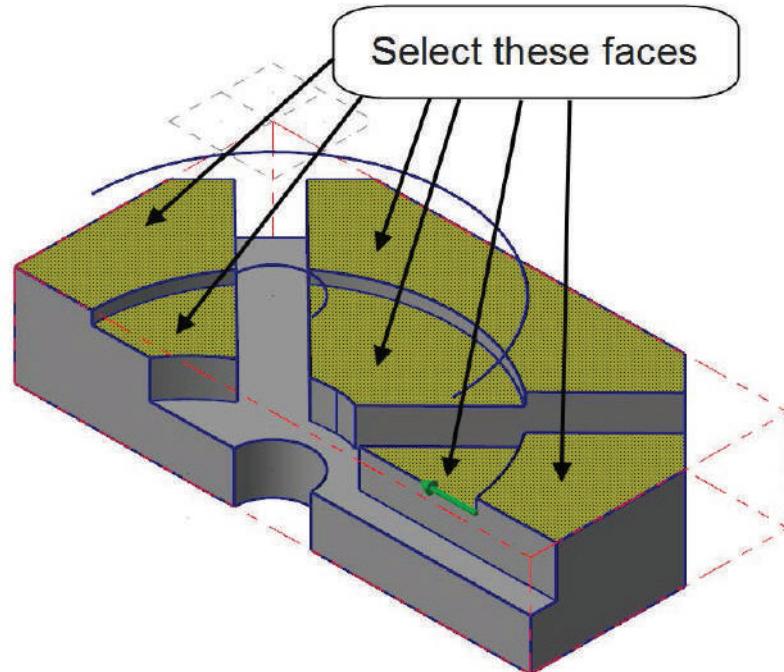




- ♦ In the **Chaining** dialog box, select **Solids** and enable the **Face** button only as shown.



- ♦ Select the top faces of the solid as shown.

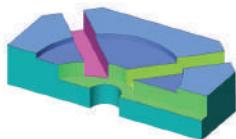


- ♦ Select the **OK** button to exit the **Chaining** dialog box.

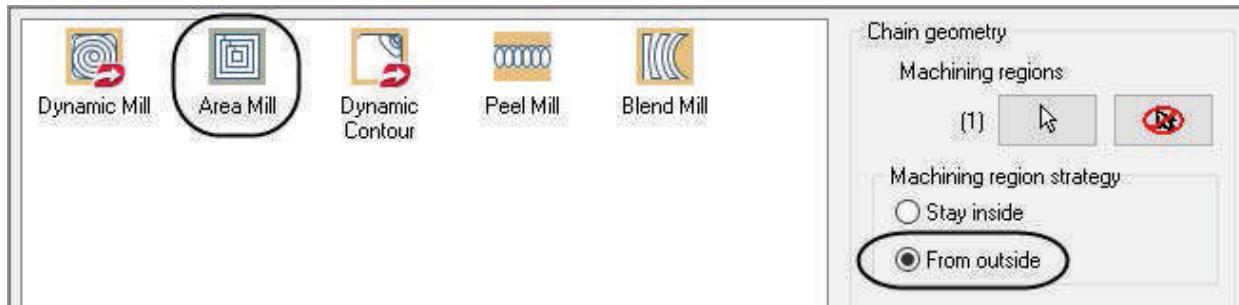


- ♦ Select the **OK** button to exit the **Chain Options** dialog box.



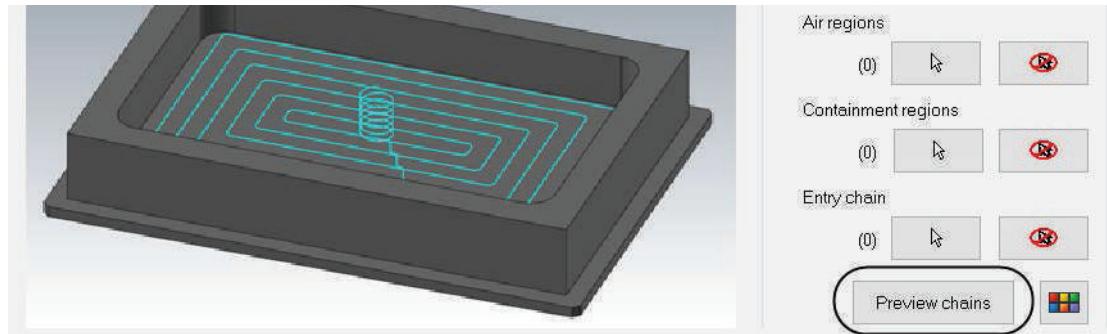


- On the **Toolpath Type** page, **Area Mill** will be selected and **From outside** enabled as shown.

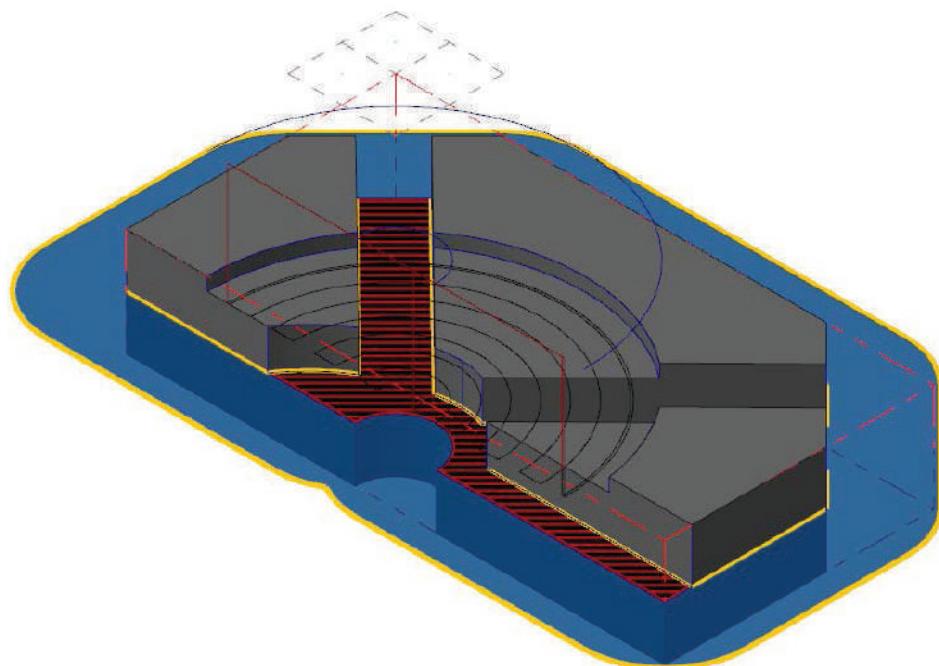


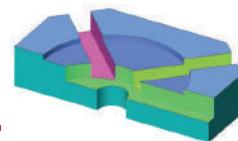
11.2 Preview Chains

- Select the **Preview chains** button as shown.



- Select the **Hide dialog** button to see the preview in the graphics window.
Press **Alt + S** to unshade the solid if needed. The **Preview chains** should look as shown.



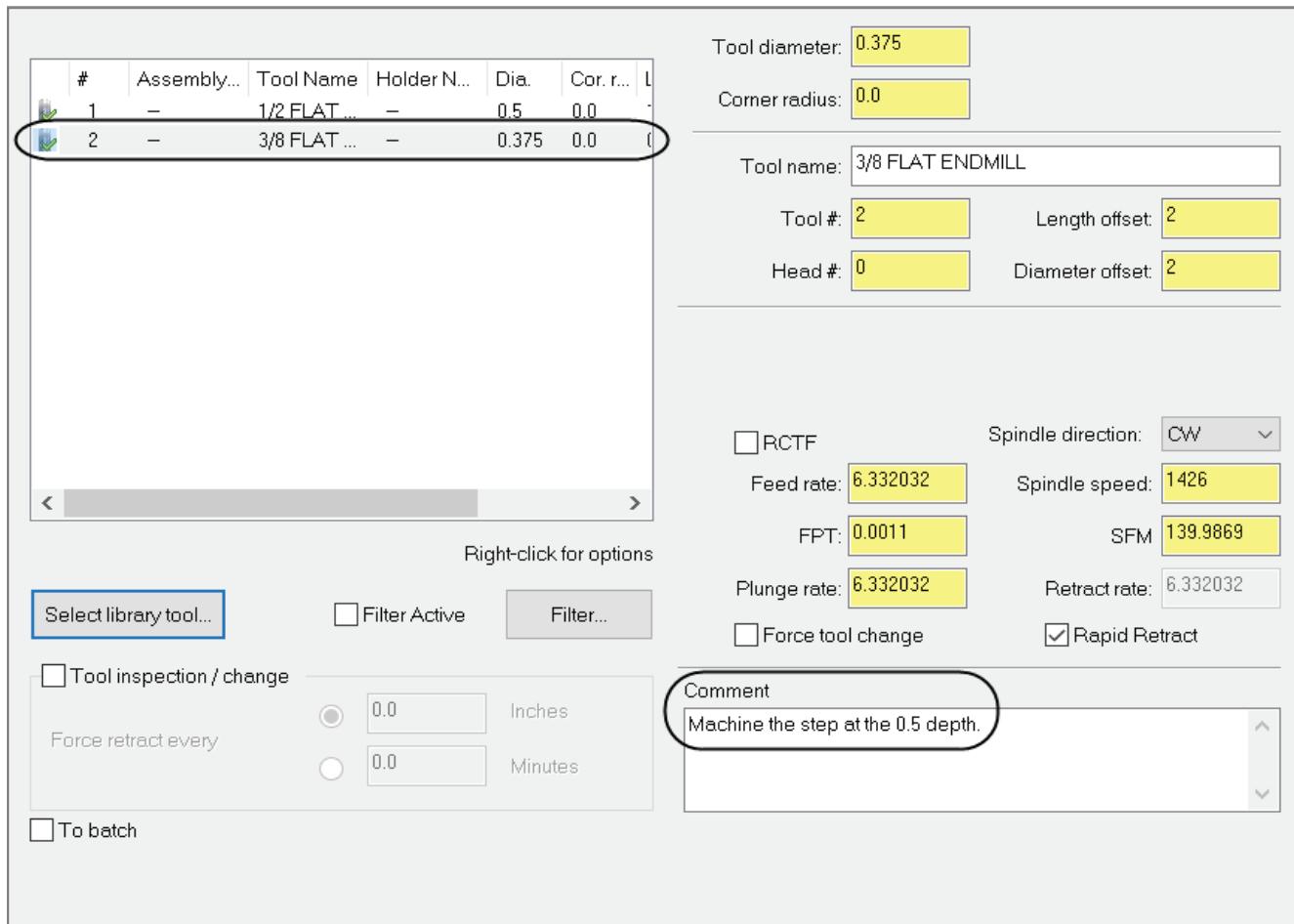


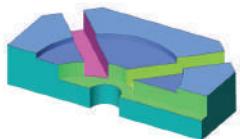
- ◆ Press **Esc** key to return to the toolpath parameters.
- ◆ Press **Alt + S** to return the solid to the shaded mode.
- ◆ Click on the **Preview chains** button again to clear the **Preview chains** display.



- ◆ Select the **Tool** page and make all the necessary changes as shown in [Figure: 11.2.1](#).
- ◆ From the existing list of tools, select the **3/8" Flat Endmill**.

Figure: 11.2.1

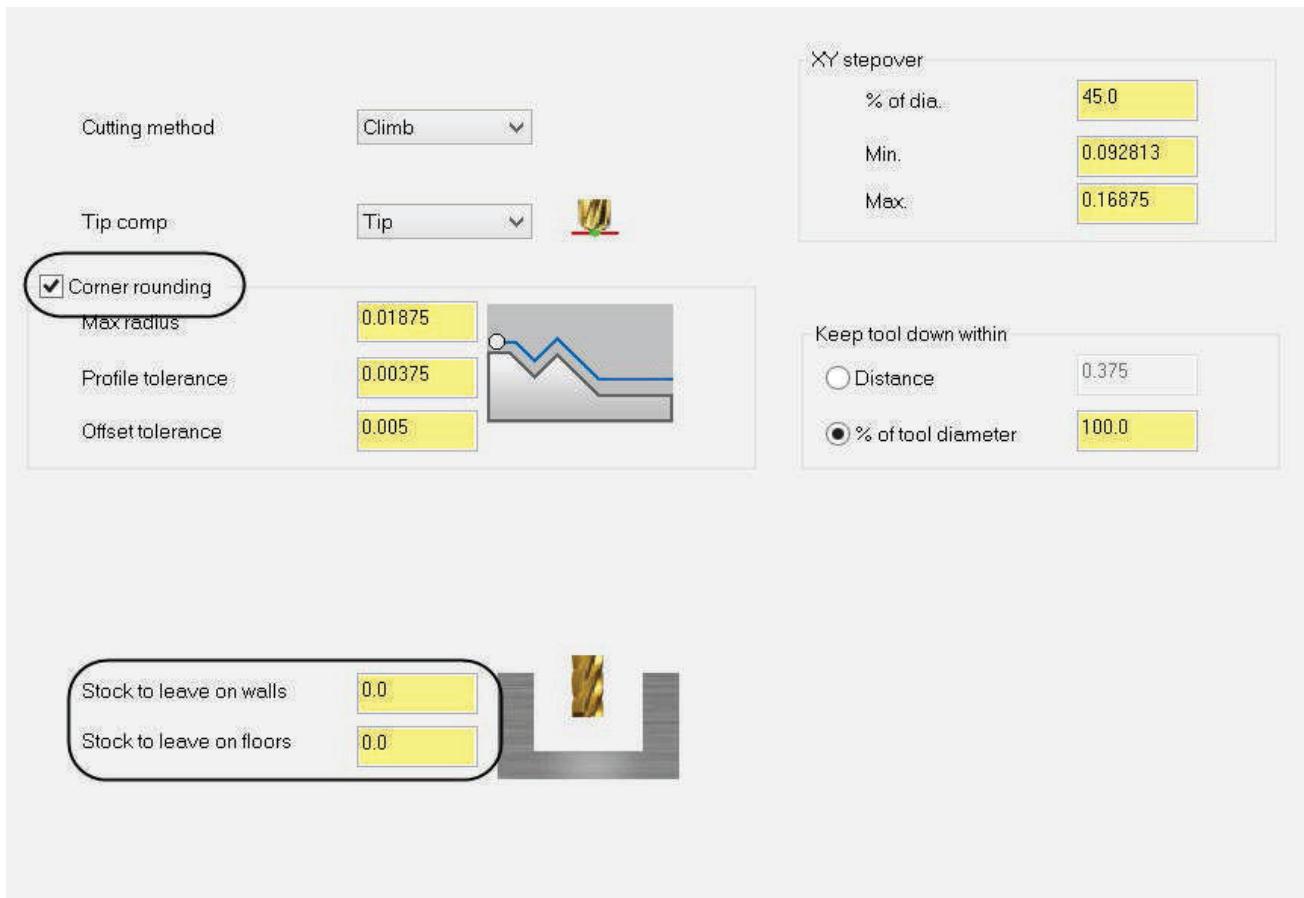


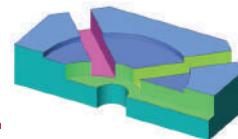


11.3 Set the Cut Parameters

- ◆ Choose **Cut Parameters** and change the parameters as shown in [Figure: 11.3.1](#).

Figure: 11.3.1

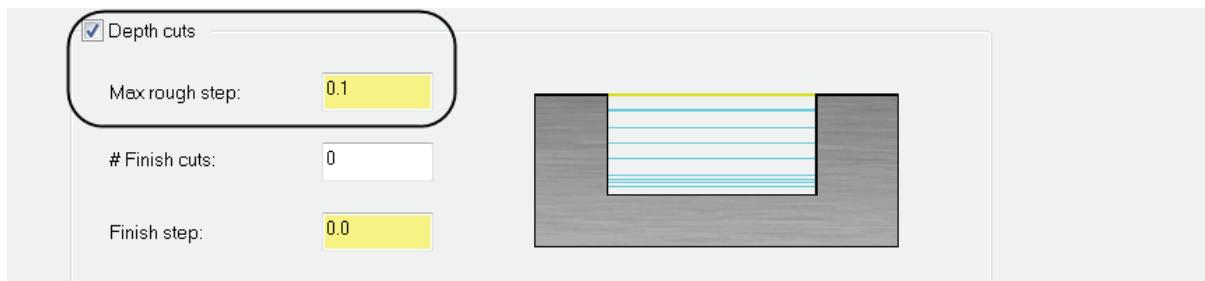




11.4 Set the Depth Cuts Parameters

- From the Tree View list, select the **Depth Cuts** parameters and enable **Depth cuts** as shown in [Figure: 11.4.1](#).

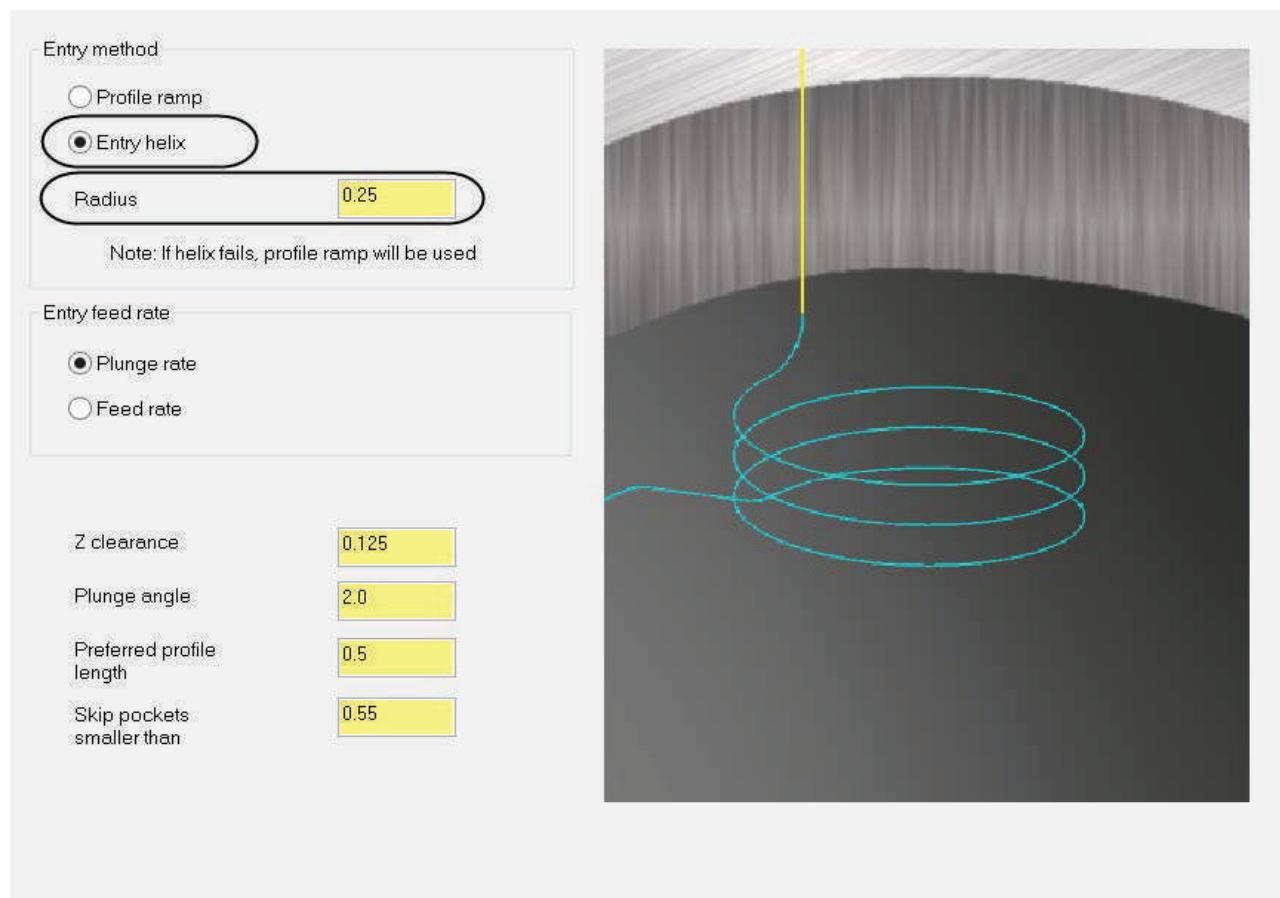
Figure: 11.4.1

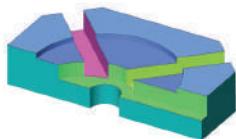


11.5 Set the Transitions

- Select **Transitions** and enable **Entry helix** and enter a **Radius** of **0.25** as shown in [Figure: 11.5.1](#).

Figure: 11.5.1

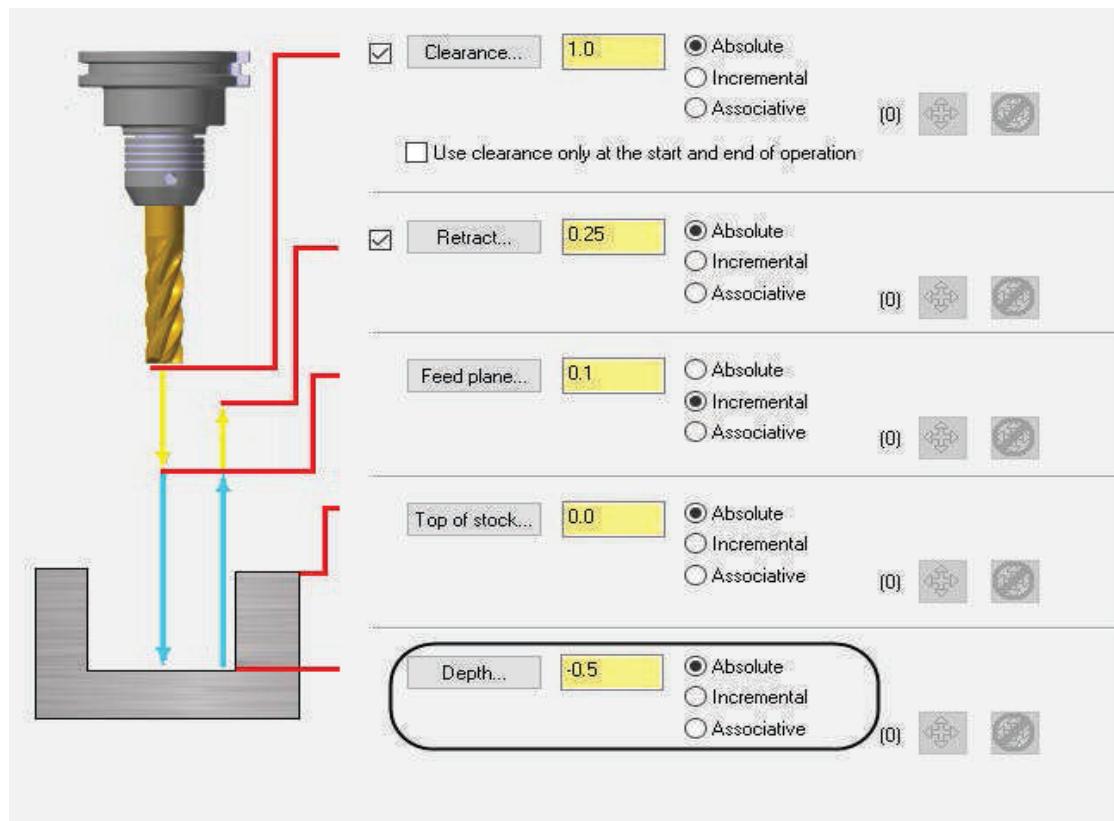




11.6 Set the Linking Parameters

- ♦ Select **Linking Parameters** and input a **Depth** of **-0.5** as shown in [Figure: 11.6.1](#).

Figure: 11.6.1



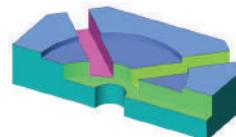
11.7 Preview the Toolpath

- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

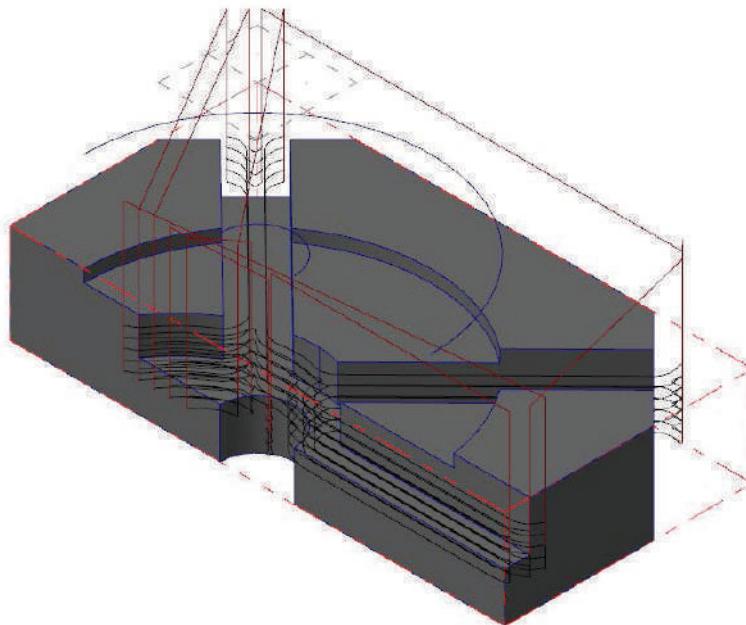


- ♦ Select the **Hide dialog** button to see the preview in the graphics window.





- ◆ The toolpath should look as shown.



- ◆ Press **Esc** key to exit the preview.

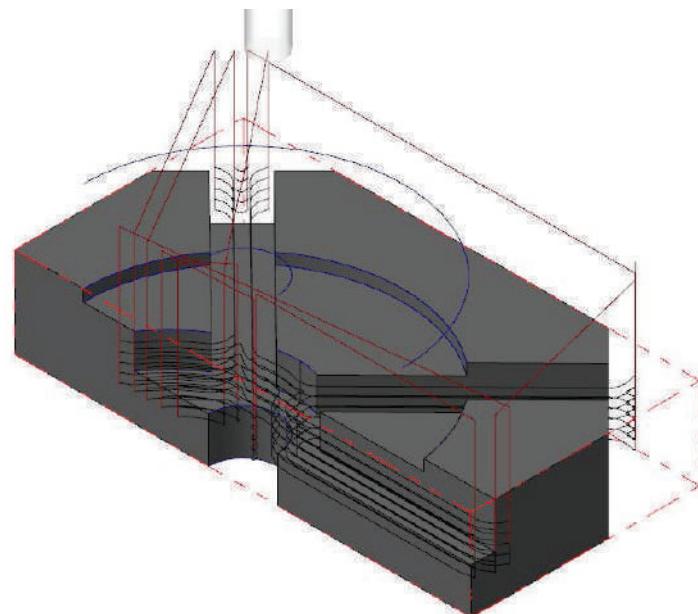
NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

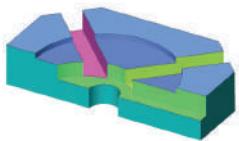
- ◆ Select the **OK** button to exit the **Area Mill** parameters.



11.8 Backplot the toolpath

- ◆ See page 586 to review **Backplot** procedure.

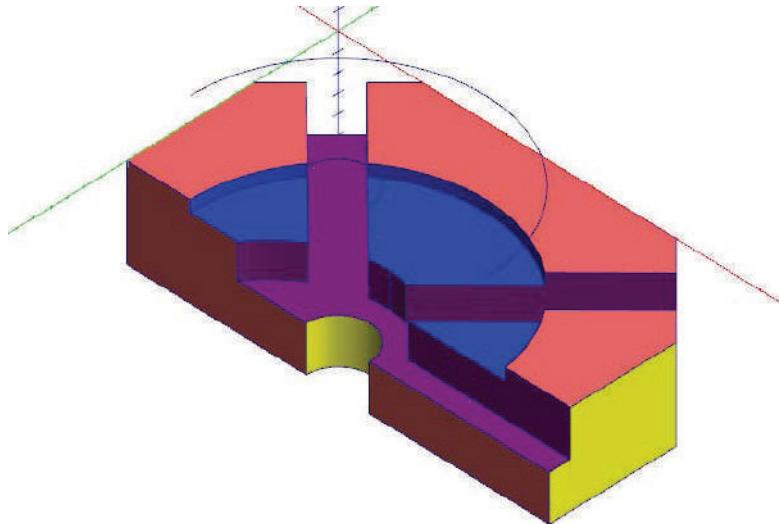




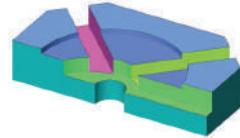
- ♦ Select the **OK** button to exit the **Backplot**.

11.9 Verify both toolpaths

- ♦ See **page 588** for more info.
- ♦ To verify all toolpaths, from the **Toolpaths Manager**, choose the **Select all operations** icon.



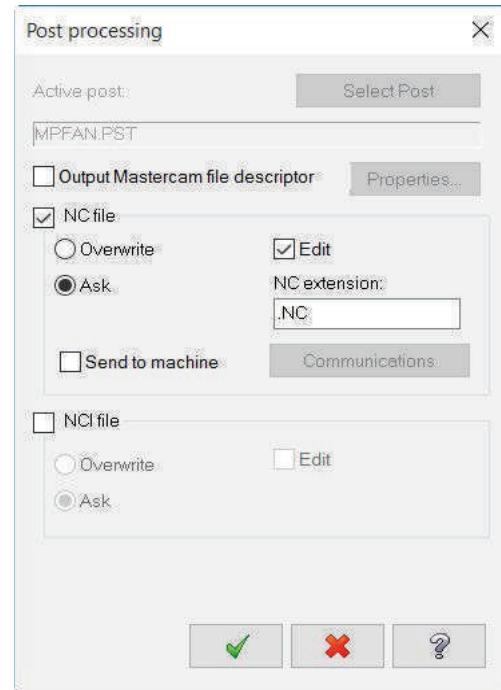
- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.



STEP 12: POST THE FILE

- ♦ Ensure all operations are selected; if they are not, use the **Select all operations** button in the **Toolpaths Manager**.
- ♦ Select the **Post selected operations** button from the **Toolpaths Manager**.
- ♦ In the **Post processing** window, make the necessary changes as shown in [Figure: 12.0.1](#).

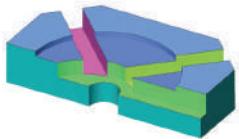
Figure: 12.0.1



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

Edit enabled allows you to automatically launch the default editor.

- ♦ Select the **OK** button to continue.
- ♦ Save your file and name it "Your Name_6.NC".



- ♦ A window with **Mastercam Code Expert** will be launched and the NC program will appear as shown.

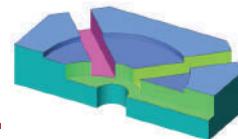
The screenshot shows the Mastercam Code Expert application window. The title bar reads "Tutorial #6.NC - Mastercam 2018 Code Expert". The menu bar includes File, Home, View, and NC Functions. The toolbar has icons for Insert Block Numbers, Insert Block Skip, Go To, Remove Block Numbers, Remove Block Skip, Remove Spaces, Remove Comments, Send File, Send, Receive, First, Previous, Next, Last, Mark, Tools, Multi-Stream, and NC Configuration. The main area displays the NC code for Tutorial #6.NC:

```
1   $  
2   O0006 (TUTORIAL #6)  
3   (DATE=DD-MM-YY - 14-02-17 TIME=HH:MM - 16:14)  
4   (NCX FILE = \\\LESSERVER06\BOOKCEN\PRODUCTION TUTORIALS\2018\TRAINING TUTORIALS\T CAD IMPORT MILL ESSENTIALS TOOLPATHS\NCX FILES\INCH\TUTORIAL #6.NC  
5   (NC FILE - C:\USERS\NINGYUE.MDO\DESKTOP\TUTORIAL #6.NC)  
6   (MATERIAL - ALUMINUM INCH - 2024)  
7   ( T1 | 1/2 FLAT ENDMILL | H1 )  
8   ( T2 | 3/8 FLAT ENDMILL | H2 )  
9   N100 G20  
10  N110 G0 G17 G40 G49 G80 G90  
11  ( MACHINE THE MATERIAL SURROUNDING THE PART USING DYNAMIC MILL. )  
12  N120 T1 M6  
13  N130 G0 G90 G84 X-.0579 Y-3.2088 A0. S1069 M3  
14  N140 G43 H1 Z1.  
15  N150 Z1.  
16  N160 G1 Z-.99 F6.42  
17  N170 X-.1522 Y-2.8595  
18  N180 X-.2055 Y-1.8293 Z-.9941  
19  N190 X-.2222 Y-2.7563 Z-.997  
20  N200 X-.2328 Y-2.7656 Z-.9985  
21  N210 X-.2475 Y-2.7475 Z-1.  
22  N220 X-.2594 Y-2.7311  
23  N230 X-.2705 Y-2.715  
24  N240 X-.2815 Y-2.6967  
25  N250 X-.2919 Y-2.6766  
26  N260 X-.3013 Y-2.655  
27  N270 X-.3093 Y-2.6324  
28  N280 X-.3152 Y-2.6111  
29  N290 X-.3195 Y-2.5917  
30  N300 X-.3225 Y-2.5574  
31  N310 X-.3223 Y-2.5261  
32  N320 X-.3303 Y-2.4974  
33  N330 X-.3313 Y-2.47  
34  N340 X-.3318 Y-2.444  
35  N350 X-2.4189  
36  N360 X-.3316 Y-2.3944  
37  N370 X-.3313 Y-2.3703  
38  N380 X-.3308 Y-2.3664  
39  N390 X-.3303 Y-2.3527  
40  N400 X-.3297 Y-2.2991  
41  N410 X-.329 Y-2.2755  
42  N420 X-.3284 Y-2.252  
43  N430 X-.3277 Y-2.2284  
44  N440 X-.327 Y-2.2049  
45  N450 X-.3264 Y-2.1514  
46  N460 X-.3257 Y-2.1579
```

At the bottom, there is a "Find Extents" button and a status bar showing "Ready", "80.67KB", "100%", and zoom controls.

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

STEP 13: SAVE THE UPDATED MCAM FILE

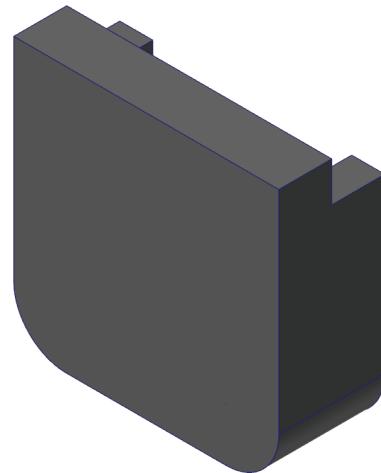


REVIEW EXERCISE - STUDENT PRACTICE

IMPORT THE GEOMETRY FOR TUTORIAL #6 EXERCISE

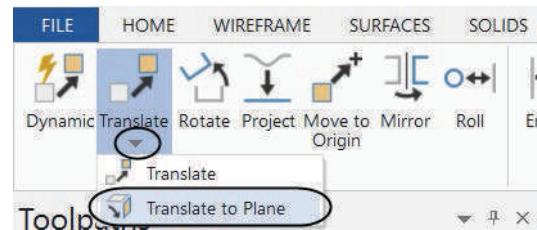
Import CAD Model and prepare it for machining.

- ◆ Open the file from SolidWorks.

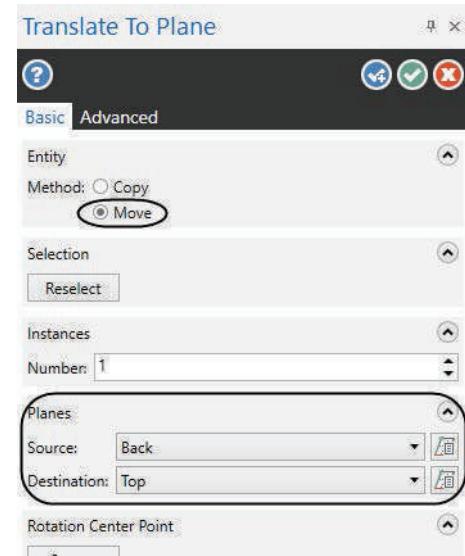


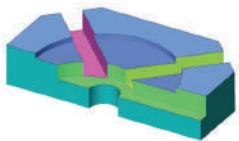
Rotate between views.

- ◆ Use TRANSFORM/Translate to Plane to rotate the part between views.

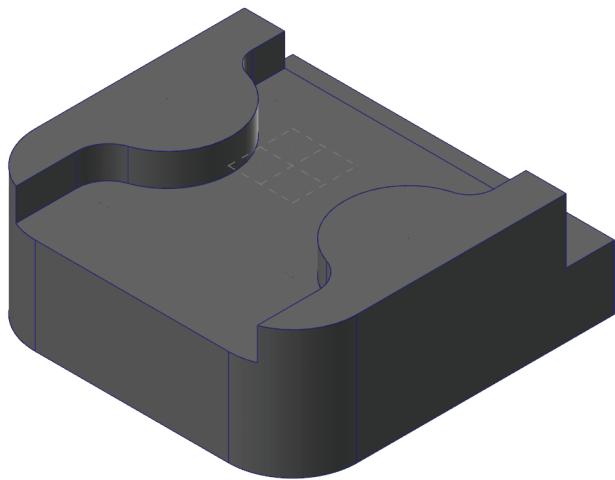


- ◆ Ensure Move is enabled. Set the Source to Back and the Destination to Top as shown.

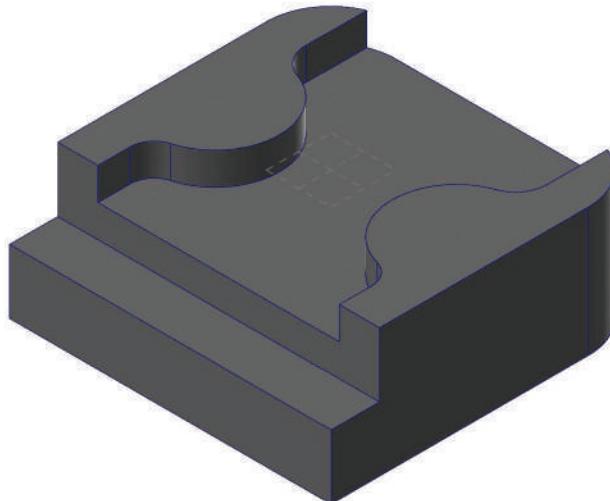




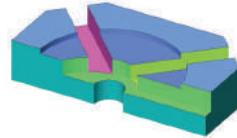
- The geometry should look as shown.

**Rotate the part in top view.**

- Use **TRANSFORM/Rotate** to rotate the part 180 degrees.

**Move to Origin**

- Use **TRANSFORM/Move to Origin** to move the upper left corner to the **Origin**.
- Press **Space** and enter the coordinates **-2.5, 2.5**.



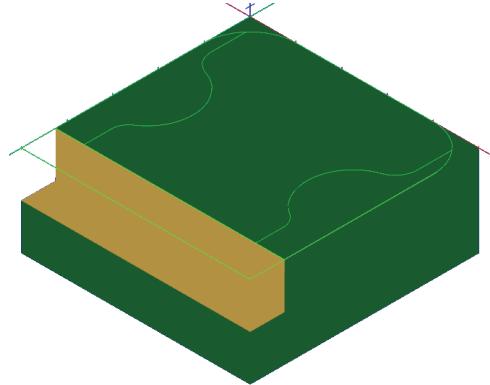
CREATE THE TOOLPATHS FOR TUTORIAL #6 EXERCISE

Create the Toolpaths for Tutorial #6 Exercise as per the instructions below.

Set the machine properties including the Stock Setup.

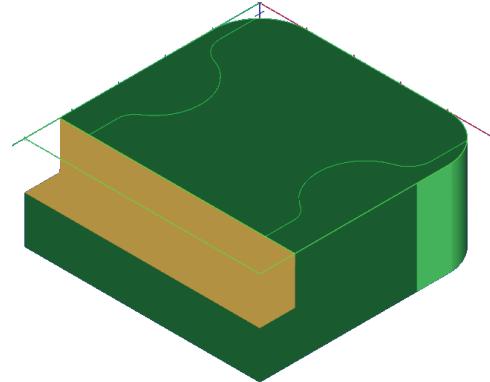
Remove the material on the step using Contour (2D).

- ◆ Use a **7/8" Flat Endmill**.
- ◆ Based on your chaining direction, ensure the **Compensation direction** is set correctly.
- ◆ Enable **Depth Cuts**.
- ◆ **Lead In/Out**, ensure the **Arc Radius** is set to zero.
- ◆ **No Break Through, Multi Passes**.
- ◆ Set the **Depth** according to the part.



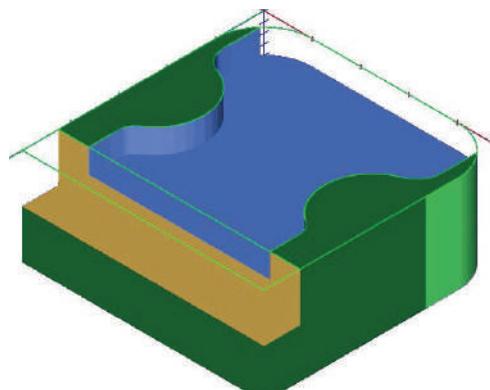
Remove the material around the fillets using Contour (2D).

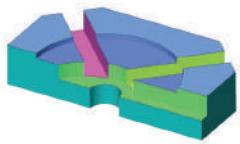
- ◆ Use the **7/8" Flat Endmill**.
- ◆ Based on your chaining direction, ensure the **Compensation direction** is set correctly.
- ◆ Enable **Depth Cuts**.
- ◆ Set a **Lead In/Out and Break Through**.
- ◆ **No Multi Passes**.
- ◆ Set the **Depth** according to the part.



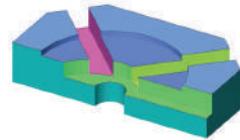
Create a 2D High Speed Blend toolpath to remove the material in the center of the part.

- ◆ Select the two chains.
- ◆ Use a **1/2" Flat Endmill**.
- ◆ Select **Along** for the tool cutting direction.
- ◆ Set **% of stepover to 25.0**.
- ◆ Set the **Compensation direction to Inside**.
- ◆ **Extend Exit/Entry 0.5"**
- ◆ **Stock to leave on walls/floors = 0.0**.
- ◆ Disable **Depth Cuts** and **Break Through**.
- ◆ Set the **Depth** according to the part.
- ◆ Your part will appear as shown.





NOTES:



TUTORIAL #6 QUIZ

- ◆ What does the Analyze Position command do?
 - ◆ How does a Blend Mill toolpath work?
 - ◆ What does the Along parameter in the Blend Mill toolpath do?

TUTORIAL #7





OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

Import the CAD Model and prepare it to generate Toolpaths from:

- ◆ The student will open the SolidWorks file in Mastercam.
- ◆ The student will use Translate 3D to rotate the part in the proper plane.
- ◆ The student will use Translate to rotate the part in the Top plane.

Create the necessary Toolpaths to machine the part:

- ◆ The student will set up the stock size to be used and the clamping method used.
- ◆ The 2D HS Dynamic Area Mill toolpath will be used to machine the pockets.
- ◆ The Feature Based Drill toolpath will be used to machine the four holes.
- ◆ The 2D HS Dynamic Mill toolpath will be used to machine the outside profile.
- ◆ The Contour toolpath will be used to finish the outside profile.

Backplot and Verify the file:

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

Post Process the file to generate the G-code:

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



This tutorial takes approximately one hour to complete.



GEOMETRY CREATION

STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

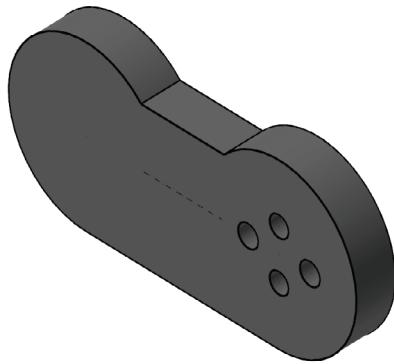
Please refer to the **Getting Started** section to set up the graphical user interface.

STEP 2: IMPORTING THE SOLIDWORKS FILE GEOMETRY

Mastercam lets you read (import) a variety of CAD file types into the Mastercam database. You can also write (export) Mastercam files to a variety of different file formats.

To import a SolidWorks file in Mastercam, you have to use the Open function and then select SolidWorks files from the list of file types.

Step Preview:

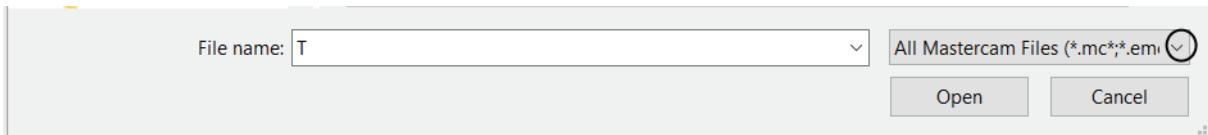


Download the files from www.emastercam.com/trainingfiles.

Save the file at a preferred location.

FILE

- ♦ **Open.**
- ♦ In the file name extension, click on the drop down arrow as shown.





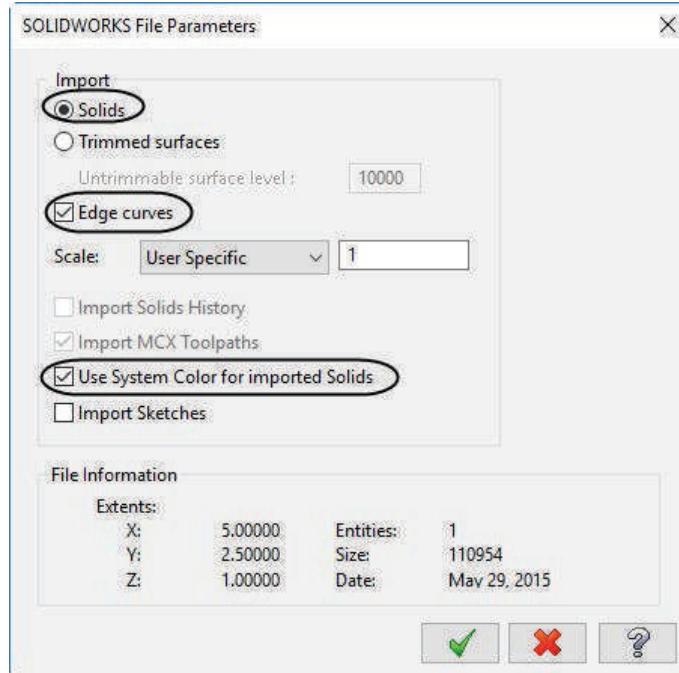
- From the list of file types, select **SolidWorks Files (*.sldprt;*.sldasm)** as shown.

Mastercam Files (*.mcam)
 Mastercam X Files (*.mcx*)
 Mastercam Edu X Files (*.emcx*)
 All Mastercam Files (*.mc*,*.emc*)
 IGES Files (*.igs;*.iges)
 AutoCAD Files (*.dwg;*.dxr;*.dxf;*.dwf;*.dwfx)
 Parasolid Files (*.x_t;*.x_b;*.xmt_txt)
 ProE/Creo Files (*.prt;*.asm;*.prt.*;*.asm.*)
 ACIS Kernel SAT Files (*.sat;*.sab)
 STEP Files (*.stp;*.step)
 VDA Files (*.vda)
 Rhino 3D Files (*.3dm)
SOLIDWORKS Files (*.sldprt;*.sldasm;*.slddrw)

- Find and select **TUTORIAL #7.sldprt**.
- Click on the **Options** button.



- Leave the **Solids** enabled to import the file as a solid and enable **Edge curves** for Mastercam to automatically create curves at the edges of the solid. To better see the curves, enable also **Use System Color for imported Solids** as shown.

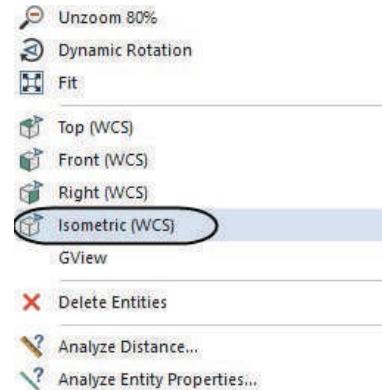


- Select the **OK** button to exit the **SOLIDWORKS File Parameters** dialog box.
- Open the file.

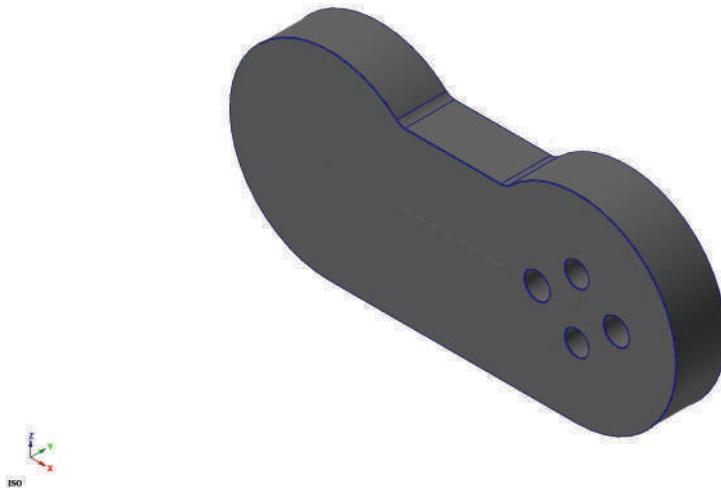




- ♦ Right mouse click in the graphics window and select the **Isometric** view.



- ♦ Press **Alt + F1** to fit the geometry in the graphics window.
- ♦ The geometry should look as shown.



NOTE: In preparation to add toolpaths to the part, the geometry has to be oriented such that the part sits on the machine table. In the next step, you will rotate the part using the **Translate 3D** command.



STEP 3: ORIENT THE PART USING TRANSLATE 3D

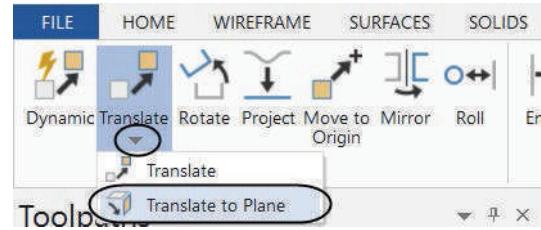
Translate 3D allows you to move the geometry between views (from one plane to another). In this case, you will move the Back plane to Top.

Step Preview:

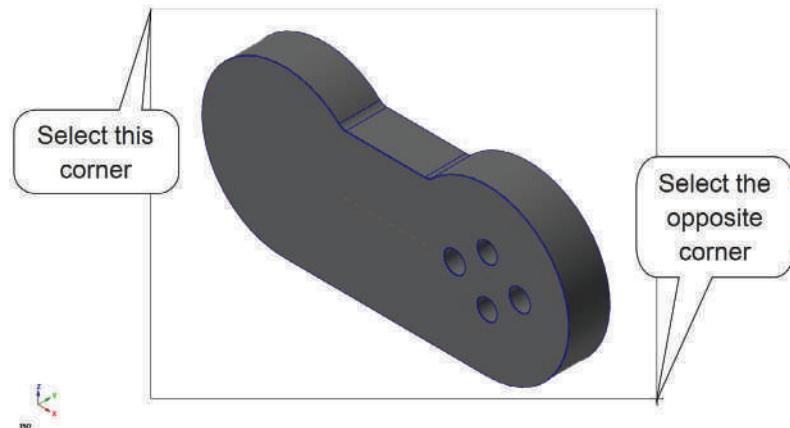


TRANSFORM

- From the **Position** group, click on the drop down arrow under **Translate** and select **Translate to Plane** as shown.

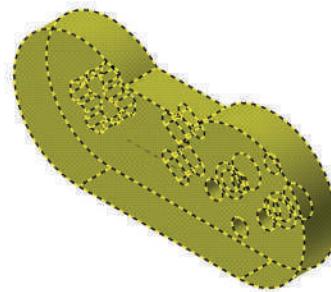


- [Translate: select entities to translate]: Make a window around the solid as shown.



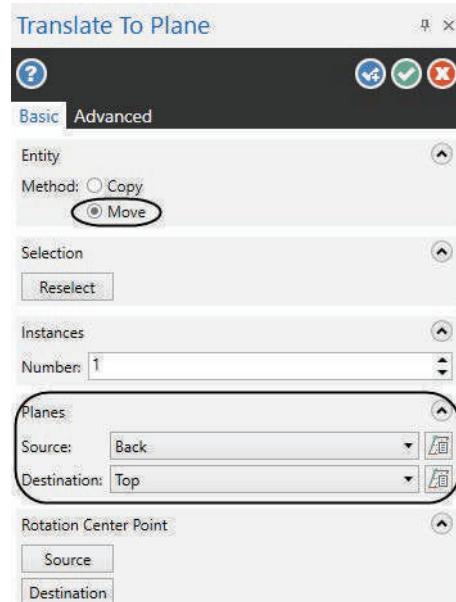


- The entire geometry should be selected as shown.

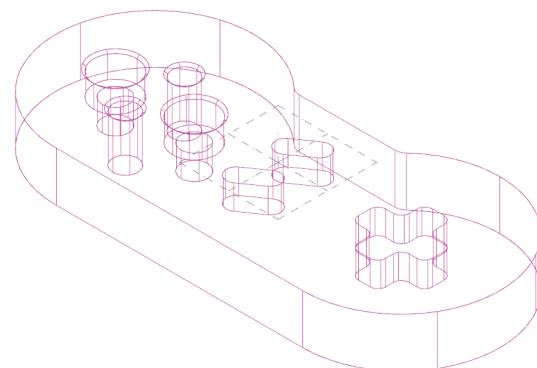


- Click on the **End Selection** button or press **Enter** to finish the selection.
- In the **Translate To Plane** panel, enable **Move** and change the **Source View to Back** and leave the **Destination View to Top** as shown in [Figure: 3.0.1](#).

Figure: 3.0.1



- The geometry preview should look as shown.



- Select the **OK** button to exit **Translate To Plane**.





STEP 4: ROTATE THE PART

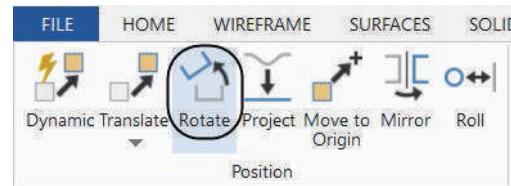
In this step, you will learn how to rotate the part about the origin 180 degrees in the Top plane.

Step Preview:



TRANSFORM

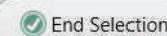
- From the **Position** group, select **Rotate** as shown.



- [Select entities]: Make a window around all entities as shown.



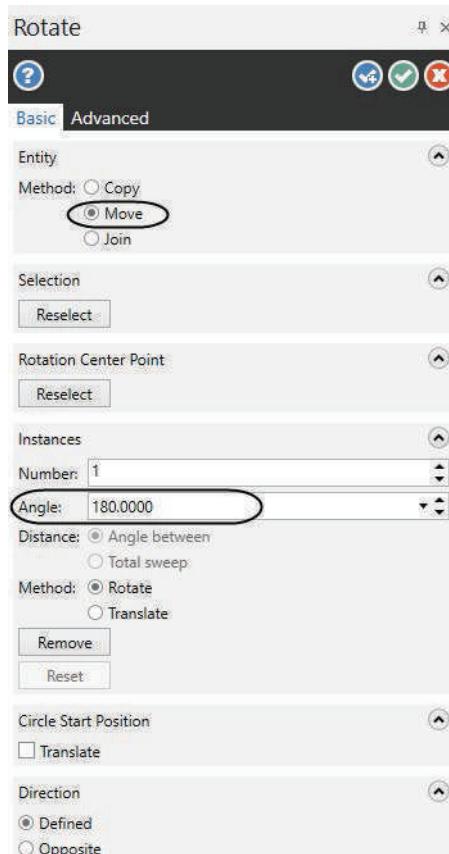
- Click on the **End Selection** button or press **Enter** to finish the selection.



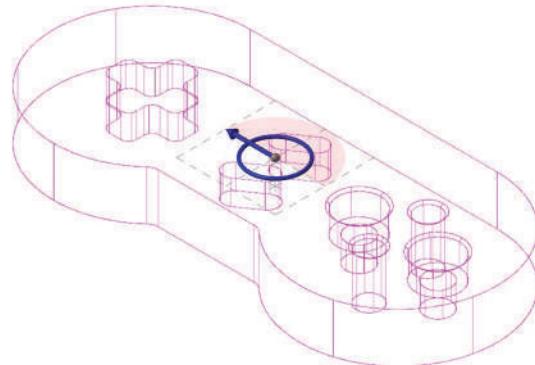


- In the Rotate panel, enable Move and change the angle to 180 as shown in [Figure: 4.0.1](#).

Figure: 4.0.1



- The preview should look as shown.

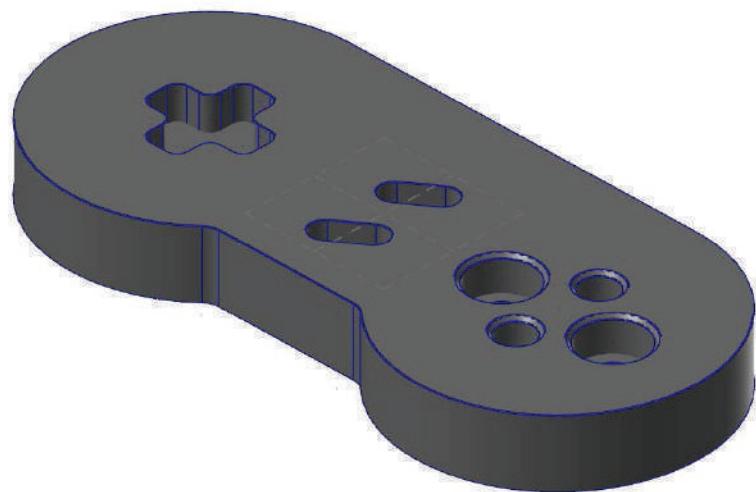


- Select the OK button to exit Rotate.
- Right mouse click on the graphics window and select Clear Colors as shown to remove the result color.





- ♦ The geometry should look as shown.



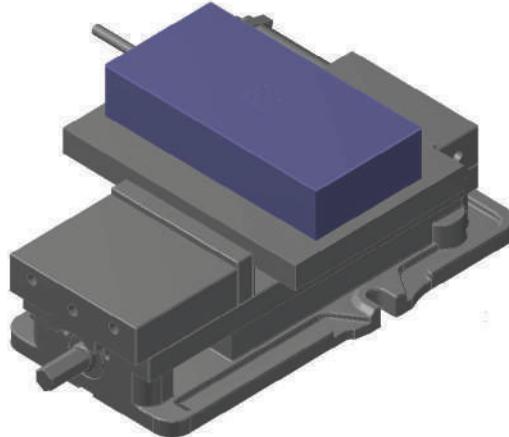
STEP 5: SAVE THE FILE

- ♦ Save As.
- ♦ File name: "Your Name_7."



TOOLPATH CREATION

SUGGESTED FIXTURE:

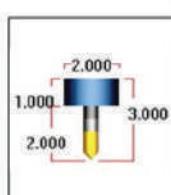


SETUP SHEET

TOOL LIST

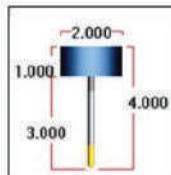
	TYPE: Endmill1 Flat DIA OFFSET: 1 HOLDER: DEFAULT HOLDER NUMBER: 1 LENGTH OFFSET: 1 #1 - 0.1563 ENDMILL1 FLAT - 5/32 FLAT ENDMILL	FLUTE LENGTH: 0.375 OVERALL LENGTH: 2.25 CORNER RAD: 0.0 # OF FLUTES: 4
	TYPE: Spot Drill DIA OFFSET: 2 HOLDER: DEFAULT HOLDER NUMBER: 2 LENGTH OFFSET: 2 #2 - 0.5000 SPOT DRILL - 1/2 SPOTDRILL	FLUTE LENGTH: 2.0 OVERALL LENGTH: 3.0 CORNER RAD: 0.0 # OF FLUTES: 2
	TYPE: CSink DIA OFFSET: 3 HOLDER: DEFAULT HOLDER NUMBER: 3 LENGTH OFFSET: 3 #3 - 0.3300 CSINK -	FLUTE LENGTH: 1.0 OVERALL LENGTH: 2.0 CORNER RAD: 0.0 # OF FLUTES: 4
	TYPE: CSink DIA OFFSET: 4 HOLDER: DEFAULT HOLDER NUMBER: 4 LENGTH OFFSET: 4 #4 - 0.3300 CSINK -	FLUTE LENGTH: 1.0 OVERALL LENGTH: 2.0 CORNER RAD: 0.0 # OF FLUTES: 4

TUTORIAL #7



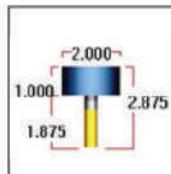
TYPE: CSink
DIA: OFFSET: 6
HOLDER: DEFAULT HOLDER
NUMBER: 6
LENGTH OFFSET: 6
#6 - 0.5363 CSINK -

FLUTE LENGTH: 1.0
OVERALL LENGTH: 2.0
CORNER RAD: 0.0
OF FLUTES: 4



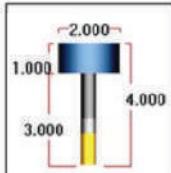
TYPE: Drill
DIA: OFFSET: 7
HOLDER: DEFAULT HOLDER
NUMBER: 7
LENGTH OFFSET: 7
#7 - 0.2500 DRILL - 1/4 DRILL

FLUTE LENGTH: 0.75
OVERALL LENGTH: 3.0
CORNER RAD: 0.0
OF FLUTES: 2



TYPE: Endmill Flat
DIA: OFFSET: 8
HOLDER: DEFAULT HOLDER
NUMBER: 8
LENGTH OFFSET: 8
#8 - 0.4375 ENDMILL FLAT -

FLUTE LENGTH: 1.375
OVERALL LENGTH: 1.875
CORNER RAD: 0.0
OF FLUTES: 4



TYPE: Endmill Flat
DIA: OFFSET: 9
HOLDER: DEFAULT HOLDER
NUMBER: 9
LENGTH OFFSET: 9
#9 - 0.5000 ENDMILL FLAT - 1/2 FLAT ENDMILL

FLUTE LENGTH: 1.0
OVERALL LENGTH: 3.0
CORNER RAD: 0.0
OF FLUTES: 4



STEP 6: SELECT THE MACHINE AND SET UP THE STOCK

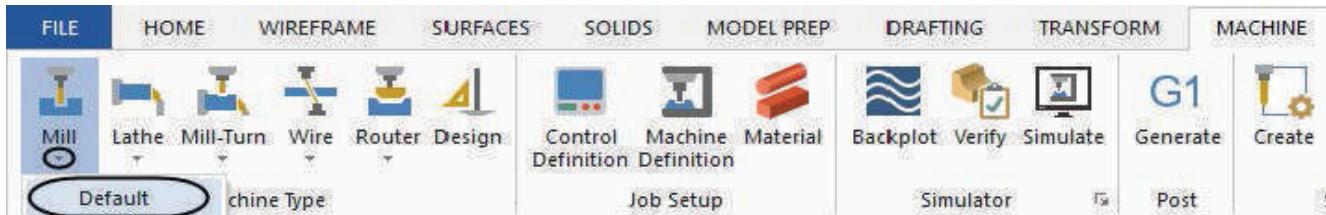
In Mastercam, you select a **Machine Definition** before creating any toolpath. The **Machine Definition** is a model of your machine's capabilities and features. It acts like a template for setting up your machine. The machine definition ties together three main components: the schematic model of your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

NOTE: For the purpose of this tutorial, we will be using the **Default Mill** machine. Select the **Mill Default** only if there is no **Machine Group** in the **Toolpaths Manager**.

- ♦ Press **Alt + F1** to fit the geometry to the screen.

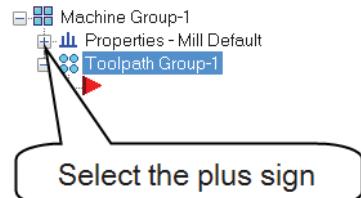
MACHINE

- ♦ From the **Machine Type** group, click on the drop down arrow below **Mill** and select the **Default**.

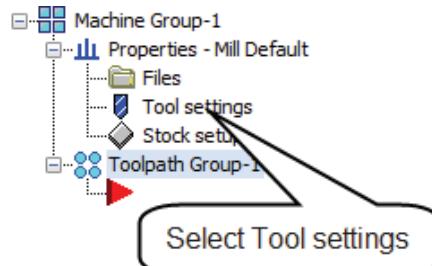


NOTE: Once you select the **Mill Default**, the ribbon changes to reflect the toolpaths that could be used with **Mill Default**.

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



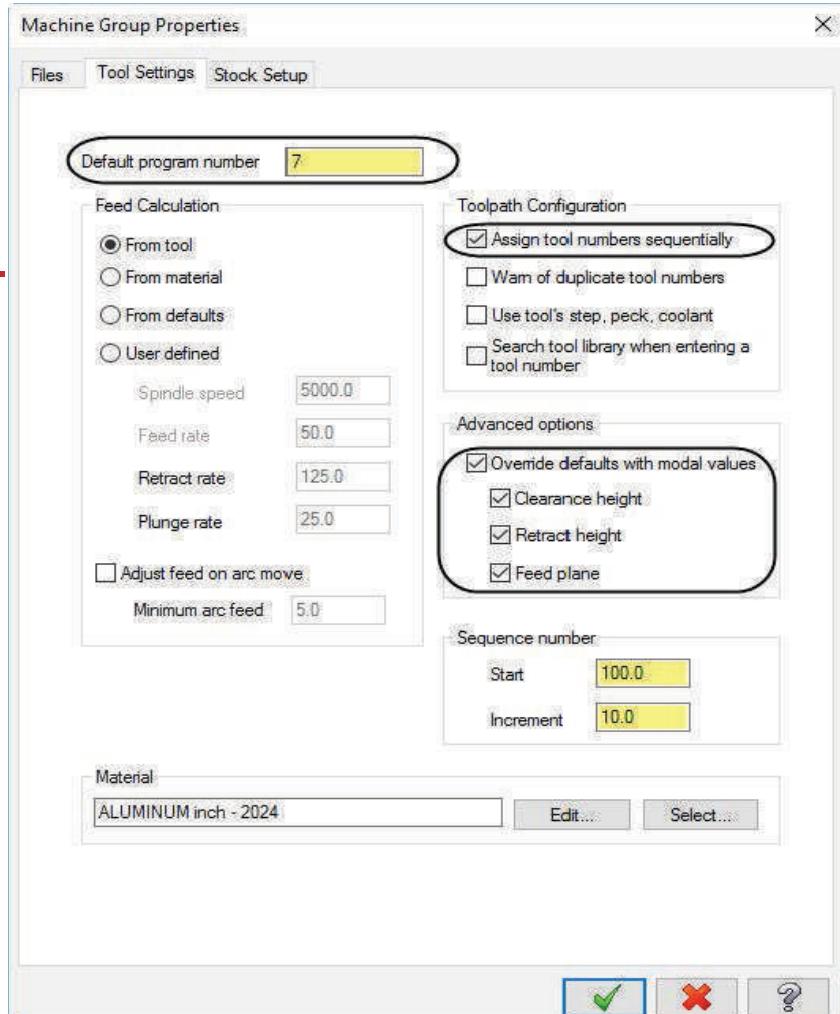
- ♦ Select **Tool settings** to set the tool parameters.





- Change the parameters to match the screenshot as shown in [Figure: 6.0.1](#).

Figure: 6.0.1



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

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

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

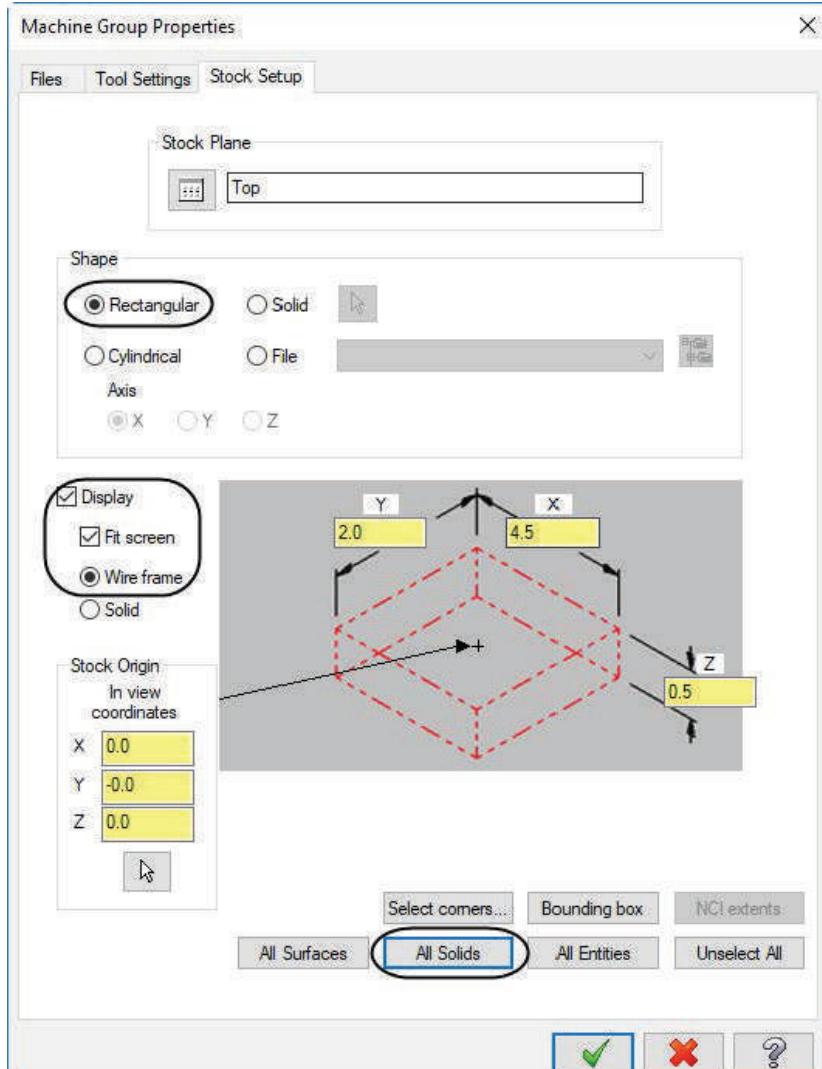
Override defaults with modal values enables the system to keep the values that you enter.

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



- ♦ Select the **Stock Setup** tab to define the stock.
- ♦ Pick the **Rectangular** shape option.
- ♦ Choose the **All Solids** button and the stock size will be input as shown in [Figure: 6.0.2](#).

Figure: 6.0.2



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

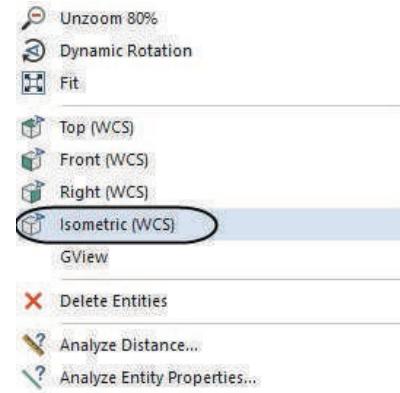
Display options allow you to set the stock as **Wireframe** and to fit the stock to the screen. (Fit Screen)

NOTE: The stock model that you create is used when viewing the file or the toolpaths, during backplot, or while verifying toolpaths. In the graphics, the plus shows you where the stock origin is. The default position is the middle of the stock.

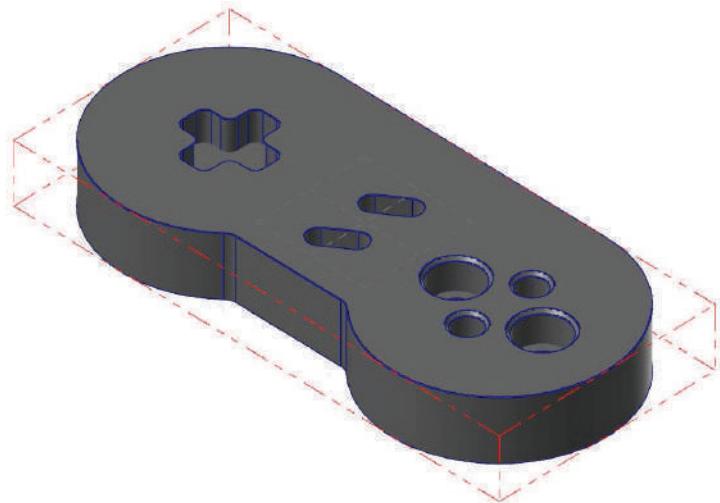
- ♦ Select the **OK** button to exit **Machine Group Properties**.



- ♦ Right mouse click in the graphics window and select the **Isometric** view to see the stock.



- ♦ Press **Alt + F1** to fit the geometry to the screen.
- ♦ The stock model should appear as shown.



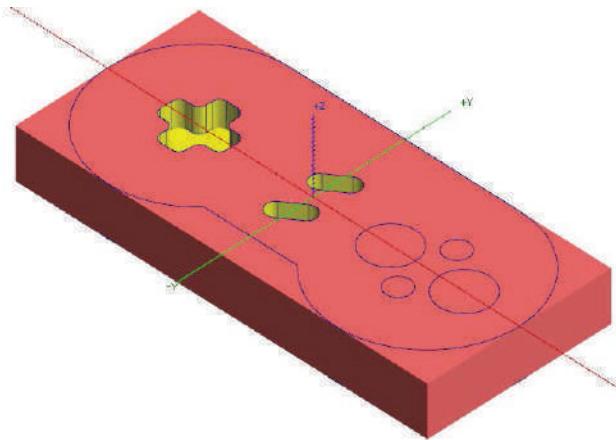
NOTE: You may not be able to see the stock very clearly due to the fact that the stock is the same size as the part. The stock is not geometry and cannot be selected.



STEP 7: 2D HIGH SPEED DYNAMIC MILL

2D High Speed Dynamic Mill utilizes entire flute lengths of cutting tools to produce the smoothest, most efficient tool motion for high speed pocketing. The toolpath supports a custom entry method and many others. Micro lifts further refine the dynamic milling motion and avoid excessive heat build up. Custom feeds and speeds optimize and generate safe tool motion. **Dynamic Mill** machines pockets, material that other toolpaths leave behind, and standing bosses or cores. The toolpath depends on the **Machining strategy** that you choose in the **Chain Options**. The outside chain contains the toolpath; all inside chains are considered islands.

Toolpath Preview:

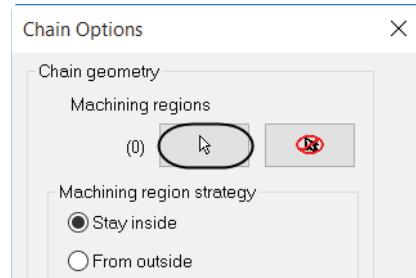


TOOLPATHS

- From the **2D** group, select **Dynamic Mill** as shown.

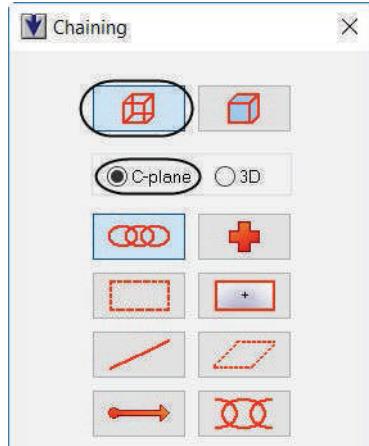


- From the **Chain Options** dialog box, click on the **Select** button in the **Machining regions** as shown.

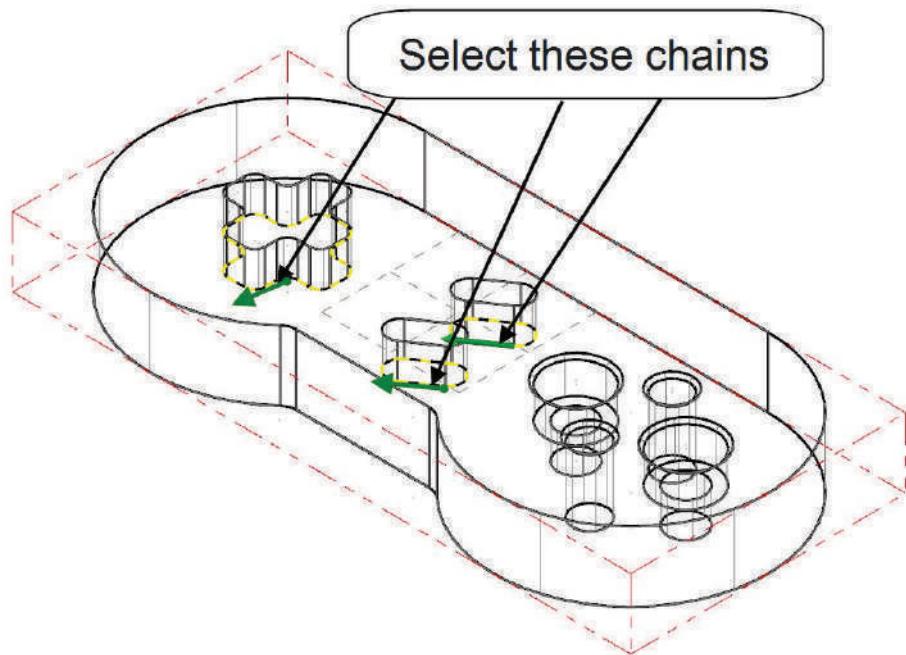




- In the Chaining dialog box, make sure that **Wireframe** selection is selected and then enable **C-plane** as shown.



- Press **Alt + S** if needed, to display the solid in unshaded mode.
- Select the bottom of the pockets as shown.

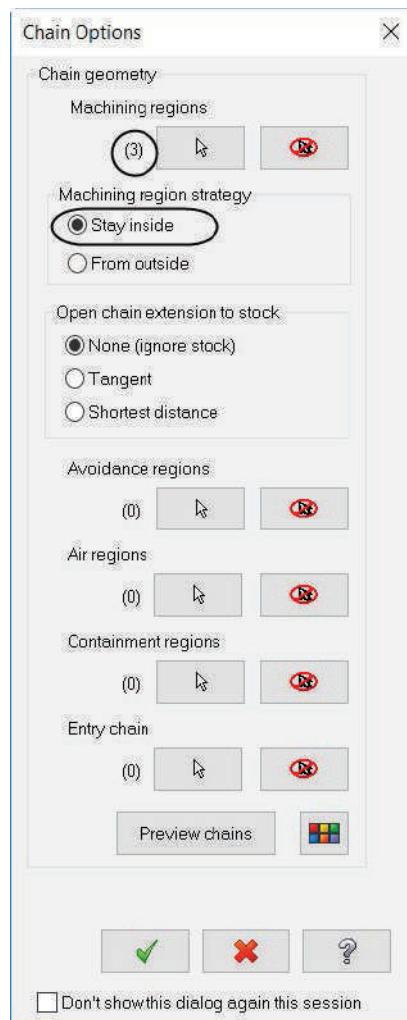


- Select the **OK** button to exit the Chaining dialog box.





- In the **Chain Options** dialog box, **Machining regions** will have 3 chains and make sure that **Stay inside** is enabled as shown.



- Select the **OK** button to exit the **Chain Options** dialog box.
- In the **Toolpath Type** page, **Dynamic Mill** will be already selected as shown in [Figure: 7.0.1](#).

Figure: 7.0.1

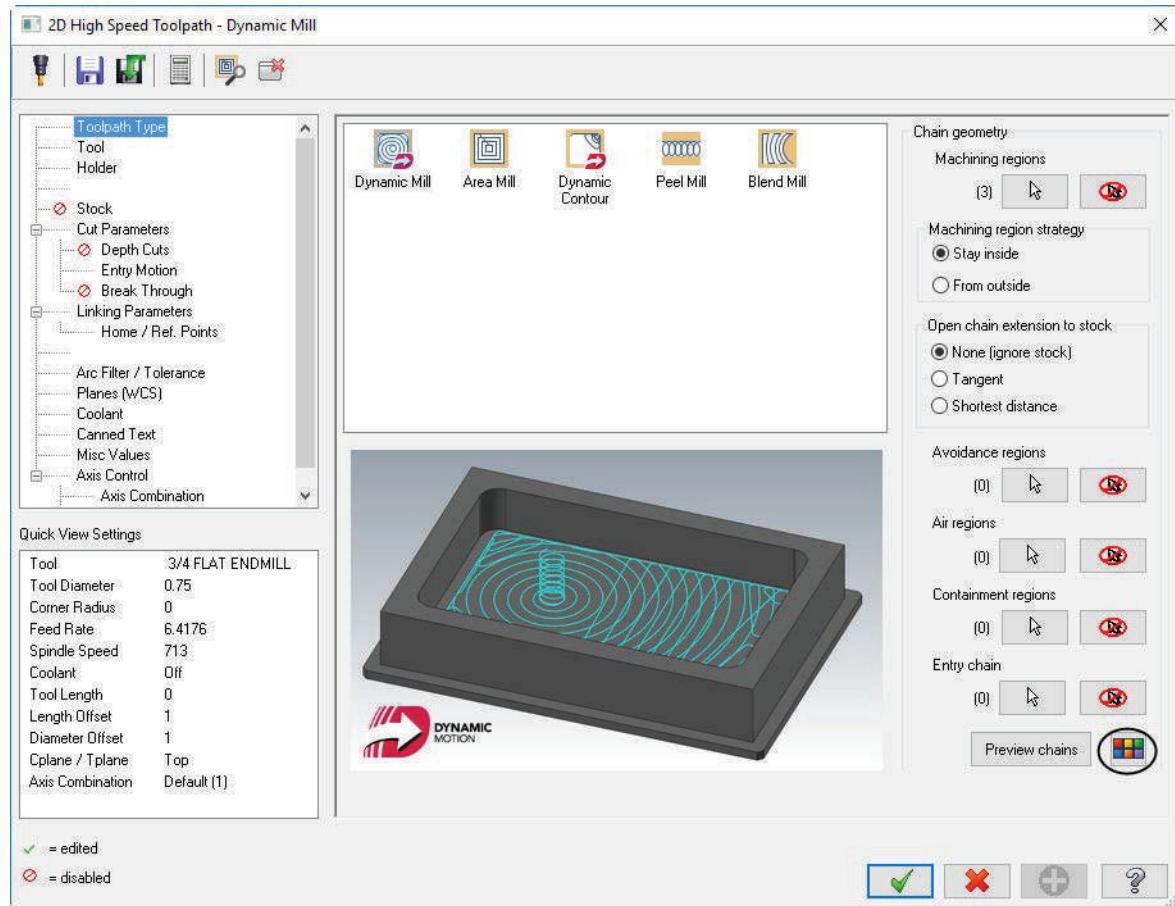




7.1 Preview Chains

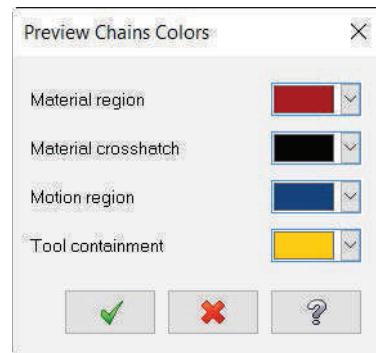
The **Preview Chains** function is intended to give the user a quick visual representation of how Mastercam sees the various pieces of geometry that have been selected, how they interact with one another, and a general overview of how the toolpath will be calculated with the selections presently made.

- ♦ Click on the **Color** icon to see the legend for **Preview chains** as shown.





- The Preview Chains Colors dialog box should look as shown.

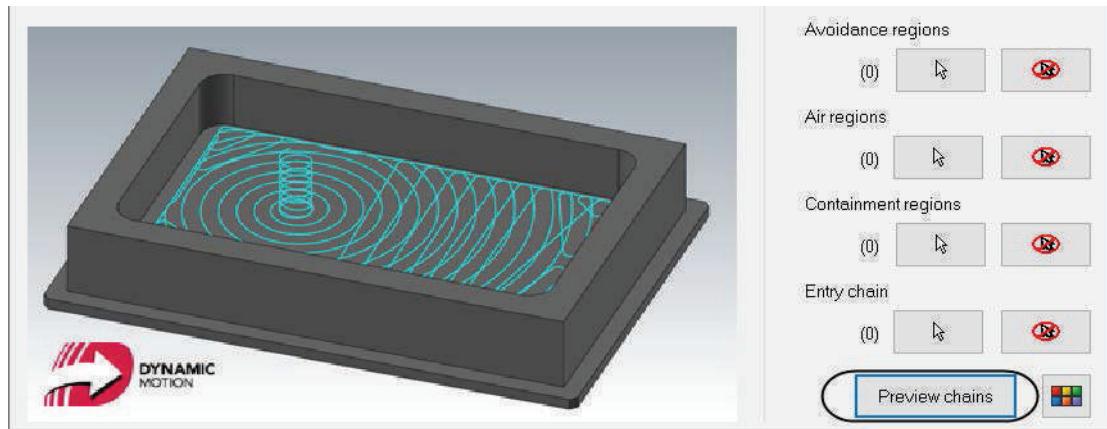


The **Material region** and **Material crosshatch** are the two colors that are used to define the material to be cut. The default colors are red for the background and black for the crosshatch.

The **Motion region** displays the area that Mastercam is making available to the toolpath for motion if it needs it. The color to represent it is dark blue. The primary reason for the display of the entire available (but not necessarily used) Motion region is to help the user visualize how the tool may move near or interact with any adjacent geometry.

The **Tool containment** is what you have selected as the Containment region in the chain geometry. If you have not selected a containment region, it will default to the outside of the Motion region since that is currently the default area the toolpath is being contained to. The color used to represent the Tool containment is yellow.

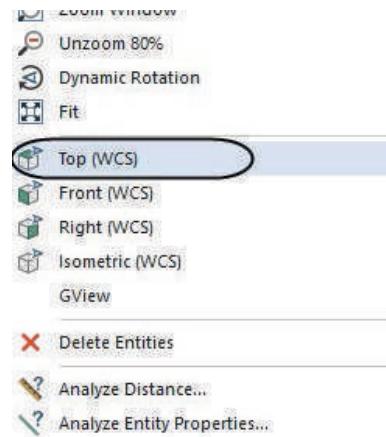
- Select the **OK** button to exit **Preview Chains Colors**.
- Select the **Preview chains** button as shown.



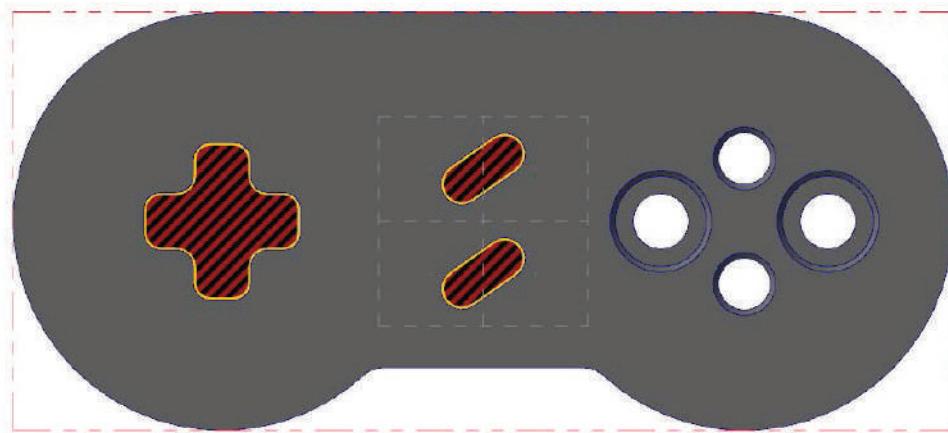
- Select the **Hide** dialog button to see the preview in the graphics window.



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



- ♦ The **Preview chains** should look as shown.



- ♦ Press **Esc** key to return to the toolpath parameters.
- ♦ Click on the **Preview chains** button again to clear the **Preview chains** display.





7.2 Select a 5/32" Flat Endmill from the library and set the Tool Parameters

- ♦ Select **Tool** from the **Tree View** list.

- ♦ Click on the **Select library tool** button.

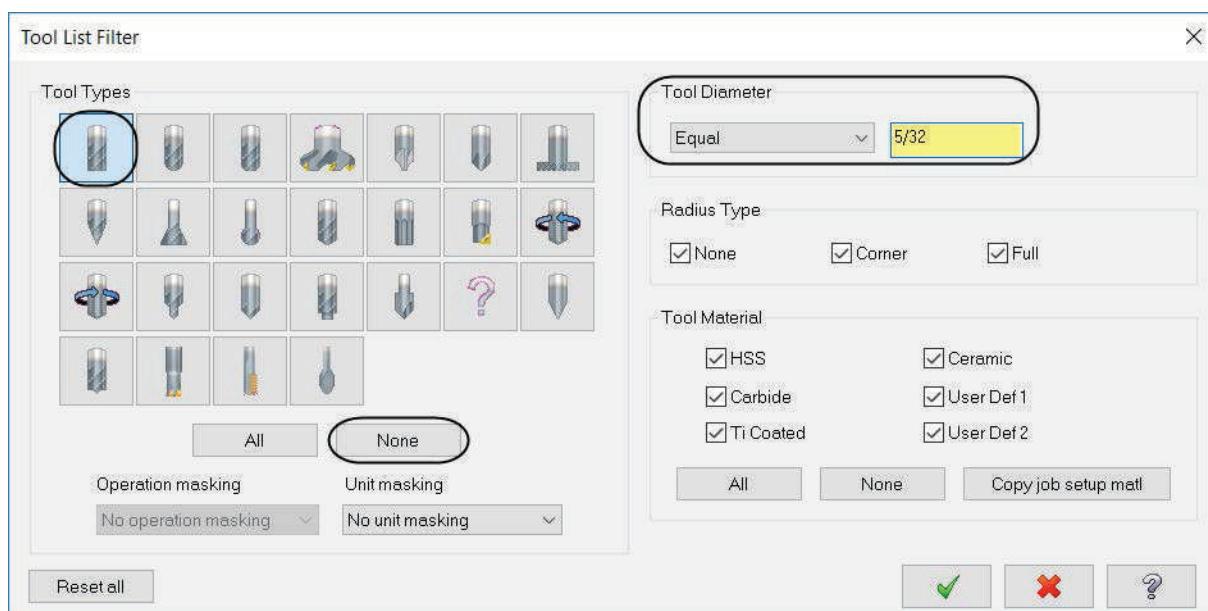
Select library tool...

- ♦ Select the **Filter** button.



- ♦ Select the **None** button and then under **Tool Types**, choose the **Flat Endmill** icon.
- ♦ Under **Tool Diameter**, pick **Equal** and input a value of **5/32** as shown in [Figure: 7.2.1](#).

Figure: 7.2.1



- ♦ Select the **OK** button to exit the **Tool List Filter**.
- ♦ In the **Tool Selection** dialog box, you should only see a **5/32" Flat Endmill**.

#	Assembly...	Tool Name	Holder Name	Dia.	Cor. r...	Length	# Flut...	Type	Rad...
283	-	5/32 FLAT ENDMILL	-	0.15...	0.0	0.375	4	End...	None

- ♦ Select the **5/32" Flat Endmill** in the **Tool Selection** page and then select the **OK** button to exit.

TUTORIAL #7**2D HIGH SPEED DYNAMIC MILL**

- ♦ Make all the necessary changes as shown in [Figure: 7.2.2.](#)

Figure: 7.2.2

The screenshot shows a software interface for tool setup. On the left, a table lists a single tool entry:

#	Assembly Name	Tool Name	Holder Name	Dia.
1	-	5/32 FLAT...	-	0....

On the right, various tool parameters are displayed in input fields:

- Tool diameter: 0.15625
- Corner radius: 0.0
- Tool name: 5/32 FLAT ENDMILL
- Tool #: 1
- Length offset: 1
- Head #: 0
- Diameter offset: 1
- RCTF:
- Spindle direction: CW
- Feed rate: 6.160896
- Spindle speed: 3422
- FPT: 0.0005
- SFM: 139.9705
- Plunge rate: 6.160896
- Retract rate: 6.160896
- Force tool change:
- Rapid Retract:

A comment box at the bottom contains the text: "Machine the 3 pockets using dynamic mill."

Other interface elements include:

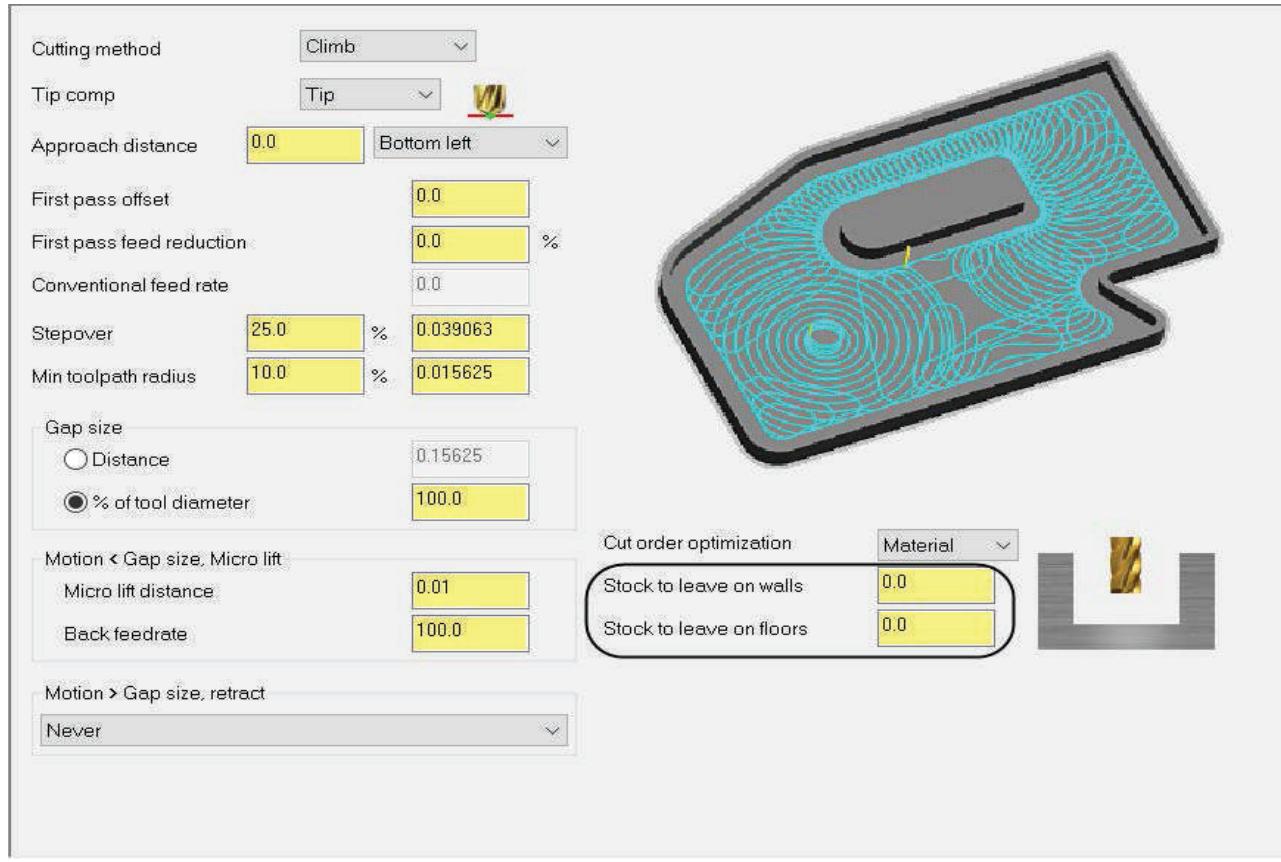
- Select library tool... button
- Filter Active checkbox
- Filter... button
- Right-click for options button
- To batch checkbox



7.3 Set the Cut Parameters

- From the Tree View list, select **Cut Parameters**.
- Change the settings for this toolpath as shown in [Figure: 7.3.1](#).

Figure: 7.3.1



Stepover sets the distance between cutting passes in the X and Y axis.

Toolpath radius reduces sharp corner motion between cut passes.

Micro lift distance enters the distance the tool lifts off the part on the back moves. Microlifts are slight lifts that help clear chips and minimize excessive tool heating.

Back feedrate controls the speed of the backfeed movement of the tool.

Motion > Gap size, retract controls retracts in the toolpath when making a non-cutting move within an area where the tool can be kept down or microlifted.

Cut order optimization defines the cut order Mastercam applies to different cutting passes in the dynamic mill toolpath.



7.4 Set the Entry Motion

- From the Tree View list, select **Entry Motion**.
- Entry Motion** configures an entry method for the dynamic mill toolpath which determines not only how and where the tool enters the part, but also the cutting method/machining strategy used by the toolpath. The previous settings will be saved.
- All we want to do is change the **Entry method** to **Profile** as shown in [Figure: 7.4.1](#).

Figure: 7.4.1



Entry method set to **Profile** creates a boundary based on the shape of the selected chain and uses the tool to ramp into the part. The slot is cleared by taking lighter cuts in the Z axis until the tool reaches the full depth.

Z clearance adds extra height used in the ramping motion down from a top profile. It ensures that the tool has fully slowed down from rapid speeds before touching the material.

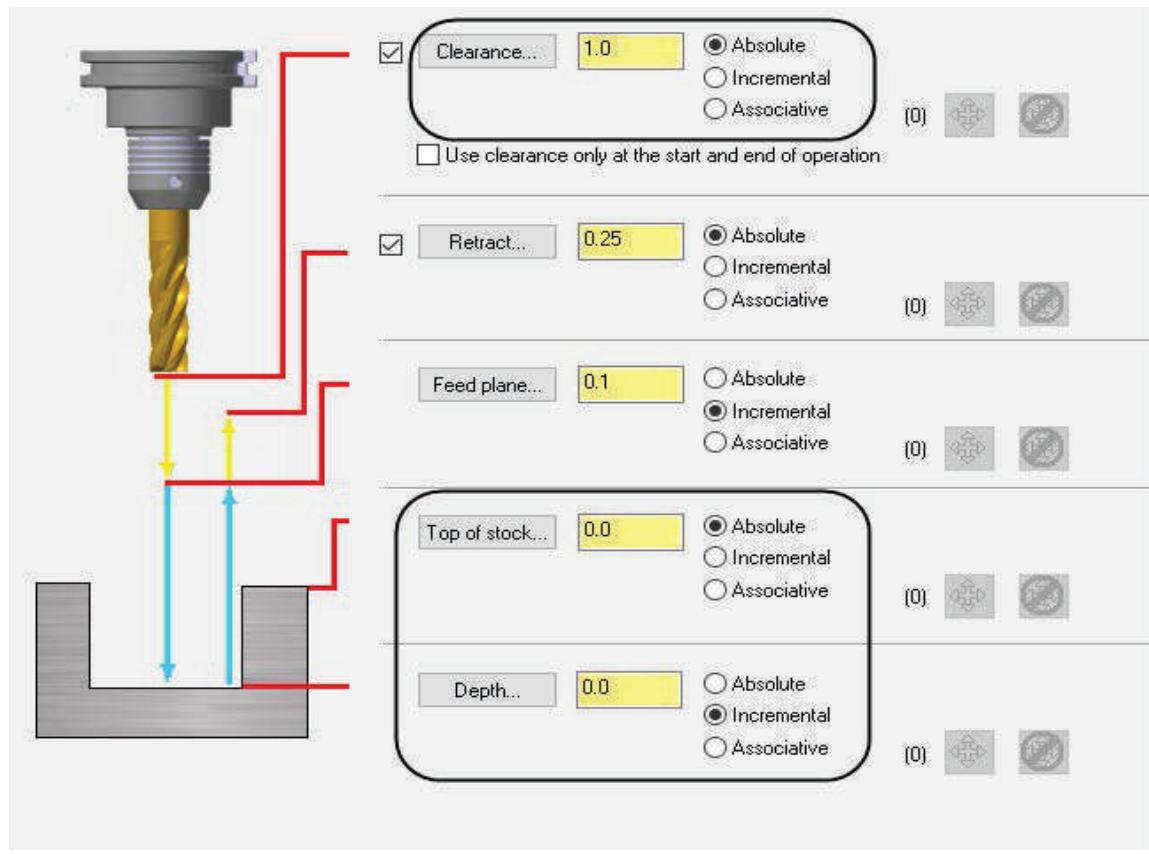
Plunge angle sets the angle of descent for the entry move and determines the pitch.



7.5 Set the Linking Parameters

- ♦ Select **Linking Parameters** and enable **Clearance** and set it to **Absolute 1.0**. Change the **Top of stock** value to **Absolute 0.0** and the **Depth** to **Incremental** and **0.0** as shown in [Figure: 7.5.1](#).

Figure: 7.5.1



Incremental value for the **Depth** is measured at the chained geometry. In this tutorial, the pocket chains were selected at the bottom, which is their final depth.

7.6 Preview the Toolpath

- ♦ To quick check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

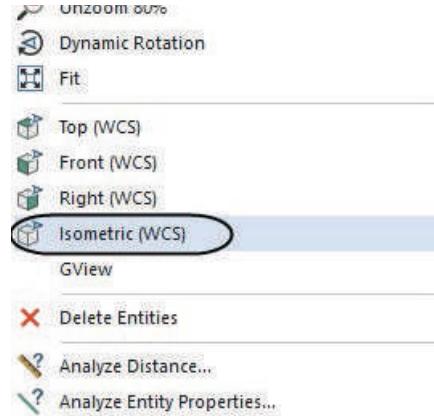


- ♦ To hide the dialog box, click on the **Hide dialog** icon as shown.

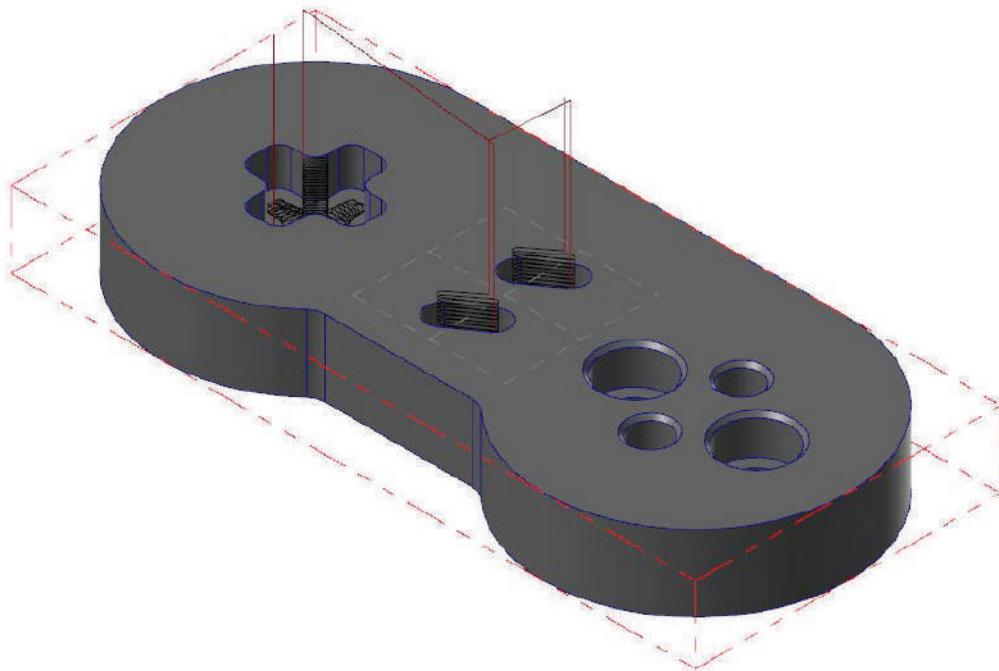




- ♦ To see the part from an **Isometric** view, right mouse click in the graphics window and select **Isometric** as shown.



- ♦ The toolpath should look as shown.



- ♦ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- ♦ Select the **OK** button to generate the toolpath.

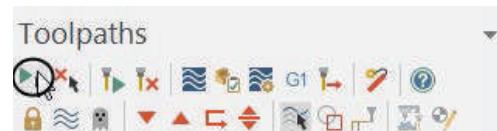




STEP 8: BACKPLOT THE TOOLPATHS

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 additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates, and the cycle time. It also shows any collision between the workpiece and the tool.

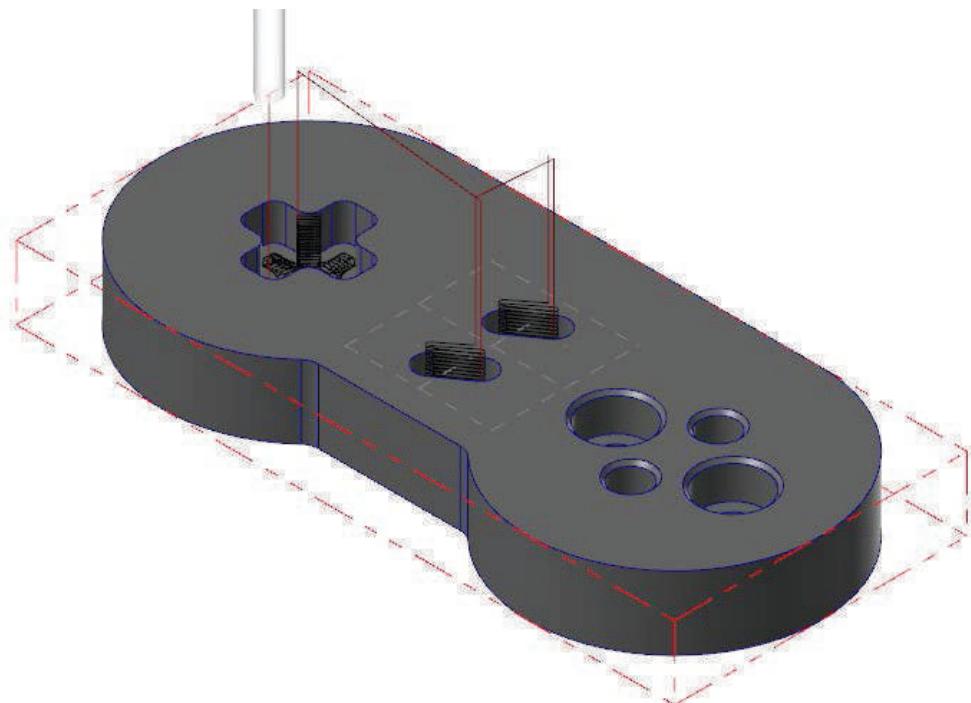
- ♦ Press **Alt + S** to shade the geometry.
- ♦ Make sure that the toolpaths are selected (signified by the green check mark on the folder icon). If the operation is not selected, choose the **Select all operations** icon.



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



- ♦ Select the **Play** button to run **Backplot**.
- ♦ The toolpath should look as shown.



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





STEP 9: SIMULATE THE TOOLPATH IN VERIFY

Verify Mode shows the path the tools take to cut the part with material removal. This display lets you spot errors in the program before you machine the part. As you verify toolpaths, Mastercam displays additional information such as the X, Y, and Z coordinates, the path length, the minimum and maximum coordinates, and the cycle time. It also shows any collisions between the workpiece and the tool.

- From the **Toolpaths Manager**, select the **Verify selected operations** icon as shown.

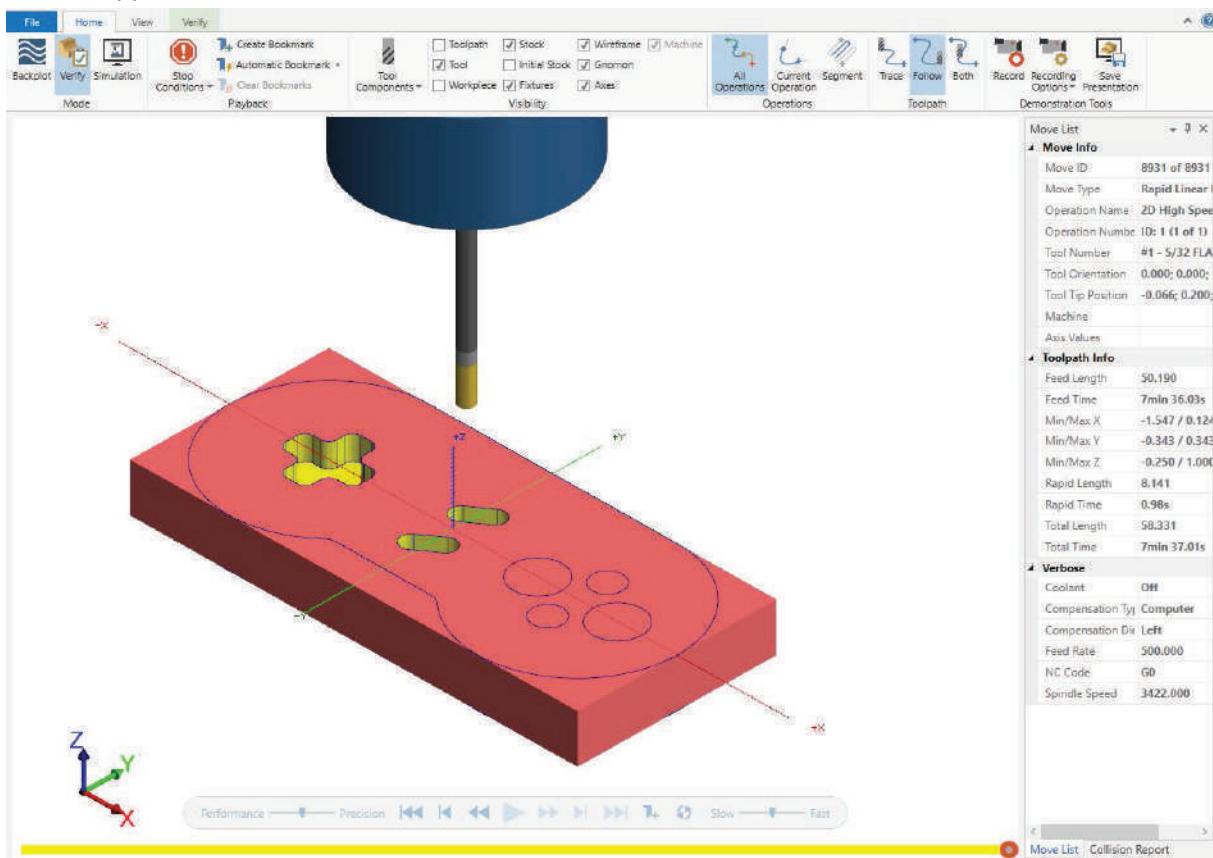


NOTE: Mastercam launches a new window that allows you to check the part using **Backplot** or **Verify**.

- Select the **Play** button to run **Verify**.



- The part should appear as shown.



- To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.

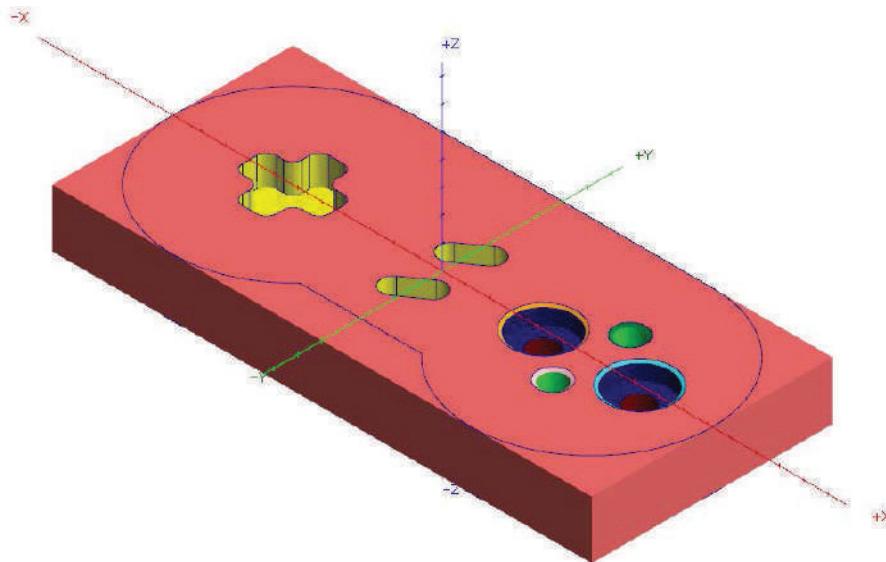




STEP 10: DRILL THE HOLES USING FBM DRILL

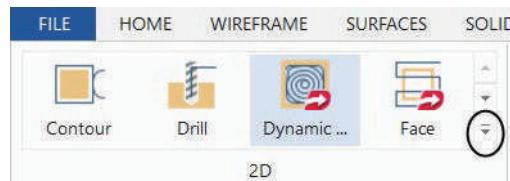
FBM Drill automatically detects holes in a solid based on your specific criteria and generates a complete series of drilling and chamfering. FBM drill also generates circle mill or helix bore operations for large-hole features when you activate these settings.

Toolpath Preview:

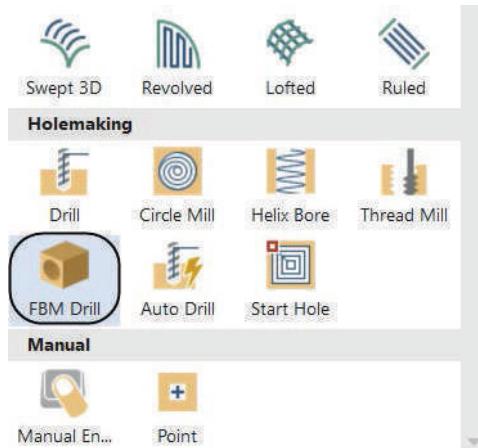


TOOLPATHS

- From the **2D** group, select the **Expand gallery** arrow as shown.



- From the **Toolpaths Gallery**, select the **FBM Drill**.



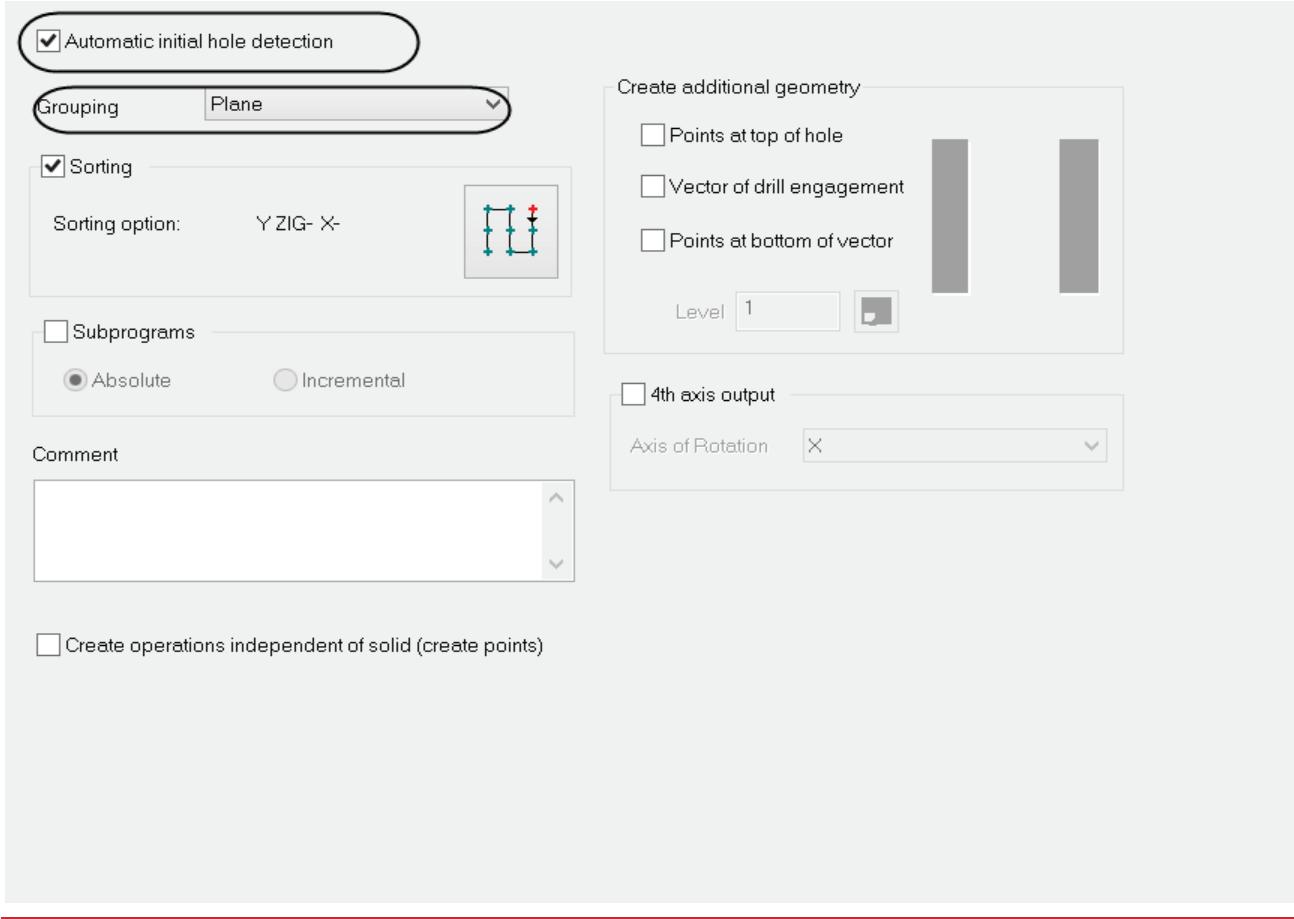
TUTORIAL #7

DRILL THE HOLES USING FBM DRILL



- When the **FBM Toolpaths - Drill** page appears, enable **Automatic initial hole detection**.
- Change the **Grouping** to **Plane** as shown in [Figure: 10.0.1](#) to group the operations by the plane in which the holes lie.

Figure: 10.0.1



Automatic initial hole/feature detection automatically detects features when you choose the FBM operation. If you save the settings to your toolpath .DEFAULTS file and use that final in the active machine group, Mastercam automatically detects features every time you choose the FBM operation.

Grouping controls how the drill cycles that FBM Drill creates are organized in the **Toolpaths Manager**. Mastercam orders operations within groups into subgroups by operation type.

Plane groups all operations based on the plane of the hole.

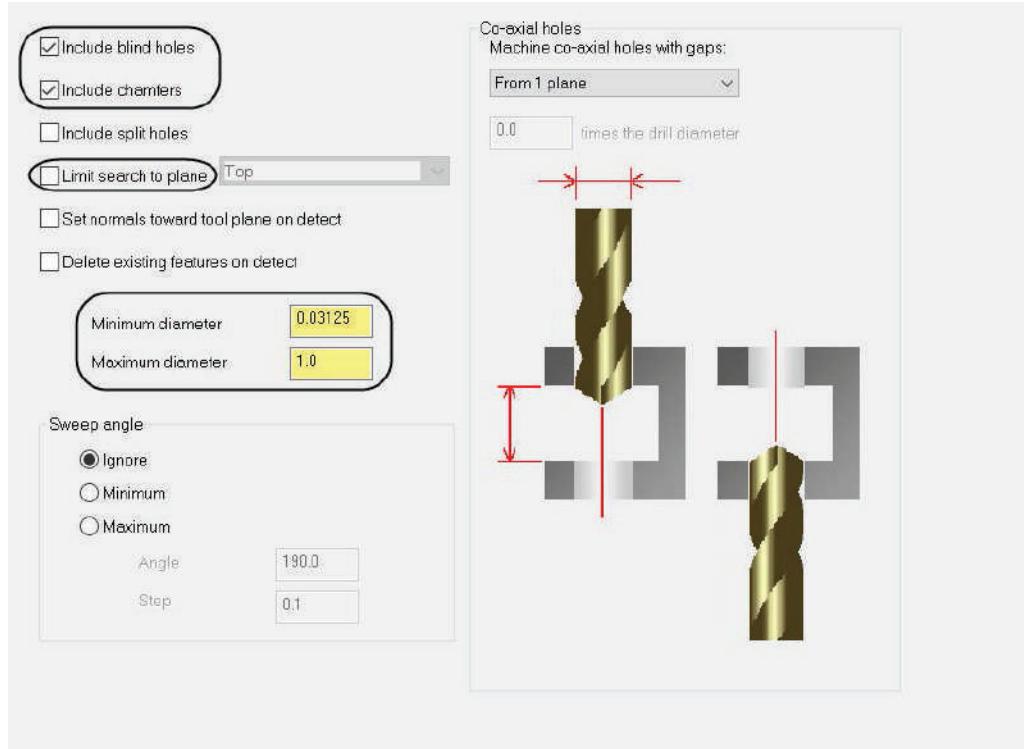
Create additional geometry selects one or more options to create geometry for detected hole features without generating toolpaths. The geometry is saved to a level you choose in this section and is non associative.



10.1 Hole Detection

- Choose **Hole Detection** to control the types of holes FBM Drill detects. Enable/disable the options as shown in [Figure: 10.1.1](#).

Figure: 10.1.1



Include blind holes will search your part for blind holes (holes that do not go through the entire solid).

Include chamfers searches for holes with chamfers.

Include split holes searches the part for holes that are incomplete.

Limit search to plane detects features that can only be machined in the selected plane.

Minimum diameter finds holes which are equal to or greater than this value.

Maximum diameter finds holes which are equal to or less than this value.

Sweep angle lets you set a tolerance for how complete holes need to be in terms of their included angle to be detected by and included in the FBM Drill operation.

Machine co-axial holes with gaps determines whether Mastercam treats multiple holes that share a common axis as a single hole, or as multiple holes from different planes.

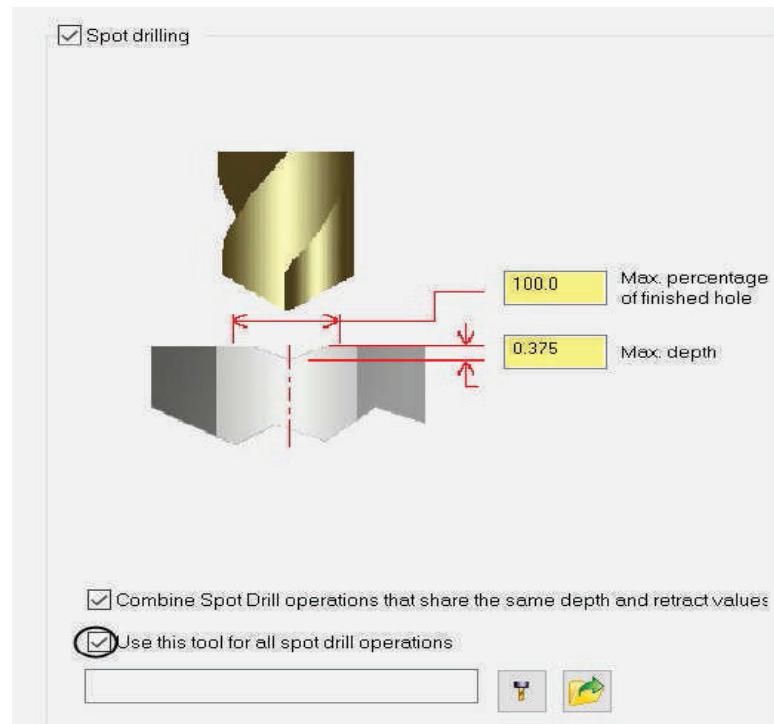


10.2 Spot Drilling

- ♦ Select **Spot Drilling** to activate and define the spot drilling toolpaths for the **FBM Drill** operation.
- ♦ Enable the option **Use this tool for all spot drill operations** as shown in [Figure: 10.2.1](#).

NOTE: It takes a couple of minutes to enable the **Use this tool for all spot drill operations** button.

Figure: 10.2.1



Use this tool for all spot drill operations allows you to choose a specific spot drill cycle generated by the FBM Drill operation.

- ♦ Pick the button **Select tool from library**.
- ♦ This will let you choose a tool from the current tool library.
- ♦ Find and select the **1/2"** **Spot drill** from the list.

#	Assembly...	Tool Name	Holder Name	Dia.	Cor. r...	Length	# Flut...	Type	Rad....
22	-	1/4 SPOTDRILL	-	0.25	0.0	2.0	2	Spot...	None
23	-	3/8 SPOTDRILL	-	0.375	0.0	2.0	4	Spot...	None
24	-	1/2 SPOTDRILL	-	0.5	0.0	2.0	2	Spot...	None
25	-	3/4 SPOTDRILL	-	0.75	0.0	2.0	4	Spot...	None
26	-	1. SPOTDRILL	-	1.0	0.0	2.0	4	Spot...	None
27	-	1/64 DRILL	-	0.01...	0.0	1.0	2	Drill	None
28	-	NO. 78 DRILL	-	0.016	0.0	1.0	2	Drill	None

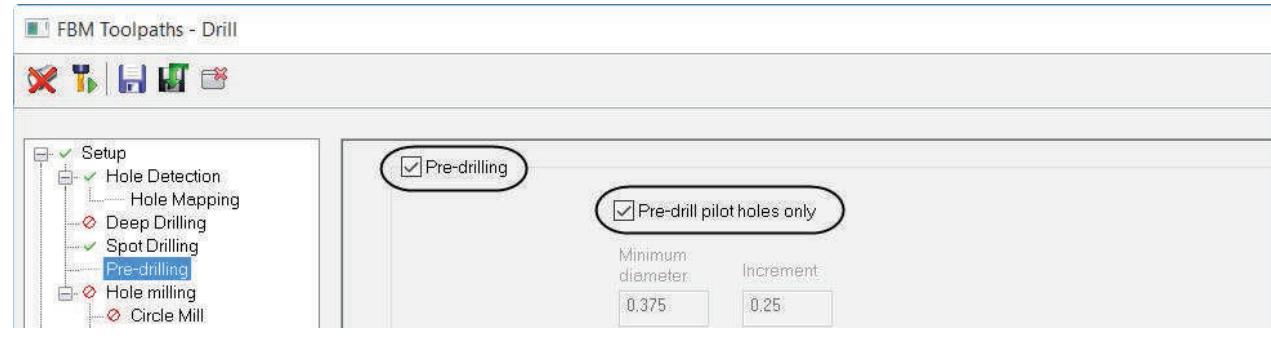
- ♦ Select the **OK** button and the **Spot drill** will appear in the box to the left of the buttons.



10.3 Pre-Drilling

- ♦ Select **Pre-drilling**.
- ♦ This page defines pre-drilling cycles that rough out the drilled holes before the finish drill cycle.
- ♦ Leave the **Pre-drilling** settings as shown in [Figure: 10.3.1](#).

Figure: 10.3.1



Pre-drilling are operations that rough out the detected holes before creating any finished drill and chamfer operation.

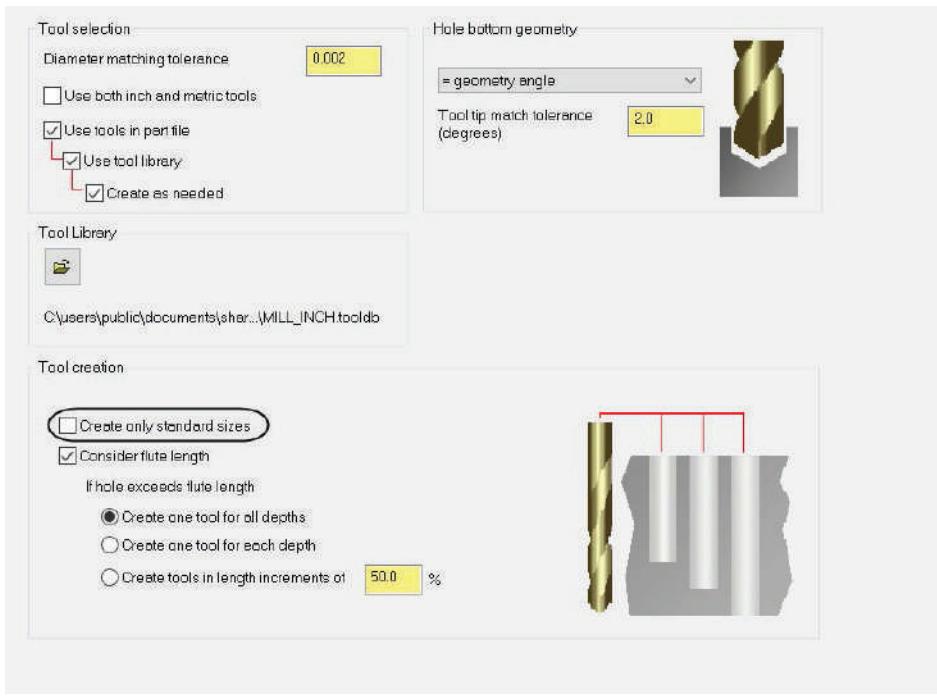
Pre-drill pilot holes only deactivate all pre-drill roughing cycles except for assigned pilot holes cycles. This also deactivates the parameters for minimum diameter, increment and stock to leave on the page because they are not applicable. FBM Drill generates only pilot holes pre-drill cycles followed by finish hole cycles.



10.4 Tools

- ♦ **Select Tools.** This page controls the tools Mastercam selects for the drill cycles that the **FBM Drill** operation creates.
- ♦ Enable/disable the parameters as shown in [Figure: 10.4.1](#).

Figure: 10.4.1



Diameter matching tolerance determines how closely the diameter of the tool and the hole must match before selecting an appropriate tool.

Tool selection lets you pick one or more of the following options to tell Mastercam where to locate tools for the FBM-generated toolpaths.

Use tools in part file looks in the current Mastercam file for an appropriate tool. The tools do not have to be used in previous operations to be available to the FBM operations.

Use tool library searches the selected tool library for the necessary tools.

Create as needed creates the necessary tools using the tool creation parameters you define.

Hole bottom geometry defines the relationship of the bottom hole geometry to the tool tip geometry.

= **geometry angle** the tool tip angle must match the hole bottom geometry within the specified tool tip match tolerance.

> **geometry angle** the tool tip must be greater than the floor angle geometry.

< **geometry angle** the tool tip angle must be smaller than the floor angle.

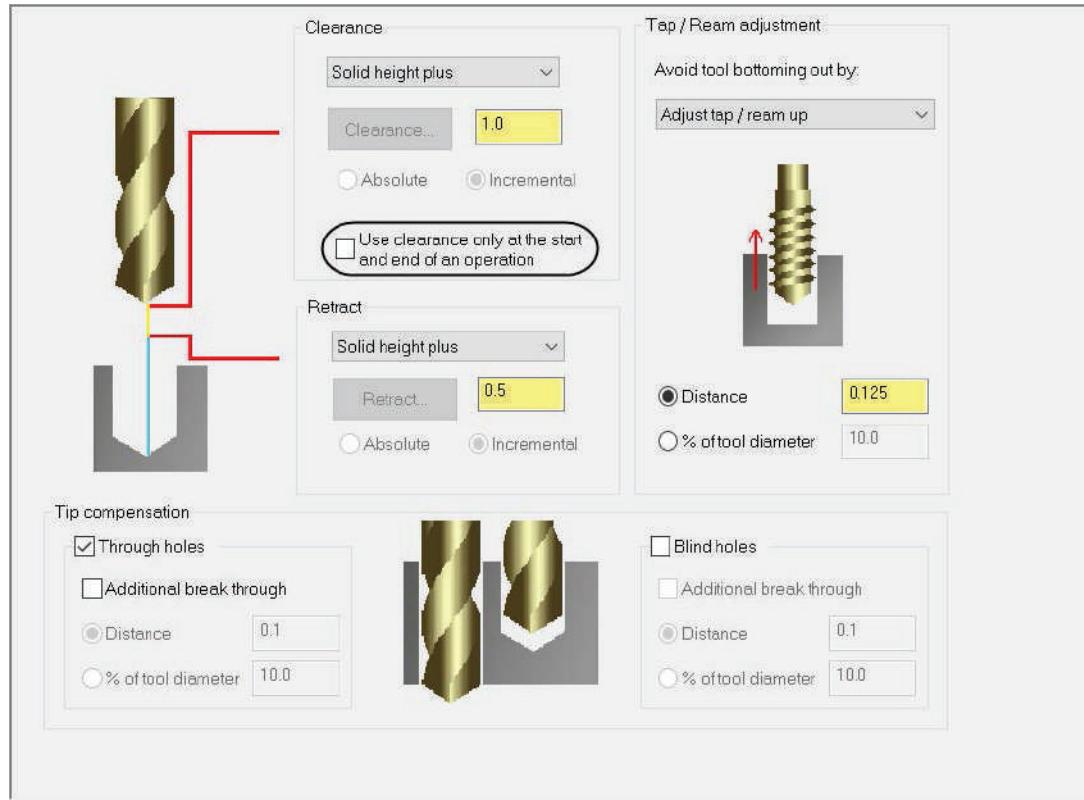


10.5 Linking Parameters

This page defines how **FBM Drill** calculates clearance height and retract height for the drilling cycles.

- ♦ Select **Linking Parameters** from the **Tree View list** and set the parameters as shown in [Figure: 10.5.1](#).

Figure: 10.5.1



Clearance is the height at which the tool moves to and from the part. There are several options available from where the height is measured:

Solid height plus adds a fixed height above the highest point of the solid model.

Stock model plus adds a specified height above the stock model.

Top of hole plus adds clearance distance above the top of each hole.

Top of Coaxial holes plus sets the clearance above the highest hole on the shared axis for holes that share the same axis.

Manual allows you to set the clearance using all options in any combination.

Retract is the height at which the tool moves before the next tool pass. The same options are available as for **Clearance** height.

Tap/Ream adjustment determines whether tapped and reamed holes are fully finished.

Tip compensation compensates for the tool tip.

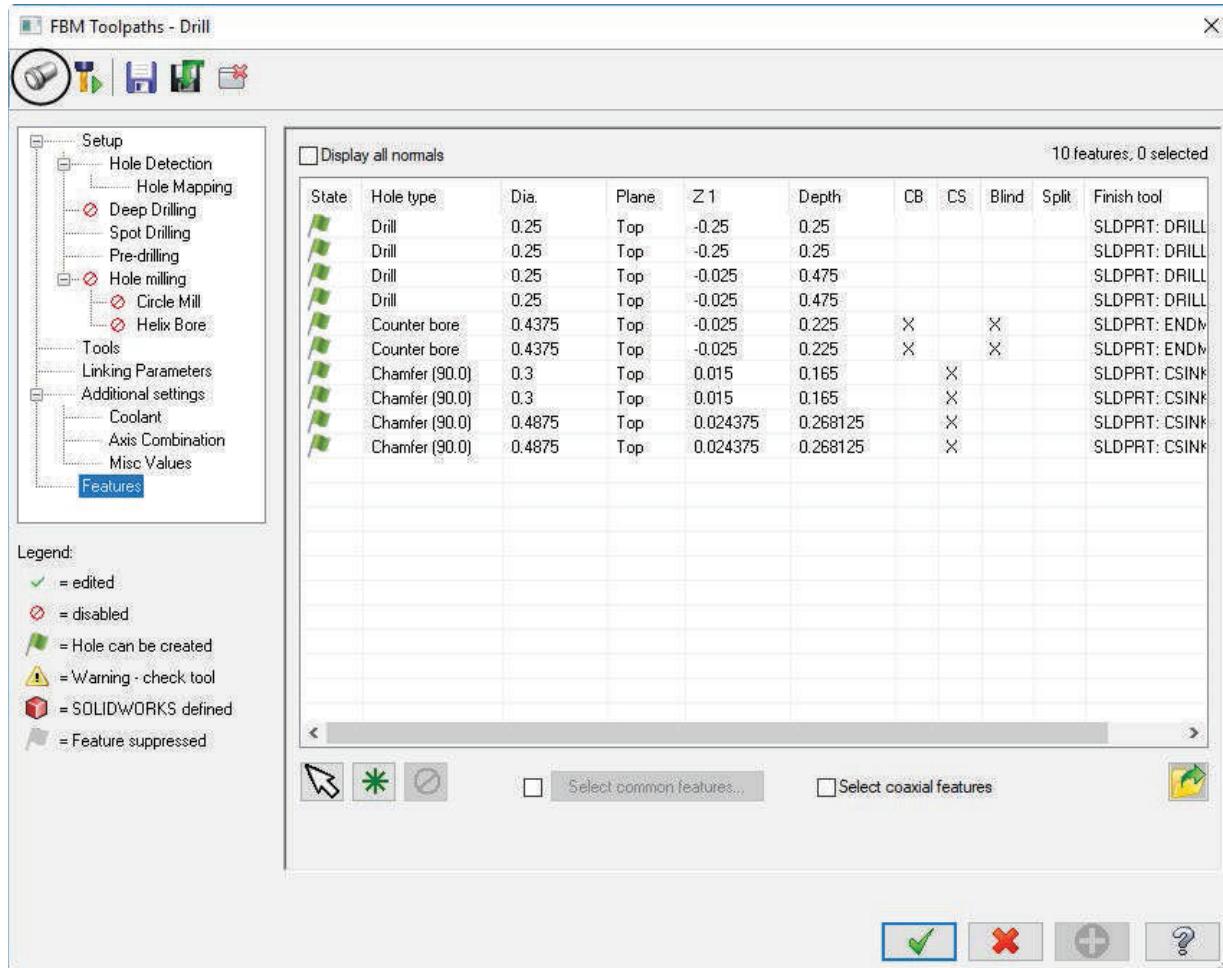


10.6 Features

This page allows you to manage the list of hole features that FBM Drill detects in the solid model.

- ♦ Select **Features** from the **Tree View** list.
- ♦ Choose the button to **Detect the Features** on the top left corner of the dialog box for Mastercam to detect the holes as shown in [Figure: 10.6.1](#).

Figure: 10.6.1

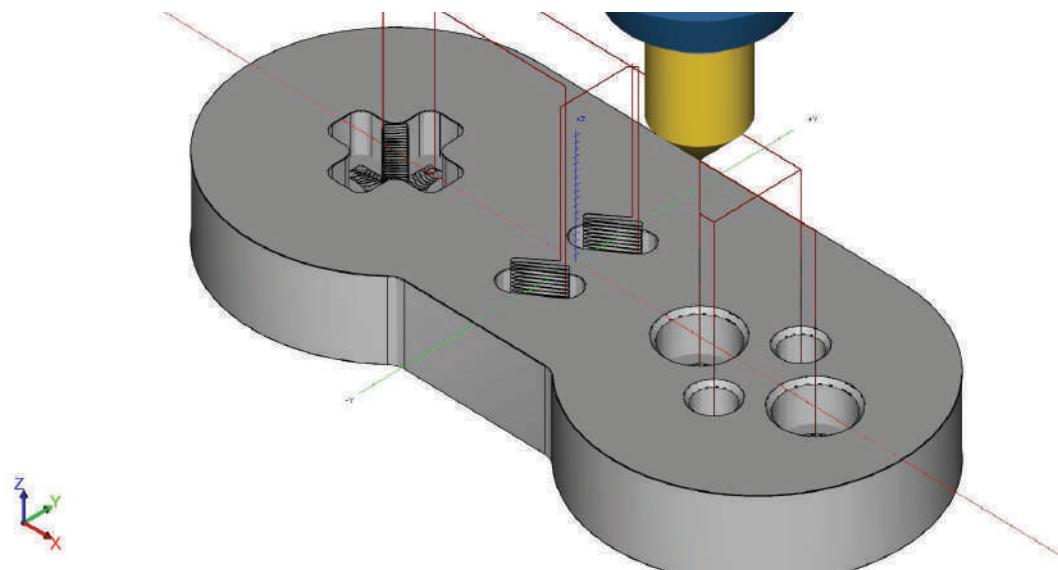


- ♦ Choose the **OK** button to exit the **FBM Toolpaths - Drill** parameters.



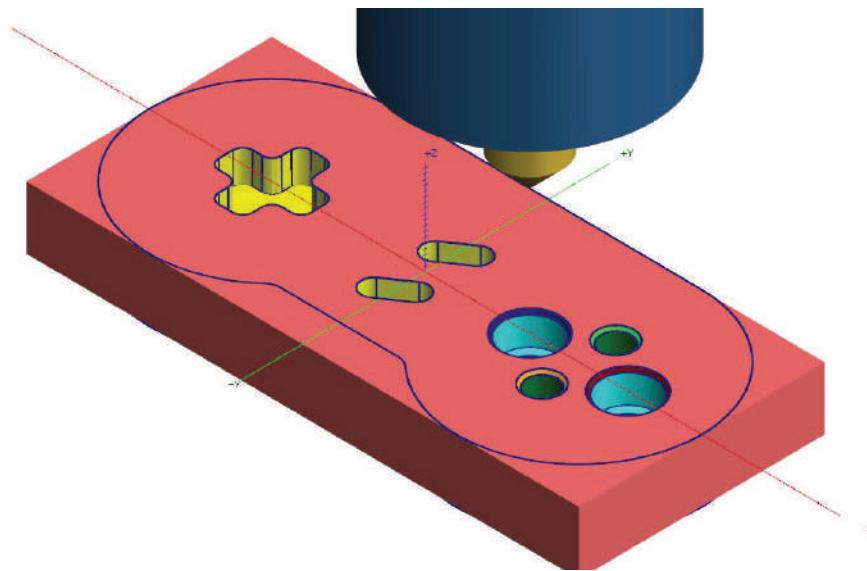
10.7 Backplot the toolpaths

- ♦ Click on the **Select all operations** icon in the **Operations Manager**.
- ♦ To **Backplot** the toolpaths, see [page 655](#) to review the procedure.
- ♦ The toolpaths should look as shown.



10.8 Verify the toolpaths

- ♦ To **Verify** the toolpaths, see [page 656](#) to review the procedure.
- ♦ The part should look as shown.



- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.

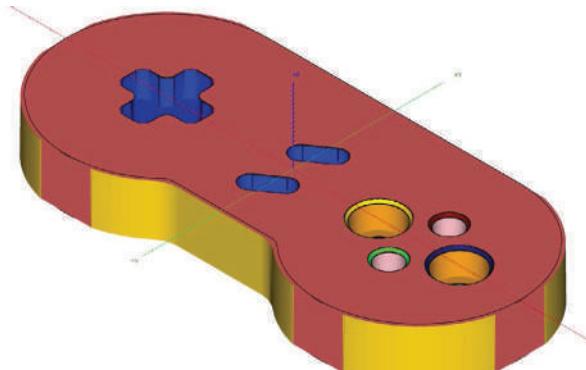


STEP 11: MACHINE THE OUTSIDE OF THE PART USING DYNAMIC MILLING

2D High Speed Dynamic Mill utilizes the entire flute length of their cutting tools to produce the smoothest, most efficient tool motion for high speed pocketing and core milling.

The **Dynamic Mill** toolpath machines pockets, material that other toolpaths left behind, and standing bosses or cores. The toolpath depends on the **Machining strategy** that you choose in the **Chain Options**. If the strategy chosen is **From outside**, the toolpaths start at the outmost chain and moves freely outside of this area; the inner chain defines the limit of the toolpath. You can also machine pockets in which case the strategy selected is **Stay inside** which keeps the tool inside the machining regions.

Toolpath Preview:

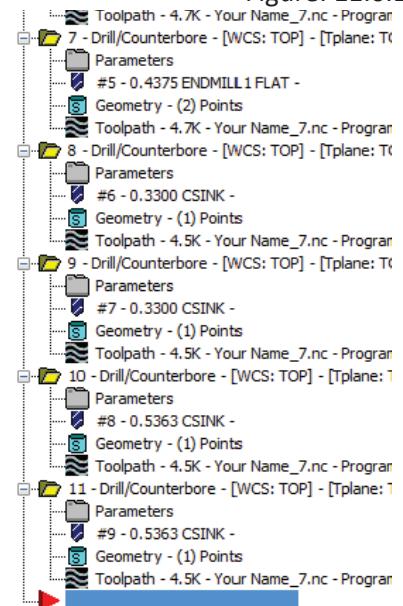


- ♦ Click on the **Move insert arrow down one item** icon to move the arrow at the end of the toolpaths as shown in



Figure: 11.0.1.

Figure: 11.0.1



- ♦ In the **Toolpaths Manager**, click the **Select all operations**.
- ♦ Press **Alt + T** to remove the toolpath display.

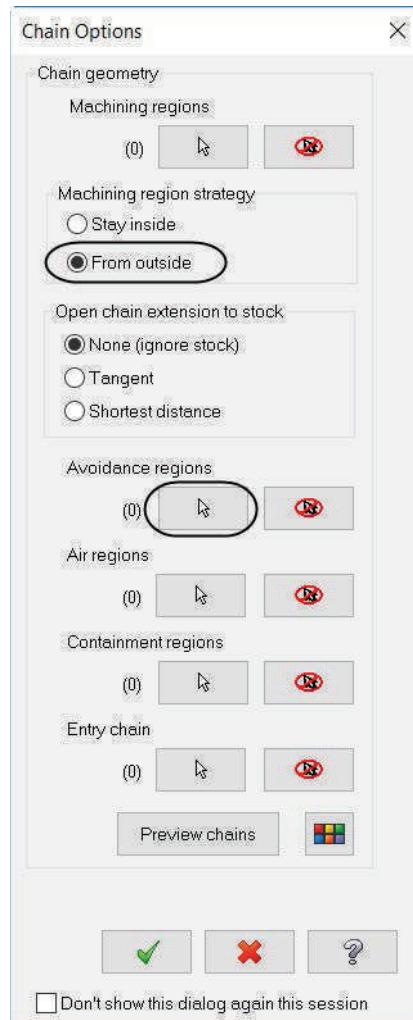
**TOOLPATHS**

- From the **2D** group, select the **Expand gallery** arrow and then from the **Toolpath Gallery**, select **Dynamic Mill**.



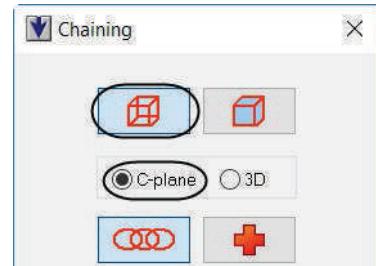
- In the **Chain Options**, **Machining region strategy**, enable **From outside** as shown in [Figure: 11.0.2](#).
- Click on the **Select** button in the **Avoidance regions** as shown in [Figure: 11.0.2](#).

Figure: 11.0.2

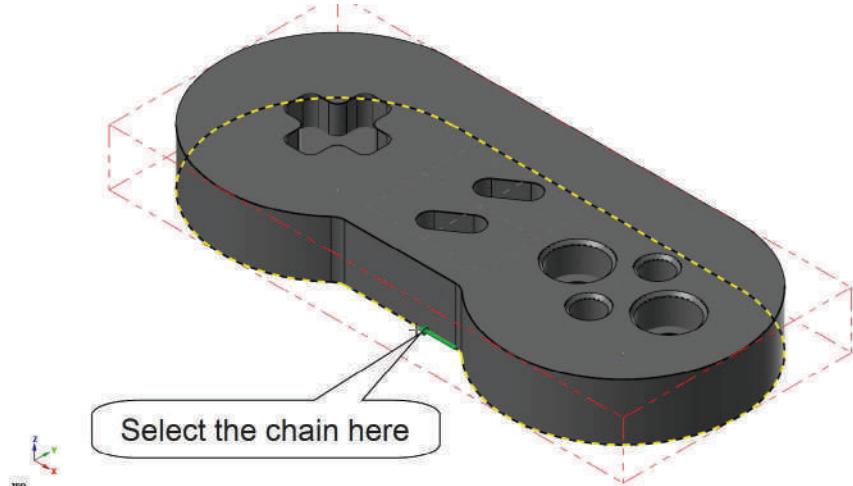




- ◆ Enable **C-plane** in the **Chaining** dialog box as shown.



- ◆ [Select 2D HST avoidance chain 1]: Select the chain as shown.

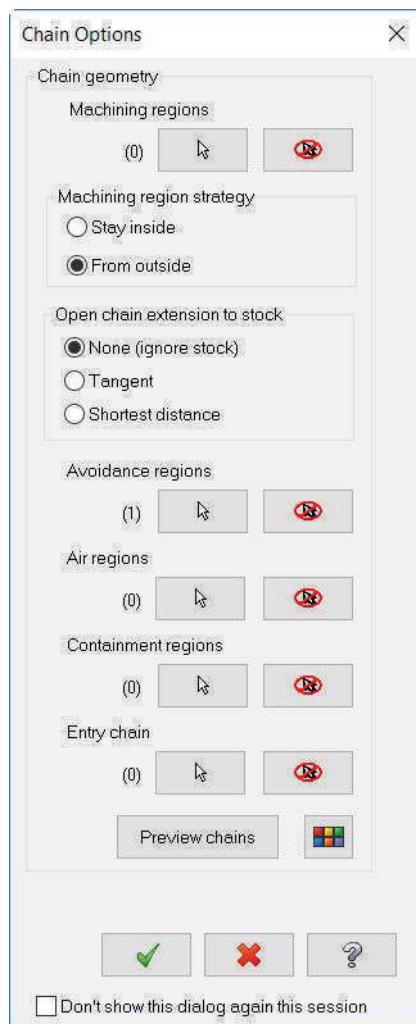


- ◆ Select the **OK** button to exit the **Chaining** dialog box.





- The **Chain Options** dialog box should look as shown.



From outside enabled ensures that the tool will approach the part from the outside.

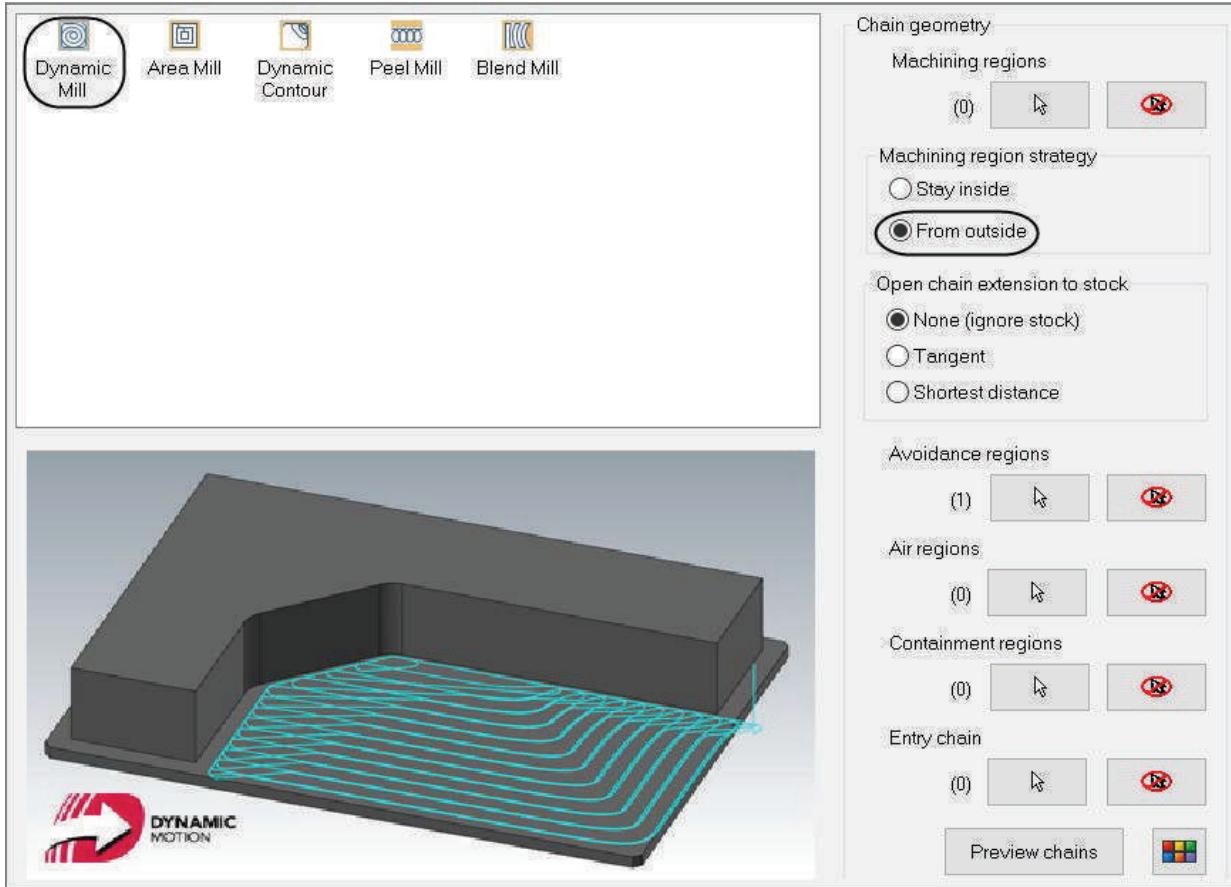
Avoidance regions allows you to select the profile that describes the shape up to where the material will be removed.

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



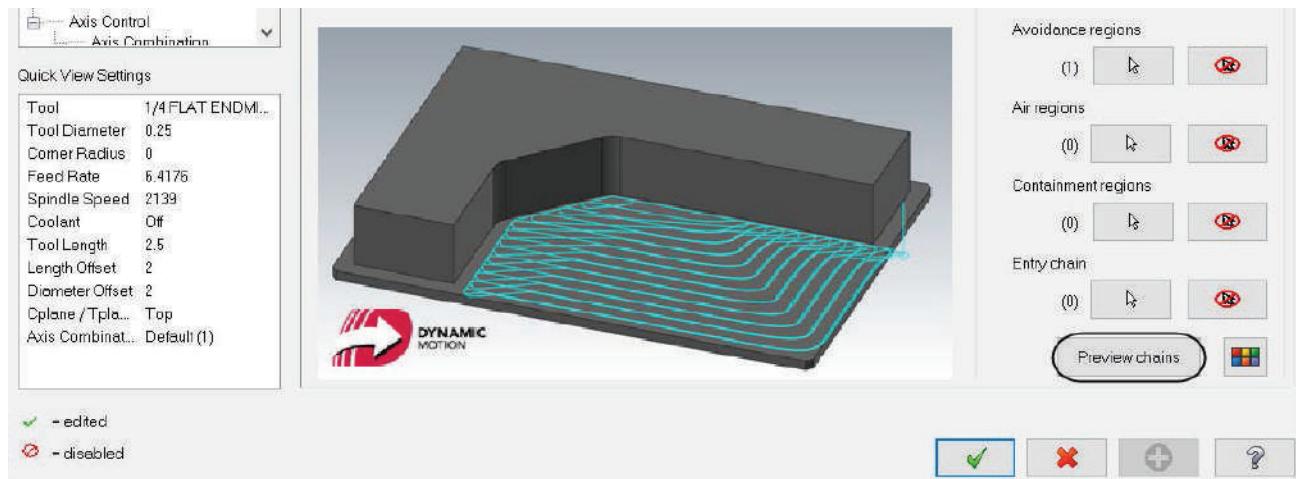


- In the Toolpath Type page, Dynamic Mill with From outside should be already selected.



11.1 Preview Chains

- Select the Preview chains button as shown.

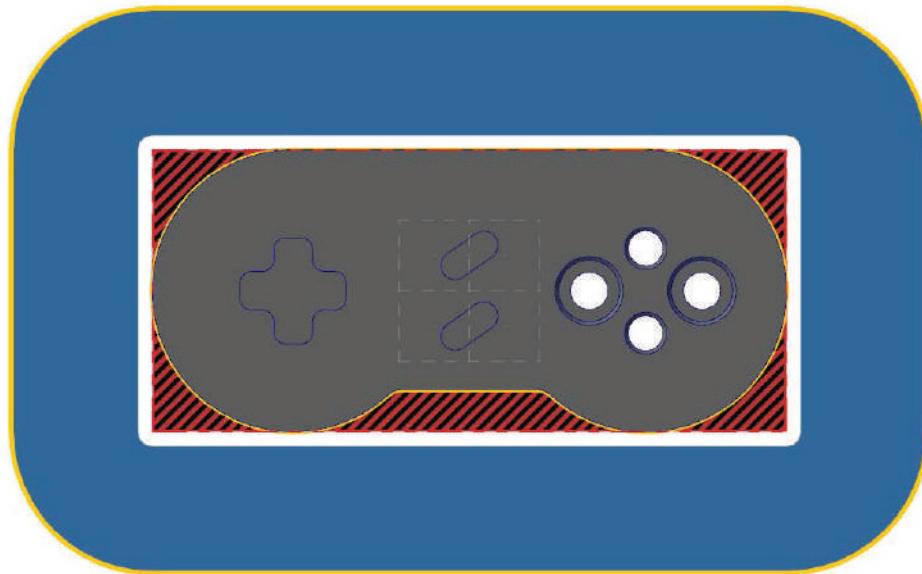


- Select the Hide dialog button to see the preview in the graphics window.





- ♦ The **Preview chains** should look as shown.



- ♦ Press **Esc** key to return to the toolpath parameters.
- ♦ Click on the **Preview chains** button again to clear the **Preview chains** display.

 Preview chains



11.2 Select a 0.5" Flat Endmill from the Library and set the Tool Parameters

- ♦ Select **Tool** from the **Tree View** list.

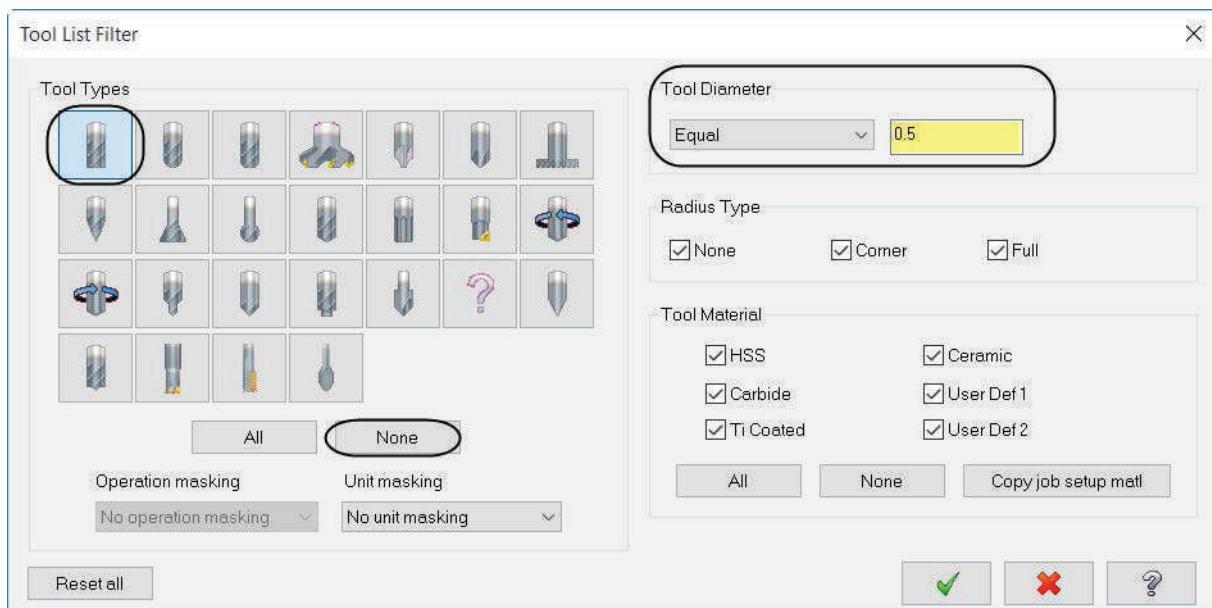
♦ Click on the **Select library tool...**

- ♦ Select the **Filter** button.



- ♦ Select the **None** button and then under **Tool Types**, choose the **Flat Endmill** icon.
- ♦ Under **Tool Diameter**, pick **Equal** and input a value of **0.5** as shown in [Figure: 11.2.1](#).

Figure: 11.2.1



- ♦ Select the **OK** button to exit the **Tool List Filter**.
- ♦ In the **Tool Selection** dialog box, you should only see a **1/2" Flat Endmill**.

#	Assembly...	Tool Name	Holder Name	Dia.	Cor. r...	Length	# Flut...	Type	Rad...
290	-	1/2 FLAT ENDMILL	-	0.5	0.0	1.0	4	End...	None

- ♦ Select the **1/2" Flat Endmill** in the **Tool Selection** page and then select the **OK** button to exit.



- ♦ Make all the necessary changes as shown in [Figure: 11.2.2.](#)

Figure: 11.2.2

The screenshot shows the Mastercam software interface for setting up a dynamic milling operation. On the left, a list of tools is displayed in a table:

#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	L...
1	-	5/32 FLA...	-	0.15...	0.0	0.0
2	-	1/2 SPOT...	-	0.5	0.0	0.0
3	-	-	-	0.33	0.0	0.0
4	-	-	-	0.33	0.0	0.0
5	-	-	-	0.53...	0.0	0.0
6	-	-	-	0.53...	0.0	0.0
7	-	1/4 DRILL	-	0.25	0.0	0.0
8	-	LTR. H D...	-	0.266	0.0	0.0
9	-	-	-	0.4375	0.0	0.0
10	-	1/2 FLAT ...	-	0.5	0.0	0.0

On the right, various parameters are set for the toolpath:

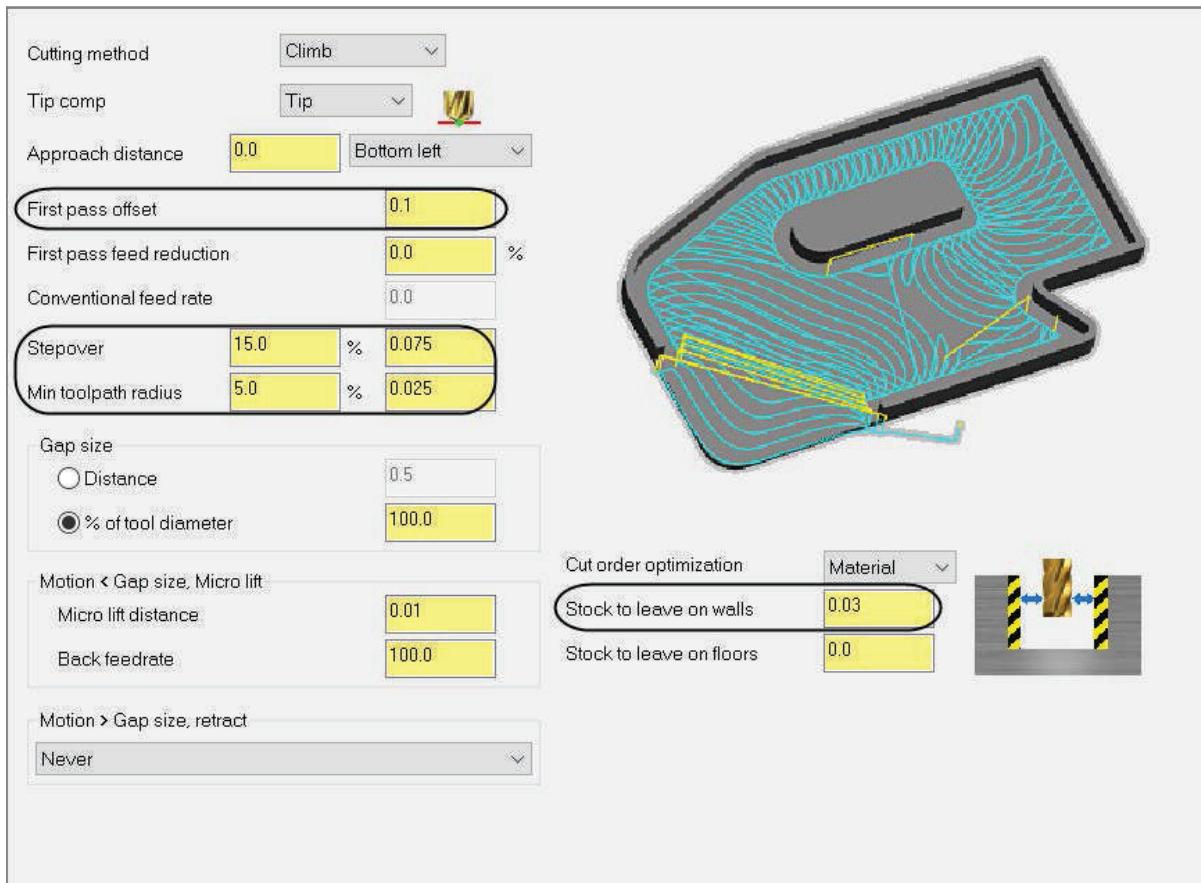
- Tool diameter: 0.5
- Corner radius: 0.0
- Tool name: 1/2 FLAT ENDMILL
- Tool #: 10
- Length offset: 10
- Head #: 0
- Diameter offset: 10
- RCTF:
- Spindle direction: CW
- Feed rate: 150.0
- Spindle speed: 8000
- FPT: 0.0047
- SFM: 1047.1204
- Plunge rate: 50.0
- Retract rate: 6.4176
- Force tool change:
- Rapid Retract:
- Comment: Machine the outside profile.
- To batch:



11.3 Set the Cut Parameters

- From the Tree View list, select **Cut Parameters** and ensure the parameters appear the same as shown in [Figure: 11.3.1](#).

Figure: 11.3.1





11.4 Set the Depth Cuts Parameters

- From the Tree View list, select the **Depth Cuts** parameters and disable **Depth cuts** as shown in [Figure: 11.4.1](#).

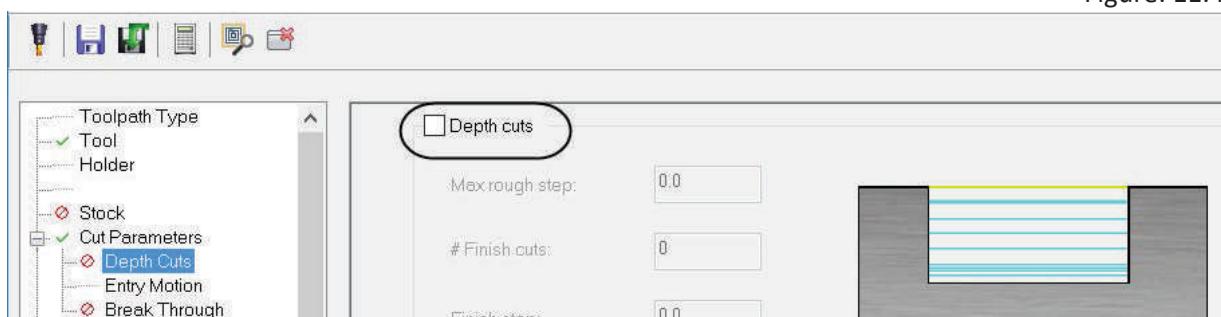


Figure: 11.4.1

11.5 Set the Entry Motion

- From the Tree View list, select **Entry Motion**.
- Set the **Entry method** to **Helix only**. Input a **Z clearance** value of **0.05** and a **Plunge angle** of **2.0** degrees as shown in [Figure: 11.5.1](#).

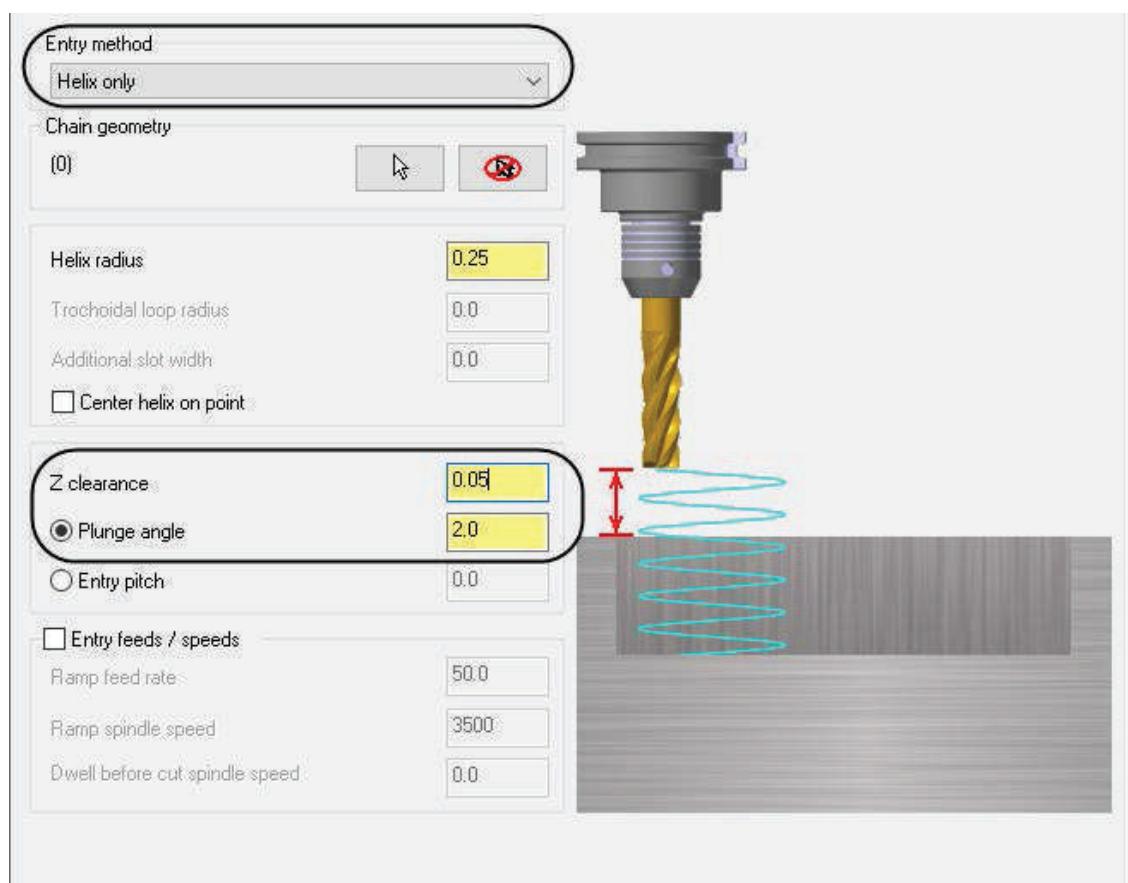


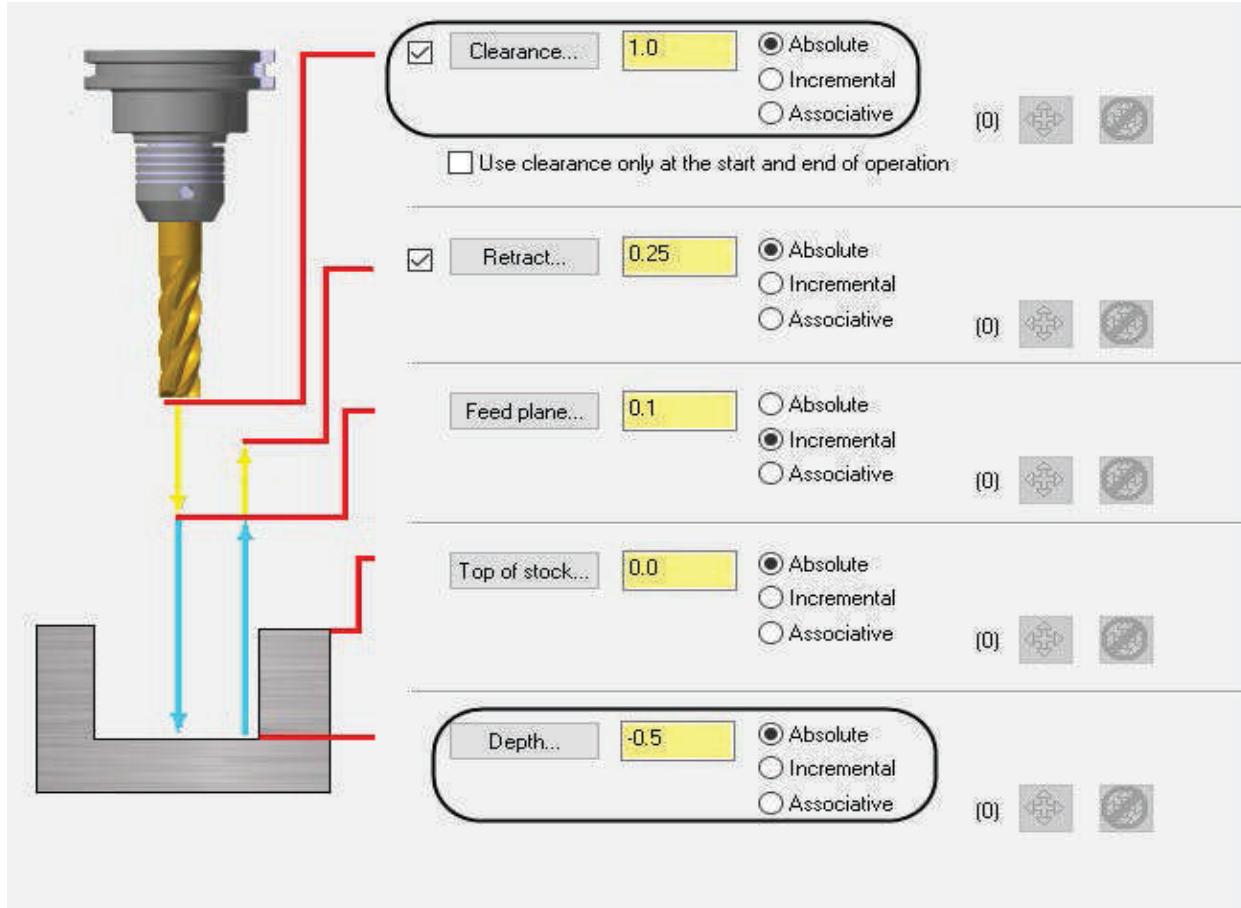
Figure: 11.5.1



11.6 Set the Linking Parameters

- ♦ Select **Linking Parameters**, enable **Clearance**, input a value of **1.0** and input a **Depth** value of **-0.5** as shown in [Figure: 11.6.1](#).

Figure: 11.6.1



11.7 Preview the Toolpath

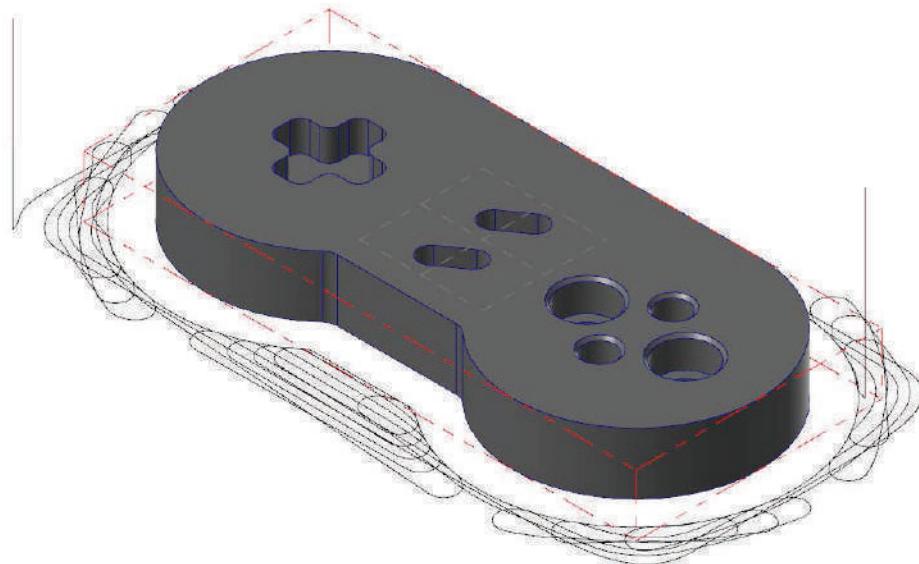
- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.



- ♦ See [page 653](#) to review the procedure.



- The toolpath should look as shown.



- Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

- Select the **OK** button to generate the toolpath.



11.8 Backplot the toolpath

- To **Backplot** the toolpath, see [page 655](#) for more information.



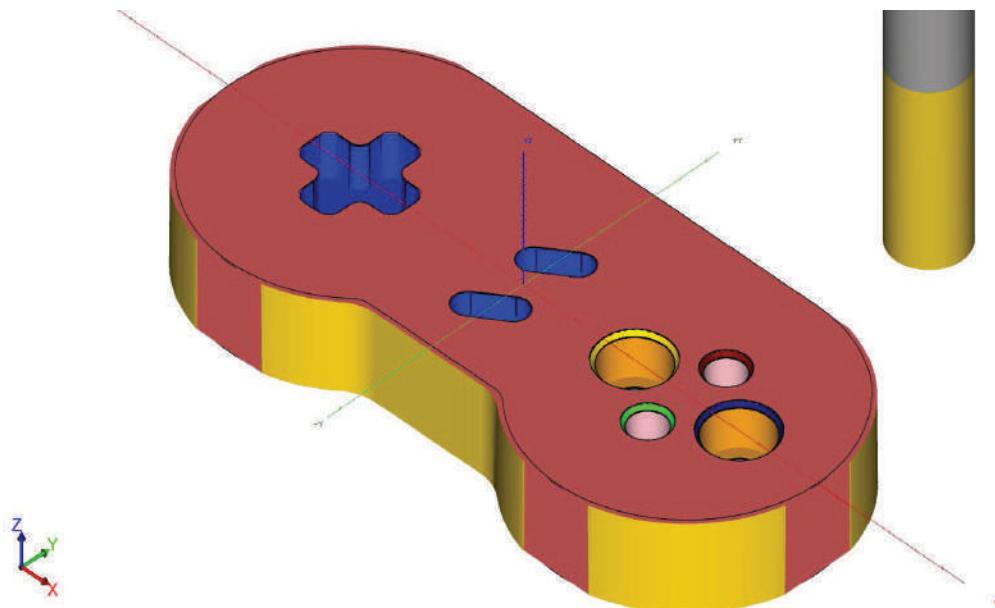
- Select the **OK** button to exit the **Backplot**.





11.9 Verify the toolpaths

- ♦ To **Verify** the toolpaths, see [page 656](#) for more information.
- ♦ To verify all toolpaths, from the **Toolpaths Manager**, choose the **Select all operations** icon.

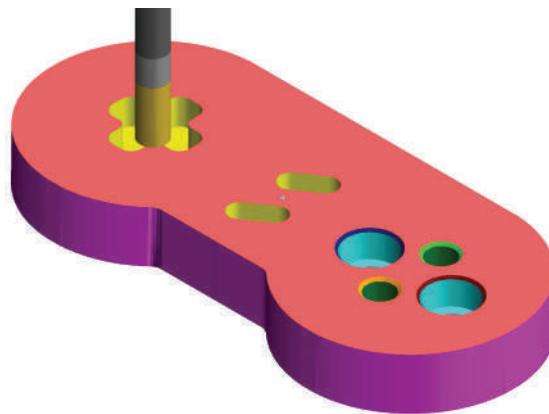


- ♦ To go back to the Mastercam window, minimize the **Mastercam Simulator** window as shown.

STEP 12: CONTOUR TOOLPATH

A **Contour** toolpath removes material along a path defined by a chain of curves. A **Contour** toolpath only follows a chain; it does not clean out an enclosed area. You will use this toolpath to finish the outside profile.

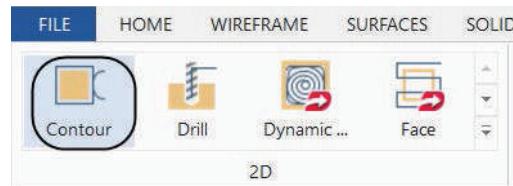
Toolpath Preview:



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

**TOOLPATHS**

- From the **2D** group, select **Contour** as shown.

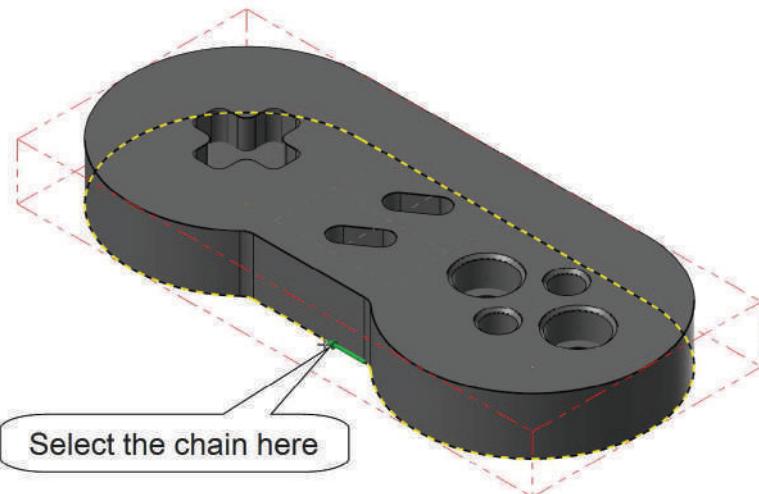


- Select **Wireframe** and enable **C-plane** in the **Chaining** dialog box as shown.



- Select the chain and ensure the chaining direction is the same as shown in [Figure: 12.0.1](#).

Figure: 12.0.1



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

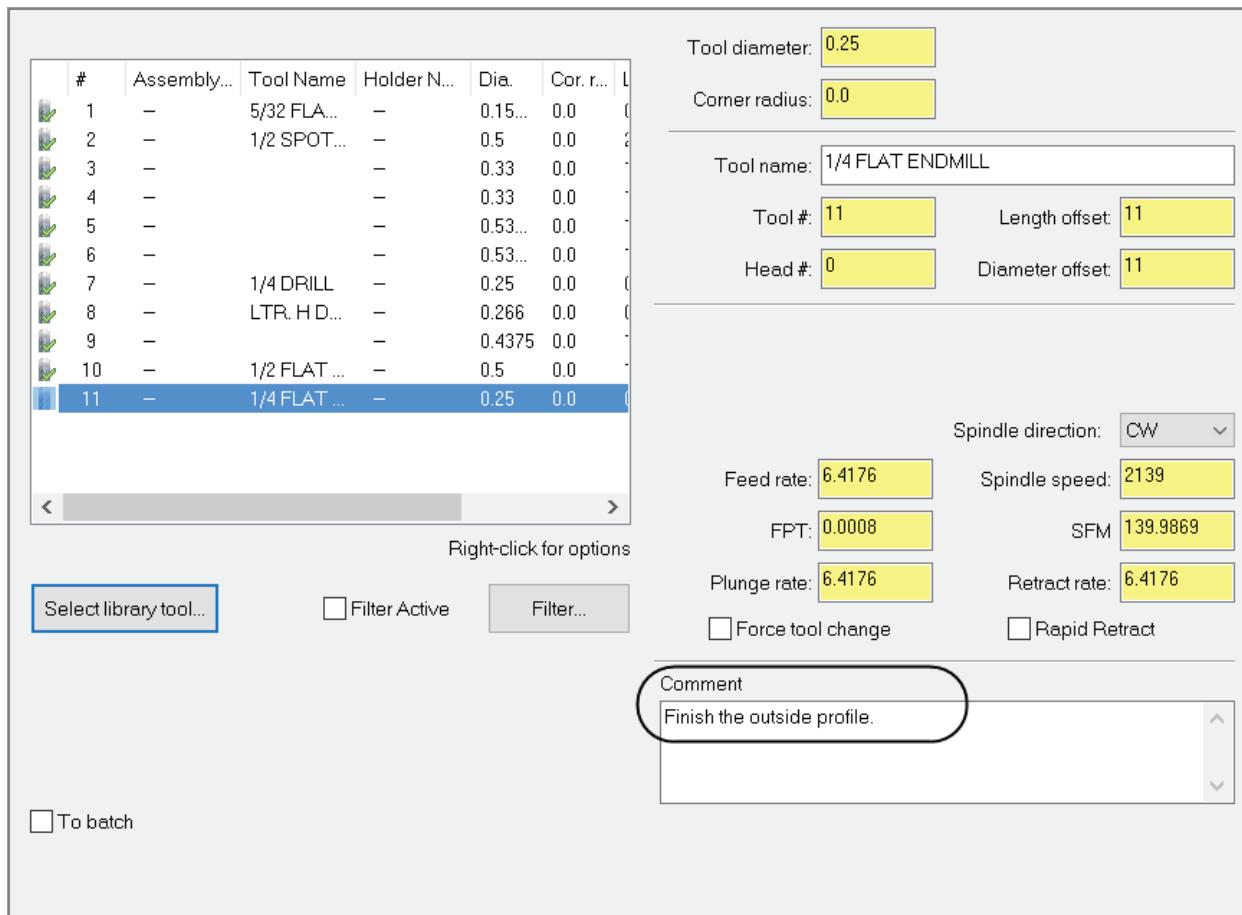




12.1 Select the 1/4" Flat Endmill from the library and set the Tool Parameters

- From the **Tree View** list, select **Tool**.
- Using the **Filter** option, select the **1/4" Flat Endmill** from the library as shown in the previous steps.
- Make all the necessary changes as shown in [Figure: 12.1.1](#).

Figure: 12.1.1

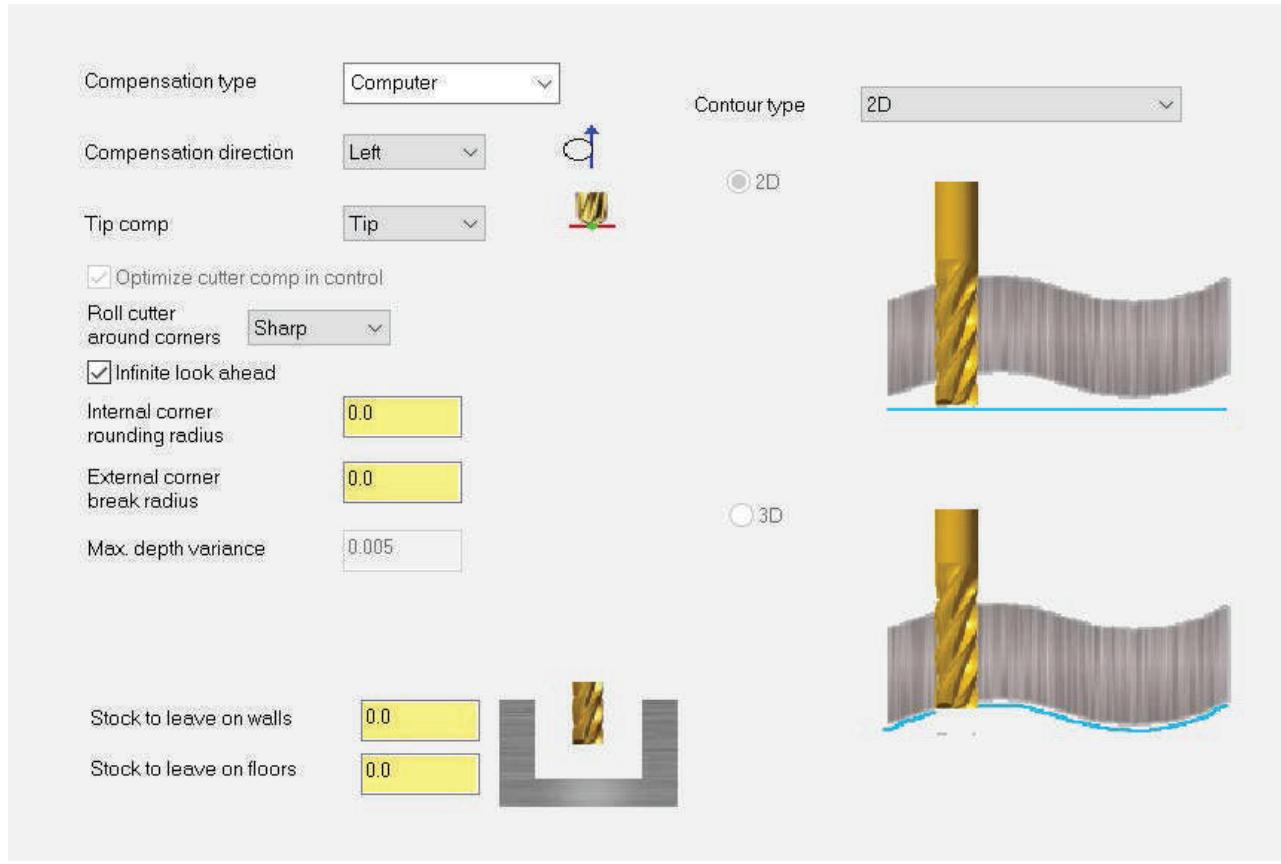




12.2 Cut Parameters

- ♦ Select the **Cut Parameters** and make the necessary changes as shown in [Figure: 12.2.1](#).

Figure: 12.2.1



Roll cutter around corners inserts arc moves around corners in the toolpath.

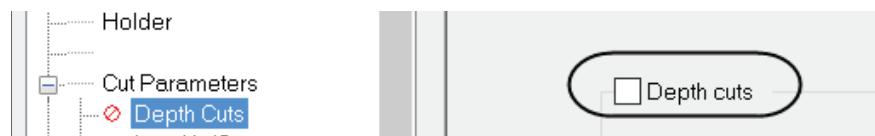
None guarantees all sharp corners.

Sharp rolls the tool around sharp corners (135 degrees or less).

All rolls the tool around all corners and creates smooth tool movement.

12.3 Depth Cuts

- ♦ Select **Depth Cuts** and make sure it is disabled as shown.

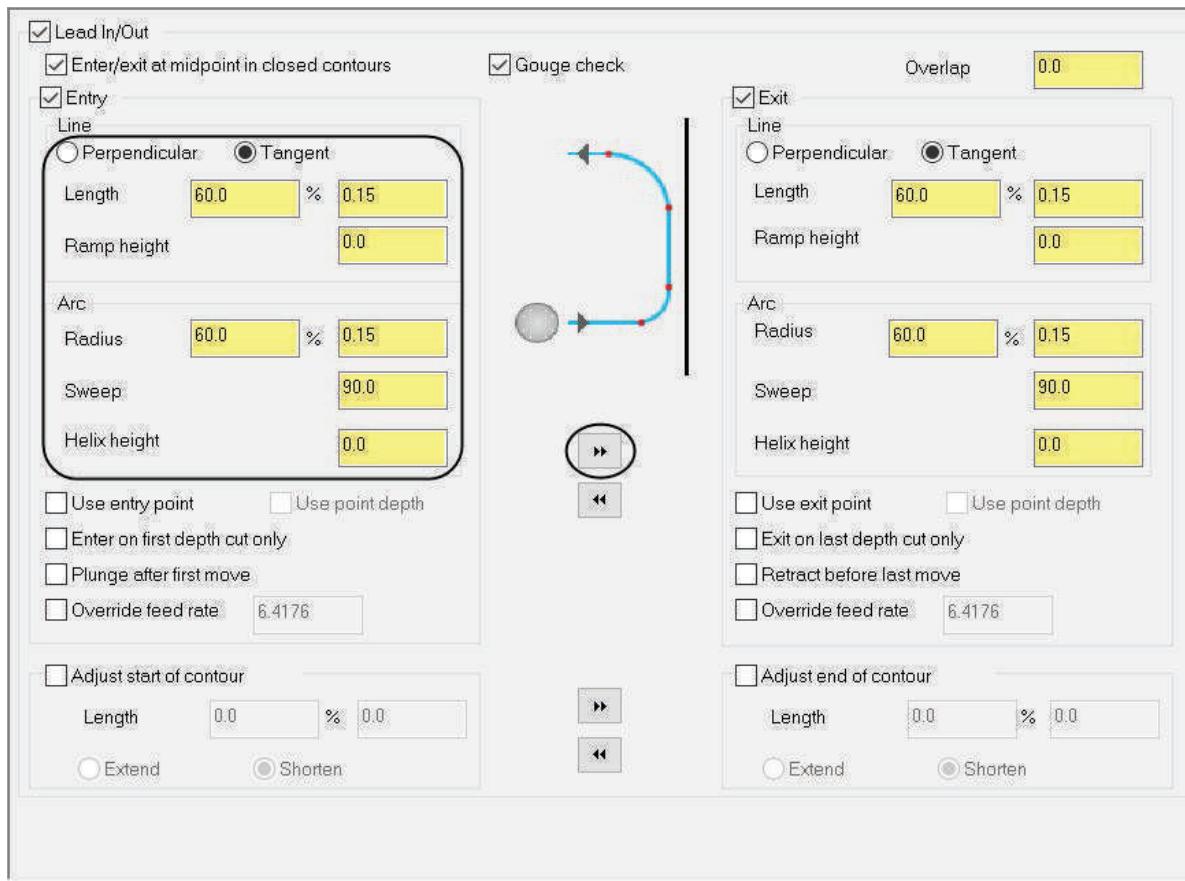




12.4 Lead In/Out

- Enable **Lead In/Out** and make any other necessary changes as shown in [Figure: 12.4.1](#).

Figure: 12.4.1



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

Length set to 60% of the tool diameter ensure that the linear movement is bigger than the tool radius in case **Cutter Compensation** in **Control** was used.

Radius set to 60% of the tool diameter ensures that the arc movement is bigger than the tool radius to generate an arc output.

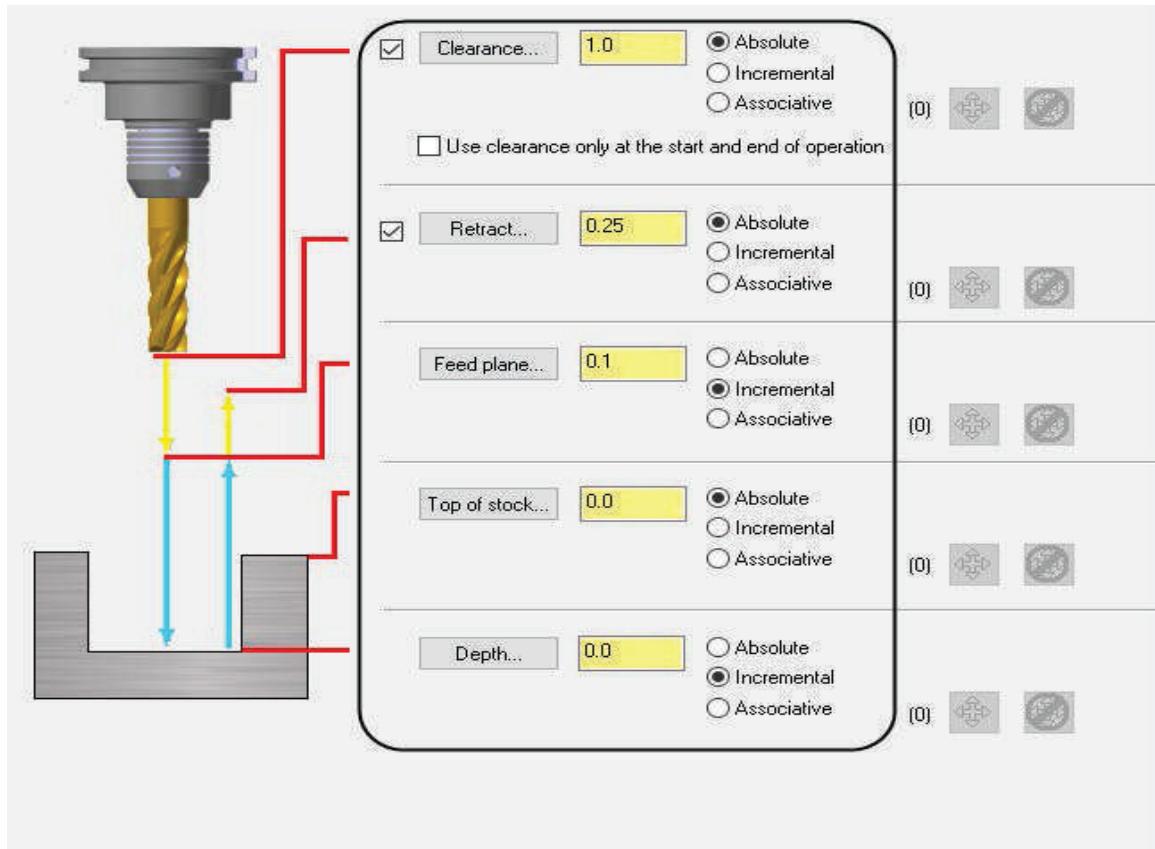
Overlap sets how far the tool goes past the end of the toolpath before exiting for a cleaner finish.



12.5 Linking Parameters

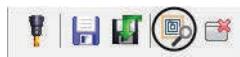
- ♦ Select **Linking Parameters** from the **Tree View** list.
- ♦ Set the **Top of stock** to **Absolute** and **0.0** and the **Depth** to **Incremental** and **0.0** as shown in [Figure: 12.5.1](#).

Figure: 12.5.1



12.6 Preview the Toolpath

- ♦ To quickly check how the toolpath will be generated, select the **Preview toolpath** icon as shown.

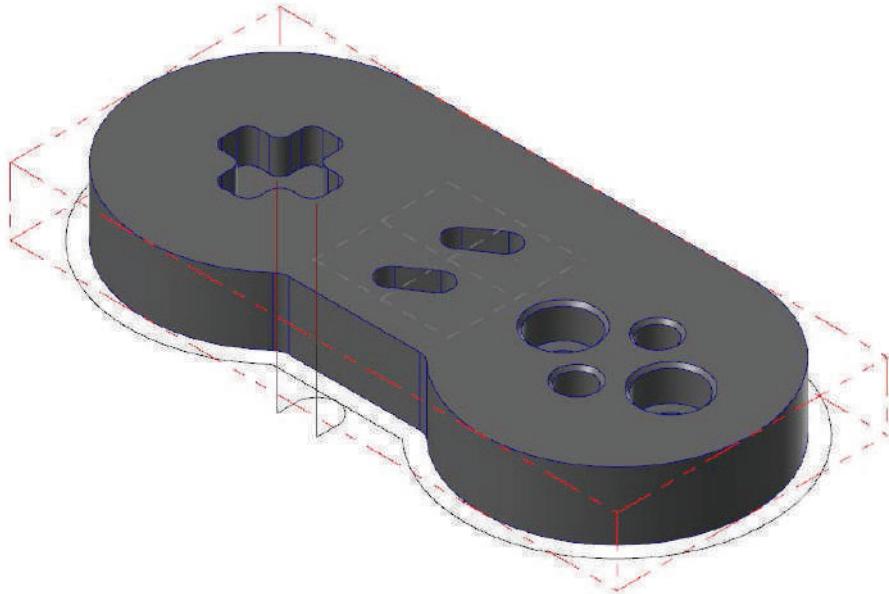


- ♦ Select the **Hide dialog** button to see the preview in the graphics window.





- ◆ The toolpath should look as shown.



- ◆ Press **Esc** key to exit the preview.

NOTE: If the toolpath does not look as shown in the preview, check your parameters again.

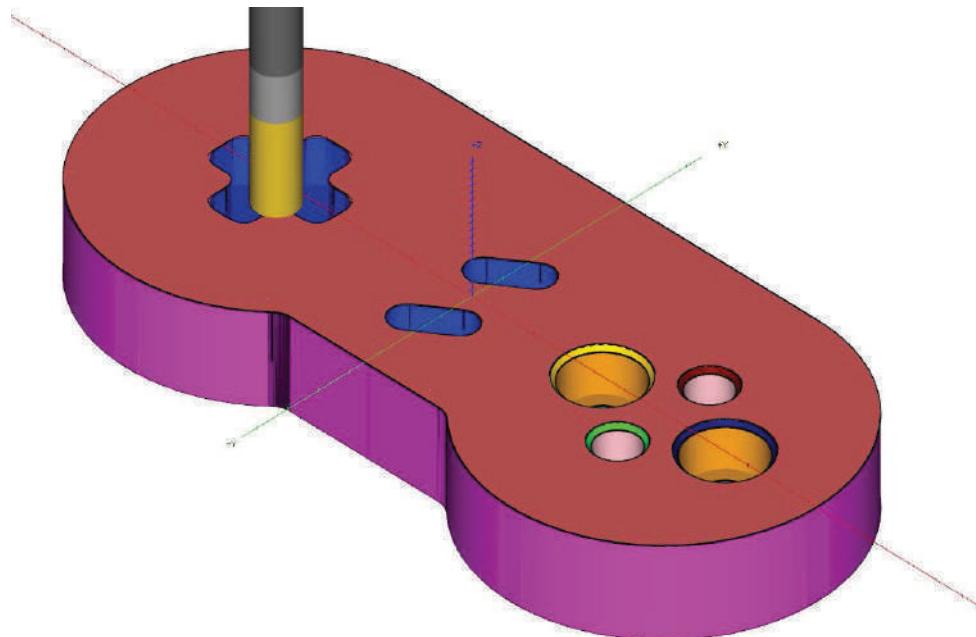
- ◆ Select the **OK** button to exit the **2D Toolpaths - Contour** parameters. 



12.7 Verify the toolpaths

- ◆ Choose **Select all operations** button.
- ◆ Click on the **Verify selected operations** icon.
- ◆ For information on how to set the verify parameters and simulate the toolpath, please check [page 147](#).
- ◆ The finished part will appear as shown in [Figure: 12.7.1](#).

Figure: 12.7.1



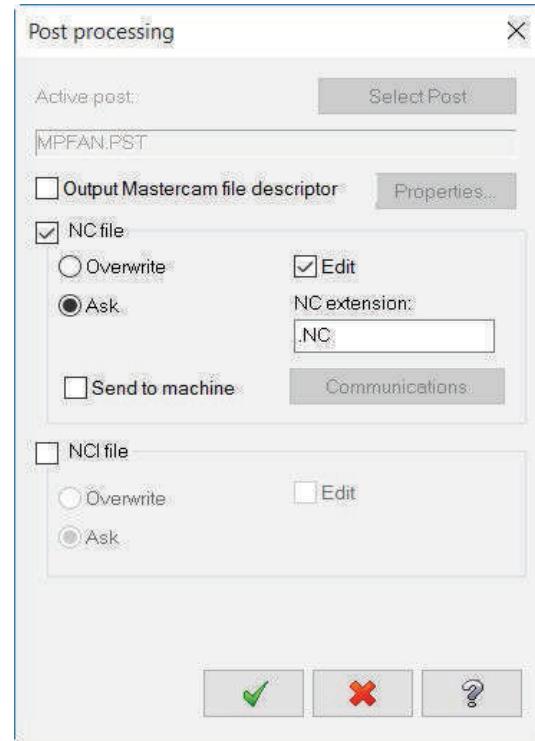
- ◆ To go back to the Mastercam window, close the **Mastercam Simulator** window as shown.



STEP 13: POST THE FILE

- ◆ Ensure all operations are selected. If they are not, use the **Select all operations** button in the **Toolpaths Manager**. 
- ◆ Select the **Post selected operations** button from the **Toolpaths Manager**. 
- ◆ In the **Post processing** window, make the necessary changes as shown in [Figure: 13.0.1](#).

Figure: 13.0.1



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

Edit enabled allows you to automatically launch the default editor.

- ◆ Select the **OK** button to continue. 
- ◆ If an **Operations Manager** box appears, select **Yes** to continue.
- ◆ Save "Your Name_7.NC" file. 



- ♦ A window with **Mastercam Code Expert** will be launched and the NC programs will appear as shown in [Figure: 13.0.2](#).

Figure: 13.0.2

The screenshot shows the Mastercam Code Expert interface with the title bar "Tutorial #7.NC - Mastercam 2018 Code Expert". The menu bar includes File, Home, View, and Editor. The Editor tab is selected, showing the NC code for the part. The code consists of several G-code commands, including tool changes (T1-T15), drilling operations (G01, G02, G03), and dynamic mill operations (G120). The code is color-coded for readability. The status bar at the bottom right indicates the file size is 304.14KB and the zoom level is 100%.

```
1 G8
2 O0007 (TUTORIAL #7)
3 (DATE=02-02-17 TIME=16:38:00)
4 (NCX FILE = \VHSSERVER06\BOOKDEV\PRODUCTION\BOOKS\2018\TRAINING TUTORIALS\TT CAD IMPORT MILL ESSENTIALS TOOLPATHS\MCX FILES\INCH\TUTORIAL TOOLPATH INCH\TUTORIAL_
5 (NC FILE = C:\USERS\NINGUE\DESKTOP\TUTORIAL #7.MC)
6 (MATERIAL = ALUMINUM INCH - 2024)
7 ( T1 | 1/2 FLAT ENDMILL | H1 )
8 ( T3 | 1/8 SPOTDRILL | H3 )
9 ( T4 | 1/4 DRILL | H4 )
10 ( T5 | 1/8 H DRILL | H5 )
11 ( T8 | H8 )
12 ( T9 | H9 )
13 ( T10 | H10 )
14 ( T11 | H11 )
15 ( T12 | H12 )
16 ( T13 | 1/2 FLAT ENDMILL | H15 | XY STOCK TO LEAVE - .05 | Z STOCK TO LEAVE - 0. )
17 ( T14 | 1/4 FLAT ENDMILL | H14 )
18 N100 G20
19 N110 G0 G17 G40 G49 G80 G90
20 ( MACHINE THE 3 POCKETS USING DYNAMIC MILL. )
21 N120 T1 M6
22 N130 G0 G90 G54 X-1.29 Y-.0377 A0. S3428 M3
23 N140 G43 H1 Z1.
24 N150 Z125
25 N160 G1 X-1.2587 Y-.039 Z.1245 F6.16
26 N170 X-1.256 Y-.0417 Z.1248
27 N180 X-1.2534 Y-.0444 Z.1247
28 N190 X-1.2508 Y-.0472 Z.1249
29 N200 X-1.2783 Y-.05 Z.1244
30 N210 X-1.2759 Y-.052 Z.1243
31 N220 X-1.2734 Y-.0556 Z.1241
32 N230 X-1.2711 Y-.0586 Z.124
33 N240 X-1.2687 Y-.0615 Z.1239
34 N250 X-1.2676 Y-.0639 Z.1238
35 N260 X-1.2665 Y-.0643
36 N270 X-1.2653 Y-.0655 Z.1237
37 N280 X-1.2641 Y-.0667 Z.1236
38 N290 X-1.2629 Y-.0678
39 N300 X-1.2618 Y-.0688 Z.1235
40 N310 X-1.2606 Y-.0696
41 N320 X-1.2595 Y-.0704 Z.1234
42 N330 X-1.2589 Y-.0708
43 N340 X-1.2583 Y-.0711
44 N350 X-1.2577 Y-.0715
45 N360 X-1.2571 Y-.0717 Z.1233
46 N370 X-1.2566 Y-.072
```

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

STEP 14: SAVE THE UPDATED MCAM FILE

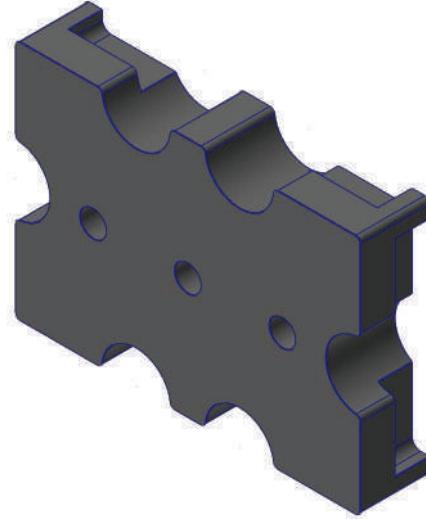


REVIEW EXERCISE - STUDENT PRACTICE

IMPORT THE GEOMETRY FOR TUTORIAL #7 EXERCISE

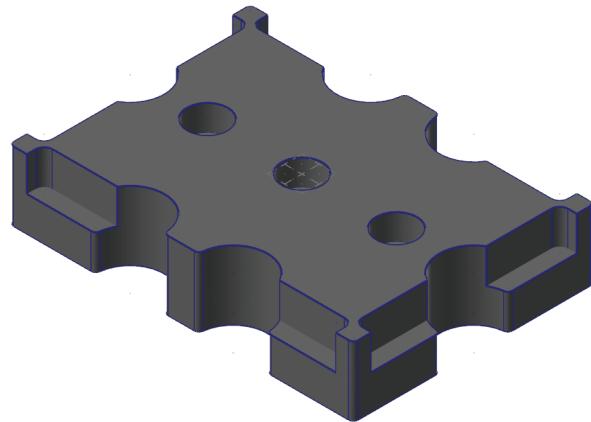
Import CAD Model and prepare it for machining.

- ◆ Open the file from SolidWorks.



Rotate between views.

- ◆ Use TRANSFORM/Translate to Plane to rotate the part between views.
- ◆ Set the Source View to Back and the Destination View to Top.

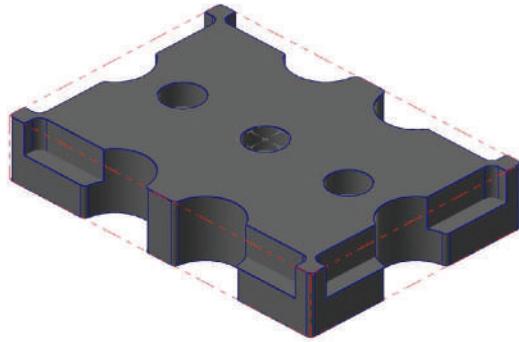




CREATE THE TOOLPATHS FOR TUTORIAL #7 EXERCISE

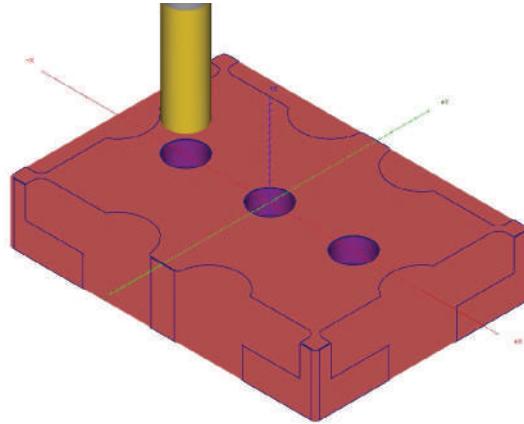
Create the Toolpaths for Tutorial #7 Exercise as per the instructions below. Some of these toolpaths may require a longer tool length. You can determine this by analyzing the toolpath through **Verify** or **Backplot**, ensuring that the tool's cutting length is greater than the depth of stock which it is cutting. Refer to [page 708](#) for more information on how to do this.

Set the Machine Properties including the Stock.



FBM Drill.

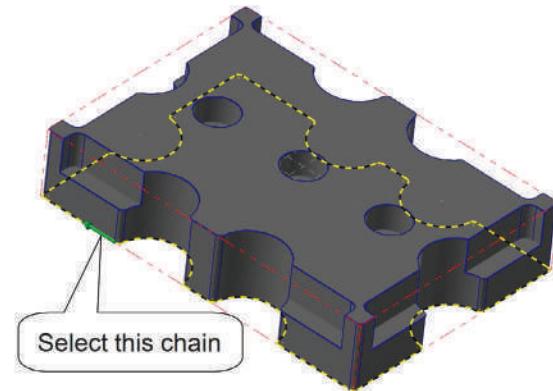
- ◆ Enable **Automatic initial hole detection**.
- ◆ **Grouping** set to **Plane**.
- ◆ Under **Hole Detection** enable **Include blind holes**.
- ◆ Enable **Include chamfers**.
- ◆ **Maximum diameter** set to **1.5**.
- ◆ **Spot drilling** and **Combine Spot Drill operations** that share the same depth and retract values enabled.
- ◆ **Max percentage of finished hole** set to **100%** and **Max depth** set to **0.375**.
- ◆ **Pre-drilling** and **Pre-drill pilot holes only** enabled.
- ◆ In **Tool selection**, have **Use tools in part file**, **Use tool library**, and **Create as needed** enabled.
- ◆ In **Tool creation**, disable **Create only standard sizes**, enable **Consider flute length**, and **Create one tool for all depths**.
- ◆ In the **Linking Parameters**, have **Clearance** set to **Solid height plus** and **1.0** and **Retract** set to **Solid height plus** and **0.5**.
- ◆ Enable **Through holes**.
- ◆ In **Features**, click on **Detect**.



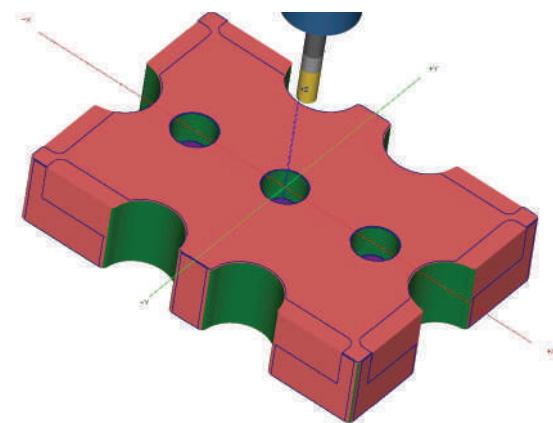


Machine the outside profile using 2D HS Dynamic Mill.

- ♦ In the Chain Options, enable **From outside** and **Shortest distance** for **Open chain extension to stock**.
- ♦ Click on the **Select** button in the **Avoidance regions** and with **C-plane** enabled, select the bottom contour.



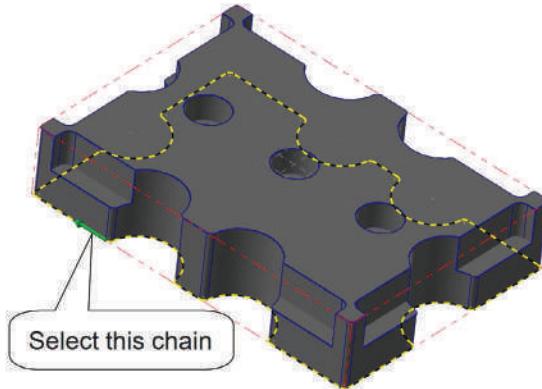
- ♦ Select the **1/2" Flat Endmill**. Change the length of this tool accordingly.
- ♦ **First pass offset = 0.1**.
- ♦ **Stepover = 25%**.
- ♦ **Min toolpath radius = 10%**.
- ♦ **Micro lift distance = 0.01**.
- ♦ **Stock to leave on walls = 0.03**.
- ♦ **Stock to leave on floors = 0.0**.
- ♦ **Entry motion set to Helix only; Radius 0.25**.
- ♦ **Clearance = 1.0 (Absolute)**.
- ♦ **Retract = 0.25 (Absolute)**.
- ♦ **Feed plane = 0.1 (Incremental)**.
- ♦ **Depth = -2.0 (Absolute)**.



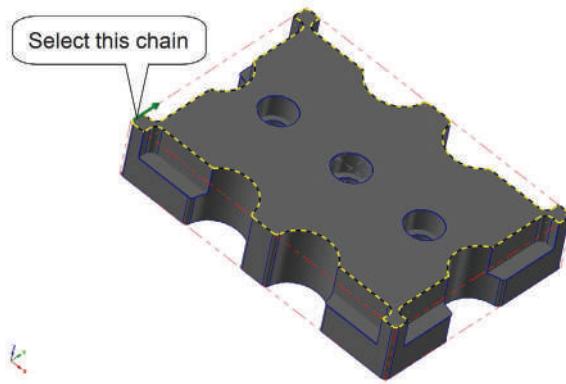


Machine the steps using 2D HS Dynamic Mill.

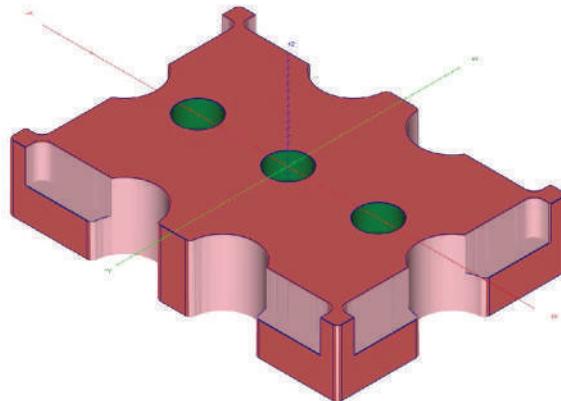
- ♦ In the **Chain Options**, enable **From outside, None (ignore stock)** for extension.
- ♦ Click on the **Select** button in the **Machining regions** and with **C-plane** enabled, select the bottom contour.



- ♦ Click on the **Select** button in the **Avoidance regions** and with **C-plane** enabled, select the top contour.

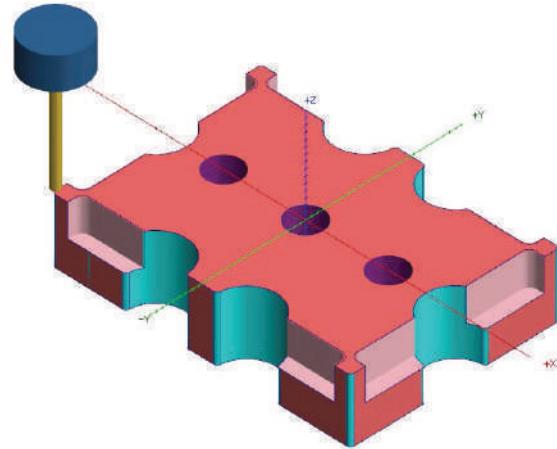


- ♦ Select the same **1/2" Flat Endmill**.
- ♦ **First pass offset = 0.1**.
- ♦ **Stepover = 25%**.
- ♦ **Min toolpath radius = 10%**.
- ♦ **Micro lift distance = 0.01**.
- ♦ **Stock to leave on walls = 0.03**.
- ♦ **Stock to leave on floors = 0.0**.
- ♦ **Entry motion set to Helix only; Radius 0.25**.
- ♦ **Clearance = 1.0 (Absolute)**.
- ♦ **Retract = 0.25 (Absolute)**.
- ♦ **Feed plane = 0.1 (Incremental)**.
- ♦ **Top of stock = 0.0 (Absolute)**.
- ♦ **Depth = -1.0 (Absolute)**.



**Finish the bottom profile using Contour toolpath.**

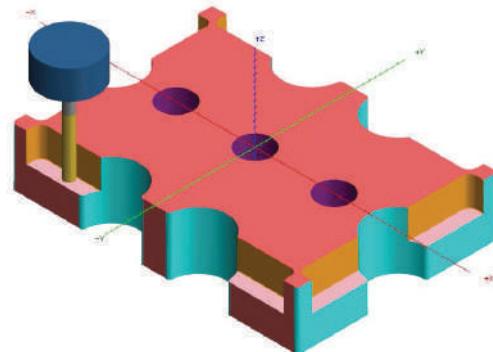
- ◆ Enable **C-plane** and select the bottom profile as shown before in the **ClockWise** direction.
- ◆ Select a **3/8" Flat Endmill**. Change the length of this tool accordingly.
- ◆ **Contour type 2D**.
- ◆ **Compensation type** set to **Computer**.
- ◆ **Compensation direction** set to **Left**.
- ◆ **Stock to leave on either walls or floors = 0.0**.
- ◆ **Lead In/Out** set to defaults.
- ◆ **Clearance = 1.0** (Absolute).
- ◆ **Retract = 0.25** (Absolute).
- ◆ **Feed plane = 0.1** (Incremental).
- ◆ **Top of stock = 0.0** (Absolute).
- ◆ **Depth = 0.0** (Incremental).



NOTE: Change the **Cutting length** and **Overall length** as needed. To change the default tool properties, in the tool list, right mouse click on the tool and select **Edit tool** to set tool parameters.

Finish the step profile using Contour toolpath.

- ◆ Enable **C-plane** and select the top profile as shown before in the **ClockWise** direction.
- ◆ Use the same **3/8" Flat Endmill**.
- ◆ **Contour type 2D**.
- ◆ **Compensation type** set to **Computer**.
- ◆ **Compensation direction** set to **Left**.
- ◆ **Stock to leave on either walls or floors = 0.0**.
- ◆ **Lead In/Out** set to defaults.
- ◆ **Clearance = 1.0** (Absolute).
- ◆ **Retract = 0.25** (Absolute).
- ◆ **Feed plane = 0.1** (Incremental).
- ◆ **Top of stock = 0.0** (Absolute).
- ◆ **Depth = -1.0** (Absolute).





NOTES:

**TUTORIAL #7 QUIZ**

- ◆ What does FBM Drill allow you to do?

- ◆ What does Translate 3D command do?

- ◆ What does Avoidance regions selection do?

CREATING/EDITING TOOLS

Objectives:

- ✓ The student will learn how to create and modify tools.

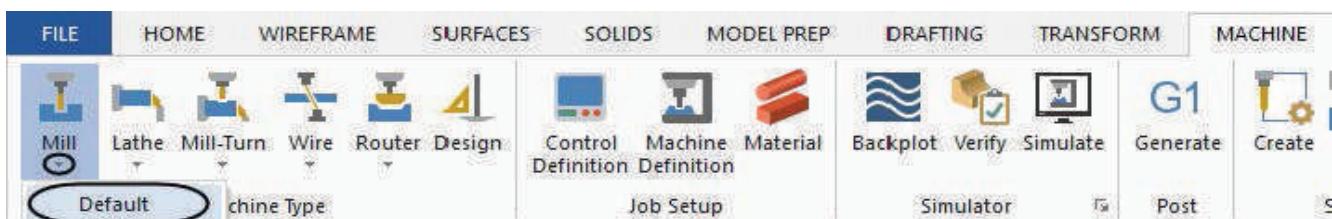
CREATING AND EDITING A MILL TOOL LIBRARY

NOTE: The purpose of tool libraries is to hold the tool data. The libraries can be edited or added to by following the directions below. Each time the **Tool Type** or **Cutter diameter** is changed in the **Toolpath parameters**, the tool library recalculates the feed rate and spindle speed. The following menu selections will allow you to create a new tool.

CREATING A NEW TOOL

MACHINE

- ♦ Default.

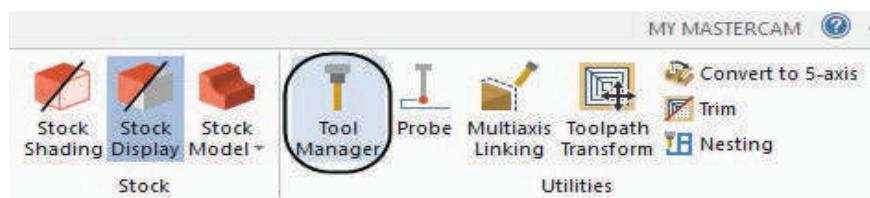


NOTE: Once you select the **Mill Default** the ribbon bar changes to reflect the toolpaths that could be used with **Mill Default**.

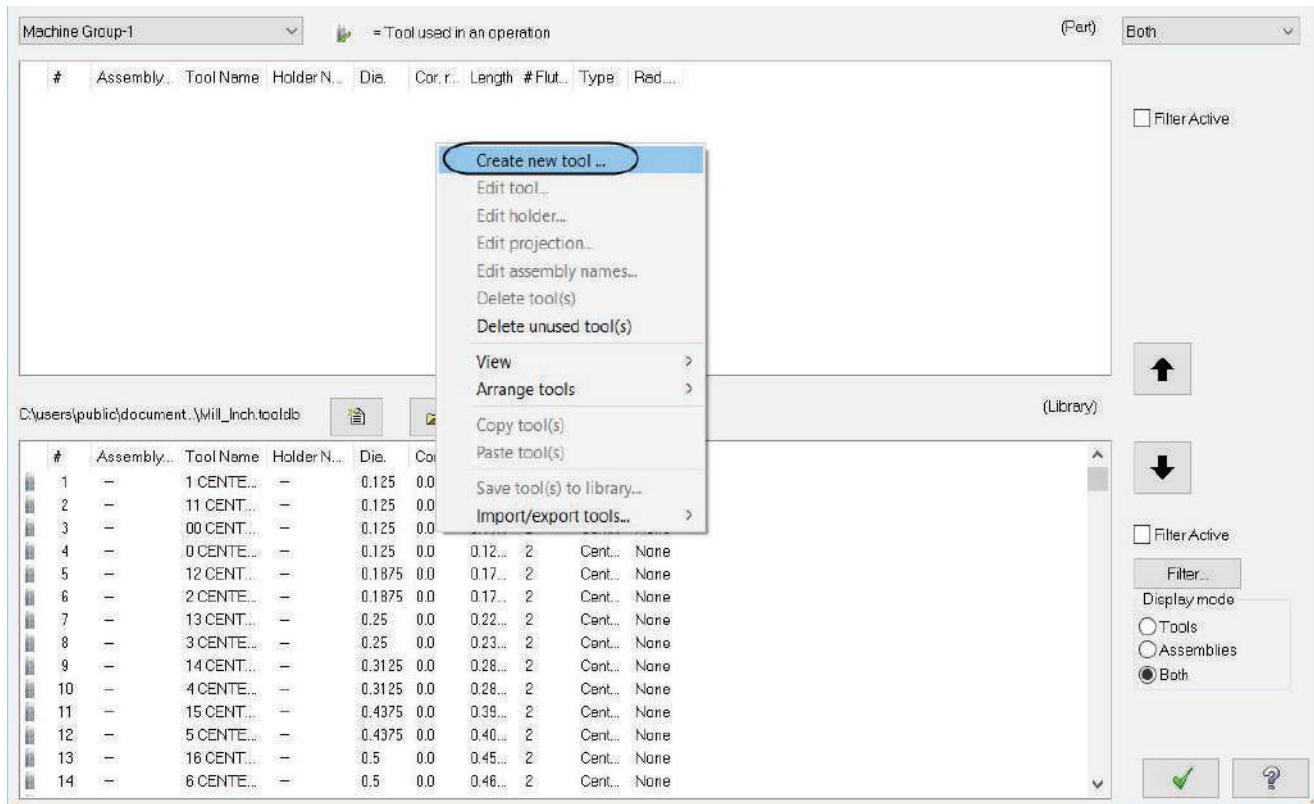
Option 1: Create a new tool using Mill Tool Manager

TOOLPATHS

- ♦ From the Utilities group, select **Tool Manager** as shown.



- ♦ Right click in the tool display area.
- ♦ Select **Create new tool** as shown.



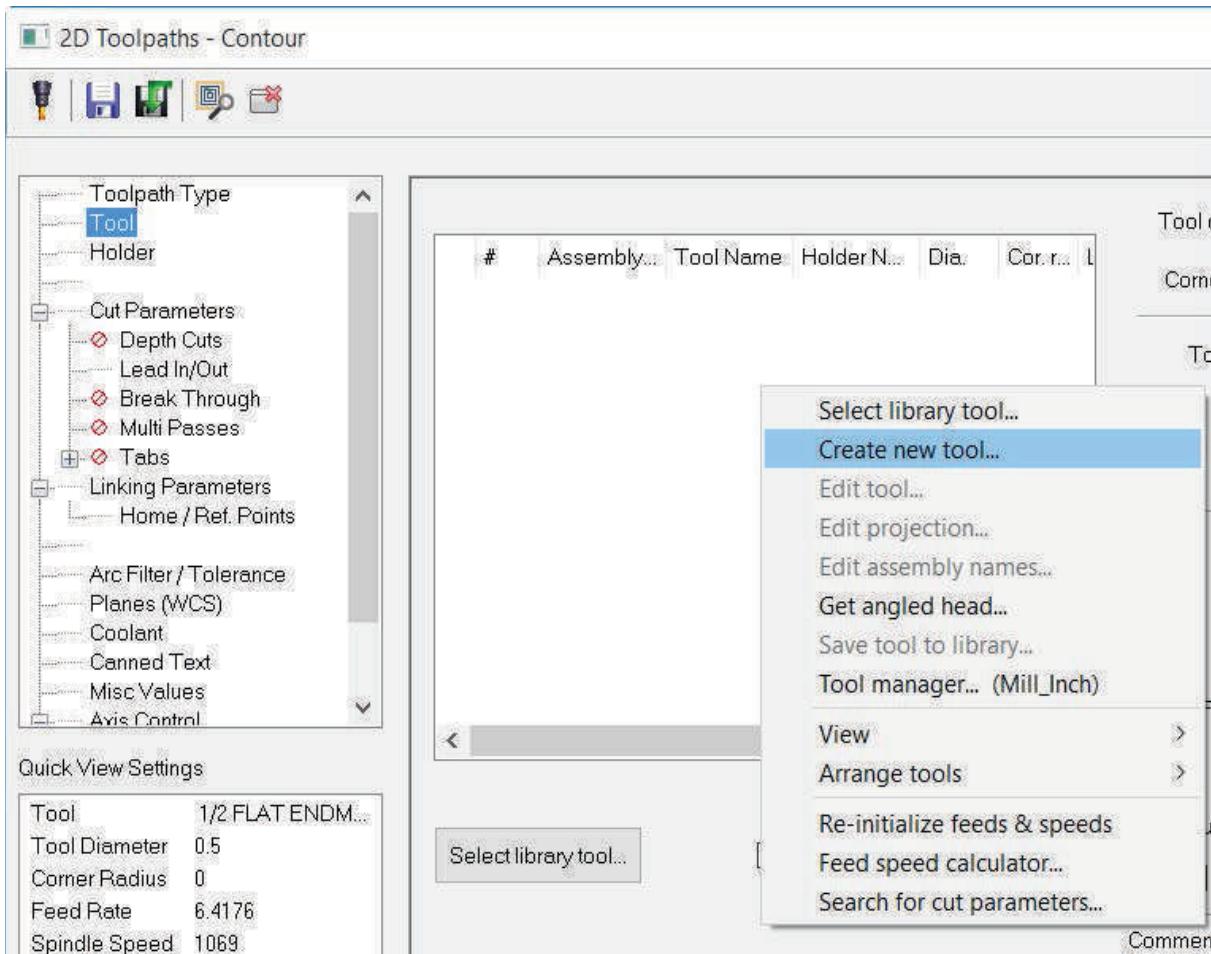
Option 2: Create a new tool inside of the Toolpath Parameters

Close the Tool Manager.

Create a rectangle.

TOOLPATHS

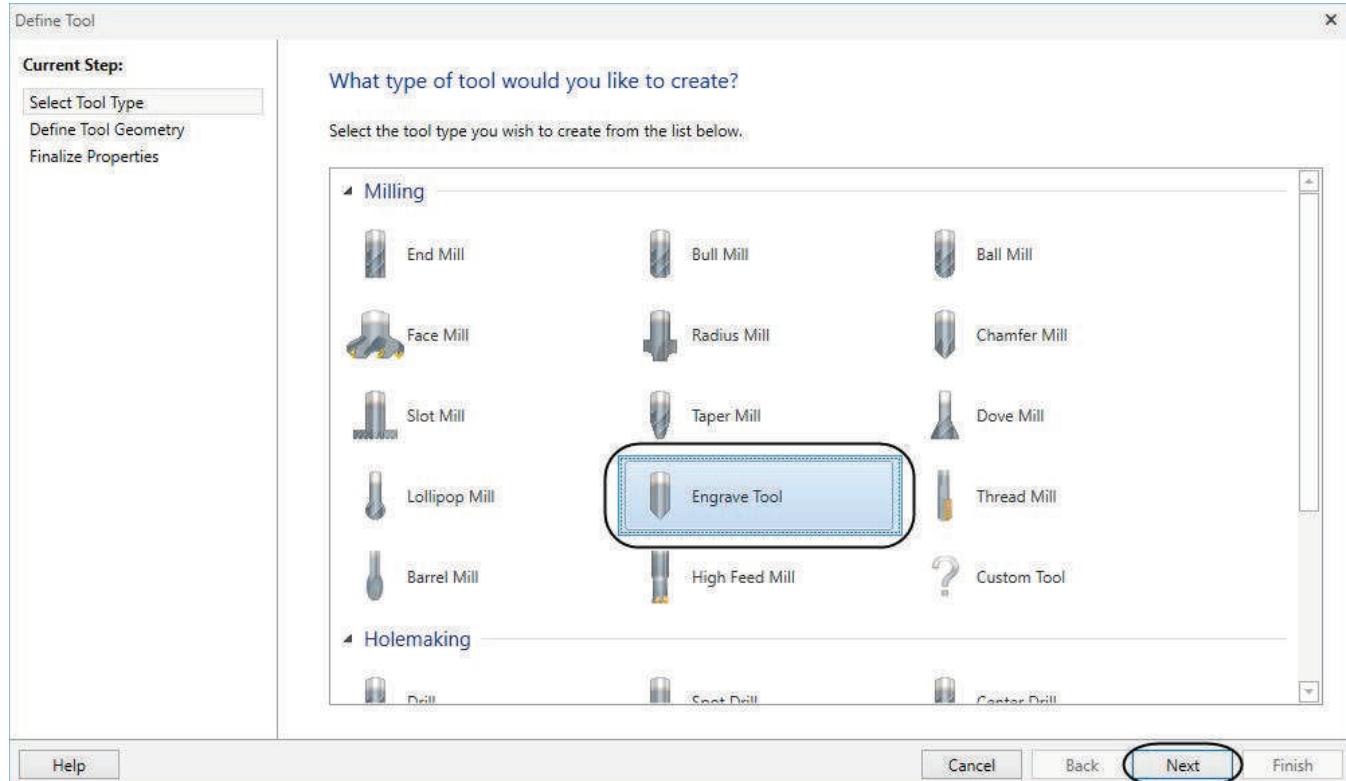
- ◆ From the **2D** area, select **Contour**.
- ◆ Chain the rectangle.
- ◆ From the **Tree View** list, select **Tool**.
- ◆ Right click in the tool display area.
- ◆ Select **Create new tool** as shown.



1. What type of tool would you like to create?

The **Create New Tool** dialog box displays all the default tool type options in Mastercam.

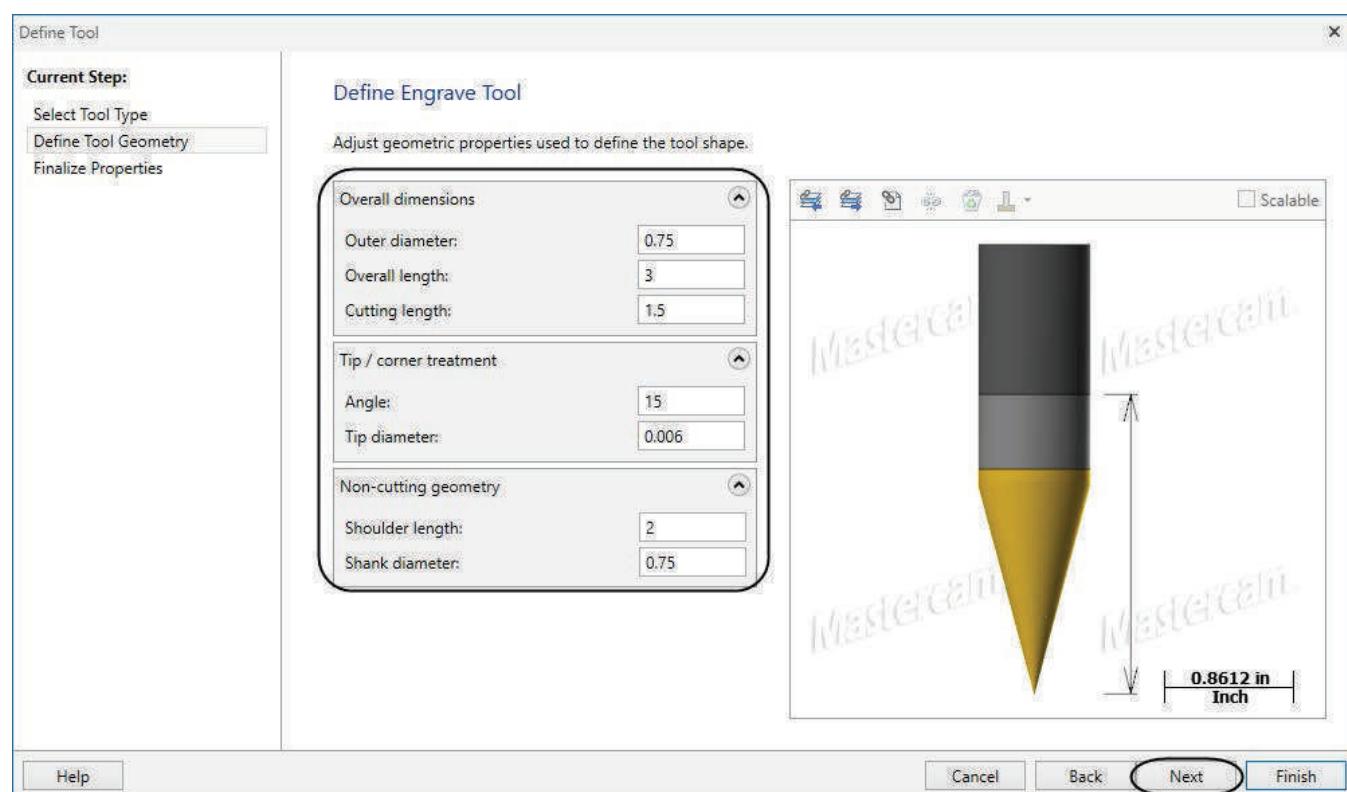
- ♦ The **Define Tool** screen will appear as shown.



- ♦ Choose the tool type you wish to modify and then select the **Next** button.

2. Define geometric tool parameters.

Define geometric tool parameters lets you enter new parameters or edit current parameters of Mill tools. A description of each parameter in the **Define geometric tool parameters** is listed below.



NOTE: The fields on the tabs change depending on the type of tool you are creating.

Overall Dimensions

Outer Diameter - sets the diameter of the tool.

Overall Length - sets the length of the tool.

Cutting Length - sets the length from the top of the flutes to the tip of the tool.

Tip/Corner Treatment

Corner Type - allows you to choose between None, Chamfer, Corner radius and Full radius.

Angle - measures the angle from the center line of the tool to the outer angle of the tool.

Tip Diameter - sets the tool tip diameter.

Non-Cutting Geometry

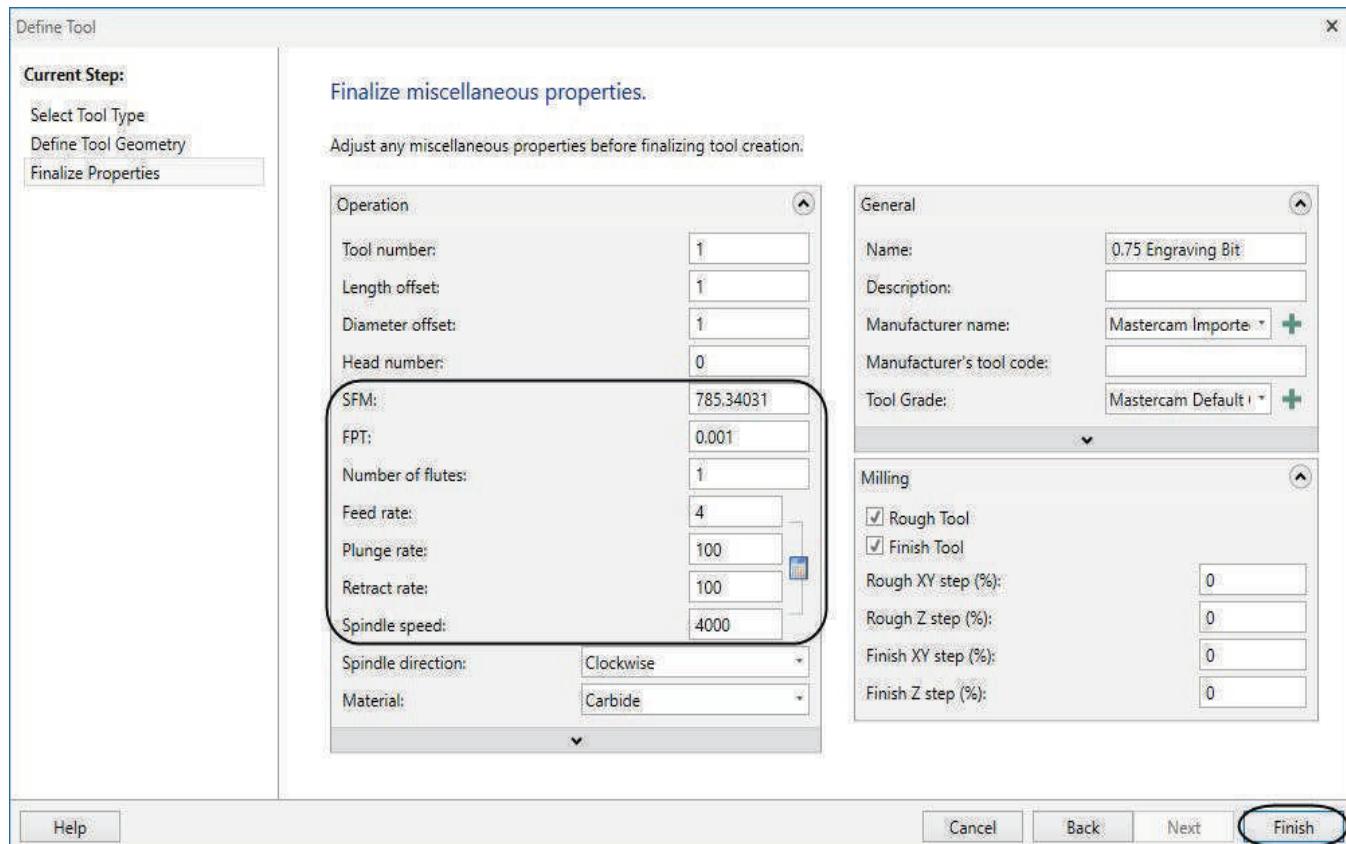
Shoulder Length - sets the distance from the top of the shoulder to the tip of the tool.

Shank Diameter - sets the diameter of the tool shank.

- ◆ Set the dimension values and then select the **Next** button.

3. Finalize miscellaneous properties

This area allows you to type information such as Name, Manufacturer Name and Manufacturer's tool code. It also allows you to enter the tool offset numbers, the feeds and speeds, the material, coolant settings and other settings.



NOTE: To set up the coolant, click on the drop down arrow.

- ◆ Make any necessary changes as shown and select the **Finish** button to complete the tool creation.

Operation

Tool Number - sets the default tool assembly number.

Length Offset - sets the number that corresponds to a register in the machine that compensates for the tool length.

Diameter Offset - sets the number that corresponds to a register in the machine that offsets the diameter.

Number of Flutes - specifies the number of flutes on the tool which is used to calculate the feed rate.

Feed Rate - sets the default feed rate.

Plunge Rate - sets the default plunge rate.

Retract Rate - sets the default retract rate.

Spindle Speed - sets the spindle speed in RPM.

Spindle Direction - allows you to choose the spindle direction between Clockwise, Counterclockwise and Static.

Material - displays the tool material.

Coolant - allows you to set the canned text coolant options.

Metric - allows you to enable metric tools.

General

Name - allows you to type the name of the tool.

Description - displays additional info.

Manufacturer Name - allows you to type the name of the Manufacturer.

Manufacturer's Tool Code - allows you to type the tool code.

Milling

Rough XY step (%)- allows you to set the size of a roughing step in the X and Y axes for the tool. The system measures this distance as a percentage of the tool diameter.

Rough Z step (%) - allows you to set the size of a roughing step in the Z axis for the tool. The system measures this distance as a percentage of the tool diameter.

Finish XY step (%) - allows you to set the size of a finish step in the X and Y axes for the tool. The system measures this distance as a percentage of the tool diameter.

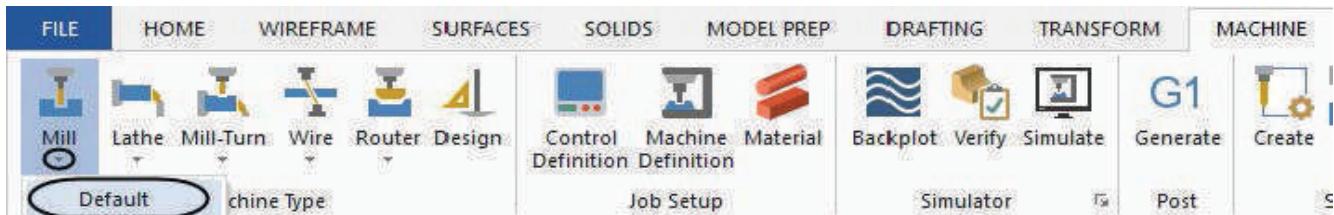
Finish Z step (%) - allows you to set the size of a step in the Z axis for finish the tool. The system measures this distance as a percentage of the tool diameter.

EDITING AN EXISTING TOOL

Option 1: Edit an existing tool using the Tool Manager

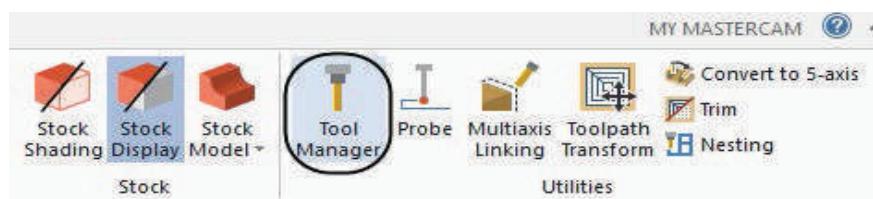
MACHINE

- From the **Machine Type** group, click on the drop down arrow below **Mill** and select the **Default**.



TOOLPATHS

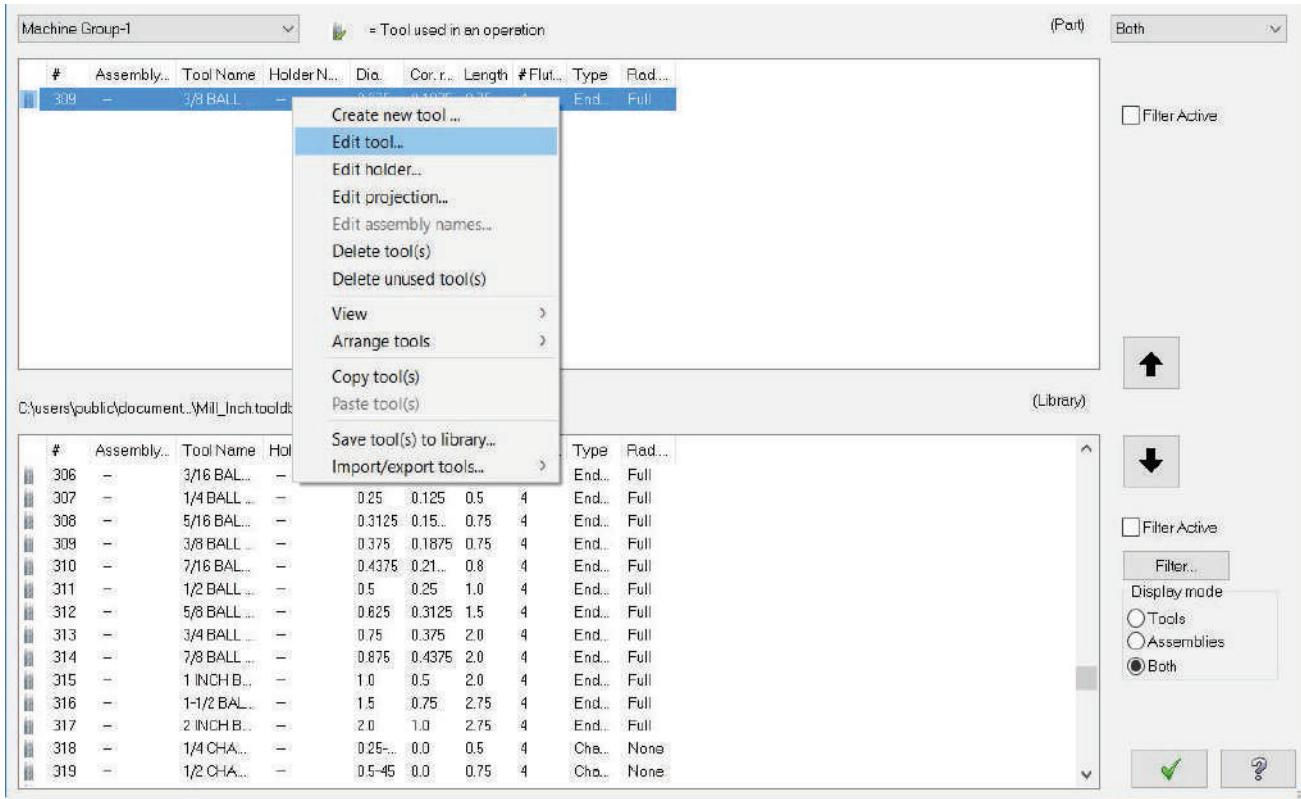
- From the **Utilities** group, select **Tool Manager** as shown.



- Find the **3/8" Ball Endmill (#309)** from the library list and click on the upward arrow to move it in the current tool list as shown.

#	Assembly...	Tool Name	Holder N...	Dia.	Cor. r...	Length	# Flut..	Type	Rod...
306	-	3/16 BAL...	-	0.1875	0.09...	0.4375	4	End...	Full
307	-	1/4 BALL ...	-	0.25	0.125	0.5	4	End...	Full
308	-	5/16 BAL...	-	0.3125	0.15...	0.75	4	End...	Full
309	-	3/8 BALL ...	-	0.375	0.1875	0.75	4	End...	Full
310	-	7/16 BAL...	-	0.4375	0.21...	0.8	4	End...	Full
311	-	1/2 BALL ...	-	0.5	0.25	1.0	4	End...	Full
312	-	5/8 BALL ...	-	0.625	0.3125	1.5	4	End...	Full
313	-	3/4 BALL ...	-	0.75	0.375	2.0	4	End...	Full
314	-	7/8 BALL ...	-	0.875	0.4375	2.0	4	End...	Full
315	-	1 INCH BL...	-	1.0	0.5	2.0	4	End...	Full
316	-	1-1/2 BAL...	-	1.5	0.75	2.75	4	End...	Full
317	-	2 INCH BL...	-	2.0	1.0	2.75	4	End...	Full
318	-	1/4 CHA...	-	0.25...	0.0	0.5	4	Cho...	None
319	-	1/2 CHA...	-	0.5-45	0.0	0.75	4	Cho...	None

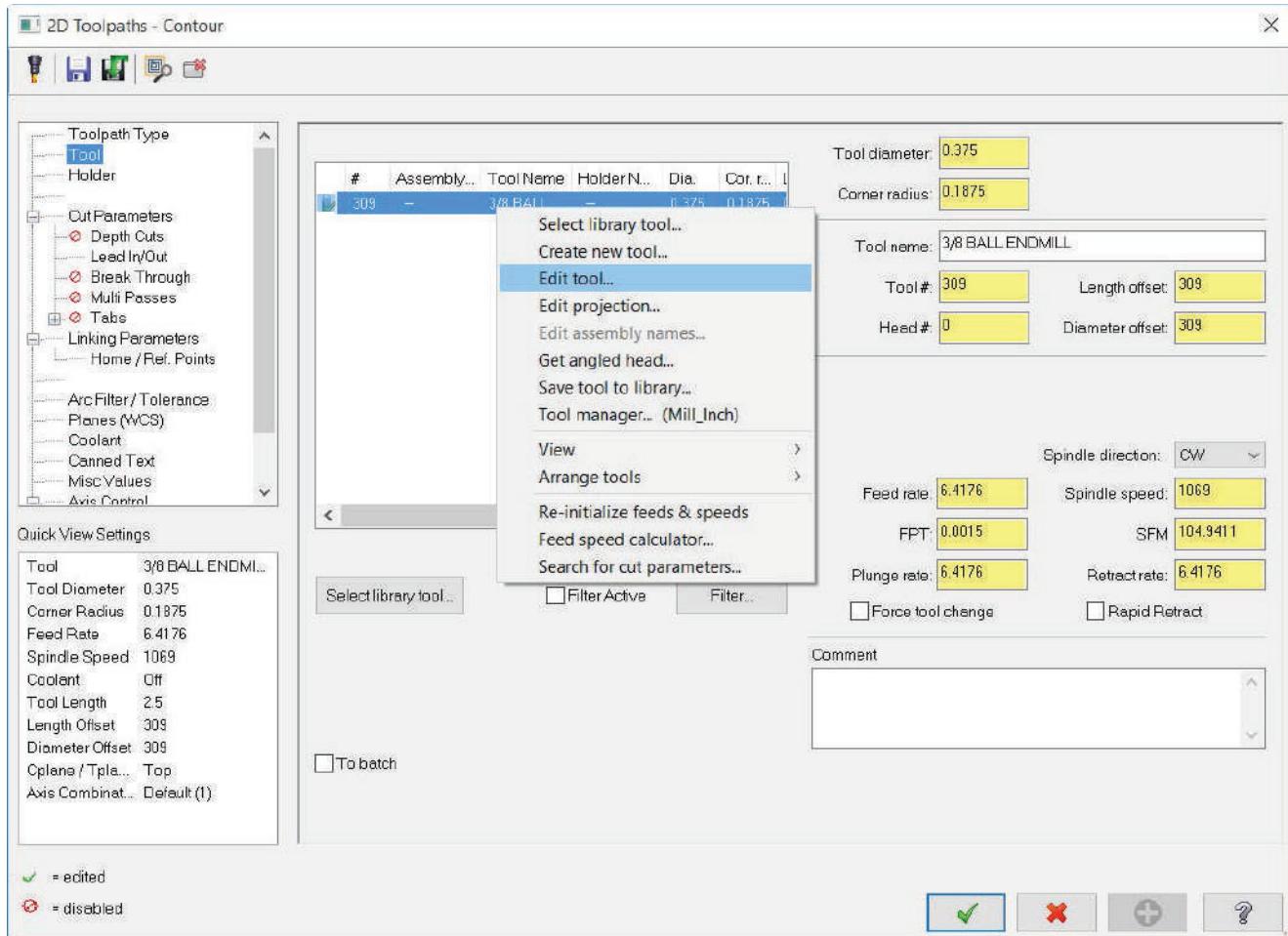
- ♦ Right click on the existing tool.
- ♦ Select **Edit tool** as shown.



- ♦ Once the **Edit Tool** dialog box opens, you can change any of the dimensions or properties of the tool that you want.

Option 2: Editing an existing tool inside of the toolpath parameters

- In the **Tool** dialog box, right click on the existing tool and then select **Edit tool** as shown.



- Make the changes to the tool.
- Once you select the **OK** button, the tool will be automatically saved with the new changes.

QUIZ ANSWERS

Objectives:

- ✓ The answers to the 7 Tutorial quizzes.

CAD IMPORT & MILL ESSENTIALS TOOLPATHS TUTORIAL QUIZ ANSWERS

Tutorial # 1 Answers

- ♦ What is a Contour toolpath used for?
- ♦ **Contour** toolpaths remove the material along a path defined by a chain of curves. Contour toolpaths only follow a chain; they do not clean out an enclosed area.

- ♦ What is a Facing toolpath used for?
- ♦ **Facing** toolpath quickly removes material from the top of the part to create an even surface for future operations.

- ♦ What does Backplot do?
- ♦ **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.

- ♦ What does Verify allow you to do?
- ♦ **Verify** allows you to use a solid model to simulate the machining of a part. The model created by verification represents the surface finish, and shows collisions, if any exist.

Tutorial # 2 Answers

- ♦ What does Slot Mill toolpath do?
- ♦ **Slot Mill** toolpath allows Mastercam to efficiently machine obround slots. These are slots that consist of 2 straight lines and two 180-degree arcs at the ends.

- ♦ What does 2D HS Dynamic Mill do?
- ♦ **2D HS Dynamic Mill** utilizes the entire flute length of the cutting tools to machine open pocket shapes, standing core shapes or pockets. To machine standing cores the toolpath uses the outmost chain as the stock boundary. The tool moves freely outside of this area; the inner chain defines the limit of the toolpath.

- ♦ What does 2D HS Dynamic Contour Mill do?
- ♦ **2D HS Dynamic Contour** toolpath utilizes the entire flute length of the cutting tools and is used to mill material off walls. It does support both closed or open chains.

- ♦ What is the process used to be able to post different operations as different programs?
- ♦ Create a new toolpath group and then rename it.

Tutorial # 3 Answers

- ◆ What does Area Mill require and what do those requirements do?
- ◆ **Area Mill** takes small cuts to machine open pocket shapes, standing core shapes or pockets based on the machining region strategies.

- ◆ What does Island Facing option do?
- ◆ **Island Facing** allows you to face the top of all the islands selected.

- ◆ What does Pocket Remachining do?
- ◆ Pocket Remachining calculates areas where the pocket roughing tool could not machine the stock and creates a remachining pocket toolpath to clear the remaining material.

Tutorial # 4 Answers

- ◆ What does a Circle Mill toolpath do?
- ◆ **Circle Mill** toolpath is used to mill circular pockets based on a single point. Mastercam will pocket out a circular area of the diameter and to the depth that you specify.

- ◆ What does a Dwell before cut spindle speed do?
- ◆ It adds a dwell after the entry ramp into the cut. This pause allows the spindle to ramp up to the desired spindle speed before starting the cutting passes.

- ◆ What does a Transform toolpath operation do?
- ◆ It allows you to run the same toolpath in different locations. You can transform a single toolpath or several at a time.

Tutorial # 5 Answers

- ◆ What settings do you need to use to remachine a pocket using High Speed Area Mill Toolpath?
- ◆ **2D High Speed Area Mill** toolpath with the **Rest Material** enabled targets material left behind by previous toolpaths.

- ◆ What is the use of **WCS** in Mastercam?
- ◆ This tells Mastercam how your part is position or orientated in the machine.

- ◆ After creating a new toolpath group why do you rename the NC file?
- ◆ To create two separate programs.

Tutorial # 6 Answers

- ◆ What does the Analyze Position command do?
- ◆ **Analyze Position** allows you to view the XYZ coordinates of a selected position or point entity.

- ◆ How does a Blend Mill toolpath work?
- ◆ **2D High Speed Blend Mill** toolpath morph smoothly between two open chains.

- ◆ What does Along parameter in the Blend Mill toolpath do?
- ◆ **Along parameter** sets the toolpath to cut in the along direction while stepping over in the across direction.

Tutorial # 7 Answers

- ◆ What does FBM Drill do?
- ◆ **FBM Drill** automatically detects holes in a solid based on your specific criteria and generates a complete series of drilling and chamfering. FBM drill also generates circle mill or helix bore operations for large-hole features when you activate these settings.

- ◆ What does Translate 3D command do?
- ◆ **Translate 3D** allows you to move the geometry between views (from one plane to another).

- ◆ What does Avoidance regions selection do?
- ◆ **Avoidance regions** allows you to select the profile that describes the shape up to where the material will be removed.

Numerics

- 2D / 3D Construction 7
 2D Chamfer 87, 95
 2D High Speed Area Mill 397, 408
 2D High Speed Blend Mill 540
 2D High Speed Core Mill 224, 555
 2D High Speed Dynamic Area Mill 317, 326
 2D High Speed Dynamic Core Mill 528

A

- Attributes 7

B

- Back Feedrate 244
 Break Through 139

C

- Chamfer Toolpath 82, 90
 Circle Mill 349
 Clearance 47, 185
 Color 7
 Comment 44, 183
 Constant Overlap Spiral 261
 Contour Remachining 154
 Corner Rounding 231
 Counterbore 480
 Creating A New Tool 634
 Creating And Renaming Toolpath Groups 174
 Creating/Editing Tools 633

D

- Depth 47, 140, 185
 Depth Cuts 54
 Drill Change At Point 461
 Drill/Counterbore 73
 Dwell 73

E

- Editing An Existing Tool 640
 Entry Feed Rate 233
 Entry Method 233

F

- Facing Toolpath 40, 180
 Feed Plane 47, 140, 185
 Function Prompt 4

G

- Graphic Area 4
 Grid 11
 Groups 7
 GUI - Graphical User Interface 4

- Gview 7

H

- High Speed Loops 46, 184

I

- Insert Arrow 410

K

- Keep Tool Down Within 231

L

- Level 7

- Line Style 7

- Line Width 7

M

- Menu 4
 Micro Lift Distance 244
 Move Between Cuts 46, 184
 MRU Toolbar 4

O

- Offset Tolerance 231
 Optimize Cut Order 244
 Origin 4
 Output 3d Arc Moves 233

P

- Planes 7
 Pocket Remachining 264
 Point Style 7
 Profile Tolerance 231

Q

- Quick Mask Toolbar 4
 Quiz Answers 643

R

- Renaming The NC File 187
 Reorganizing Toolbars 6
 Retract 47, 140, 185
 Ribbon Bar 4

S

- Scale 4
 Slot Mill 491
 Spot Drilling 69, 337, 426
 Start Angle 167
 Status Bar 4
 Stepover Percentage/Stepover Distance 137

T

- Tip Offset 87
 Tool Inspection/Change 230

INDEX

Toolbars	4
Toolpath Groups	450
Toolpaths/Solid Manager	4
Top Of Stock	47, 140, 185
U	
Using the Menu	5
Using the Toolbars	5
V	
View Port XYZ Axes	4

W

Work Coordinate System (WCS)	7, 366
------------------------------------	--------

X

XY Stepover	231
-------------------	-----

Z

Z Clearance	233
-------------------	-----

Z Depth	7
---------------	---

Did you like this book?

Discover other Mastercam Training Solutions at

www.eMastercam.com



eMastercam has the largest Mastercam community on the web with over 56,000 members and over a million forum posts. While there, be sure to join the conversation!

Training Solutions Available:

Training Tutorials

Handbooks

Instructor Kits

Professional Courseware

Multimedia

Online Video eCourses

CNC STEM Activities

Site Licenses

Sitewide Subscriptions!

Mastercam® eCourses



LOG IN & LEARN TODAY!

Expand your Mastercam knowledge, anytime and at your own pace, with our selection of step-by-step online video eCourses.



Highlights Include:

- Online previews with table of contents including the time it takes to complete each session.
- Mastercam 2018 Home Learning Edition Demo Software download is included so you can follow along with our instructors.
- Corresponding eBook is provided.
- Quizzes follow each tutorial.
- Personalized certificate of completion for each successfully completed eCourse.
- Lifetime access

www.eMastercam.com/ecourses

Only \$149 each!

NEW! eMastercam.com

Sitewide Subscription



ALL eBook* and eCourses
only \$**39.99**/Month!

LEARN MORE: www.eMastercam.com



OCTOPUZ®

Robot Software

Program and simulate all your robots
in the same cell, for all your applications,
while your robots are still working.

Book a demo and see for yourself!

www.OCTOPUZ.com



Does your school have robots? You'll want to check out OCTOPUZ.



As easy to teach as it is to learn, OCTOPUZ is an offline robot programming and simulation software used world-wide in industry. This powerful solution prepares your students for the real world of robotics and provides them with the necessary skills to not only be hired, but to excel.

With
OCTOPUZ
you can:

1

Create your custom cell.

2

Program your application.

3

Verify using simulation.

4

Generate and post code to ALL your robot brands.



OCTOPUZ®
Robot Programming & Simulation Software

www.OCTOPUZ.com/for-education

Post processors that are requested worldwide.



We speak your machine's language.

Implementing successful post processors for decades, the post department at In-House Solutions has earned a reputation for quality, resulting in one of the largest post departments in the world. We have an extensive library that is continually developed and expanded to include current machines and controllers. If a post is not already available, we will develop a custom one for nearly any machine. We have great relationships with OEMs who provide technical information for both machines and controls, which in turn allows us to produce post processors that will generate edit-free code and run your machine efficiently.

www.inhousesolutions.com/posts

