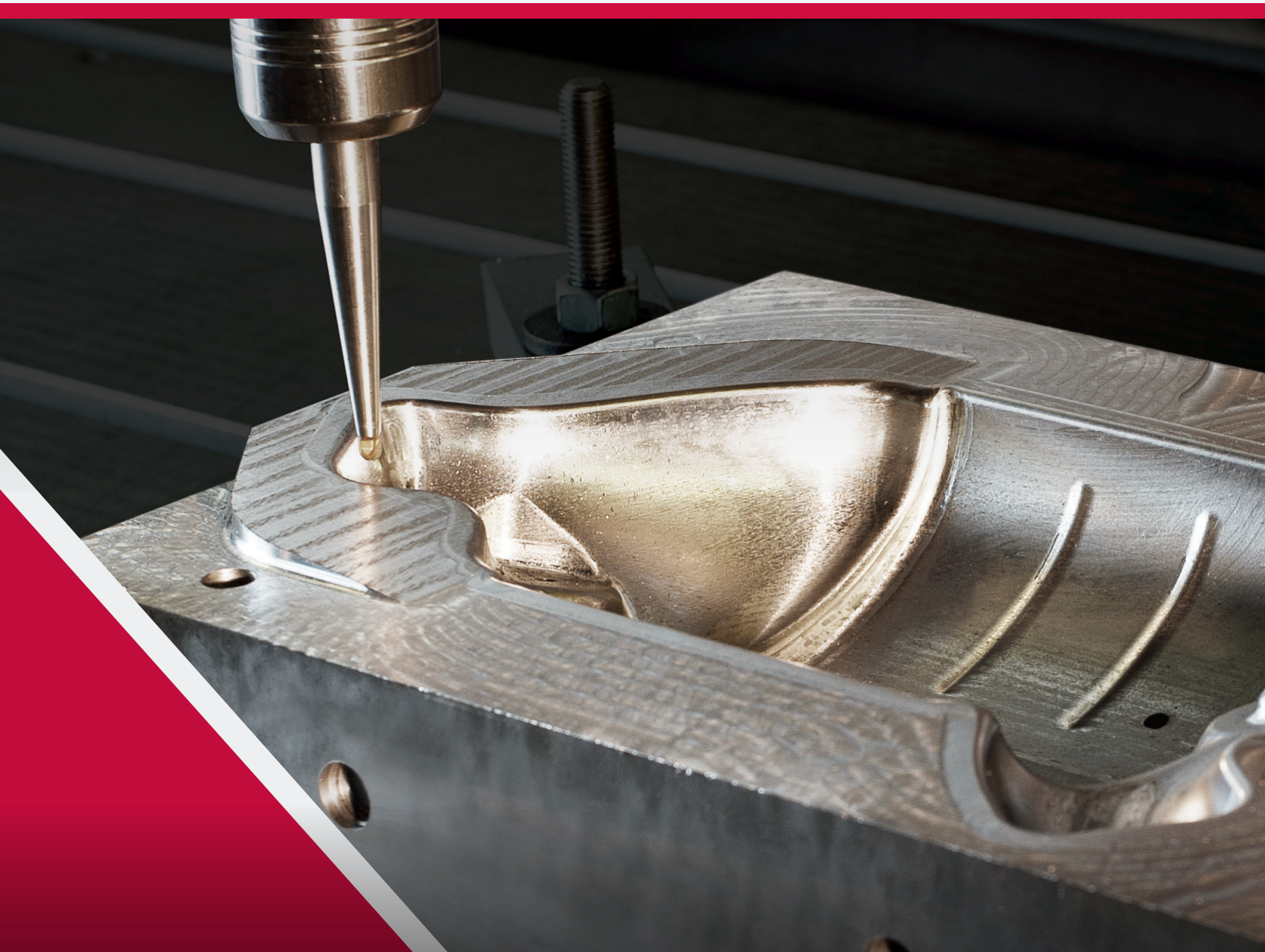
Mastercam® 2022

VIDEO ECOURSE PROMO CODE INCLUDED



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Mastercam 2022 Mill Advanced Training Tutorial

Software: Mastercam 2022

Authors: Mariana Lendel

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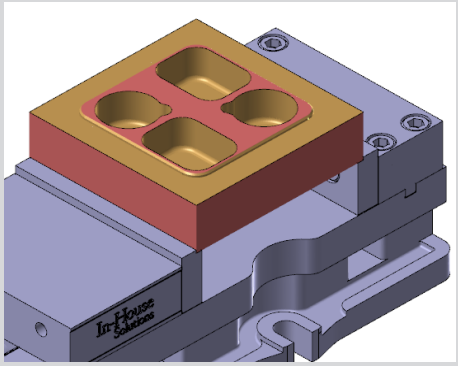
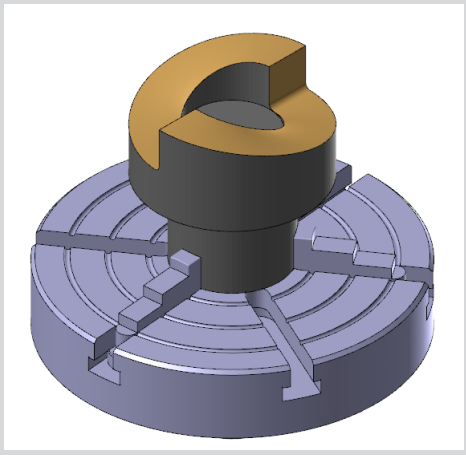
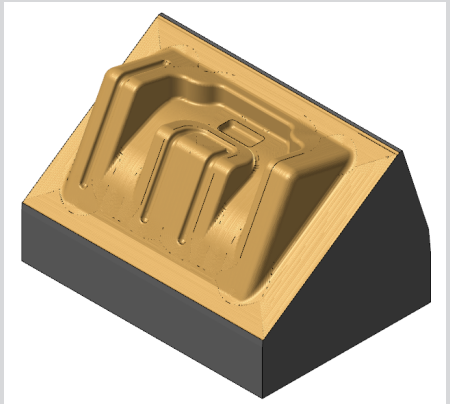
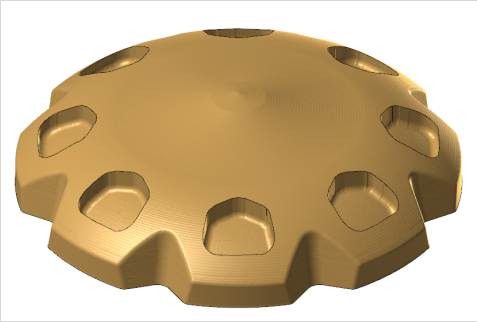
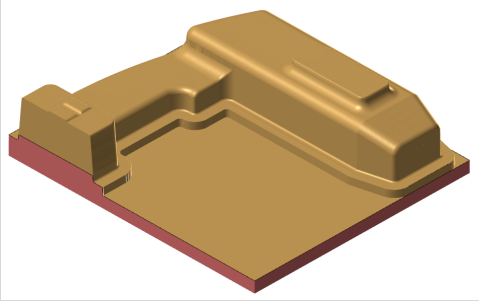
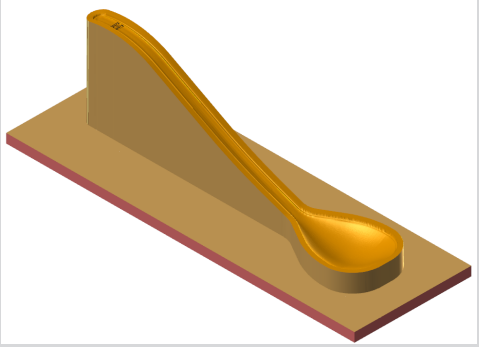
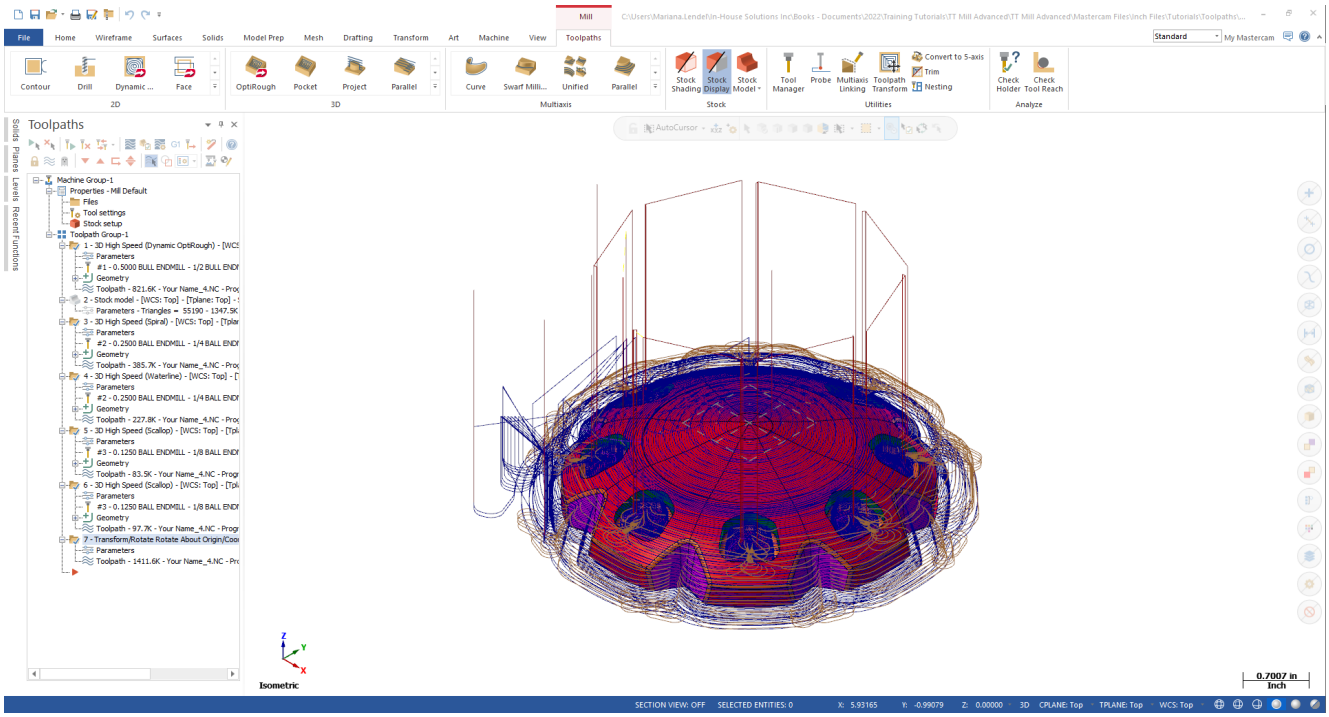
#1 	Solid Extrude Create Body Solid Extrude Cut Body Solid Fillet Solid Chamfer	High Speed Area Roughing High Speed Horizontal Compare Toolpath to Workpiece High Speed Waterline
#2 	Swept Surface Solid Extrude Solid Trim To Surface Solid Boolean Add Use Levels	Stock Model High Speed Area Roughing - Rest Material Stock Model After Roughing High Speed Radial High Speed Surface Finish Blend Transform - Rotate By Coordinate
#3 	Add History Push-Pull Bounding Box Silhouette Boundary Solid Extrude Curve All Edges Levels	High Speed Surface Dynamic OptiRough Stock Model High Speed Surface Dynamic OptiRough with Rest Material Stock Model High Speed Surface Equal Scallop Edit Projection Edit Tool

Tutorial	Geometry Functions	Toolpath Creation
#4 	Revolved Surface Project Curve Onto Surface Ruled/Loft Surface Curve At Intersection Trim Surface To Curves Surface Fillet Draft Surface Fillet Blend Surface	High Speed Surface OptiRough Stock Model High Speed Surface Spiral High Speed Surface Waterline High Speed Surface Scallop Transform- Rotate
#5 	Mold Cavity Solid Extrude Solid Draft to Face Solid Constant Radius Fillet Solid Shell Mold Core Curves All Edges Solid Impression	2D High Speed Dynamic Surface High Speed OptiRough Surface High Speed Hybrid Surface High Speed Pencil 2D High Speed Contour Dynamic
#6 	Net Surface Loft Surface Flat Boundary Surface	2D High Speed Dynamic Surface High Speed OptiRough Surface High Speed Scallop Surface High Speed Pencil Surface High Speed Project

Getting Started



OBJECTIVES

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

STEP 1: STARTING MASTERCAM

1.1 For Windows 7

- Select the **Start** button.
- Select **All Programs** and click on Mastercam 2022.

1.2 For Windows 8

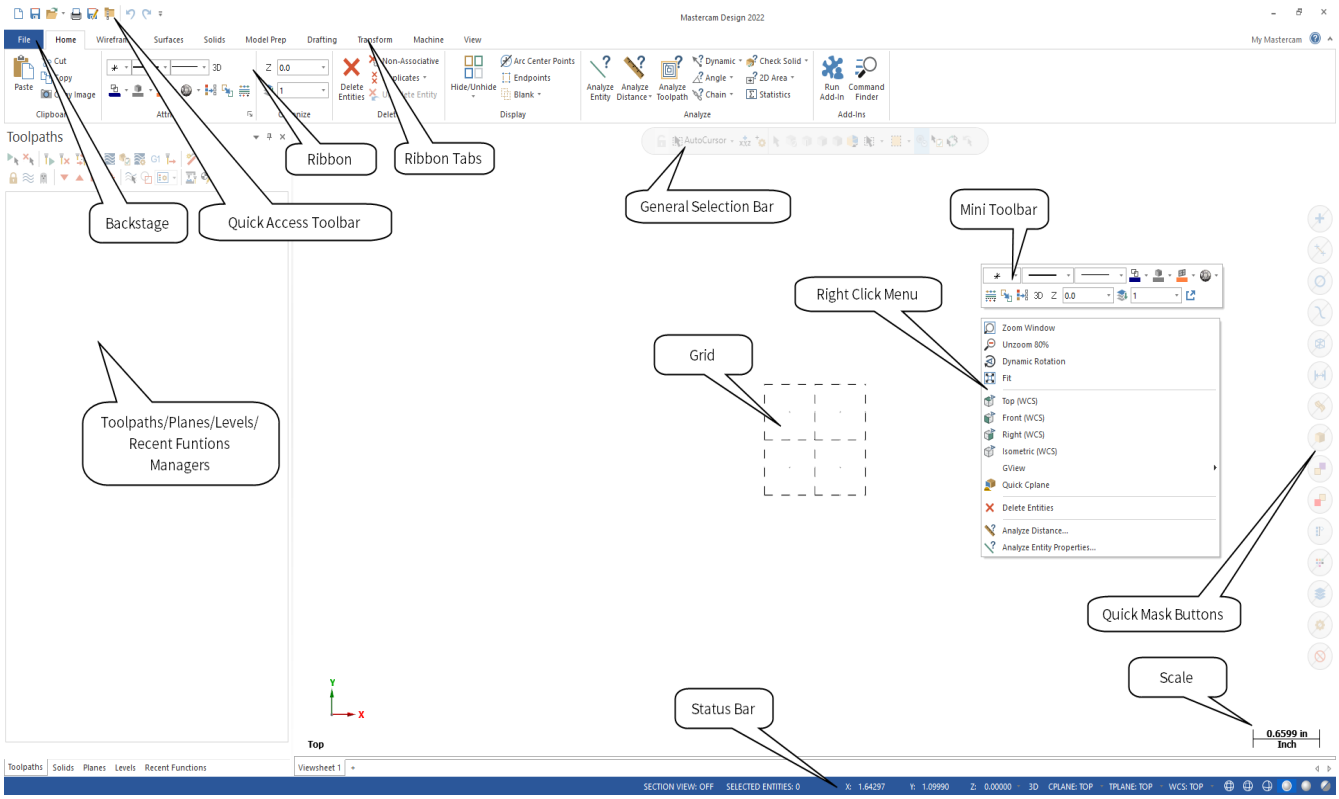
- Select the **Start** button.
- Click on the drop down arrow to open Apps.
- Find and click on Mastercam 2022.

1.3 For Windows 10

- Select the **Start** button.
- Click on the drop down arrow to open Apps.
- Find and click on Mastercam 2022.

- To start the software, from Desktop, click on the shortcut icon as shown.

STEP 2: GUI - GRAPHICAL USER INTERFACE



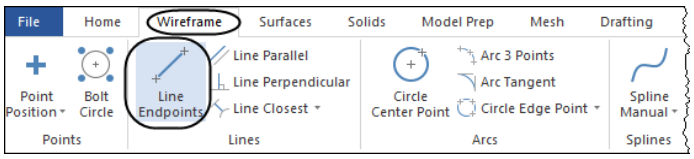
Quick Access Toolbar	**QAT** contains a fully customizable set of functions that can be quickly accessed by the user.
Backstage (File)	Allows you to manage files. You can insert information about files, start a new file, open an existing one or merge files together. You can also save, convert or print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

STEP 3: NAVIGATE THROUGH MASTERCAM

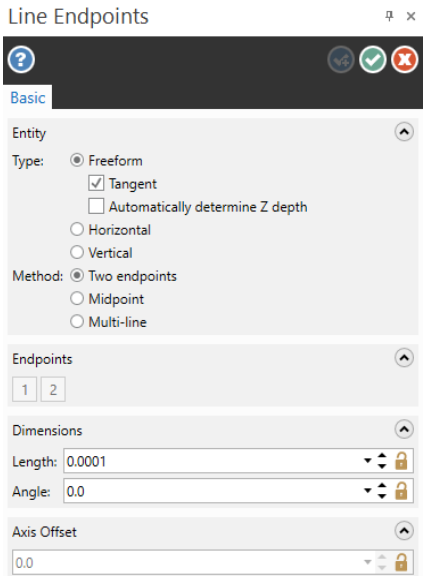
In this step, you will learn how to use the menu functions in Mastercam to create geometry.

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

- Left click on **Wireframe**.
- Left click on the **Line Endpoints** icon as shown.



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



Function Prompt

Prompts the user to execute a command.

Sketching a line

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

Creating a line knowing the endpoint coordinates

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

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

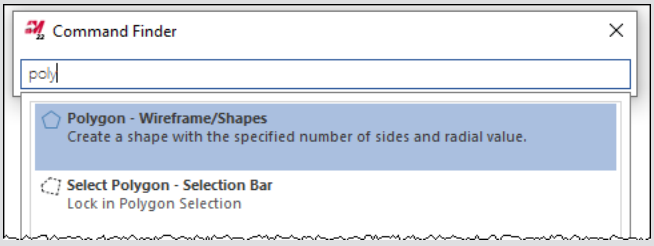
- Press **Enter** to continue.
- Select the **AutoCursor Fast Point** icon again and enter in the coordinates of the second endpoint and then press **Enter**.

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

- You can also enter the coordinates of the first endpoint, then enter the **Length** and **Angle** if necessary.
- To continue making lines, choose the **OK and Create New Operation** button from the dialog box or press **Enter**.
- To exit the current command, select the **OK** button or press the **Esc** button.
- To undo the last command, from the **QAT** (**Q**uick **A**ccess **T**oolbar) select the **Undo** button. The Undo button can be used to go back to the beginning of geometry creation or to the last point of the saved file. Mastercam also has a **Redo** button for your convenience.

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

Note: To find a command, from the **Home** ribbon, select the **Command Finder** icon and type the function name in the field that opens up. For example, to find the **Polygon** command type "polygon" in the text field. From the list, select the desired command.



STEP 4: SET THE ATTRIBUTES

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

4.1 Attributes Group

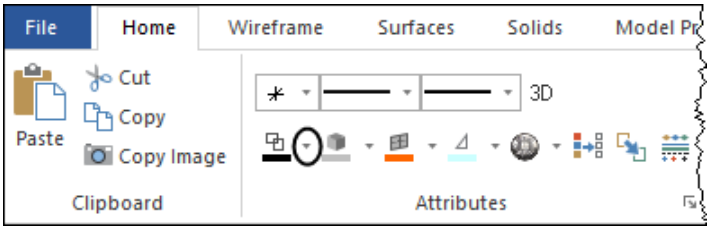
Point Style	Displays and sets the system's point style.
Line Style	Displays and sets the system's line style.
Line Width	Displays and sets the current system's line width.
Color	Assigns the current color to wireframe, solid and surface entities. To change the current color, click in the specific color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

4.2 Organize Group

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

Change The Wireframe Color

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



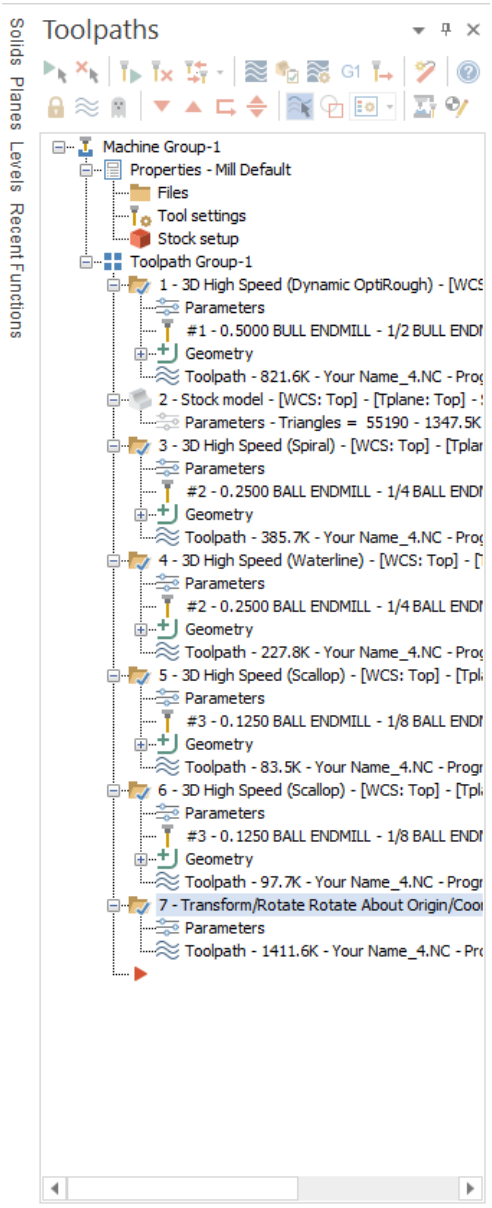
- Select the desired color from the dialog box as shown.

Note: Any geometry on your screen will remain in the previous system color. This change will only affect the geometry you create going forward.
To change the color of existing geometry, select the entities first and then click on the drop down arrow next to the Wireframe Color and select the desired color. The same method can be applied for any other attribute that you want to set or change.

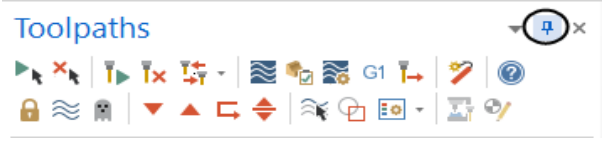
STEP 5: MANAGER PANELS

5.1 The Toolpaths Manager

The **Toolpaths Manager** displays all the operations for the current part. You can sort, edit, regenerate, verify and post any operation as shown. For more information on the **Toolpaths Manager**, please refer to the **Help** icon.

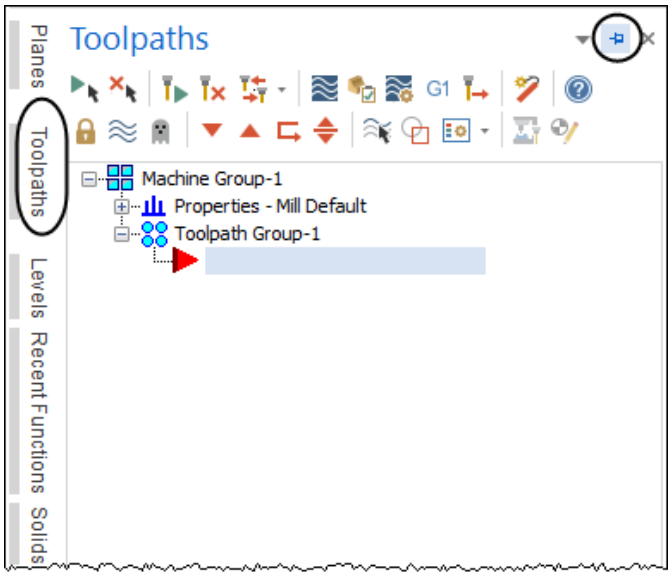


- The **Toolpaths Manager**, **Solids Manager**, or **Planes Manager** can be hidden to gain more space in the graphics area for creating geometry. Use **Auto Hide** icon to close all **Toolpaths**, **Solids**, **Planes** and **Levels Manager** panels.



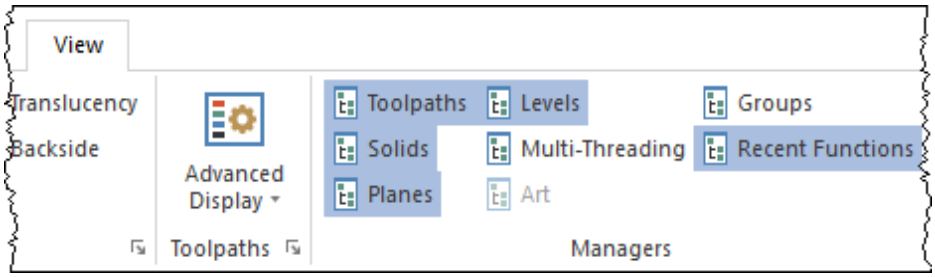
- The panels will be hidden to the left of the graphics window as shown.

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



- Selecting the **X** (**Close** icon) instead of the **Auto Hide**, you will close the manager panel.

- To re-open them, from the **View** tab, select **Toolpaths**, **Solids**, **Planes** or **Levels** as shown.



STEP 6: SETTING MASTERCAM TO IMPERIAL

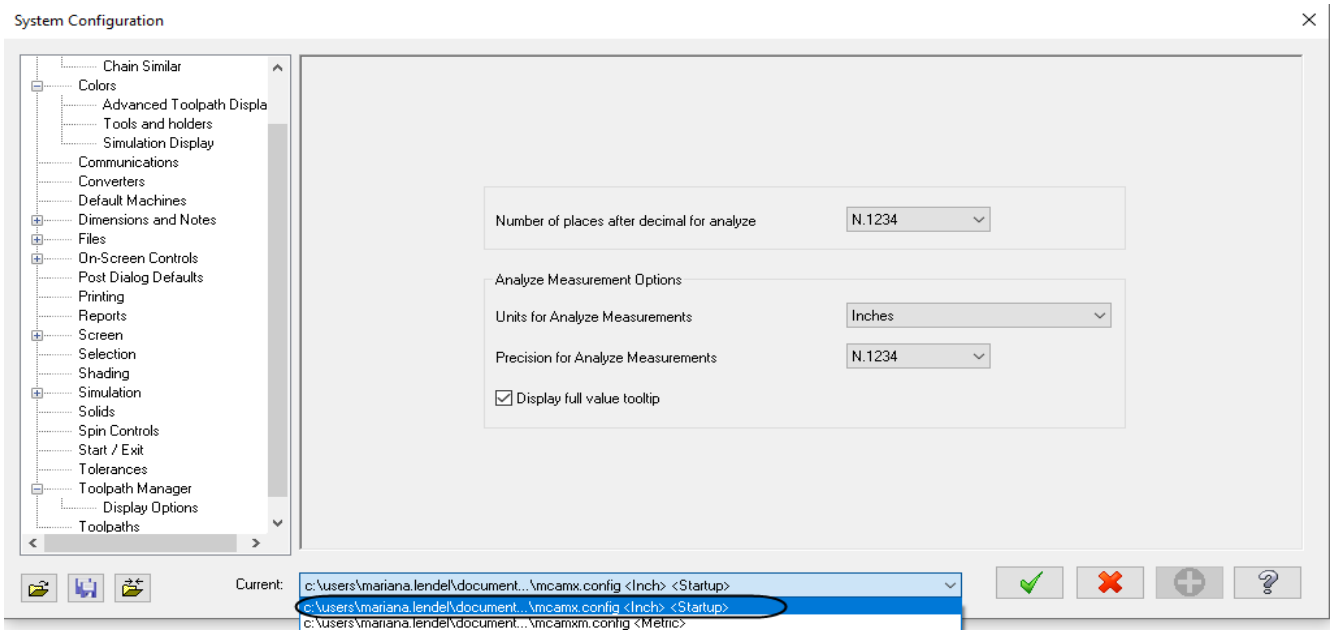
In this step you will learn how to set the imperial system as your default. You will have to select the Backstage options and select the system configuration.

6.1 Setting Mastercam to Imperial for the current session only

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

File

- **Configuration.**
  - Select the drop down arrow beside **Current** as shown.
  - Select **mcamx.config <Inch>** as shown in.



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

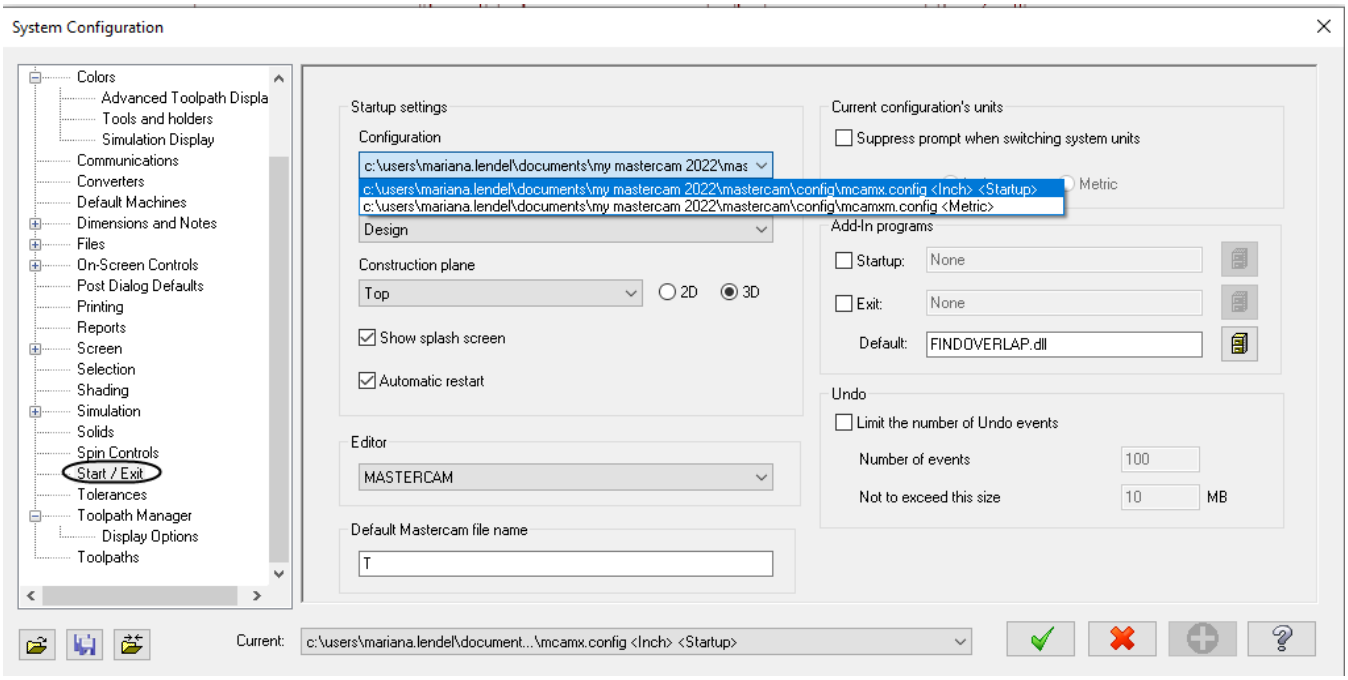
Note: If you have a drawing on the screen it may ask you to scale the current part to Imperial. Choose Yes if you wish to do this.

6.2 Setting Mastercam to imperial as a default

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

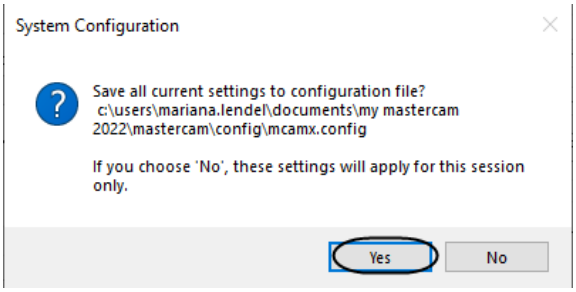
File

- **Configuration.**
  - Select **Start/Exit** from the configuration topics.
  - Select the drop down arrow below **Configuration** in the **Startup settings** area as shown
  - Select **mcamx.config <Inch>** as shown.



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

- Mastercam will then prompt you to save these settings to your current configuration file, select **Yes**.



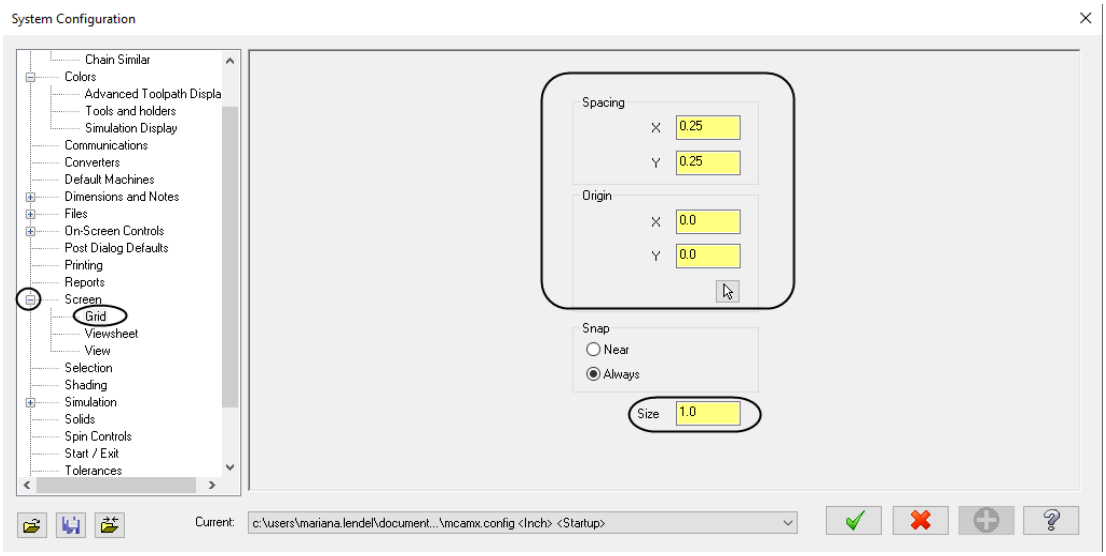
STEP 7: SET THE GRID

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

File

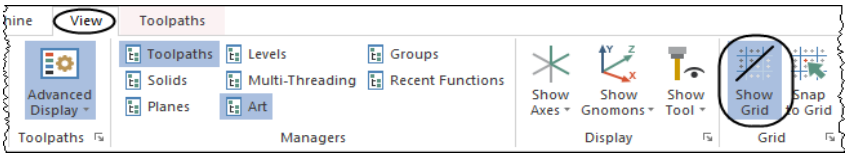
- **Configuration.**
  - Select **Screen** from the configuration **Topics**.

- Select the plus sign (**+**) beside **Screen** as shown.



- In **Grid Setting**s, change the **Spacing** to **X = 0.25** and **Y = 0.25**.
- Set the **Size** to **1.0**.
- Choose the **OK** button to exit. 
- Select the **Yes** button to save the settings in the **System Configuration**.

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



- The grid should look as shown.

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 following:

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

STEP 8: STEP TITLES

8.1 Sub step titles

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

bold text - Represents menu commands, dialog box settings or other similar items from the screen.

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

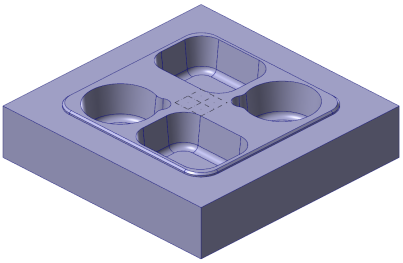
- Bulleted text is step by step instructions that have to be followed.

The files used in this book are available for download at http://www.emastercam.com/files/.

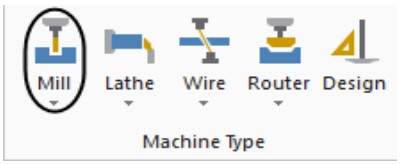
MASTERCAM WORK FLOW

The process to create or import the geometry and 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:

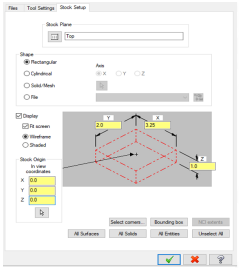
1. Create or import the part geometry.

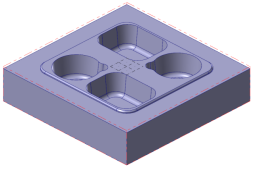


2. Select the Machine type.

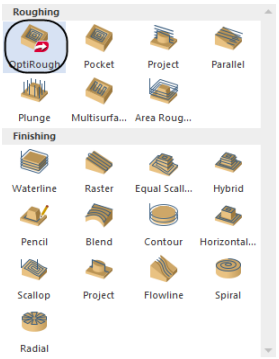


3. Define the stock size that your part will be cut from and set tool information.

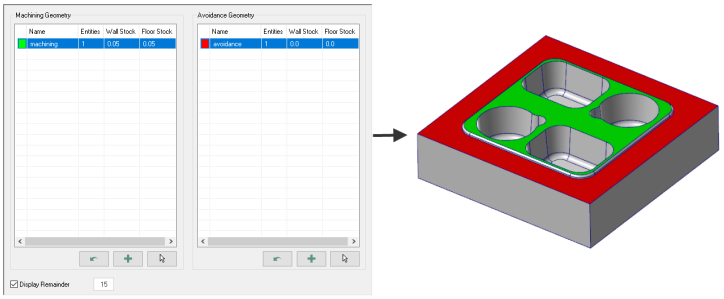




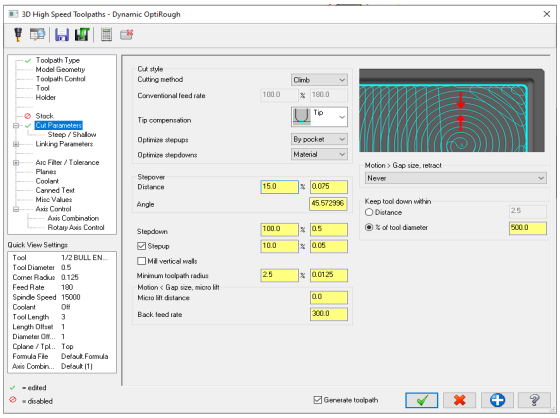
4. Select a toolpath type such as OptiRough.



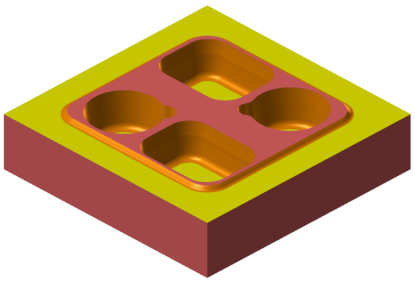
5. Select the Machining geometry and the Avoidance geometry.



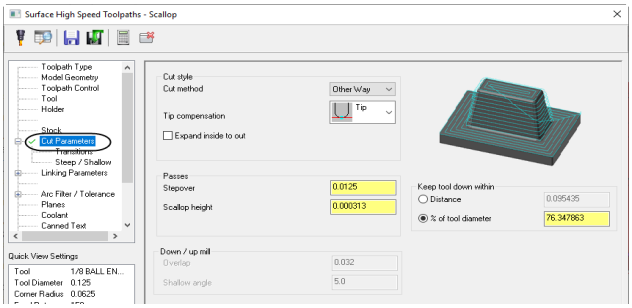
6. Fill in the necessary information on the parameters pages that appear for the toolpath type you selected.



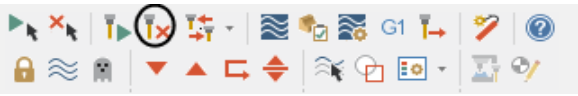
7. Verify the toolpath on your computer screen to confirm the results are as you expected, using Backplot and/or Solid Verify.



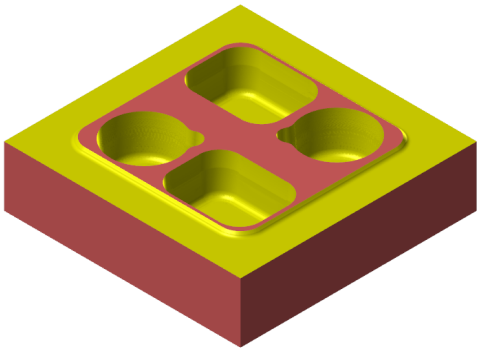
8. Make any changes as required by changing parameters.



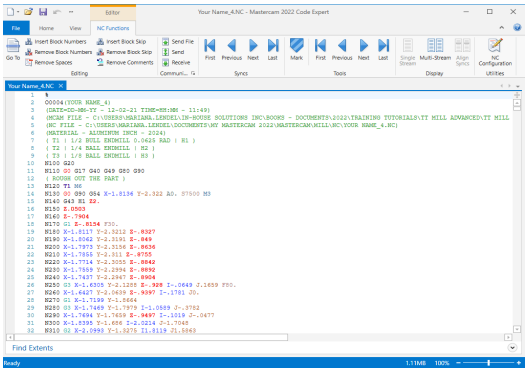
9. Regenerate the “Dirty” operation to update the parameter changes.



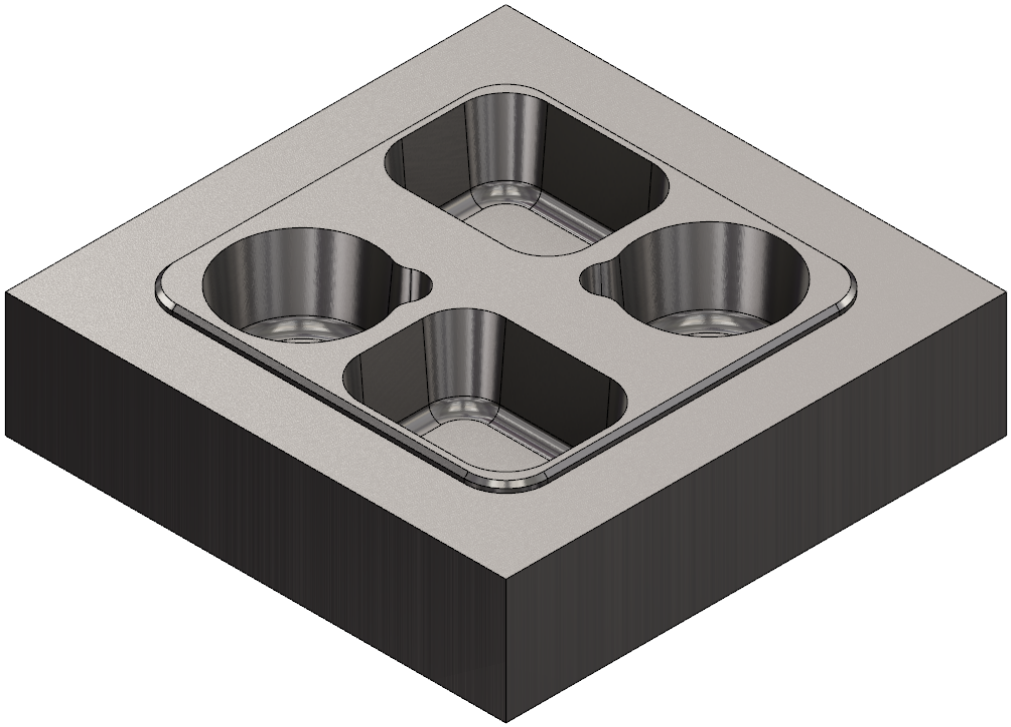
10. Verify again to make sure the toolpath is correct.



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



Tutorial 1:
Geometry Creation



OVERVIEW OF STEPS TAKEN TO CREATE THE PART GEOMETRY:

From Drawing to CAD Model:

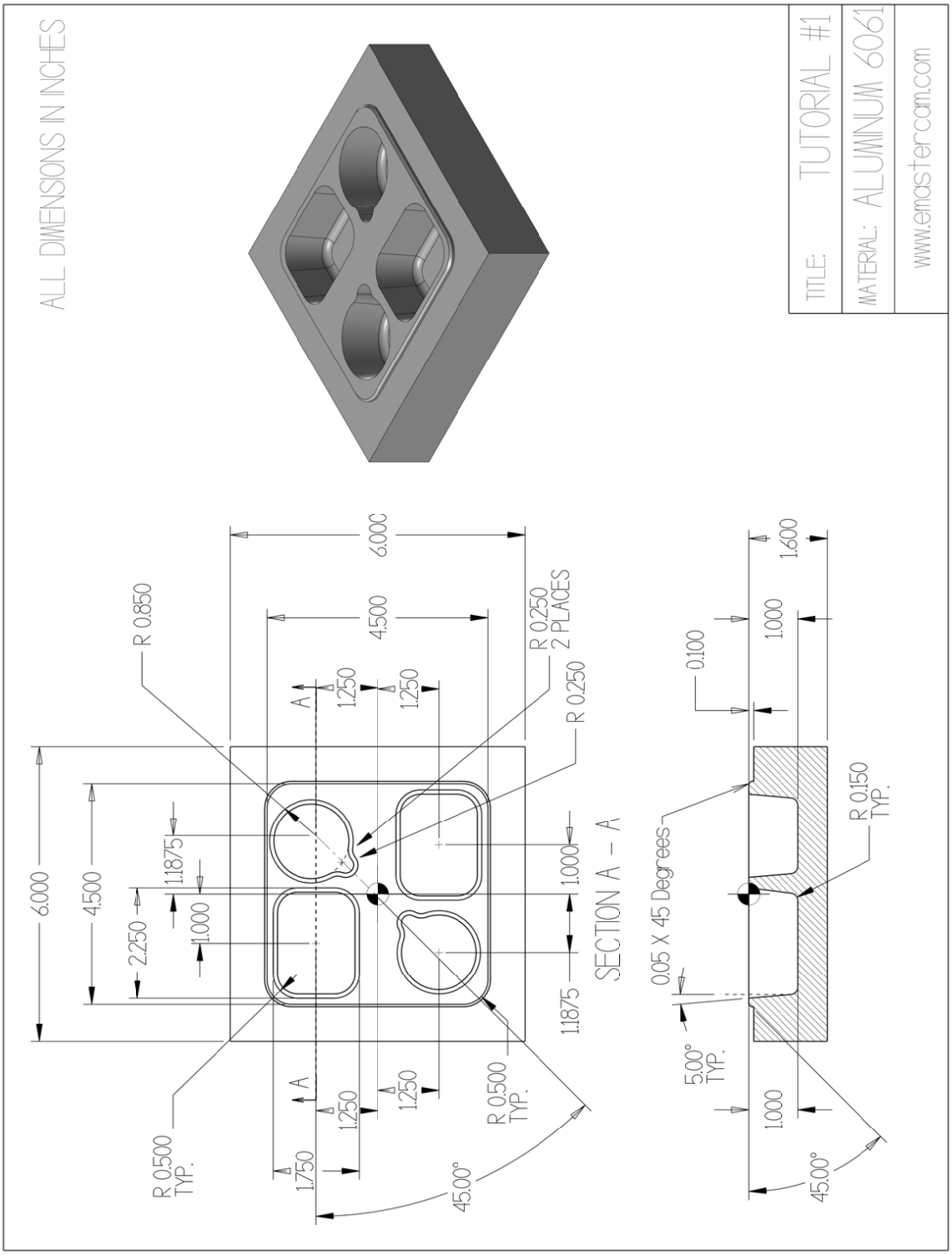
- The student should examine the drawing on the following page to understand what part is being created in the tutorial.
- From the drawing we can decide how to create the geometry in Mastercam.

Create the 3D CAD Model:

- The student will open the wireframe file needed to create the solid.
- Geometry creation commands such as Extrude Create Body, Extrude Cut Body, Fillet and Chamfer will be used.

This tutorial takes approximately forty minutes to complete.

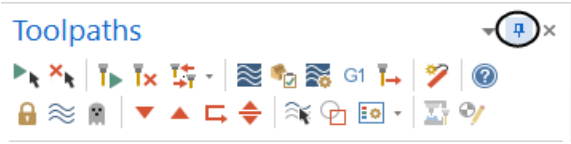
TUTORIAL #1 DRAWING



STEP 1: SETTING UP THE GRAPHICAL USER INTERFACE

Please refer to the Getting Started section for more info on how to set up the graphical user interface. In this step, you will learn how to hide the manager panels to gain more space in the graphics window.

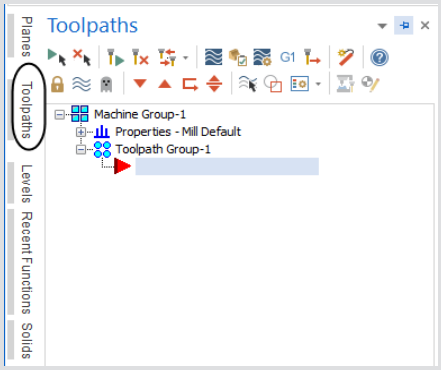
- Use **Auto Hide** icon to hide all **Manager** panels.



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

Note: To un-hide them temporally, you can click on one of the Managers to open it as shown.

While creating the geometry, keep the Manager panels hidden. This ensures more space in the graphics window for the geometry.



SOLID CREATION

A **Solid** is a geometric entity that occupies a region of space and consists of one or more faces, which define the closed boundary of the solid. A solid operation requires closed boundary geometry.

STEP 2: OPEN THE FILE WITH THE WIREFRAME

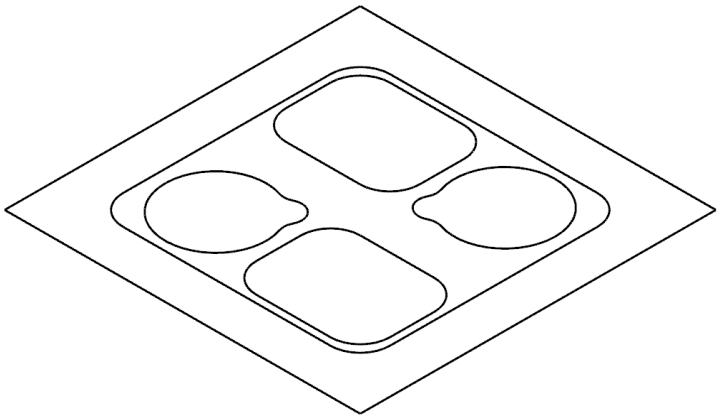
In this step you will open the file with the wireframe.

Note: The wireframe geometry of the part is already created. During this tutorial we will concentrate on the solid geometry creation. The file can be downloaded from www.emastercam.com/trainingfiles.

Resources - Download the file from www.emastercam.com/trainingfiles

- From the **QAT** select the **Open** icon as shown.

- Select the file **TUTORIAL 1 WIREFRAME.MCAM**.
- The geometry should look as shown.

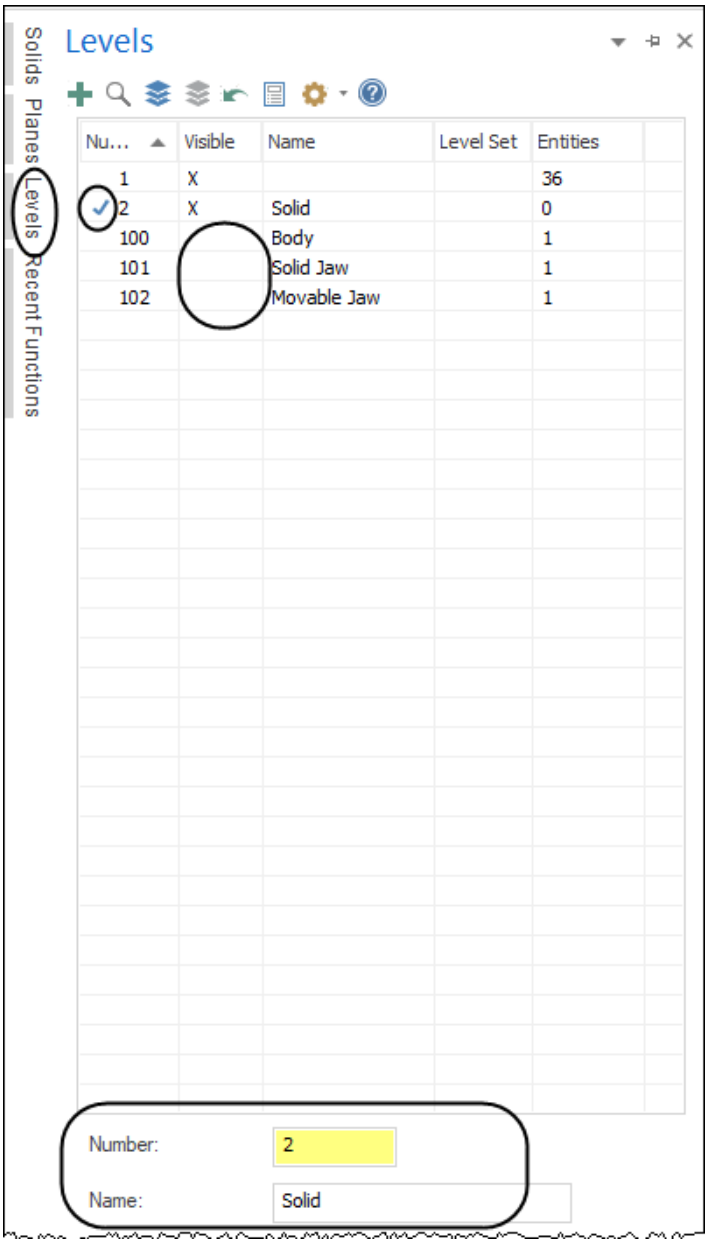


Note: If the vise is also on the screen, follow next step to remove it.

STEP 3: CHANGE THE MAIN LEVEL TO 2

Levels are a primary organizational tool in Mastercam. A Mastercam file can contain separate levels for wireframe, surfaces, drafting entities and solids. By organizing your files into levels, you can easily control which areas of the drawing are visible at any time and which parts are selectable. By doing so, you will not inadvertently make changes to areas of the drawing you do not want to change.
In this step you will change the **Main Level to 2** to create the solid on **Level 2**.

- From the left side of graphics window, select the **Levels** tab as shown.
- Click in the **Number** area and enter in the level number **2** and type in the name "**Solid**" as shown.
- Make sure that **Levels 100, 101, 102** are invisible, otherwise, click to remove the **X**s in the **Visible** column next to them as shown.

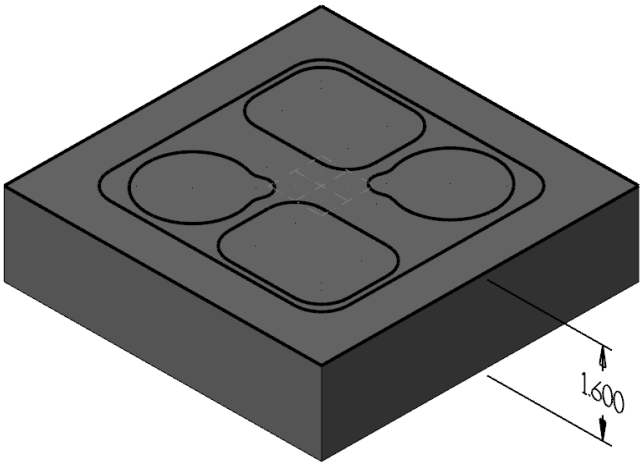


STEP 4: CREATE THE SOLID BODY

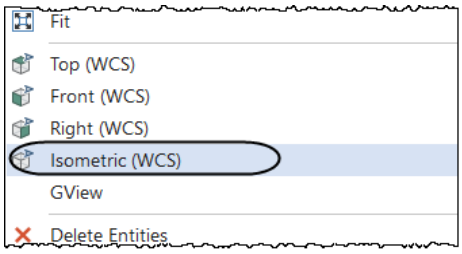
Unlike wireframe models, which are a collection of curves, and surface models, which are a collection of surfaces, a solid model is a closed single entity.

This step shows you how to create a **Solid body** using the **Solid Extrude** command.

Step Preview:

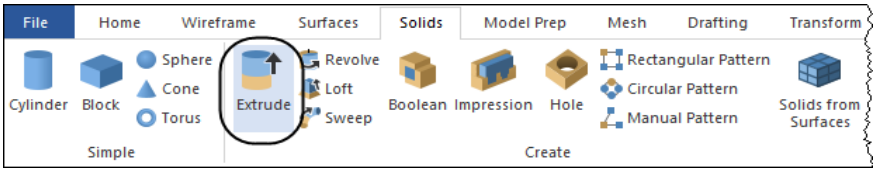


- Right mouse click in the graphics window and select **Isometric** view as shown.



Solids

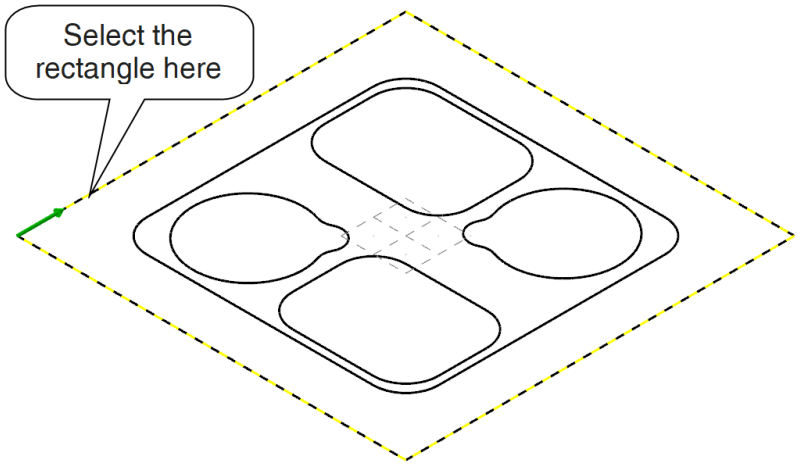
- From the **Create** area, select the **Extrude** icon as shown.



- Leave the default settings in the **Chaining** dialog box and select the outside rectangle as shown.

Chaining is the process of selecting and linking geometry entities such that they form the foundation of a toolpath, a surface, or a solid. When you chain the geometry, you can select one or more sets of curves (lines, arcs, and splines) that have adjoining endpoints.
Chaining differs from other selection methods because it assigns order and direction to the selected curves. Chaining order and direction determine how surfaces, solids, and toolpaths are generated.

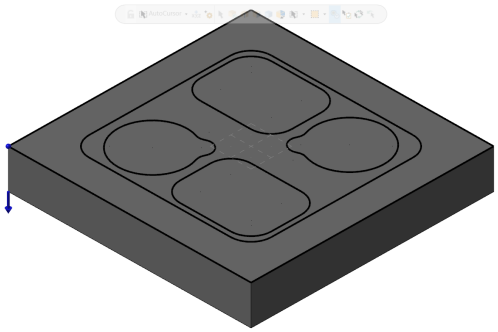
- [Select chain(s) to extrude 1]: Select the outer rectangle as shown.



Note: If you did not select the chain correctly, from the Chaining dialog box, click on the **Unselect** button to undo the previous selection. If you need to change the chain direction, click on the **Reverse** button.

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

- Press **Alt + S** if needed to see the solid in shaded mode as shown below.
- Press **Alt + F1** to fit the geometry inside the graphics window if needed. Make sure the arrow in the graphics window points downwards as shown.
- Otherwise, in the **Solid Extrude** panel, click on the **Reverse All** icon as shown. 



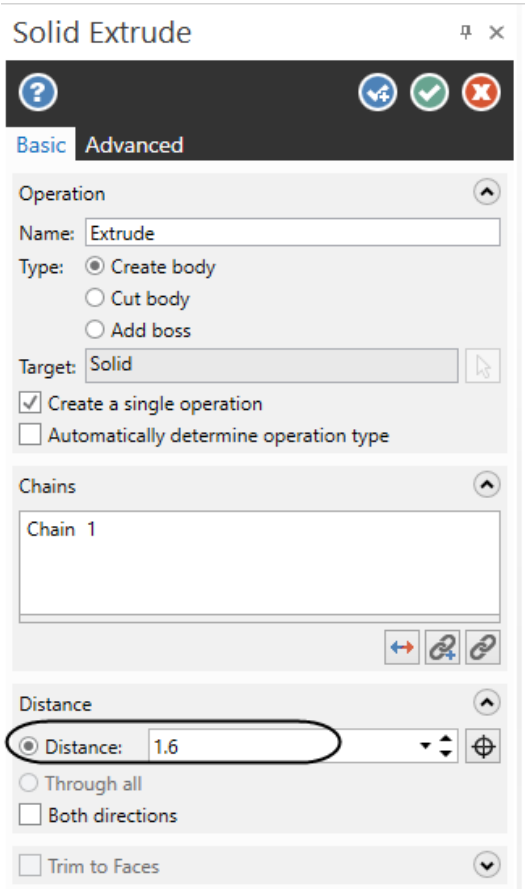
- Change the **Distance** to **1.6**.

Extrusion Operation is used to create a solid body, cut a solid body, or add a boss to another solid.

Reverse Direction extrudes the solid in the opposite direction from the arrow on the chain indicating the extrusion direction.

Distance allows you to control the length of extrusion, by specifying a Distance, extending **Through all**, extending in Both directions, or trimming to selected faces.

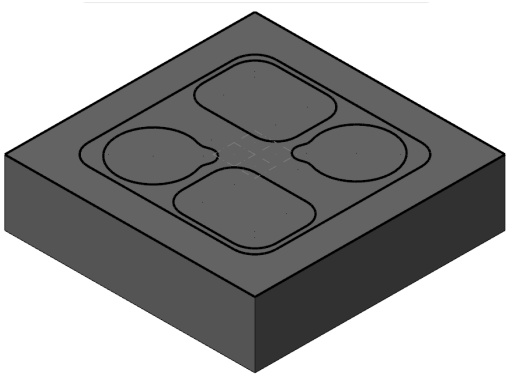
Both Directions allows you to extrude in both directions from the chain.



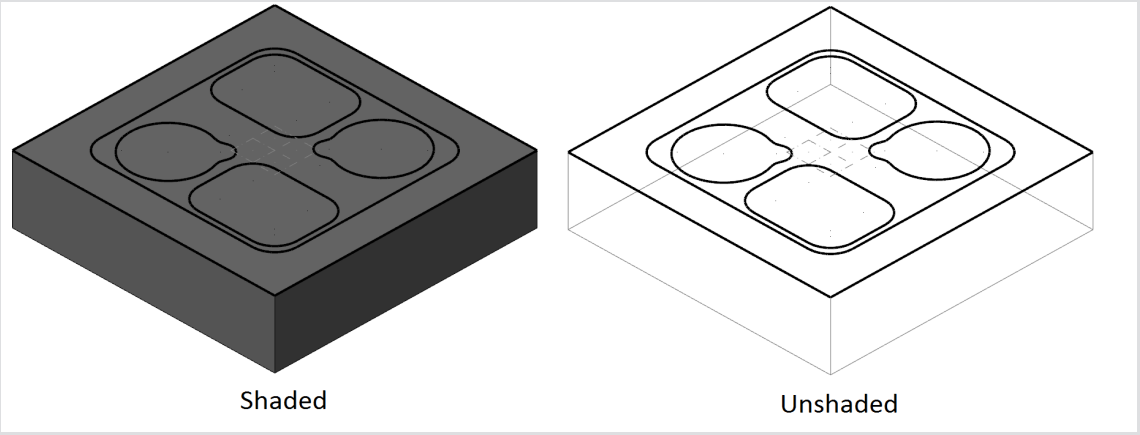
Note: The **Type** can only be set to **Create body** as this is the first solid operation.

- Select the **OK and Create a New Operation** button to remain in the same command. 
- From the **Status bar**, select **Outline Shaded** to see the solid edges better.

- The solid should look as shown.



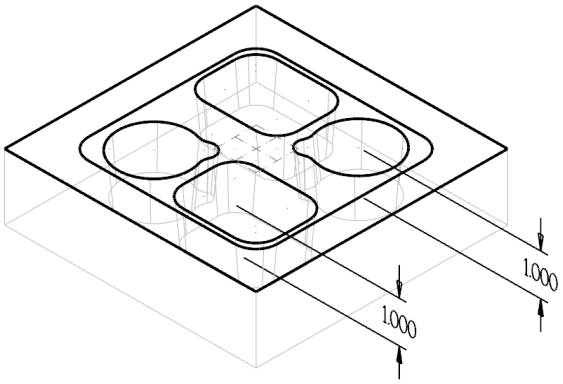
Note: To view the part shaded or unshaded at any time, you can press **Alt + S** on the keyboard**.**



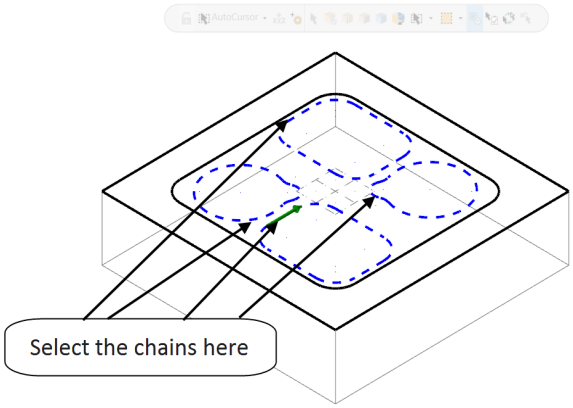
STEP 5: CREATE THE POCKETS

This step shows you how to create the 1.0" deep pockets using **Solid Extrude Cut body** operation.

Step Preview:

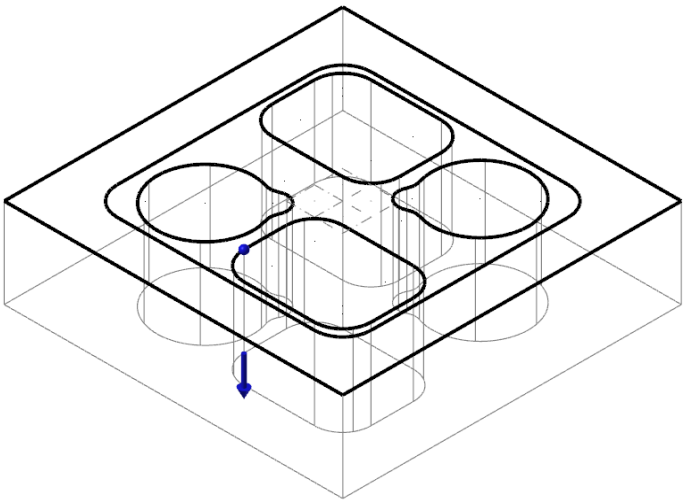


- If you have accidentally exited the solid extrude command, start the command as previously shown.
- Otherwise, press **Alt + S** to unshade the solid and select the chains as shown.

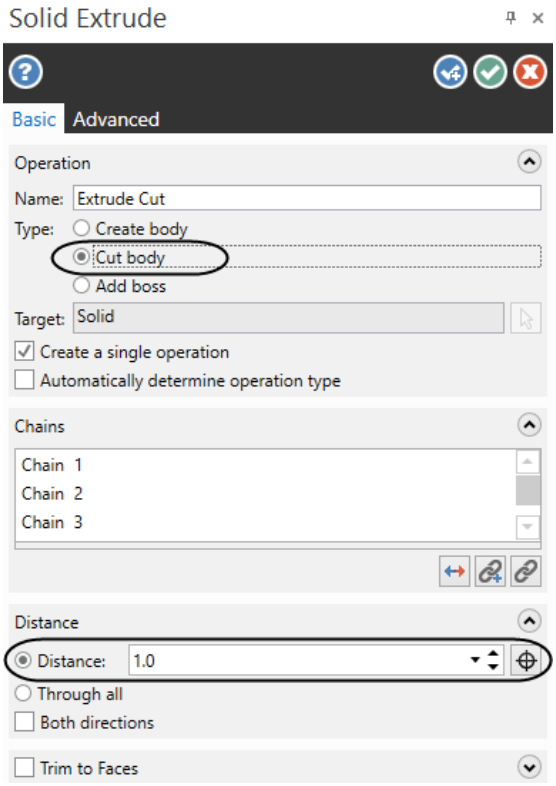


- Select the **OK** button to exit the **Chaining** dialog box. 

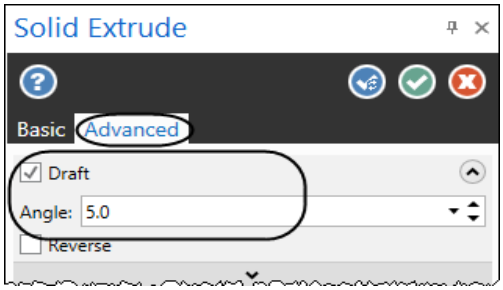
- The arrow should point downwards as shown, otherwise click on the **Reverse All** icon. 



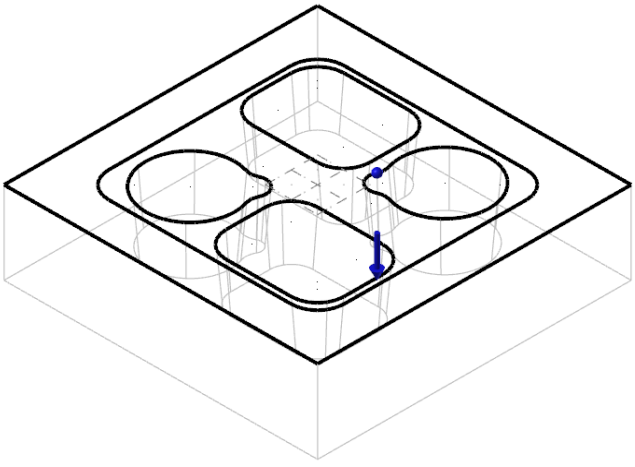
- In the **Solid Extrude** panel, enable **Cut body** and change the **Distance** to **1.0**.



- From the **Solid Extrude** panel, select the **Advanced** tab, and enable **Draft** and change the **Angle** to **5** degrees as shown.

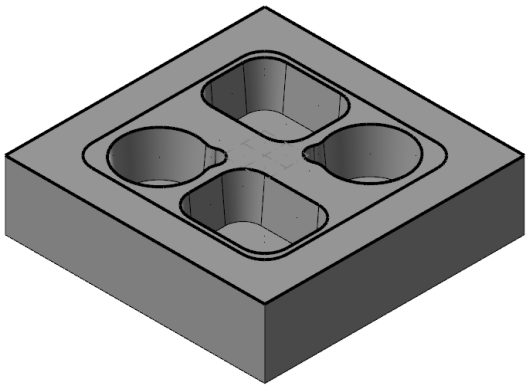


- The pockets with the draft angle should look as shown.



- If the draft angle is outwards, enable **Reverse**.

- Press **Alt + S** to see the solid in a shaded mode.

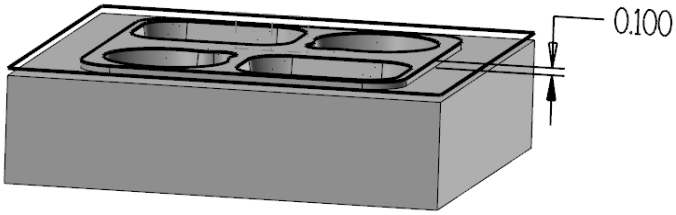


- Select the **OK and Create New Operation** button to remain in the same command. 

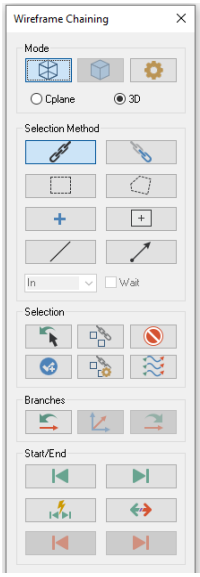
STEP 6: CREATE THE TOP BOSS

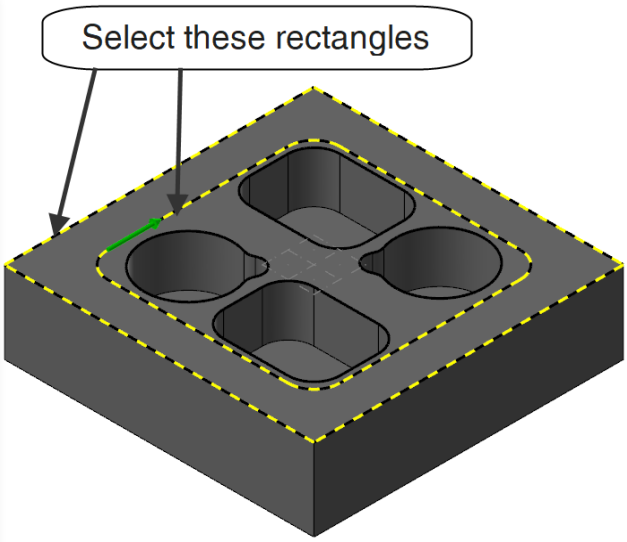
This step shows you how to create the top boss using the **Solid Extrude Cut** body operation.

Step Preview:



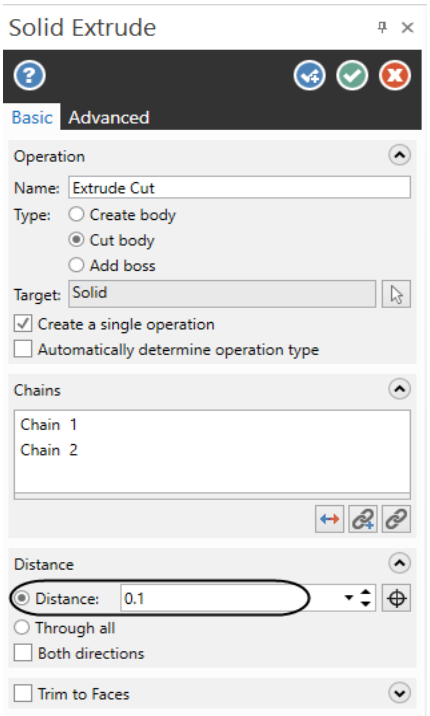
- If you accidentally exit the solid extrude command, start the command as previously shown.
- Otherwise, select the outside and inside rectangles as shown.



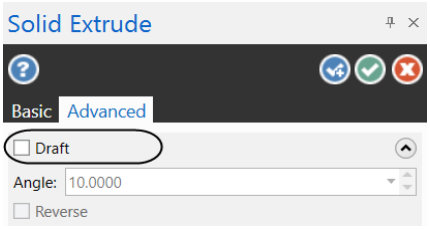


- Select the **OK** button to exit the **Chaining** dialog box.
- The arrow should point downwards, otherwise click on the **Reverse All** icon.

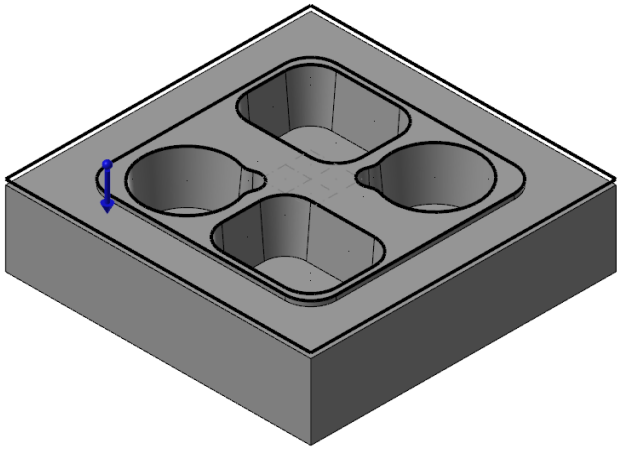
- In the **Solid Extrude** panel, select **Basic** tab and make sure that **Cut body** is enabled and the **Distance** is set to **0.1** as shown.



- From the **Solid Extrude** panel, select the **Advanced** tab, disable **Draft** as shown.

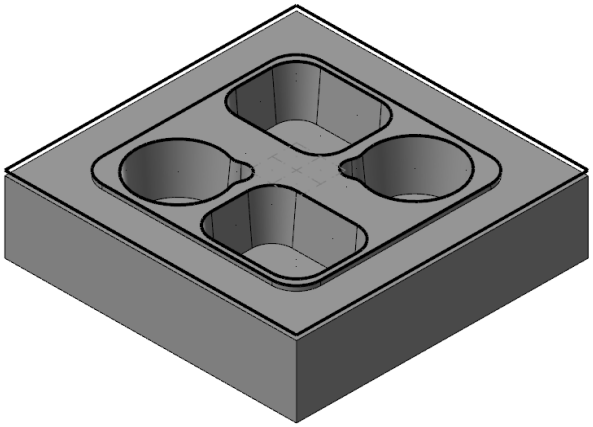


- The solid should look as shown.



- Select the **OK** button to exit the **Extrude** command. 

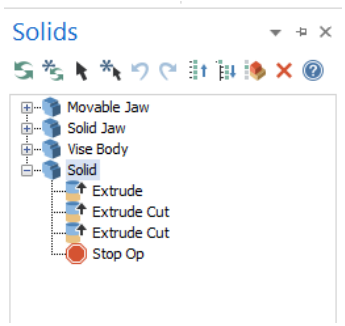
- The solid should look as shown.



STEP 7: USING SOLIDS MANAGER

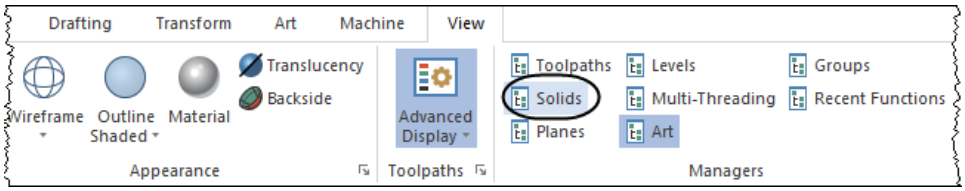
This step shows you how to check the solid and modify it if needed using the Solids Manager.

Step Preview:

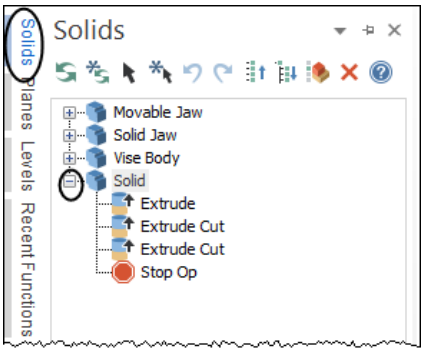


View

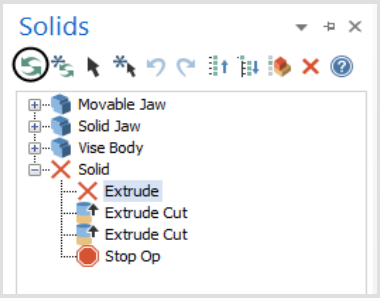
- From the **Managers** group, click on the **Solids** as shown.



- Or, select the **Solids** tab from the left of the interface and the **Solids Manager** opens as shown.
- To check the solid history, click on the plus in front of the solid and the operations that were created until now should be listed as shown.



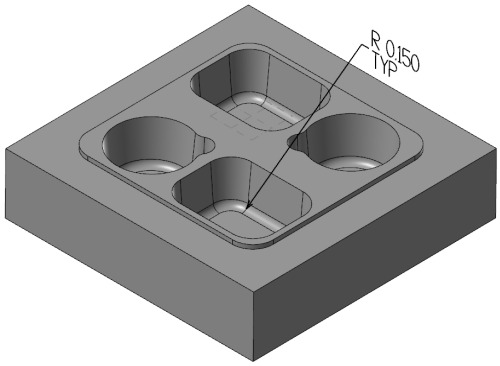
Note: To modify any of the solids operation, double click on the operation. The corresponding panel will appear on the screen, and hence the parameters can be modified. To update the solid after modifying the parameters, click on the **Regen all** button from the **Solids Manager**.



STEP 8: FILLET THE POCKETS - SOLID CONSTANT FILLETS

In this step we will fillet the part using the **Solid Constant Fillet** command. You will select the faces at the bottom of the pockets.

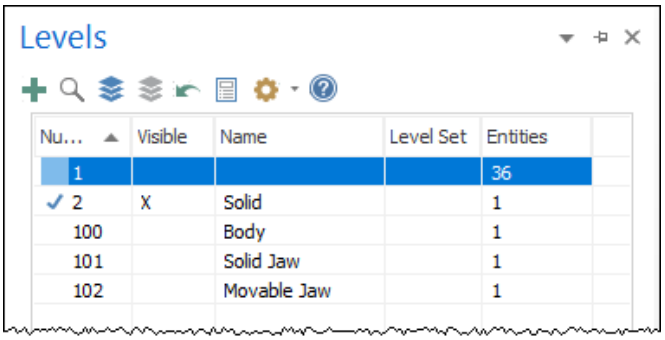
Step Preview:



8.1 Make Level 1 invisible

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

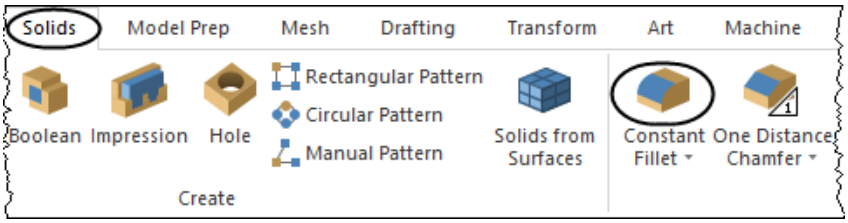
- In the **Levels** panel, click in the **Visible** column next to 1 to remove the **X** as shown.



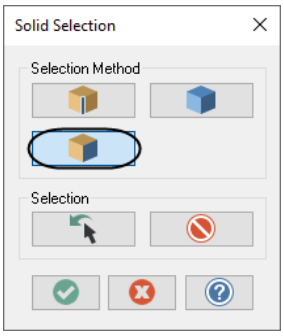
8.2 Fillet the solid with a 0.15" Radius

Solids

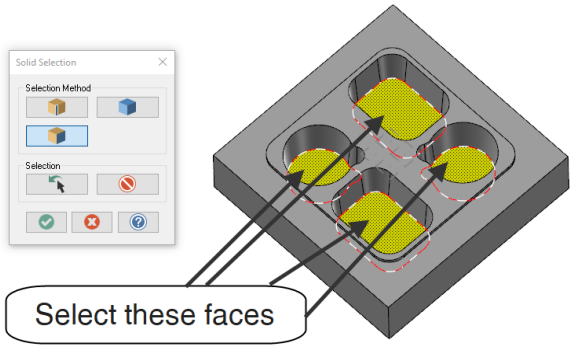
- From the **Modify** group, select the **Constant Fillet** icon as shown.



- In the **Solid Selection** dialog box, enable only the **Face** button as shown.



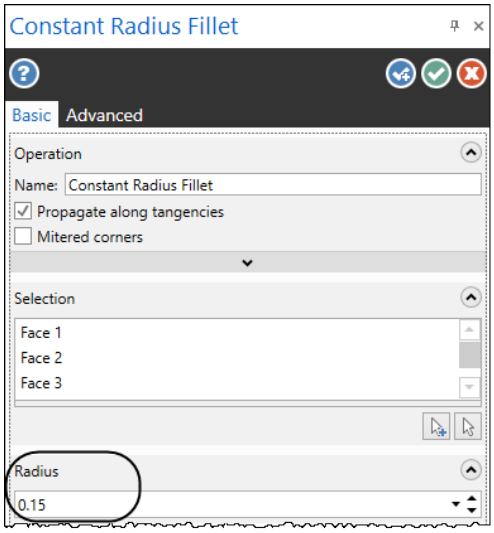
- To rotate the part, move the cursor to the center of the part.
- Click with the mouse wheel and hold it down while slowly moving it in one direction until the part is rotated as shown.
- [Select entities to fillet]: Select the faces as shown.



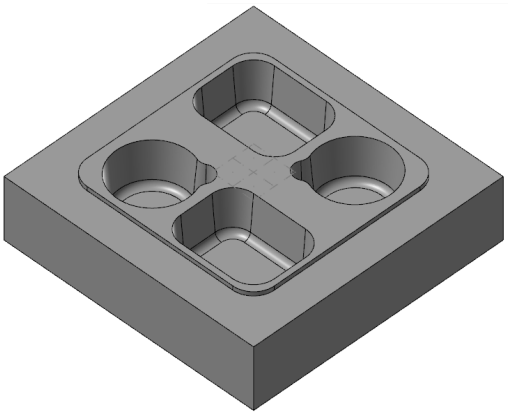
Note: Ensure the cursor cue changes to this  , indicating that you are selecting a face.

- In the **Solid Selection** dialog box, click the **OK** button to continue. 

- In the **Constant Radius Fillet** dialog box, make sure that the **Radius** is set to **0.15** as shown.



- The solid should appear as shown.

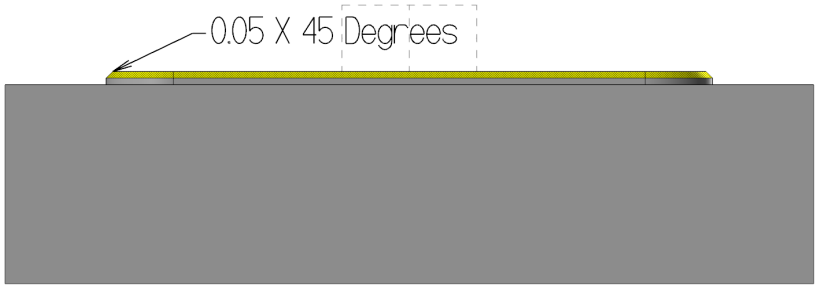


- Select the **OK** button to exit the command. 

STEP 9: CHAMFER THE TOP BOSS

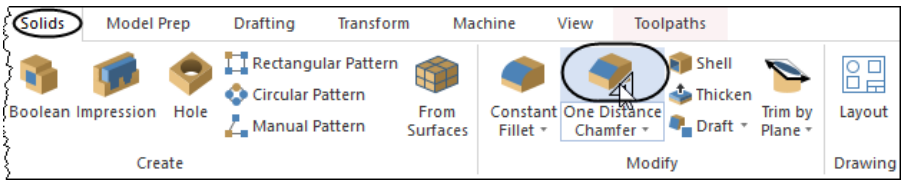
In this step you will use **One Distance Chamfer** command to chamfer the top boss with a 45 degrees angle and 0.05" width.

Step Preview:

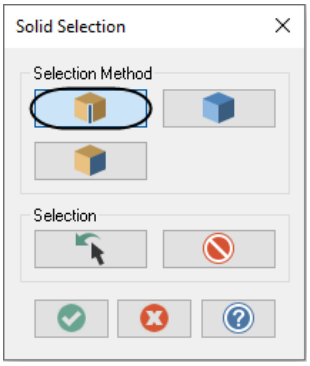


Solids

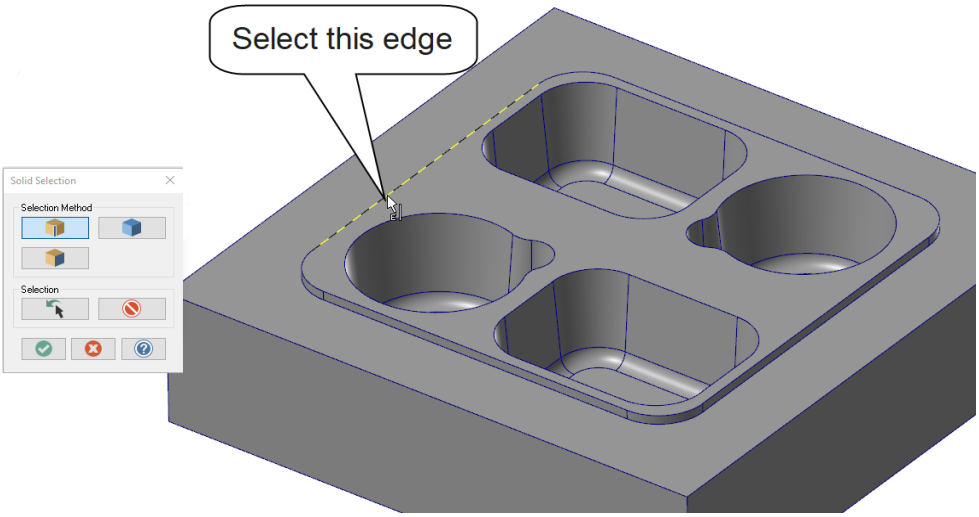
- From the **Modify** group, select **One Distance Chamfer** icon as shown.



- In the **Solid Selection** dialog box, unselect the **Body** and **Face** buttons and leave enabled only the **Edge** button as shown.



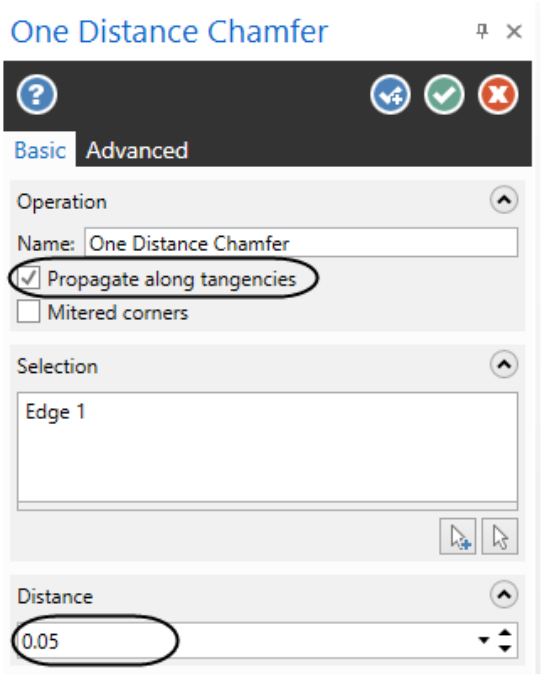
- [Select entities to chamfer]: Select the edge of the boss as shown after the **Solid Selection** dialog box appears.



Note: Ensure the cursor cue changes to this , indicating you are selecting an edge.

- In the **Solid Selection** dialog box, click on the **OK** button to continue.

- In the **One Distance Chamfer** panel, enable **Propagate along tangencies** and change the **Distance** to **0.05** as shown.



- Select the **OK** button to exit **One Distance Chamfer** command.