

Mastercam 2022



LATHE C&Y AXIS

TRAINING TUTORIAL SERIES | IMPERIAL



2022

HLE DOWNLOAD
INSTRUCTIONS INSIDE

***Mastercam*® 2022**

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



HOW TO DOWNLOAD:

Mastercam Home Learning Edition (HLE)
Demo Software

Visit <https://signup.mastercam.com/try-mastercam>

Proceed to download!

Mastercam 2022 Lathe C & Y Axis Training Tutorial

Copyright: 1998 - 2022 In-House Solutions Inc. All rights reserved

Software: Mastercam 2022

Author: Mariana Lendel

Date: July 13, 2021

ISBN: 978-1-77146-948-7

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.

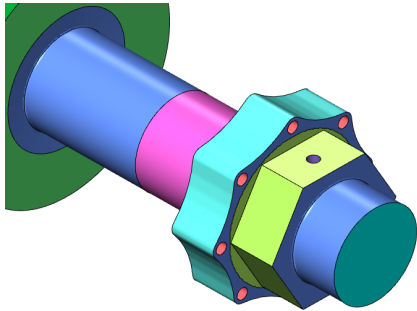
Lathe C&Y Axis Projects

Tutorial #1

Topics Covered

Tutorial #1: Geometry Creation

- Create the 2D geometry in the Top plane on Level 1
- Create the 3D geometry using the Right plane and Translate command



Tutorial #1: Toolpath Creation

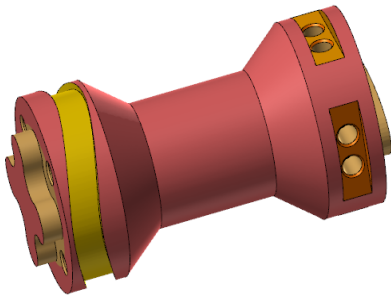
- Generate the 2D toolpaths to machine the OD such as: Face, Rough, Finish and Groove
- General overview of Construction Planes/Graphic Views using the Status Bar and the Planes Manager.
- Generate the C-axis toolpaths such as: Face Contour, Face Drilling, and C-Axis Drilling used to machine the part from the Right plane.

Tutorial #2

Topics Covered

Tutorial #2: Geometry Creation

- Create the 2D geometry in the Top plane on Level 1
- Create the 3D geometry using the Right plane and Translate command



Tutorial #21: Toolpath Creation

- Generate the 2D toolpaths to machine the OD such as: Face, Rough, Finish and Groove
- General overview of Construction Planes/Graphic Views using the Status Bar and the Planes Manager.
- Generate the C-axis toolpaths such as: Face Contour, Cross Contour, Cross Drilling and C-Axis Contour to machine the part.

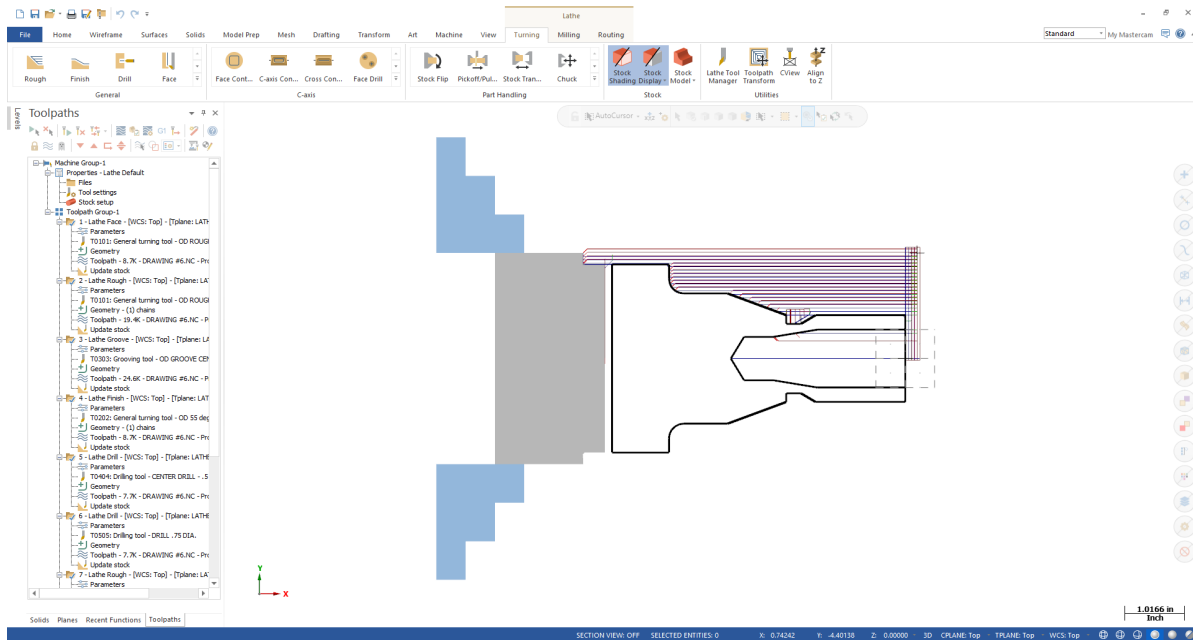
Table Of Contents

Lathe C & Y Axis Training Tutorial	1
Getting Started	7
Introduction “What Is “Mastercam“?	8
Mastercam® User Interface:	9
Setting The Grid	10
Home tab Attributes and Organize And the Mini Toolbar	11
Data entry shortcuts	13
The Right Mouse click Menu	14
AutoCursor	15
Graphic views, Planes & The Work Coordinate System	17
Setting The Construction And Tool Plane To +D+Z	18
Mastercam Parameter Dialog Boxes:	19
Mastercam® Work Flow:	20
Conventions used in this book:	24
Lathe C & Y Axis Part #1	25
Introduction:	26
New Toolpath Information	27
Instructor Demonstration:	28
Create the 2D Geometry	30
Create The 3D Geometry	36
Step 1: Select The Machine And Set Up The Stock	47
Step 2: Lathe face toolpath	52
Step 3: Lathe rough OD toolpath	54
Step 4: Lathe finish OD toolpath	57
Step 5: Machine The Leftover Material - Groove	58
Step 6: Machine The Hexagon - C-Axis Face Contour	60
Step 7: Machine second Hexagon - C-Axis Face Contour	67
Step 8: C-Axis Face Drill The Part	71
Step 9: Create Another C-Axis Face Drill Toolpath	74

Step 10: Spot Drill The Hole Using Cross Drill	77
Step 11: Cross Drill The Hole To Size	81
Step 12: Cutoff The Part	84
Step 13: Post Process the file	87
Step 14: Save The Updated MCAM File	87
Review Exercise - Student Practice	88
Lathe C & Y Axis Part #2	93
Introduction:	94
New Toolpath Information	95
Instructor Demonstration:	96
Geometry Creation	97
Step 1: Setting Up The Graphical User Interface	98
Step 2: Create And Organize Levels	99
Step 3: Create Lines To Represent The Outside Profile	103
Step 4: Change The Construction Plane, Main Level And Color	112
Step 5: Create The Ellipse On The Right Face	113
Step 6: Create A Circle	115
Step 7: Create A Rectangle	117
Step 8: Trim The Rectangle	119
Step 9: Break The Lines	121
Step 10: Create Arcs And Rotate Them	123
Step 11: Create The Circles On The Left Face	127
Step 12: Insert Fillets between The R 0.250" Circles	131
Step 13: Trim The Rounded Contour	134
Step 14: Create Points	136
Step 15: Create The Spline	137
Step 16: Select The Machine And Set Up The Stock	140
Step 17: Machine The Ellipse On The Right Face	147
Step 18: Backplot The Toolpaths	152
Step 19: Simulate the Toolpath in Verify	153

Step 20: Machine the Slots Using Y axis Cross Contour	155
Step 21: Transform Rotate toolpath	160
Step 22: Spot Drill The Holes Using Cross Drill	164
Step 23: Drill The Holes Using Cross Drill	167
Step 24: Transfer The Stock To The Right Spindle	169
Step 25: Machine The Profile in The Right Spindle	172
Step 26: Face Drill The Holes In The Right Spindle	178
Step 27: Create Another Face Drill Toolpath	181
Step 28: Machine The Spline	184
Step 29: Post Process the file	188
Step 30: Save The Updated MCAM File	188
Review Exercise - Student Practice	189
Quiz Answers	195
	195

Getting Started



INTRODUCTION “WHAT IS “MASTERCAM“?”

Mastercam® is the world's most widely-used CAM software from CNC Software, Inc.

Machine tool programmers can program parts graphically in Mastercam, using the innovative, time tested, proven toolpaths and machining strategies preferred the most by shops around the world. This book is a step-by-step “hands-on” book of examples to have you programming quickly without knowing NC Machine programming languages.

WHO THIS BOOK IS FOR:

This book is for the machine tool programming student who wants to learn how to program parts using Mastercam. This book is intended to be used as reinforcement exercises as part of classroom training. While a student can learn from books or videos if they are highly motivated, there is no substitution for the interaction with a “live” instructor.

This book covers Mastercam Lathe (Turning toolpaths) and assumes you are new to the Mastercam programming environment. This book also assumes that the student using this book is familiar with machining processes, tools and how Numerical Control (NC) machines operate. This book is intended to teach you how to get your job done using Mastercam.

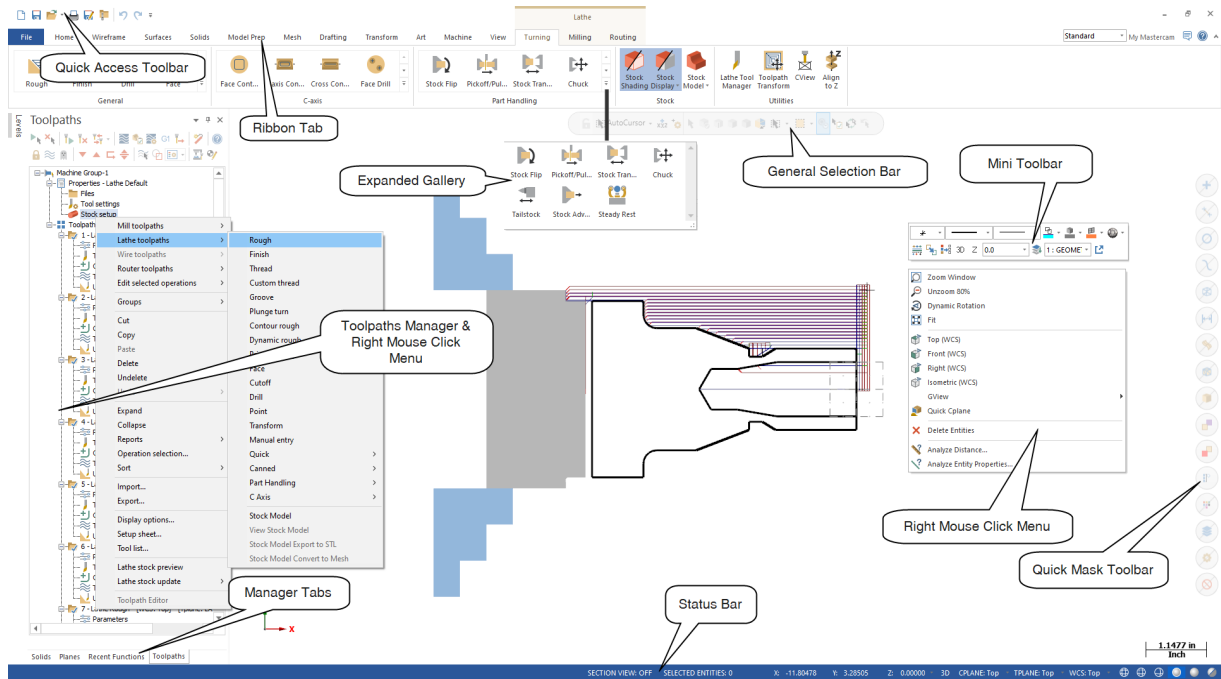
PROGRAMMING TECHNIQUES USED IN EXAMPLES:

The example exercises in this book can be programmed an infinite number of ways. In fact, any two programmers would likely approach each part a different way and probably use different tools or toolpaths. For the exercises in this book, we used settings that were the personal preference or that came from the experience of multiple Mastercam authorized instructors. All of the examples and exercises have options that can be set or changed to suit your experiences or company standards.

We chose one method rather than clutter the book with all of the possible options and disrupting the workflow of the exercises. Utilize the Mastercam Help that is accessible from most dialog boxes whenever you need more information about options.

MASTERCAM® USER INTERFACE:

Mastercam is a completely graphic programming interface that includes built in CAD tools. You will find the interface to be very streamlined and easy to access all of the toolpath generating commands. There are several places you need to know to access all of the Mastercam commands. The locations for the Mastercam commands and tools are shown in the screen display below.



Quick Access Toolbar	Quick Access Toolbar (QAT) allows you to quick access the most common used functions.
Ribbon Tabs	Access to all Mastercam commands, features and functions.
Expanded Gallery and Tooltips	Additional icons to start toolpaths and hints on the toolpath.
General Selection Bar	Used to access AutoCursor settings and selection options.
Mini Toolbar	Quick access to attributes.
Right Mouse Click Menu	Quick access to commands relevant to the graphics view and analyze commands.
Quick Mask Toolbar	Used to control entity masking for easier selection.
Mastercam Toolpath Manager	This is where all toolpath operations you create are stored. They can be accessed here for editing the Parameters, Tools, and the Geometry of the toolpaths.
Toolpath Manager Right Mouse Click Menu	Special commands for working with the operations you have created.
Managers Tabs	Tabs to control which panel is displayed in the Manager panel.
Status bar	Displays the current WCS, Tool and Construction plane info and current cursor location and also displays the Section View.

SETTING THE GRID

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

File

■ Configuration.

- Select **Screen** from the configuration **Topics**.
- Select the plus sign (+) beside screen as shown.

■ Spacing

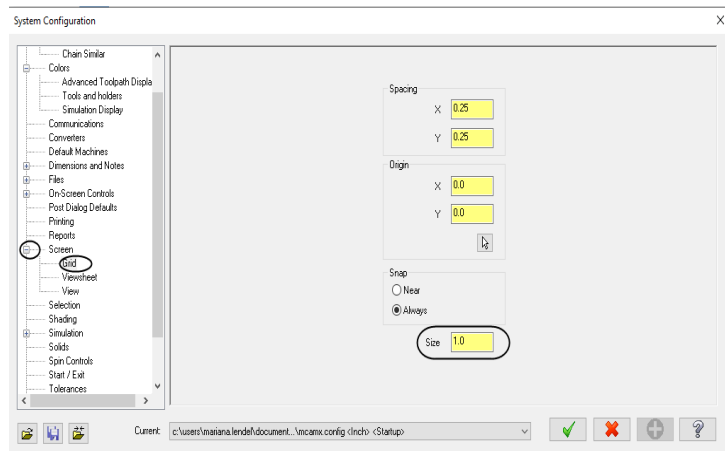
- **X = 0.25.**
- **Y = 0.25.**

■ Origin

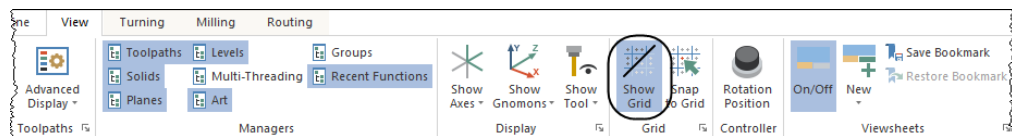
- **X = 0.0.**
- **Y = 0.0.**

■ Size = 1.0.

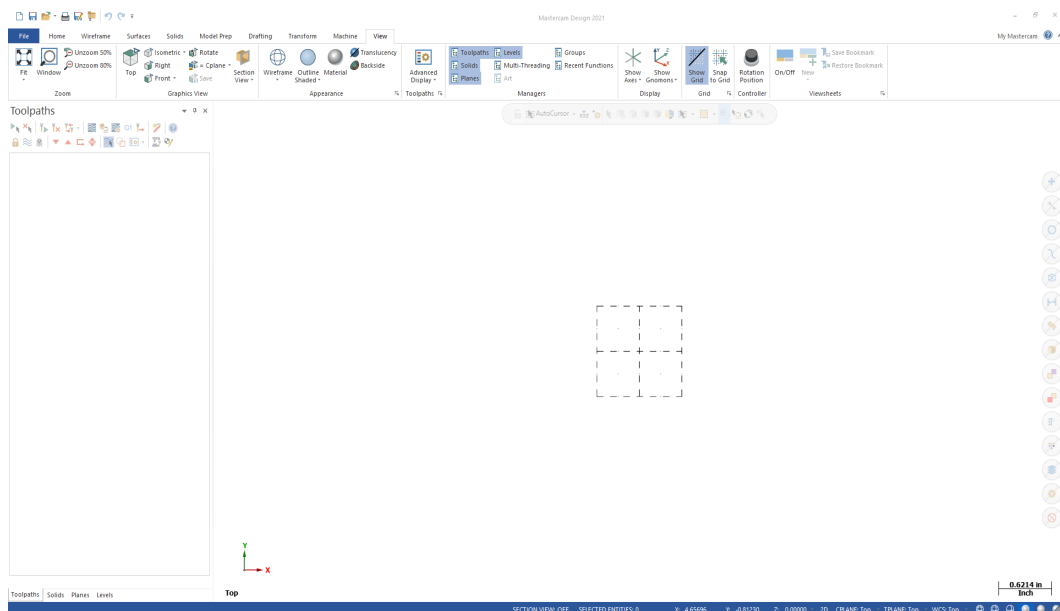
- Select the **OK** button to exit.
- Select the **Yes** button to save the settings in the **System Configuration**.



- To see the **Grid** in the graphics window, from the **View** ribbon enable **Show Grid** as shown.



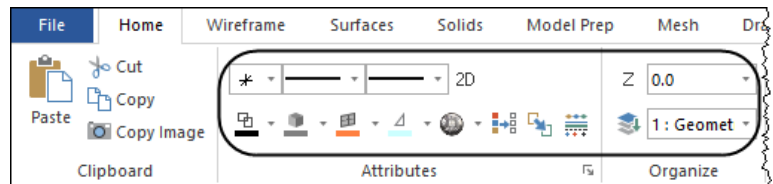
- The **Grid** should look as shown.



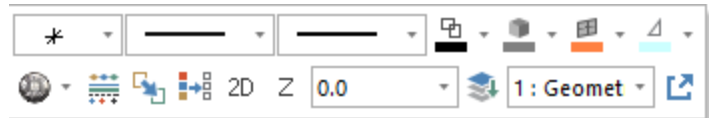
HOME TAB ATTRIBUTES AND ORGANIZE AND THE MINI TOOLBAR

The **Home** tab **Attributes** and **Organize** groups or the **Mini Toolbar** allows you to set the attributes (color, level, style and width) and Z depth currently used.

The **Home** tab with the **Attributes** and **Organize** groups should look as shown.



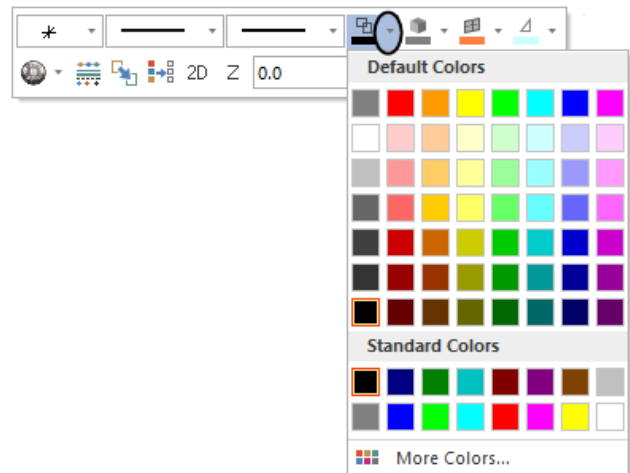
Example of the default **Mini Toolbar**.



Point Style	Sets the display of points. The box or circle point styles are especially useful when working with 3D Drawings.
Line Style	Line style, such as solid, hidden, or phantom.
Line Width	Line width and surface UV line width.
Wireframe, Solids & Surface Color	You can choose to set or change the Wireframe color, the Solids color and the Surface color. Colors help organize and simplify the drawing. Up to 256 colors are available. Select from the color dialog box by clicking on the colored area. The number inside the box represents the color palette number.
Clear Colors	Removes the group and result colors from affected entities and from the database. When performing a transform function, Mastercam creates a temporary group from the originals (red) and a result (purple) group from the transformed entities. However, the groups stay in effect only until you use the Screen, Clear Colors function or perform another transform function.
Select All	Changes multiple attributes.
Set from entity	Allows you to select an existing entity and sets the current attributes based on the entity attributes.
2D/3D Mode	2D or 3D drawing mode. Determines whether Z-inputs are from the Working Depth field or from mouse.
Z	Sets the Z depth of the Construction plane.
Change Level	Change the level of the selected entities.
Set Material	Displays and sets the material for selected objects.

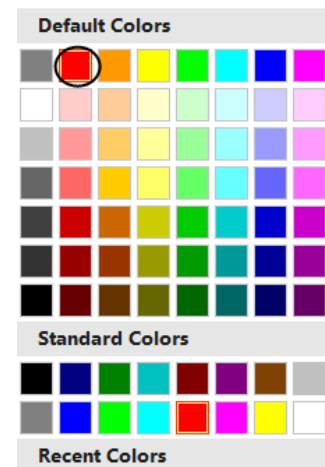
Example Of Setting The Current Wireframe Color

- First, make sure that no entities are selected in the graphics window.
- Right mouse click to open the **Mini Toolbar** and click on the drop down arrow next to the **Wireframe** color 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.



DATA ENTRY SHORTCUTS

Mastercam lets you use several shortcuts to enter information into text boxes when creating geometry. These shortcuts let you get data from an existing entity and use it when creating new geometry.

The shortcuts are as follows:

A	Use " A " to input an existing angle.
D	Use " D " to input an existing diameter.
L	Use " L " to input an existing line length.
R	Use " R " to input an existing radius.
S	Use " S " to input the distance between two existing points.
X	Use " X " to input an existing X coordinate.
Y	Use " Y " to input an existing Y coordinate.
Z	Use " Z " to input an existing Z coordinate.

To Use A Shortcut

- Type a letter (from the list above) instead of a value into the text box.
- Press **Enter**. Mastercam prompts you to select the existing entity from which you want to get the data.
- Select the entity you want to use.
- Mastercam displays the data it gets from the existing entity.
- Press **Enter** to accept the data. Type a different value and press **Enter**.
- Mastercam uses the data from the existing entity in creating the new entity.

Data Entry Shortcut Example

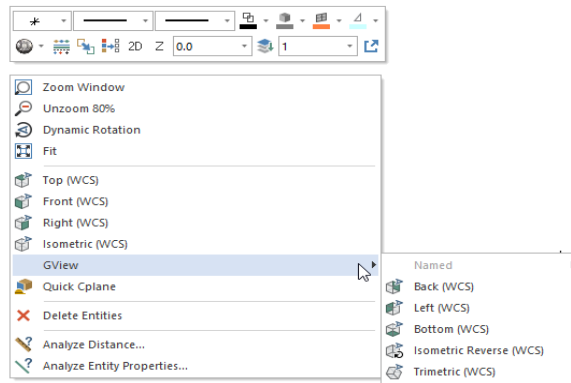
This example uses the "**L**" data entry shortcut to create a line.

Create a line with **Endpoints** of **0,0,0** and **3,0,0**.

This will be the existing line from which you get data by using "**L**".

- Select the **Wireframe** tab and click on the **Line Endpoints** icon.
- Select the **Origin** as the first line **Endpoint**.
- Set **45** as the angle of the line and press **Enter**.
- Type "**L**" to activate the **Length** field and then highlight the existing value and type "**L**" in the field and press **Enter** again.
- Mastercam prompts you to select an existing line.
- Select the existing line.
- Mastercam displays the data it gets from the existing line.
- Press **Enter** to accept this data.
- Select the **OK** button to exit.
- Mastercam creates the new line.

THE RIGHT MOUSE CLICK MENU

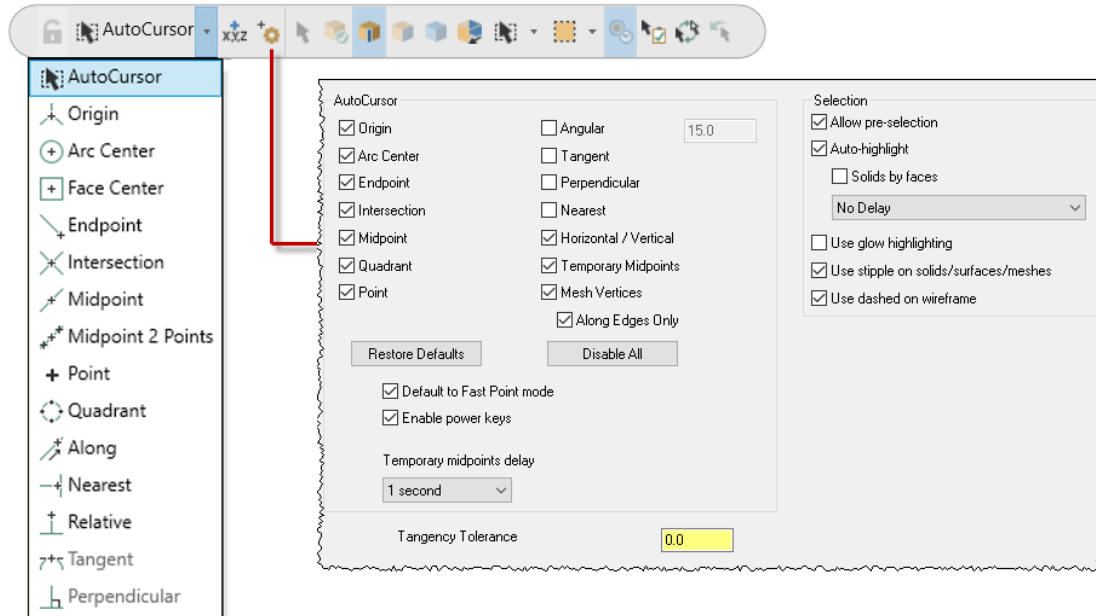


Zoom Window	Magnifies a portion of the graphics window. When you use this function, Mastercam prompts you to draw a rectangular window around the portion of the graphics area that you want to magnify.
Unzoom 80%	Reduces the size of the displayed geometry to 80% of its current size.
Dynamic Rotation	Lets you dynamically rotate entities in the graphics window around a selected point or position. When you choose this function from the menu or toolbar, you first select an Endpoint, Point, or a position in space, and then move the mouse to rotate entities around the selected point. Mouse click when the drawing is in the desired position.
Fit	Fit screen maximizes your view of visible geometry in the graphics window.
Top	Sets the graphics view to the top view.
Front	Sets the graphics view to the front view.
Right	Sets the graphics view to the right view.
Isometric	Sets the graphics view to an isometric view.
Gview	Allows you to set the graphics view from Back, Left, Bottom, Isometric Reverse or Trimetric.
Delete Entities	Erases the selected entities.
Analyze Distances	Analyzes the distance between two points.
Analyze Entity	Analyzes and allows you to edit selected entity lengths, angles, positions and attributes.

AUTOCURSOR















Allows the user to select Endpoints, Midpoints, Arc Center Points or various other entity points. Mastercam prompts you to enter a point. To select a point move the cursor over the geometry on the screen. A little square will appear on the screen and a corresponding cursor icon will be displayed.

AutoCursor options are available each time Mastercam prompts you to enter a point. You can select to override the **AutoCursor** or to set the defaults from the **General Selection Bar**.



Note: You can set the **AutoCursor** defaults. This will determine which entities are shown when your cursor hovers near by.

Cursor Icons

-  Origin.
-  Center.
-  Face Center.
-  Endpoint.
-  Intersection.
-  Midpoint.
-  Midpoint 2 Points.
-  Point.
-  Quadrant.
-  Along.
-  Nearest.
-  Relative.
-  Tangent.
-  Perpendicular.

GRAPHIC VIEWS, PLANES & THE WORK COORDINATE SYSTEM

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.

The main tool for applying coordinate systems in Mastercam is the plane. A plane consists of two main parts: a slice through the coordinate system and an origin, or zero point.

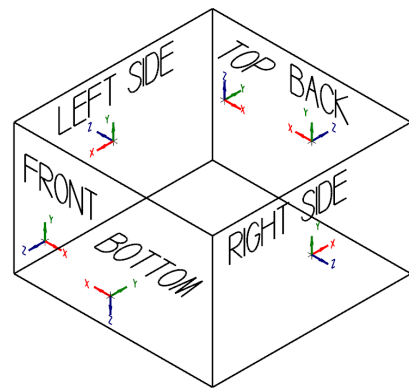
The **Construction Plane (Cplane)** is the plane in which new geometry is created.

The **Tool Plane (Tplane)** is the cutting plane for a toolpath, typically normal to the tool axis.

The **Work Coordinate System (WCS)** 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.

There are 9 standard Graphics Views or Planes in Mastercam.

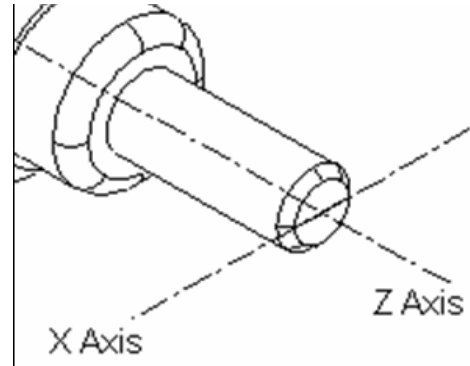
View 1	Top.
View 2	Front.
View 3	Back.
View 4	Bottom.
View 5	Right.
View 6	Left.
View 7	Isometric.
View 8	Isometric Reverse.
View 9	Trimetric.



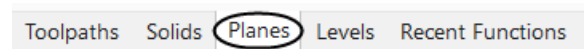
SETTING THE CONSTRUCTION AND TOOL PLANE TO +D+Z

The **Planes** settings **+D+Z** apply to construction methods for geometry requiring two axes of motion, the Z axis and a diameter value (the D value represents the diameter of X). Set the Cplane to +D+Z and Mastercam interprets X axis values as diameter values (as opposed to radius values).

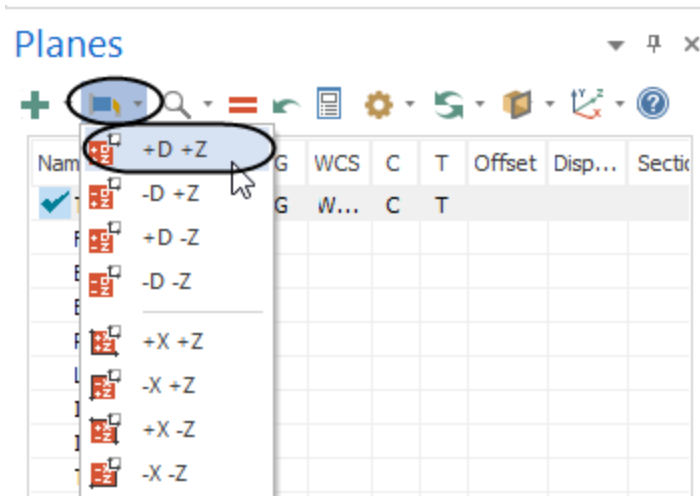
- The absolute zero or origin of the part is normally at the front face and center line of the part. The center line forms the Z axis as shown in the following example.



- Select the **Planes** tab on the bottom left side of the screen as shown.



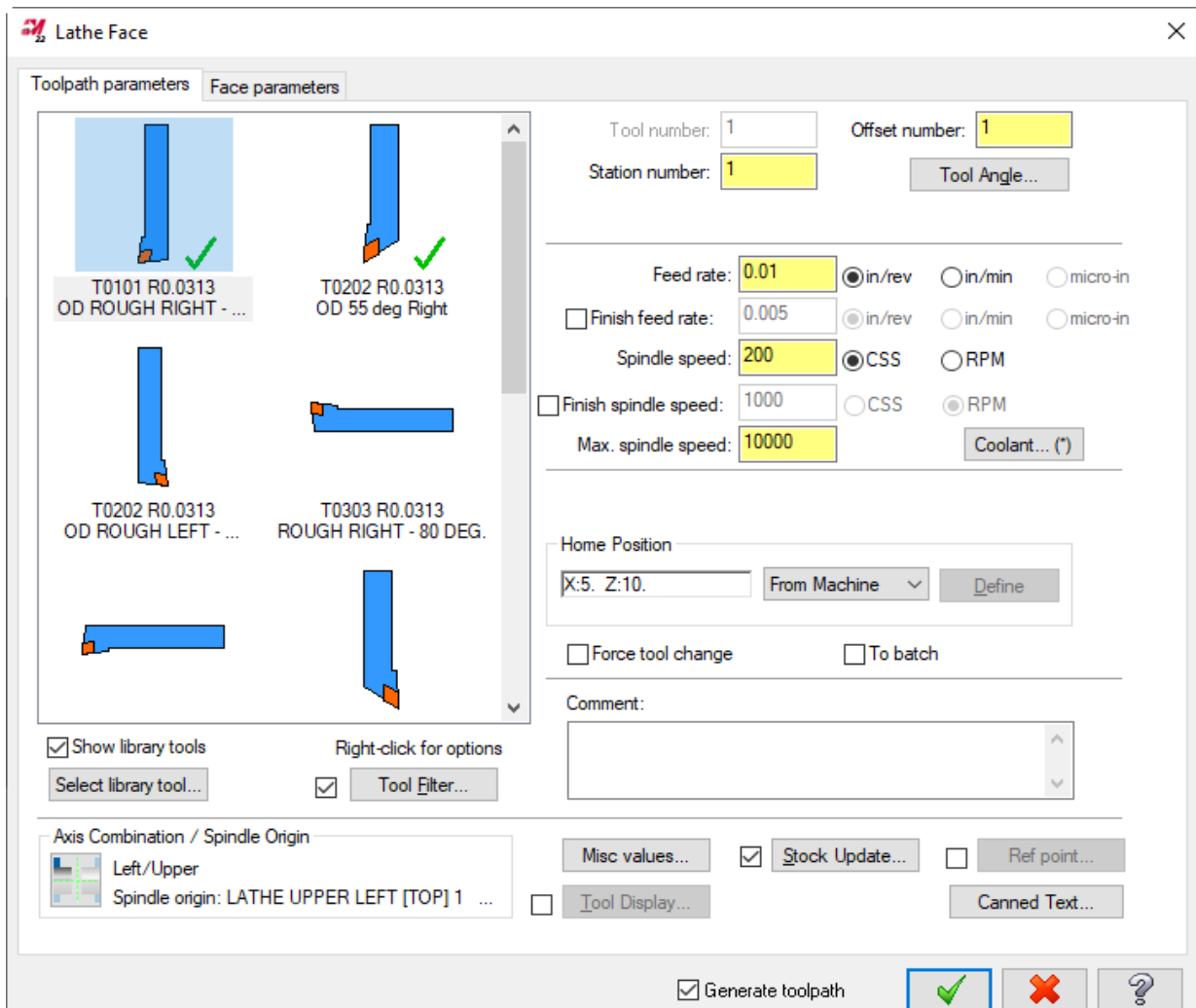
- From the **Planes** manager, click on the **Select lathe plane** button and select +D+Z as shown.



- Select **Lathe Diameter** and **+D+Z**.

MASTERCAM PARAMETER DIALOG BOXES:

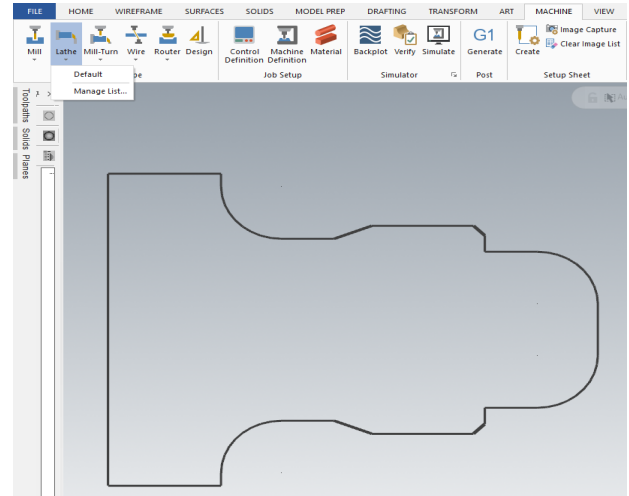
When you begin creating toolpaths with Mastercam® you will be required to supply information so you can create the toolpath exactly the way you want it. Mastercam® utilizes a tree-style dialog box interface that is made up of several distinct areas as shown.



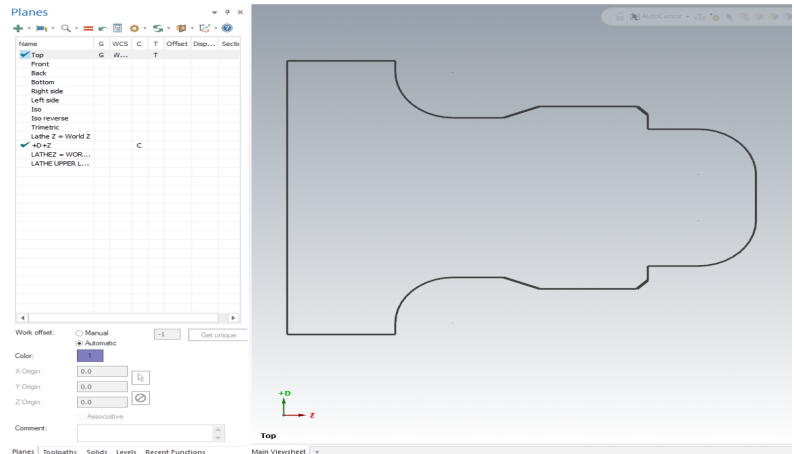
MASTERCAM® WORK FLOW:

The process to generate a toolpath will be repeated over and over through the tutorials in this book. You will find the process simple and straightforward once you have programmed a few parts. Following is an outline of the process we will follow to create programs:

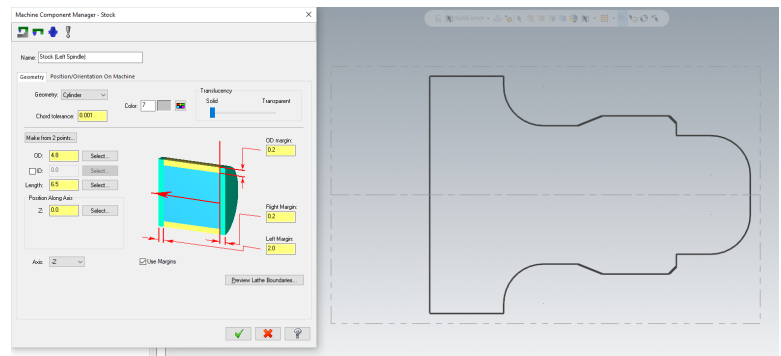
1. Create the Geometry or Open the Mastercam file and select the Machine if it is different than Default



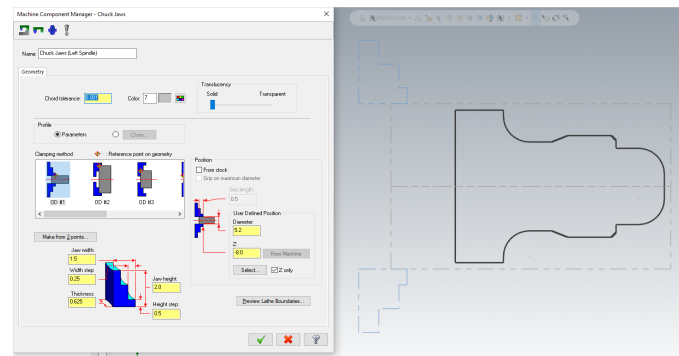
2. Transform the Geometry if needed, identify the plane and locate the origin where you want it.



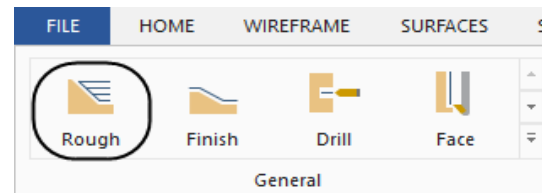
3. Define the stock size that your part will be cut from and set tool information.



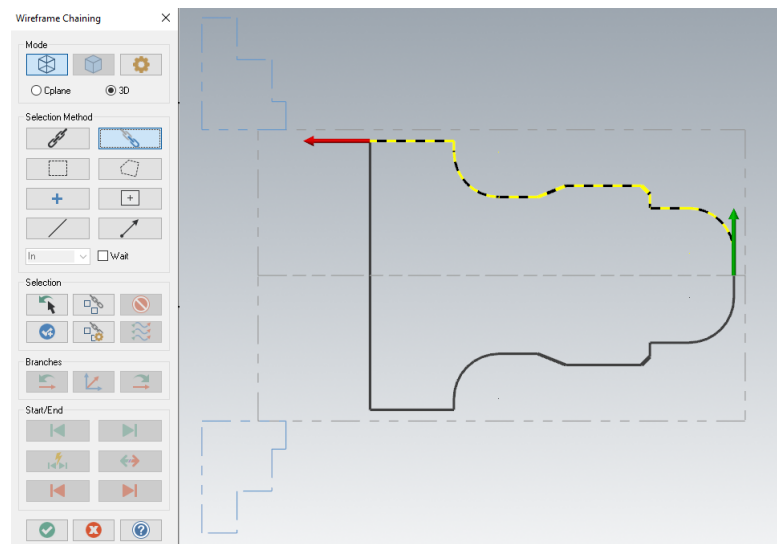
4. Define the Chuck Jaws to hold the part in place .



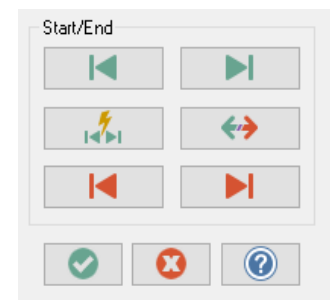
5. Select a toolpath operation type such as Facing, Roughing or Drilling.



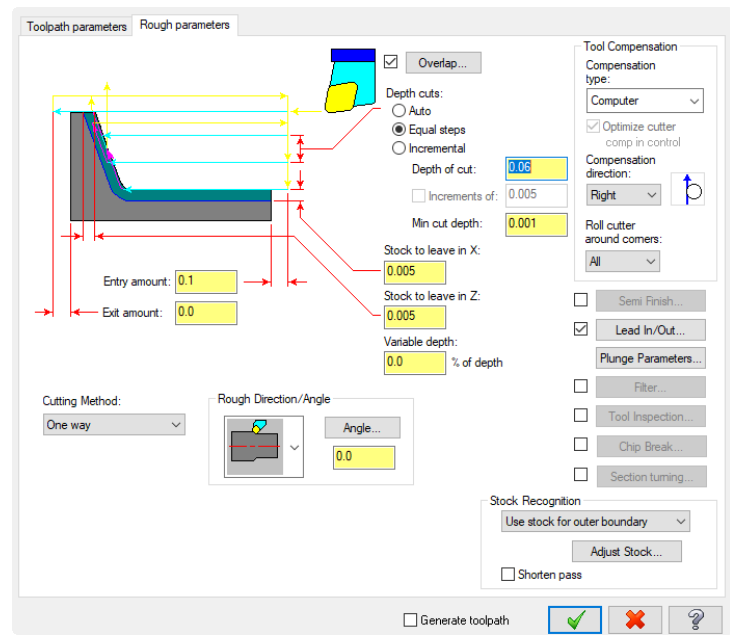
6. Select the geometry of the part you will cut with the selected operation using geometry selection tools.



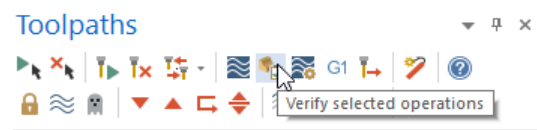
7. Confirm or adjust the Chain direction, Start Point, End Point, etc., if needed.



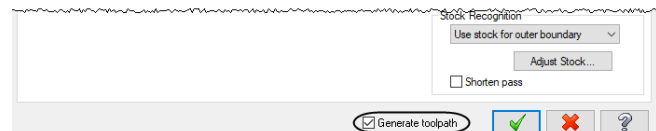
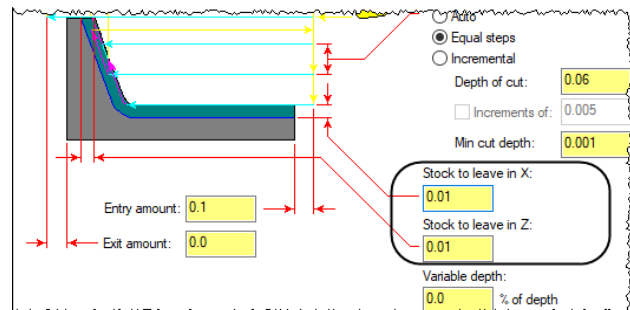
8. Fill in the necessary information on the Parameters pages of the dialog box that appears for the toolpath type you choose.



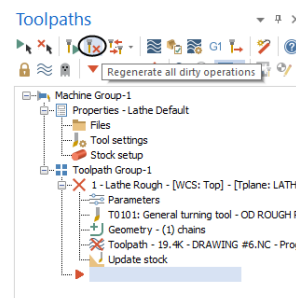
9. Verify the toolpath on your computer screen to confirm the results are as you expected, using Backplot and/or Solid Verify.



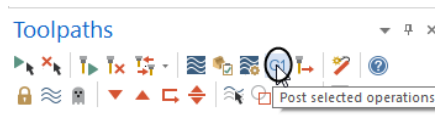
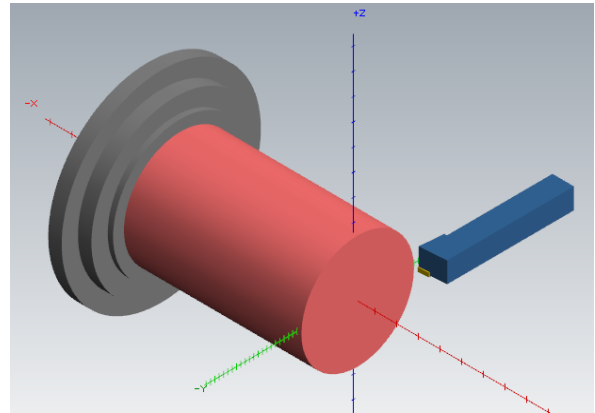
10. Make any changes as required by changing parameters.



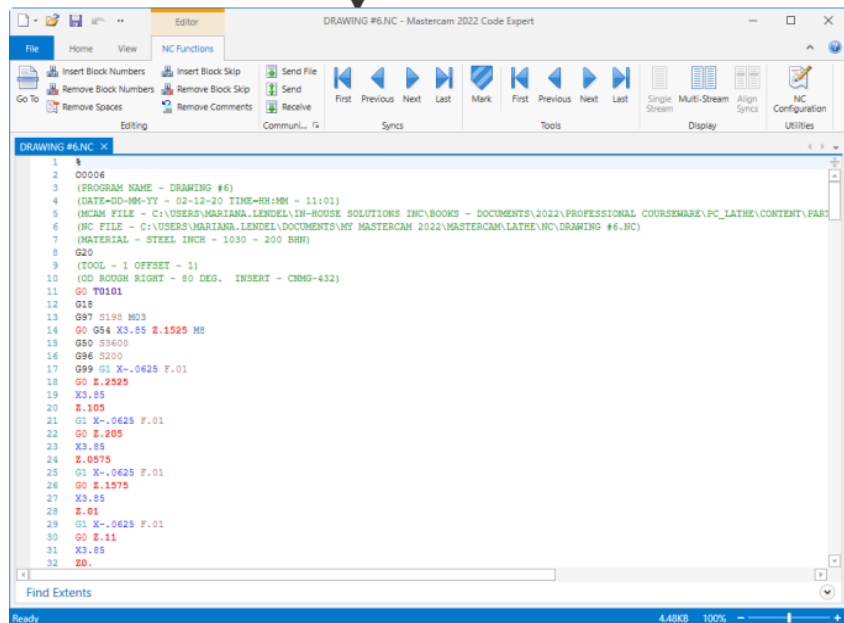
11. You can update the toolpath by enable Generate toolpath in the parameters or by clicking on the Regenerate the "Dirty" operation in the Toolpaths Manager.



12. Verify again to make sure the toolpath is correct.



13. Convert the graphical toolpath information into machine code by Post Processing and sending it to the CNC machine.



CONVENTIONS USED IN THIS BOOK:

We have attempted to make this manual as uncluttered as possible and provide you with reference information when it is appropriate. It is not intended to be a Reference Guide or all-encompassing user manual.

The text styles used are the followings:

Bold Text - Represents menu commands, dialog box settings or other similar items from the screen

Standard Text - Represents normal wording needed to provide you the instruction.

STEP 1: STEP TITLES

1.1 Sub step titles

Information about the current step or terms or parameter definitions describing the parameters and description.

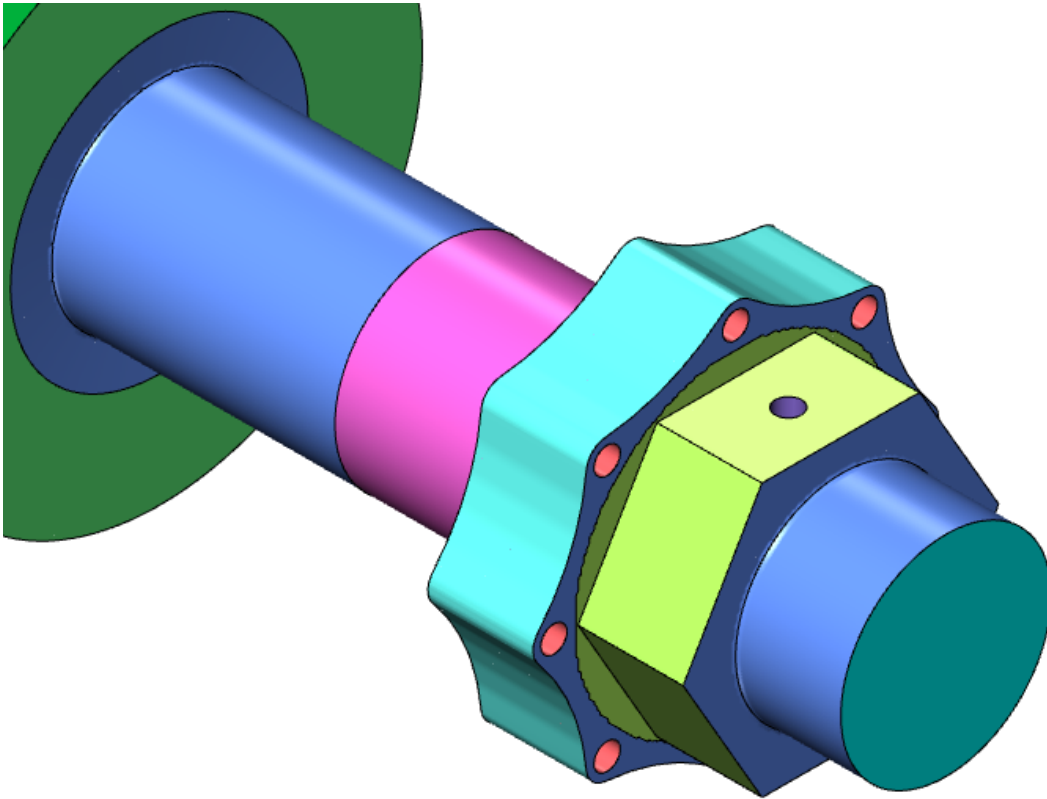
Note: Represents information about the process/step that is important or may require an explanation

- Bulleted text are step by step instructions that have to be followed.

TUTORIALS FILES:

The training files are mostly Mastercam files. These files are available for download at <http://www.e-mastercam.com/files/>.

Lathe C & Y Axis Part #1



INTRODUCTION:

This lesson will teach you how to create milling toolpaths in Lathe.

You can use the Lathe C-axis toolpaths that are included with Mastercam Lathe for the most common contour and drill applications. These are available to anyone with a Lathe license and do not require a separate Mill license. When you select one of these toolpaths, Mastercam automatically sets the Tplane and Cplane to the appropriate settings for the selected application. These toolpaths also support Y-axis rotation and axis substitution. Your machine definition must include the appropriate rotary axis components.

If your workstation is licensed for both Mastercam Mill and Mastercam Lathe, you can also create Mill toolpaths. Use Mill toolpaths to machine in planes or orientations not available with the standard Lathe C-axis toolpaths, or to create toolpaths other than contour or drill toolpaths—for example, pocket toolpaths. You can set the Tplane to any view available in your part, or use the CView utility as a guide. Choose Mill from the Machine tab, and select the desired toolpath. The toolpaths that are available to you depend on the level of Mill license and the capabilities of your machine definition.

OVERVIEW OF EXERCISE:

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. 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.

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 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.

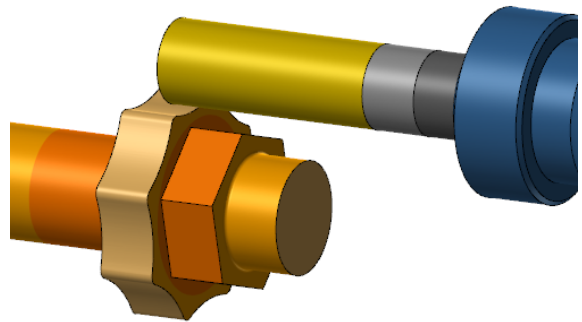
NEW CONCEPTS COVERED IN THIS LESSON:

- Planes Manager
- Face Contour
- Face Drill
- C-Axis Drill

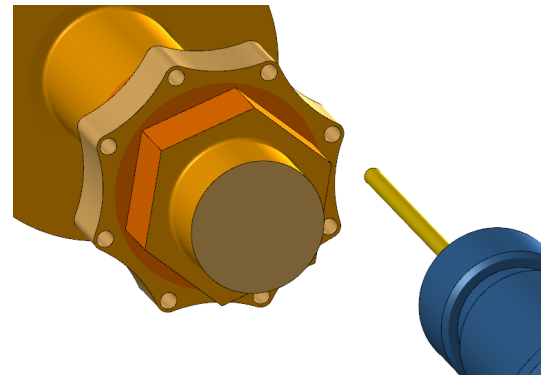
NEW TOOLPATH INFORMATION

The following is brief description of the new toolpath used in this exercise.

Face contour creates a toolpath on the face of the part with the tool parallel to the axis of rotation. Mastercam automatically sets the Tool plane (Tplane) and Construction plane (Cplane) to the side so that the tool is placed perpendicular to the face of the part.



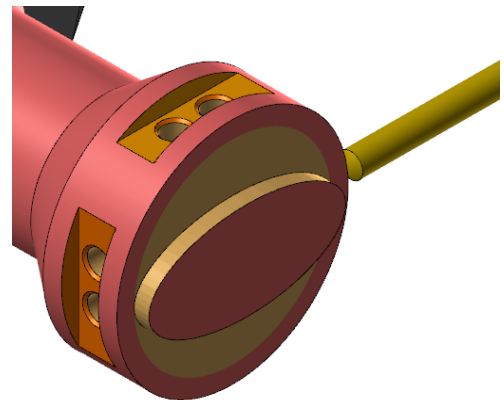
Face drill is used to drill holes into the face of the part. When selected, Mastercam sets the Tool plane (Tplane) and Construction plane (Cplane) to the side so that the tool is placed perpendicular to the face of the part.



C-axis drill is used to drill holes that are perpendicular to the axis of rotation, as when drilling holes in a cylinder. You can select all points that you want to drill at the same time without having to specify the tool plane for every hole. The points may lie in the Top construction plane, or be in their correct 3D positions around the Z axis.

If the points you have selected are in their correct 3D positions, you must check 'Unroll' in the rotary axis dialog (choose the Rotary Axis button in the C-axis drill dialog box to access this check box). When 'Unroll' is checked, Mastercam unrolls the points from the cylinder to the Top Cplane.

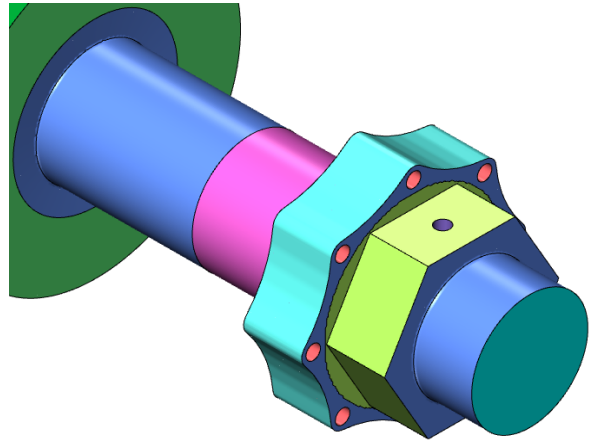
The C-axis toolpath is created by converting the X coordinate of each point in the top Cplane to an angle about the Z axis, using the rotary axis diameter entered in the rotary axis dialog box. The resulting toolpath is then displayed in its rolled state



INSTRUCTOR DEMONSTRATION:

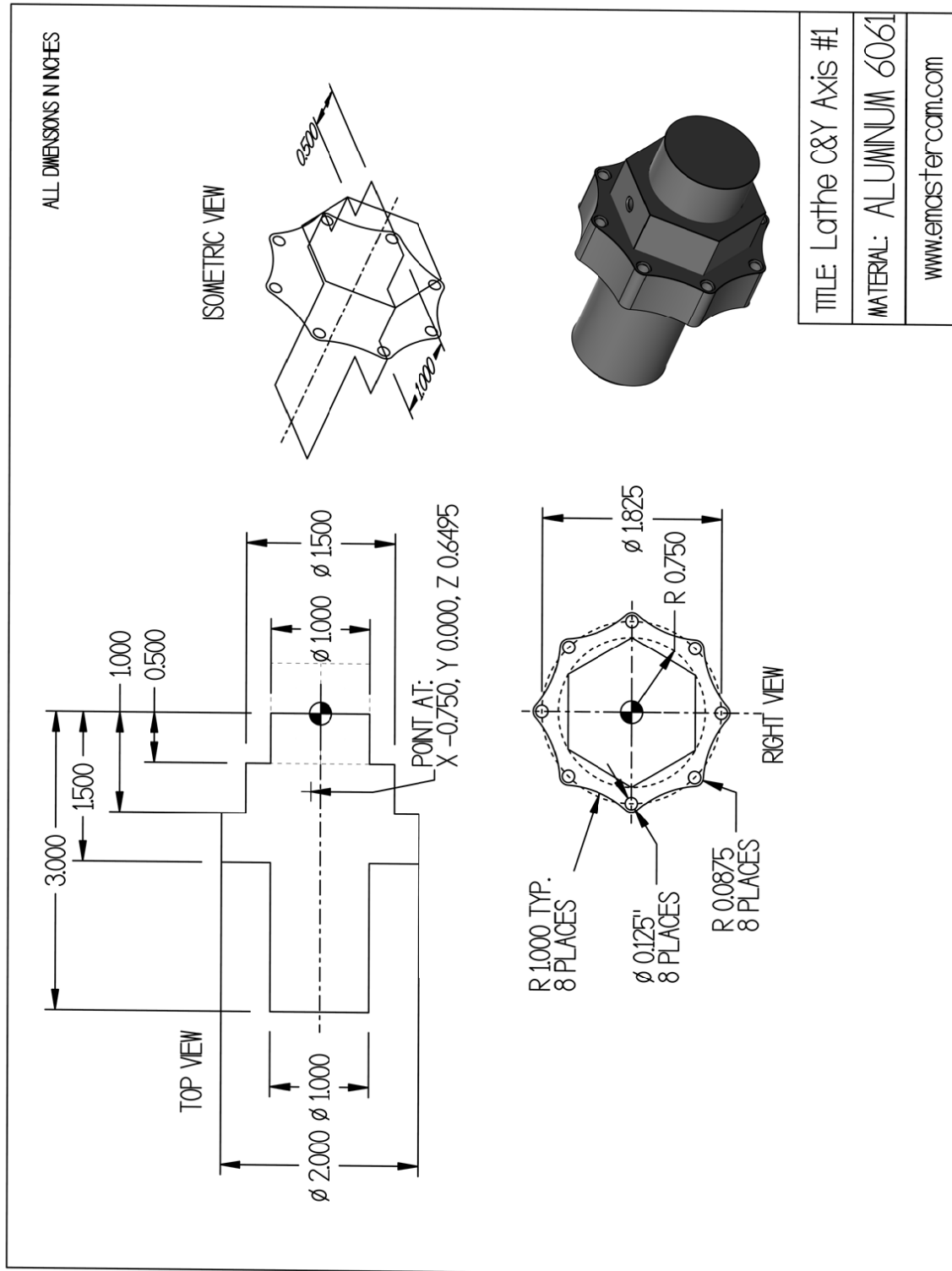
Topics:

- Create the 2D geometry in the Top plane on Level 1
- Create the 3D geometry using the Right plane and Translate command
- Generate the 2D toolpaths to machine the OD such as: Face, Rough, Finish and Groove
- General overview of Construction Planes/Graphic Views using the Status Bar and the Planes Manager.
- Generate the C-axis toolpaths such as: Face Contour, Face Drilling, and C-Axis Drilling used to machine the part from the Right plane.



NOTES:

GEOMETRY CREATION

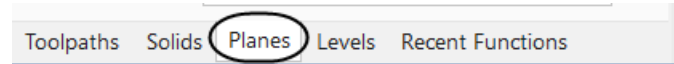


CREATE THE 2D GEOMETRY

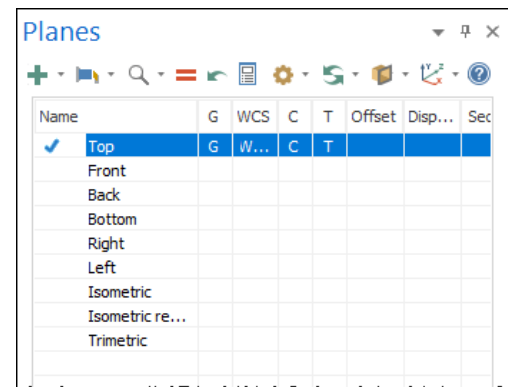
1. Setting Up The Graphical User Interface

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.



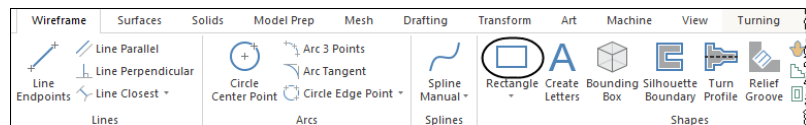
- Make sure that **WCS**, planes and the graphics view are all set to **TOP** as shown.



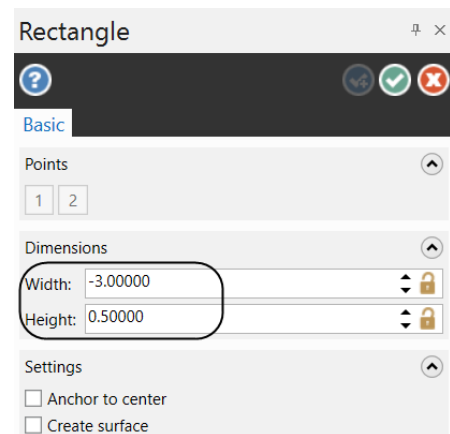
2. Create A Rectangle

Wireframe

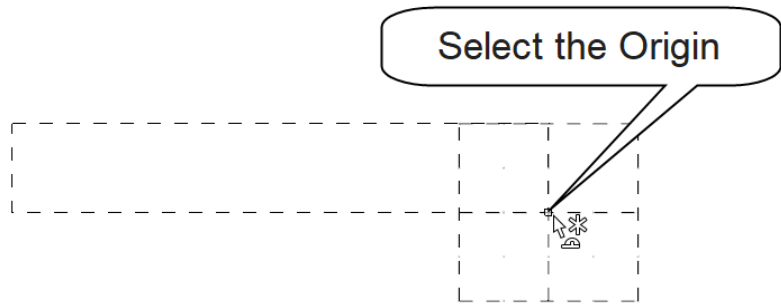
- From the **Shapes** group, select the **Rectangle** icon as shown.



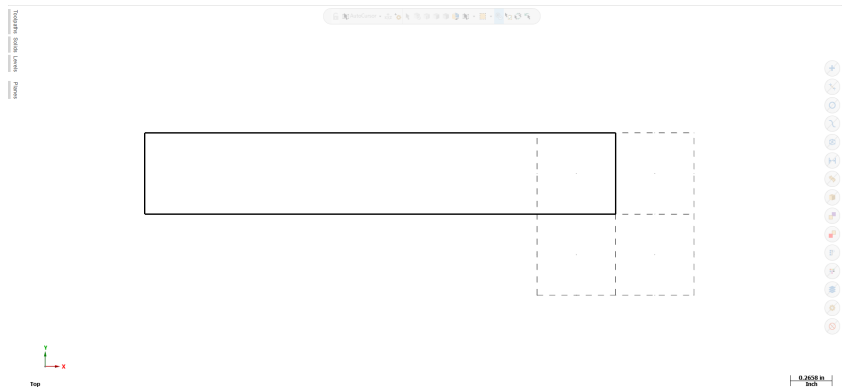
- Enter the **Width** and the **Height** and make sure that **Anchor to center** and **Create surface** buttons are not selected (highlighted) as shown.
- Press **Enter** after typing the values to see a preview of the rectangle.



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

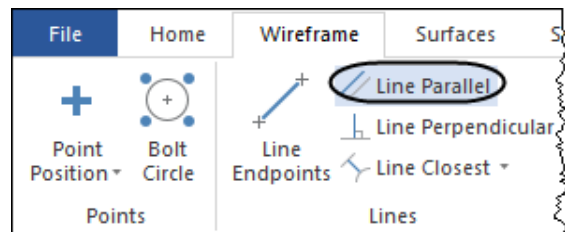


3. Create Parallel Lines

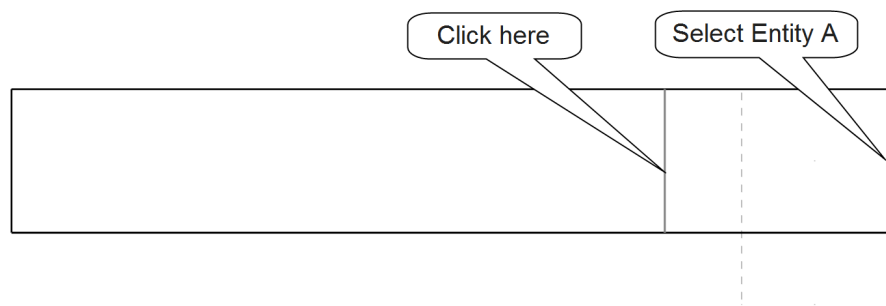
Create The Vertical Lines

Wireframe

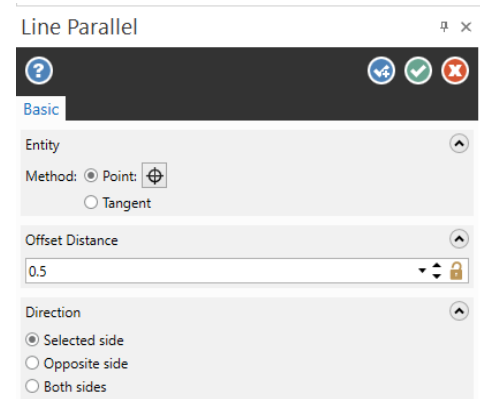
- From the **Lines** group, select the **Line Parallel** icon.



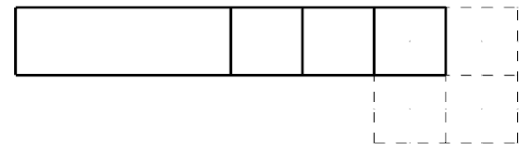
- [Select a line]: Select Entity A as shown.
- [Select the point to place a parallel line through]: Click on a point to the left of **Entity A** as shown.



- 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.

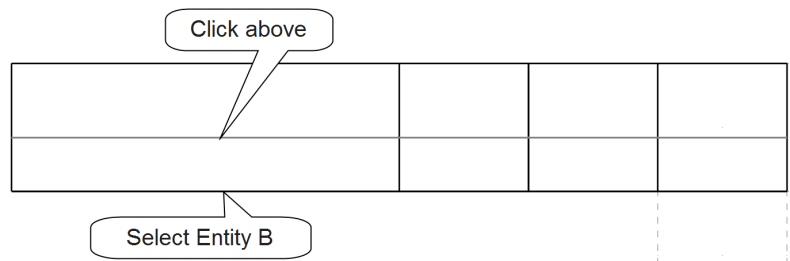



- [Select a line]: Select **Entity A** as shown before.
- [Select the point to place a parallel line through]: Click on a point to the left of **Entity A** as shown before.
- In the **Line Parallel** panel, enter the **Offset Distance 1.0** and press **Enter**.
- Select the **OK and Create New Operation**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

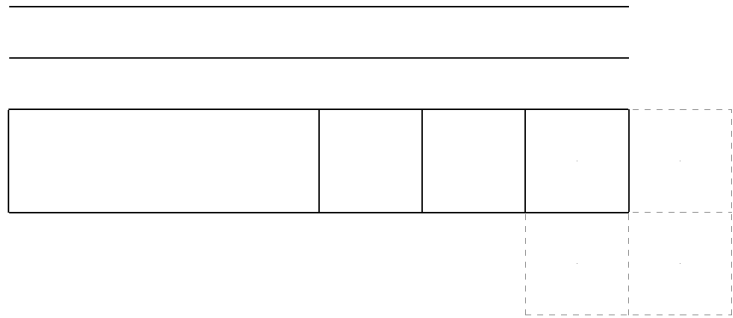


Create The Horizontal Lines

- [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**.
- Press Enter again 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.

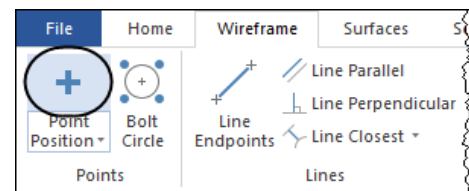


4. Create A Point

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.

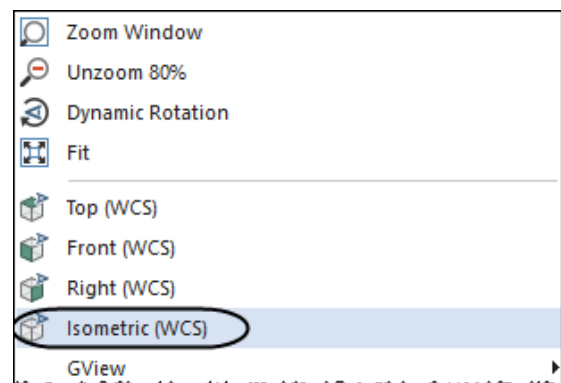
Wireframe

- 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.

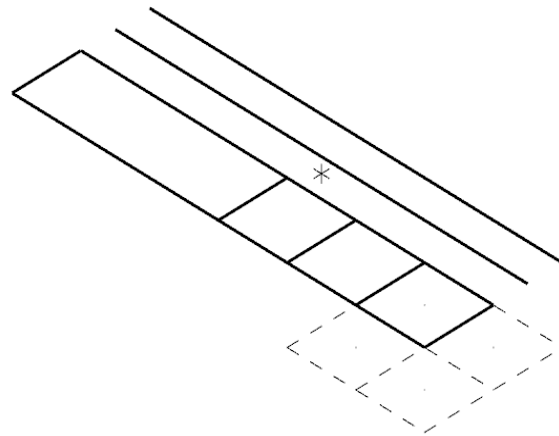


-0.75, 0.0, 0.6495

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



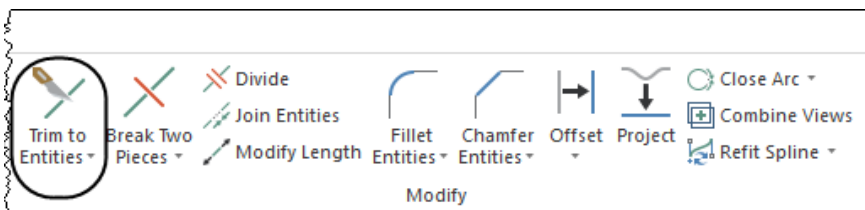
- The point should appear as shown.
- Right mouse click in the graphics window and select the **Top** view.



5. Trim The Geometry In The Top View

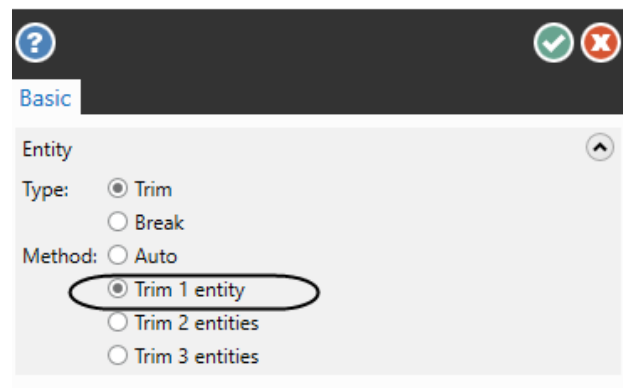
Wireframe

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

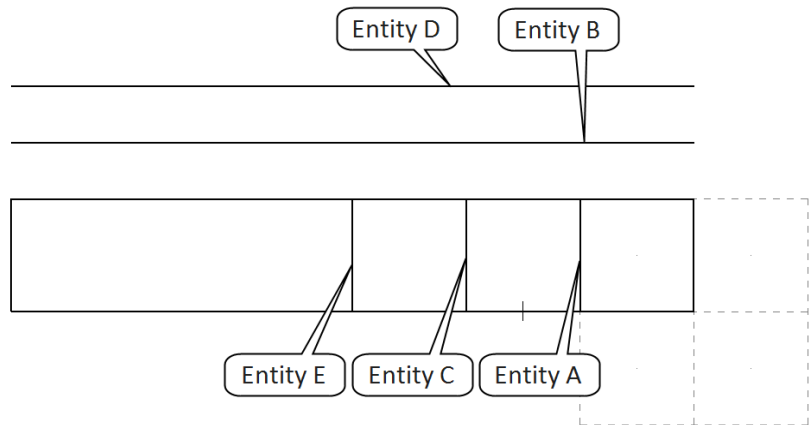


- In the **Trim to Entities** panel, enable **Trim 1 entity** as shown.

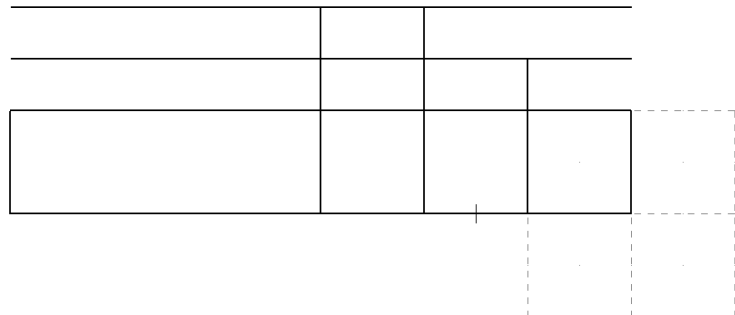
Trim to Entities



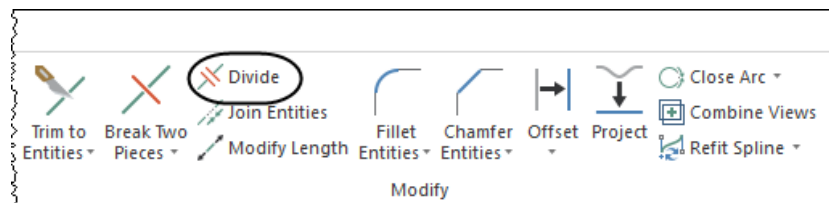
- [Select the entity to trim/extend]:
Select **Entity A** as shown.
- [Select the entity to trim/extend to]:
Select **Entity B** as shown.
- [Select the entity to trim/extend]:
Select **Entity C** as shown.
- [Select the entity to trim/extend to]:
Select **Entity D** as shown.
- [Select the entity to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



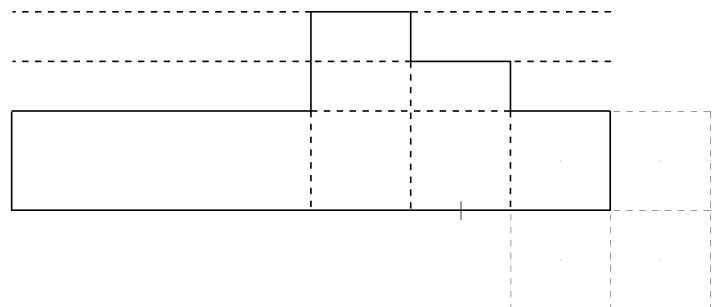
The part should appear as shown.



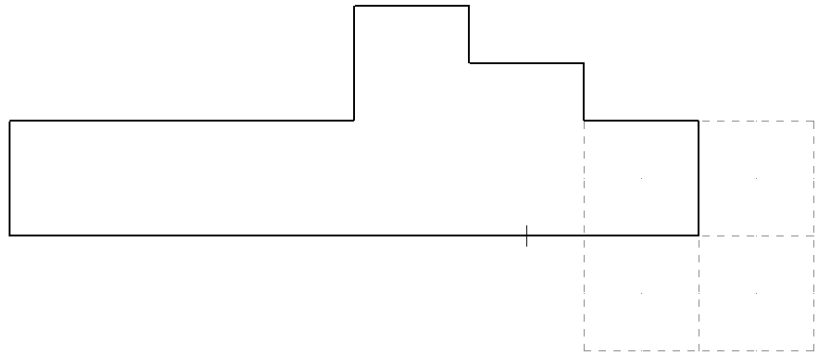
- From the **Modify** group, select the **Divide** icon as shown.



- 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.



CREATE THE 3D GEOMETRY

1. Set Levels Manager

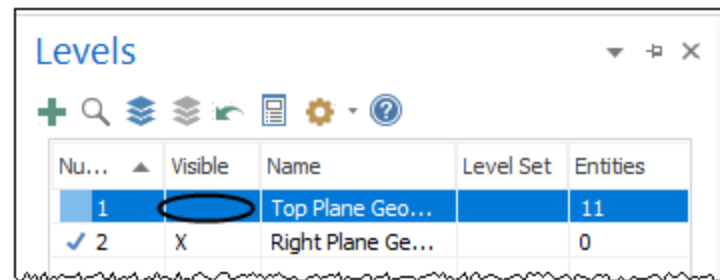
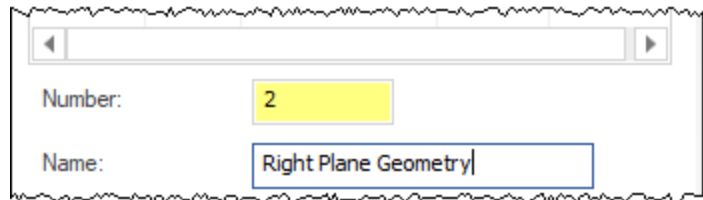
In this step, we will set the levels before we create the rest of the part to organize our geometry.

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

Toolpaths Solids Planes **Levels** Recent Functions

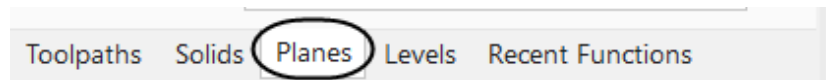
- Click in the **Name** area in the **Levels Manager** and enter in the name "Top Plane Geometry" as shown.

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

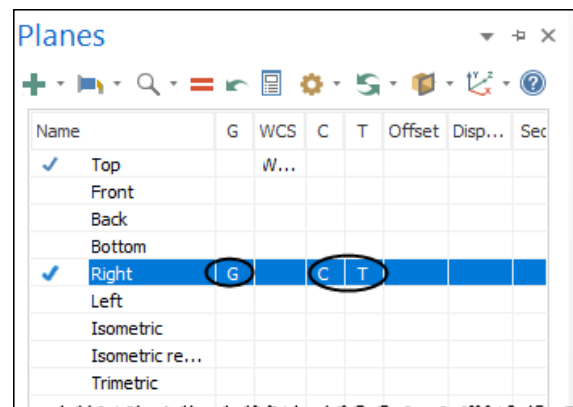


2. Change The Construction Plane To Right

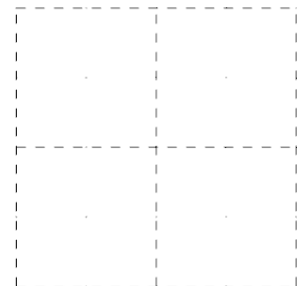
- Select the **Planes** tab to open the **Planes Manager** as shown.



- 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.

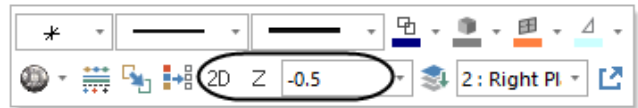


- Position the **Grid** (Hold down the mouse wheel and the **Shift** key to pan) so that you can see it on screen as shown.



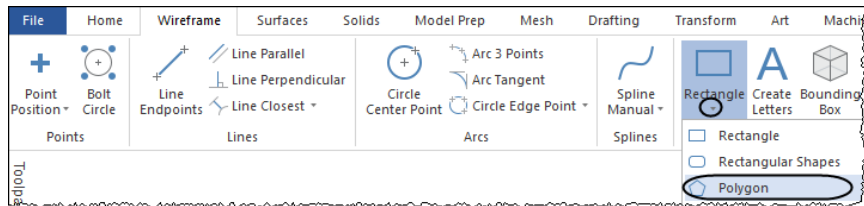
3. Create A Polygon

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



Wireframe

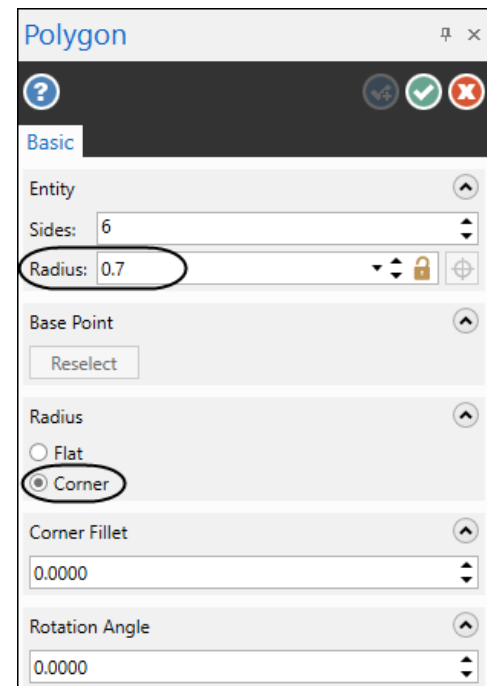
- From the **Shapes** group, click on the drop down arrow below the **Rectangle** and select the **Polygon** as shown.



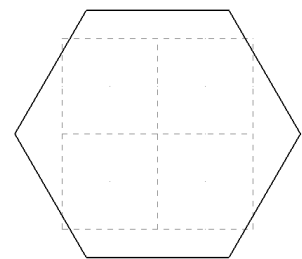
- [Select position of base point]: Select the **Origin** as shown.
- Select and click on the screen a point in the graphics window to sketch a polygon.



- In the **Polygon** panel enter in the **Radius** and enable **Corner** as shown.
- Select the **OK** button to exit the **Polygon** command.

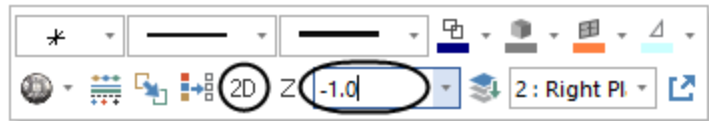


- Press **Alt + F1** to fit the geometry to the graphics window if needed.
- The part should appear as shown.



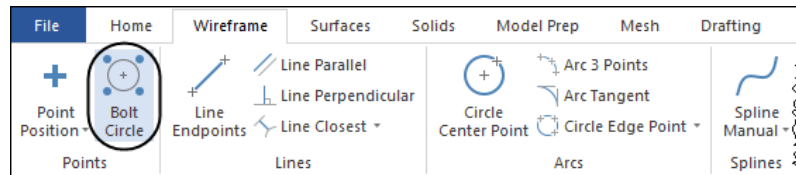
4. Create Bolt Circles

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

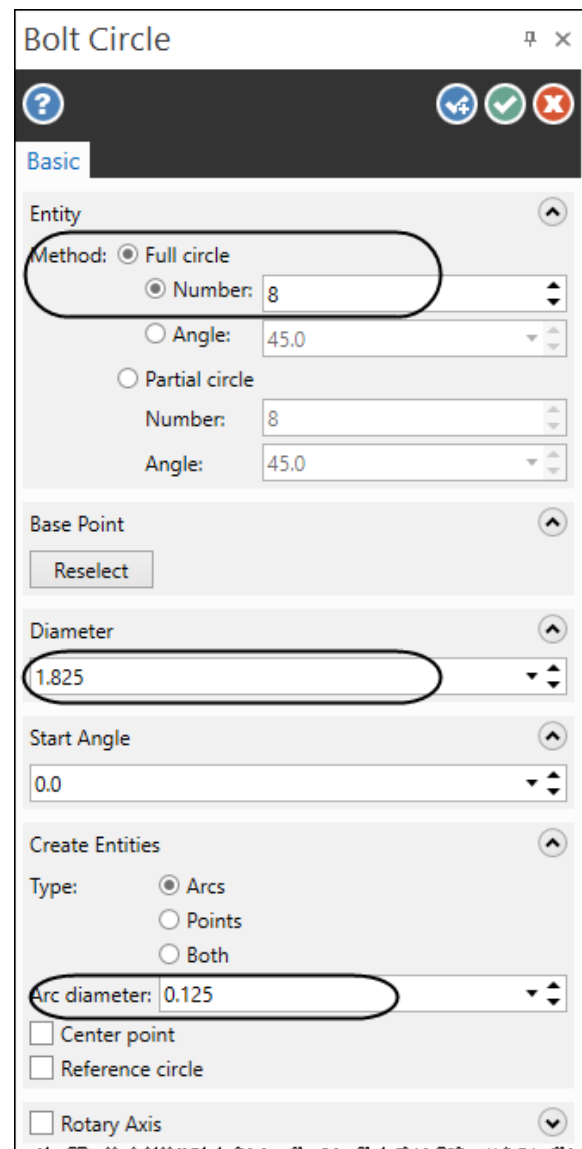


Wireframe

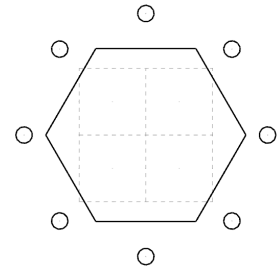
- From the **Points** group, select **Bolt Circle** as shown.
- [Select position of base point] Select the **Origin**.



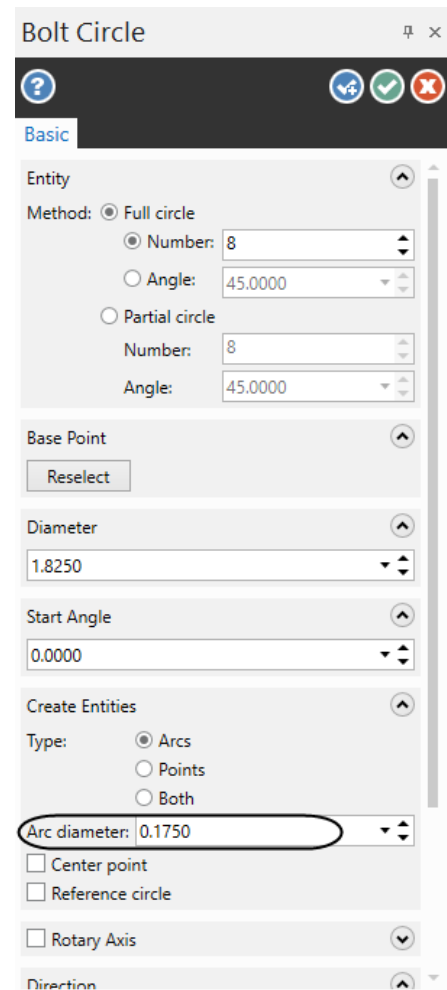
- The **Bolt Circle** panel will appear. Enable **Full Circle** and enter in values for **Diameter** of Guide Circle and **Number**. Enable **Arcs** and input an **Arc diameter** of 0.125" as shown.
- Select the **OK and Create New Operation** button to continue using the **Bolt Circle** command.



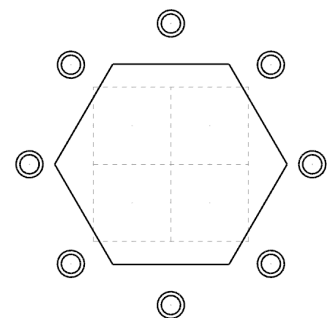
- Press **Alt + F1** to fit the geometry onto the graphics window.
- The part should appear as shown.



- [Select position of base point] Select the **Origin** as shown.
- Change the **Arc diameter** value to **0.175"** and keep the other settings in the **Bolt Circle** panel as shown.
- Select the **OK** button to exit the **Bolt Circle** command.



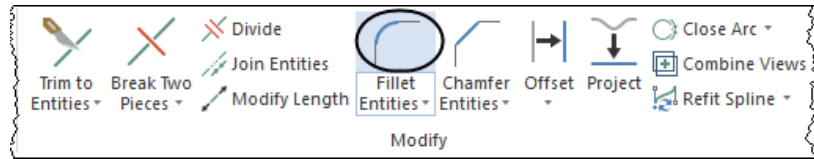
- Press **Alt + F1** to fit the geometry onto the graphics window.
- The part should appear as shown.



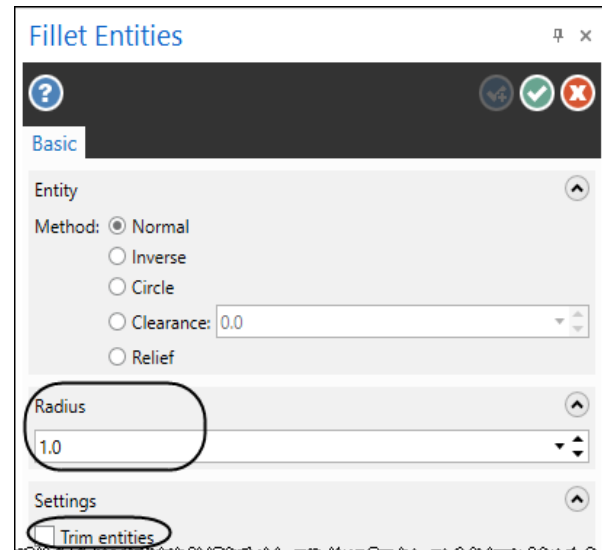
5. Create A Fillet Knowing The Radius And Endpoints

Wireframe

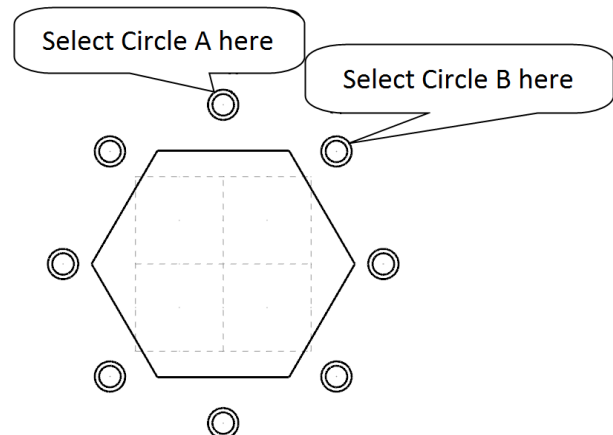
- From the **Modify** group, select the **Fillet Entities** as shown.



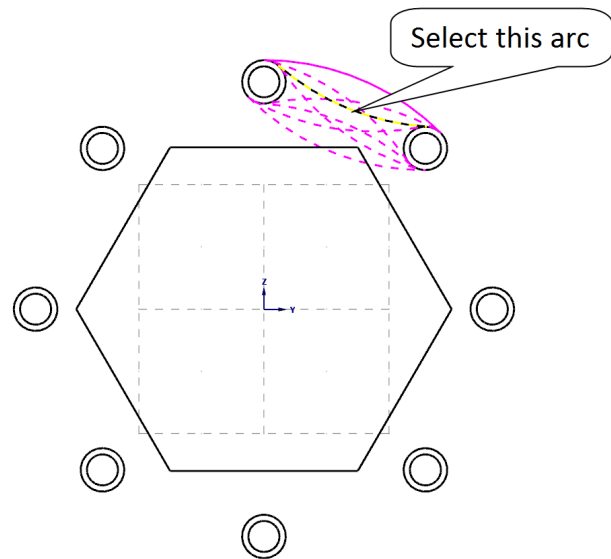
- In the **Fillet Entities** panel, enter the **Radius** and make sure **Normal** is enabled and **Trim entities** is disabled as shown.



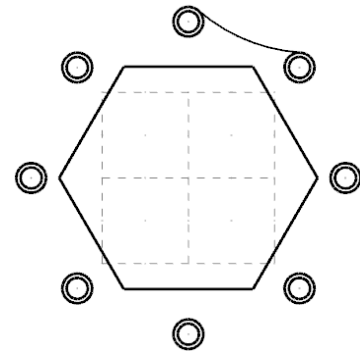
- [Fillet: Select an entity]: Select **Circle A** as shown.
- [Fillet: Select another entity]: Select **Circle B** as shown.



- [Select an arc] Select the arc as shown.
- Select the **OK** button to exit the command.



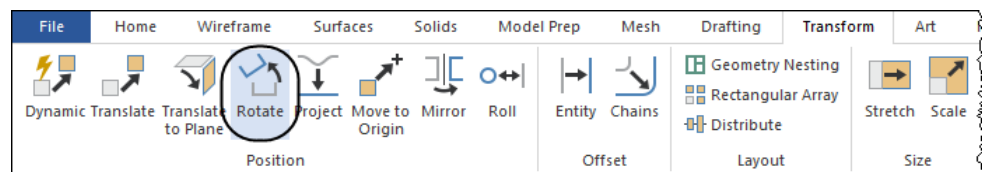
- The arc should appear as shown.



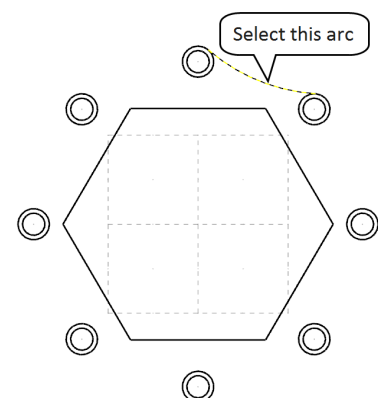
6. Rotate The Arc

Transform

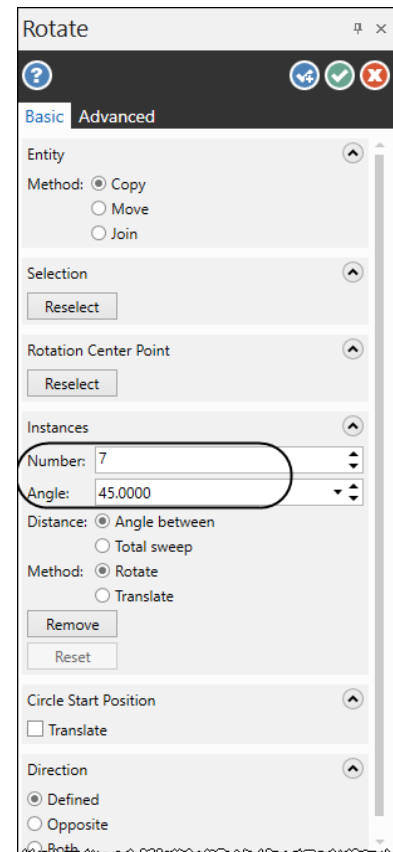
- From the **Position** group, select the **Rotate** icon.



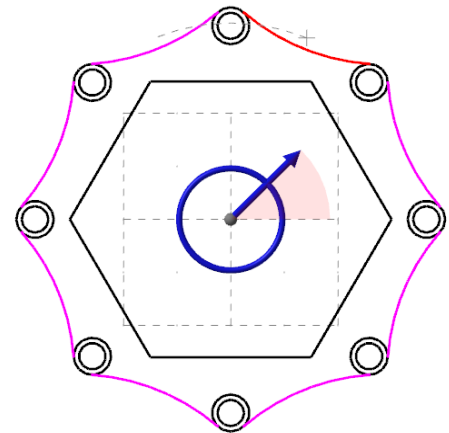
- [Rotate: select entities to rotate]: Select the **Arc** as shown.
- Press **Enter** to complete selection.



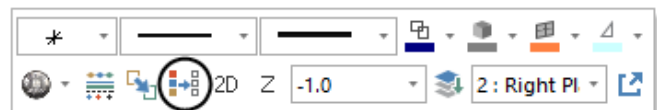
- The **Rotate** panel will appear. Match the parameters as shown and press **Enter**.



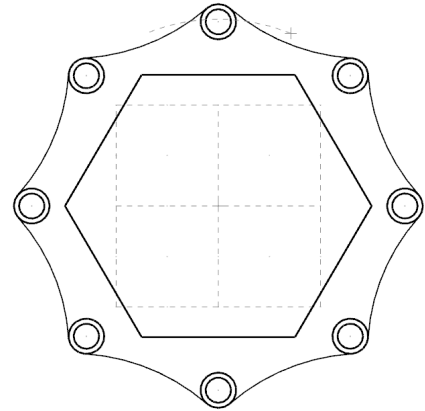
- A preview of the result will be shown. Confirm this is the result you want as shown.
- Click on the **OK** button to exit the command.



- Right mouse click in the graphics window and select the **Clear Colors** icon to return the geometry to its original color.



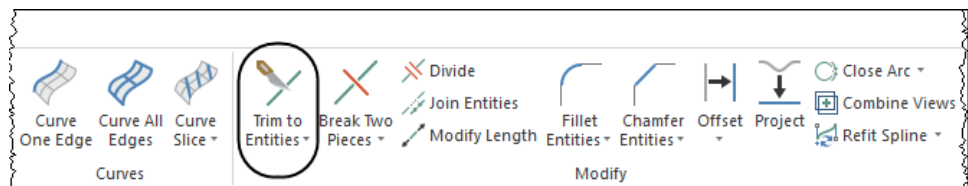
- The part should appear as shown.



7. Trim the Circles

Wireframe

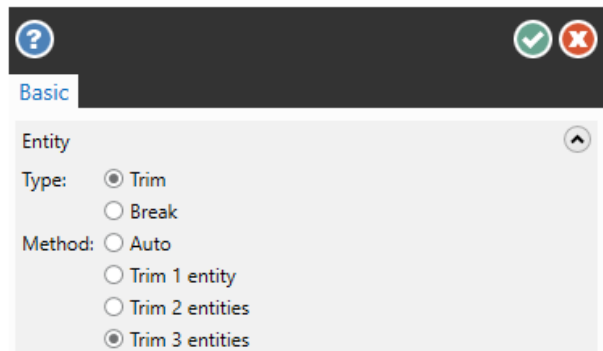
- From the **Modify** group, select the **Trim to Entities** as shown.



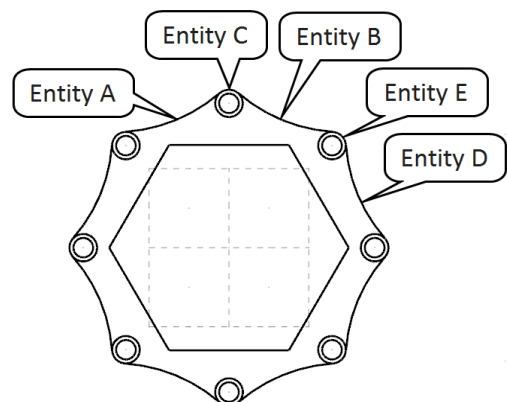
- In the **Trim to Entities** panel, enable **Trim 3 entities** as shown.

Note: It is very important to select the arcs in the order and the positions shown. The first two entities that you select are trimmed to the third, which acts as a trimming curve. The third entity is then trimmed to the first two.

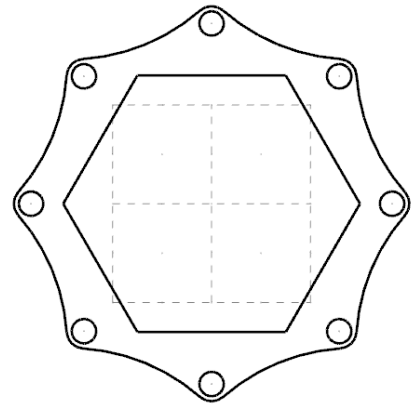
Trim to Entities



- [Select the first entity to trim/extend]: Select **Entity A** as shown.
- [Select the second entity to trim/extend]: Select **Entity B** as shown.
- [Select the entity to trim/extend to]: Select **Entity C** as shown.
- [Select the first entity to trim/extend]: Select **Entity B** as shown.
- [Select the second entity to trim/extend]: Select **Entity D** as shown.
- [Select the entity to trim/extend to]: Select **Entity E** as shown.



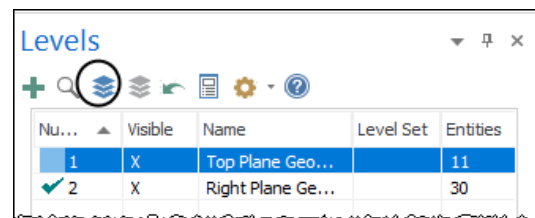
- Trim the other arcs using the same order shown previously.
- Select the **OK** button to exit the command.
- After the trimming is completed, the part should look shown.



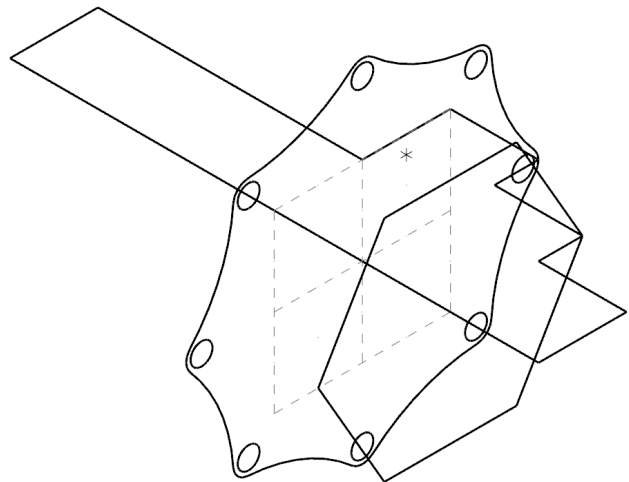
- Select the **Levels** tab as shown.



- In the **Levels Manager** panel, select the **Turn all levels on** icon to see the entire geometry.



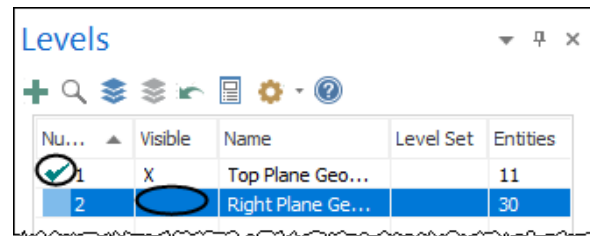
- Right mouse click in the graphics window and select the **Isometric** view.
- Press **Alt + F1** to fit the geometry into the graphics window.
- The part should appear as shown.



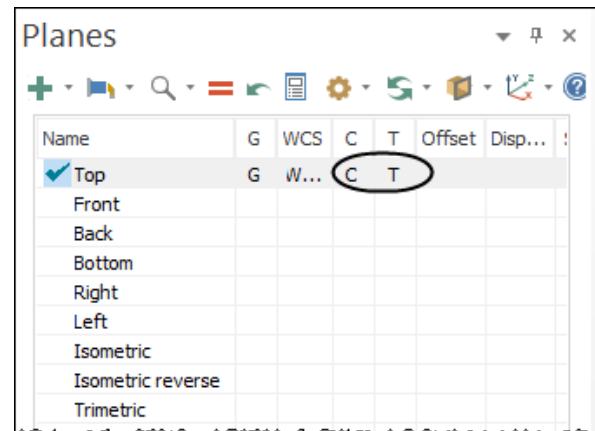
- Right mouse click in the graphics window and select the **Top** view as shown.

Note: We will only make **Level 1** visible right now to generate the Roughing, Finishing and Grooving toolpaths before creating the **C-Axis** toolpaths.

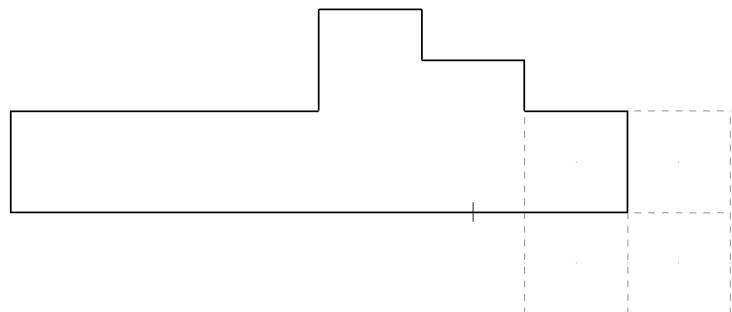
- In the **Levels Manager** panel, click on **1** in the **Number** column to make **Level 1** the **Main level**.
- Click in the **Visible** column next to number **2** to remove the **X** and make **Level 2** invisible as shown.



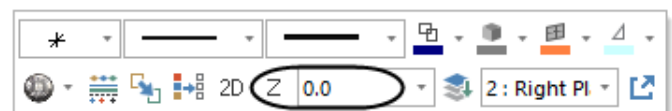
- Open **Planes Manager** and make sure the **Cplane** and **Tplane** are set to **Top** as shown.



- The part should appear as shown.



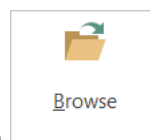
- Right mouse click in the graphics window and change the **Z** value to **0** as shown.



8. Save The File

File

- **Save As.**



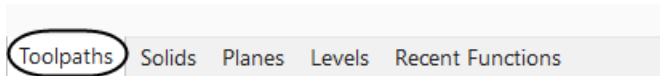
- Click on the **Browse** icon as shown.
- Find a location on the computer to save your file. File name: "LatheCY_1".

STEP 1: 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 tool's capabilities and features and acts like a template for setting up machining jobs. The machine definition ties together three main components: the schematic model of your machine tool's components, the control definition that models your control unit's capabilities, and the post processor that will generate the required machine code (G-code). For a Lathe C-Axis exercise, we need just a basic machine definition.

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

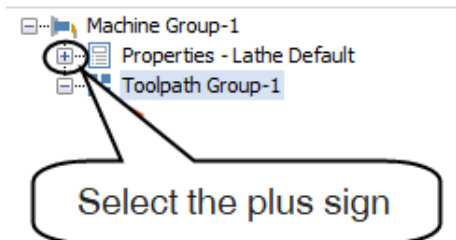
- To display the **Toolpaths Manager** panel, select **Toolpaths** tab as shown.



Machine

- In the **Machine Type**, click on the drop down arrow below **Lathe** and select **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.

