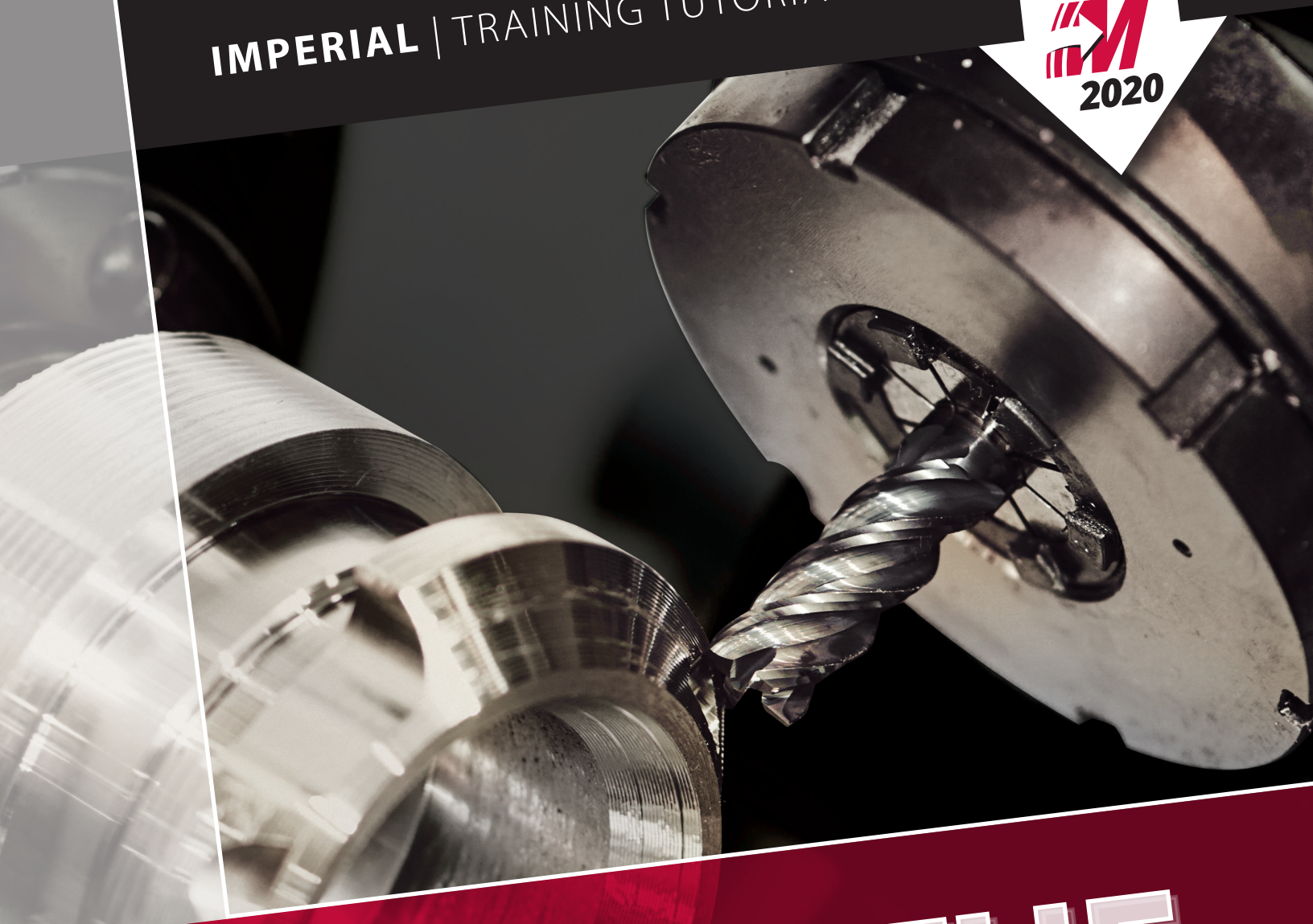
Mastercam 2020

IMPERIAL | TRAINING TUTORIAL SERIES

HLE DOWNLOAD
INSTRUCTIONS INSIDE
2020



LATHE C & Y AXIS

Mastercam 2020

LATHE C & Y AXIS TRAINING TUTORIAL

To order more books:

Call 1-800-529-5517 or

Visit www.emastercam.com or

Contact your Mastercam dealer

Mastercam 2020 Lathe C & Y Axis Training Tutorial

Copyright: 1998 - 2020 In-House Solutions Inc. All rights reserved

Software: Mastercam 2020

Authors: Mariana Lendel

ISBN: 978-1-77146-848-0

Date: July 17, 2019

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 are registered trademarks of Microsoft Corporation;

Windows 10 is a registered trademarks of Microsoft Corporation.

Lathe C & Y Axis Projects

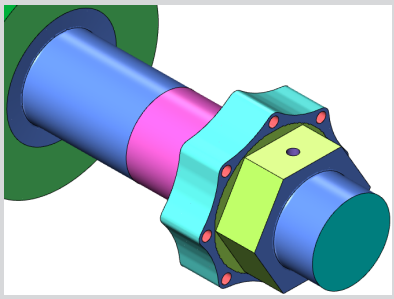
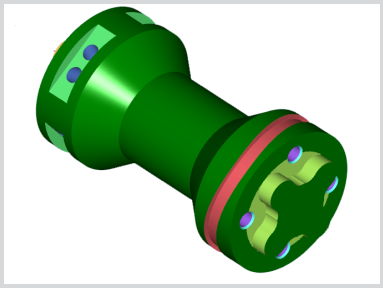
Tutorial	Geometry Functions	Toolpath Creation
#1 	Create Rectangle. Create Line Parallel. Create Point. Edit Trim. Change Cplane. Create Polygon. Create Arc Endpoints. Transform Rotate. Create Fillet Chains. Create Bolt Circle.	Face. Rough. Finish. Groove. C-Axis Face Contour. C-Axis Face Drill. C-Axis Drill. Cutoff.
#2 	Levels Manager. Planes. Create Line Endpoints. Transform Mirror. Manual Spline. Create Ellipse. Create Circle Center Point. Edit Trim. Transform Rotate. Create Fillet. Create Point Position. Create Spline.	C-Axis Face Contour. C-Axis Cross Contour with Y-Axis. C-Axis Cross Drill. C-Axis Face Drill. Stock Transfer. C-Axis Face Contour. C-Axis Face Drill. C-Axis Contour.

Table Of Content

Getting Started **9**

Step 1: Starting Mastercam 10

Step 2: GUI - Graphical User Interface 11

Step 3: Navigate Through Mastercam 12

Step 4: Set The Attributes 14

Step 5: Manager Panels 16

Step 6: Setting Mastercam To Imperial 18

Step 7: Set The Grid 20

Tutorial 1 ..... **21**

Overview Of Steps Taken To Create The Final Part: 22

Geometry Creation 24

Step 1: Setting Up The Graphical User Interface 24

Step 2: Create A Rectangle 24

Step 3: Create Parallel Lines 27

Step 4: Create A Point 30

Step 5: Trim To Complete The Geometry In The Top View 32

Step 6: Set Levels Manager 35

Step 7: Change The Construction Plane To Right 37

Step 8: Create A Polygon 38

Step 9: Create Bolt Circles 40

Step 10: Create A Fillet Knowing The Radius And Endpoints 43

Step 11: Rotate The Arc 46

Step 12: Trim The Circles 48

Step 13: Save The File 52

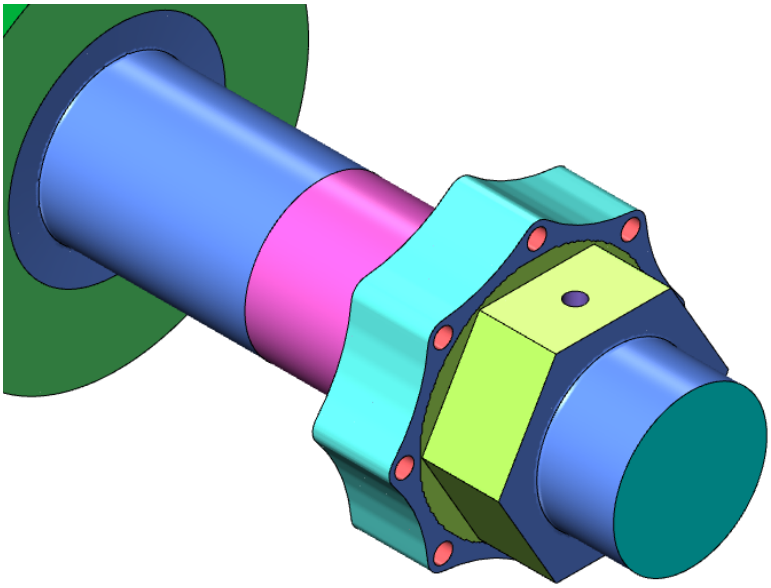
Toolpath Creation 53

Part Setup: 53

Setup Sheet: 54
Step 14: Select The Machine And Set Up The Stock 55
Step 15: Face The Part 60
Step 16: Backplot The Toolpath 64
Step 17: Simulate The Toolpath In Verify 65
Step 18: Rough The OD 68
Step 19: Backplot The Toolpaths 74
Step 20: Simulate The Toolpath In Verify 75
Step 21: Finish The OD 76
Step 22: Machine The Leftover Material Using A Groove Toolpath 82
Step 23: Machine The Hexagonal Face Using The C-Axis Face Contour 89
Step 24: Machine The Rounded Contour Using C-Axis Face Contour 100
Step 25: C-Axis Face Drill The Part 107
Step 26: Create Another C-Axis Face Drill Toolpath 111
Step 27: Spot Drill The Hole Using Cross Drill 116
Step 28: Cross Drill The Hole To Size 123
Step 29: Cutoff The Part 127
Step 30: Run The Post Processor To Obtain The G-Code File 134
Step 31: Save The Updated MCAM File 135
Review Exercise - Student Practice 136
Create The Geometry For Tutorial #1 Exercise 137
Create The Toolpaths For Tutorial #1 Exercise 138
Tutorial 2 ..... **143**
Overview Of Steps Taken To Create The Final Part: 144
Step 1: Setting Up The Graphical User Interface 146
Step 2: Create And Organize Levels Before Creating Geometry 147
Step 3: Create Lines To Represent The Outside Profile 151
Step 4: Change The Construction Plane, Main Level And Color 163

Step 5: Create The Ellipse On The Right Face 165
Step 6: Create A Circle 167
Step 7: Create A Rectangle 170
Step 8: Trim The Rectangle 173
Step 9: Break The Lines 175
Step 10: Create Arcs And Rotate Them 177
Step 11: Create The Circles On The Left Face 182
Step 12: Insert Fillets Between The R 0.250" Circles 187
Step 13: Trim The Rounded Contour 191
Step 14: Create Points 193
Step 15: Create The Spline 195
Step 16: Select The Machine And Set Up The Stock 200
Step 17: Machine The Ellipse On The Right Face 210
Step 18: Backplot The Toolpaths 218
Step 19: Simulate The Toolpath In Verify 219
Step 20: Machine The Slots Using Y Axis Cross Contour & Transform Toolpaths 221
Step 21: Transform Rotate Toolpath 229
Step 22: Spot Drill The Holes Using Cross Drill 233
Step 23: Drill The Holes Using Cross Drill 237
Step 24: Transfer The Stock To The Right Spindle 241
Step 25: Machine The Profile In The Right Spindle 245
Step 26: Face Drill The Holes In The Right Spindle 254
Step 27: Create Another Face Drill Toolpath 258
Step 28: Machine The Spline On The Left Face 262
Step 29: Run The Post Processor To Obtain The G-Code File 267
Step 30: Save The Updated MCAM File 269
Quiz Answers **277**

Tutorial 1



OVERVIEW OF STEPS TAKEN TO CREATE THE FINAL PART:

From Drawing to CAD Model:

- From the drawing we can gain an idea as to how to go about creating the geometry in Mastercam.
- The student will need to create the geometry used to machine the part from two planes: Top and Right.
- A point will also be created to indicate a center point for C-Axis drilling.

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

- The student will create the upper profile of the part in the Top view on its own level.
- The student will create geometry from the Right plane for the C-Axis Toolpaths.
- Transform Rotate and Create Polygon geometry creation commands will be used.

Create the necessary Toolpaths to machine the part:

- The student will Face, Rough, Finish, and Groove the part to create the outside profile.
- C-Axis Face Contour, Face Drilling, and C-Axis Drilling toolpaths will be used to machine the part from the Right plane.
- The part will then be cutoff using a Cutoff toolpath with a clearance cut enabled.

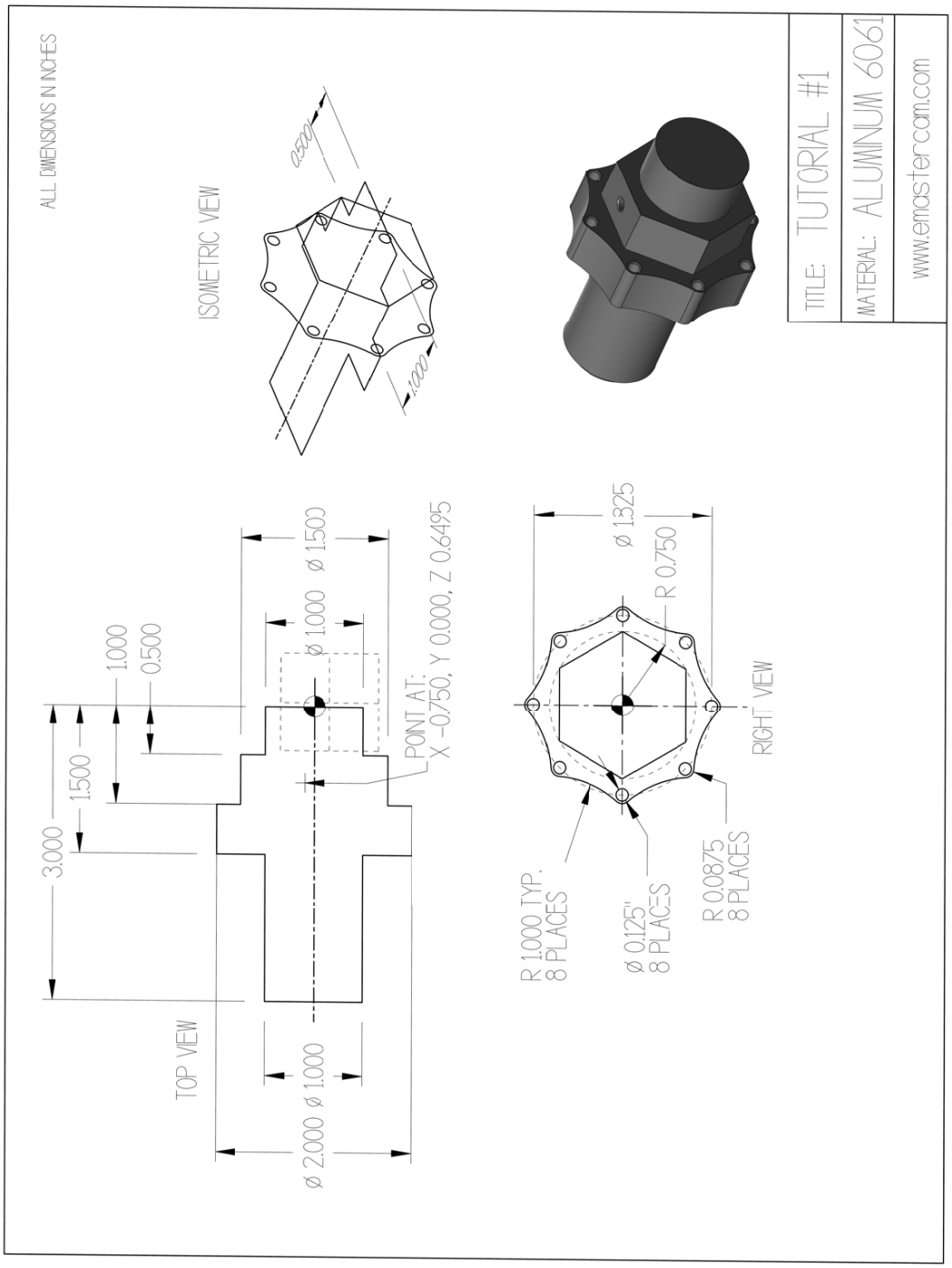
Backplot and Verify the file:

- 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 and thirty minutes to complete.


ALL DIMENSIONS IN INCHES
ISOMETRIC VIEW
0.500
1.000
TITLE: TUTORIAL #1
MATERIAL: ALUMINUM 6061
www.emastercam.com
ø 1.500
ø 1.000
POINT AT:
X -0.750, Y 0.000, Z 0.6495
1.000
0.500
1.500
3.000
TOP VIEW
ø 2.000 ø 1.000
ø 1.325
R 0.750
R 1.000 TYP.
8 PLACES
ø 0.125"
8 PLACES
R 0.0875
8 PLACES
RIGHT VIEW

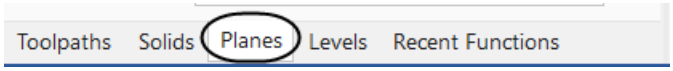
GEOMETRY CREATION

STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

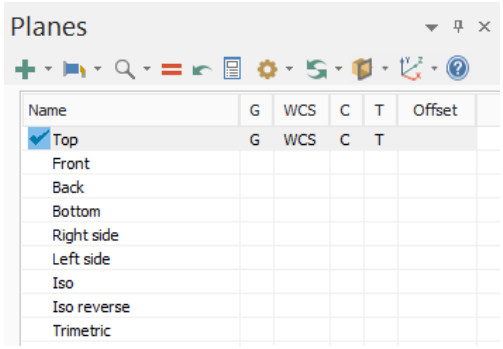
Before starting the geometry creation, you should customize the toolbars to see the toolbars required to create the geometry and machine a 3D part. See **Getting Started** for details.

***Note:** Because we will be creating **C-Axis** toolpaths, we will be using the **Top** and **Right** construction planes for this tutorial.*

- To open the **Planes Manager** panel, select the **Planes** tab as shown.



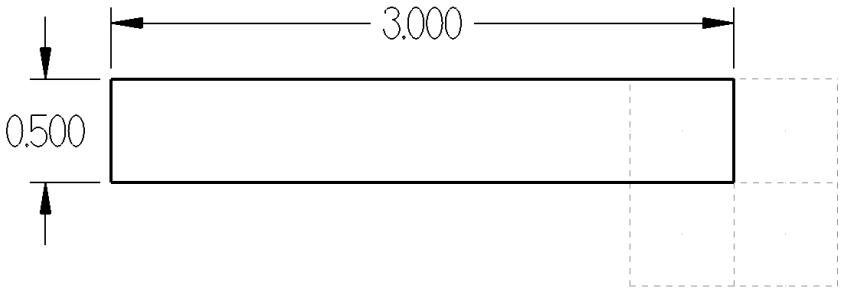
- Make sure that **WCS**, planes and the graphics view are all set to **TOP** as shown.



STEP 2: CREATE A RECTANGLE

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

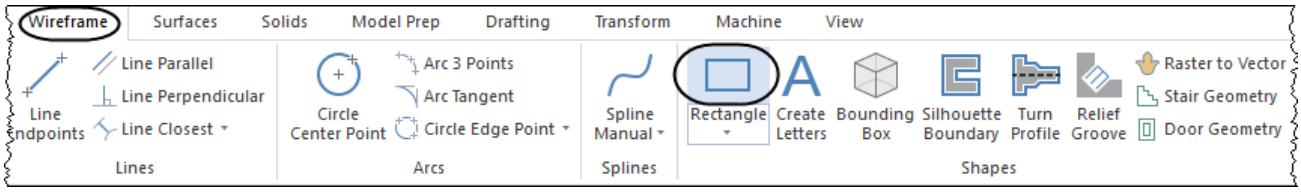
Step Preview:



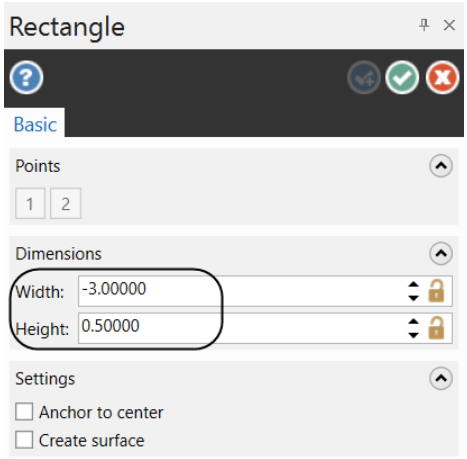
2.1 Create the -3.0" by 0.5" rectangle

WIREFRAME

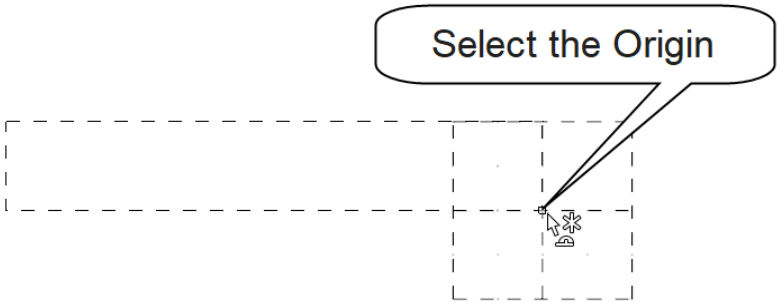
- From the **Shapes** group, select the **Rectangle** icon as shown.



- Enter the **Width** and the **Height** and make sure that **Anchor to center** and **Create surface** buttons are not selected (highlighted) as shown.

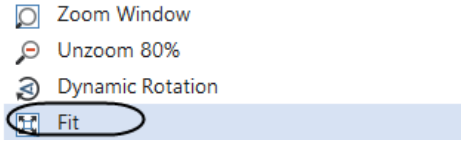


- Press **Enter** after typing the values to see a preview of the rectangle.
- [Select position for first corner]: Select the **Origin** as shown.



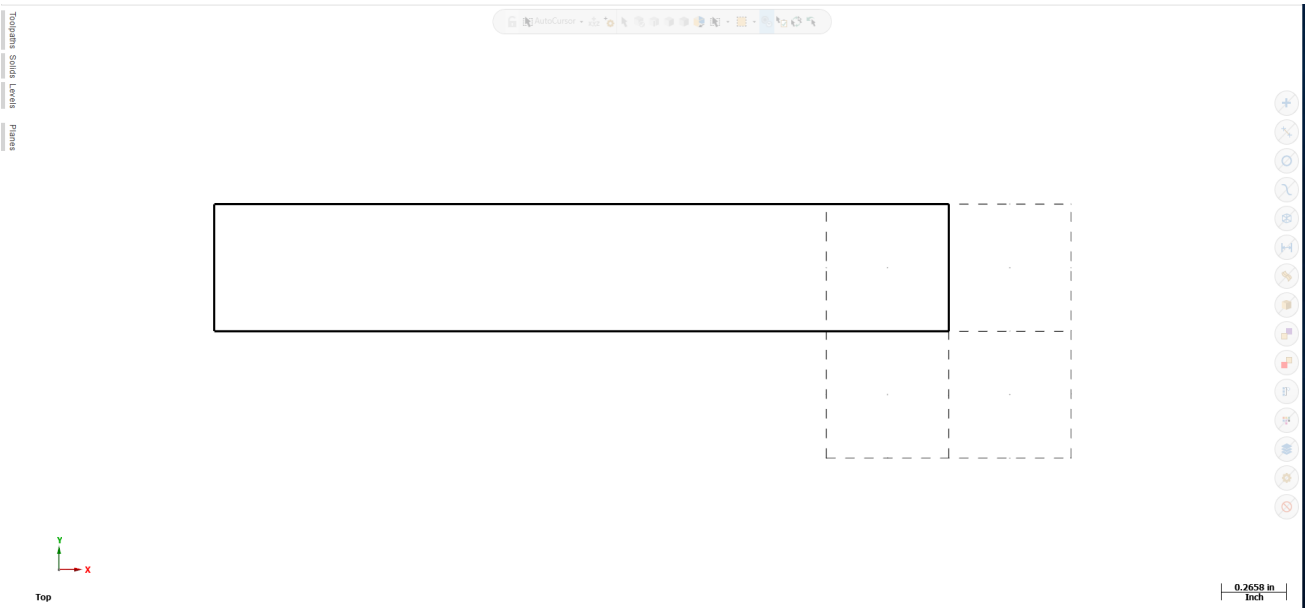
- Make sure that when selecting the origin, the visual cue of the cursor changes as shown.
- Select the OK button to exit the Rectangle panel.

- Right mouse click in the graphics window and select **Fit** to fit the drawing to the screen. You can also press **Alt + F1**.



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

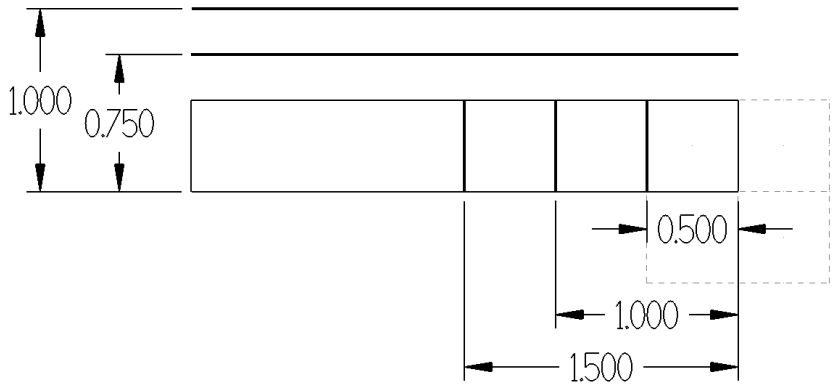
- Hover the mose at the center of the geometry and scroll the mouse down to unzoom the geometry.
- The geometry should look as shown.



STEP 3: CREATE PARALLEL LINES

In this step, you will create parallel lines to represent the grooves on the left hand side of the part.

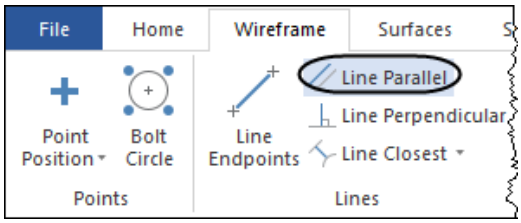
Step Preview:



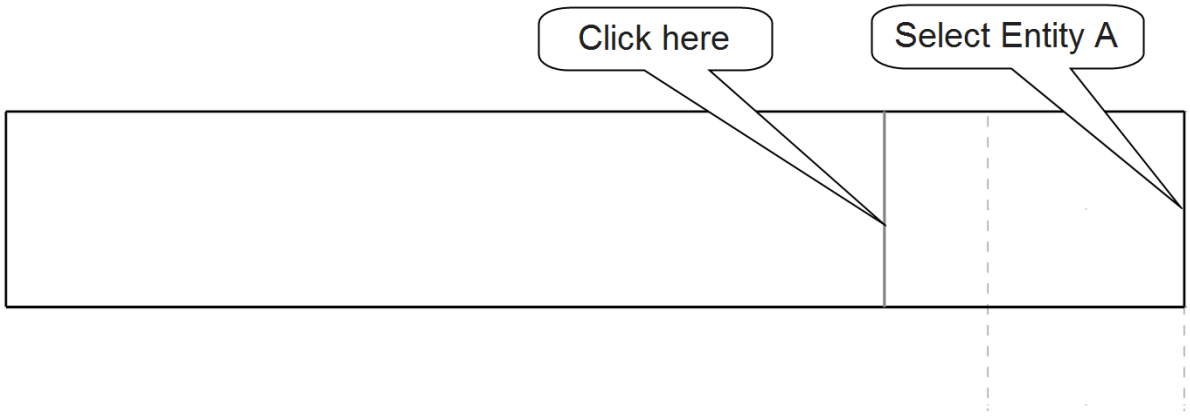
Create The Vertical Lines

Wireframe

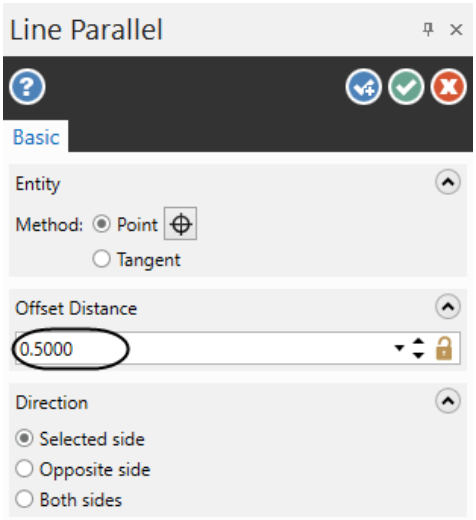
- From the **Lines** group, select the **Line Parallel** icon.



- [Select a line]: Select Entity A as shown.
- [Select the point to place a parallel line through]: Click on a point to the left of **Entity A** as shown.



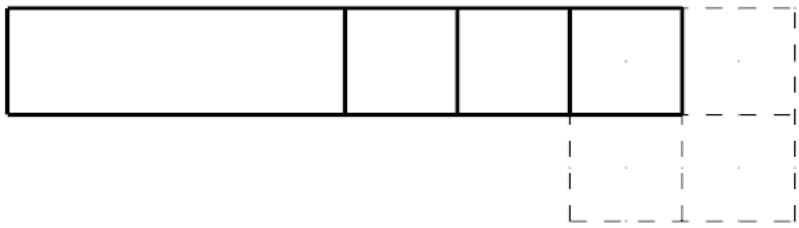
- In the **Line Parallel** panel, enter the **Offset Distance 0.5** and press **Enter**.



- Select the **OK and Create New Operation** button to stay within the command.
- [Select a line]: Select **Entity A** as shown before.
- [Select the point to place a parallel line through]: Click on a point to the left of **Entity A** as shown before.
- In the **Line Parallel** panel, enter the **Offset Distance 1.0**and press **Enter**.

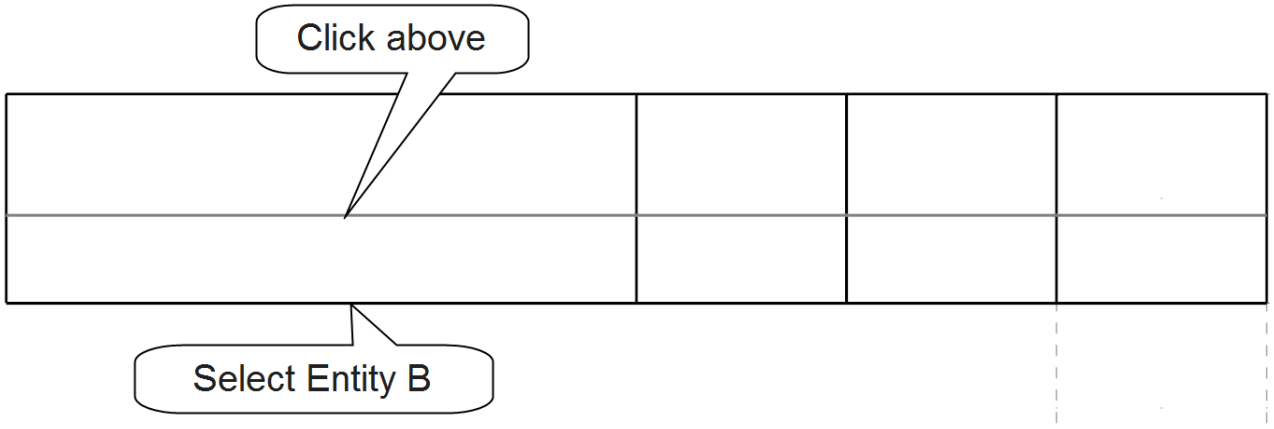
- Select the **OK and Create New Operation** button to stay within the command.
- [Select a line]: Select **Entity A**.
- [Select the point to place a parallel line through]: Click on a point to the left of **Entity A**.
- In the **Line Parallel** panel, enter the **Offset Distance 1.5** and press **Enter**.

- Select the **OK and Create New Operation** button to stay within the command.
- The part should appear as shown.

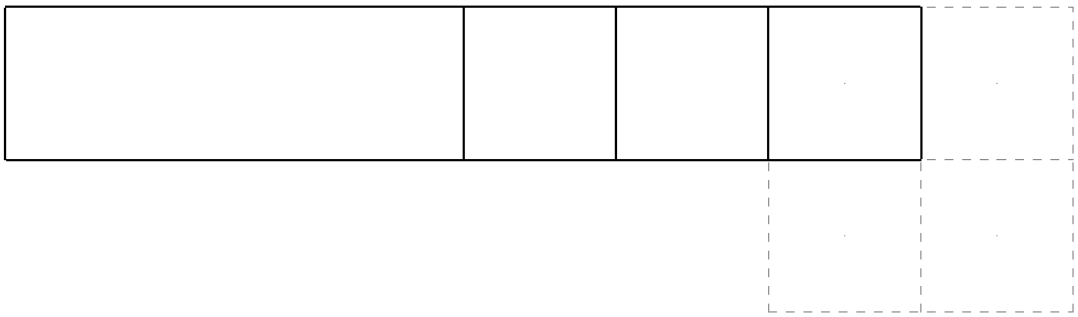


3.1 Create the horizontal lines

- [Select a line]: Select **Entity B** as shown.
- [Select the point to place a parallel line through]: Click on a point above **Entity B** as shown.



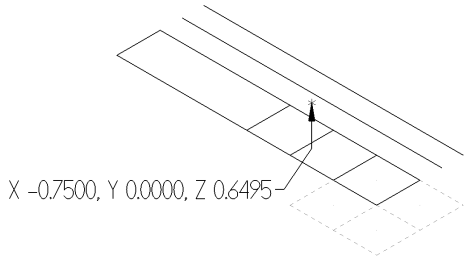
- In the **Line Parallel** panel, enter the **Offset Distance 0.75** and press **Enter**.
- Select the **OK and Create New Operation** button to stay within the command.
- [Select a line]: Select **Entity B** as shown.
- [Select the point to place a parallel line through]: Click on a point above **Entity B** as shown.
- In the **Line Parallel** panel, enter the Offset Distance **1.0** and press **Enter**.
- Select the **OK** button to exit the command.
- The part should appear as shown.



STEP 4: CREATE A POINT

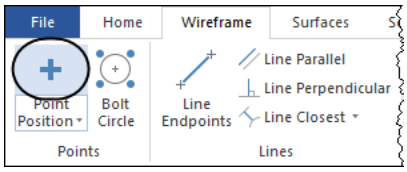
In this step, we will use the **Create Point Position** command to create a point. This point will be used by Mastercam as a drill position locator while creating the **C-Axis Drill** toolpath.

Step Preview:



Wireframe

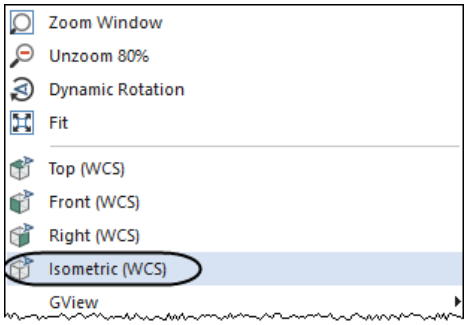
- From the **Points** group, select the **Point Position** icon as shown.



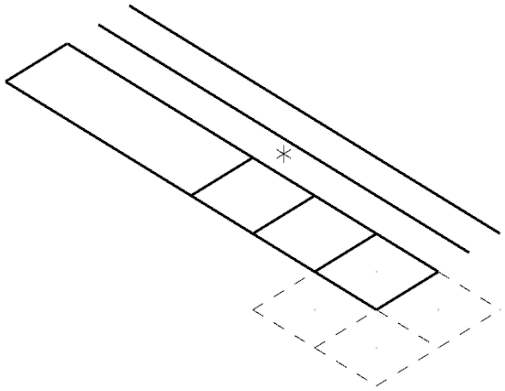
- [Create point position]: Select the **AutoCursor Fast Point** icon in the **General Selection** toolbar as shown.

- In the coordinate field, enter the coordinates **-0.75, 0.0, 0.6495** as shown. Press the **Enter** key once you have finished.

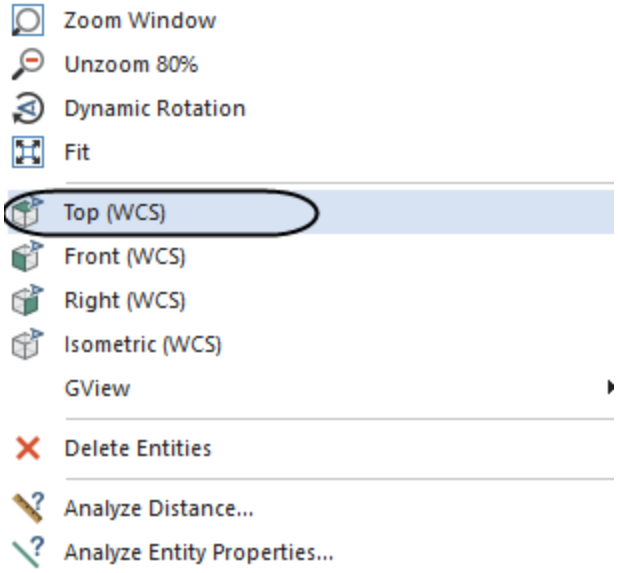
- Select the **OK** button to exit **Point Position** panel.
- Right mouse click in the graphics window and select the **Isometric** view as shown.



♦ The point should appear as shown.



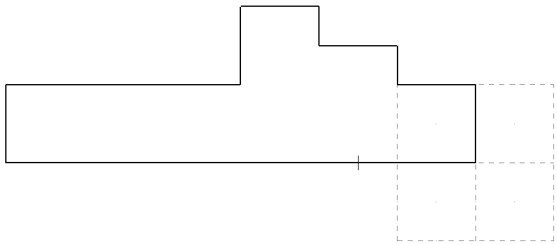
♦ Right mouse click in the graphics window and select the **Top** view as shown.



STEP 5: TRIM TO COMPLETE THE GEOMETRY IN THE TOP VIEW

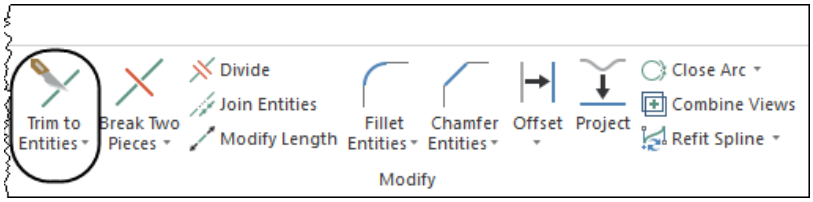
In this step, we will use the **Trim Break Extend** command to trim the lines to complete the geometry in the top view.

Step Preview:

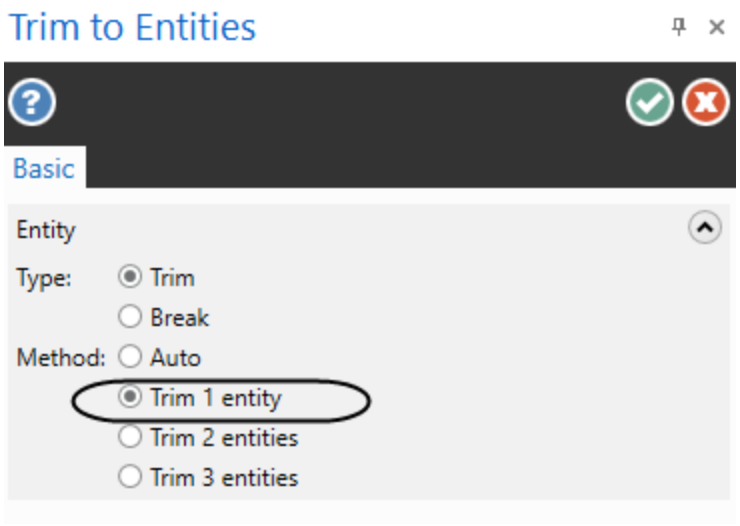


Wireframe

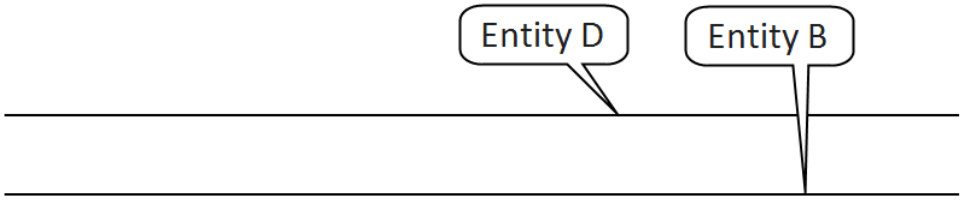
- From the **Modify** group, select the **Trim to Entities** icon as shown.

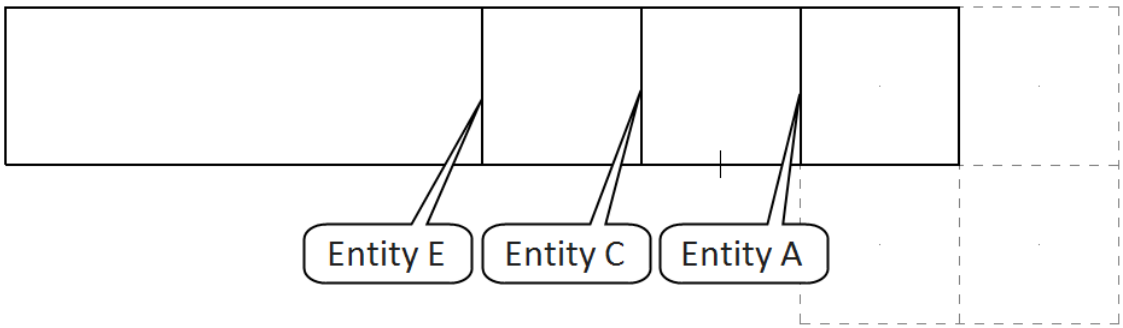


- In the **Trim to Entities** panel, enable **Trim 1 entity** as shown.

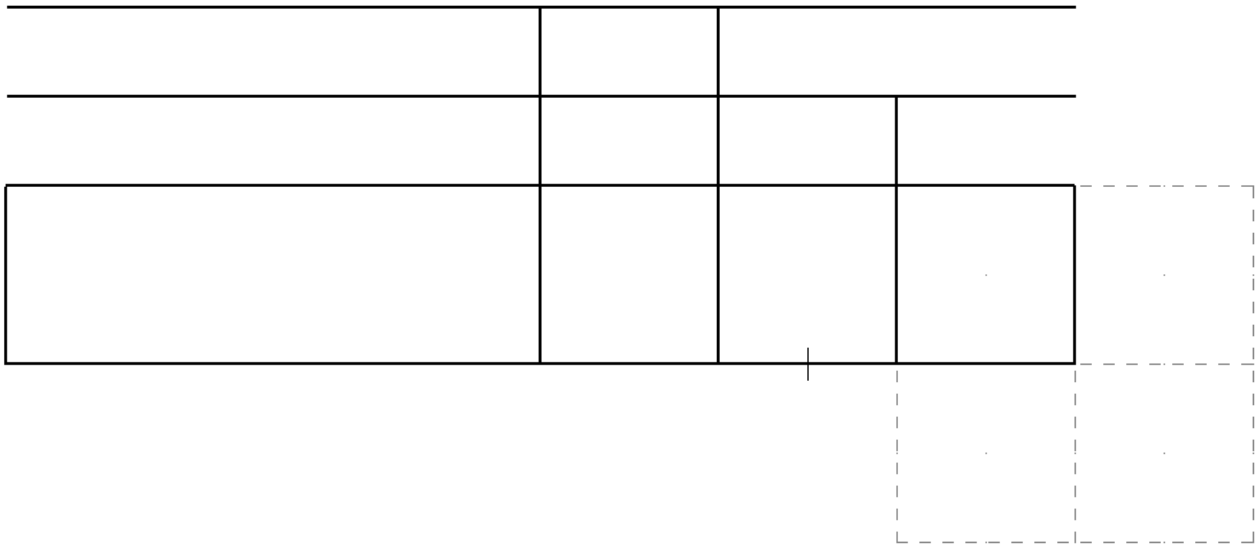


- [Select the entity to trim/extend]: Select **Entity A** as shown.
- [Select the entity to trim/extend to]: Select **Entity B** as shown.

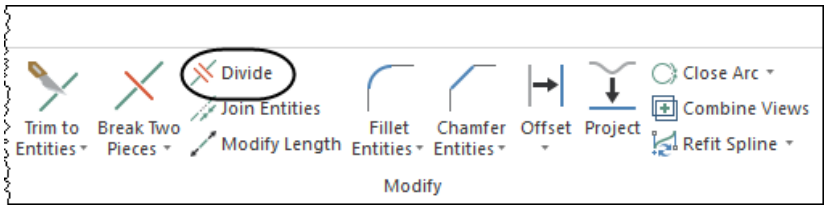




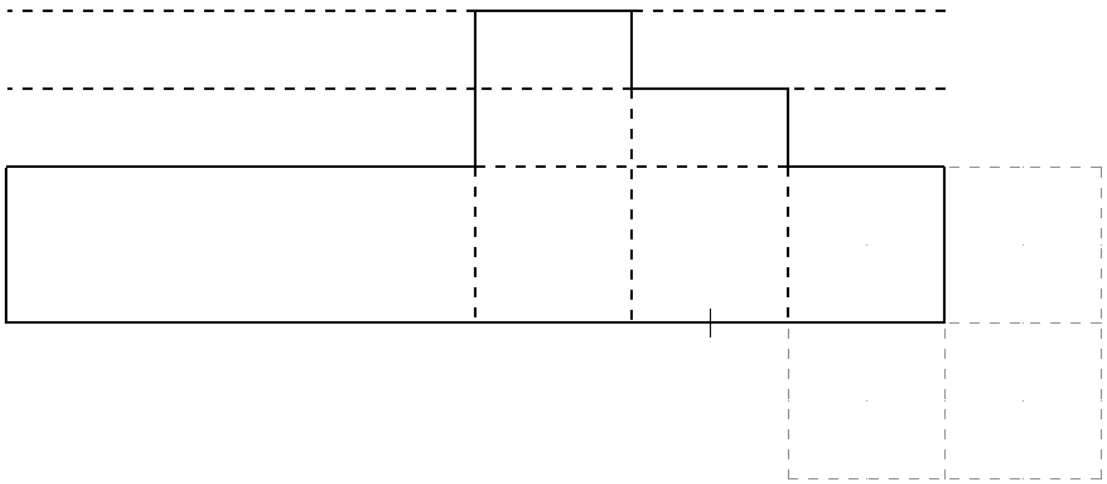
- [Select the entity to trim/extend]: Select **Entity C** as shown.
- [Select the entity to trim/extend to]: Select **Entity D** as shown.
- [Select the entity to trim/extend]: Select **Entity E** as shown.
- [Select the entity to trim/extend to]: Select **Entity D** as shown.
- Select the **OK** button to exit the command.
- The part should appear as shown.



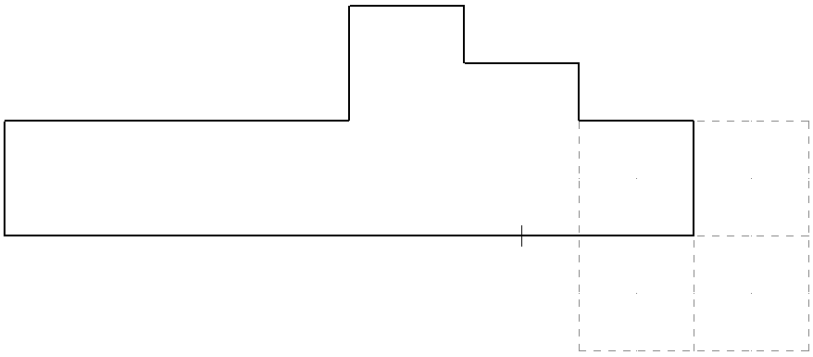
- From the **Modify** group, select the **Divide** icon as shown.



- In the **Divide** panel, make sure that **Trim** is enabled.
- [Select the curve to divide/delete]: Select the portions of line that appear as hidden (dotted) lines below to delete them with the divide function.



- Select the **OK** button to exit the command.
- The part should appear as shown.

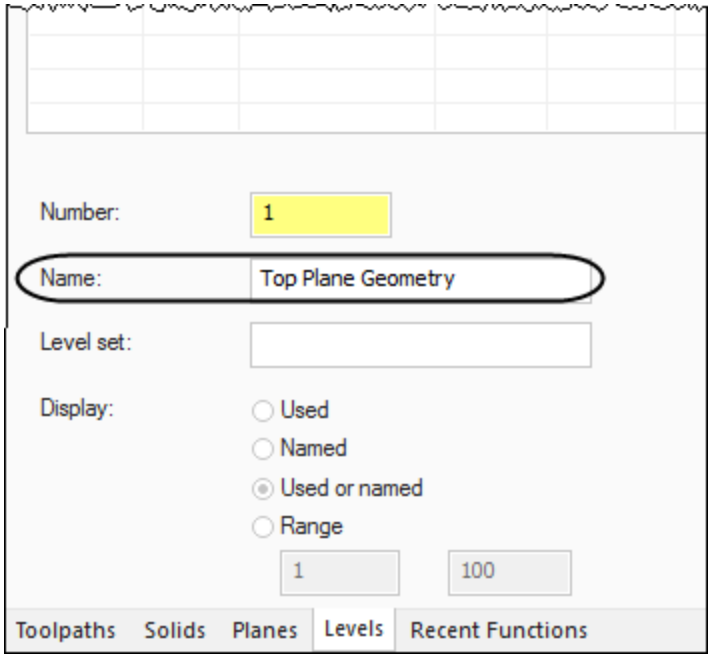


STEP 6: SET LEVELS MANAGER

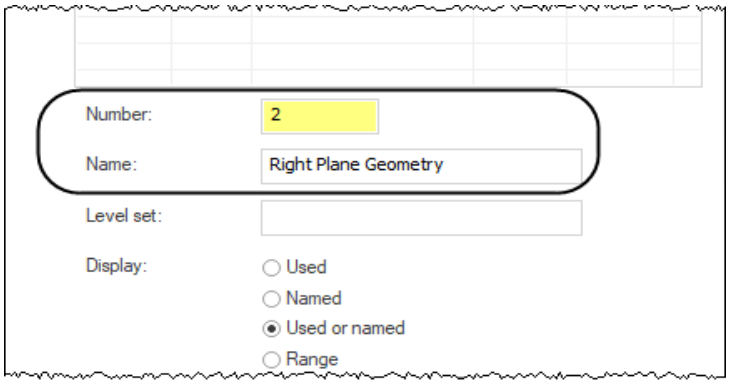
In this step, we will set the levels before we create the rest of the part to organize our geometry.

- Select the **Levels** tab to open **Levels Manager** as shown.

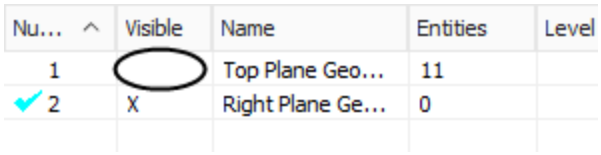
- Click in the **Name** area in the **Levels Manager** and enter in the name "**Top Plane Geometry**" as shown. Press **Enter** once complete.



- Click in the **Number** area and enter **2** as the level number. Then click in the **Name** area and type in the name "**Right Plane Geometry**" as shown. Press **Enter** once complete.

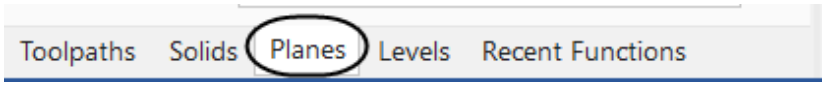


- Select the **X** in the **Visible** column of **Level 1** to make it invisible.

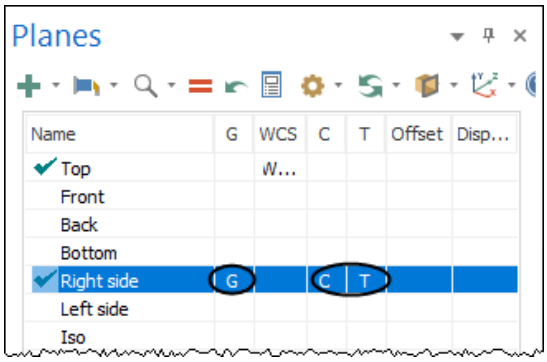


STEP 7: CHANGE THE CONSTRUCTION PLANE TO RIGHT

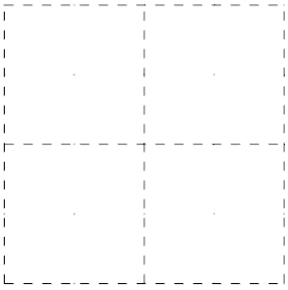
- Select the **Planes** tab to open the **Planes Manager** as shown.



- Set the **G**(Graphic view), **C**(Construction plane) and **T**(Tool plane) columns to **Right Side**. Leave the **WCS** set to **Top** plane as shown.



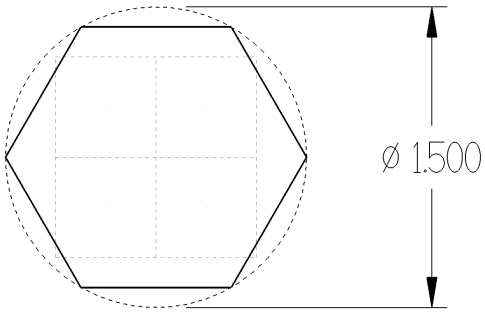
- Position the **Grid** (Hold down the mouse wheel and the **Shift** key to pan) so that you can see it on screen as shown.



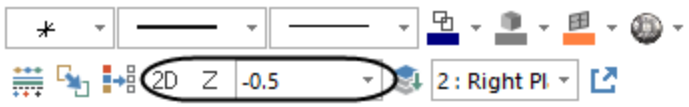
STEP 8: CREATE A POLYGON

In this step, we will create the six sided polygon, using the 2D construction mode.

Step Preview:

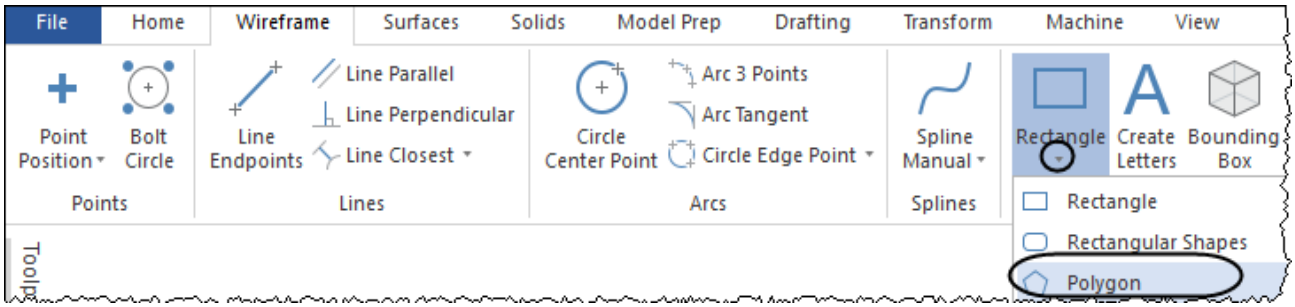


- Right mouse click in the graphics window and change the **construction mode** to **2D** and the **Z Depth** to **-0.5** as shown.



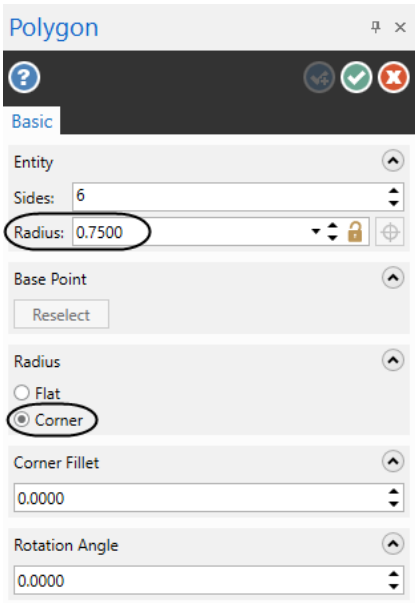
Wireframe

- From the **Shapes** group, click on the drop down arrow below the **Rectangle** and select the **Polygon** as shown.

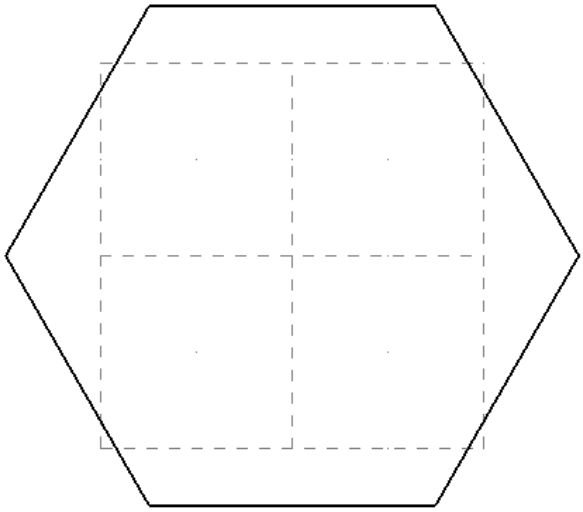


- [Select position of base point]: Select the **Origin** as shown.
- Select and click on the screen a point in the graphics window.

- In the **Polygon** panel enter in the **Radius** and enable **Corner** as shown.



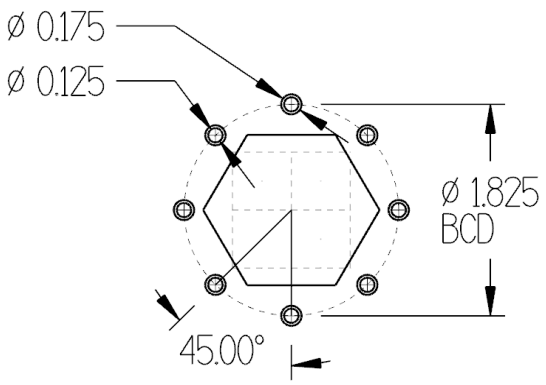
- Select the **OK** button to exit the **Polygon** command.
- Press **Alt + F1** to fit the geometry to the graphics window if needed.
- The part should appear as shown.



STEP 9: CREATE BOLT CIRCLES

In this step we will create bolt circles that will be used as construction lines to create the final contour needed to finish the part and as drilling positions.

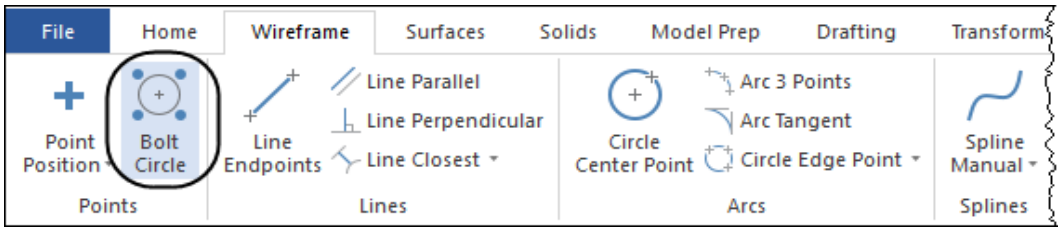
Step Preview:



- Right mouse click in the graphics window and change the construction mode to 2D and the **Z Depth** as shown.

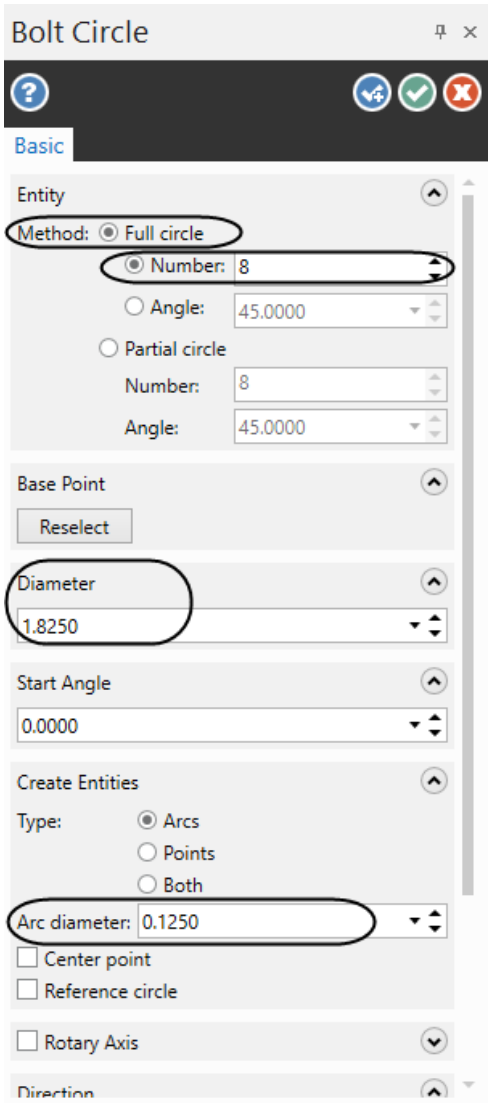
Wireframe

- From the **Points** group, select **Bolt Circle** as shown.



- [Select position of base point] Select the **Origin**.

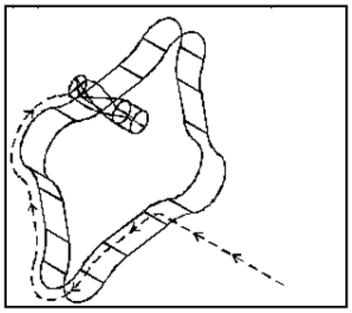
- The **Bolt Circle** panel will appear. Enable **Full Circle** and enter in values for **Diameter** of Guide Circle and **Number**. Enable **Arcs** and input an **Arc diameter** of **0.125"** as shown.



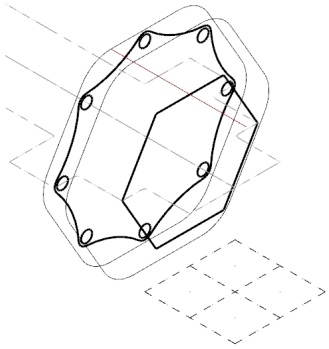
- Select the **OK and Create New Operation** button to continue using the **Bolt Circle** command.
- Press **Alt + F1** to fit the geometry onto the graphics window.

STEP 23: MACHINE THE HEXAGONAL FACE USING THE C-AXIS FACE CONTOUR

The **C-Axis Face Mill Type** allows you to machine the face of the part with the tool parallel to the main spindle's axis of rotation. In this case, the main spindle holding the part rotates (C-Axis) while the tool moves along the X axis. The Z axis is positioned based on the depth of the cut and the Y axis is locked in position at 0. Mastercam automatically sets the tool plane (TPlane) and construction plane (CPlane) so that the tool is placed perpendicular to the face of the part as shown.

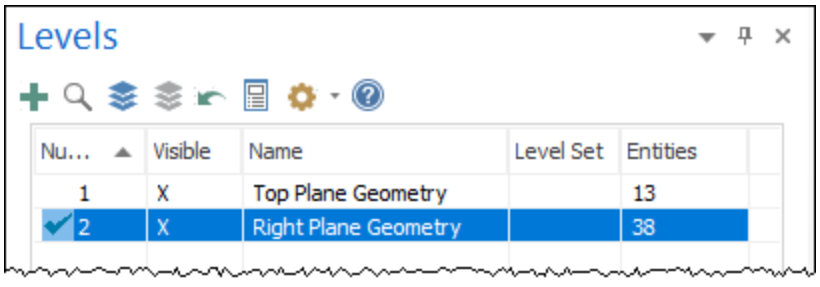


Toolpath Preview:

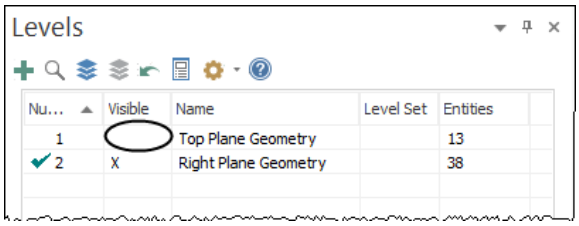


♦ From the managers panel, select the **Levels** tab as shown.

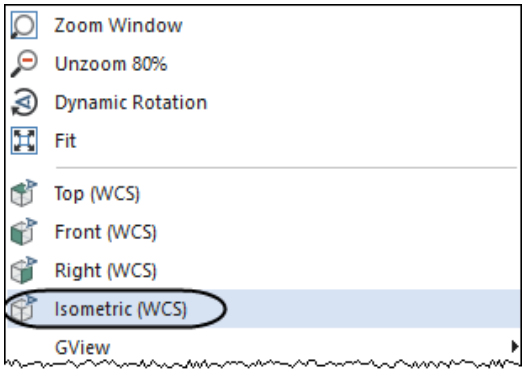
♦ In the **Levels Manager**, click on **2** in the **Number** column to make it the main level as shown.



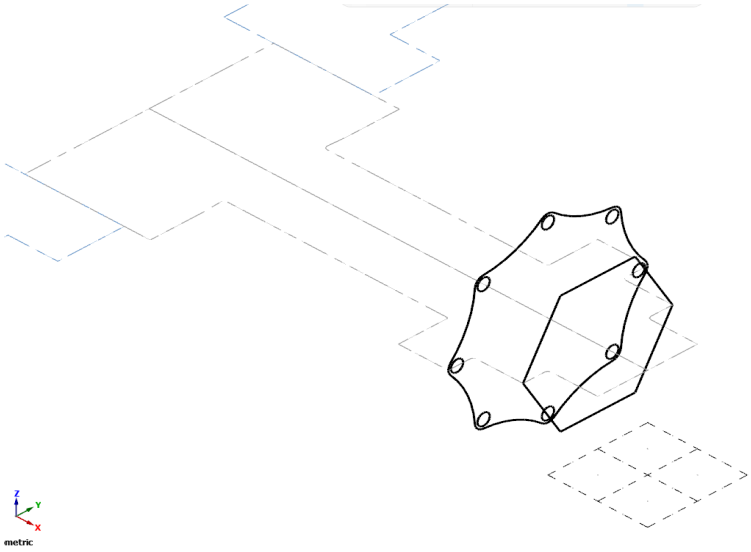
♦ Click in the **Visible** column next to **Level 1** to make it invisible and leave **Level 2** the only one visible.



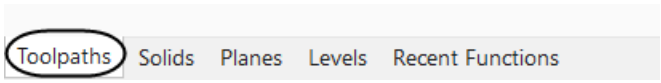
♦ Right mouse click in graphics window and select the **Isometric** view as shown.



♦ Press **Alt + F1** to fit the geometry and the stock to the graphics window.
♦ The part should appear as shown.

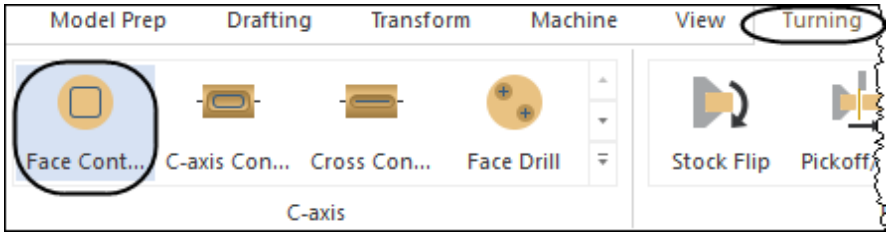


♦ To open the **Toolpaths Manager** panel, select the **Toolpaths** tab.

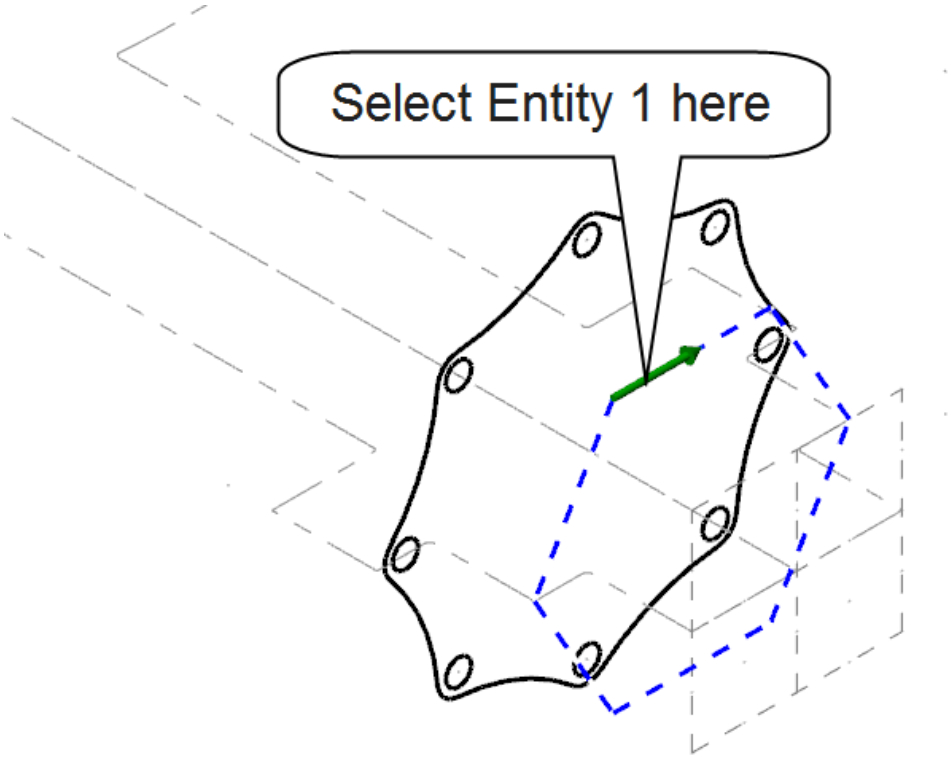


Turning

- From the **C-axis** group, select the **Face Contour** icon as shown.

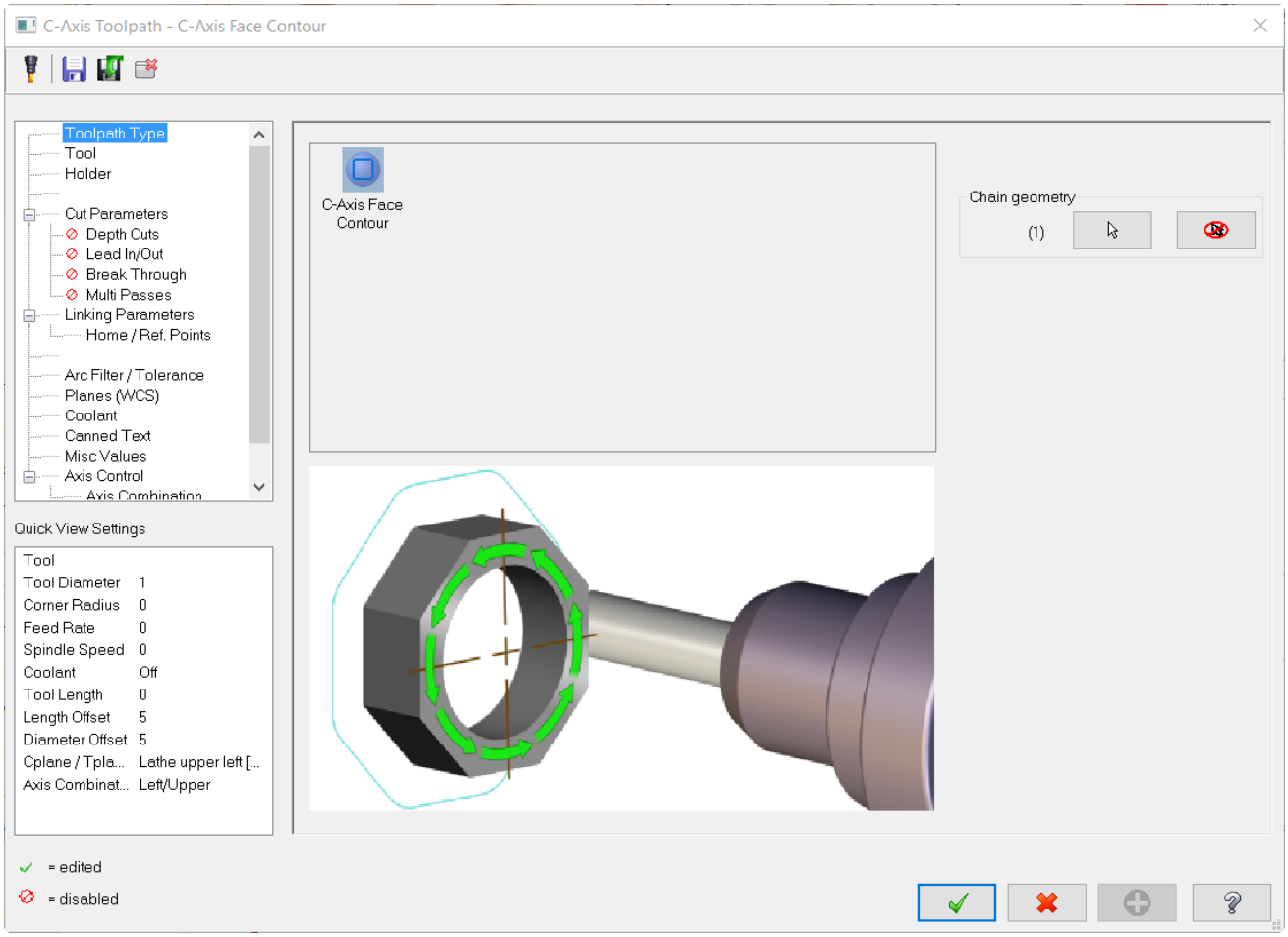


- The **Chaining** dialog box will open. Leave the default settings.
- [Select contour chain 1]: Select **Entity A** as shown so that the chaining direction is **Clockwise**.

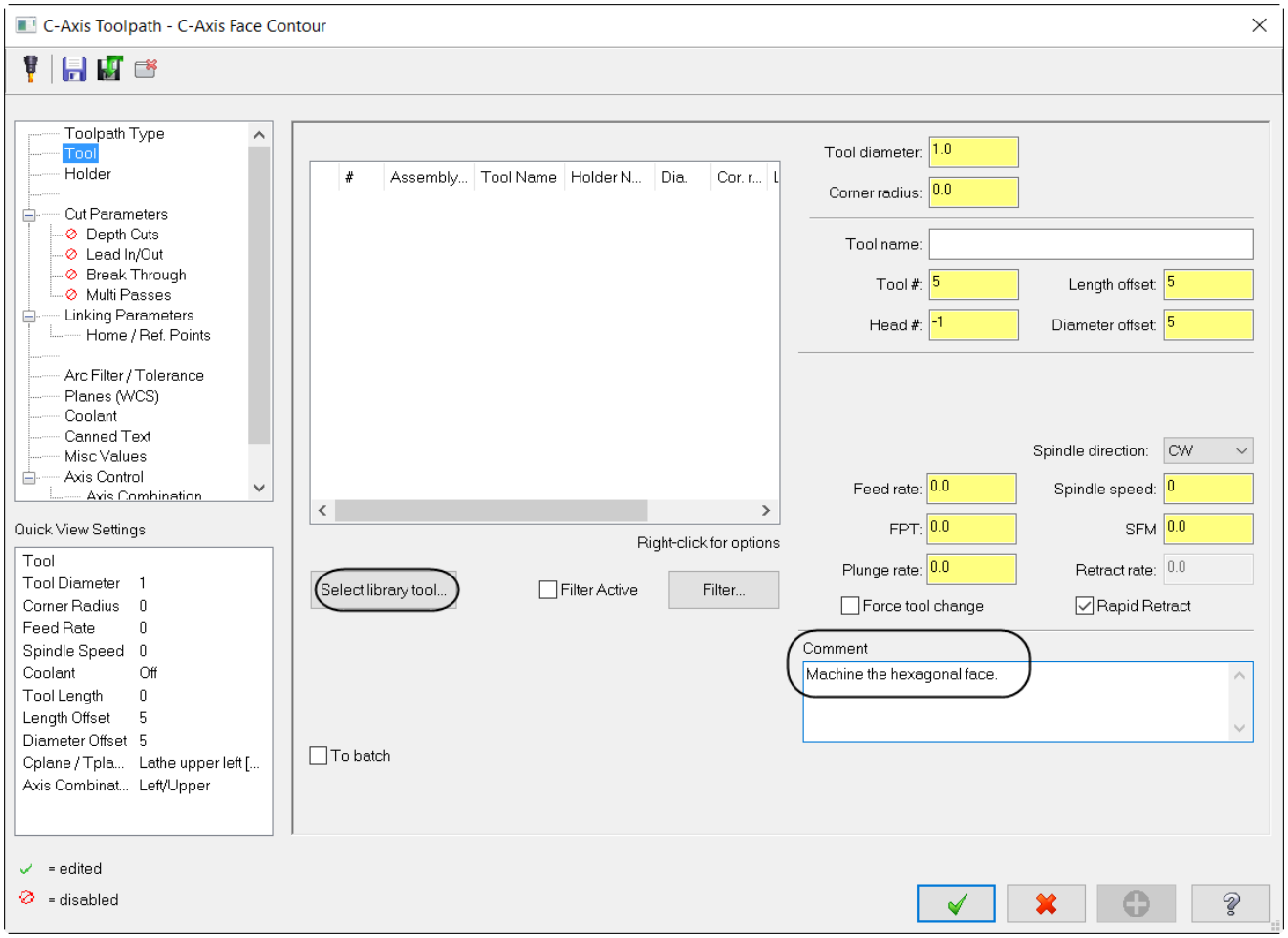


- Select the **OK** button to exit the **Chaining** dialog box.

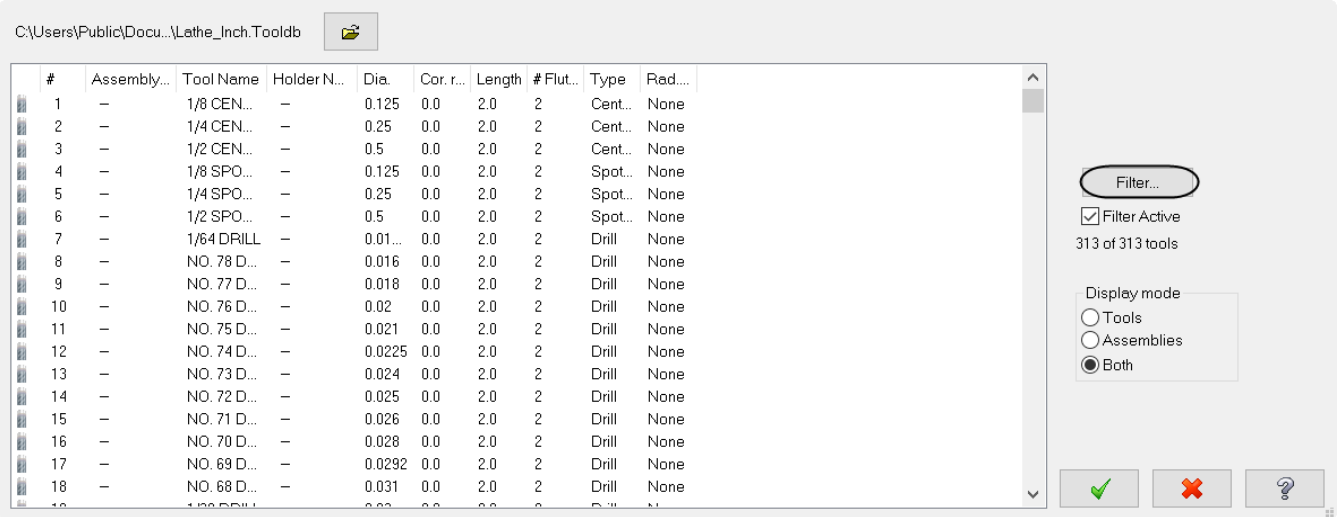
♦ The **Toolpath Type** page should look as shown.



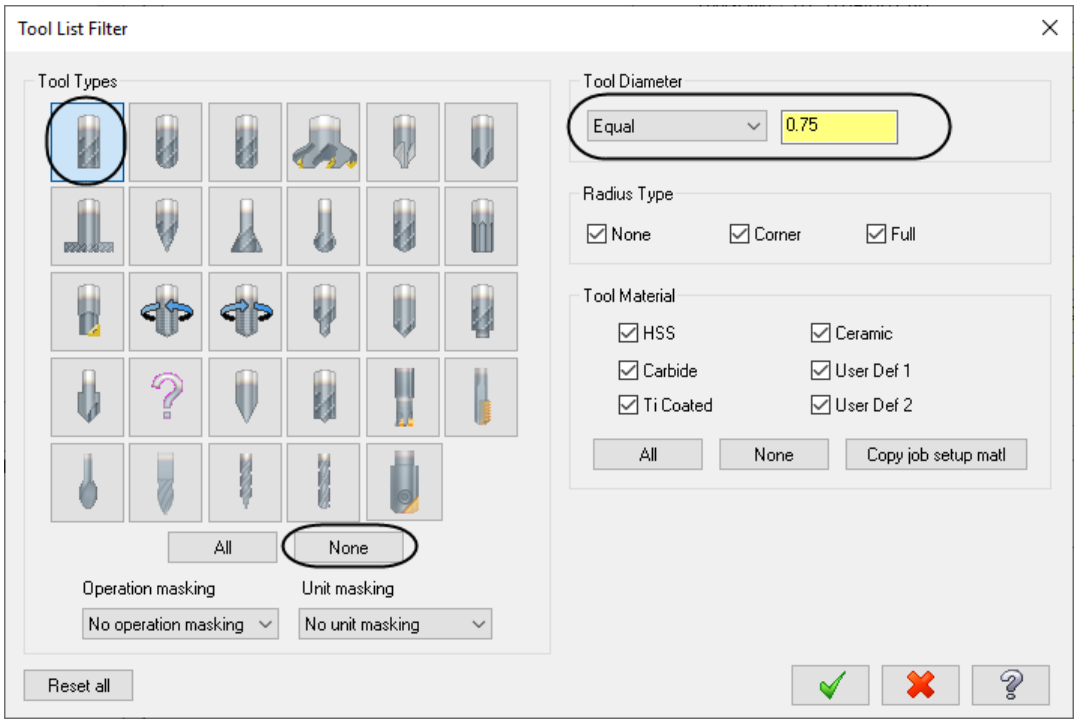
- From the **Tree View List**, on the left side of the window, select **Tool**.
- Enter a comment and click on the **Select library tool** button as outlined below.



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

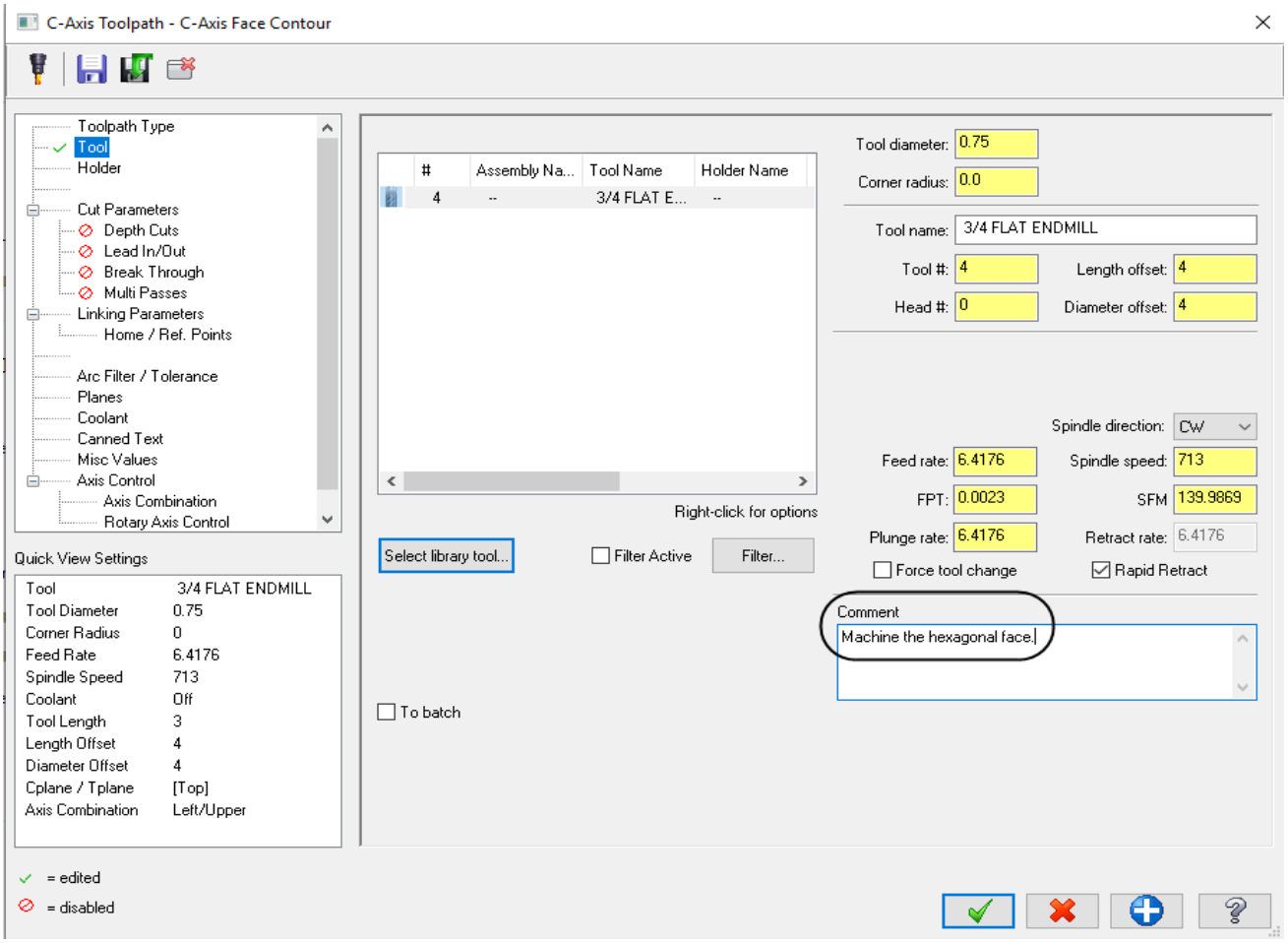


- Select the **None** button in the **Tool Types** area to disable any previous tool type selection.
- Select the **Flat Endmill** type.
- Select the drop down arrow in the **Tool Diameter** field and select **Equal**.
- Enter **0.75** in the **Diameter** field.



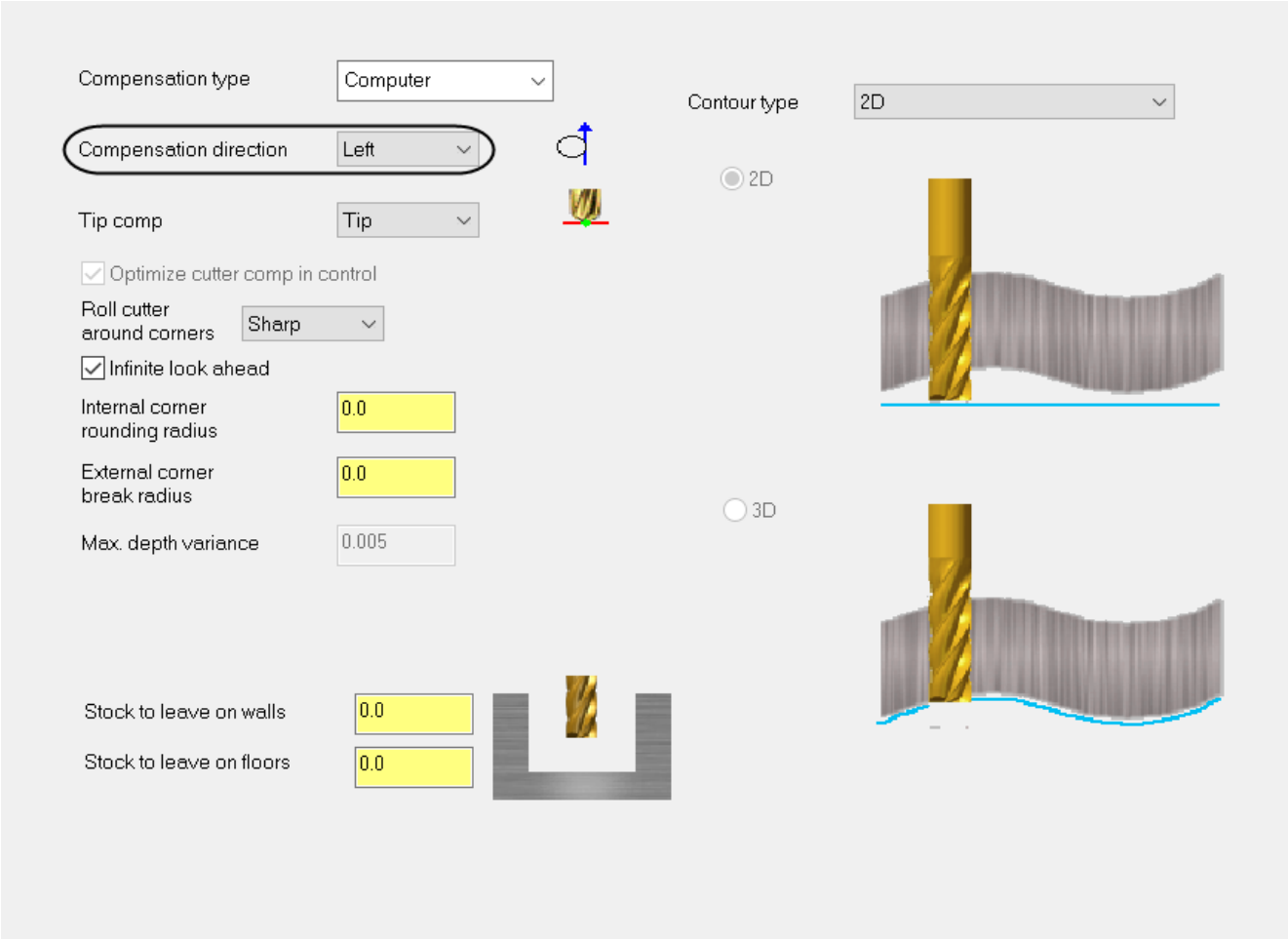
- Select the **OK** button to exit the **Tool List Filter** dialog box.
- Make sure that you click on the **3/4" Flat Endmill** in the tool list. Then click **OK** to exit the dialog box.

♦ The **Flat Endmill** should now appear in the tool list as shown.



Note: *The Feeds and Speeds are based on the tool definition. You may change them as you wish.*

♦ Select the **Cut Parameters** from the **Tree View List** and make the changes as shown.



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

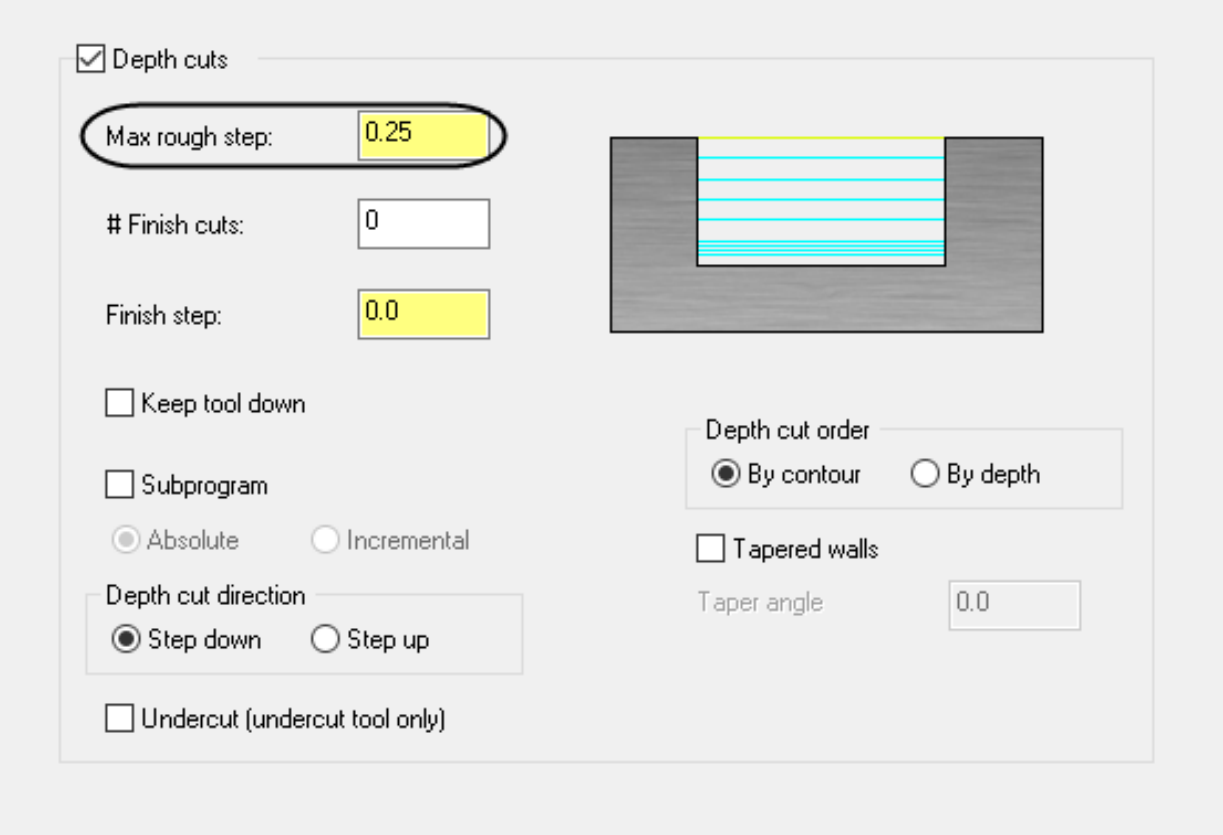
Compensation Direction set to **Left** compensates the toolpath to the left of the chain based on the chaining direction.

Roll cutter around corners set to **Sharp** inserts arc moves around corners in the toolpath. The radius of the arc moves is equal with the radius of the tool. Set to **None** to not create any extra arcs.

Infinite look ahead prevents the toolpath from crossing itself (Fish Tails).

Contour type allows the user to select what type of contour they would like to create.

♦ Select **Depth Cuts** from the **Tree View List** and make the changes as shown.



Depth cuts are the Z axis cuts that the tool makes in a contour toolpath.

Max rough step sets the maximum amount that can be machined in one step.

By contour enables the system to complete all depth cuts in the first contour before moving to the next one.

In-House Solutions

Mastercam® 2020

TRAINING SOLUTIONS

- Training Tutorials
- Instructor Materials
- Handbooks
- Professional Courseware
- Pro. Cert. Curriculum
- Online Video eCourses
- Site Licenses
- Sitewide Subscription
- Online Community

TRAINING TUTORIALS

Our popular line of Mastercam Training Tutorials cover every Mastercam application, offering the most extensive collection of Mastercam training solutions available. Each book contains a series of tutorials that guide you through creating geometry, applying then verifying toolpaths and generating G-Code. Helpful hints, review exercises and quizzes complete the learning experience.

The Home Learning Edition (HLE) Demo Software is available for download with each eBook, and can be purchased on DVD for $5.

ARE YOU AN INSTRUCTOR?

Buy 10 or more of the Mill Essentials, Mill Advanced or Lathe Training Tutorials, get the corresponding Instructor Guide for Free! (see page 5)

Mill Essentials

The Mill Essentials Training Tutorial takes a very comprehensive look at 2D machining with Mastercam.

This book begins with the absolute basics by introducing the Mastercam user interface and will have you creating geometry, mastering 2D toolpaths and posting code before you know it.

- 7 Step-by-Step Tutorials with Review Exercises
- Includes Video Training DVD and Mastercam HLE Demo Software

Price $65 (eBook)
ISBN: 978-1-77146-870-1
Price $90 (Print)
ISBN: 978-1-77146-833-6

Lathe

The Lathe Training Tutorial is ideal for both beginners and current Mastercam users.

This book introduces 2D geometry creation and lathe 2D toolpaths providing an excellent foundation for many lathe applications.

- 7 Step-by-Step Tutorials with Review Exercises
- Includes Video Training DVD and Mastercam HLE Demo Software

Price $45 (eBook)
ISBN: 978-1-77146-872-5
Price $65 (Print)
ISBN: 978-1-77146-835-0

ORDER: 1.800.529.5517 • books@inhousesolutions.com • www.eMastercam.com

Mill Advanced

The Mill Advanced Training Tutorial picks up where Mill Essentials leaves off, introducing 3D concepts including surface high speed toolpaths.

It is strongly recommended that you have a good understanding of 2D machining before beginning this book.

- 6 Step-by-Step Tutorials with Review Exercises
- Includes Video Training DVD and Mastercam HLE Demo Software

Price $60 (eBook)
ISBN: 978-1-77146-871-8
Price $80 (Print)
ISBN: 978-1-77146-834-3

Multiaxis Essentials

The Multiaxis Essentials Training Tutorial is intended for the advanced Mastercam user looking to learn Mastercam multiaxis programming.

A firm grasp on both 2D and 3D machining is required which can be attained by completing the Mill Essentials and Mill Advanced Training Tutorials.

- 10 Step-by-Step Tutorials
- Includes Video Training DVD and Mastercam HLE Demo Software

Price $85 (eBook)
ISBN: 978-1-77146-878-7
Price $90 (Print)
ISBN: 978-1-77146-841-1

Multiaxis Advanced

The Multiaxis Advanced Training Tutorial is intended for the advanced Mastercam user.

This book contains projects demonstrating a variety of Multiaxis Advanced milling applications such as pattern application toolpaths.

- 6 Step-by-Step Tutorials
- Includes Mastercam HLE Demo Software

Price $75 (eBook)
ISBN: 978-1-77146-879-4
Price $80 (Print)
ISBN: 978-1-77146-842-8

Lathe C & Y Axis

The Mastercam 2020 Lathe C & Y Axis Training Tutorial is intended for the advanced lathe user that wants to learn how to program lathe parts that include milling operations.

Advanced C-Axis toolpaths as well as the Y-Axis rotation will be described in this book. Examples on how to use the Mill toolpaths on a Lathe with Live Tooling are also incorporated.

Price $65 (eBook)
ISBN: 978-1-77146-887-9
Price $85 (Print)
ISBN: 978-1-77146-848-0

Solids

The Solids Training Tutorial provides concise step-by-step instructions on creating and manipulating 3D wireframe and solid geometry.

This book details commands such as extrude, loft, revolve, sweep, boolean add, fillet, chamfer and more.

- 9 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $40 (eBook)
ISBN: 978-1-77146-873-2
Price $60 (Print)
ISBN: 978-1-77146-836-7

Mastercam for SOLIDWORKS

The Mastercam 2020 for SOLIDWORKS Programming Exercises book provides a comprehensive "hands on" method of learning Mastercam for SOLIDWORKS. You will learn how to program a variety of different parts that require most of the toolpath types available in Mastercam for SOLIDWORKS. Extensive emphasis is put on making parametric changes and toolpath updates to match the SOLIDWORKS model changes. Primary focus is on toolpath creation on SOLIDWORKS models.

Price $65 (eBook)
ISBN: 978-1-77146-881-7
Price $85 (Print)
ISBN: 978-1-77146-844-2

Router

The Router Training Tutorial offers detailed coverage of 2D geometry creation, drilling, contouring, pocketing, nesting, block drilling and importing graphics to machine.

Instruction on tool settings, stock setup and custom profile tool creation are also included.

- 8 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $65 (eBook)
ISBN: 978-1-77146-876-3
Price $85 (Print)
ISBN: 978-1-77146-839-8

Beginner

The Beginner Training Tutorial provides a comprehensive step-by-step approach to learning the basics of three Mastercam modules: Mill Essentials (2D), Lathe and Solids.

It is an excellent choice for new Mastercam users looking to get a broader overview of the software.

- 8 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $65 (eBook)
ISBN: 978-1-77146-874-9
Price $85 (Print)
ISBN: 978-1-77146-837-4

Design

The Design Training Tutorial provides a comprehensive step-by-step approach to learning geometry creation within Mastercam.

Its focus is 2D and 3D geometry creation with explanations given on how to dimension the parts.

- 12 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $45 (eBook)
ISBN: 978-1-77146-875-6
Price $65 (Print)
ISBN: 978-1-77146-838-1

CAD Import & Mill Essentials

The CAD Import & Mill Essentials Toolpaths Training Tutorial is intended for anyone looking to understand the ins and outs of Mastercam Mill Essentials toolpaths, while learning the best practices for importing geometry from various CAD software packages.

- 7 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $70 (eBook)
ISBN: 978-1-77146-878-7
Price $90 (Print)
ISBN: 978-1-77146-846-6

CAD Import & Mill Advanced

The CAD Import & Mill Advanced Toolpaths Training Tutorial consists of 6 projects and 6 accompanying practice exercises. It is intended for intermediate to experienced Mastercam users who are primarily importing 3D geometry from another CAD software package.

- 6 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $60 (eBook)
ISBN: 978-1-77146-884-8
Price $80 (Print)
ISBN: 978-1-77146-847-3

Wire

The Wire Training Tutorial provides users with an excellent resource for learning how to use Mastercam to program wire EDM machines.

In addition to geometry creation, the book focuses on wirepaths for dies, taper angle projects and more.

- 6 Step-by-Step Tutorials with Review Exercises
- Includes Mastercam HLE Demo Software

Price $40 (eBook)
IBSN: 978-1-77146-877-0
Price $60 (Print)
ISBN: 978-1-77146-840-4

Project Workbook

The Mastercam 2020 Project Workbook is ideal for High School students, hobbyists and those who prefer engaging in project-based learning. The Workbook includes an overview of CAD/CAM and basic machining followed by a series of step-by-step projects for both mills and lathes. *Note: no machining instructions included.*

- Five projects included: Art Lithophane, Bowling Pin, Shield, F1 Car, and Jewelry Box
- Includes Mastercam HLE Demo Software

Price $60 (eBook)
ISBN: 978-1-77146-880-0
Price $75 (Print)
ISBN: 978-1-77146-843-5

Nesting eBook

This tutorial includes a variety of projects that are using Nesting options in Mastercam to fit parts onto a sheet of material for best yield.

You will learn how Nesting operates on geometry and how you can create and use tabs to assist in holding down the nested parts. You will also learn how Nesting operates on toolpaths and how you can use WCS (Work Coordinate System) to set the parts of an assembly in the proper view for machining.

Price $35
(eBook only)
ISBN: 978-1-77146-885-5

SITE LICENSE

An eBook site license allows an educational facility to purchase one or several of our Training Solutions in eBook format (PDF). This in turn grants lifetime access to the content from any workstation on campus for unlimited users.

For more information on eBook Site Licenses, email **eBooks@inhousesolutions.com** or call **1.800.529.5517**.

INSTRUCTOR KITS

These are the ultimate toolkits for instructors looking to enhance their 2D mill, 3D mill and lathe classes while minimizing prep time. Each kit contains tests, quizzes, mid-terms, finals, powerpoint presentations, lesson plans, answer sheets, and more.

Instructor Guide & Training Tutorial are included in each kit. Each print kit includes an Instructor DVD, Video Training DVD, and Mastercam HLE Demo Software. Download links are provided for eBook kits.

eBook

Bundle of all three:
$200
ISBN: 978-1-77146-882-4

Print

$100 Mill Essentials Kit
ISBN: 978-1-77146-864-0
$100 Mill Advanced Kit
ISBN: 978-1-77146-865-7
$100 Lathe Kit
ISBN: 978-1-77146-866-4

Bundle of all three:
$250
ISBN: 978-1-77146-845-9

ORDER: 1.800.529.5517 • books@inhousesolutions.com • www.eMastercam.com

NEW!

MASTERCAM PROFESSIONAL CERTIFICATION CURRICULUM

This 3-part series is designed to help you prepare for your Professional Level Mastercam Certification™ Test. The skills exam must be administered by a Mastercam Certified Instructor or a Mastercam Certified Reseller.

Part 1 – Mill 2D

Part 1 – Mill 2D begins at the absolute basics by introducing the Mastercam user interface. It will have you creating geometry, drilling and creating contour toolpaths before you know it. Its gradual progression leads to more advanced concepts such as multiple setup scenarios and 2D High Speed Toolpaths. All of the parts within the book are designed with machinability in mind.

A multitude of topics are covered including 2D geometry, tool settings, stock setup, drilling, tapping, contouring, pocketing, circle milling. You will learn the 2D High Speed Toolpaths such as dynamic mill, area mill, dynamic rest mill, how to import a solid and machine it and how to use the WCS in multiple fixture applications.

Price $100

ISBN: 978-1-77146-867-1

Part 2 – Solids

Part 2 – Solids will teach a variety of useful solids commands, that include extrude, loft, revolve, sweep, boolean add and remove, fillet, chamfer, shell, trimming, and creating solid patterns. You will learn how to change a solids color and create a solid geometry from surfaces, as well as how to manipulate a solid by using commands from the Model Prep menu such as Push-Pull, Move and Split Solid Face. You will use create Layout and learn how to dimension parts to create blueprints.

Price $100

ISBN: 978-1-77146-868-8

Part 3 – Advanced Mill 2D

Part 3 – Advanced Mill 2D is dedicated to the power user. You will learn how to machine an imported solid model using the WCS for multiple setups. To better organize the parts, you will master how to use levels and view sheets. You will practice indexing a 4-axis part using the tool planes defined in the Plane manager. Fixtures will be used and the part will be align to their faces. You will learn how to save operations to a library and how to import them for different parts. A brief introduction to the basic 3D machining will be also covered.

Price $100

ISBN: 978-1-77146-869-5

Part 1, 2 & 3 Bundle

This part 1, 2 & 3 bundle provides you all three Mastercam Professional Certification Curriculum at a better price.

Price $250

ISBN: 978-1-77146-904-3

ORDER: 1.800.529.5517 • books@inhousesolutions.com • www.eMastercam.com

HANDBOOKS

The Mastercam 2020 Handbooks provide an academic approach to teaching the theory and application of Mastercam. The Handbook series is designed to teach the fundamentals of Mastercam, gradually working up to more complex material with each volume. Each Handbook comes with a Student DVD that contains files referenced within the book, and the Mastercam 2020 HLE Demo Software.

Handbook Volume 1
Mill 2D & Solids

The Mastercam 2020 Handbook Volume 1 is an excellent resource that teaches the theory of Mastercam Mill and Solids. The material intertwines conceptual subject matter with practical applications suitable for beginners and intermediates alike. You not only learn how to use Mastercam but why things work as they do.

Price $90 (eBook)
ISBN: 978-1-77146-893-0
Price $98 (Print)
ISBN: 978-1-77146-855-8

Handbook Volume 2
3D Modeling & Machining

The Mastercam 2020 Handbook Volume 2 takes an academic approach to teaching Mastercam 3D modeling and machining. The material is most suitable for intermediates (including individuals that have completed Volume 1). The book teaches more advanced CAD modeling techniques and explains surface creation.

Price $90 (eBook)
ISBN: 978-1-77146-894-7
Price $98 (Print)
ISBN: 978-1-77146-856-5

Handbook Volume 3
Multiaxis & Machining

The Mastercam 2020 Handbook Volume 3 is an excellent resource that teaches the theory of multiaxis machining with Mastercam. This book covers the classic family of multiaxis toolpaths including drill 5-axis, curve 5-axis, swarf 5-axis, multiaxis flowline and multiaxis multi-surfaces as well as the drill & circle mill family.

Price $90 (eBook)
ISBN: 978-1-77146-895-4
Price $98 (Print)
ISBN: 978-1-77146-857-2

FREE eBOOKS

How-To Tips Collection

The In-House Solutions support specialists take note of the questions that are asked. We realize that if you're asking a question there are ten more people who want to know the answer but haven't asked. Our goal is to make your life easier, so we've collected the answers to your most popular questions here, in one convenient eBook.

Price – FREE!
(eBook only)

Horizontal Machining with Indexing

The Indexing Training Tutorial explains how to use the Plane Manager for working with tool planes and work offsets.

Examples include horizontal machining using the WCS and a multiple fixtured tombstone project.

Price – FREE!
(eBook only)

Download at www.**eMastercam**.com/**freebooks**

Mastercam @Courses

Included with each eCourse: HLE demo software, corresponding PDF eBook, and personalized final certificate.

Mastercam CAD/CAM Systems 2020 Home Learning Edition Demo Software DOWNLOAD

LOG IN & LEARN TODAY!

Expand your Mastercam knowledge, anytime and at your own pace, with our selection of step-by-step online video eCourses.

Mastercam eCourses provide both novice and seasoned Mastercam users with the tools and information they need to excel.

The *"log in & learn"* format of each eCourse allows users to set up their own online classroom, where each user's experience is customized and tracked. Because the eCourses are not subscription-based and have no expiration date, users can log on and off at any time, and finish the program at their convenience without additional expense.

Approximate completion time ranges from 7.5 to 15 hours depending on the eCourse, making it possible for employees or students to gain new skills outside of work or studies.

Highlights:

- Online previews with table of contents including the time it takes to complete each session.
- Mastercam 2020 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.
- **Instant, lifetime access**

$149 each

ORDER: 1.800.529.5517 • books@inhousesolutions.com • www.eMastercam.com

Mill Essentials eCourse

The Mill Essentials eCourse introduces students to 2D CAD and milling toolpaths. It covers wireframe and solids creation as well as 2D mill toolpaths such as contour, drilling, blend, peel, dynamic area, transform, Feature Based Drilling, and more. This course serves as an excellent introduction to Mastercam.

Mill Advanced eCourse

The Mill Advanced eCourse builds on what students have learned in the Mill Essentials eCourse. It moves into more advanced CAD and demonstrated 3D wireframe, solid, and surface creation commands. 3 axis toolpaths such as Area Roughing, Dynamic OptiRough, Scallop, Pencil, Waterline, Radial, Hybrid, and more are covered.

Lathe eCourse

The Mastercam Lathe eCourse covers wireframe creation, working with imported part files, stock setup, facing, roughing, finishing, grooving, drilling, and cutoff toolpaths. Stock operations such as advance, flip, and tailstock are also covered. You will also learn how to program parts in a VTL.

Multiaxis Essentials eCourse

The Multiaxis Essentials eCourse covers 4 & 5 axis toolpaths. Toolpaths include contour with axis substitution, drilling with axis substitution, drilling with rotary axis positioning, rotary 4-axis, curve 5-axis, swarf 5-axis, drill 5-axis, circle mill 5-axis, flow 5-axis, and multisurface 5-axis. This course skips most CAD in favor of focusing on toolpaths.

www.**eMastercam**.com/**eCourses**

2020

ORDER: 1.800.529.5517 • books@inhousesolutions.com • www.**eMastercam**.com

ORDER: 1.800.529.5517 • books@inhousesolutions.com • www.eMastercam.com

www.**eMastercam**.com

Sitewide Subscription

An eMastercam sitewide subscription will grant you instant access to all of our online resources, including our full library of eBooks and eCourses. Once subscribed, you will be able to watch all of our streaming eCourses in your browser or read any of our Mastercam eBooks using our Webviewer.

$39.99 /1 month

$129.99 /4 months (save $29.97)

BEST VALUE:

$359.99 /12 months (save $119.89)

www.**eMastercam**.com/
sitewide-subscription

PROFESSIONAL COURSEWARE

The Mastercam Professional Courseware titles are intended for industrial training settings. Instead of step-by-step instructions, these books introduce concepts through sequences of specialized training exercises followed by parts the users are expected to produce with minimal guidelines.

Professional Courseware Mill Essentials

The Mastercam 2020 Mill Essentials Professional Courseware provides in-depth coverage of 2D wireframes and solids geometry, as well as contour, pocket, drilling, circle milling and slot milling toolpaths. More advanced exercises explain the use of the Work Coordinate System (WCS), 2D high speed toolpaths, Feature Based Machining (FBM) and more.

Price $60 (eBook)
ISBN: 978-1-77146-896-1
Price $70 (Print)
ISBN: 978-1-77146-858-9

Professional Courseware Mill Advanced

The Mastercam 2020 Mill Advanced Professional Courseware covers a multitude of features that teach a user to create 3D wireframes, surfaces and solids for 3D modeling and toolpaths. Interactive training exercises introduce 3D geometry functionality, while newer surface high speed toolpaths are thoroughly investigated along with their various parameter settings.

Price $50 (eBook)
ISBN: 978-1-77146-897-8
Price $60 (Print)
ISBN: 978-1-77146-859-6

Professional Courseware Lathe

The Mastercam 2020 Lathe Professional Courseware offers an in-depth look at Mastercam Lathe geometry and toolpath creation. Advanced toolpaths such as Misc Ops and C-Axis toolpaths are also described. Additional Mastercam files are provided along with guidelines for creating the toolpaths to machine each part.

Price $55 (eBook)
ISBN: 978-1-77146-898-5
Price $65 (Print)
ISBN: 978-1-77146-860-2

Professional Courseware Multiaxis

The Mastercam 2020 Multiaxis Professional Courseware details numerous toolpaths that allow a user to successfully machine 4-axis and 5-axis parts. Multiaxis Advanced toolpaths have been included with more complex parts along with instructions on how to machine them. Machine Simulation is used to check for any collisions between the part, the tool and any of the machine's components.

Price $65 (eBook)
ISBN: 978-1-77146-899-2
Price $75 (Print)
ISBN: 978-1-77146-861-9

Join the eMastercam community!

eMastercam is one of the largest and oldest online forums for swapping knowledge of CNC machines, tools, manufacturing processes and technology. After hours, eMastercam continues to be a place for Machinists, Engineers and others in the manufacturing industry to connect, share stories, opinions and get to know each other beyond the parts we make and the tools we use.

- Active user forums with an education-specific sub forum
- Free downloads and educational resources
- Free post processors and advanced development requests
- Overviews of Mastercam 2020 and related products
- The place to find our Mastercam Training Solutions
- Industry insights and much more!

www.**eMastercam**.com/**forums**

eMastercam.com
YOUR ONLINE MASTERCAM SOURCE

We don't just educate, we inspire!

eMastercam
We appreciate your feedback

TELL US WHAT YOU THINK

Our goal is to provide you the best Mastercam Training Solutions, and we do that with your help.

- Tell us what you like about our training solutions
- Let us know what you think can be improved
- Give us suggestions for future products

www.**eMastercam**.com/**feedback**

In-House Solutions

240 Holiday Inn Drive, Unit A
Cambridge, Ontario, Canada N3C 3X4
T: 800.529.5517 F: 519.658.1335
books@inhousesolutions.com
www.**inhousesolutions**.com