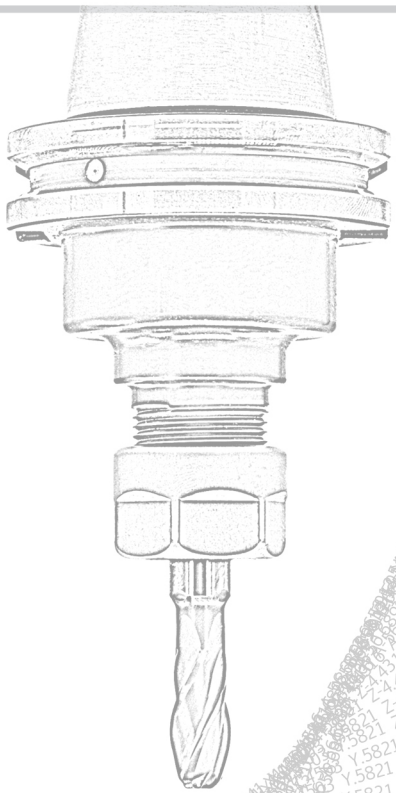
Mastercam®



Mastercam Basics Tutorial

APRIL 2022

MASTERCAM BASICS TUTORIAL

April 2022
© 2022 CNC Software, LLC – All rights reserved.
Software: Mastercam 2023

Terms of Use

Use of this document is subject to the Mastercam End User License Agreement. The Mastercam End User License Agreement can be found at:

https://www.mastercam.com/legal/

Be sure you have the latest information!

Information might have changed or been added since this document was published. The latest version of the document is installed with Mastercam or can be obtained from your local Reseller. The Resolved Issues website—installed with each release—includes the latest information about Mastercam features and enhancements.

TABLE OF CONTENTS

Table of Contents ..... **iii**

Introduction **7**

General Tutorial Requirements 7

Mastercam Interface Overview **9**

Exercise 1: Exploring Mastercam’s Interface 9

Common Interface Controls 13

Exercise 2: Exploring Mastercam's Backstage 13

Exercise 3: Customizing the Quick Access Toolbar 14

Exercise 4: Using Mastercam Resources 17

Using myMastercam 18

Giving Feedback in Mastercam 19

Using Mastercam Help 20

Exercise 5: Exploring the Ribbon Interface 23

Exercise 6: Customizing the Ribbon 26

Exercise 7: Moving and Docking Managers 30

Exercise 8: Setting Entity Attributes in the Current File 35

Exercise 9: Understanding the Selection Bar and AutoCursor 41

Exercise 10: Using Quick Masks 45

Exercise 11: Editing Geometry from the Top-Level 49

Editing with line controls 49

Editing with arc controls 51

Exercise 12: Exploring the Status Bar 54

Working with Configuration Files **57**

Exercise 1: Creating a Configuration File 57

Exercise 2: Changing System Colors 59

Exercise 3: Changing CAD Settings 62

Exercise 4: Changing the Size and Opacity of On-screen Controls 63

Exercise 5: Setting up AutoSave and Backup 65

Customizing Mastercam **67**
Exercise 1: Customizing the Graphics Window Context Menu 67
Exercise 2: Mapping Functions to Keyboard Shortcuts 70
Exercise 3: Setting Mastercam Display Options 72
Working with Files **75**
Exercise 1: Opening Non-Mastercam Part Files 76
Exercise 2: Exporting a Single File 79
Exercise 3: Using Zip2Go 82
Viewing Your Part **87**
Exercise 1: Changing Standard Views 87
Exercise 2: Zooming In and Out 91
Exercise 3: Rotating Your Part View 94
Exercise 4: Panning Your Part 95
Exercise 5: Analyzing Your Part 96
Exercise 6: Using viewsheets 101
Exercise 7: Using Materials 105
Exercise 8: Managing Section Views 109
Working With Levels **113**
Exercise 1: Exploring the Levels Manager 113
Exercise 2: Showing or Hiding Levels 116
Exercise 3: Changing the Active Level 117
Exercise 4: Creating a Level and Assigning Entities 119
Exercise 5: Moving Entities to a Different Level 122
Understanding Views and Planes **127**
Views and Planes 127
Exercise 1: Exploring the Planes Manager 128
Exercise 2: Creating a New Plane Using the Dynamic Gnomon 130
Exercise 3: Creating a New Plane Using Planes Manager 134
Exercise 4: Using a Custom Plane for Drawing Geometry 138
Working With Chain Selection **143**

Exercise 1: Creating Wireframe from a Solid 143
Exercise 2: Exploring Wireframe Chaining 146
Exercise 3: Exploring Solid Chaining 151
Introduction to Toolpaths 159
Exercise 1: Selecting a Machine and Setting Up Stock 160
Exercise 2: Choosing a Toolpath and Geometry 164
Exercise 3: Programming a Toolpath 167
Exercise 4: Backplotting and Verifying 171
Exercise 5: Post Processing 175
Conclusion 179
Mastercam Resources 179
Contact Us 179

INTRODUCTION

Welcome to the *Mastercam Basics Tutorial*. This tutorial helps you explore Mastercam's interface and learn basic concepts of the software. This tutorial includes the following:

- A broad look at the Mastercam interface.
- An introduction to Mastercam's File menu, also known as the Backstage.
- An introduction to creating and managing levels and planes.
- An exploration of solid and wireframe chaining.
- An introduction to creating toolpaths.

Tutorial Goals

- Understand basic Mastercam functions.
- Customize a configuration file and Mastercam's interface for your working style.
- Chain geometry and create toolpaths for your part.
- Manipulate Mastercam's graphics window, levels, and planes to view your part.

General Tutorial Requirements

WARNING
This tutorial is for educational purposes only. We have not verified the values and settings in this tutorial with our machines or tools. Do not use the values in this tutorial to cut parts without proper verification and testing for your environment.

NOTE
Screen colors in the tutorial pictures were modified to enhance image quality; they may not match your Mastercam 2023 settings or the tutorial results. These color differences do not affect the exercises or your results.

Estimated time to complete this tutorial: 5 hours

All Mastercam 2023 tutorials have the following general requirements:

- You must be comfortable using the Windows® operating system.
- If you are completing this tutorial using Mastercam Learning Edition, you will be unable to post code.
- Each lesson in the tutorial builds on the mastery of the preceding lesson's skills. We recommend that you complete them in order.
- Additional files may accompany a tutorial. Unless the tutorial provides specific instructions on where to place these files, store them in a folder that can be accessed from the Mastercam 2023 workstation, either with the tutorial or in any location that you prefer.

- You will need an internet connection to view videos that are referenced in the tutorials. All videos can be found on our YouTube channel: www.youtube.com/user/MastercamTechDocs
- All Mastercam tutorials require you to configure Mastercam to work in a default Metric or Inch configuration. The tutorial provides instructions for loading the appropriate configuration file.

CHAPTER 1

MASTERCAM INTERFACE OVERVIEW

Mastercam’s ribbon interface is based on standard Windows design guidelines. The ribbon comprises familiar controls, including a Quick Access Toolbar (QAT), tabs, contextual tab groups, galleries, buttons, and the Backstage. It also features special on-screen controls, and movable, dockable Managers and function panels.

This section briefly touches on the elements that make up the Mastercam interface to give you a better understanding of how to use these elements to create and machine your parts.

Goals

- Explore Mastercam's interface.
- Customize the Quick Access Toolbar.
- Access Mastercam resources.
- Configure entity attributes.
- Understand Selection Bar, AutoCursor, and Fast Point.
- Use Quick Masks.
- Explore the Status Bar.

Exercise 1: Exploring Mastercam’s Interface

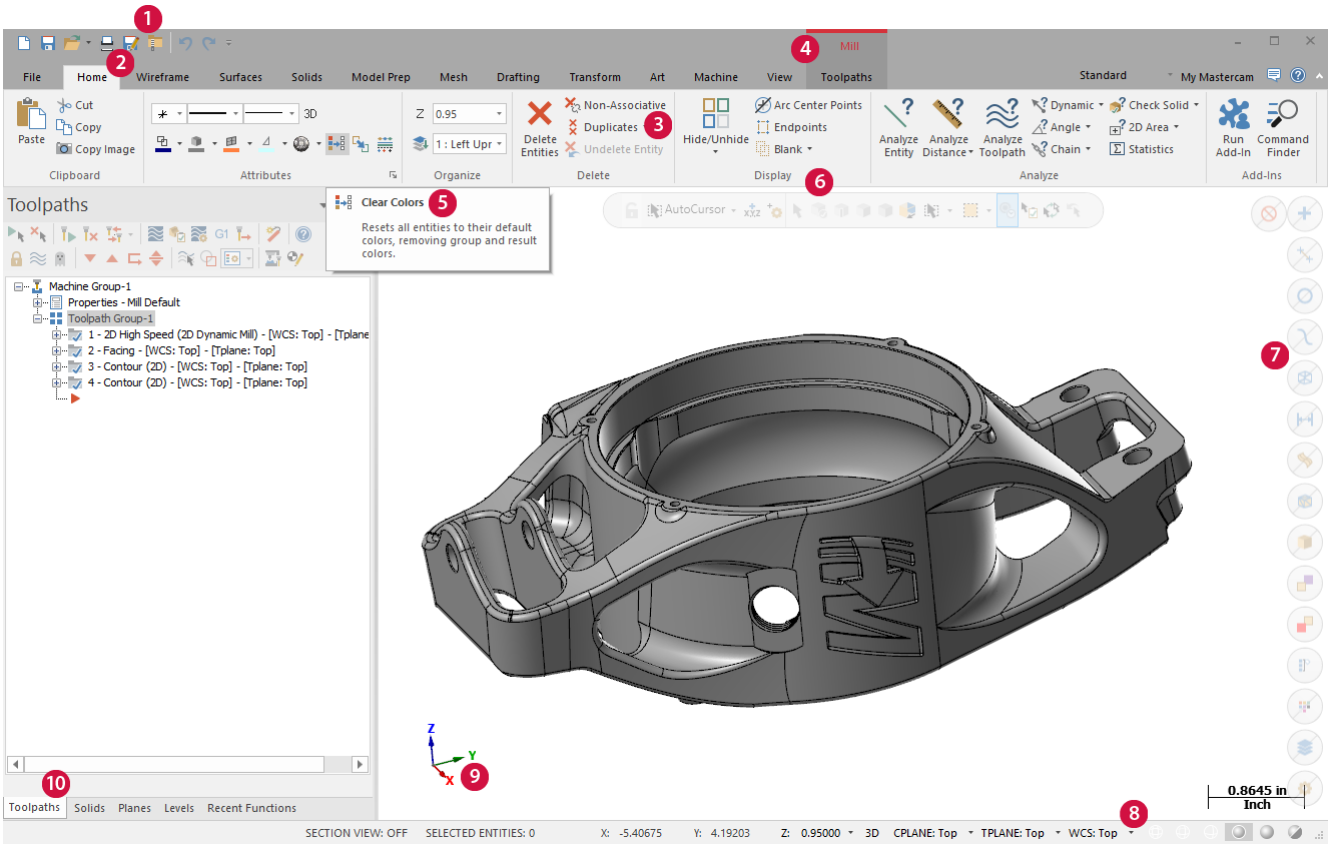
Start Mastercam using your preferred method:

- Double-click Mastercam's desktop icon.

  Or,

- Launch Mastercam from the Windows **Start** menu.

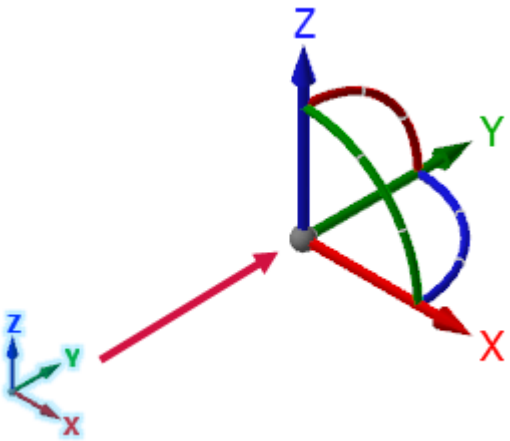
The following image displays the elements that make up the Mastercam interface. The corresponding sections provide brief descriptions of each element and its use.



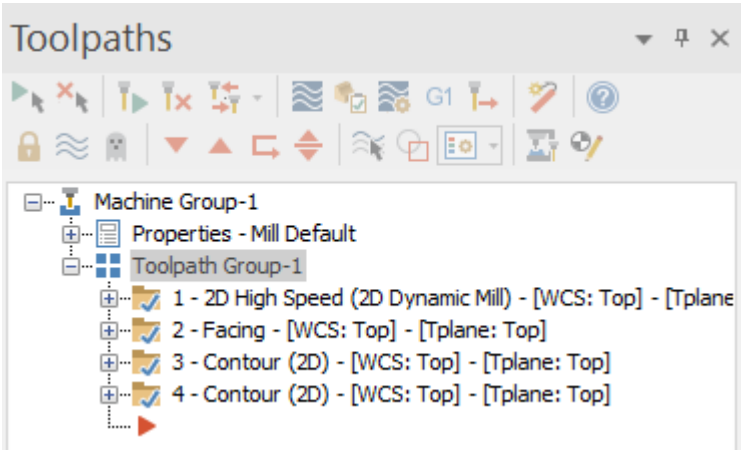
1. **Quick Access Toolbar (QAT)**: A customizable set of commonly used functions that are always available in the interface. The QAT can be located above or below the ribbon.
2. **Tab**: Groupings of related controls. Tabs are organized in a simple-to-complex workflow from left to right.
3. **Tab Group**: A region of the tab that contains a set of related controls.
4. **Contextual Tab**: A tab that displays when you make a specific selection in Mastercam. A contextual tab presents controls and commands relevant to your current activity.
5. **Tooltip**: A small window with descriptive text that displays when you hover over a command or control.
6. **Selection Bar**: A toolbar that combines AutoCursor controls and general selection tools used to select entities in the graphics window. There are two selection modes, Standard Selection and Solid Selection, which are activated based on the function you are using. AutoCursor controls allow you to detect and snap to locations as you move the cursor over geometry in the graphics window. AutoCursor becomes active whenever Mastercam prompts you to select a position in the graphics window.
7. **Quick Masks**: A group of controls that helps you to select *all* entities of a certain type, or to select *only* entities of certain type. Most Quick Mask controls are divided in half. Clicking the left or right side of the control toggles between selection modes. When a Quick Mask control is selected it highlights to indicate the selection mask is on. You can toggle multiple Quick Masks at a time.

8. **Status Bar**: A bar across the bottom of the workspace that shows the coordinate position of the cursor and provides quick access to modify planes, construction mode, and Z depth. The right side of the Status Bar has wireframe, shading, and translucency controls that change the appearance of your part. The left side of the Status bar indicates the number of selected entities and the status of Section View.

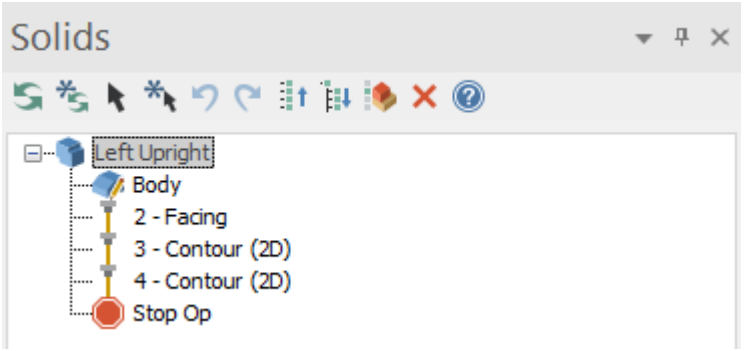
9. **Dynamic Gnomon**: The on-screen gnomon allows you to manipulate views and planes interactively. The gnomon comprises three axes connected at the origin, with selection points that let you choose different types of transformations.



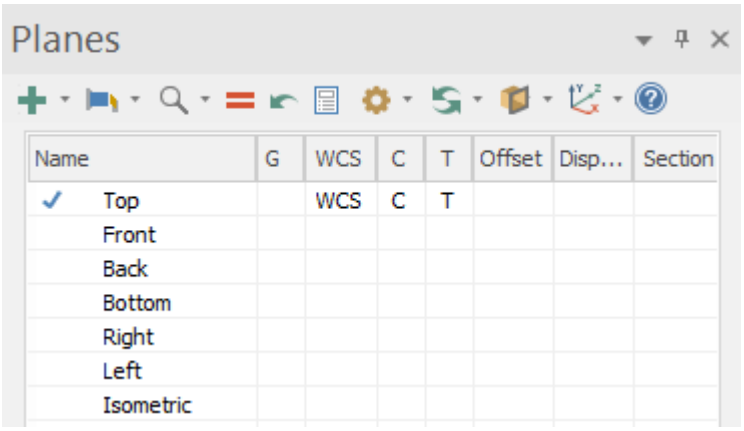
10. **Managers**: Managers include controls for toolpaths, solids, planes, levels, recent functions, and Art. The Managers can be rearranged to suit your workflow.
    - **Toolpaths Manager**: Lists the toolpath groups and machine types for the current file. Use the Toolpaths Manager to control your operations.



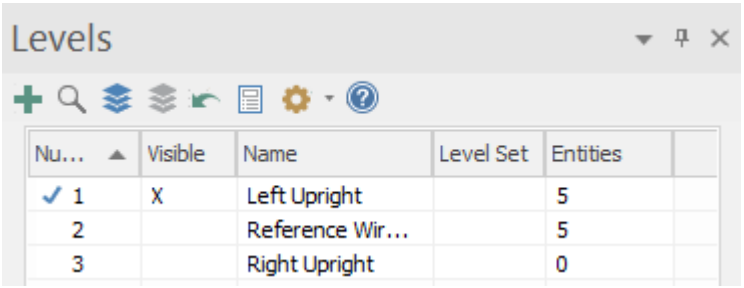
- **Solids Manager**: Lists each solid in the current file along with its operation history and associated toolpaths. Use the Solids Manager to edit solids and their operations.



- **Planes Manager**: Shows the planes in the current file. Use the Planes Manager to adjust planes and work offsets.



- **Levels Manager**: Shows the levels in the current file. Use the Levels Manager to control your levels.



You can learn more about Managers in "Moving and Docking Managers" on page 30.

11. **Graphics window**: The space in which you view, create, and modify your parts. The graphics window also displays information about the current units (inches or millimeters), and the coordinate axes for the current view or plane.

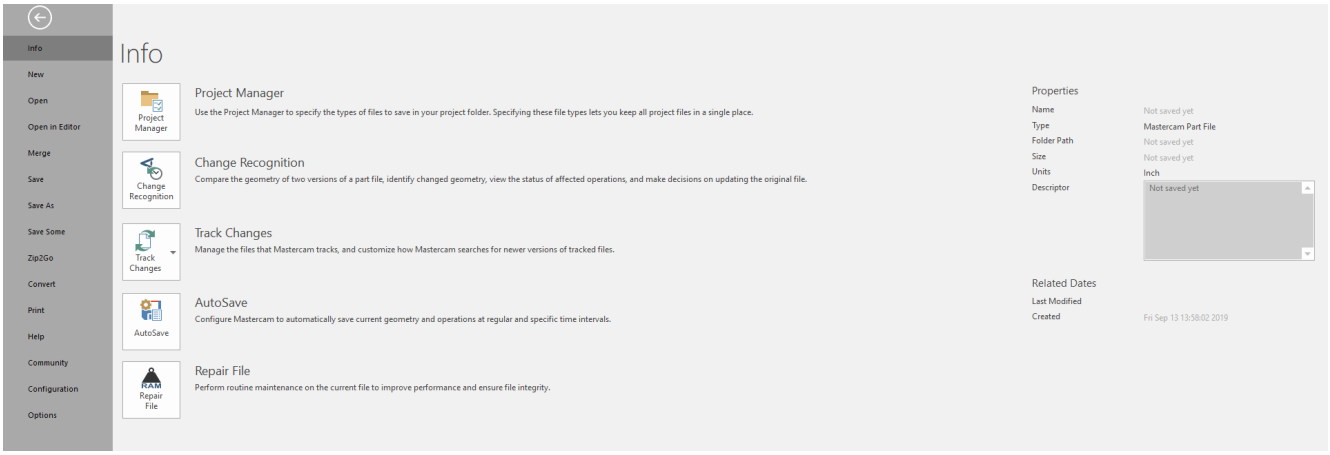
Common Interface Controls

The following table shows common controls used in the Mastercam interface. The icons may vary slightly depending on their location in Mastercam.

Icon	Option	Icon	Option
	OK and Create New		Cancel
	OK/Save		Help
	Apply		About Mastercam

Exercise 2: Exploring Mastercam's Backstage

The **File** drop-down in Mastercam is also known as the Backstage. Click **File** to open the Backstage view. The image below shows the **Info** page of the Backstage with links on the left side to other pages and functions.



The following list provides brief descriptions of some of the functions and pages located in the Backstage. Click each page to explore the Backstage.

- **Info**: Presents information and properties of the current Mastercam file. The **Info** page gives you access to functions that take action on the currently open file, including: Project Manager, Change Recognition, Track Changes, AutoSave, and Repair File.
- **New**: Opens a new file.
- **Open**: Accesses the **Open** page where you can select a recent document or browse for a file to open. You can pin frequently used files or folders to the Recent Documents and Recent Folders sections.
- **Open in Editor**: Opens a file using a file editor of your choice.
- **Merge**: Imports and merges entities from an existing part file into the current file.
- **Save/Save As**: Saves the current part file or saves the file with a new name. You can pin frequently used folders to the **Recent Folders** section of the **Save As** page.
- **Save Some**: Saves only entities that you select in the graphics window.

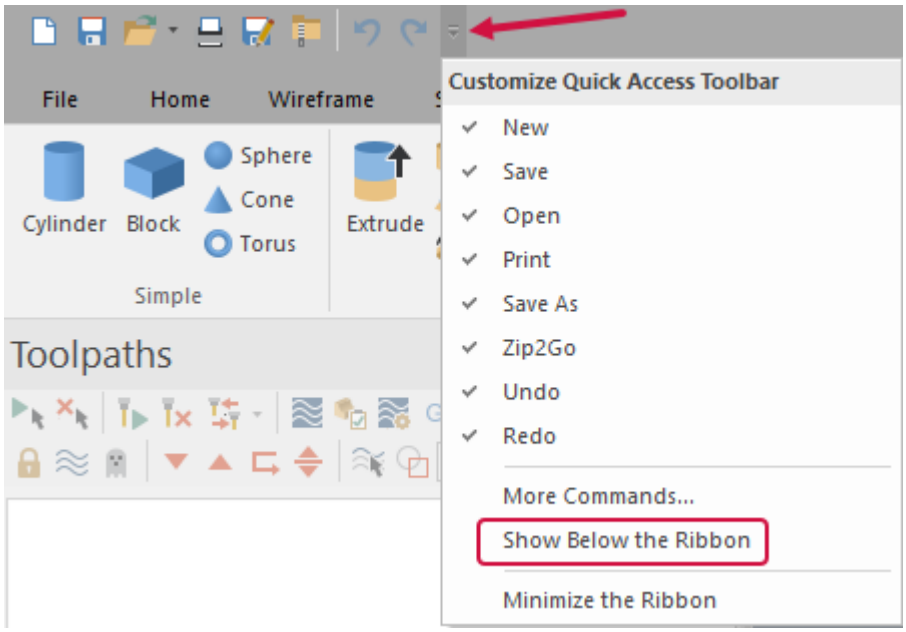
- **Zip2Go**: Opens the Zip2Go Wizard. The Zip2Go Wizard compresses the currently open Mastercam part and its associated files (configuration, machine definition, post files, tool libraries, etc.) into a .Z2G, .ZIP, or .MCAM-CONTENT file. A Zip2Go file is helpful when sharing file information with other users or Technical Support.
- **Convert**: Provides access to Import and Export functions, as well as the Migration Wizard so that you can update older Mastercam files to the latest version.
- **Print**: Previews, configures, and prints the current Mastercam file.
- **Help**: Presents information about Mastercam and licensing. Includes links to the latest *What's New* and *Resolved Issues*, tutorials, and the Help system. Visit this page to check for software updates. Access to some of these resources requires an active Internet connection.
- **Community**: Links your account to Mastercam.com, Mastercam University, the Mastercam Community App, Mastercam Forums, Mastercam Knowledge Base, and the Customer Feedback Program. These resources require an active internet connection.
- **Configuration**: Opens the **System Configuration** dialog box where you set system defaults for Mastercam.
- **Options**: Opens the **Options** dialog box where you customize the Mastercam interface.

Exercise 3: Customizing the Quick Access Toolbar

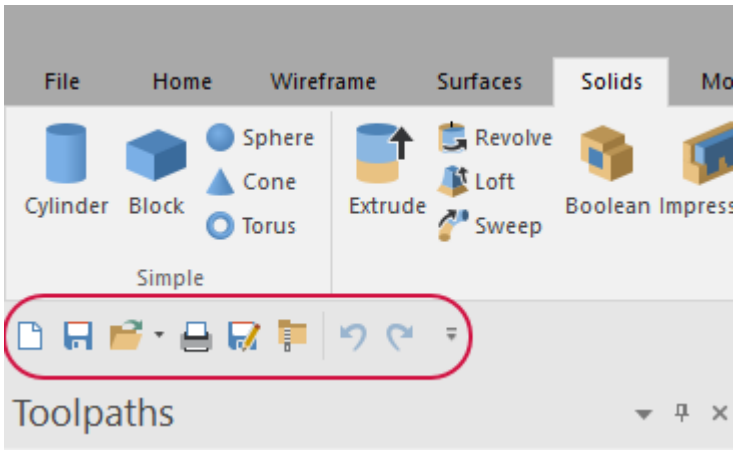
The Quick Access Toolbar (QAT) is a collection of frequently used functions. The QAT is always available and can be displayed above or below the ribbon. You can add to or remove functions from the QAT.

In this exercise you move and customize the QAT.

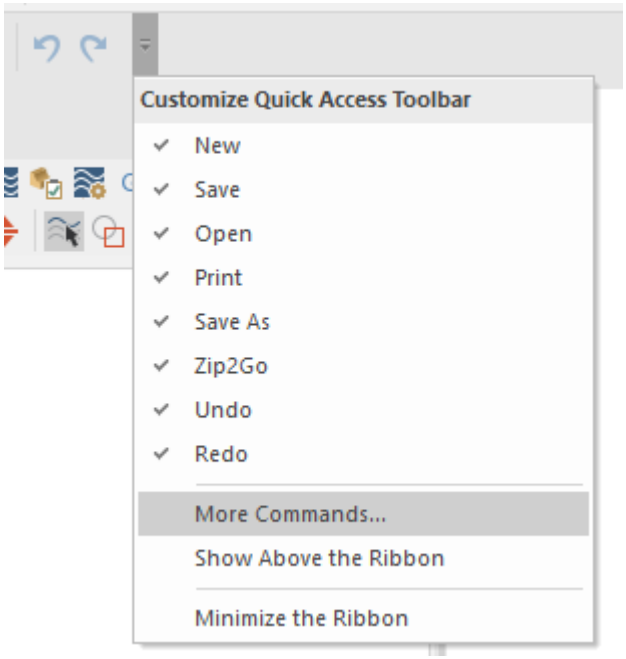
1. Click the **Customize Quick Access Toolbar** drop-down, and select **Show Below the Ribbon**.



The QAT displays between the ribbon and the managers.

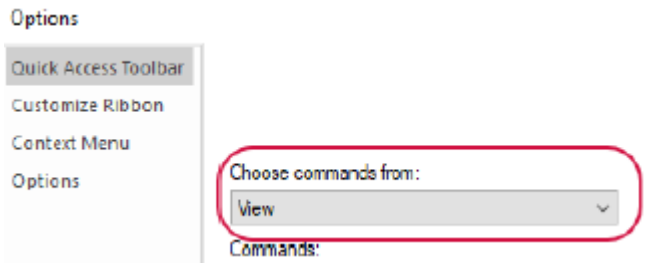


2. Click the drop-down again, and select **More Commands**.

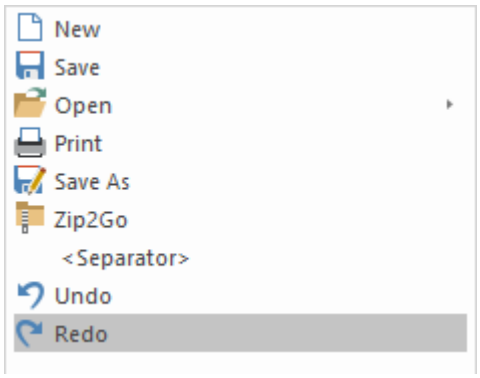


The **Options** dialog box opens.

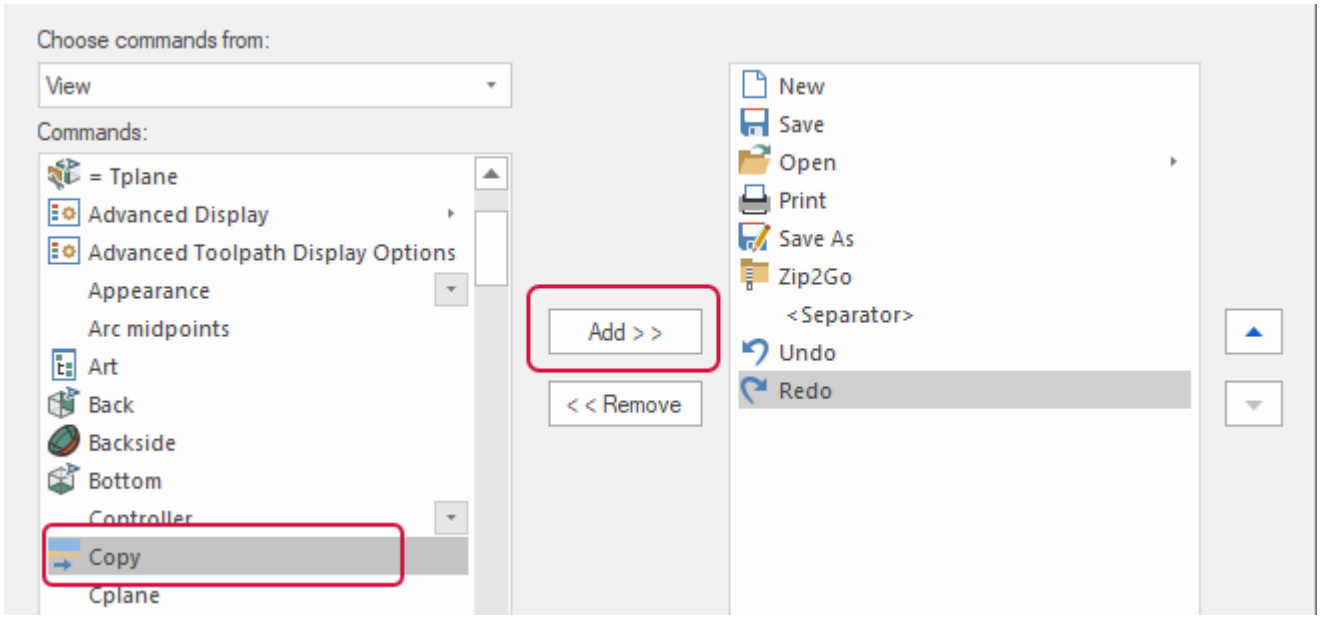
3. On the **Quick Access Toolbar** page, select **View** from the **Choose commands from** drop-down to see a list of commands that appear in the **View** tab.



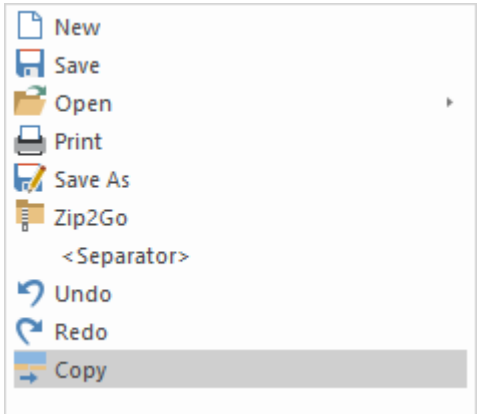
4. Select **Redo** in the right-hand pane. The command you add to the QAT will appear below this selection.



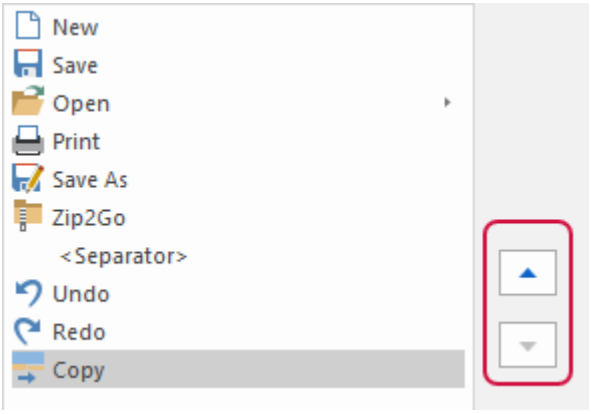
5. Select **Copy** from the **Commands** list, and then click **Add**.



The command appears below the **Redo** command in the right-hand pane.



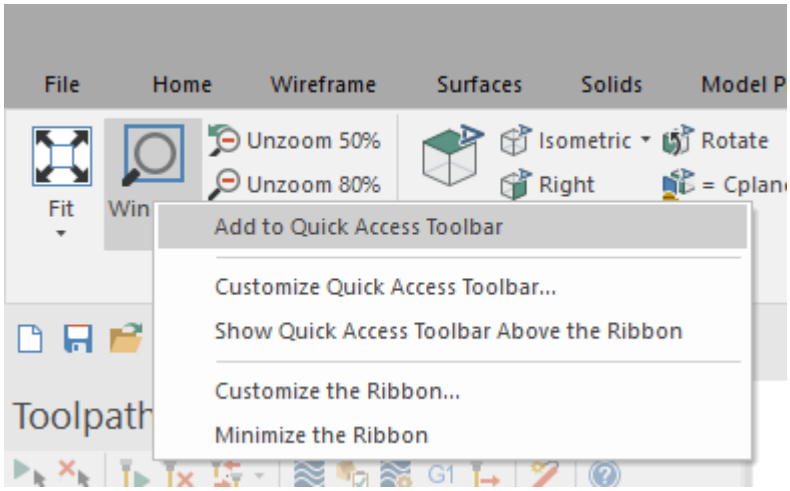
You can use the up and down arrows to rearrange the list.



NOTE
The **Commands** list is in alphabetical order.

6. Click **OK**, and the **Copy** command shows in the QAT.

Alternatively, you can add any command in the ribbon to the QAT by right-clicking the command in the ribbon and selecting **Add to Quick Access Toolbar**.



7. Return the QAT to its placement above the ribbon.

Exercise 4: Using Mastercam Resources

Your Mastercam license gives you access to resources that can improve your project and advance your Mastercam skills. Learn about using Mastercam and connect with the community to share ideas.

These resources include:

- **myMastercam:** Connect to the Mastercam community to share ideas, get news, seek assistance, and download the latest versions.
- **Feedback:** Give feedback about your Mastercam experience.

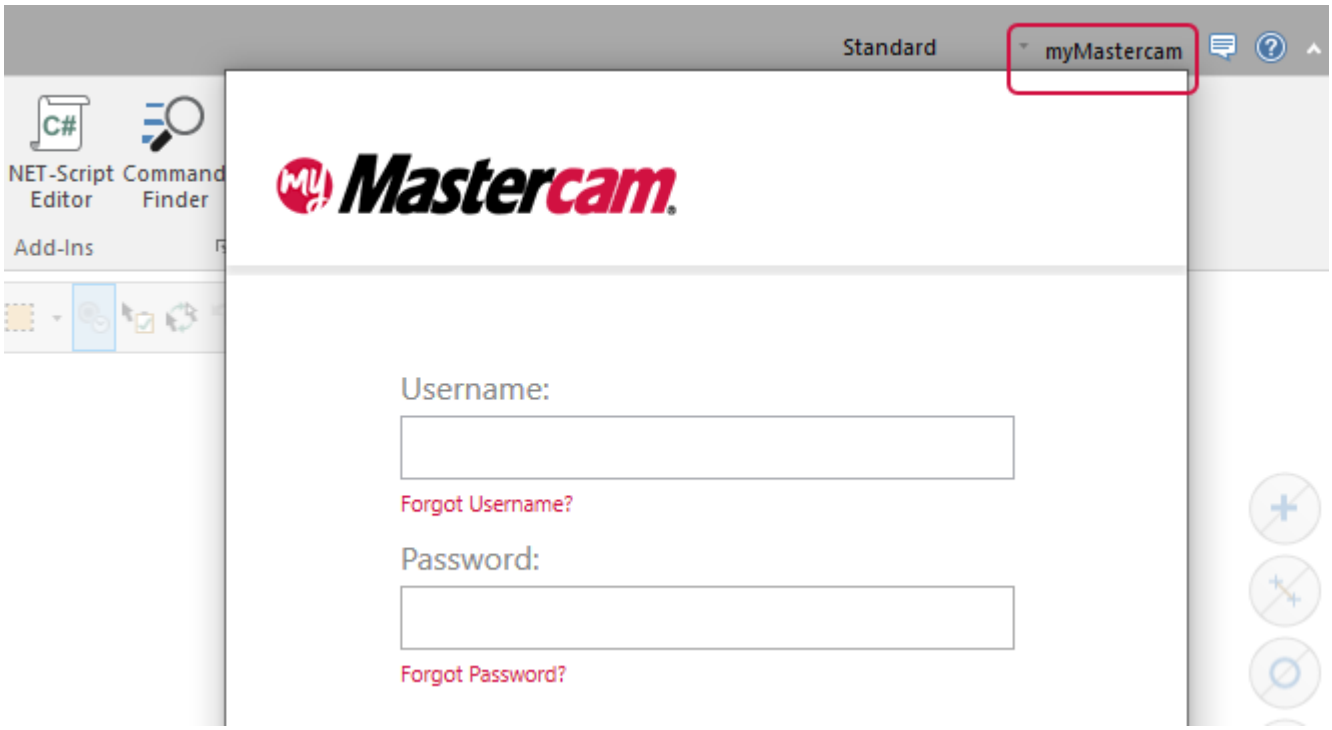
- **Help:** Use Help to learn about Mastercam functions, gain information about parameters, and expand your concept knowledge.
- ***What's New* and *Resolved Issues*:** Read the *What's New* to learn about the latest features in a release and the *Resolved Issues* to keep up-to-date on fulfilled enhancement requests.
- **Tutorials:** Bolster your Mastercam skills with step-by-step tutorials that lead you through many of Mastercam's products.

In this exercise you visit each resource and explore the options available to you.

NOTE
You must have an internet connection to access myMastercam and its associated links.

Using myMastercam

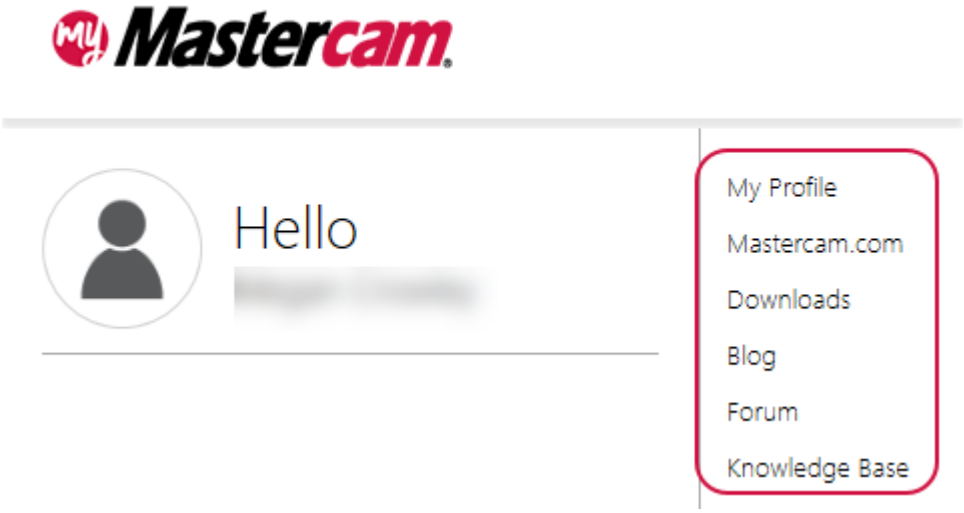
1. Click **myMastercam** to open the login dialog box.



2. Enter your credentials and click **Login**.

NOTE
If you do not have a myMastercam account, create one at https://users.mastercam.com/signup.

Your myMastercam account loads. Use this feature to quickly access the CAD/CAM community, Mastercam downloads, and your personal account.

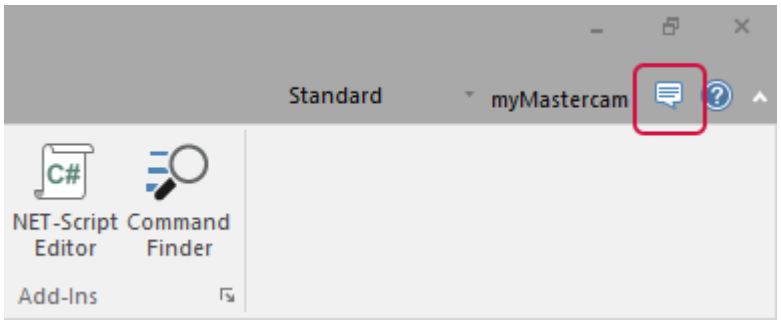


You can also use this feature to access Mastercam social media sites.

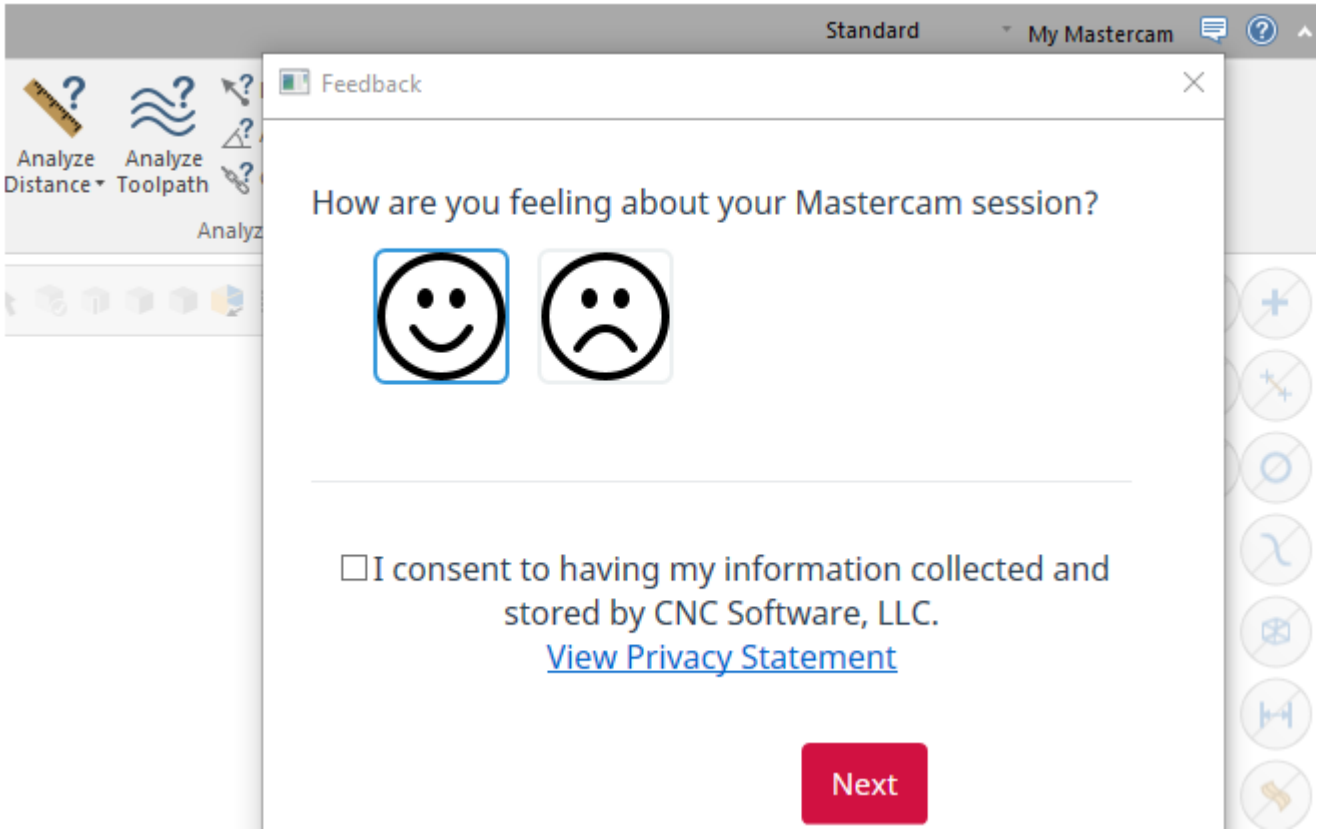
3. Click outside of the account dialog box to close it.

Giving Feedback in Mastercam

1. Mastercam allows you to give feedback about your experience with the software. Select **Feedback** in the upper right corner of Mastercam.



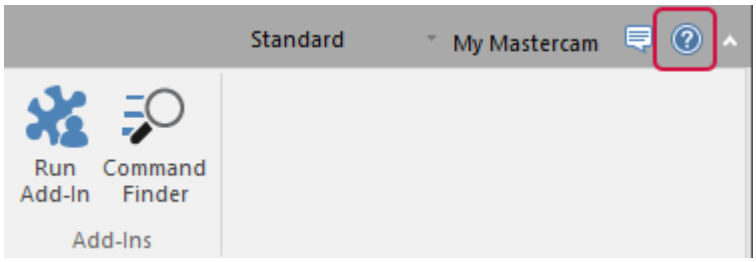
The in-app feedback browser opens.



2. Follow the prompts to submit your feedback.

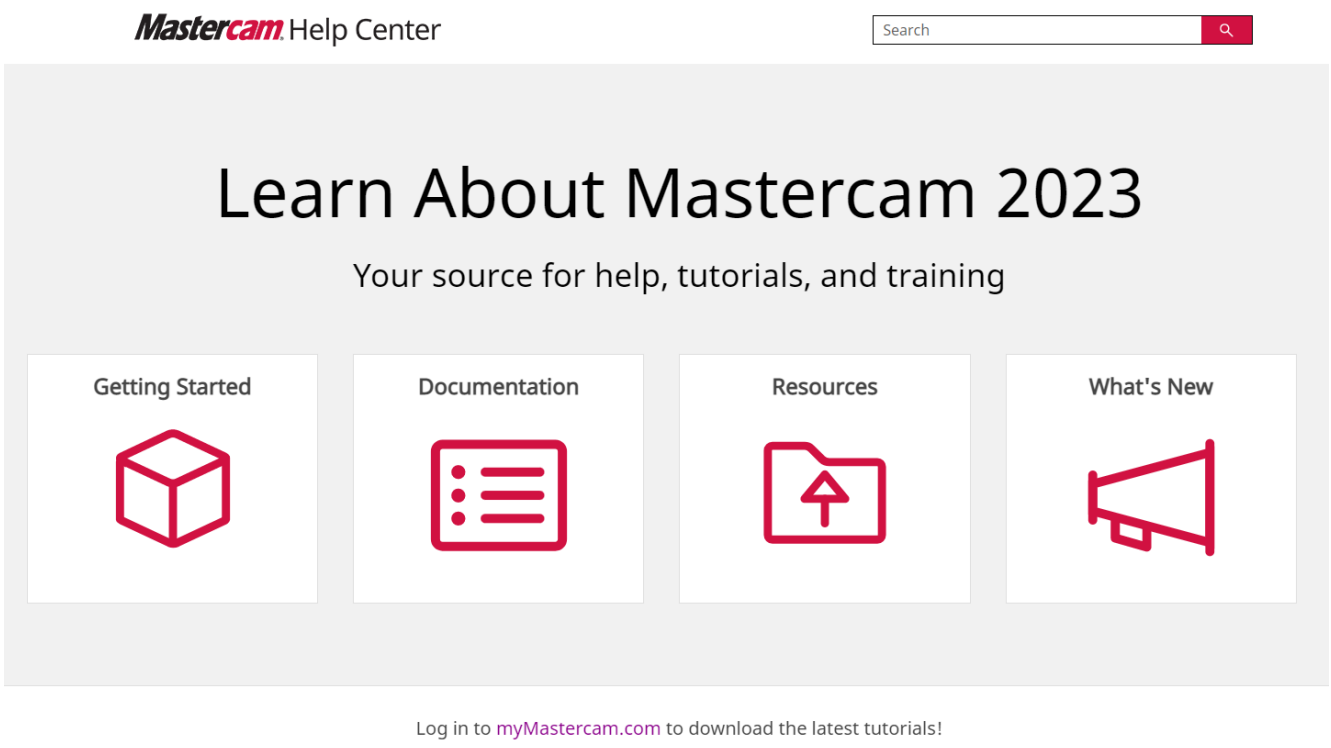
Using Mastercam Help

1. Mastercam Help is an excellent resource for learning about the software's functions. To access the Mastercam Help Center, select the **Help** icon in the upper right corner of Mastercam.



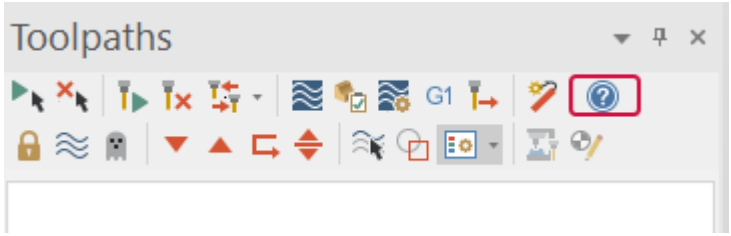
> **NOTE**
> You can also access the Mastercam Help Center by selecting **File**, **Help**, **Contents** in Mastercam's Backstage.

The Mastercam Help Center opens in your internet browser.



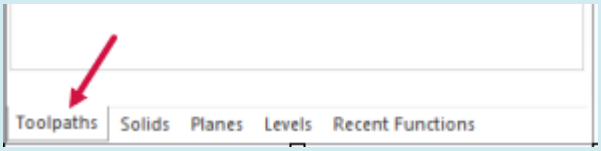
Use the Mastercam Help Center to access information about CAD/CAM topics and other Mastercam resources such as the *What's New*, *Resolved Issues*, and tutorials. Click on each of the tiles to explore your options.

2. When you are done exploring the Mastercam Help Center, return to Mastercam.
3. Context-sensitive Help is related to the location in which you click the Help button. To view an example of context-sensitive Help, click the **Help** icon in the **Toolpaths Manager**. Use context-sensitive Help topics to learn about the purpose and options for the dialog box in which you click **Help**.



NOTE

If necessary, select the **Toolpaths** tab of the Managers.

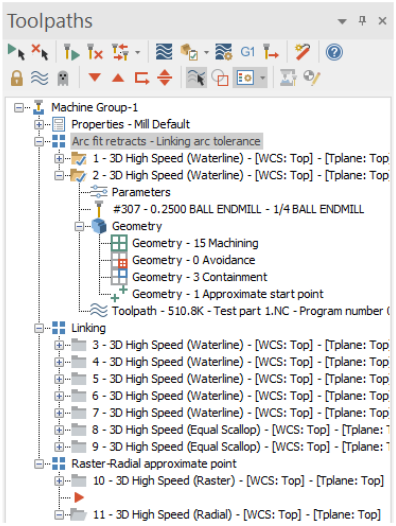


Mastercam Help Center

Search

Toolpaths Manager

The Toolpaths Manager is a central location where you can manage all the elements of the current job. Use the Toolpaths Manager to generate, sort, edit, regenerate, verify, backplot, and post any operation.

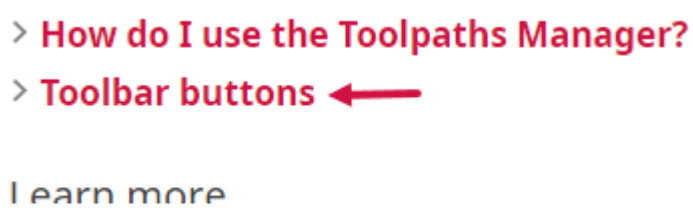


> **How does the Toolpaths Manager affect my part file?**
> **How do I use the Toolpaths Manager?**
> **Toolbar buttons**

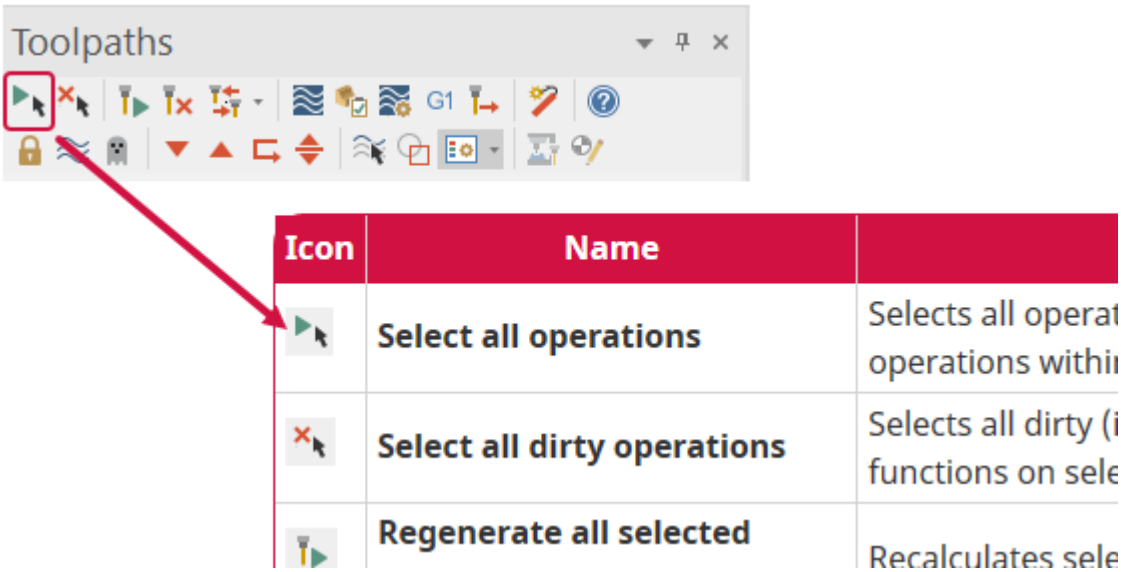
Learn more

- Toolpath Display
- Toolpaths Manager Right-click Menu
- Toolpaths Manager Icons
- Toolpaths Manager Shortcut Keys

4. In the topic, select **Toolbar buttons**.

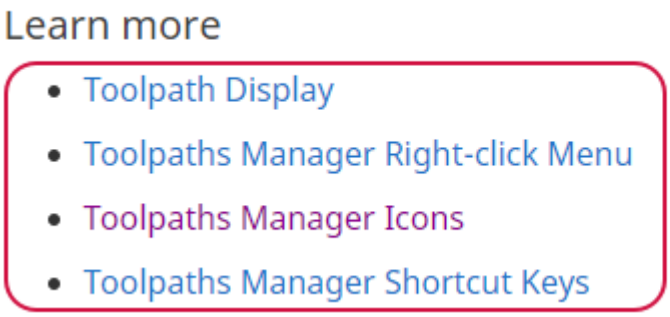


The expanded Help menu drops down to identify and describe Mastercam interface icons. Drop-down headings in Help topics may also describe menu options and procedures.



When using Mastercam, context-sensitive help is an excellent resource for learning more about interface options.

5. Scroll down to the bottom of the topic to review the links in the **Learn more** section. This section is located at the bottom of each help topic with information about similar topics. Use these links to enhance your understanding of CAD and CAM concepts.



6. When you are done exploring the **Learn more** links, return to Mastercam.

Exercise 5: Exploring the Ribbon Interface

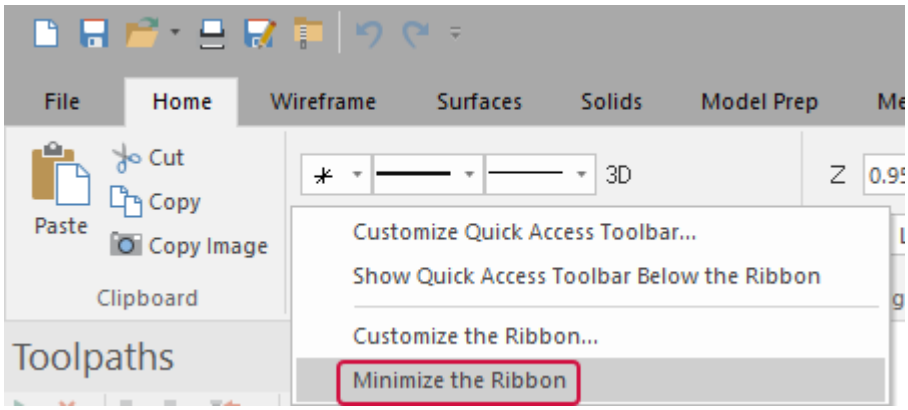
The ribbon interface comprises multiple tabs that group together related functions and controls. Tabs are organized in a simple-to-complex workflow from left to right.

Some tabs are contextual and only display when needed in the workflow. For instance, you must select a piece of wireframe geometry for the **Wireframe Selection** contextual tab to display on the ribbon.

You can choose to show all available ribbon functions, or simplify your view by showing only the most commonly used functions.

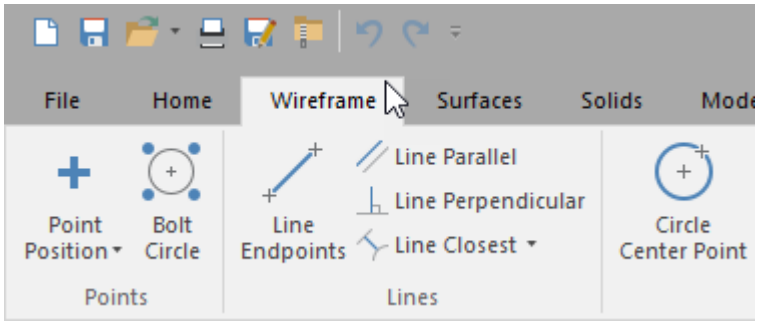
In this exercise you personalize the ribbon tabs.

1. Right-click the ribbon, and select **Minimize the Ribbon**.

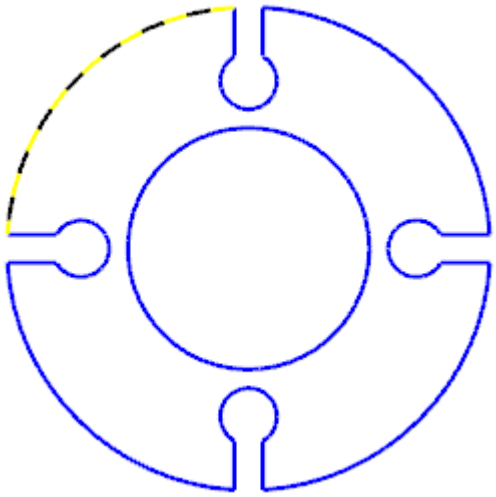


Only the tab names display in the interface.

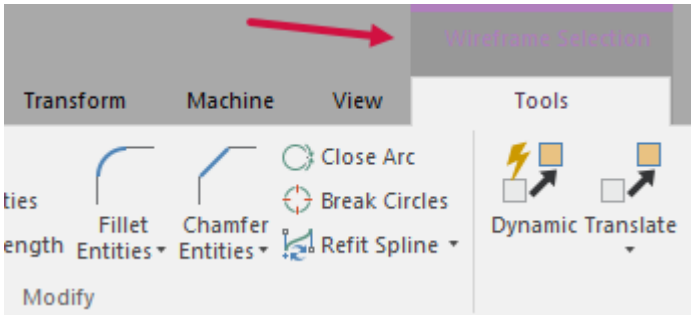
2. Click any tab to display it. Click in the graphics window to hide the tab again.



3. To redisplay the full-sized tabs, simply double-click any tab or right-click the ribbon and deselect **Minimize the Ribbon**.
4. Open BLANK_PART.mcam, which was provided with this tutorial. If you see a message for switching units from inch to metric, click **OK** to allow the change.
5. Select the geometry shown below.

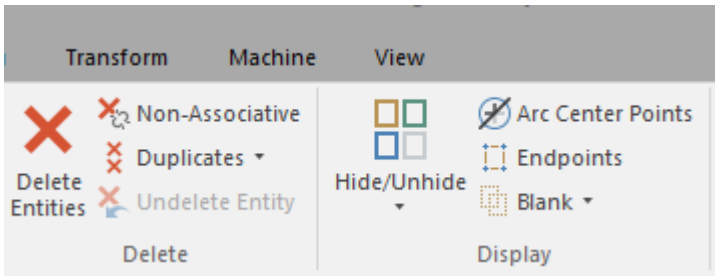


6. The **Wireframe Selection** contextual tab displays. This tab contains Mastercam functions relevant to your selected wireframe geometry.

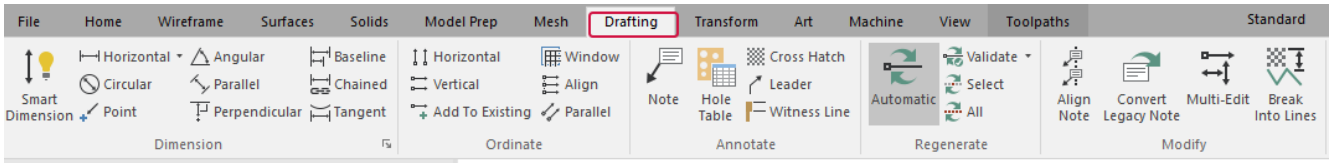


7. Press [**Esc**] to deselect the geometry in the graphics window.

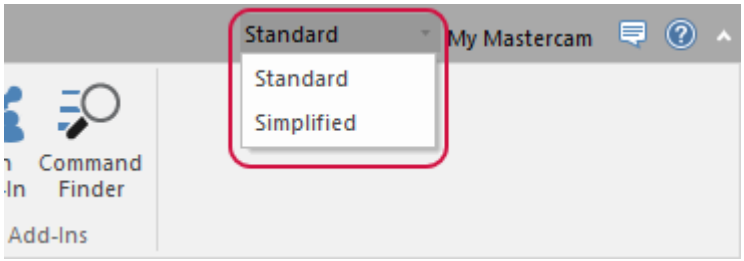
   The **Wireframe Selection** contextual tab disappears from the ribbon.



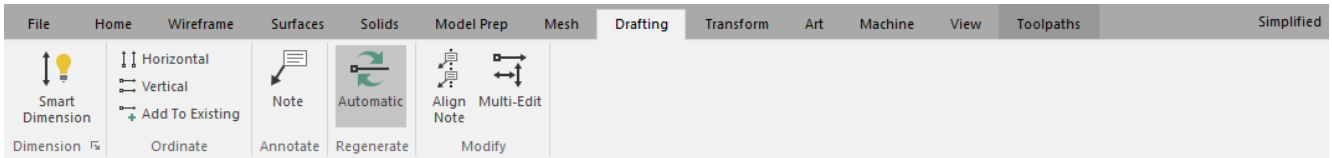
8. Select the **Drafting** tab.



9. Click the **Ribbon mode** drop-down. **Ribbon mode** alters the number of functions displayed on the ribbon.
   - **Standard**: All Ribbon functions display.
   - **Simplified**: The most commonly used Ribbon functions display. Some of the more complex Analyze functions, Drafting functions, Curve and Line functions, Delete functions, and View options are no longer available. Wire and Mill-Turn are not available in Simplified mode.



10. Select **Simplified**. The number of functions on the **Drafting** tab changes. Mastercam remembers your Ribbon mode selection between sessions.

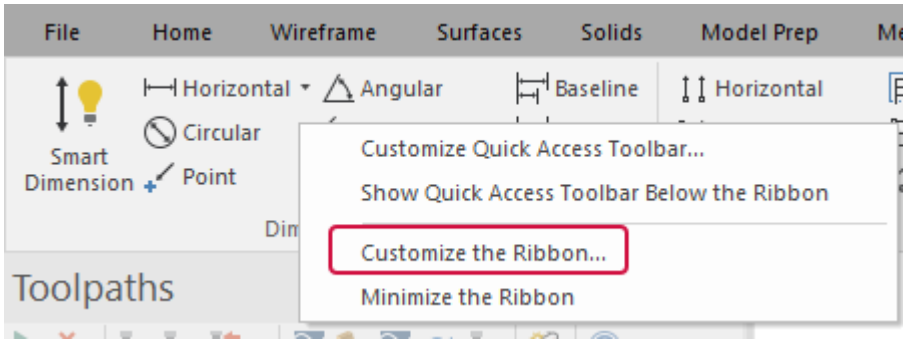


11. Return **Ribbon mode** to **Standard**.

Exercise 6: Customizing the Ribbon

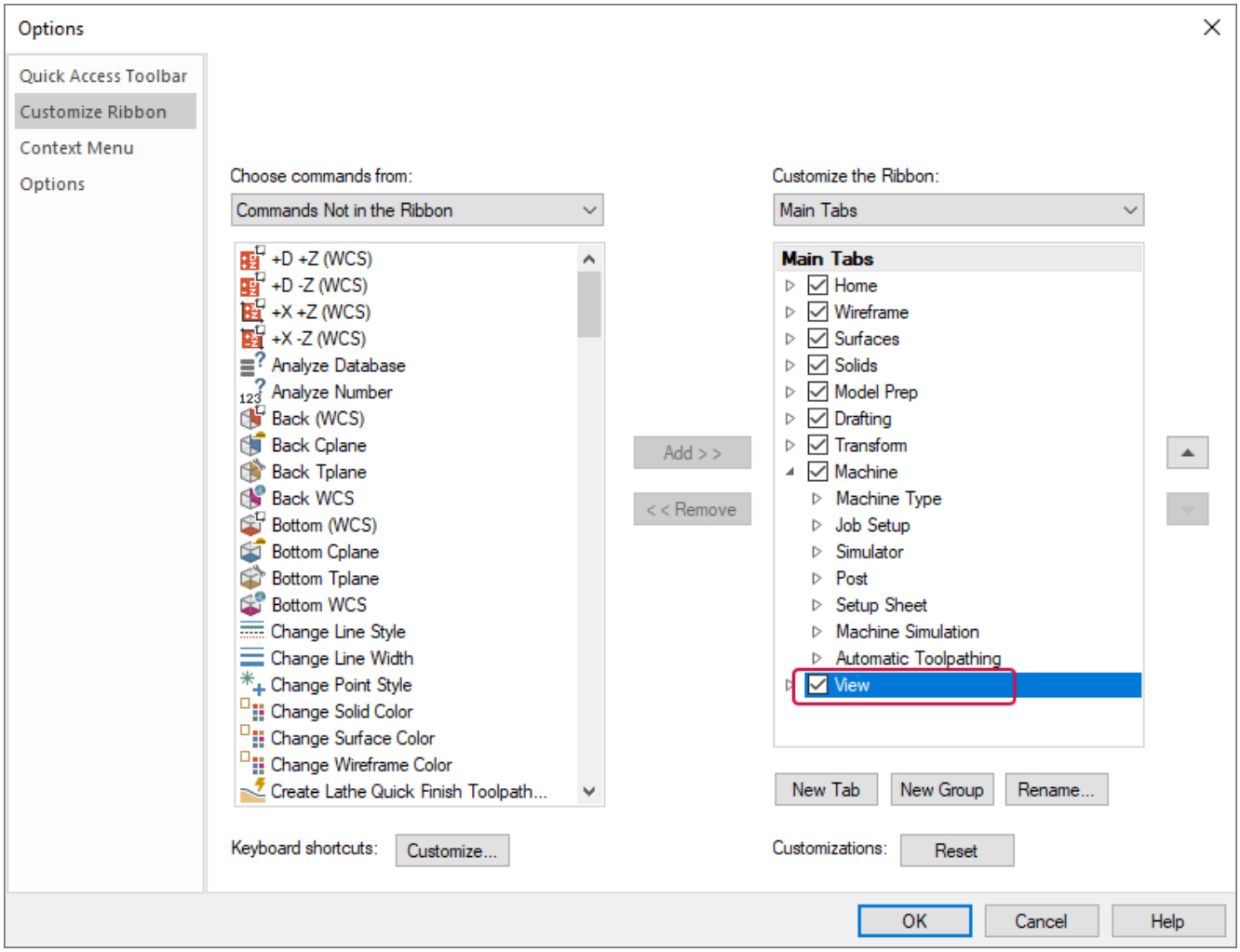
You can create a new ribbon tab and customize it with functions of your choice. In this exercise, you create a new tab, add several functions, and change the tab's position on the ribbon.

1. Right-click the ribbon, and select **Customize the Ribbon**.

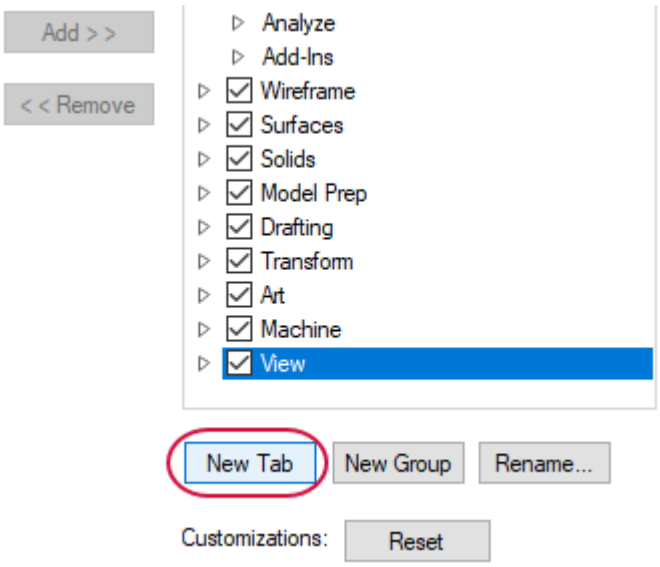


The **Options** dialog box opens.

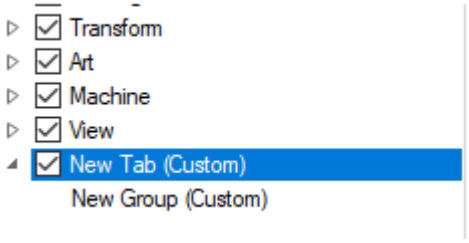
2. If necessary, highlight **View** in the **Customize the Ribbon** list.



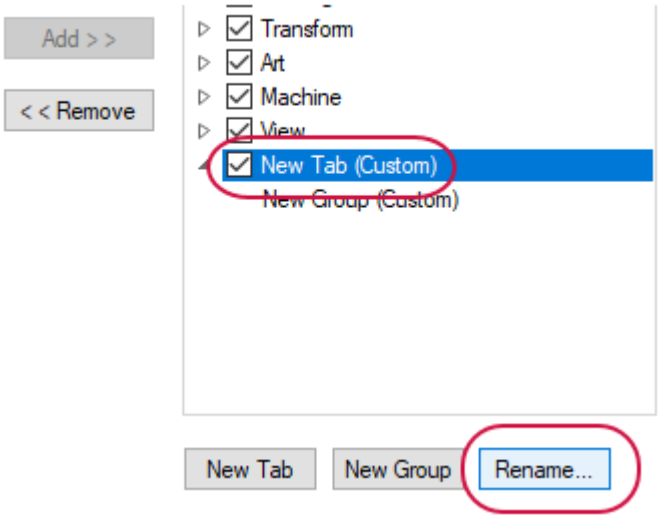
3. Click **New Tab**.



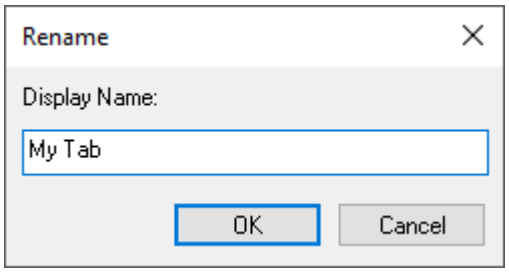
New Tab (Custom) and **New Group (Custom)** appear in the **Main Tabs** list.



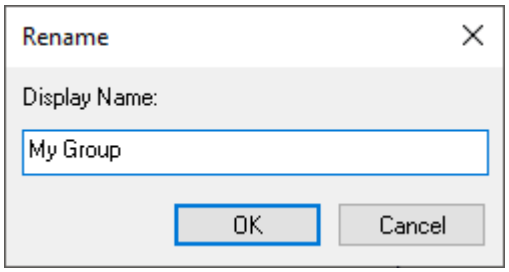
4. Select **New Tab (Custom)** and then click **Rename**.



5. Enter **My Tab** for the **Display Name** and click **OK**.

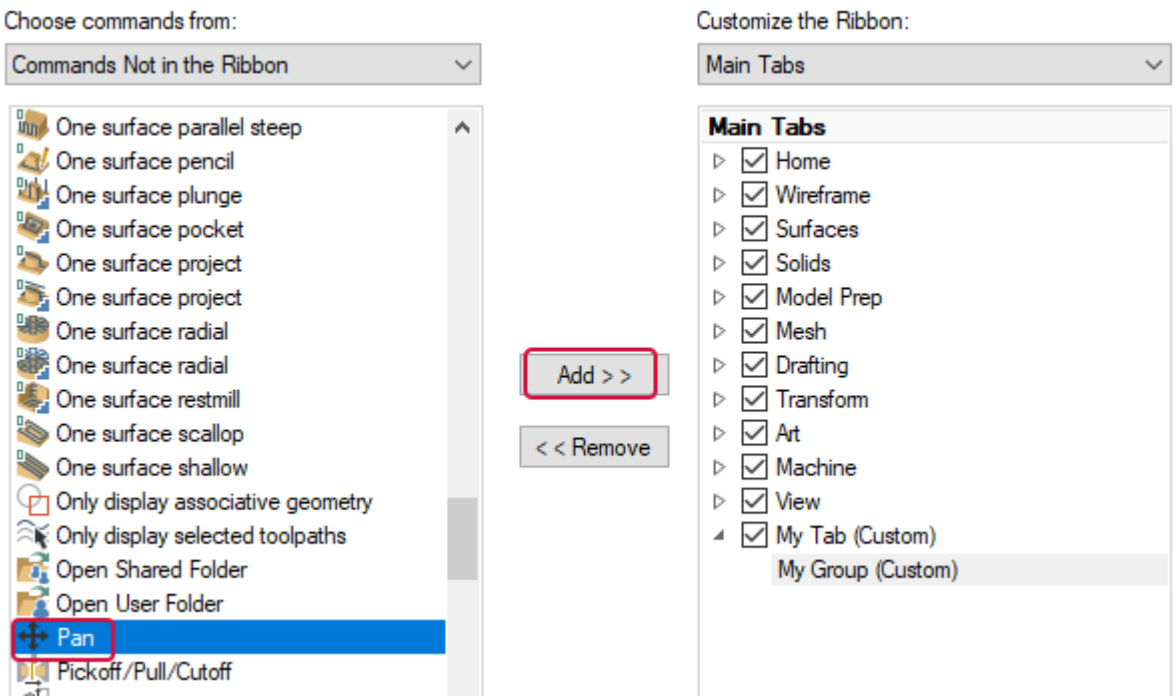


6. Right-click **New Group (Custom)** and select **Rename** from the context menu.
7. Enter **My Group** in the **Display Name** field and click **OK**.

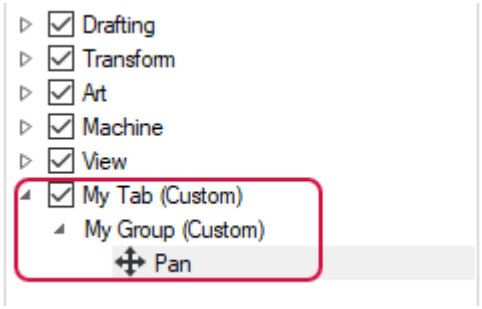


8. Select **Pan** from the **Commands Not in the Ribbon** list, and then click **Add**.

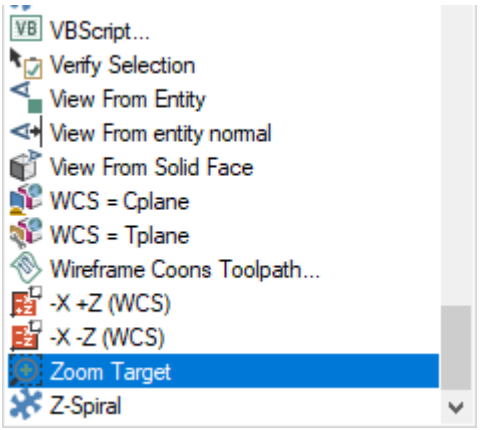
> **NOTE**
> Type the first letter of the command to jump closer to it in the **Choose commands from** list box.



The command is added to **My Group (Custom)**.

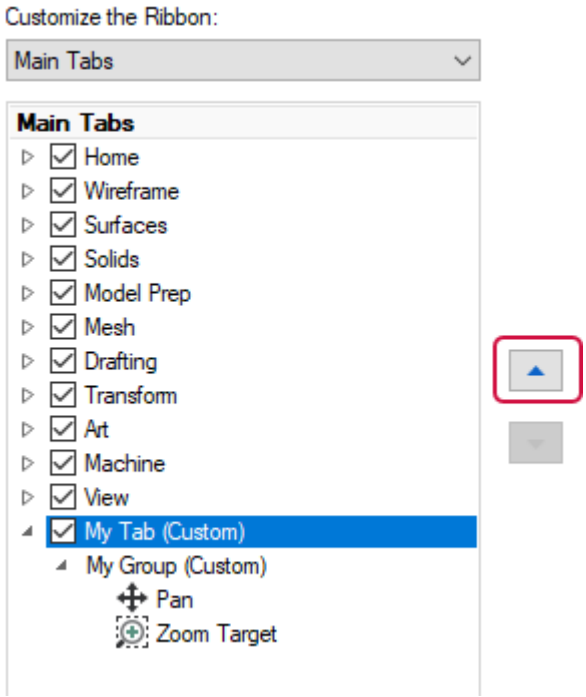


9. Select **Zoom Target** from the **Commands Not in the Ribbon** list, and then click **Add**.



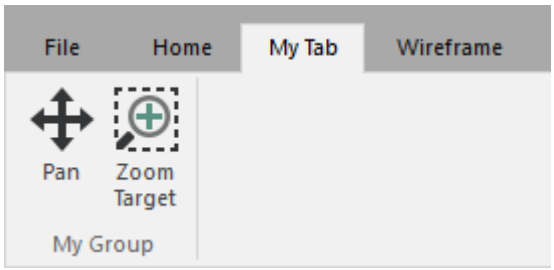
The command is added to **My Group (Custom)**.

10. Select **My Tab (Custom)**, and use the up arrow button to position the tab below the **Home** tab.



11. Click **OK** to close the **Options** dialog box.

12. Choose **My Tab** to view your customized tab with the **Pan** and **Zoom Target** commands.



13. On your own, return to the **Options** dialog box to add more commands to your tab, or rearrange the order of the ribbon tabs.

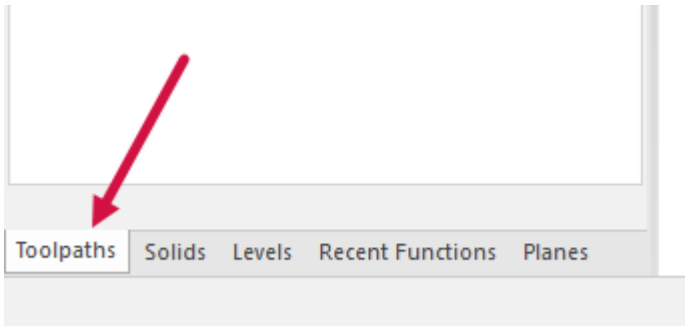
Exercise 7: Moving and Docking Managers

Mastercam has several movable, dockable managers that provide flexible access to frequently used functions including toolpaths, solids, planes, levels, and recent functions.

You can toggle the display of the managers, as well as move one or more managers to a new location. Docking behavior follows standard Windows controls. Dock a manager to a fixed location on the interface, float it anywhere on the workspace, or even move it to another monitor. Mastercam remembers the settings even after you close the application.

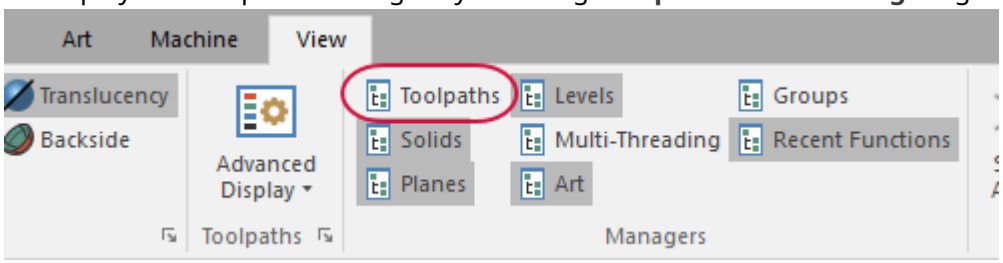
In this exercise, you customize the Mastercam interface by hiding, displaying, and moving the managers.

1. Select **File**, **New** to start with a new part file. Select **Don't Save** if you are prompted to save changes to the current Mastercam part file.
2. Click the **Toolpaths** tab to bring the Toolpaths Manager to the front.



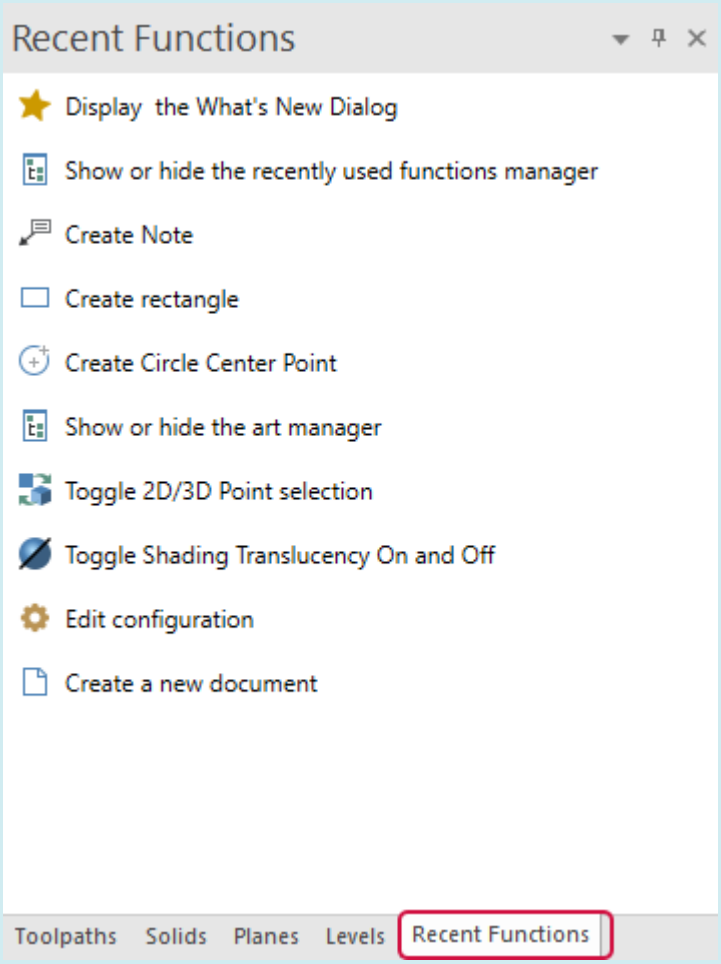
3. Hide the Toolpaths Manager by clicking the **Close** button in its upper right corner.

4. Re-display the Toolpaths Manager by choosing **Toolpaths** in the **Managers** group on the **View** tab.

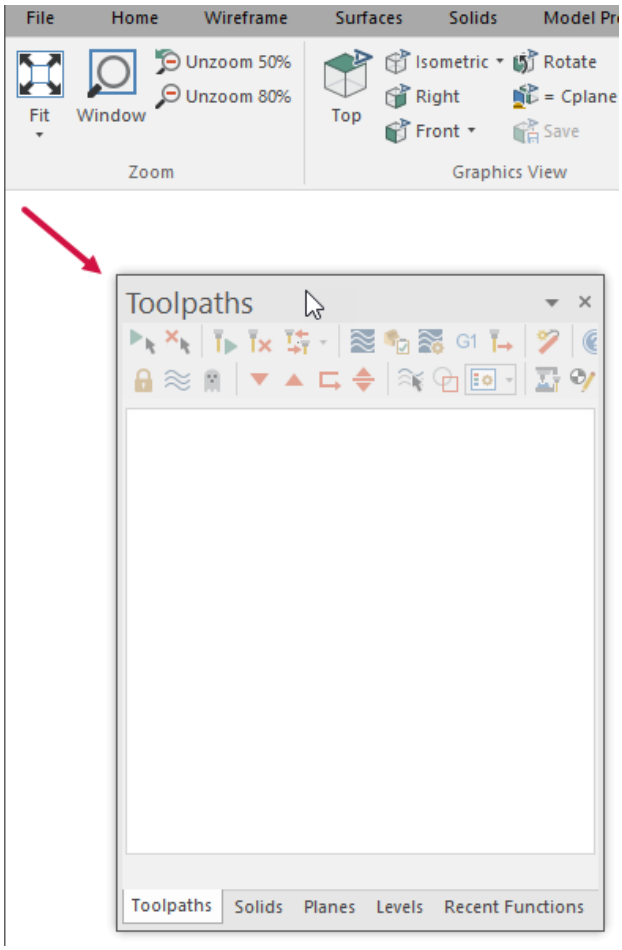


NOTE

The **Recent Functions** panel is not a manager, but it resides in the same location as the Managers for ease of use. Recently used functions are recorded and can be repeated by selecting them from the **Recent Functions** list.



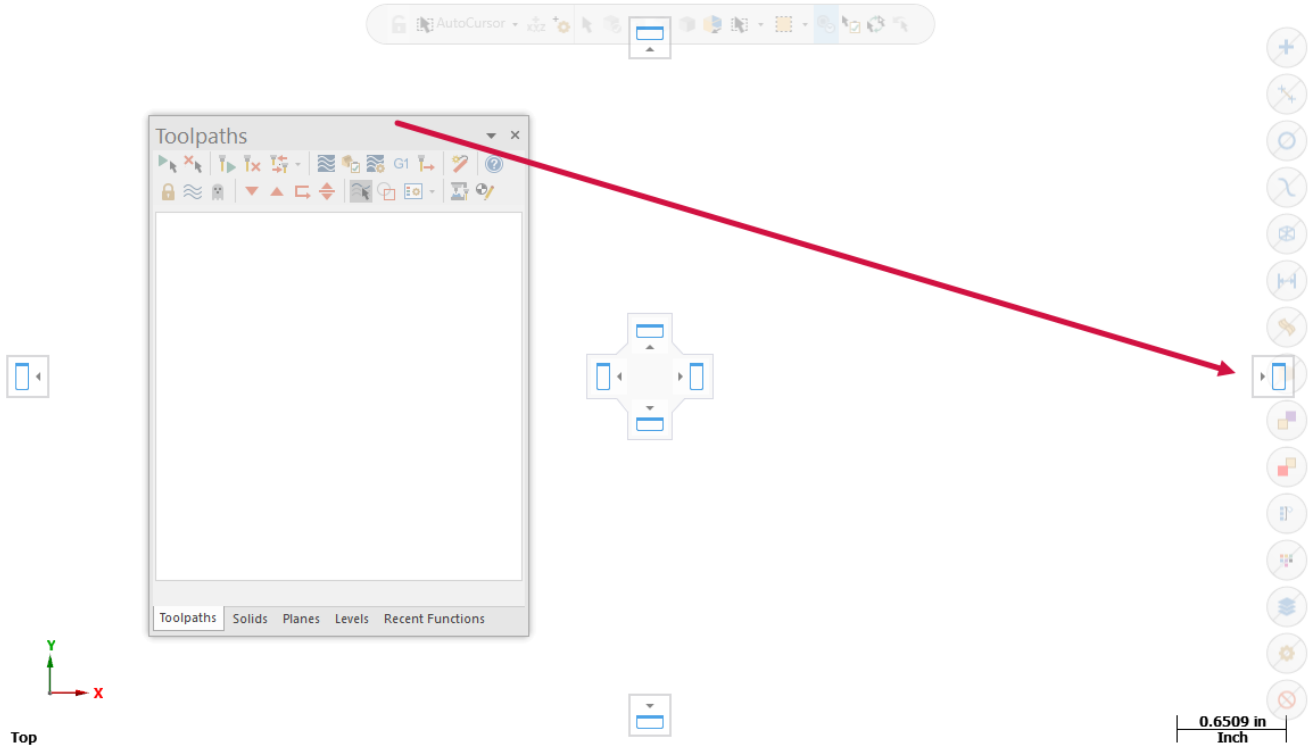
5. Select the title bar of the **Toolpaths Manager**, and drag the panel of managers into the graphics window.



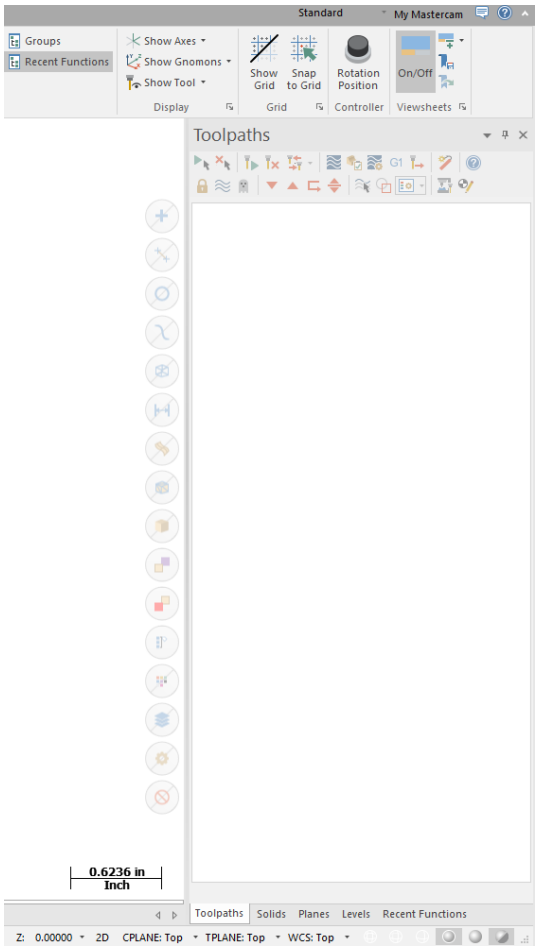
The managers are now floating in the graphics window.

6. Click and drag the managers again. Drop them onto the blue docking icon on the right side of the interface.

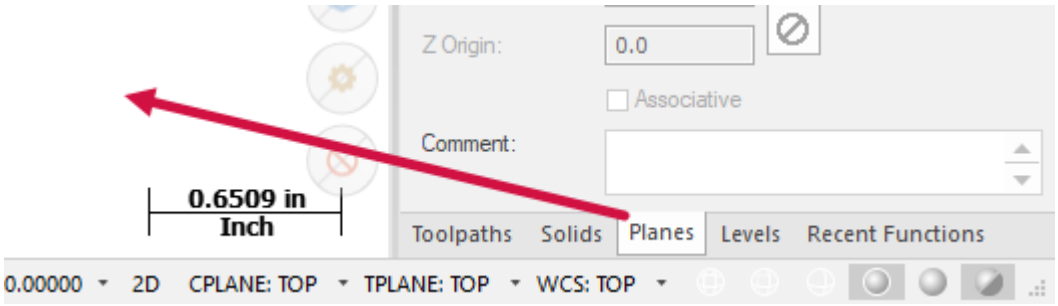
> **NOTE**
> Docking icons snap the managers into alignment with the graphics interface. Hover the stack of managers over a docking icon to see a preview of the new location.



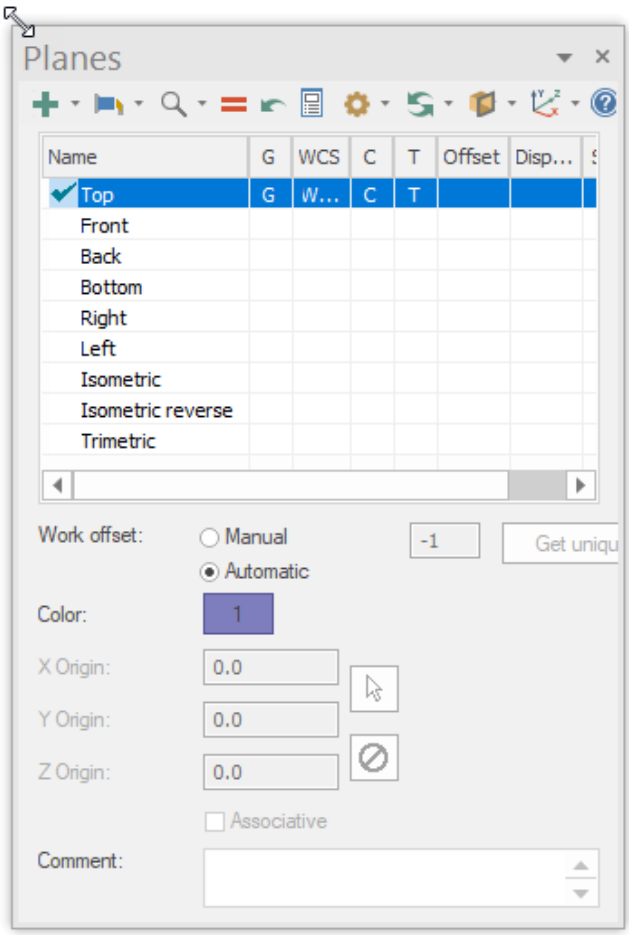
All of the managers are now docked in the new location.



7. Click the **Planes** tab, and drag and drop it anywhere on the workspace (except on a docking icon).



8. Resize the Planes Manager by clicking and dragging any of its edges.



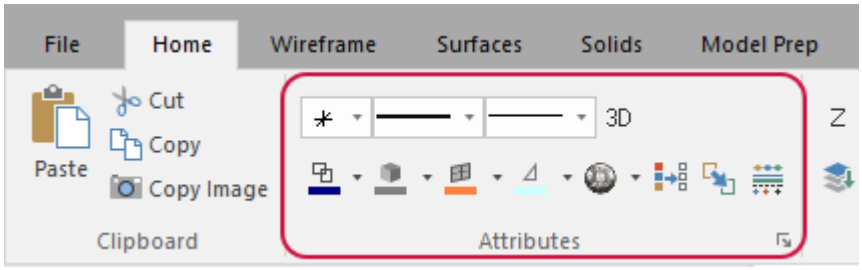
9. Double-click the Planes Manager's title bar to redock it on top of the stack of managers.
10. Click the title bar of the stack of managers, and drag it to the left side of the graphics window, where it was docked at the beginning of this exercise.

    If you work with multiple monitors, you can position the managers onto the monitor that is not running Mastercam to free the entire graphics window for drawing.

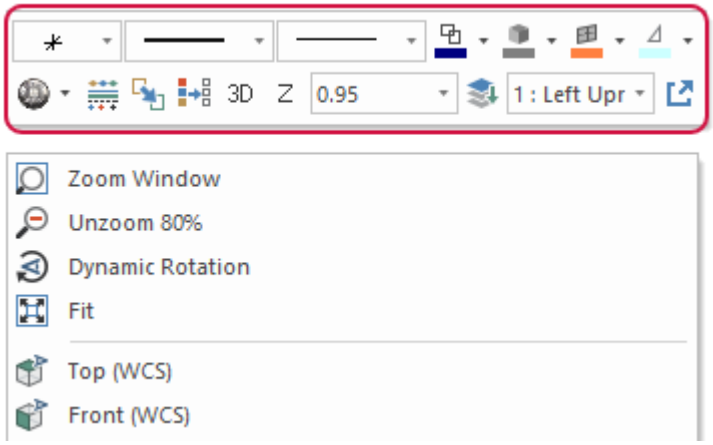
Exercise 8: Setting Entity Attributes in the Current File

Entity attributes control point and line styles, as well as the colors associated with entity types you create in Mastercam. You can change the attributes of selected entities in the graphics window two ways:

1. Use the controls on the **Home** tab.



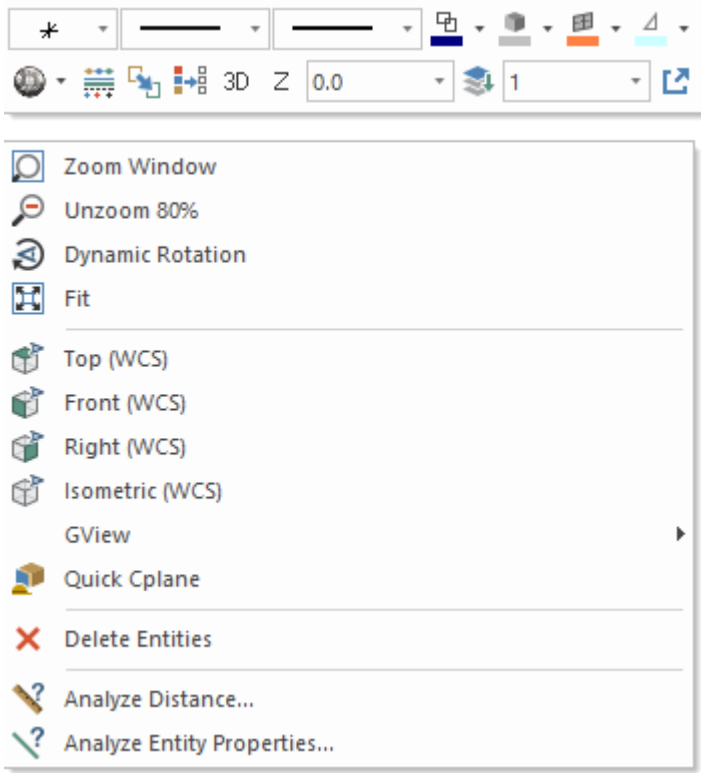
2. Use the right-click mini-toolbar.



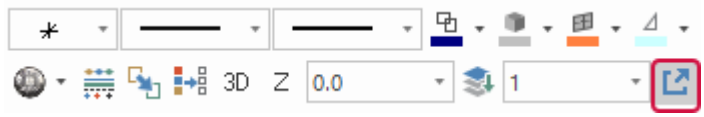
When nothing is selected in the graphics window, you can also set entity attributes for the current file using these controls. To set entity attributes that persist between sessions, use the **Colors** and **CAD** pages of the **System Configuration** dialog box.

In this exercise, you use the attributes controls located on the mini-toolbar.

1. Select **File**, **New** to start with a new part file. If prompted to save BLANK_PART.mcam, choose **Don't Save**.
2. Right-click the graphics window to open the mini-toolbar and context menu.

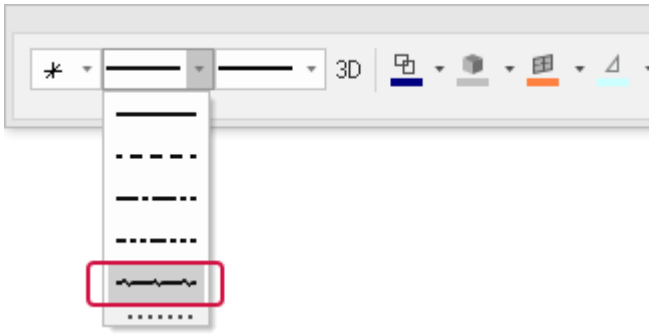


3. Select **Toggle Attributes Panel** on the mini-toolbar.



The mini-toolbar now displays independent of the right-click menu. You can resize and position the mini-toolbar anywhere in the graphics window or anywhere on your desktop.

4. With nothing selected in the graphics window, click the **Line Style** drop-down, and choose the style shown below.

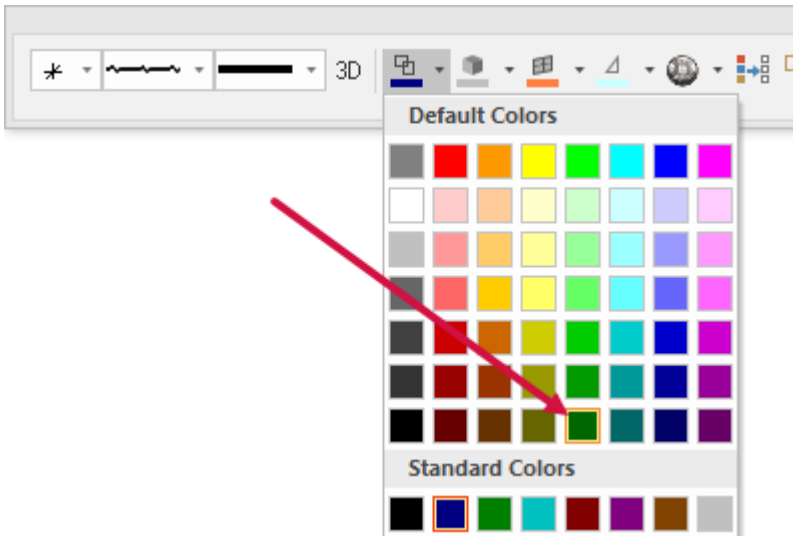


Any new geometry you create uses the line style you select.

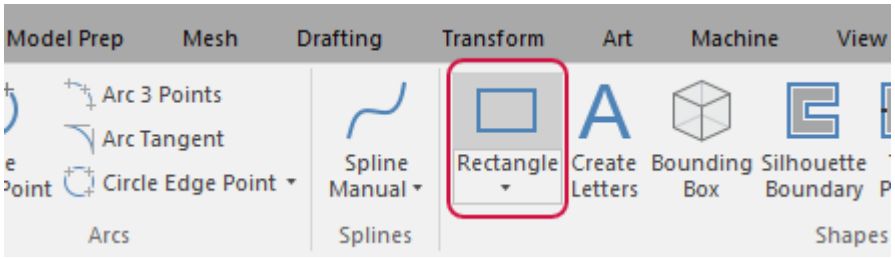
5. Select the **Line Width** shown below.

New geometry you create uses the line width you select.

6. Click the **Wireframe Color** drop-down, and select green as shown below.

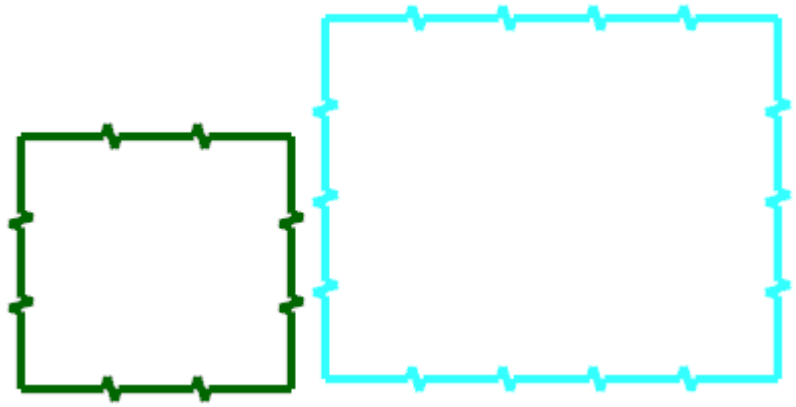


7. On the **Wireframe** tab, choose **Rectangle**.



The **Rectangle** function panel opens.

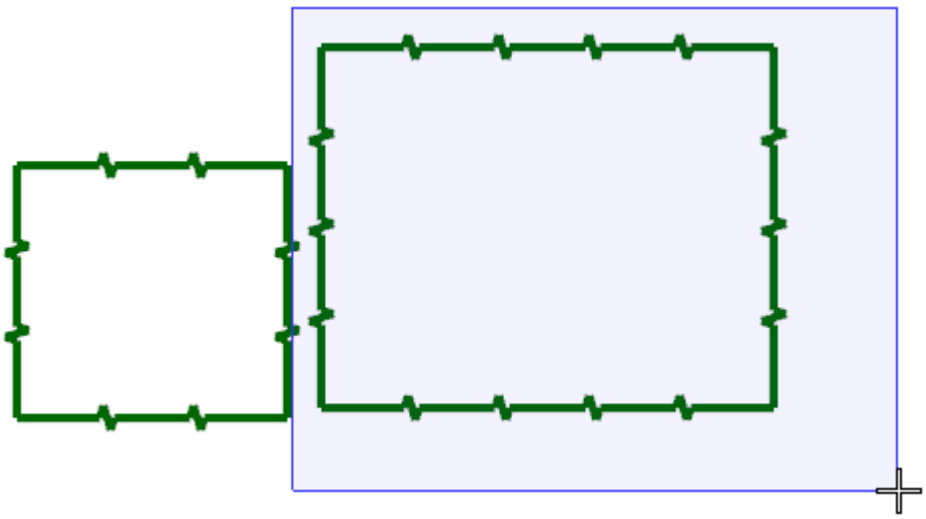
8. Click in the graphics window and draw two rectangles of any size. Follow the prompts in the graphics window.



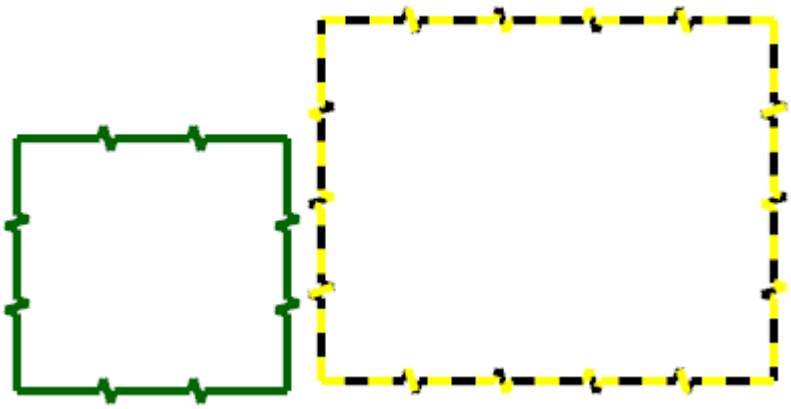
9. Click **OK**.

Your rectangle wireframe geometry is created.

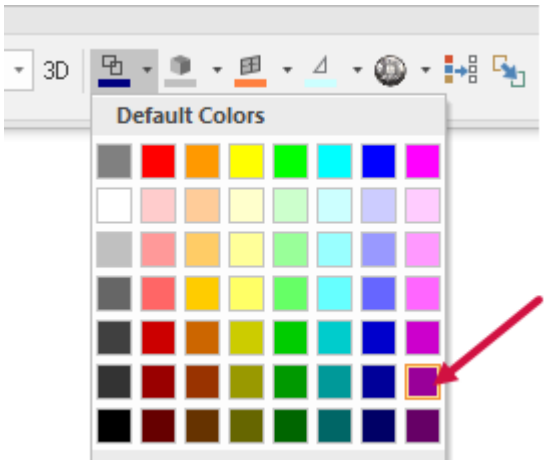
10. Hold down your left mouse button and drag to draw a window around one rectangle in the graphics window.



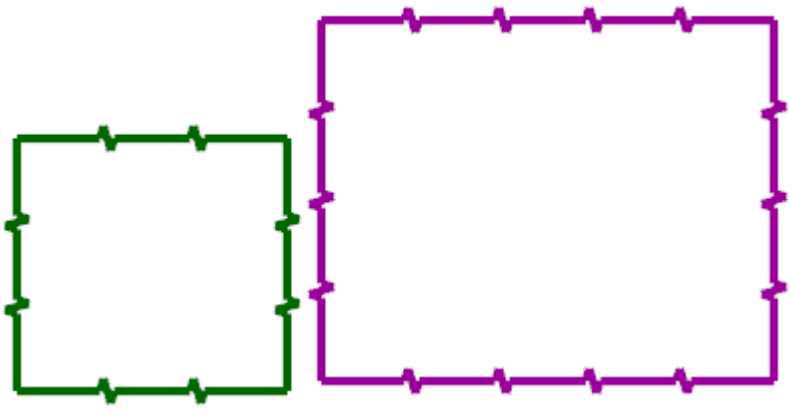
Mastercam selects the rectangle.



11. Click the **Wireframe Color** drop-down and select purple.

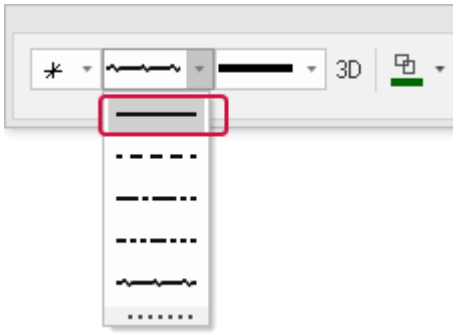


Only the rectangle you selected changes to purple.

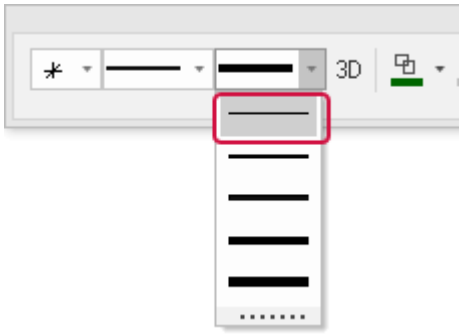


The default wireframe color does not change on the mini-toolbar. Any new wireframe geometry you create still uses the default color, which is green.

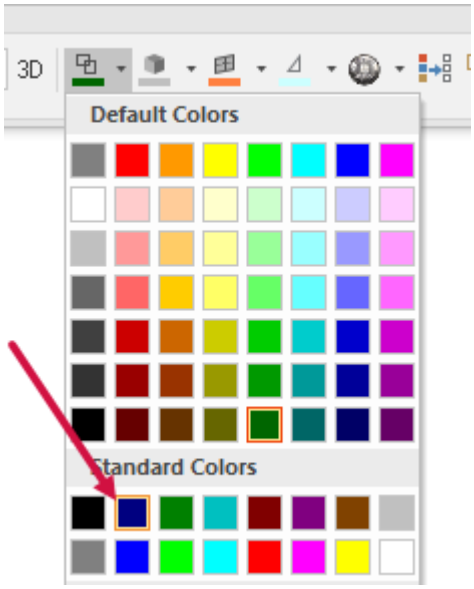
12. With nothing selected in the graphics window, set the **Line Style** to the option shown below. This resets it to default.



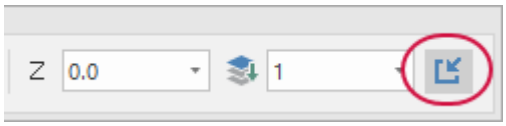
13. Set the **Line Width** to the default option shown below.



14. Set the **Wireframe Color** to the default blue shown below.



15. Select **Toggle Attributes Panel** to return the mini-toolbar to the right-click menu.



16. Click **File**, **Save**. Name your file ATTRIBUTES.mcam, and save it in the same location as the other tutorial files.

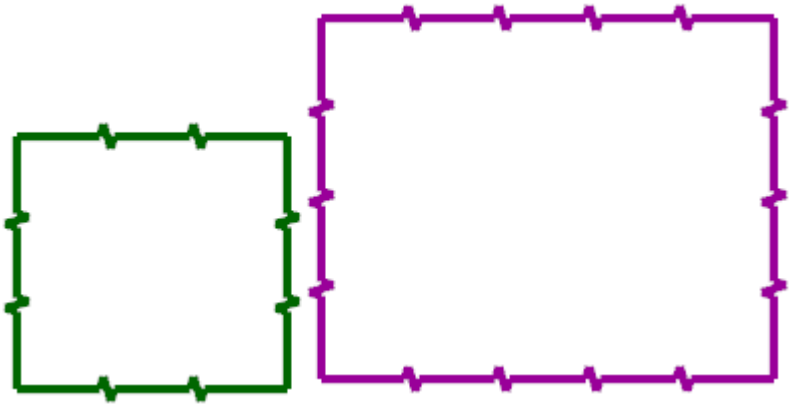
Exercise 9: Understanding the Selection Bar and AutoCursor

The **Selection Bar** combines tools used to select entities in the graphics window and AutoCursor controls. The **Selection Bar** is located across the top of the graphics window. It has two modes, Standard Selection and Solid Selection, which are activated based on the function you are using.

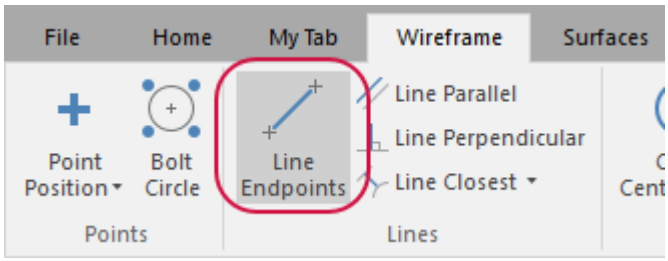
AutoCursor controls allow you to detect and snap to points as you move the cursor over geometry in the graphics window. AutoCursor becomes active whenever Mastercam prompts you to select a position in the graphics window.

In this exercise, you locate points and lines using AutoCursor and Fast Point mode. Use Fast Point Mode when AutoCursor is active to input X, Y, Z coordinates to set an endpoint.

1. If necessary open the file, ATTRIBUTES.mcam, that you saved from the previous lesson.



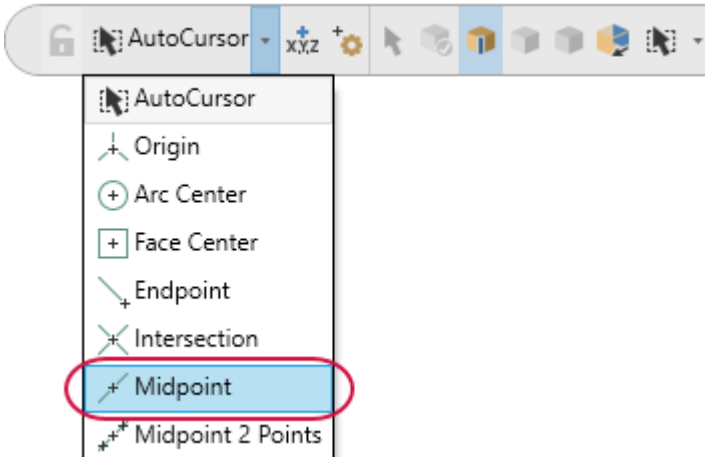
2. On the **Wireframe** tab, select **Line Endpoints**.



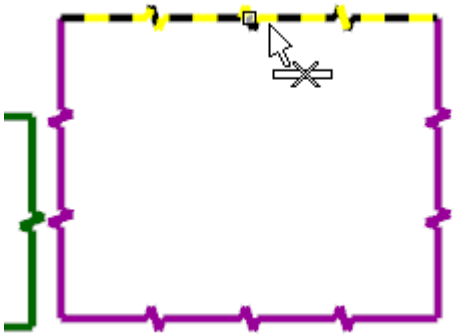
The **Line Endpoints** function panel opens, and Mastercam prompts you to specify the first endpoint.

> **NOTE**
> If you are using multiple monitors, the prompt may appear on a different monitor.

3. Click the **AutoCursor** drop-down and select **Midpoint**.

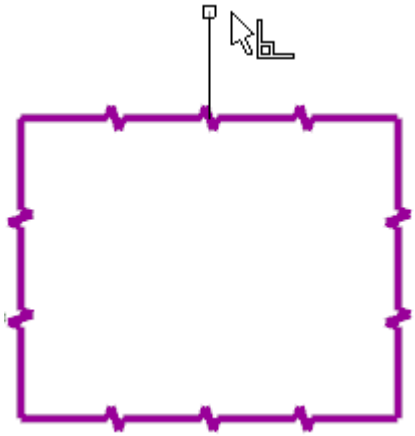


4. Click anywhere on the top line of one of your rectangles and drag up to draw a line.

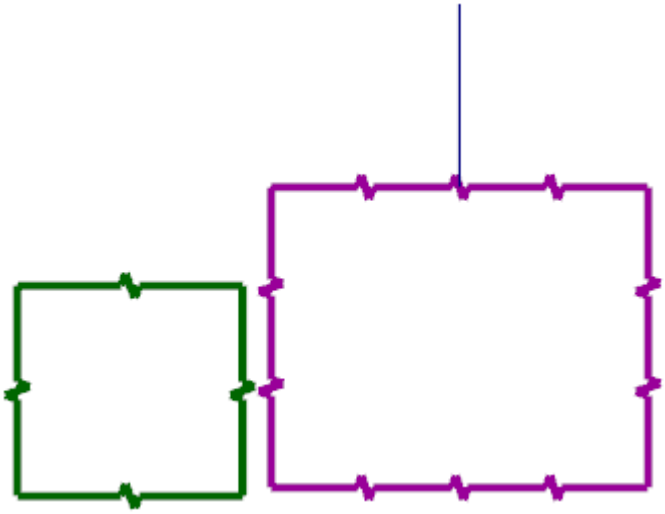


Mastercam uses the midpoint of the entity that you selected as the first endpoint.

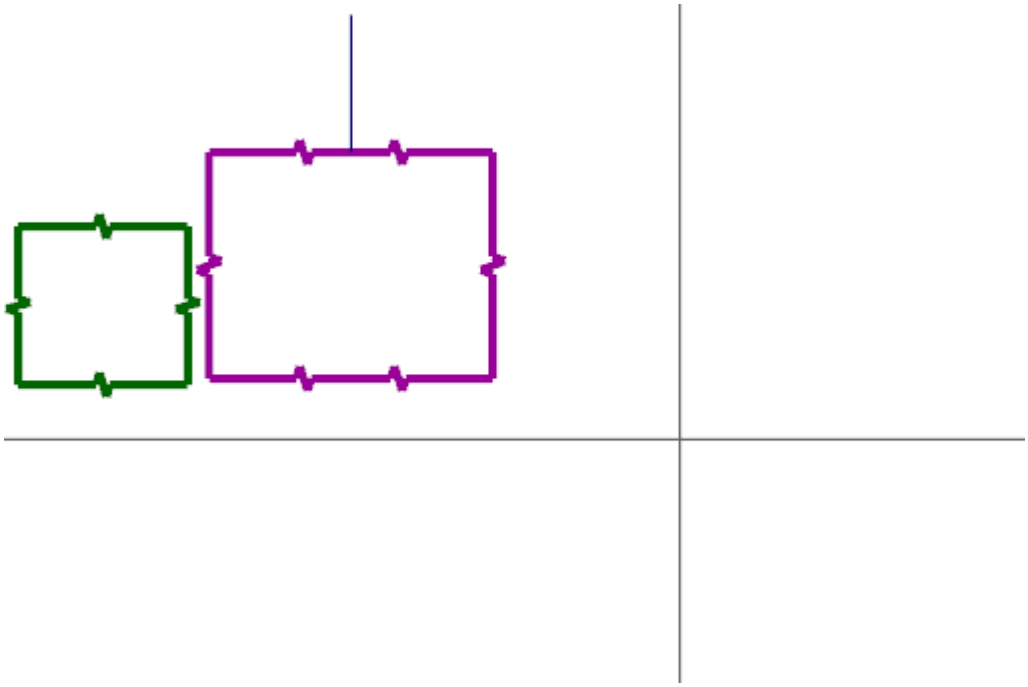
5. Click to set the second endpoint.



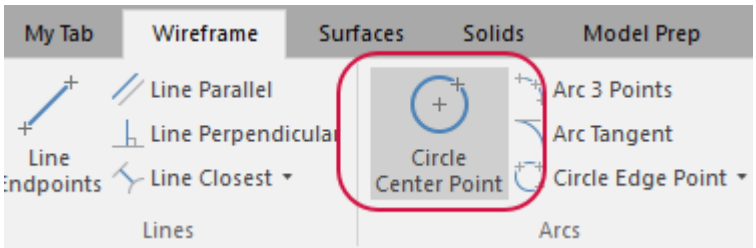
6. Select **OK** in the **Line Endpoints** function panel.



7. Press [**F9**] to show the coordinate axes in the graphics window.

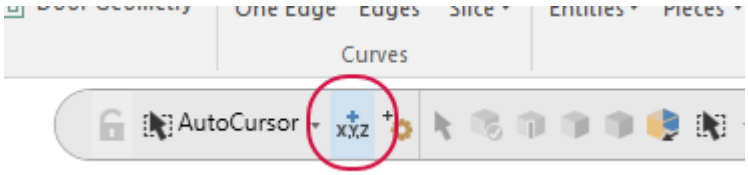


8. On the **Wireframe** tab, select **Circle Center Point**.



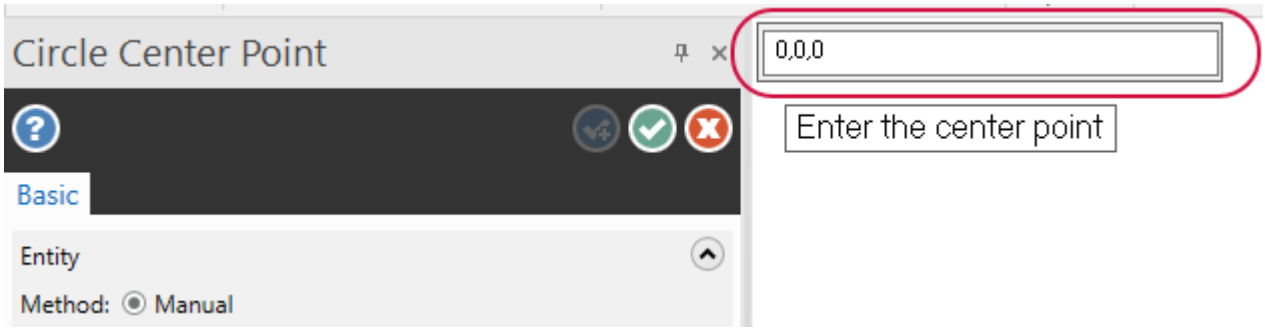
The **Circle Center Point** function panel opens, and Mastercam prompts you to enter the center point.

9. Select **AutoCursor Fast Point** on the **Selection Bar** to display the **Fast Point** field. You can enter numbers, formulas, or X,Y,Z coordinates.

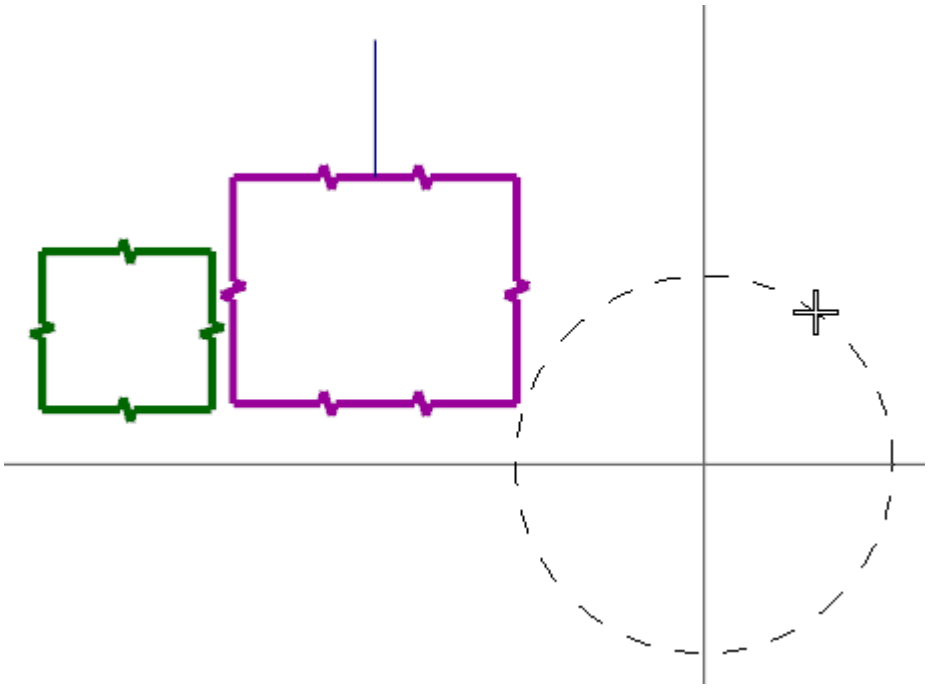


NOTE
You can also press the spacebar to display **Fast Point** mode.

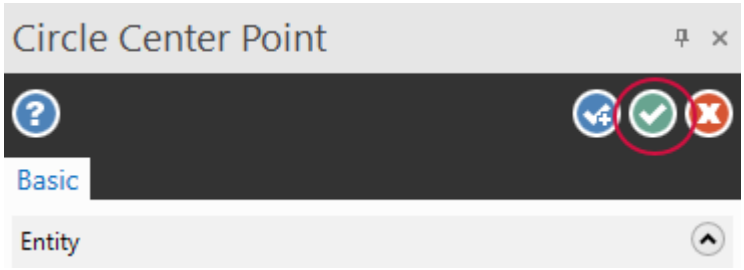
10. Type **0,0,0** to enter the coordinate location of the circle's center point, and then press [**Enter**].



11. Drag and click to set the radius of the circle. The center point is 0,0,0.



12. Choose **OK** in the **Circle Center Point** function panel.



13. If the entities are outside of the graphics window, right-click and choose **Fit**.
14. Press [**F9**] again to hide the coordinate axes.
15. Save your file.

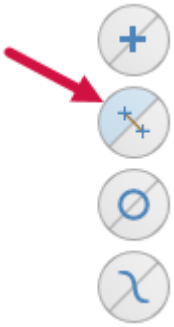
Exercise 10: Using Quick Masks

Mastercam's **Quick Masks** allow you to select all entities of a specific type, or to select only entities of a specific type. Most controls are divided in half. Clicking the left or right side of the control toggles between all/only selection modes. You can select multiple **Quick Masks**. Hover over each control to view the tooltip describing its function.

In this exercise, you select entities using several **Quick Mask** functions.

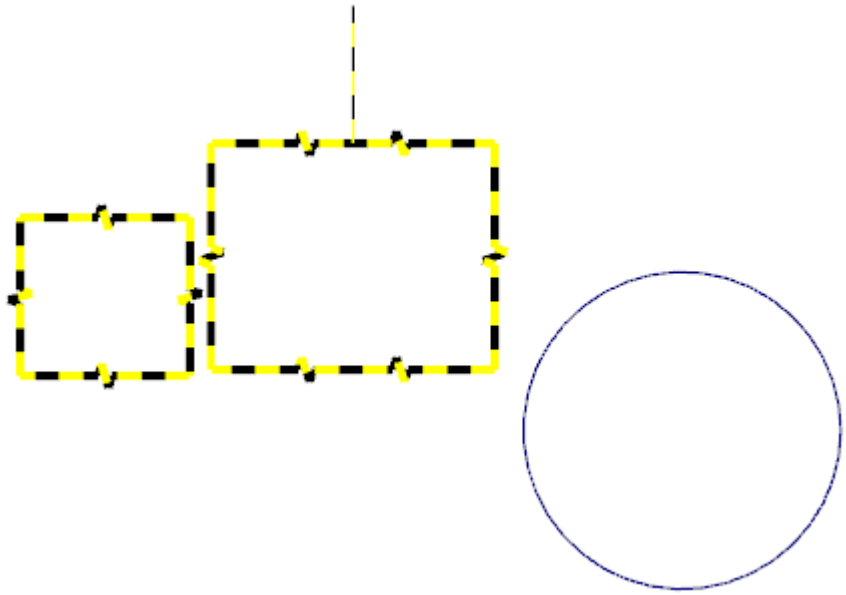
1. If necessary click **File**, **Open** and open ATTRIBUTES.mcam, which you saved in the previous lesson.
2. Click **Clear all masking**.

3. Click **Select all line entities**.



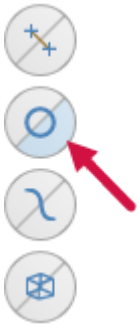
Mastercam selects the rectangles and line that you drew in the previous exercise.

> **NOTE**
> In Mastercam, a rectangle is a set of four lines.

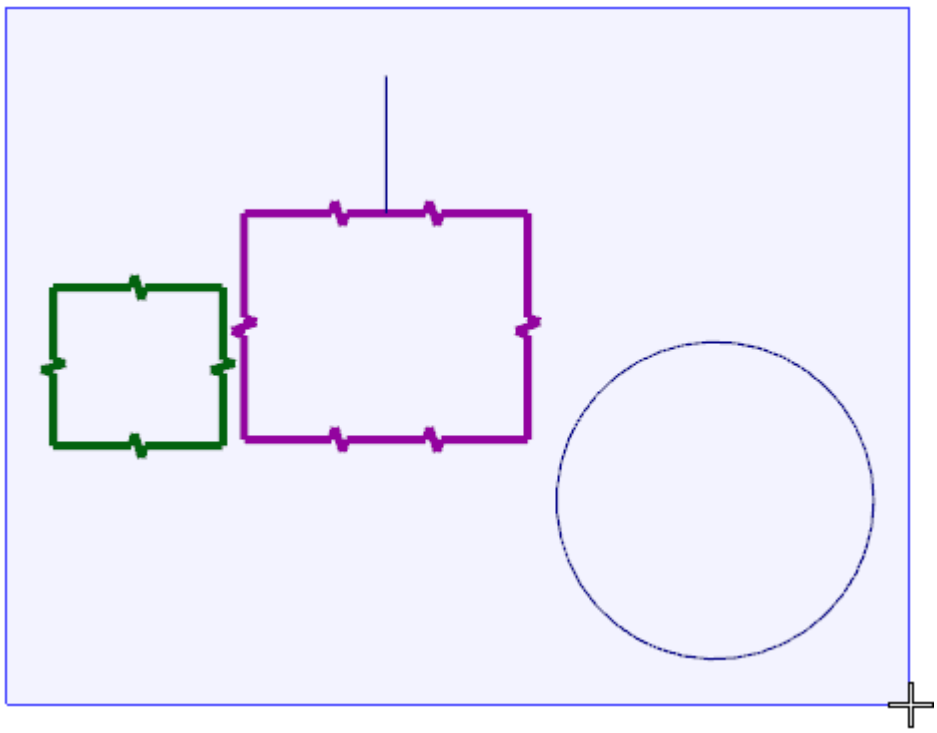


4. Click **Clear selection** to deselect the line entities.

5. Click **Select only arc entities**.

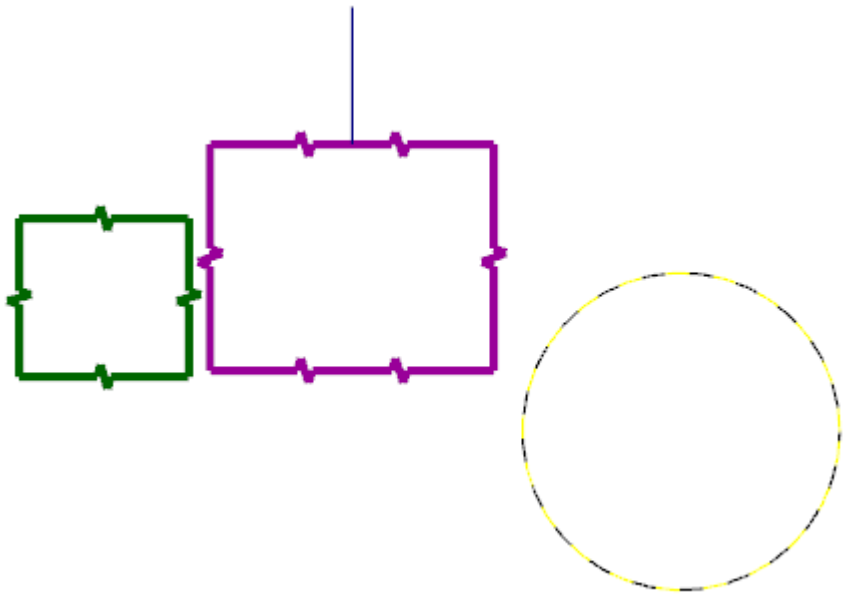


6. Click and drag to window select all entities in the graphics window.



Mastercam selects only the arc.

If there had been arcs outside of the selection window, Mastercam would not have selected them. The **Select only** side of quick mask buttons only selects geometry inside of the selection window.

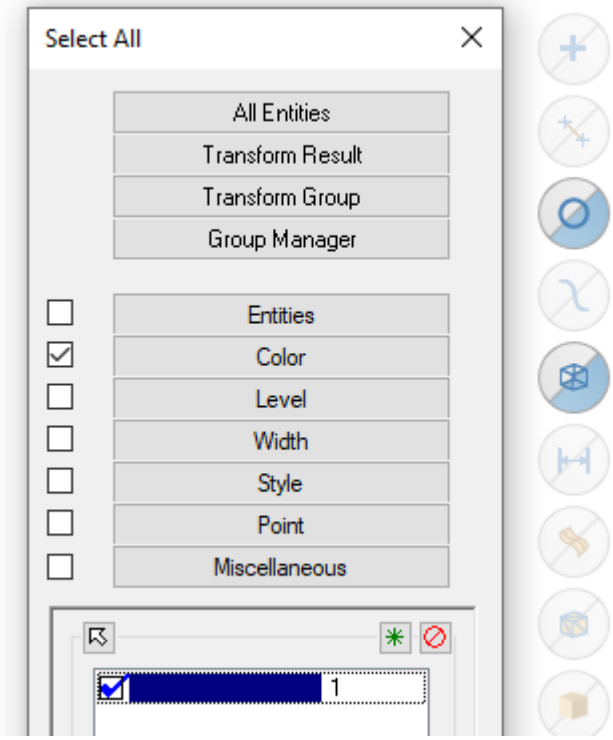


7. Click **Clear selection**.

8. Click **Select all entities by color**.

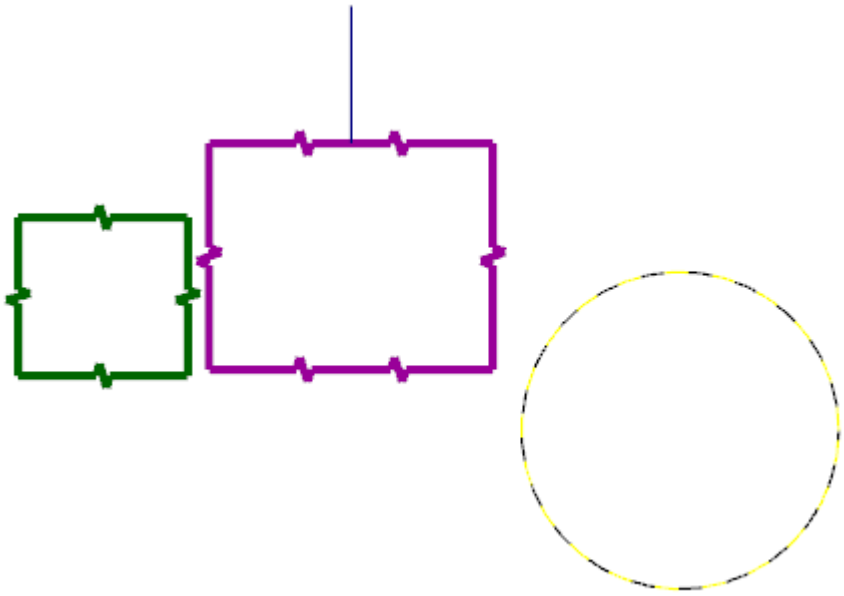
The **Select All** dialog box opens.

9. Select the blue color checkbox, and then click **OK**.



NOTE
Notice that **Select only arc entities** and **Select only wireframe entities** remain selected from the previous steps and affect the outcome.

Mastercam selects only the circle because it is the only blue arc entity.



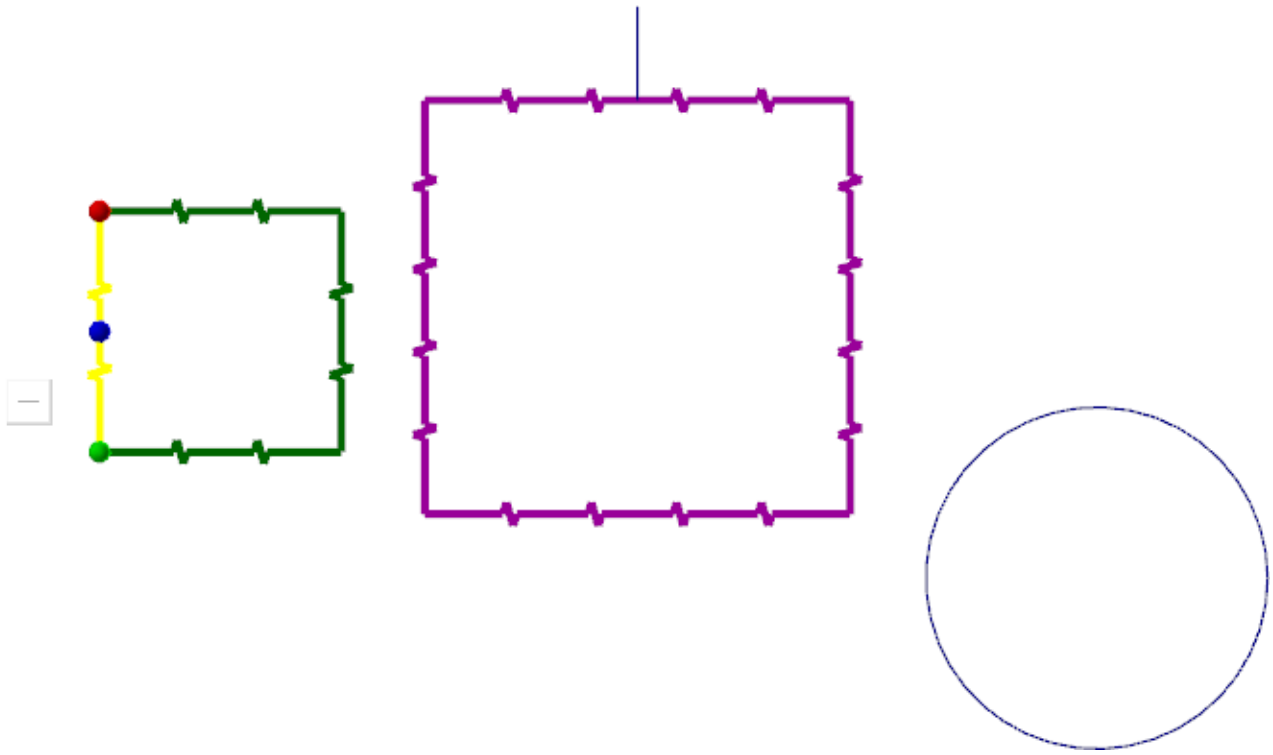
10. Select **Clear all masking**.
11. Save your file.

Exercise 11: Editing Geometry from the Top-Level

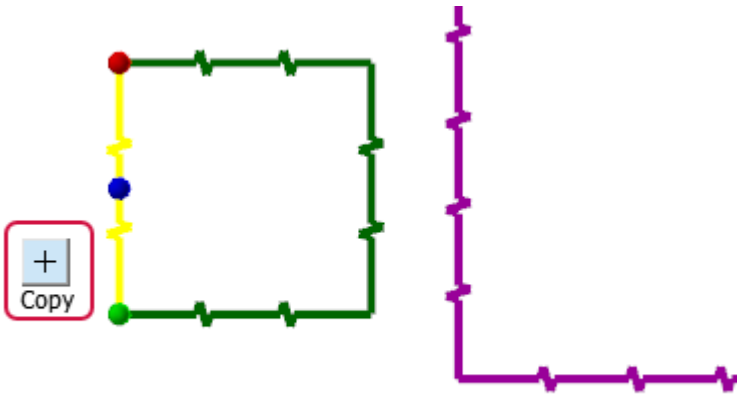
If no other function is active, you can quickly and directly edit wireframe geometry without interacting with a dialog box. In Mastercam, we call this *top-level editing*. In this exercise, you use this feature to edit lines and arcs in ATTRIBUTES.mcam.

Editing with line controls

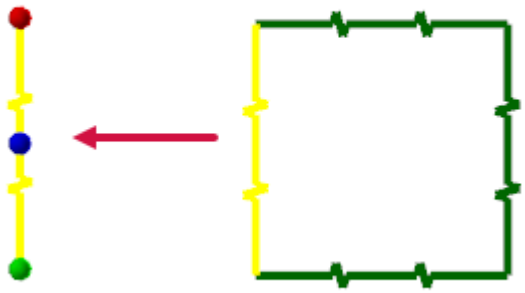
1. If necessary click **File**, **Open** and open ATTRIBUTES.mcam, which you saved in the previous lesson.
2. Double-click a line. Since no other function is active, the line highlights with a combination of spheres and a button.



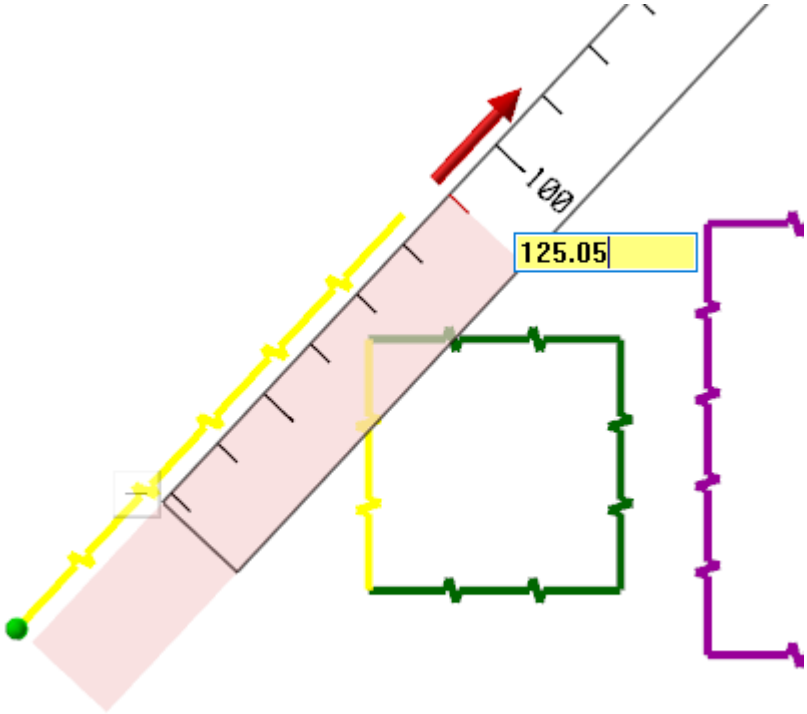
3. Toggle the button from **Move** to **Copy**.



4. Click and drag the blue sphere to create an offset parallel line.

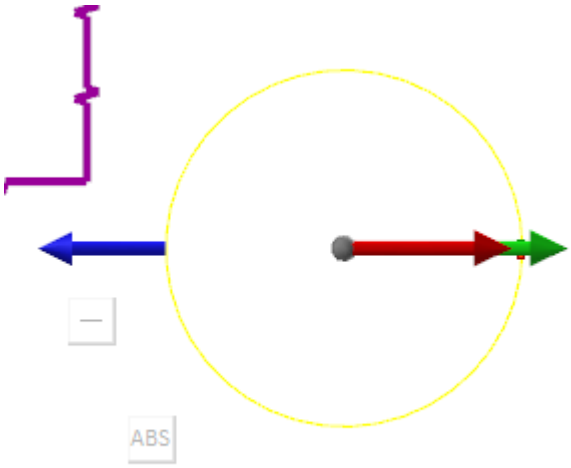


5. Click and drag either of the endpoint spheres on the newly-created line to shorten, lengthen, or change the angle of the line.
6. Hover over the green or red sphere to display its arrow.
7. Click the arrow to activate the ruler control. Type an exact value and press [**Enter**] to precisely change the length of the line.

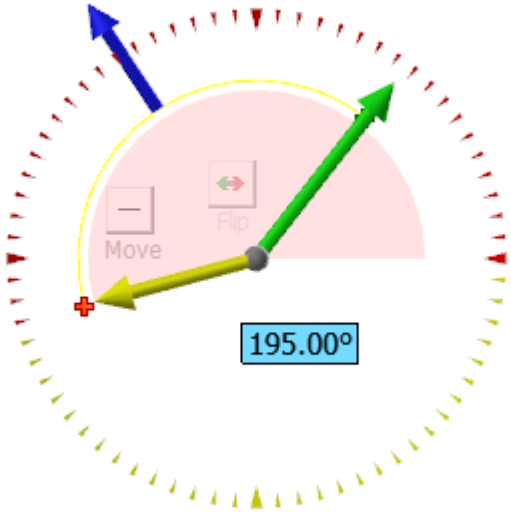


Editing with arc controls

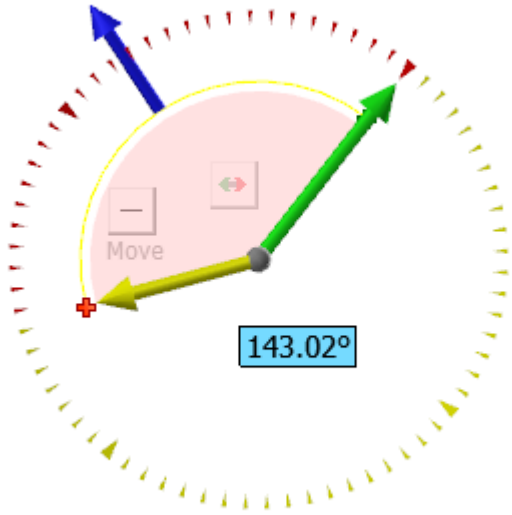
1. Double-click the circle. Since no other function is active, it highlights with a combination of arrows and buttons.



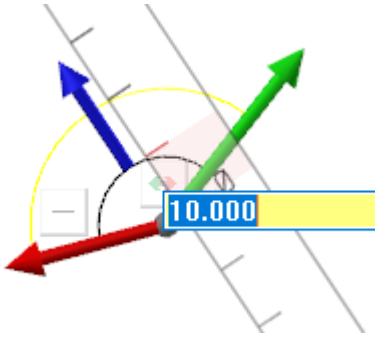
2. Click and drag the origin sphere to move the center point of the circle.
3. Click and drag the green and the red arrows to change the sweep angle of the arc.
4. Hover over the red arrow and notice that Mastercam displays the absolute value of the sweep angle. In this example, the absolute value is **195.00** degrees.



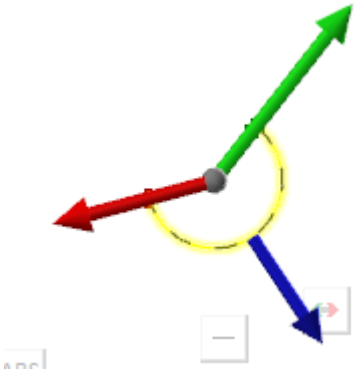
5. Toggle the **ABS/REL** button to change the display so that it shows the angle's relative value. This example now shows the relative value is **143.02** degrees.



6. Click and drag the blue arrow to change the radius of the arc. As with the line controls, you can type in an exact value.



7. Click the **Flip** button to invert the arc and display its complement.



You cannot edit multiple entities with top-level editing. To do that, use the Analyze function. To display the **Analyze** dialog box from the top-level, press the [**End**] key. You can continue to use the new graphic controls as well as change entity attributes such as level, color, and line style. Press the [**End**] key again when you are finished. (The section, "Analyzing Your Part" on page 96, later in this tutorial has more information about the Analyze function.)

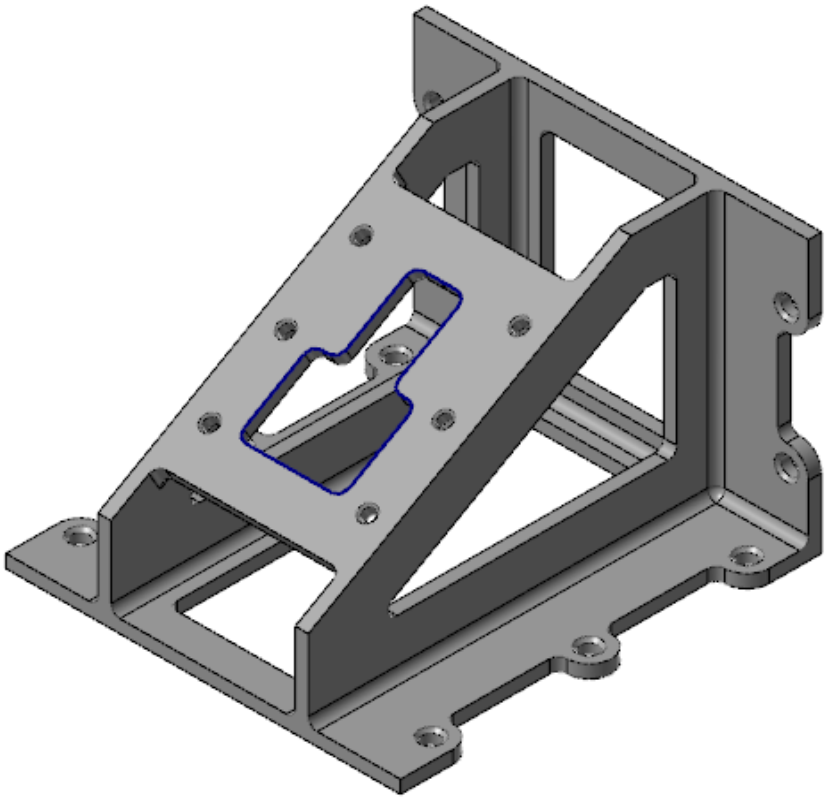
8. Press [**Esc**] to exit the top-level editing function and save your part.

Exercise 12: Exploring the Status Bar

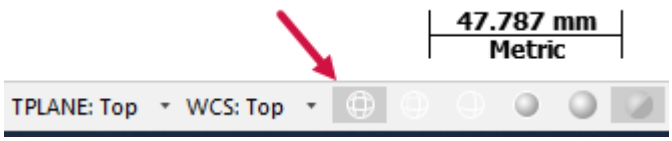
The **Status Bar** across the bottom of the Mastercam workspace displays the XYZ coordinates of the cursor in the workspace. It also provides quick access to the **2D**/**3D** switch and lets you modify the Cplane, Tplane, WCS, and Z Depth of geometry and toolpaths in the graphics window. The right side of the **Status Bar** contains wireframe, shading, and translucency controls.

In this exercise, you use the wireframe, shading, and translucency controls to change the appearance of a part.

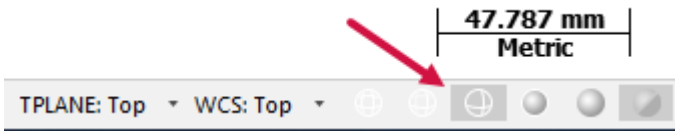
1. Open BRACE.mcam, which was provided with the tutorial. If you see a message for switching units from inch to metric, click **OK** to allow the change.



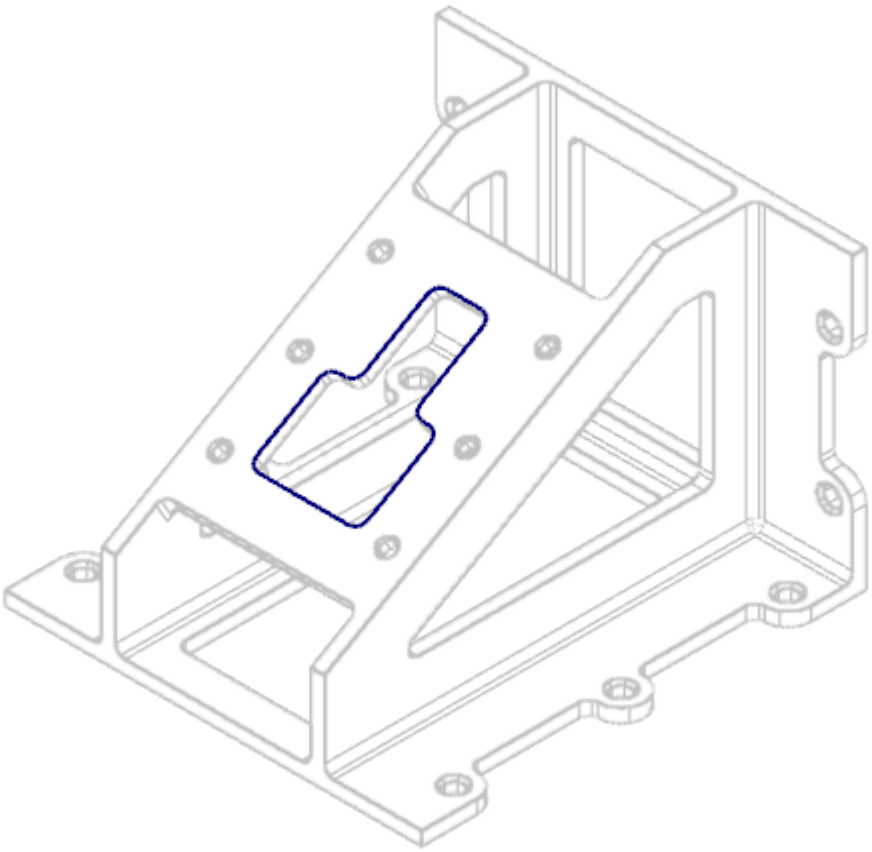
2. Choose **Wireframe** on the **Status Bar**.



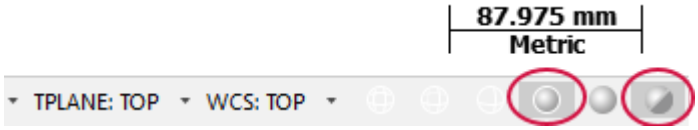
3. Choose **No Hidden** on the **Status Bar**.



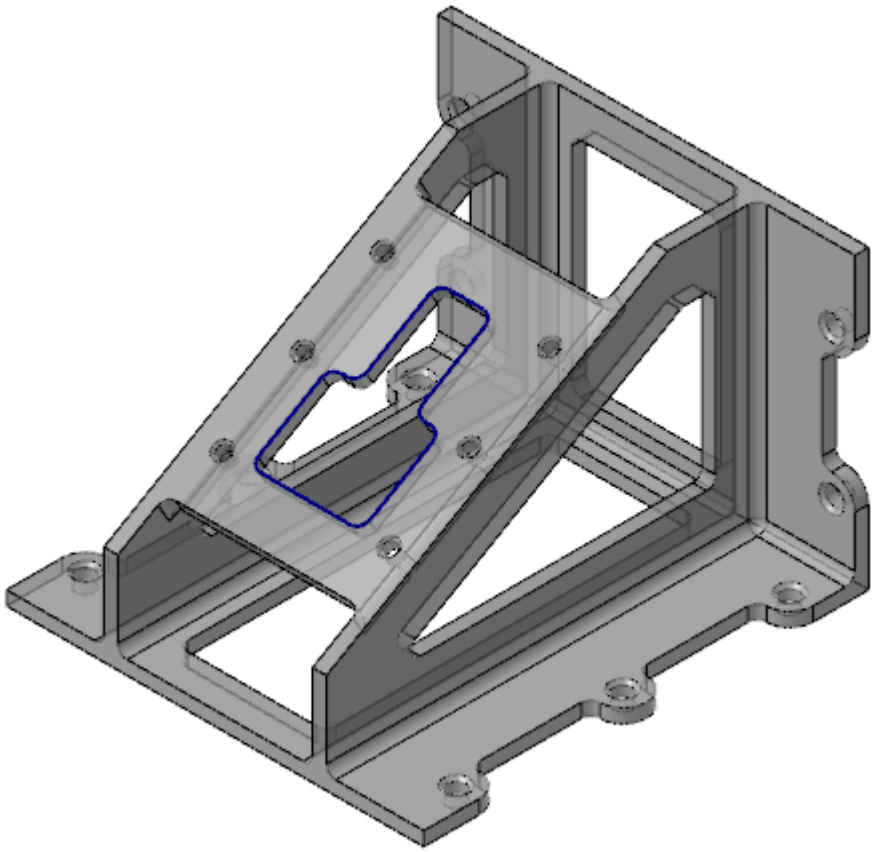
The part displays only the wireframe that is visible in the current view. No hidden wireframe is visible.



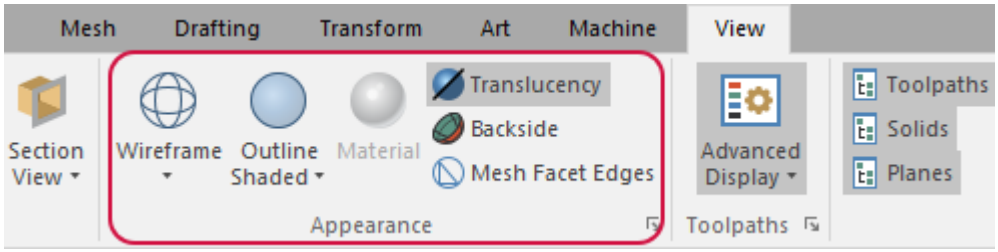
4. Choose **Outline Shaded** and **Translucency** on the **Status Bar**.



The part displays as a translucent solid.



You can also alter how a part displays using the **Appearance** options located on the **View** tab.



5. Close Mastercam without saving your changes or leave the part open for the next exercise.

You have now completed this chapter of the *Mastercam Basics Tutorial*. In the next chapter, you learn more about Mastercam configuration files.

CHAPTER 2

WORKING WITH CONFIGURATION FILES

The **System Configuration** dialog box defines and manages the configuration files that store Mastercam's settings. You can change your system configuration or create a new configuration at any time. It is easy to switch from one environment to another because each configuration is saved to a separate file.

In this chapter, you change options in the **System Configuration** dialog box to see the different kinds of settings that are stored within one configuration file.

Goals

- Create a configuration file.
- Change system colors and CAD styles.
- Modify the size and opacity of on-screen controls.
- Set up AutoSave and Backup.

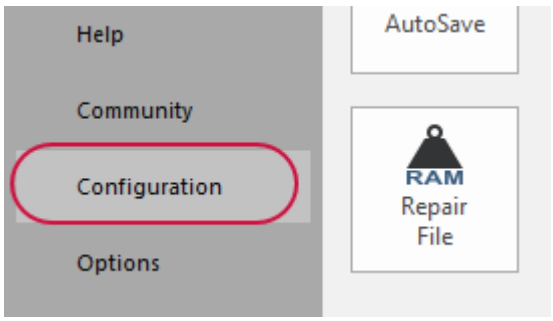
Exercise 1: Creating a Configuration File

In this exercise, you create a configuration file (.config).

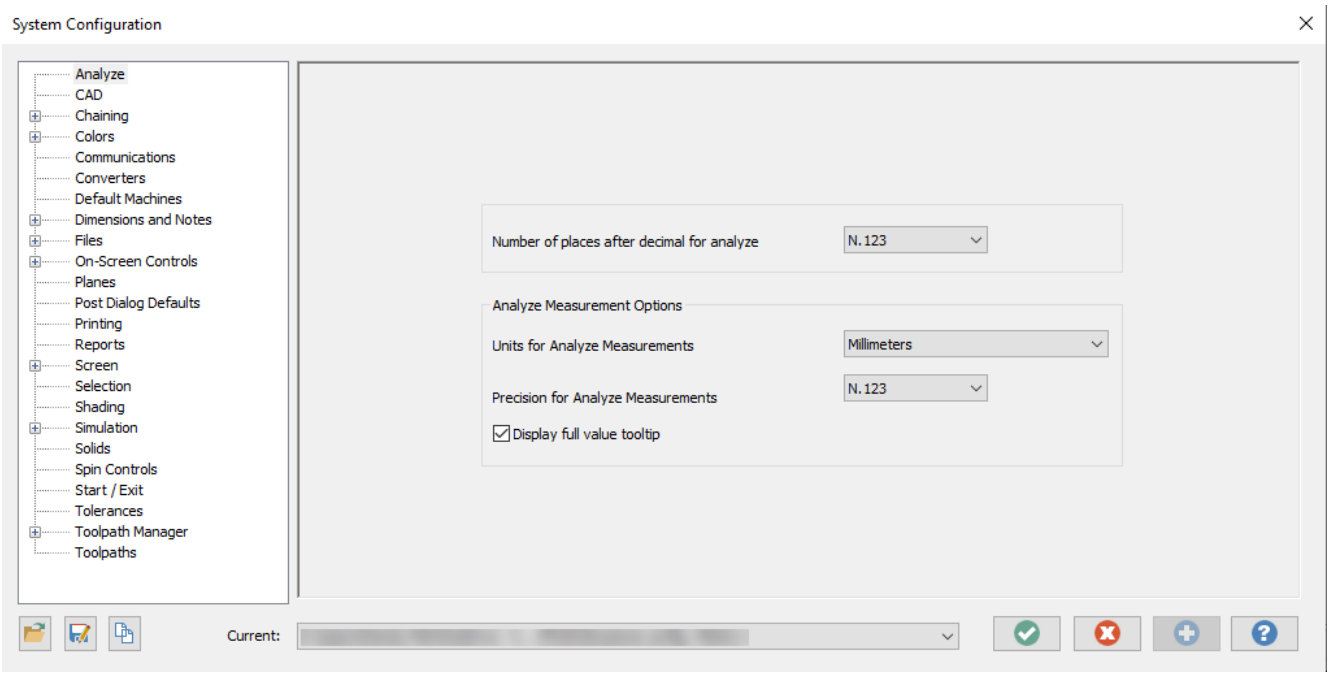
1. Press [**Ctrl** + **N**] to create a new file.

   If prompted to save your work from the last chapter, select **Don't Save**.

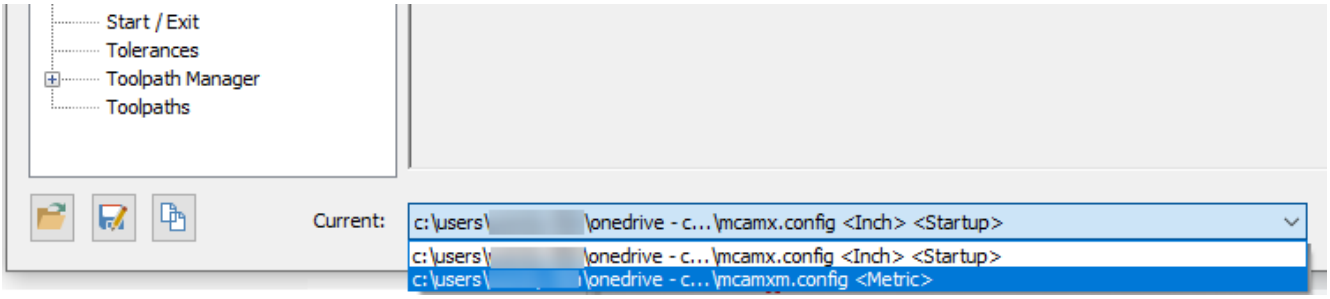
2. Select **File**, **Configuration**.



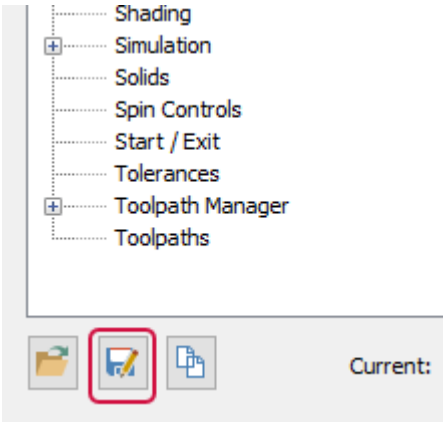
The **System Configuration** dialog box opens.



3. If necessary, select **mcamxm.config <Metric>** from the **Current** drop-down to enable Mastercam's default metric configuration.

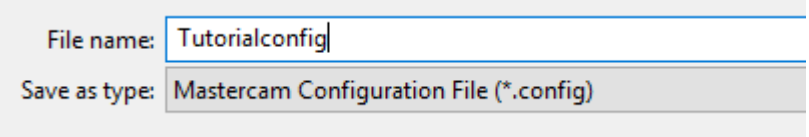


4. Click **Save As** in the lower left corner.



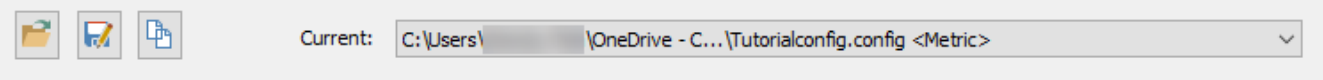
The **Save As** dialog box opens.

5. Enter **Tutorialconfig** for the **File name**.



6. Click **Save**.

   Mastercam creates a new .config file. You can create multiple .config files using the same method.

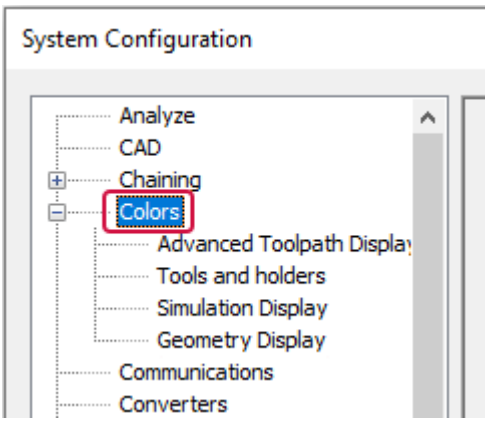


In the following exercises, you save system settings to this configuration file.

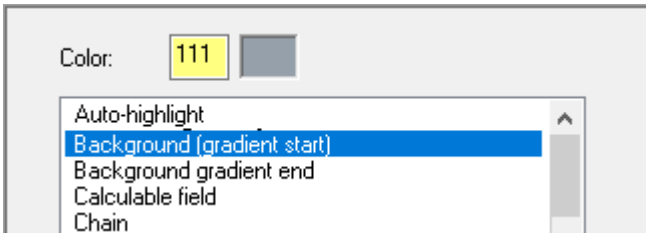
Exercise 2: Changing System Colors

In this exercise, you change some default colors to customize Mastercam's interface and geometry colors.

1. In the **System Configuration** dialog box, select the **Colors** page.

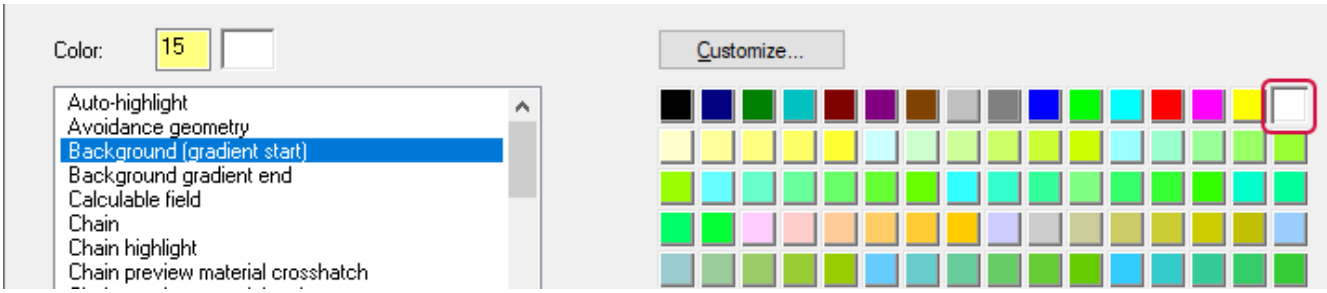


2. Select **Background (gradient start)** from the list.



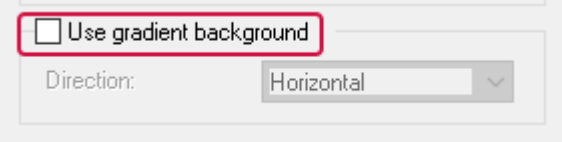
   Background (gradient start) is assigned to color **111**.

3. Select white from the color options on the right side of the dialog box.



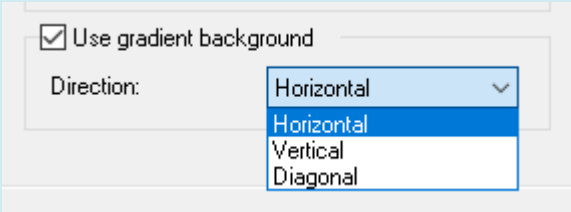
The start of the gradient background is now white.

4. Deselect **Use gradient background** to remove the gradient from the background.



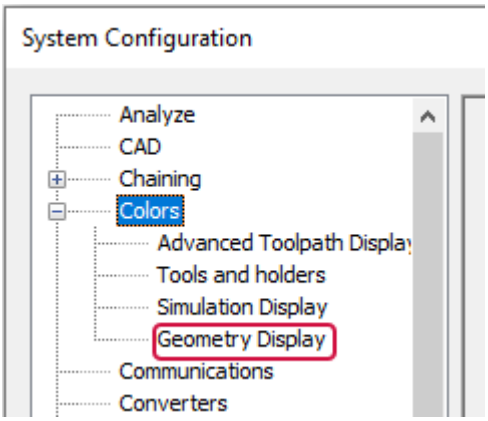
NOTE

The drop-down list next to **Direction** allows you to set gradient direction when **Use Gradient background** is selected.

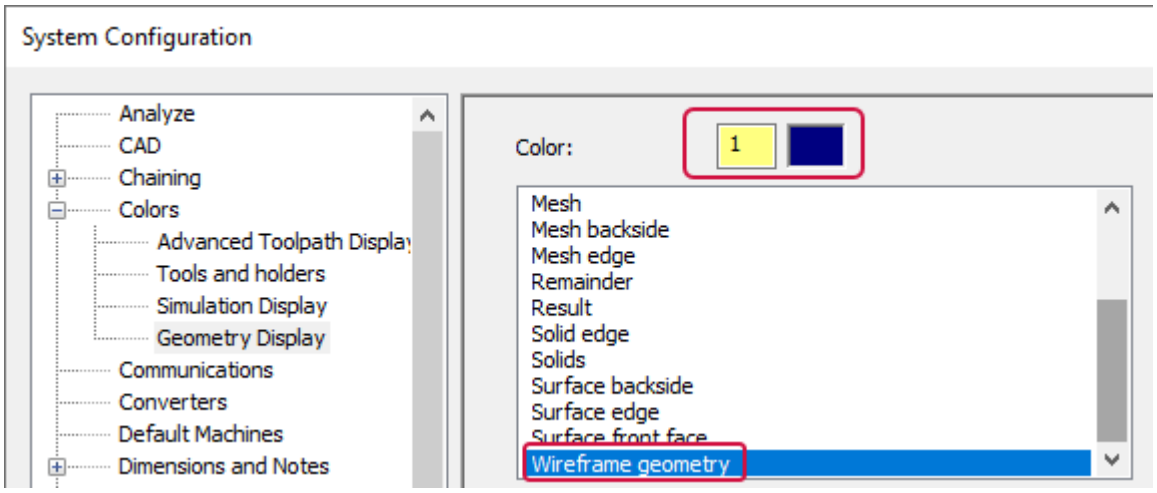


While you can change the defaults of many Mastercam interface colors on the **Colors** page, the **Geometry Display** page customizes the colors for most types of Mastercam geometry.

5. If necessary, expand the **Colors** node on the left-side of the **System Configuration** dialog box, and select the **Geometry Display** page.

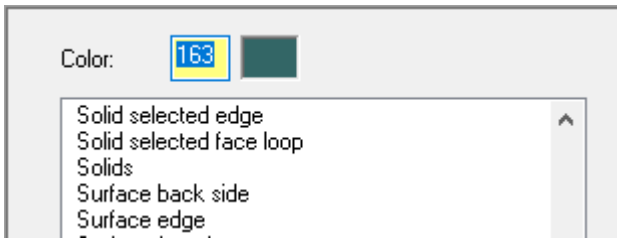


6. Select **Wireframe geometry** from the list.



Wireframe geometry is assigned to color **1**.

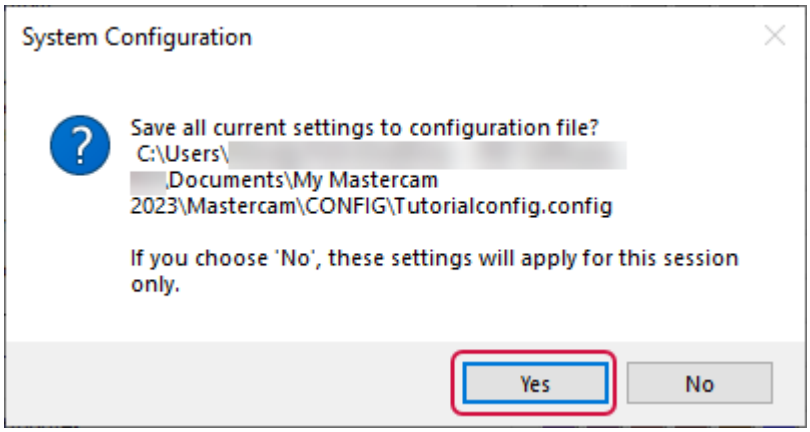
7. Type **163** for the **Color** value, and press [**Enter**].



Wireframe geometry you create now uses this color.

8. Click **Apply** to save the settings on this page and leave the **System Configuration** dialog box open.

9. Choose **Yes** to save all current settings to the Tutorialconfig.config file.

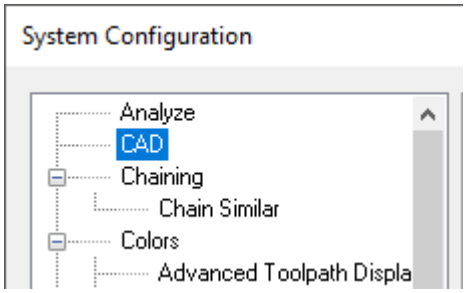


Choosing **Yes** preserves your changes. They are in effect every time you open Mastercam. If you choose **No**, the changes are in effect only until you close Mastercam.

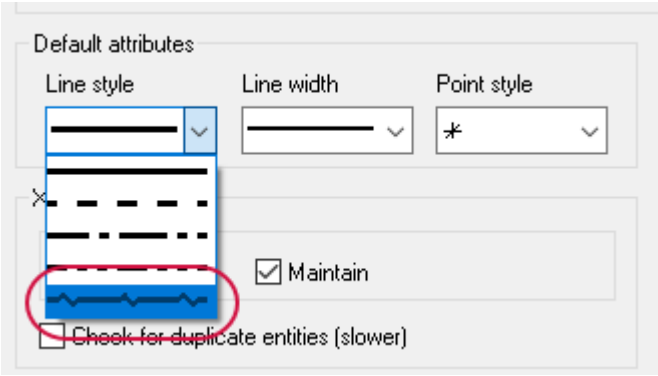
Exercise 3: Changing CAD Settings

In this exercise, you make changes to the **CAD** page. The **CAD** page specifies design defaults, such as default line and spline attributes.

1. Select the **CAD** page.

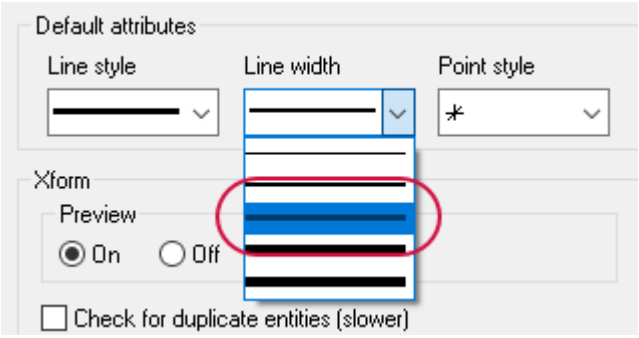


2. In the **Default attributes** section, select the **Line style** shown below.



Wireframe geometry you create now uses this line style.

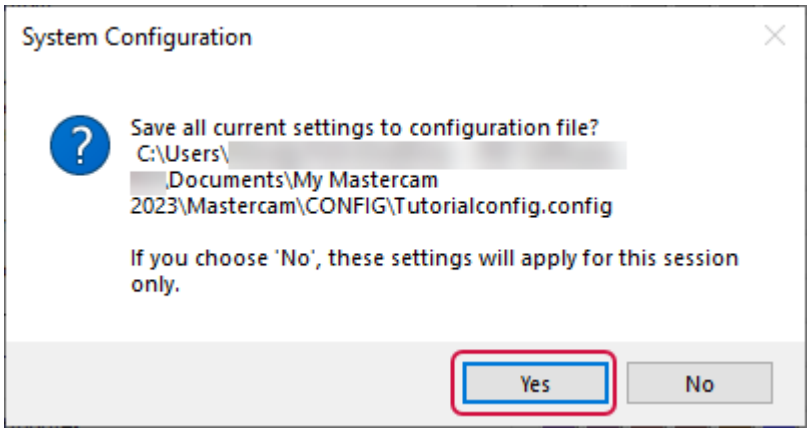
3. Select the **Line width** shown below.



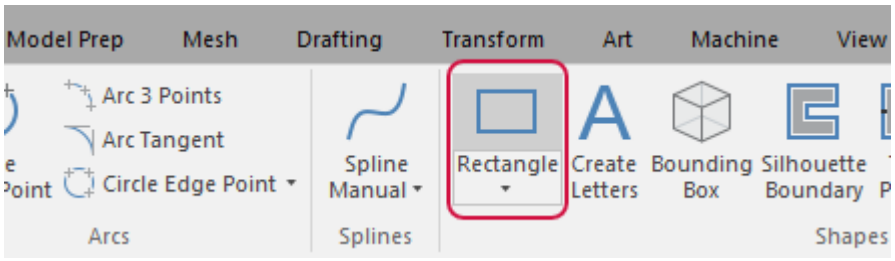
Wireframe geometry you create now uses this line width.

4. Click **OK** in the **System Configuration** dialog box.

5. Click **Yes** to save all current settings to the Tutorialconfig.config file.



6. On the **Wireframe** tab, select **Rectangle**.



The **Rectangle** function panel opens.

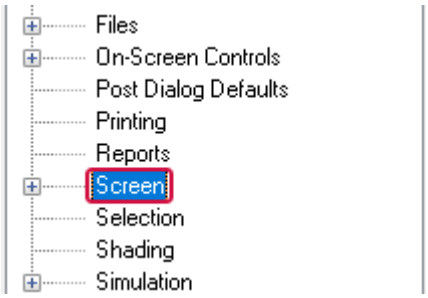
7. Follow the prompts to create a rectangle, and then click **OK** in the **Rectangle** function panel.

The line style and width of the geometry matches the selections in the **CAD** page, as well as the geometry color set in the **Colors** page.

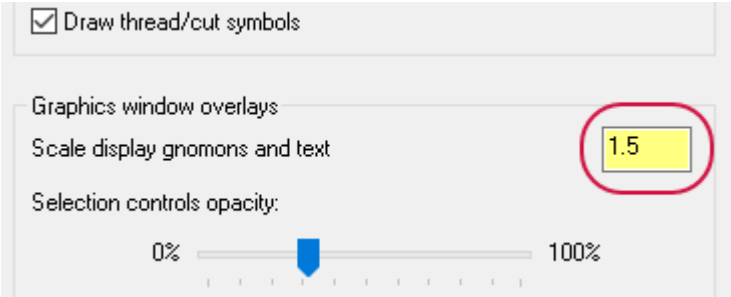
Exercise 4: Changing the Size and Opacity of On-screen Controls

In this exercise, you set the size of the on-screen gnomons and text and change the opacity of the **Selection Bar** and **Quick Mask** buttons.

1. Click **File**, **Configuration** to open the **System Configuration** dialog box.
2. Select the **Screen** page.

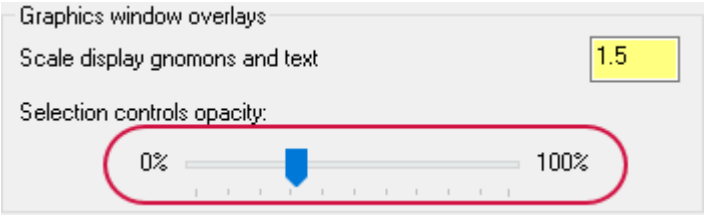


3. Enter **1.5** for **Scale display gnomons and text** and press [**Enter**].



The size of the elements in the graphics window is increased to 150%.

4. Move the **Selection controls opacity** slider from left to right.



This option adjusts the opacity of the **Selection Bar** and **Quick Masks**. Watch as the opacity changes with the slider movement, and choose an opacity setting you like. The first image below shows the slider set to 50% opacity and the second image shows the slider set to 100% opacity.

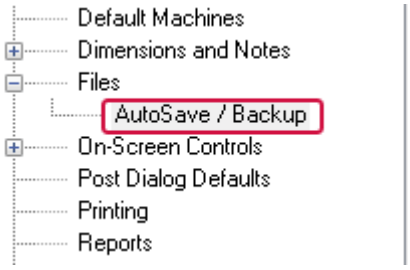
5. Click **Apply** to save the settings on this page.

6. Select **Yes** to save the settings to the configuration file.

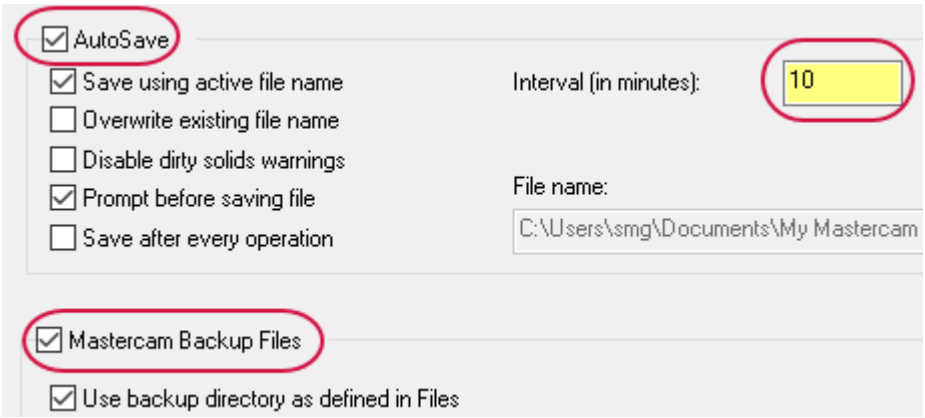
Exercise 5: Setting up AutoSave and Backup

When you are working on a part, Mastercam can save your work automatically at intervals that you specify. You can also save versions of your files as backups. In this exercise, you set up these functions.

1. Expand the **Files** category, and select the **AutoSave / Backup** page.



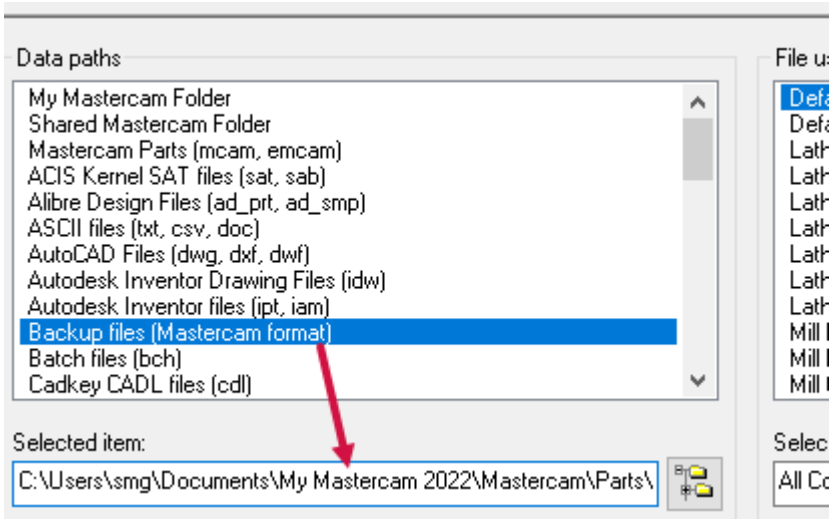
2. In the **AutoSave / Backup** page, do the following:
   - Select **AutoSave** to activate the AutoSave options.
   - Enter **10** for **Interval (in minutes)**.
   - Select **Mastercam Backup Files** to activate the backup options.



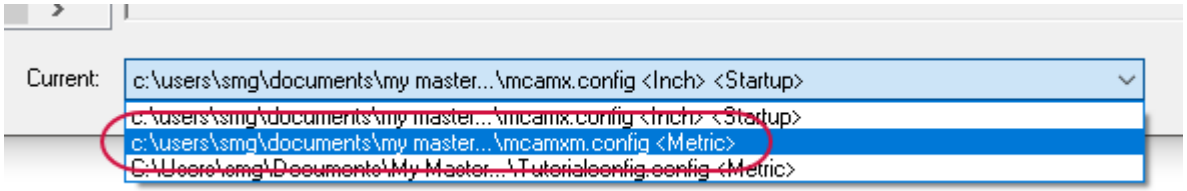
   Mastercam will save your work automatically every ten minutes and keep the ten most recent versions of your file.

3. Select the **Files** page to view the location of your backup files.

4. Choose **Backup files (Mastercam format)** in the **Data paths** list. The path displays in the **Selected item** box as shown below.



5. Select **mcamxm.config <Metric>** from the **Current** drop-down to enable the metric configuration for the remainder of this tutorial.



6. Click **OK**.

NOTE

If you would like to learn more about the individual settings on any **System Configuration** page, click the **Help** button located in the lower right-hand corner of the dialog box.

You have now completed this chapter of the *Mastercam Basics Tutorial*. In the next chapter, you learn how to customize the Mastercam workspace.

CHAPTER 3

CUSTOMIZING MASTERCAM

In this chapter, you learn how to customize your Mastercam workspace, including tab and interface options, the graphics window context menu, and keyboard shortcuts. Your settings apply each time you open Mastercam and are not specific to a configuration file.

Goals

- Customize the graphics window context menu.
- Map Mastercam functions to keyboard shortcuts.
- Set display options for tabs and interface colors.

Exercise 1: Customizing the Graphics Window Context Menu

In this exercise, you customize the graphics window context menu that displays when you right-click in the graphics window. Customizing this menu allows you quick access to your commonly used functions.

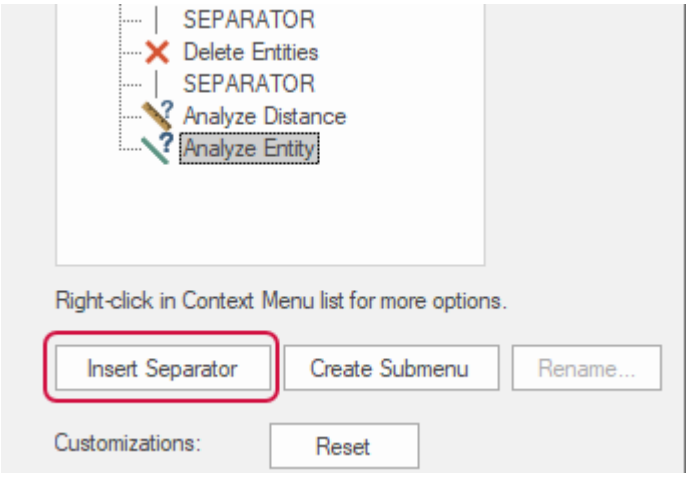
1. Select **File**, **Options**.

   The **Options** dialog box opens.

2. Select the **Context Menu** page.

   Categories of functions are displayed on the left side of the dialog box, while the right side contains the functions currently in the context menu.

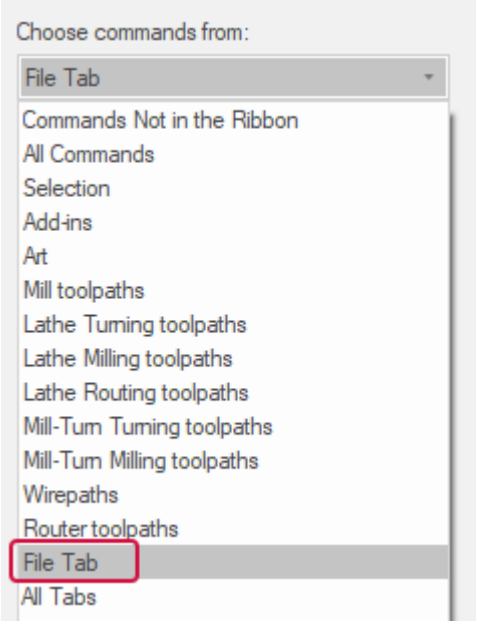
3. Select the last item in the **Context Menu** list, and click **Insert Separator** to place a line of separation after the item.



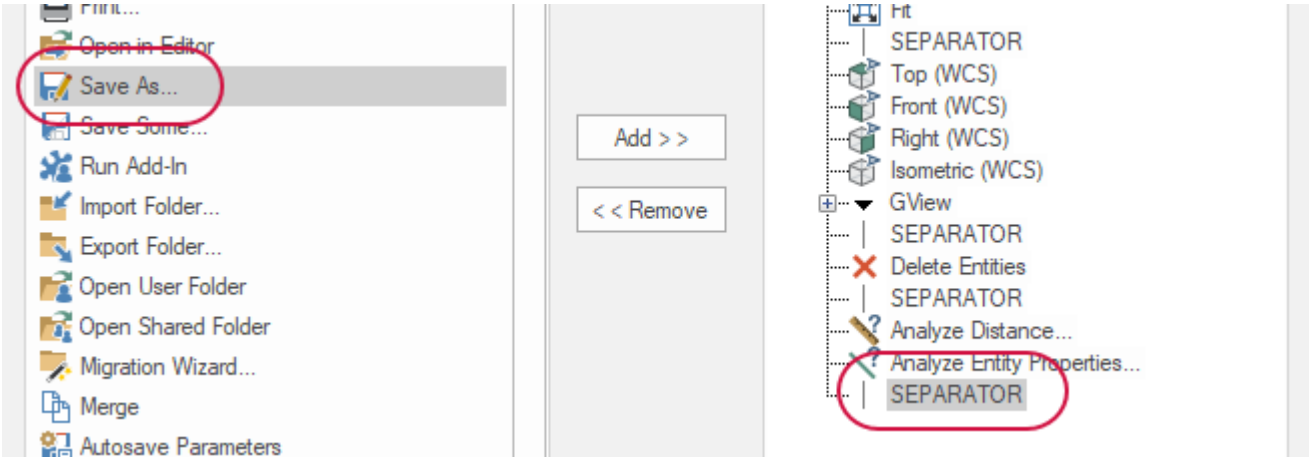
NOTE
You can also right-click the last item in the **Context Menu** list, and select **Insert Separator** from the menu to place a line of separation after the item.

A separator line appears after the last item.

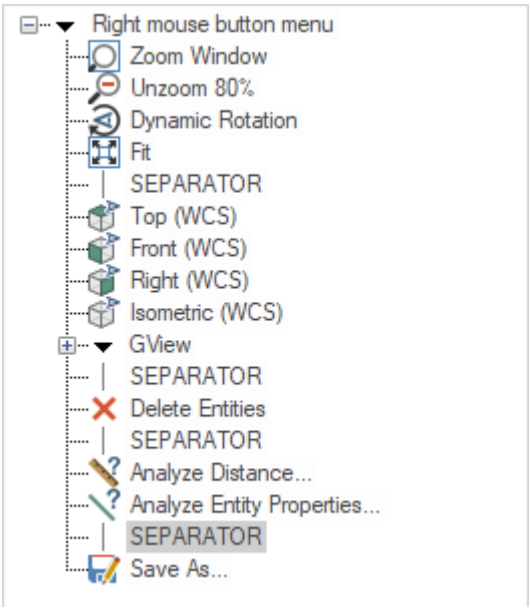
4. Select **File Tab** from the **Choose commands from** drop-down list.



5. Select the **Save As** function in the **Category** list and **SEPARATOR** in the **Context Menu** list.

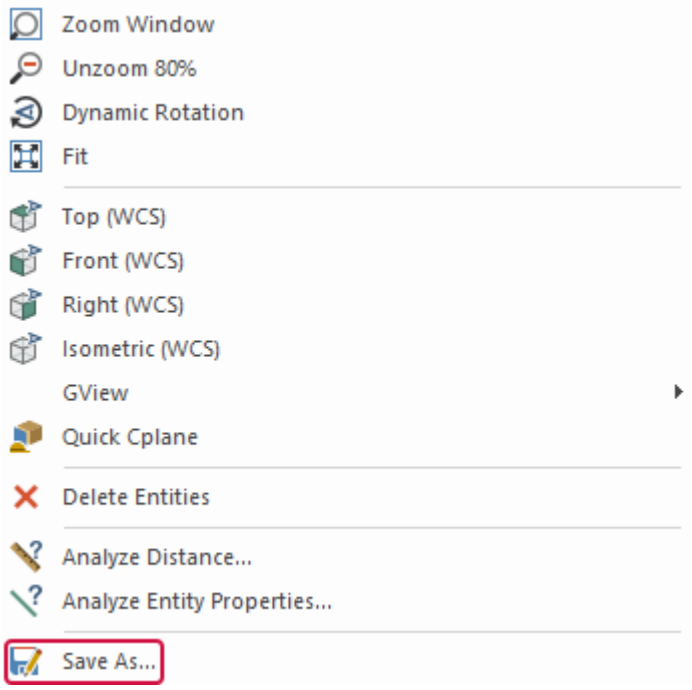


6. Click **Add** to add the Save As function after the separator.



7. Click **OK** to save these changes to the right-click menu.

8. Right-click in the graphics window to see the updated context menu.



Save As has been added to the bottom of the menu.

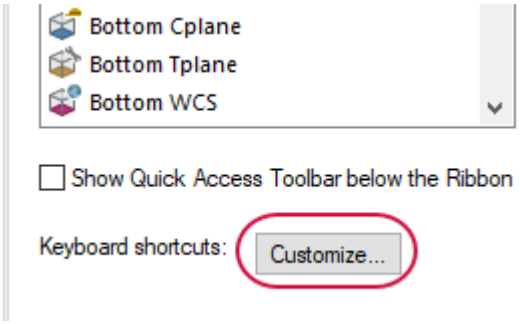
Exercise 2: Mapping Functions to Keyboard Shortcuts

In this exercise, you map a Mastercam function to a shortcut key.

1. Select **File**, **Options**.

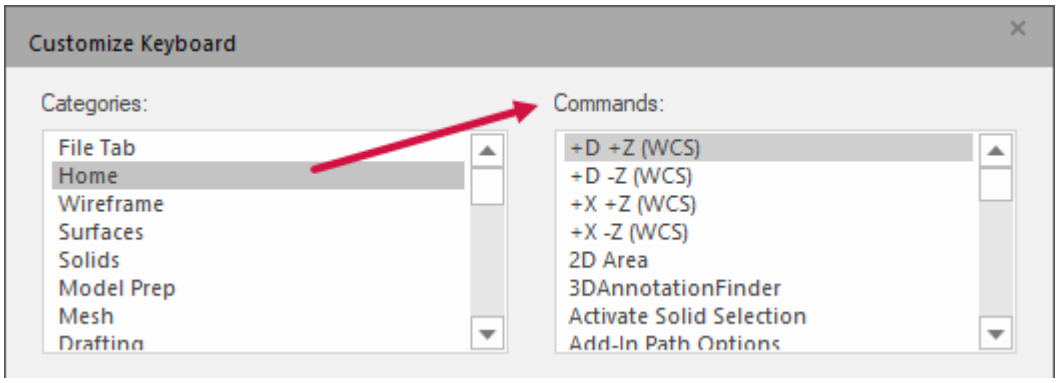
   The **Options** dialog box opens.

2. On the **Quick Access Toolbar** page, select **Customize** next to **Keyboard shortcuts**.



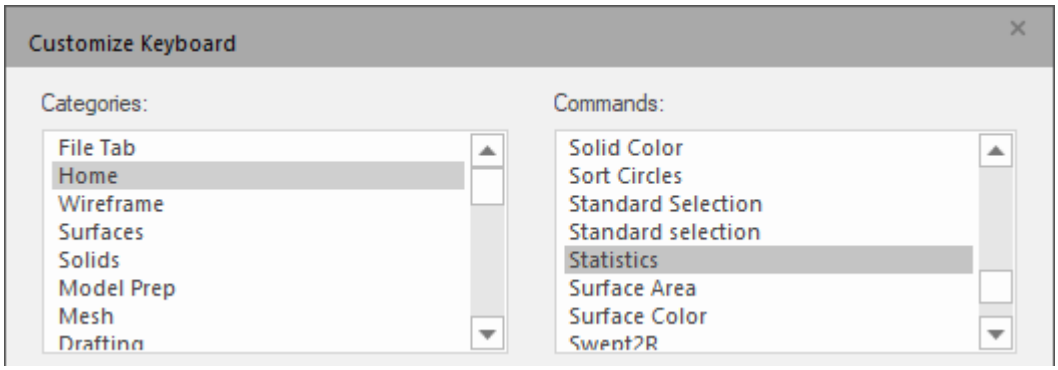
   The **Customize Keyboard** dialog box opens.

3. Select **Home** from the **Categories** list.



The **Commands** list populates with functions from the **Home** tab.

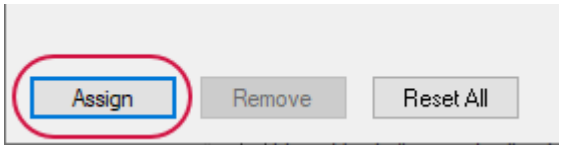
4. Select **Statistics** from the **Commands** list.



Notice that the **Current Keys** list is blank. This is because there are no shortcut keys assigned to this function.

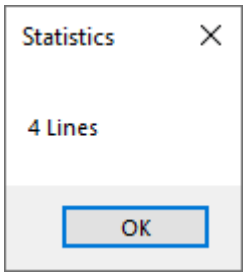
5. Click in the **Press new shortcut key** field, and press [**Ctrl+Shift+H**] to create a new shortcut key.

6. Select **Assign** to map this shortcut key to the **Statistics** function.



7. Select **Close** to exit the **Customize Keyboard** dialog box, and then **OK** to close the **Options** dialog box.

8. In the graphics window, press [**Ctrl+Shift+H**] to access the **Statistics** function.



   The **Statistics** dialog box displays a summary of the entities in the current file, including total number of visible entities by type, and the number of operations and tools. In this example, **Statistics** reports four lines because of the rectangle created in the previous lesson.

9. Click **OK** to close the dialog box.

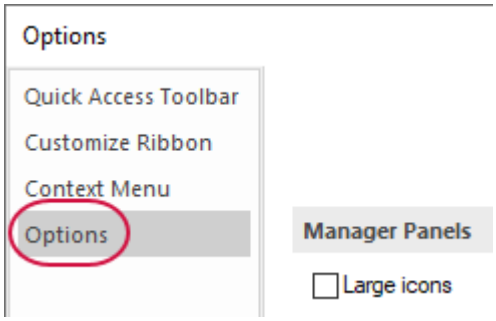
Exercise 3: Setting Mastercam Display Options

In this exercise, you set some of Mastercam's display options.

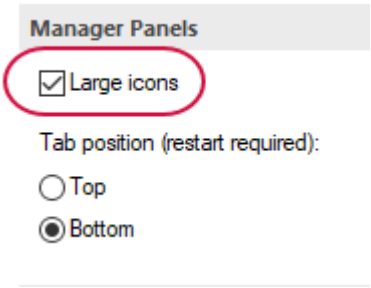
1. Select **File**, **Options**.

   The **Options** dialog box opens.

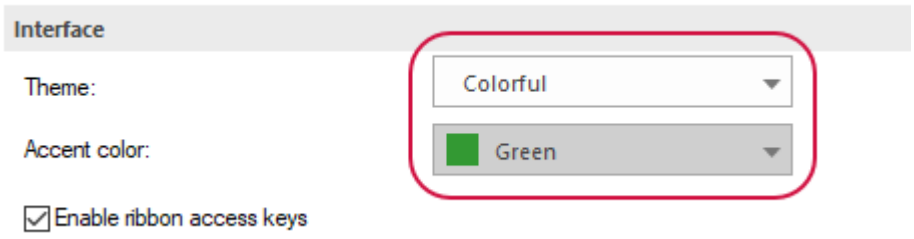
2. Select the **Options** page.



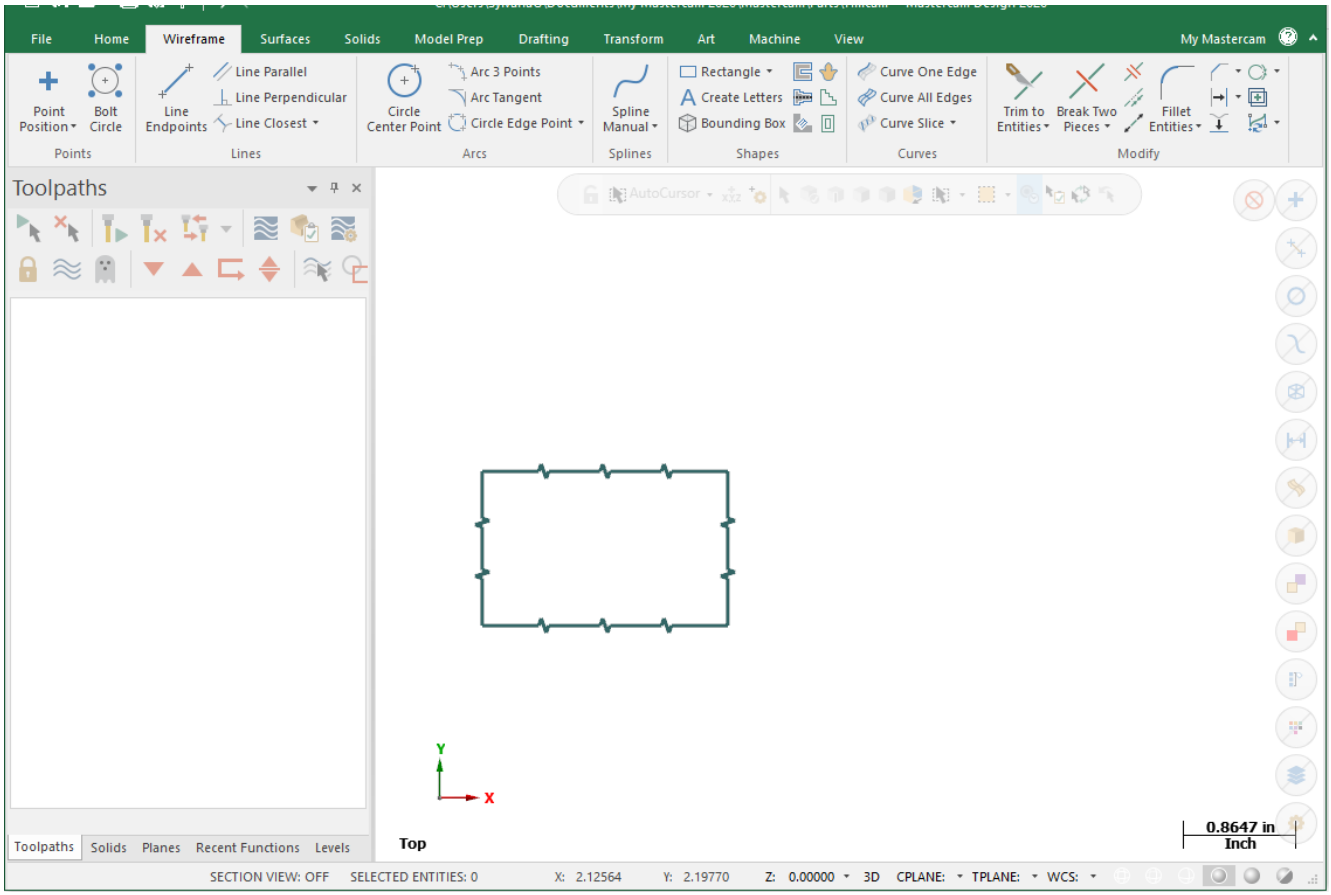
3. Select **Large icons**.



4. Set **Theme** to **Colorful** and **Accent color** to **Green**.



5. Click **OK** to close **Options**. Mastercam uses the larger icons and green theme.



6. Click **File**, **Options**, and return the options to the original settings.
7. Close Mastercam without saving your changes or leave the file open for the next exercise.

You have now completed this chapter of the *Mastercam Basics Tutorial*. In the next chapter, you learn more about working with files.

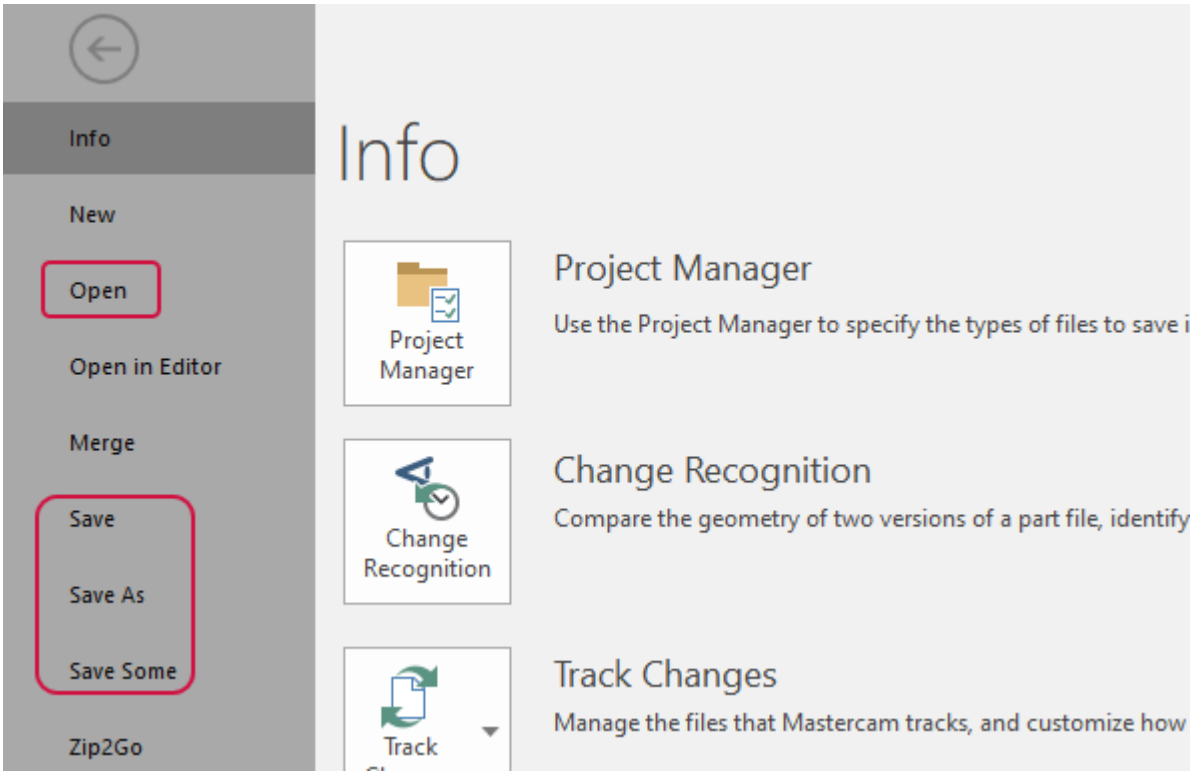
CHAPTER 4

WORKING WITH FILES

Mastercam not only saves and loads its own file types (.mcx-*, .mcam), but also many common file formats, including the following:

- SOLIDWORKS (.sldprt, .sldasm, .slddrw)
- AutoCAD (.dwg, .dxf, .dwf)
- ProE/Creo (.prt, .asm)

For the full list of available formats, refer to the Mastercam website (mastercam.com/support/technical-support/translators/). Load and save native Mastercam files by selecting **Open**, **Save**, **Save As**, or **Save Some** from the Backstage menu or from the QAT.



You can also open files by:

- Dragging a Mastercam file or third-party compatible file from Windows Explorer and dropping it in Mastercam's graphics window.
- Dragging and dropping a file onto the Mastercam desktop icon to open a new instance of Mastercam.

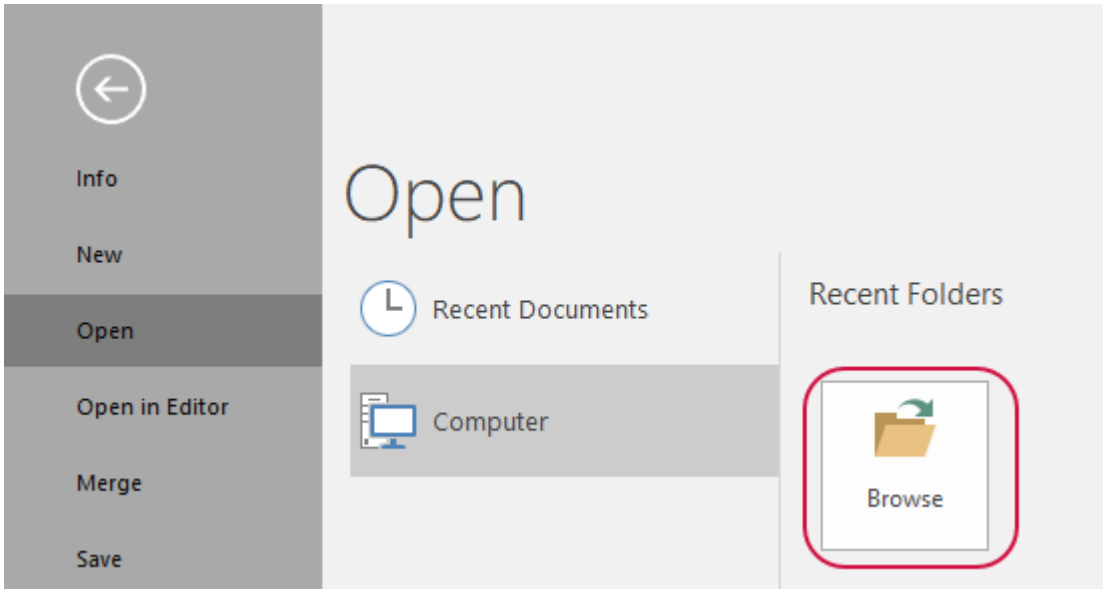
Goals

- Open a non-Mastercam file.
- Export a Mastercam file.
- Use Zip2Go files.

Exercise 1: Opening Non-Mastercam Part Files

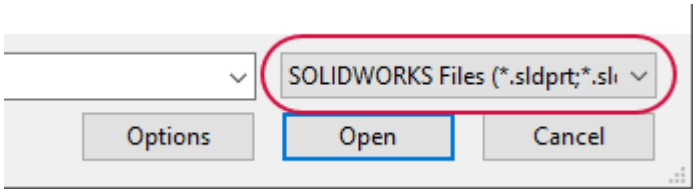
Importing files from other applications is similar to opening native Mastercam files. Depending on the file, you might need to specify how Mastercam imports elements of the file. In this exercise, you convert a SOLIDWORKS (.sldprt) file to a Mastercam part. You do not need SOLIDWORKS to open or convert a SOLIDWORKS file.

1. Select **File**, **Open**. Select **Computer**, then **Browse**.



   The **Open** dialog box displays. Browse to the tutorial parts folder.

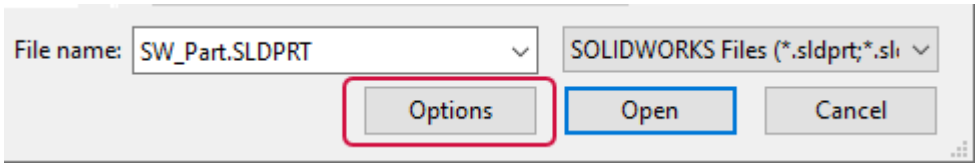
2. In the **Open** dialog box, select **SOLIDWORKS Files ("*.sldprt"; "*sldasm"; "*slddrw")** from the drop-down.



   The dialog now displays only SOLIDWORKS parts.

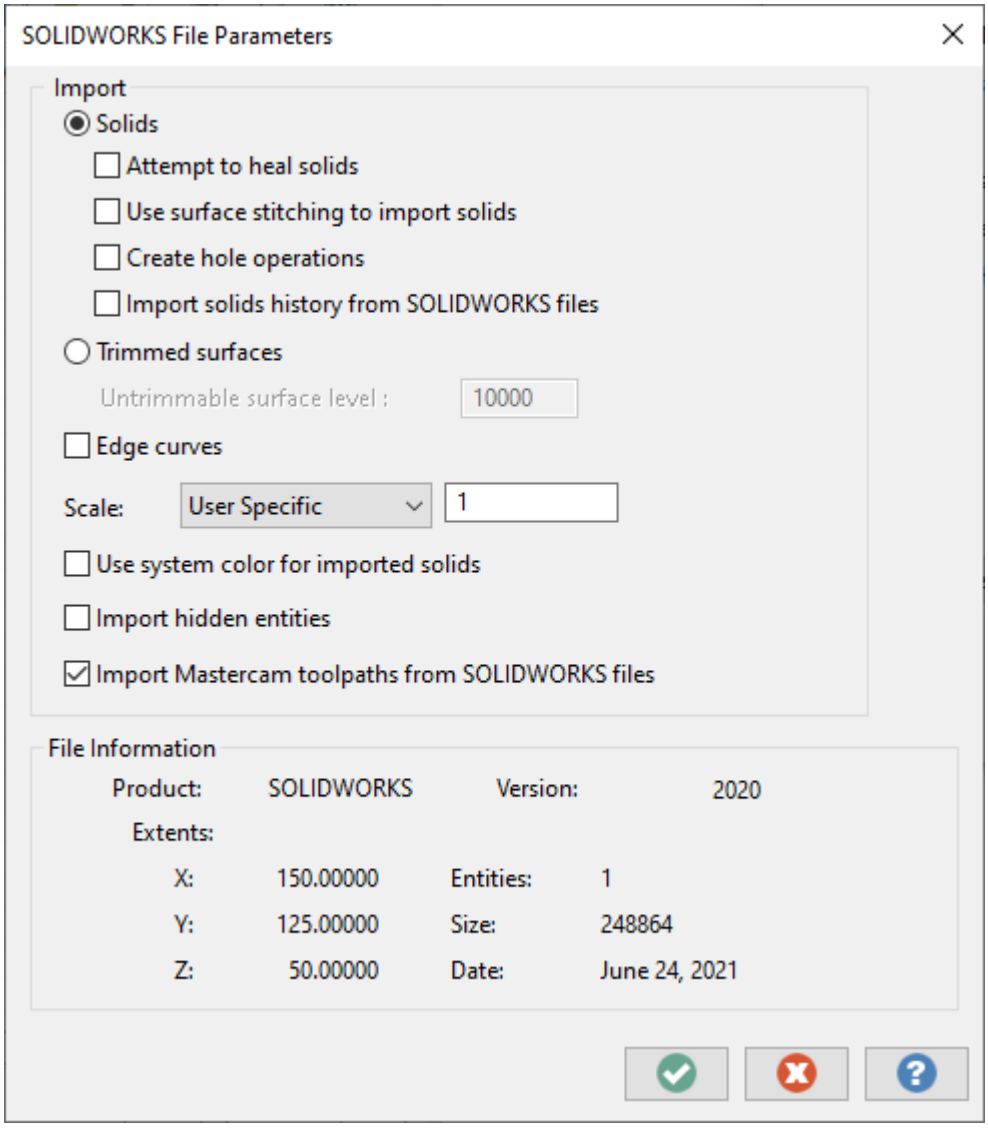
3. Select SW_PART.sldprt, which was provided with this tutorial.

4. Select **Options**.



   The **SOLIDWORKS File Parameters** dialog box opens.

You can use these options, which change based on the file type, to control how files are read into the Mastercam database.

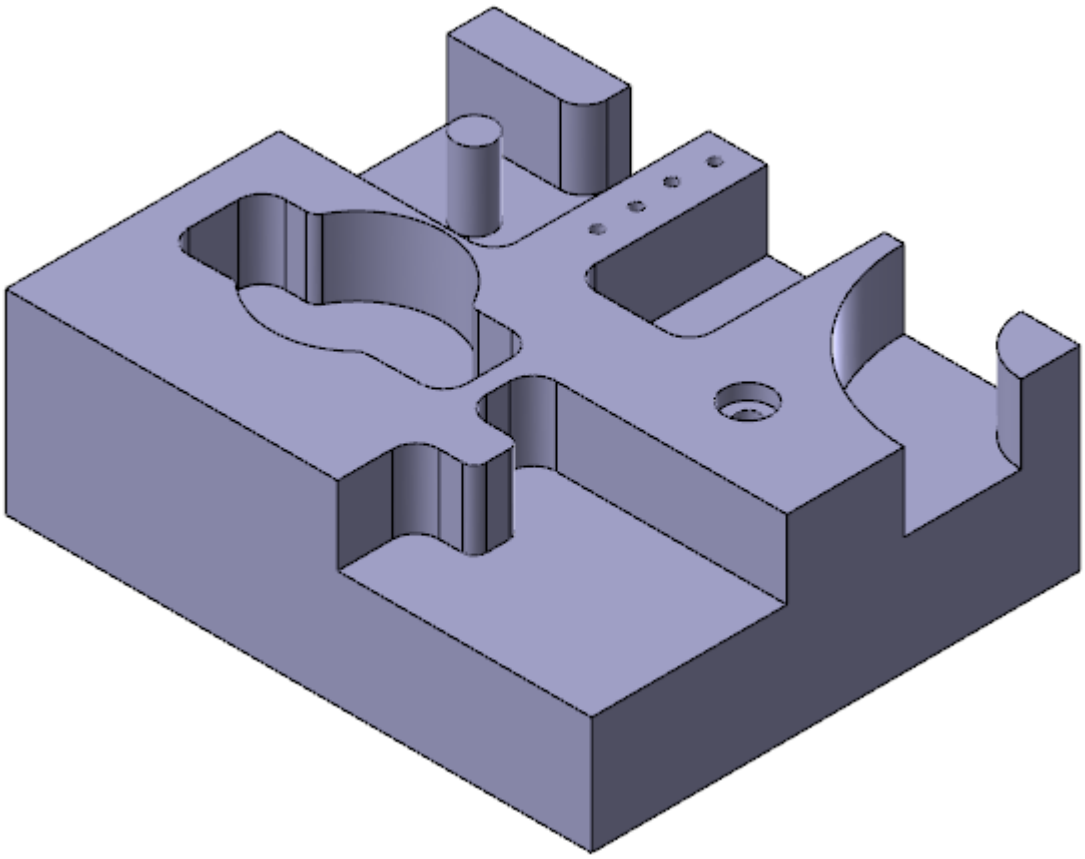


5. Select **OK** without making changes.
6. Select **Open** in the **Open** dialog box.

   Select **Don't Save** if you are asked to save changes to the current Mastercam file.

   Mastercam converts and opens the part.
7. Click **Outline Shaded** and toggle **Translucency** off to view the part as a solid.

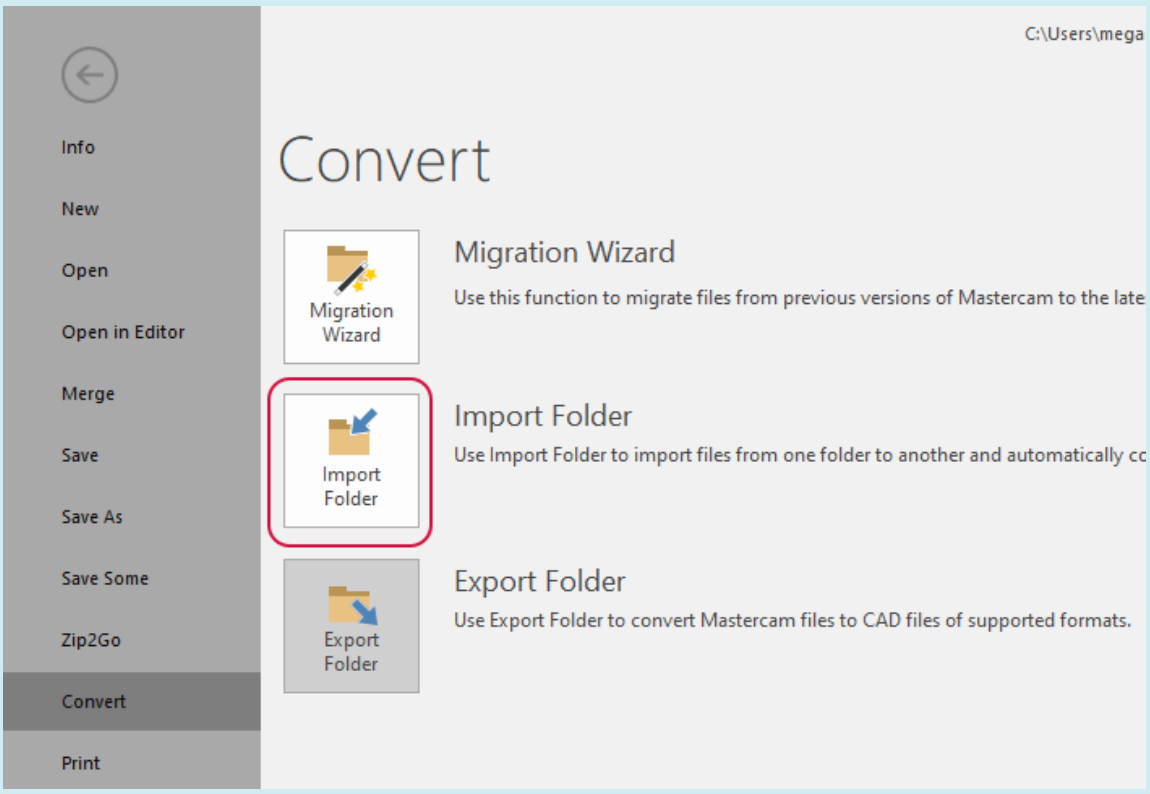
The part displays as shown below.



You learn more about the graphics window and views in later sections of the tutorial.

NOTE

To learn about converting multiple non-Mastercam files at the same time, click **File, Convert, Import Folder**. Select the help button in the **Import Folder** dialog box.

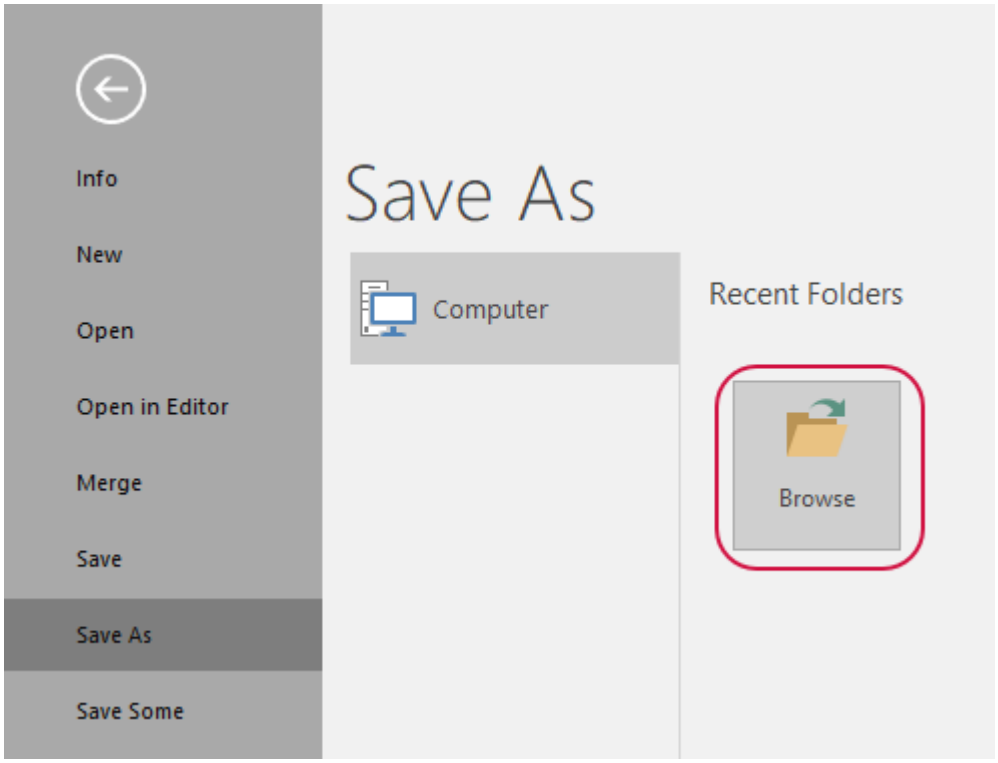


Exercise 2: Exporting a Single File

When exporting, you might need to set a few options to guarantee a successful conversion of the file data for your purposes. In most cases, you need only select the software package to which you are exporting the file. In this exercise, you learn to convert a Mastercam file to a different format.

1. Open 2D_CHAMFER_MM.mcam, which was provided with this tutorial. If prompted to save changes to the current Mastercam file, select **Don't Save**.

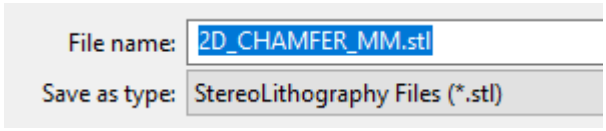
2. Select **File**, **Save As**, and then click **Browse**.



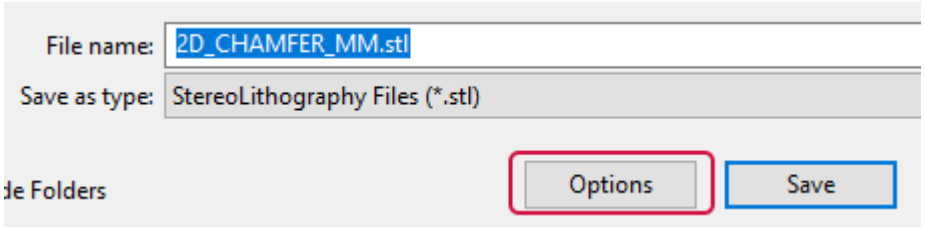
The **Save As** dialog box opens.

> **NOTE**
> You can also click the **Save As** button on the Quick Access Toolbar.
>
>
>

3. Select **StereoLithography Files ("*.stl")** from the **Save as type** drop-down list.

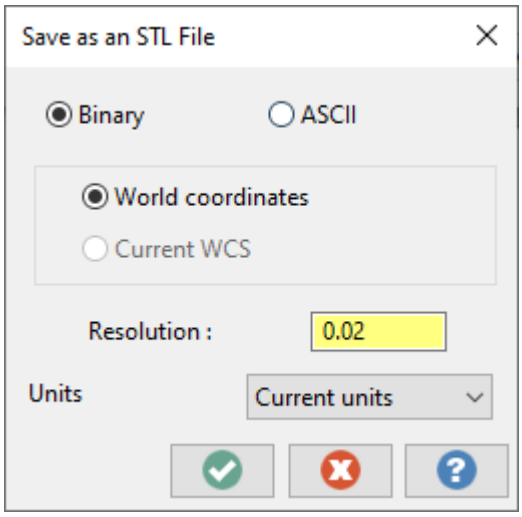


4. Select **Options**.

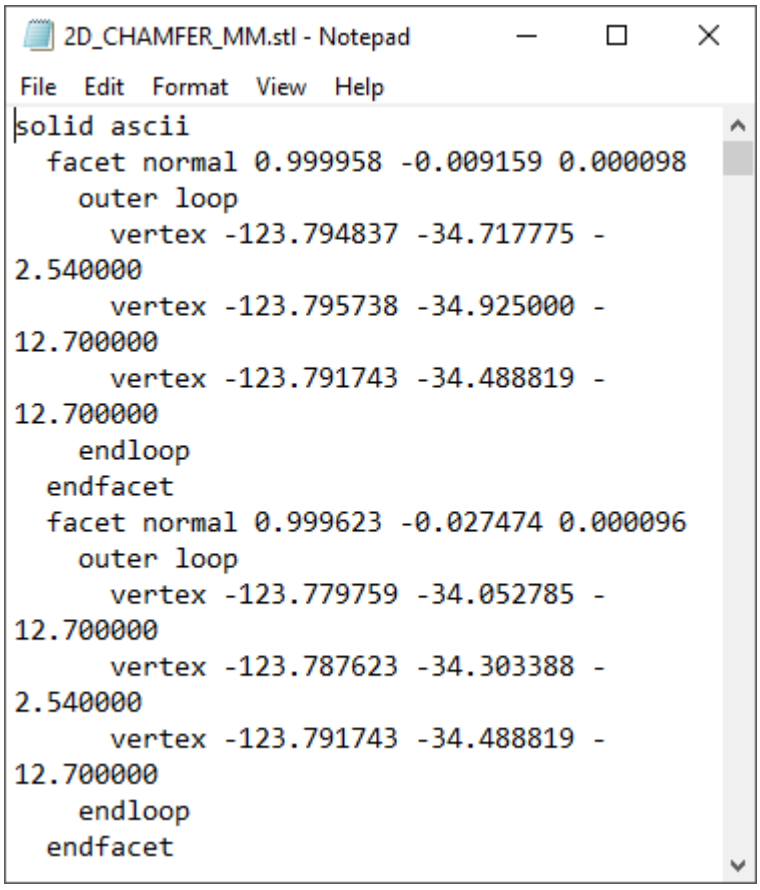


The **Save as an STL File** dialog box opens.

5. Select the **ASCII** option.

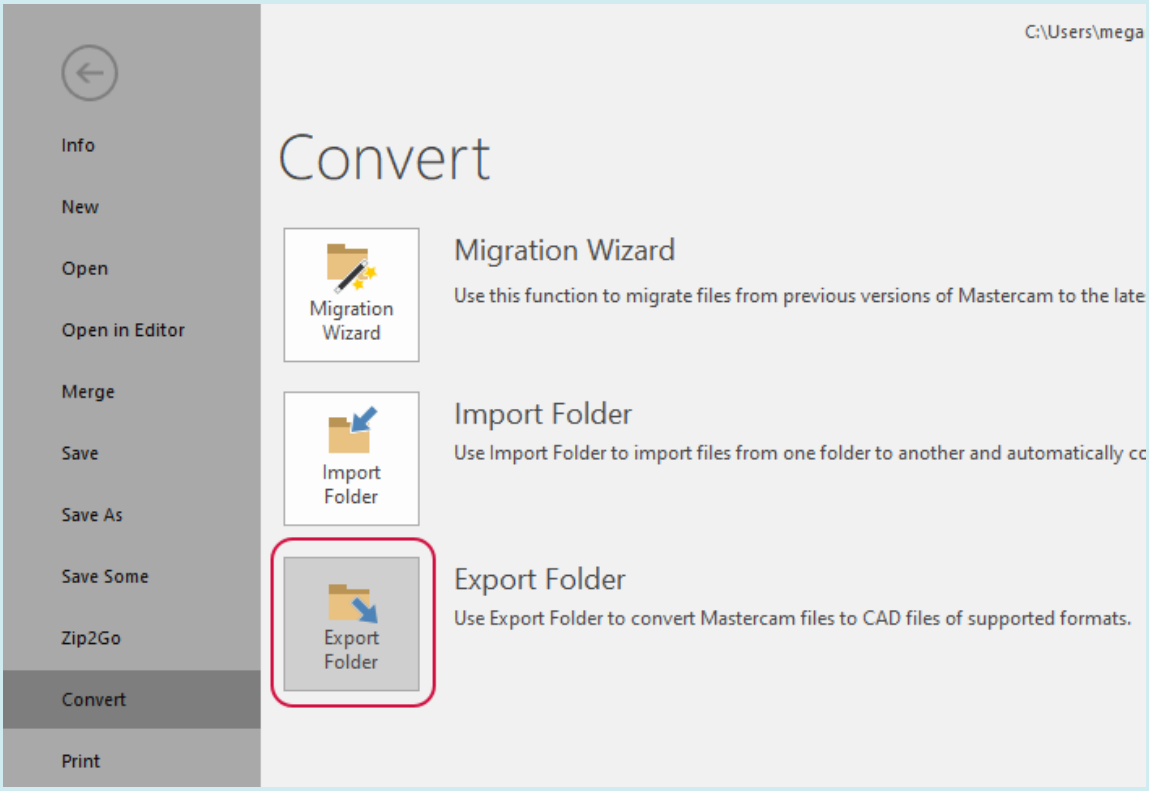


6. Click **OK** to close the **Save as an STL File** dialog box.
7. Click **Save** in the **Save As** dialog box.
8. Mastercam saves the file in ASCII StereoLithography format, which allows you to view it with any text editor, such as Notepad, as shown below.



NOTE

To learn more about exporting multiple Mastercam files at the same time, click **File, Convert, Export Folder**. Select the help button in the **Export Folder** dialog box.



Exercise 3: Using Zip2Go

The Zip2Go Wizard gathers and compresses the current Mastercam part data into a .MCAM-CONTENT, .Z2G, or .ZIP file. It scans the machine groups in your current file and captures information such as your Mastercam configuration, machine definition, post files, tool and material libraries, and other files associated with your part. This is useful if you are trying to share information with other users or Technical Support.

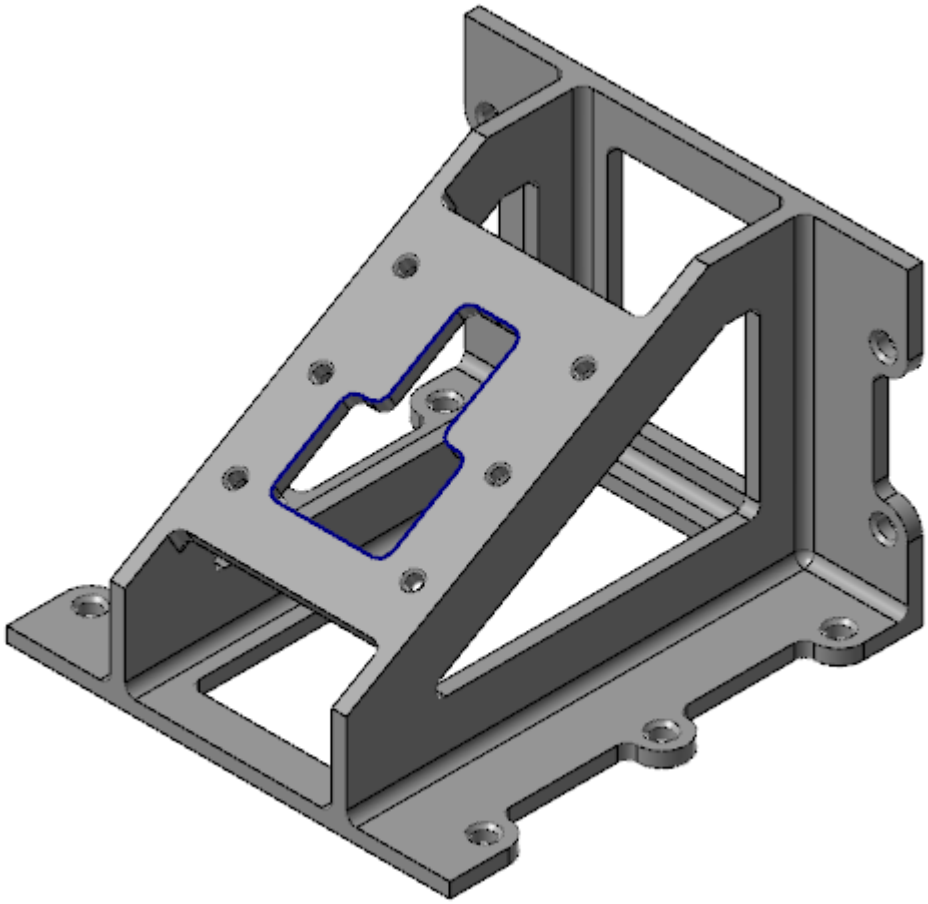
For information about each file type, see the following table.

File Type	Benefits
.MCAM-CONTENT	Can be dragged and dropped into Mastercam. This unzips the files to their proper location within the default folder structure. To view the files inside, rename it to a .zip.
.Z2G and .ZIP	Can be unzipped by anyone to view contents. These maintain the original file structure inside the compressed file.

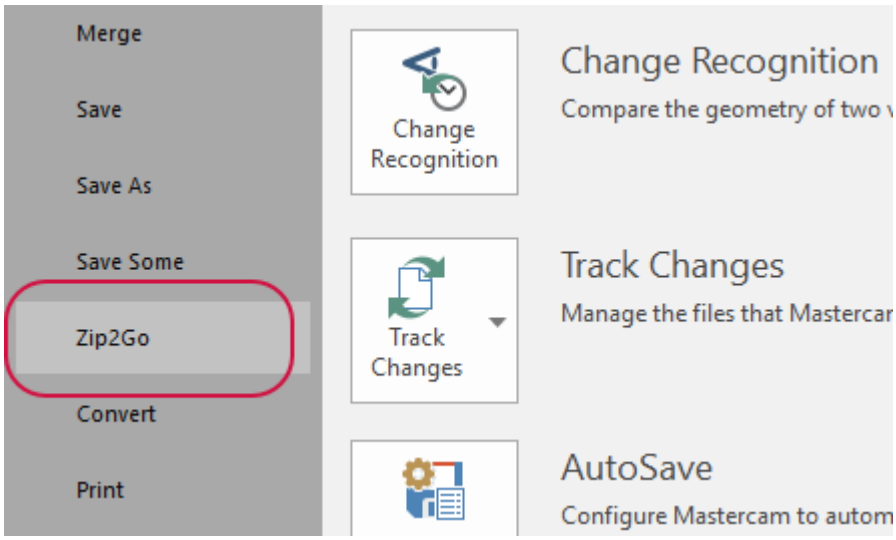
In this exercise, you run the Zip2Go Wizard to compress a Mastercam part file and all the files associated with it into an .Z2G file.

1. Open BRACE.mcam, which was provided with this tutorial.

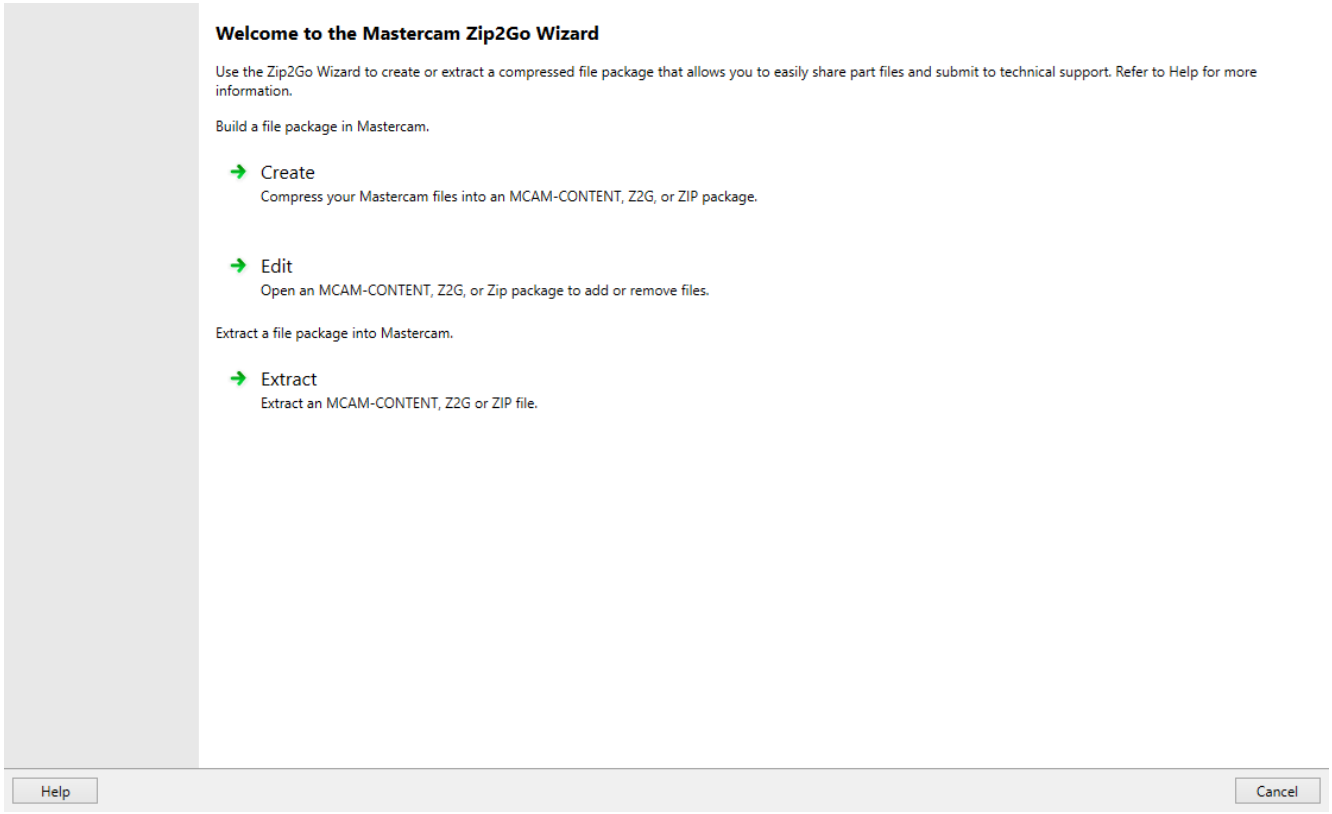
   If prompted to save changes to 2D_CHAMFER_MM.mcam, choose **Don't Save**.



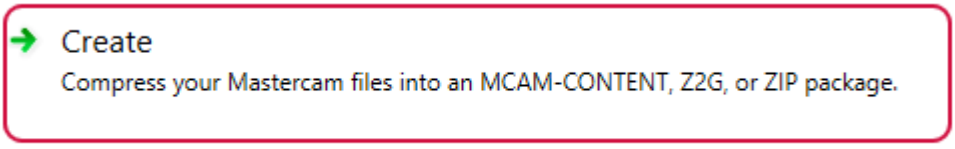
2. If necessary, click **Outline Shaded** on the **Status Bar** to view the part as a solid.
3. Select **File**, **Zip2Go**.



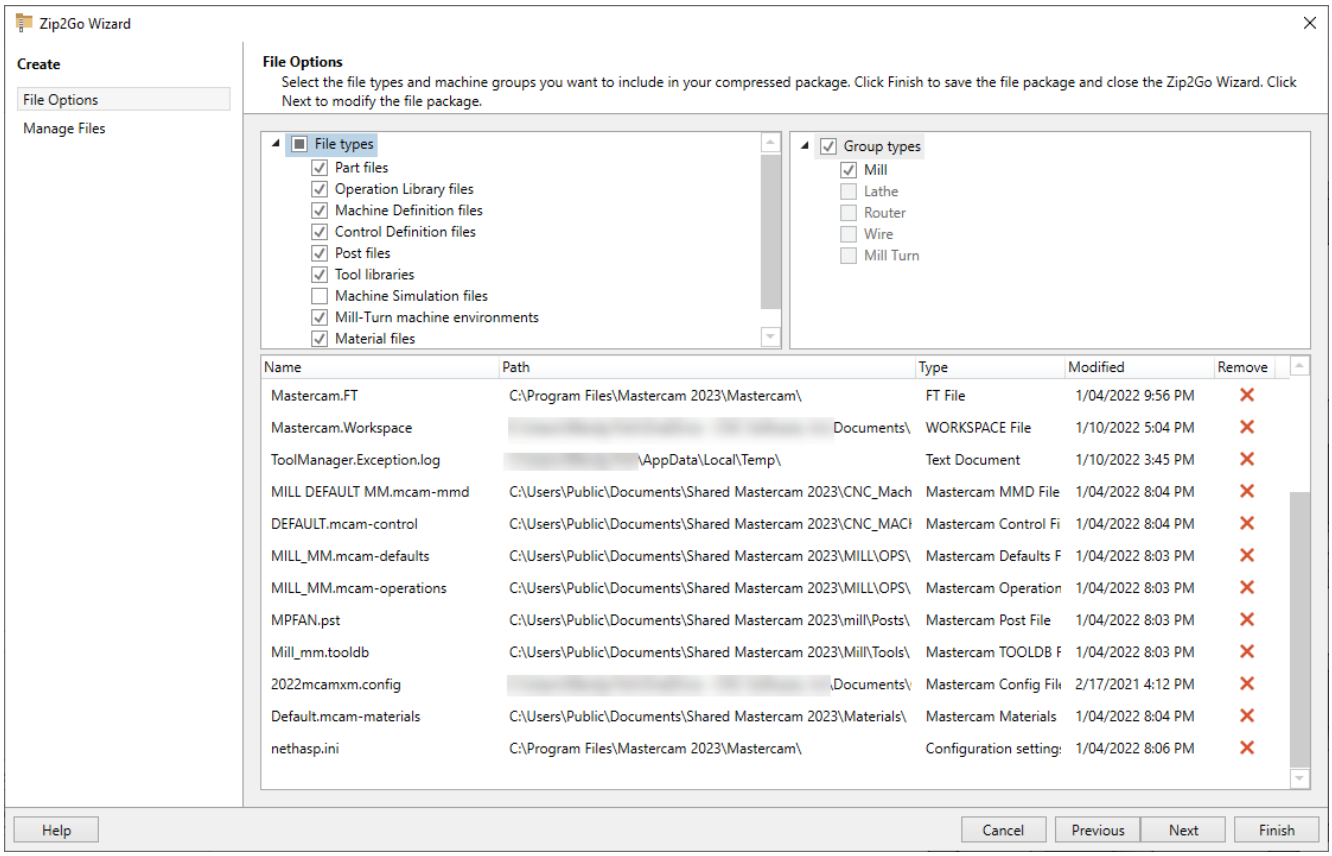
The **Zip2Go Wizard** opens.



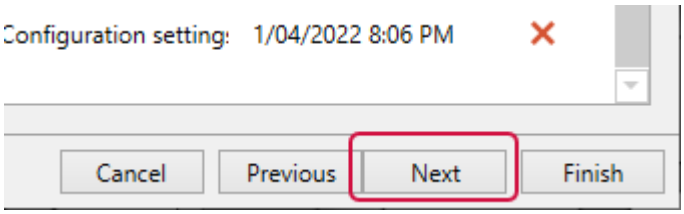
4. Click **Create**.



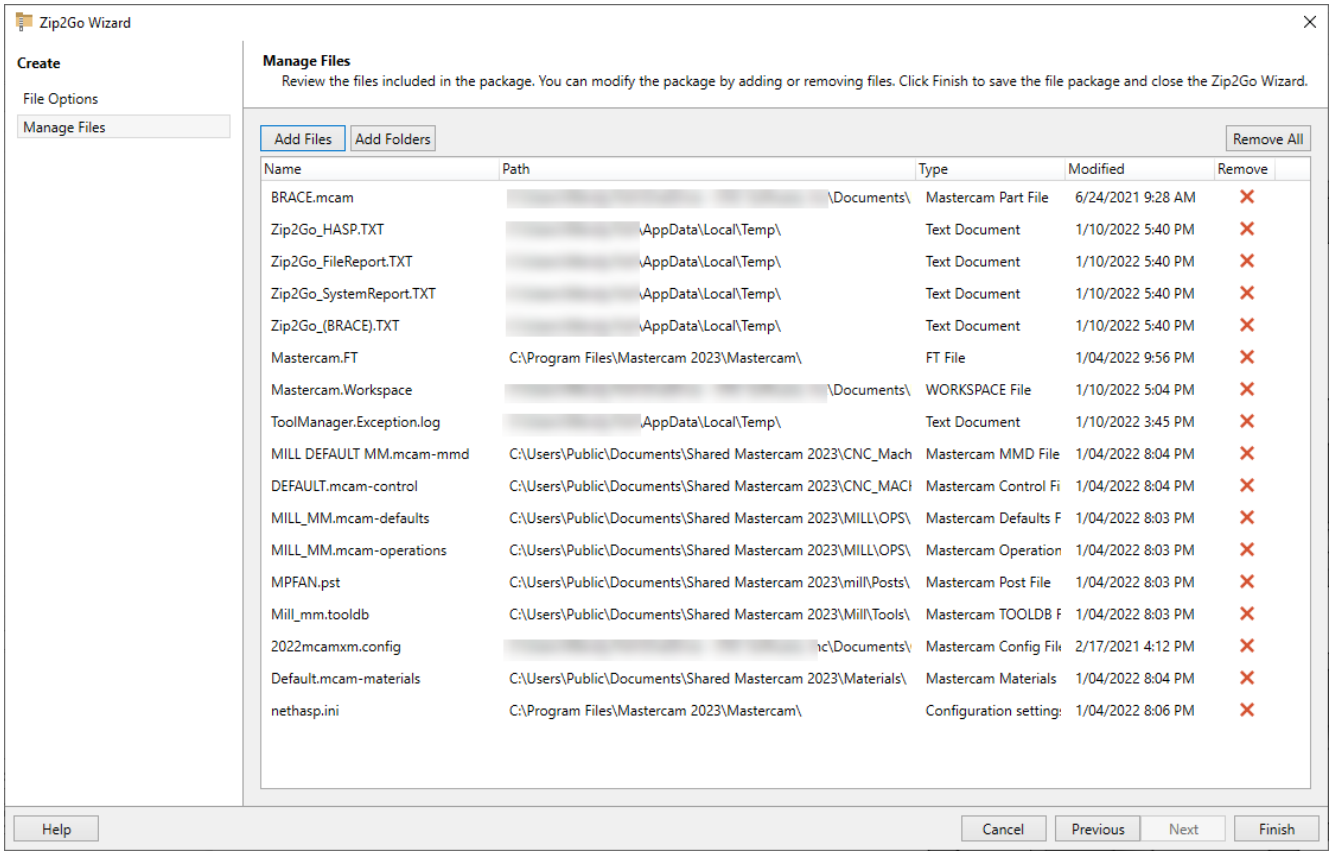
The **File Options** page displays information about the files that will be saved in the Zip2Go archive. Select the file types that you want include and deselect file types that you do not want to include in the compressed file. You can also remove specific files from the archive. For this tutorial, keep the default settings and files selected.


Zip2Go Wizard
Create
File Options
Manage Files
File Options
Select the file types and machine groups you want to include in your compressed package. Click Finish to save the file package and close the Zip2Go Wizard. Click Next to modify the file package.
File types
Part files
Operation Library files
Machine Definition files
Control Definition files
Post files
Tool libraries
Machine Simulation files
Mill-Turn machine environments
Material files
Group types
Mill
Lathe
Router
Wire
Mill Turn
Name
Path
Type
Modified
Remove
Mastercam.FT
C:\Program Files\Mastercam 2023\Mastercam\
FT File
1/04/2022 9:56 PM
Mastercam.Workspace
Documents\
WORKSPACE File
1/10/2022 5:04 PM
ToolManager.Exception.log
\AppData\Local\Temp\
Text Document
1/10/2022 3:45 PM
MILL DEFAULT MM.mcam-mmd
C:\Users\Public\Documents\Shared Mastercam 2023\CNC_Mach
Mastercam MMD File
1/04/2022 8:04 PM
DEFAULT.mcam-control
C:\Users\Public\Documents\Shared Mastercam 2023\CNC_MACH
Mastercam Control Fi
1/04/2022 8:04 PM
MILL_MM.mcam-defaults
C:\Users\Public\Documents\Shared Mastercam 2023\MILL\OPS\
Mastercam Defaults F
1/04/2022 8:03 PM
MILL_MM.mcam-operations
C:\Users\Public\Documents\Shared Mastercam 2023\MILL\OPS\
Mastercam Operation
1/04/2022 8:03 PM
MPFAN.pst
C:\Users\Public\Documents\Shared Mastercam 2023\mill\Posts\
Mastercam Post File
1/04/2022 8:03 PM
Mill_mm.tooldb
C:\Users\Public\Documents\Shared Mastercam 2023\Mill\Tools\
Mastercam TOOLDB F
1/04/2022 8:03 PM
2022mcamxm.config
\Documents\
Mastercam Config File
2/17/2021 4:12 PM
Default.mcam-materials
C:\Users\Public\Documents\Shared Mastercam 2023\Materials\
Mastercam Materials
1/04/2022 8:04 PM
nethasp.ini
C:\Program Files\Mastercam 2023\Mastercam\
Configuration setting
1/04/2022 8:06 PM
Help
Cancel
Previous
Next
Finish

5. Click **Next**.


Configuration setting
1/04/2022 8:06 PM
Cancel
Previous
Next
Finish

The **Manage Files** page displays the files that Zip2Go will include in the package. On this page, you can review, add files and folders, as well as remove specific files from the package.



6. Click **Finish** to display the **Save As** dialog box, where you can name the newly-compressed file package and save it.

   By default, Mastercam creates a zipped file with a .MCAM-CONTENT extension in the selected folder. Choose a destination and change the file type to .Z2G in the **Save as type** drop-down.

7. Click **Save**.

   Mastercam creates a compressed file in the destination folder.

8. Close Mastercam without saving your changes or leave the file open for the next exercise.

You have now completed this chapter of the *Mastercam Basics Tutorial*. In the next chapter, you learn more about viewing your part.

CHAPTER 5

VIEWING YOUR PART

Mastercam provides several tools and methods for changing the appearance of the geometry and toolpaths in the graphics window. In this chapter, you learn to view your part, use materials, analyze your part, and create and use Viewsheets.

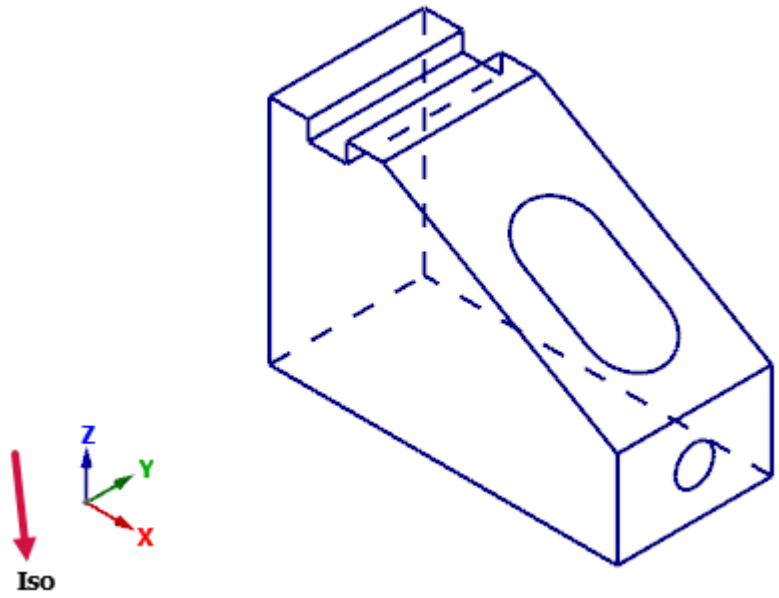
Goals

- Fit all entities in the graphics window.
- Use zoom functions to magnify your view of selected entities.
- Dynamically rotate and pan entities in the graphics window.
- Create a Viewsheet.
- Select materials.
- Analyze your part.

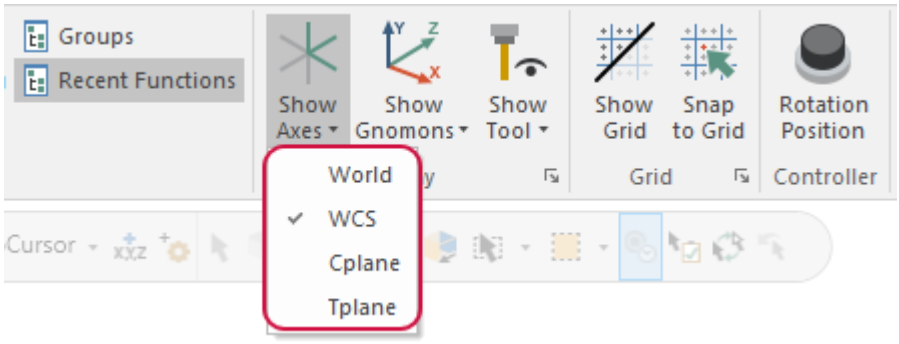
Exercise 1: Changing Standard Views

In this exercise, you learn how to look at your part from different sides and fit it to the graphics window.

1. Open ANGLEBLOCK-MM.mcam, which was provided with this tutorial. The part opens in **Isometric (WCS)** view.

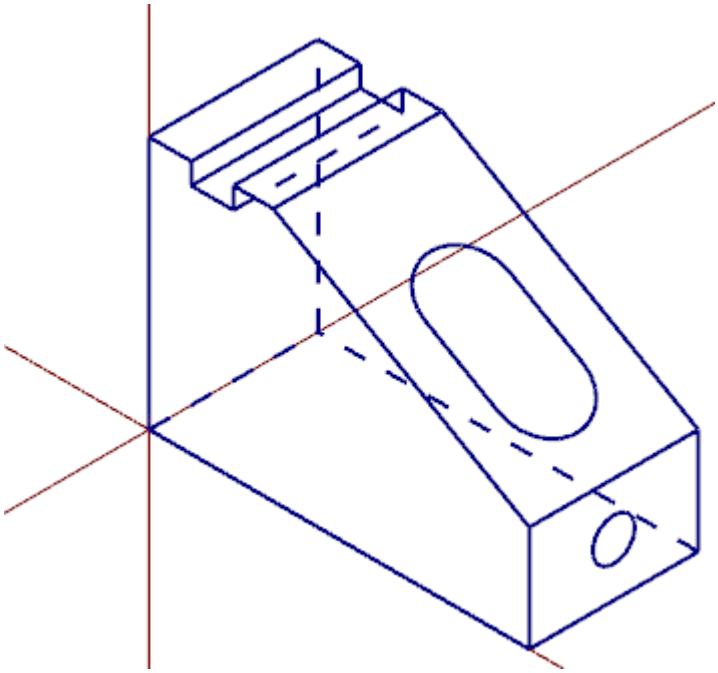


2. On the **View** tab, select the **Show Axes** drop-down. If necessary, deselect **World**, **Cplane**, and **Tplane**. Only **WCS** remains selected.

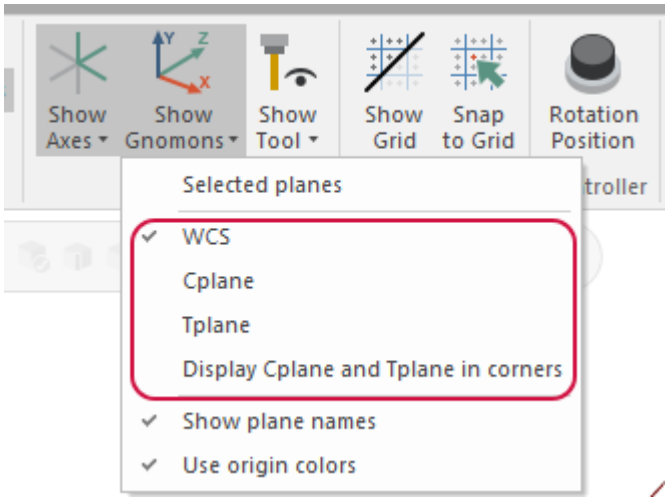


3. On the **View** tab, select **Show Axes** or press [**F9**] to toggle the display of the WCS axes.

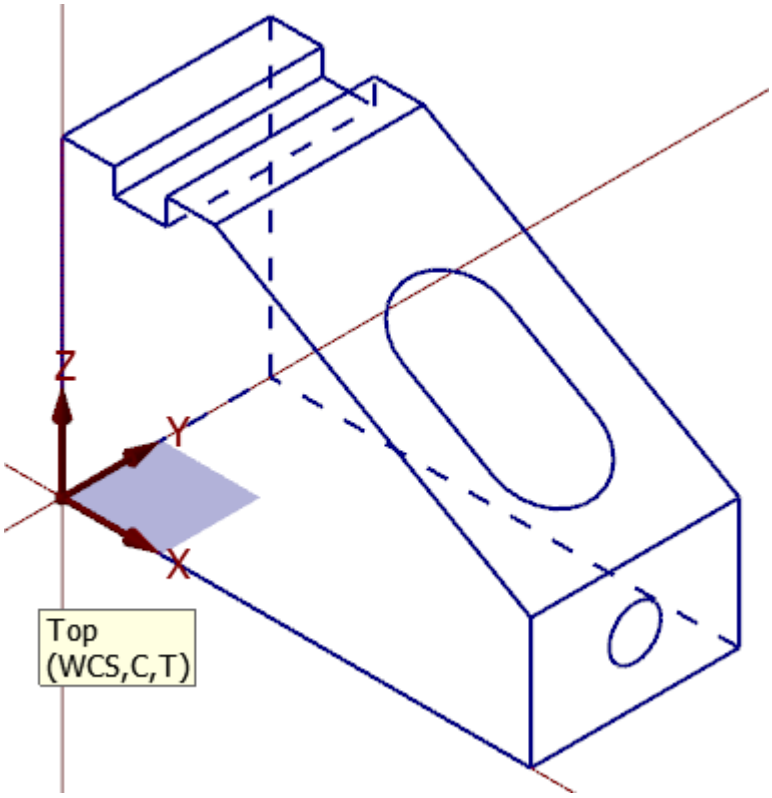
   The coordinate axes show the origin and the part orientation to help you visualize the part in 3D space.



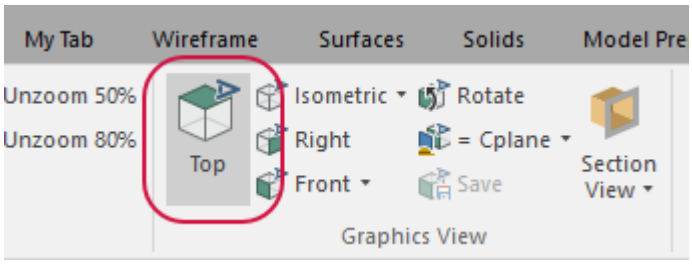
4. On the **View** tab, select the **Show Gnomons** drop-down and deselect **Cplane** and **Tplane**. Only **WCS** remains selected.



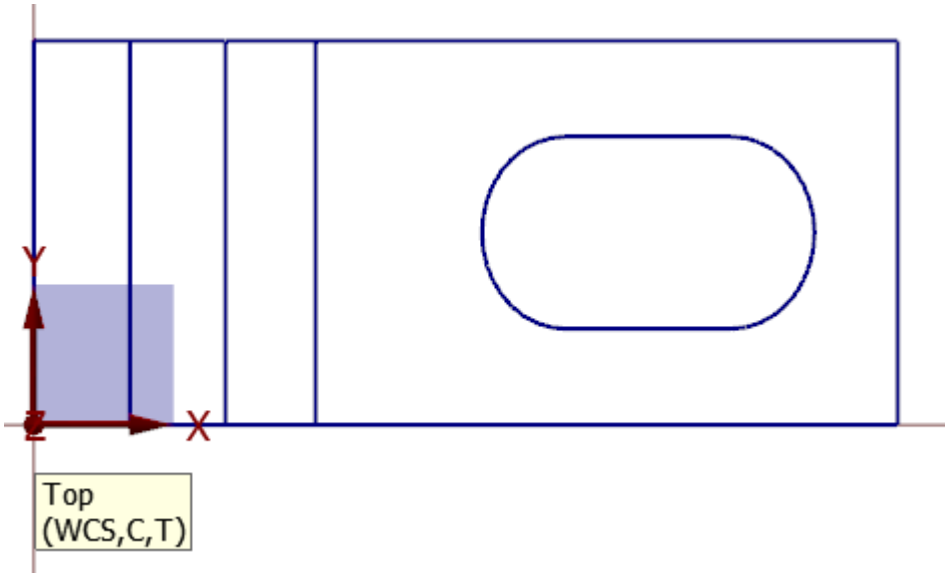
5. On the **View** tab, select **Show Gnomons**. The WCS gnomon displays. The text indicates that the WCS is set to the **Top** plane, and the Cplane and Tplane are also set to **Top**.



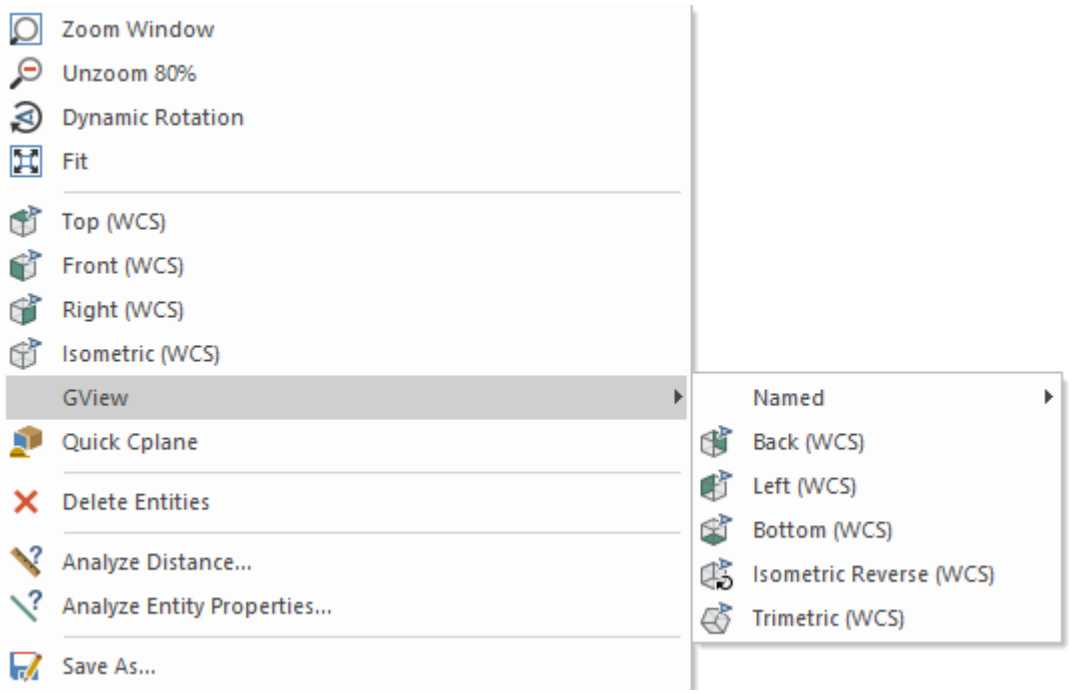
6. On the **View** tab, select **Top**.



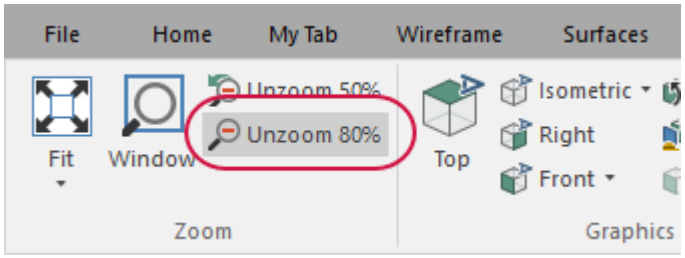
The part changes to show the **Top WCS** view.



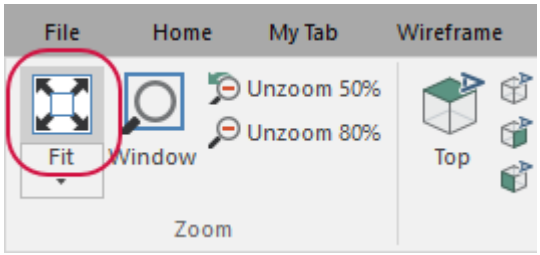
7. You can also use the right-click menu to change the view. Practice switching to other views using this menu.



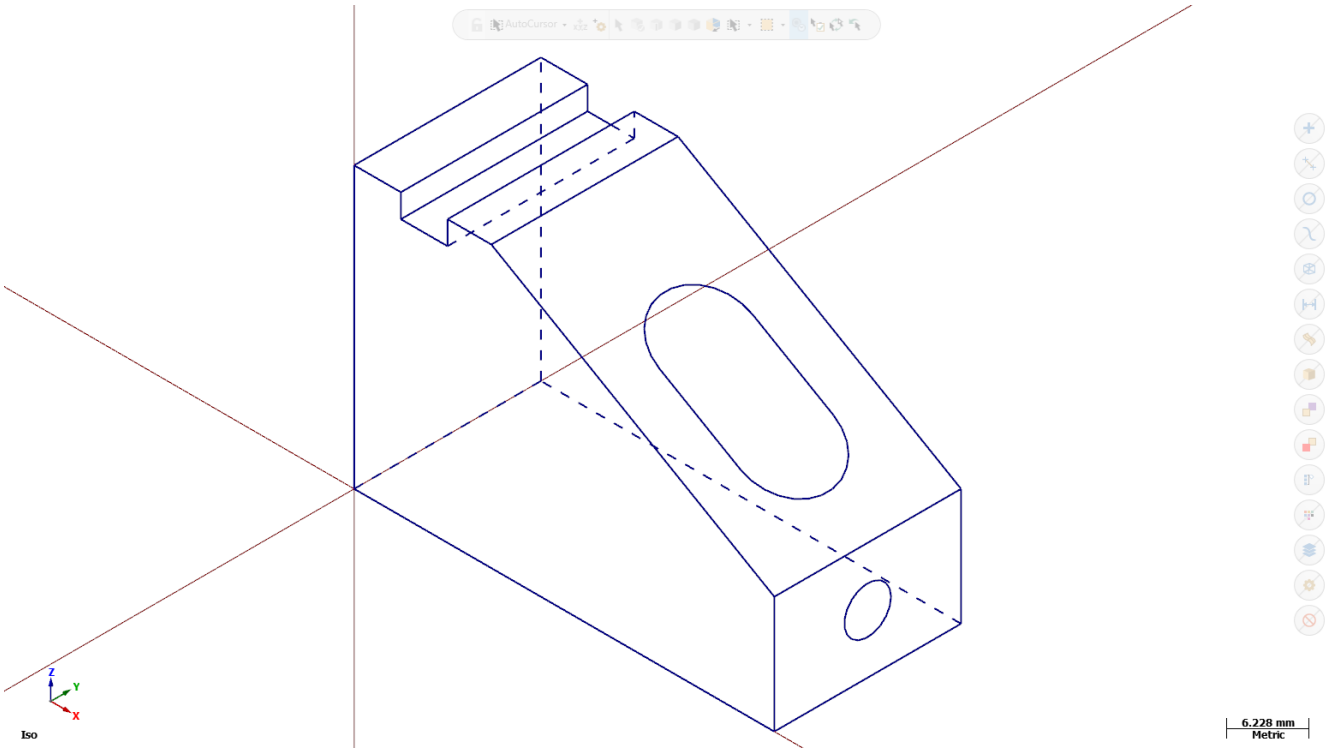
8. When finished, return the part to **Isometric (WCS)** view.
9. Click **Show Gnomons** to turn off the display of gnomons in the graphics window.
10. On the **View** tab, select **Unzoom 80%** to reduce the size of the part on the screen.



11. On the **View** tab, select **Fit**. You can also press [**Alt+F1**].



The part now fills the entire graphics window.

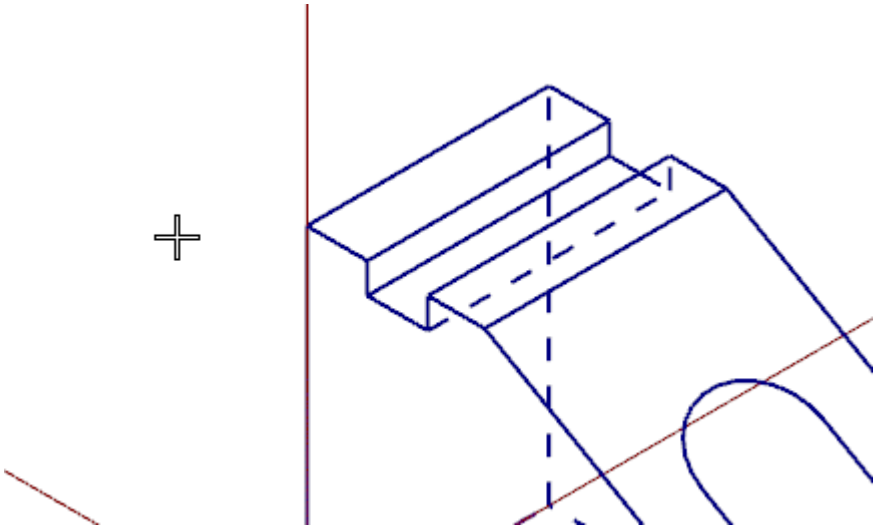


NOTE
Fit and several other **Zoom** commands are also available in the right-click menu.

Exercise 2: Zooming In and Out

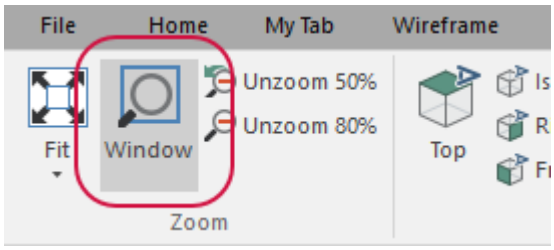
In this exercise, you practice different techniques for viewing areas of detail in your part.

1. Place your cursor in the upper left quadrant of the graphics window.

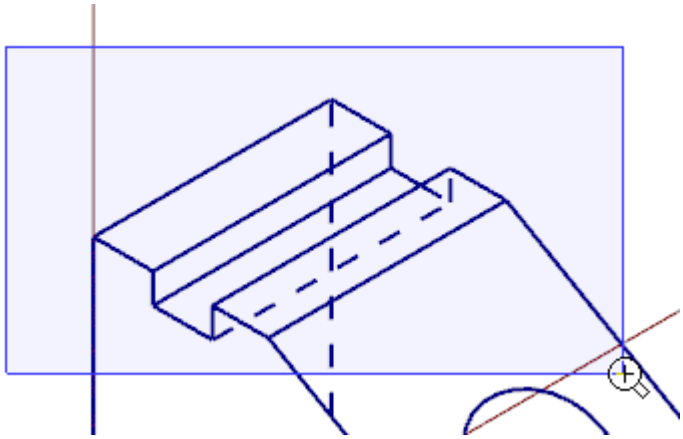


2. If your mouse has a middle mouse wheel, spin it back and forth to dynamically zoom in and out. You can also press the [**Page Up**] and [**Page Down**] keys on your keyboard to zoom in and out.
3. Click **Fit** or press [**Alt+F1**] to fit the part to the screen.

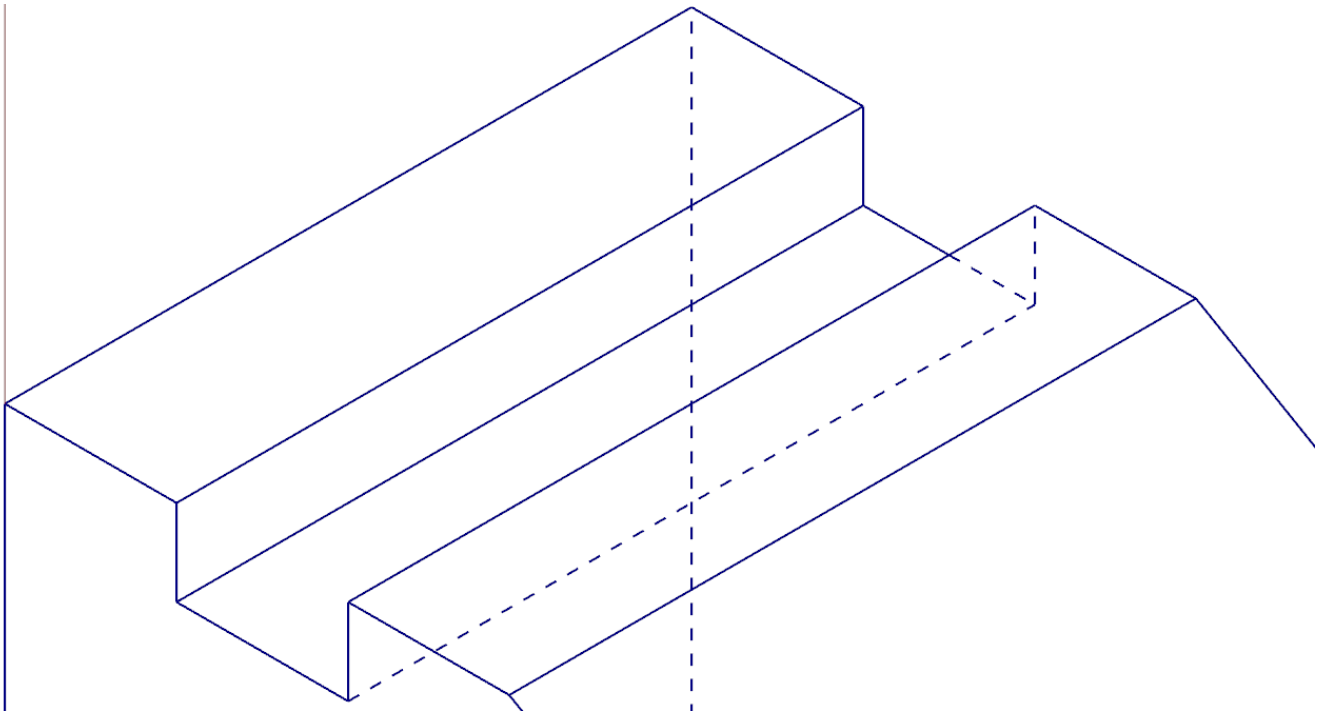
4. On the **View** tab, select **Window**.



5. Click and drag a window around the upper section of the part as shown below.

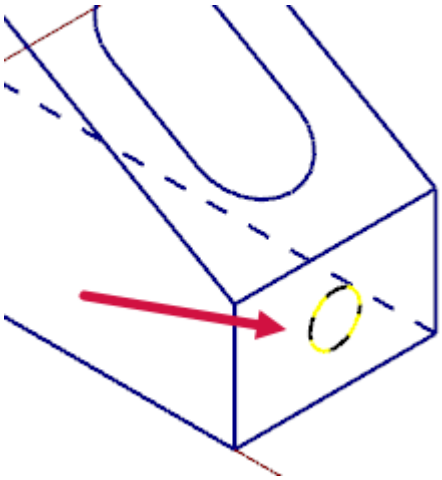


6. Click to set the zoom window. Mastercam scales the selected area to fit the entire graphics window.

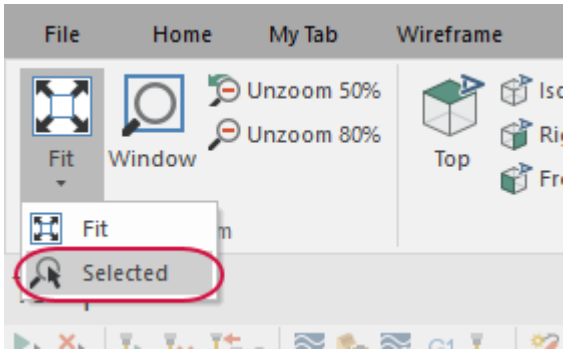


7. Right-click the graphics window and choose **Fit**.

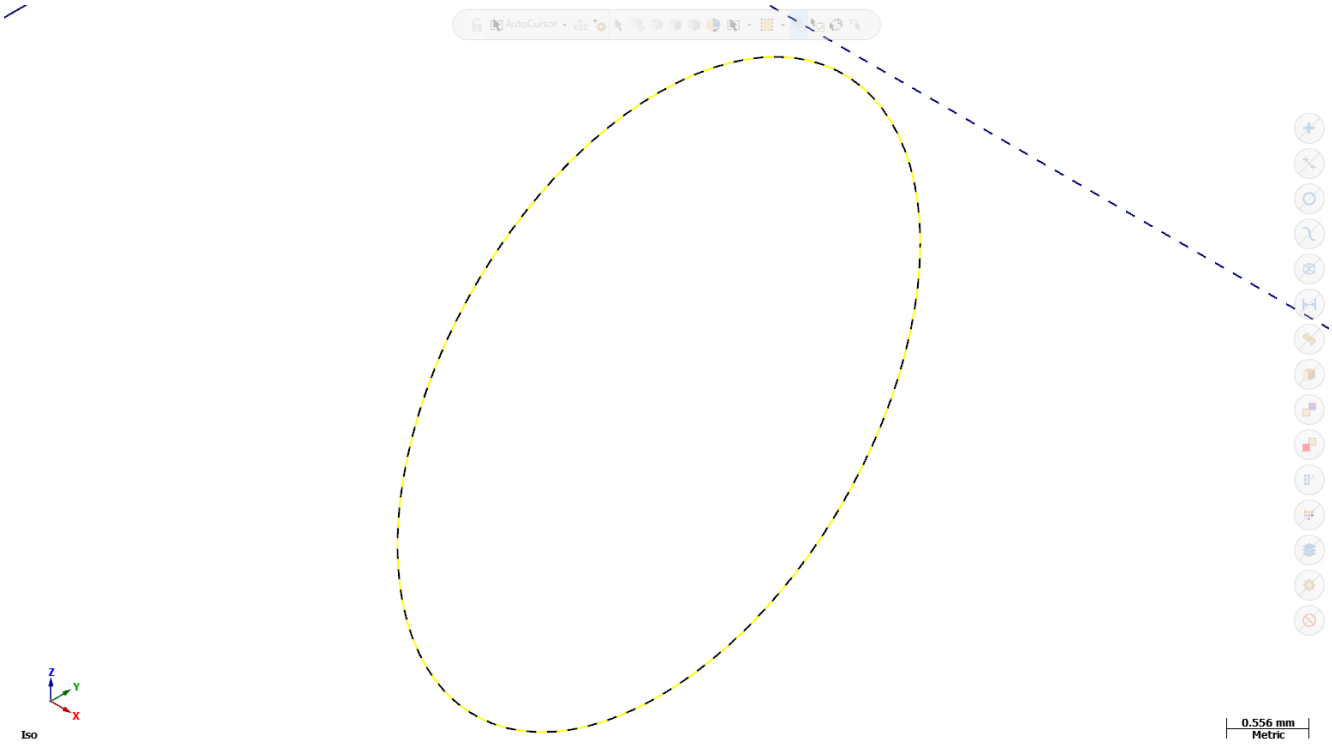
8. Select the circle on the front of the angle block.



9. On the **View** tab, choose **Selected** from the **Fit** drop-down menu.



The selected entity is scaled to fit the graphics window.



10. Press [**Esc**] to clear the selection.
11. Right-click in the graphics window and select **Fit**.

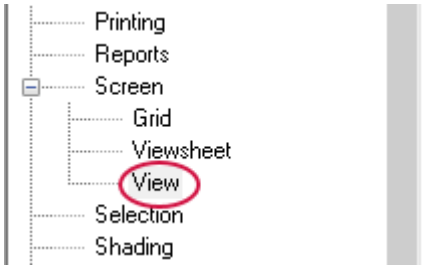
Exercise 3: Rotating Your Part View

This exercise shows how to set the preference for the action that your middle mouse button or wheel performs in the graphics window (rotate or pan). You also learn to dynamically rotate entities in the graphics window so that you can see them from any angle.

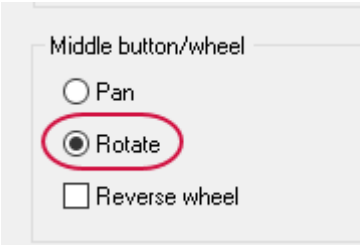
1. Select **File**, **Configuration**.

   The **System Configuration** dialog box opens.

2. Expand the **Screen** group, and select **View**.



3. If necessary, set **Middle button/wheel** to **Rotate**.



   By choosing **Rotate**, Mastercam sets dynamic rotation as the primary function when you click and hold the middle mouse button or mouse wheel.

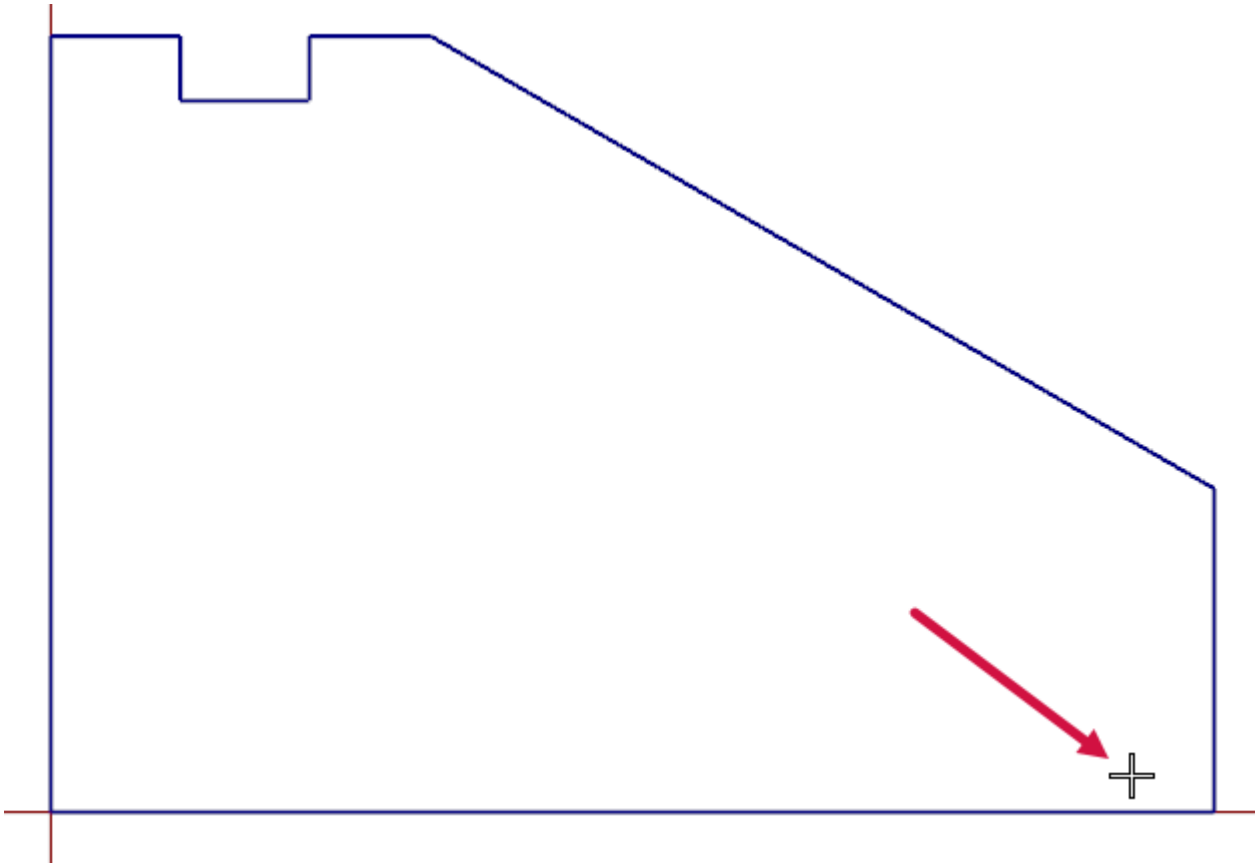
4. Click **OK** to close the **System Configuration** dialog box. If prompted to save the current settings to configuration file, click **No**.

5. Set up the part in the graphics window, as follows:

   a. Set the Gview to **Front**.

   b. Click **Unzoom 80%**.

6. Middle-click in the graphics window where indicated below and continue to hold down the middle mouse button or wheel as you move the mouse slowly in a circular motion.



This action rotates the part in space about the selected position and lets you see it from any angle.

7. Let go of the middle mouse button/wheel to finish rotating the part.

The axes symbol (gnomon) in the graphics window changes as you rotate the part. The Gview name in the graphics window no longer displays because you are not in a standard view.

Front

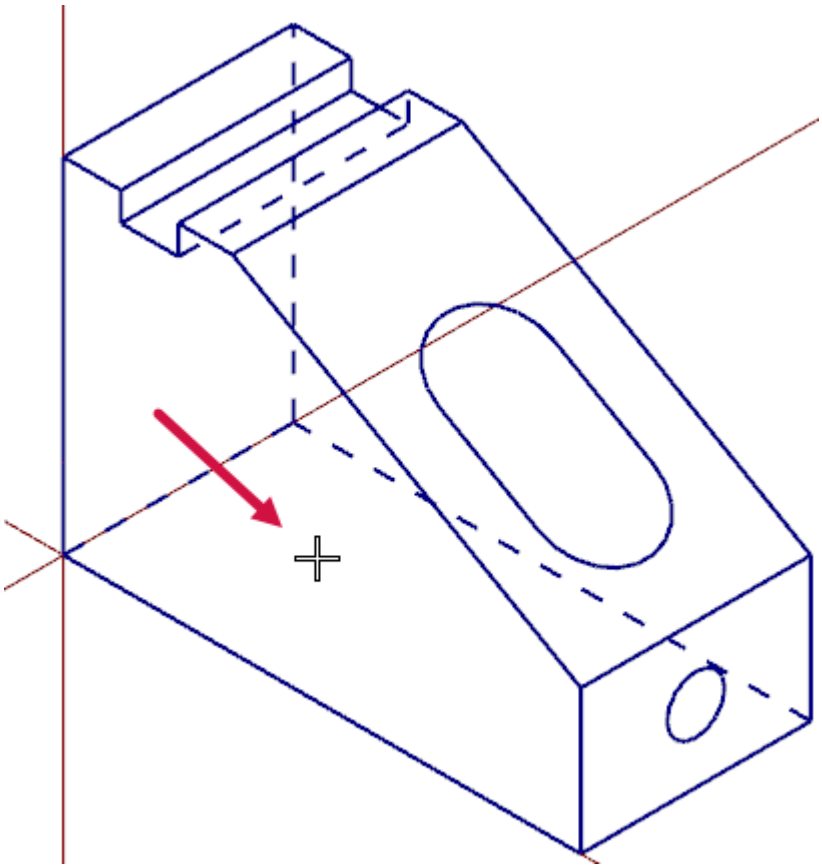
8. Return the view to **Isometric**, and fit the part to the screen.

You can also select **Dynamic Rotation** from the right-click menu and follow the on-screen prompts to rotate the part.

Exercise 4: Panning Your Part

In this exercise, you change your view in the graphics window by using shortcut keys.

1. Hold down the [**Shift**] key, middle-click in the graphics window where indicated, and continue to hold down the button/wheel while you move the mouse up, down, side to side, and around.



It appears that you have picked up the part and are moving it in the direction of the mouse. However, the entities are not physically moved in space, only the display changes. The GView does not change as you pan the entities.

2. Release the [**Shift**] key while still holding down the middle mouse button/wheel and you can rotate the part. Press the [**Shift**] key again to return to panning.
3. When you finish panning, release the [**Shift**] key and the middle mouse button/wheel at the same time to exit the function.

Exercise 5: Analyzing Your Part

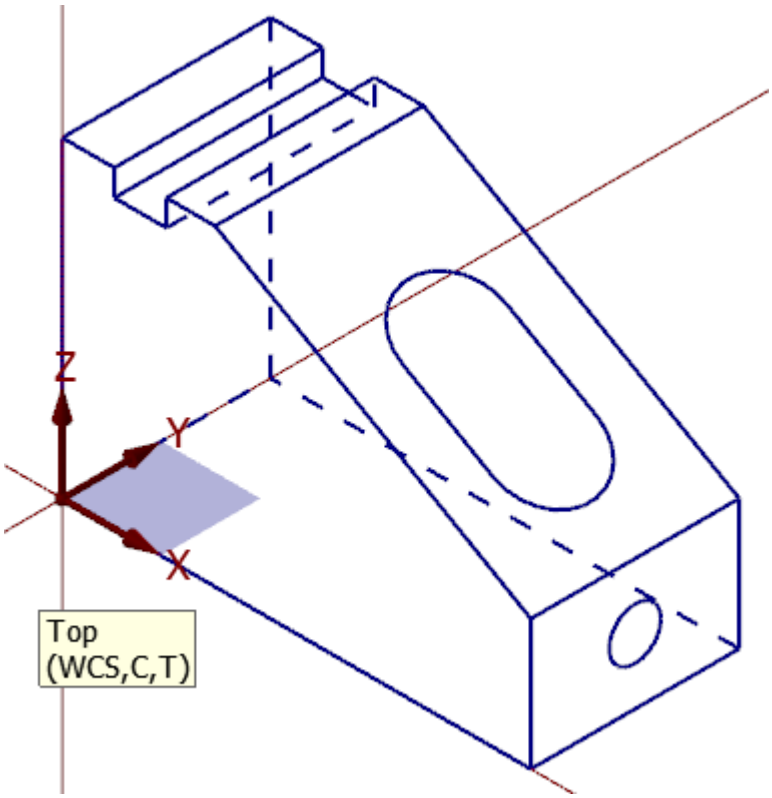
Mastercam's **Analyze** functions allow you to measure distances between lines, arcs, points, angles, and more. Depending on the type of entity you analyze, these functions can provide information about the following:

- Entity type
- Coordinates
- Operations
- Measurements
- Associated toolpaths

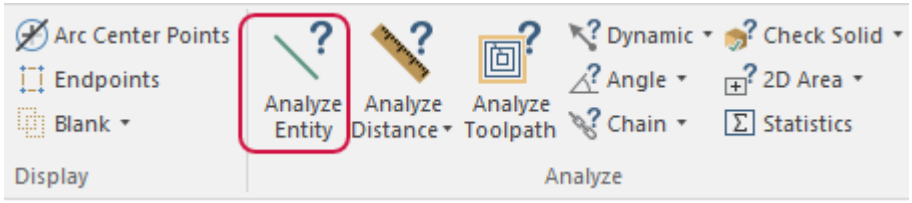
The **Analyze** functions also allow you to edit features, such as the entity's name, measurement units, and levels.

In this exercise, you use the **Analyze** functions to view and edit the properties of your wireframe.

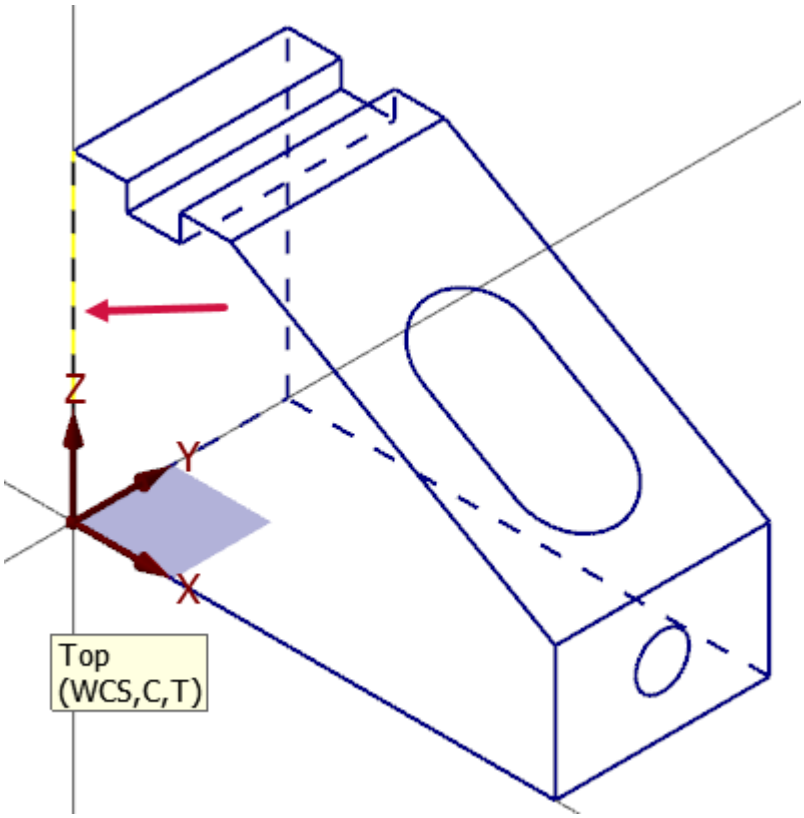
1. Set your view to **Isometric** and press [**Alt + F9**] to show gnomons.



2. On the **Home** tab, select **Analyze Entity**.



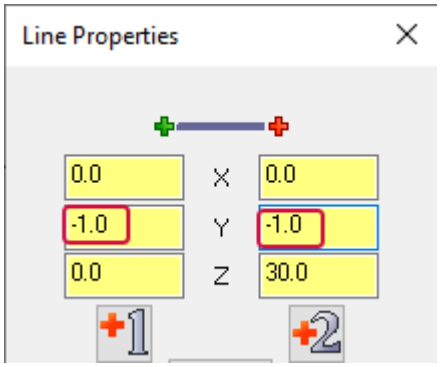
3. In the graphics window, select the line shown in the following image.



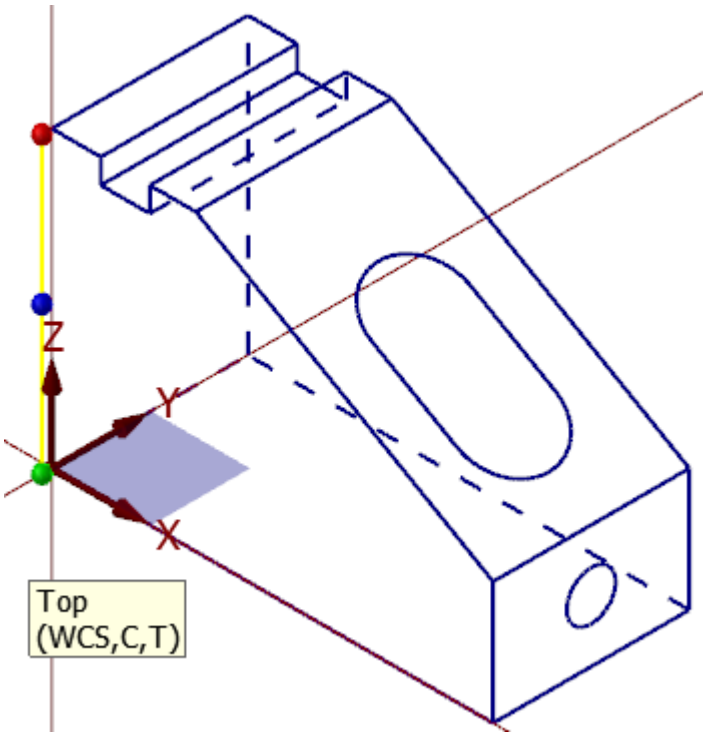
The **Line Properties** dialog box opens. **Analyze Entity** recognizes you have selected a line and shows information such as the X / Y / Z endpoint coordinates, the angle, and the level.

> **NOTE**
> Analyze functions report position coordinates in relation to the Cplane origin.

4. Enter **-1.0** in the **Y** position field for the start and end points.



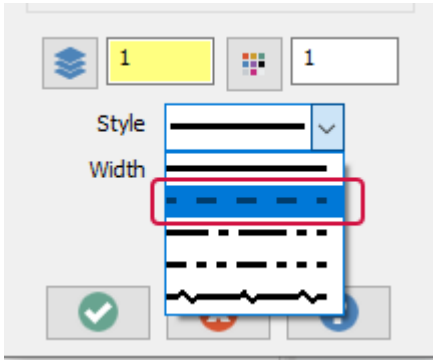
5. Press [**Enter**].



The line moves -1 units on the Y axis.

NOTE
Top-Level Editing allows you to adjust the geometry of your part from within the graphics window. To learn more about Top-Level Editing, see "Editing Geometry from the Top-Level" on page 49 in this tutorial or visit the Mastercam Help Center.

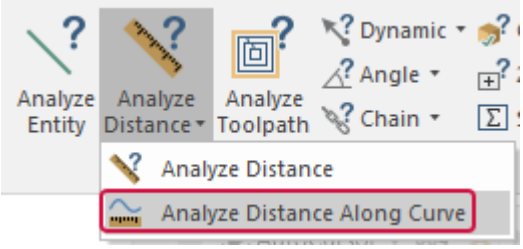
6. From the **Style** drop-down, select the option shown below.



The selected line changes to dashed. You can also change its level, color, and width. Use this function to adjust units and precision as well.

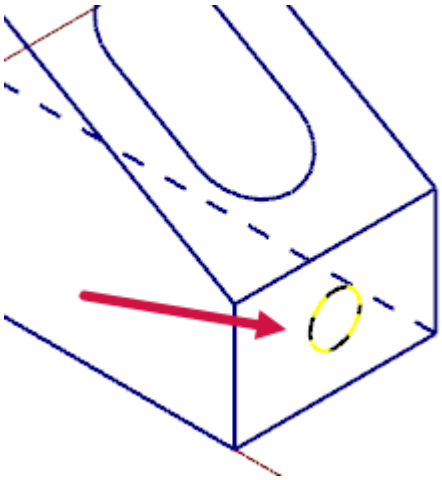
7. Click **Cancel** or press [**Esc**] to exit the **Analyze Entity** function. If you select **OK**, the changes are applied.

8. On the **Home** tab, expand the **Analyze Distance** drop-down and select **Analyze Distance Along Curve**.

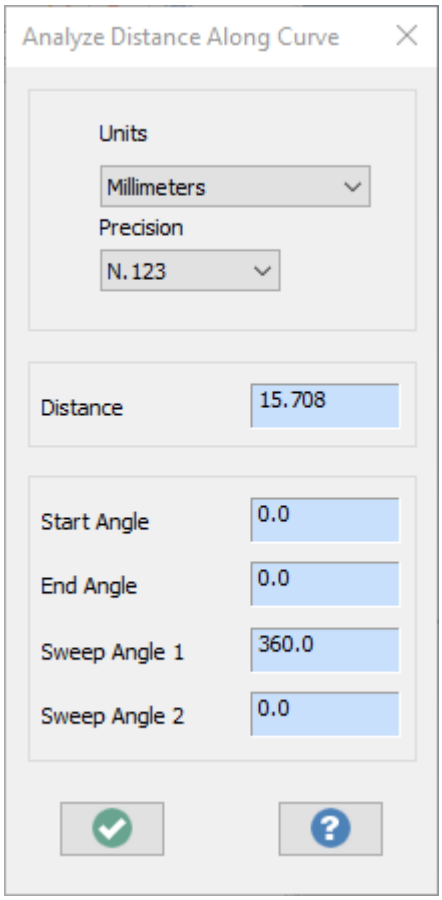


The **Analyze Distance Along Curve** dialog box appears.

9. Select the circle shown below.



The **Analyze Distance Along Curve** dialog box updates to show information about your selection. It lists the distance along the curve and the angle information.



You can use the **Units** and **Precision** drop-downs to adjust the measurements.

10. Press [**Esc**] to exit the function.
11. Press [**Alt** + **F9**] to turn off the gnomons.

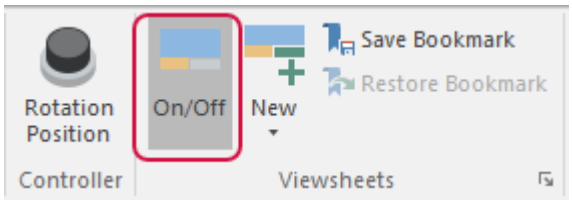
Exercise 6: Using viewsheets

Mastercam allows you to view your part in different orientations by using viewsheets. Viewsheets make viewing a large part easier, as you can set up multiple viewsheets with different views.

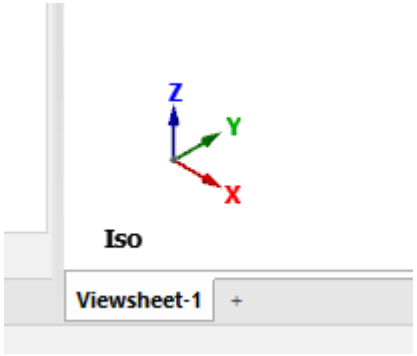
Additionally, you can create a bookmark for individual viewsheets and/or create groups of multiple viewsheets. A viewsheet bookmark holds the settings for the view displayed in the viewsheet. When you save your part file, Mastercam stores the viewsheet settings, including the bookmark, along with the part. When working with your part, you can restore the viewsheet to its saved view by restoring the bookmark.

In this exercise, you create a viewsheet.

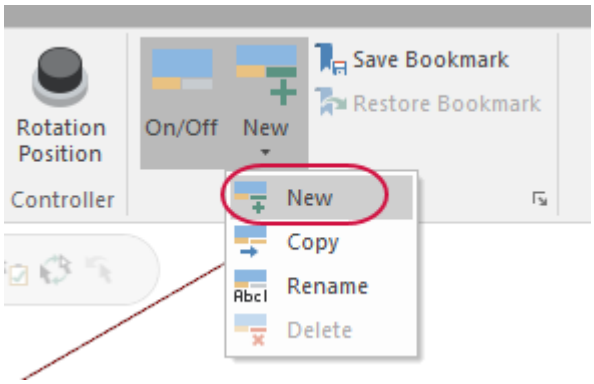
1. If necessary, fit the part to the graphics window, and set the view to **Isometric**.
2. On the **View** tab, select **On/Off** to toggle viewsheets on.



At the bottom of the graphics window, there is now a tab named **Viewsheet–1**.

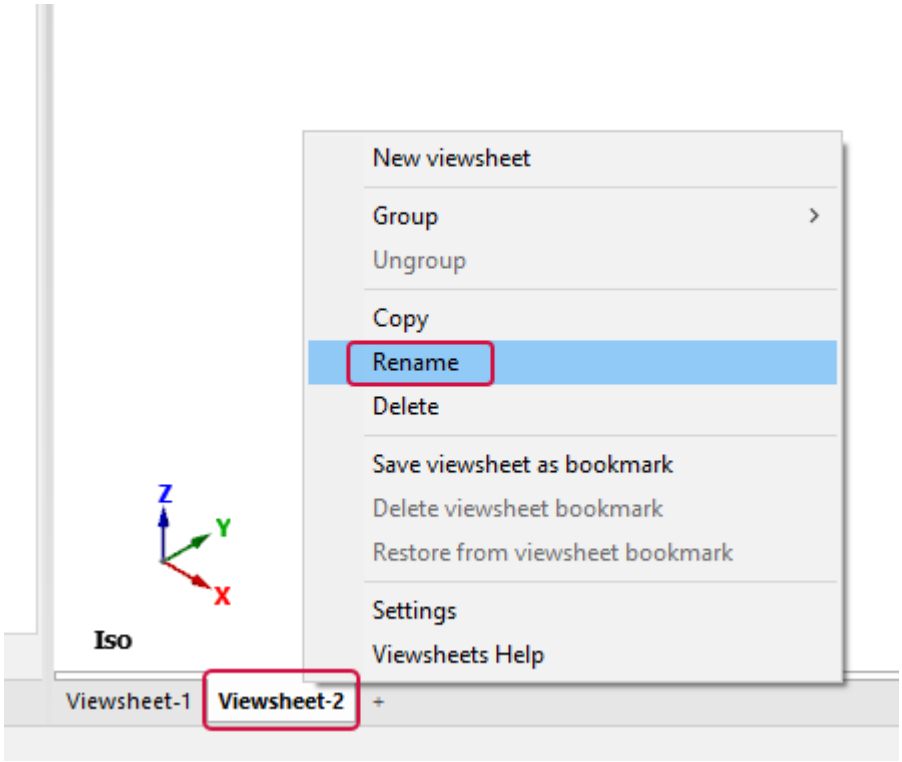


3. On the **View** tab, select **New** from the **New** drop-down.

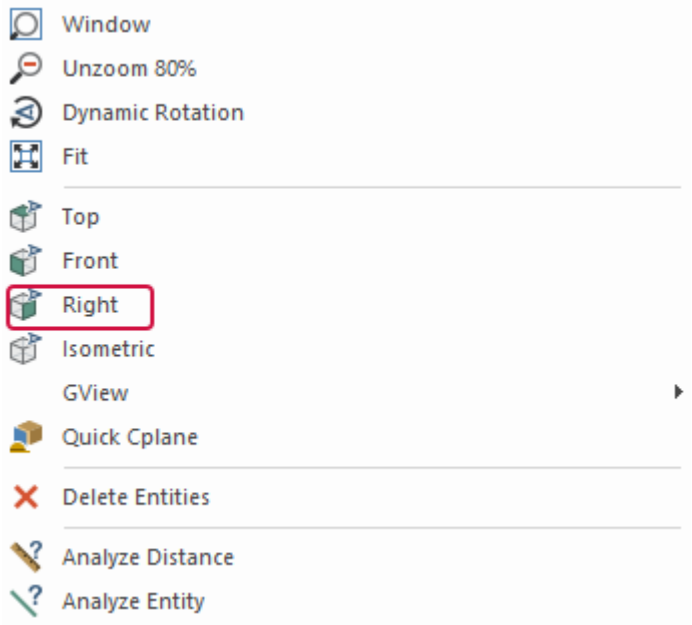


Mastercam creates a new viewsheet, **Viewsheet–2**.

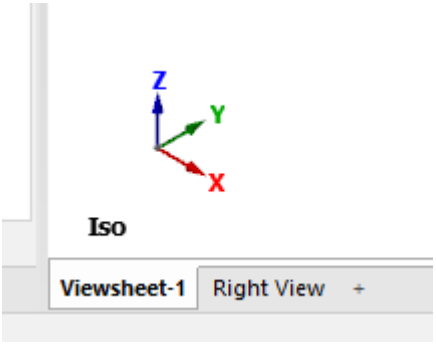
4. Right-click **Viewsheet–2** and select **Rename** from the menu. Enter **Right View** and press [**Enter**] to accept the name.



5. Right-click in the graphics window, and select **Right** to view the part from the right.

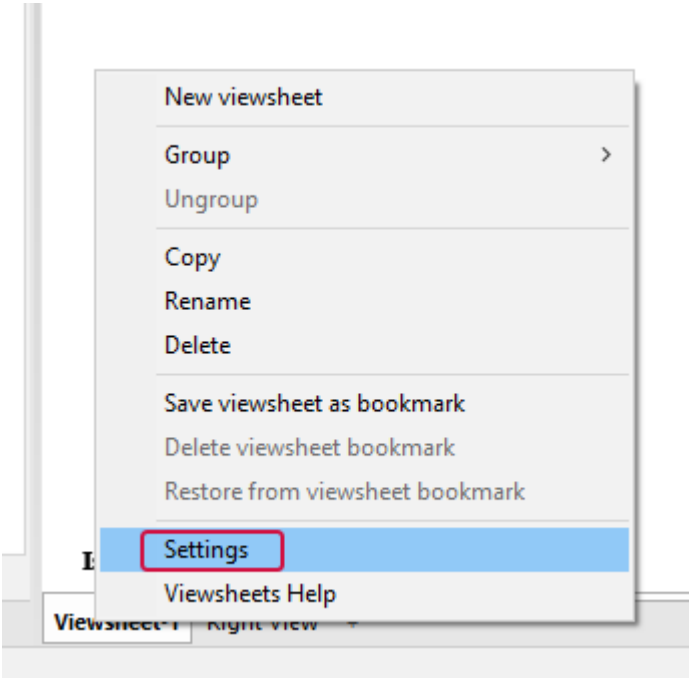


6. Select **Viewsheet–1** again.

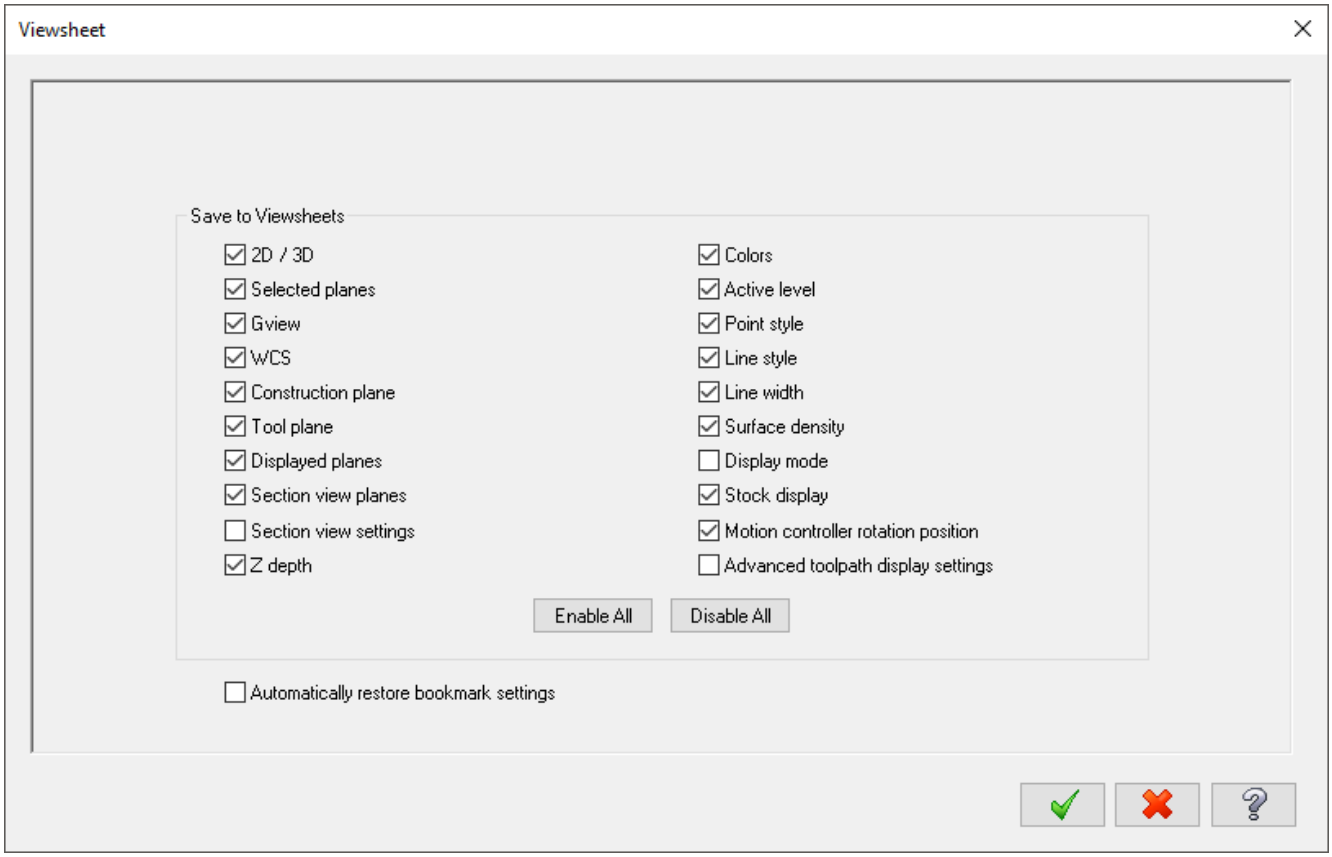


Notice that the this viewsheet still shows the isometric view. Each viewsheet can contain different views and planes.

7. Right-click **Viewsheet–1** and select **Settings**.



The **Viewsheet** dialog box opens.



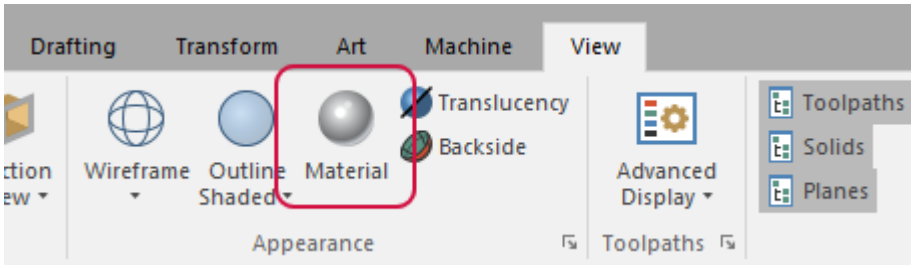
Use the **Viewsheet** dialog box to indicate which settings to save to the viewsheets. You can also access the viewsheet settings on the **Viewsheet** page of **System Configuration**.

8. Click **OK** to exit the **Viewsheet** dialog box without making changes.

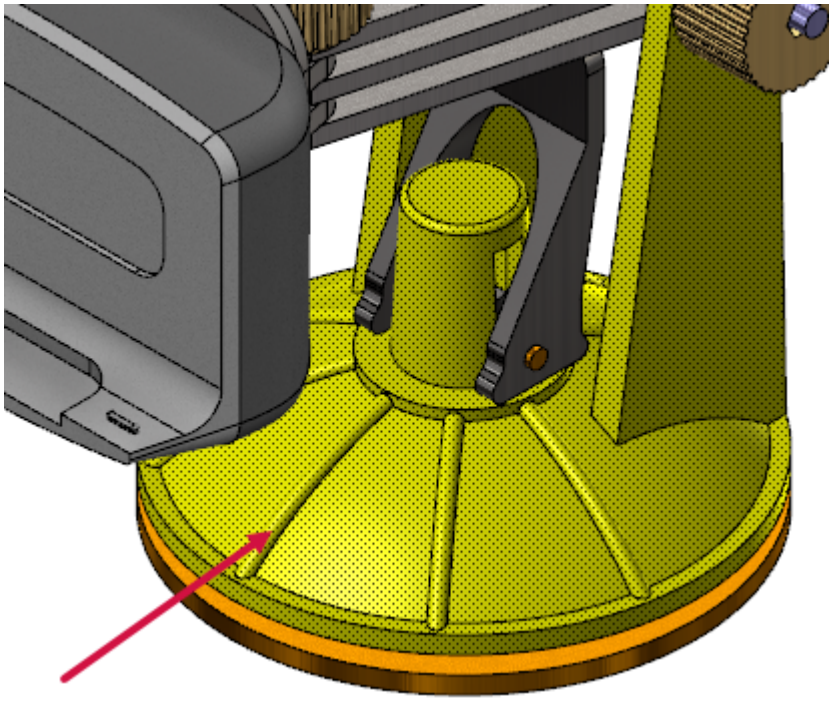
Exercise 7: Using Materials

In this exercise, you use Mastercam's material options to make your parts look more realistic.

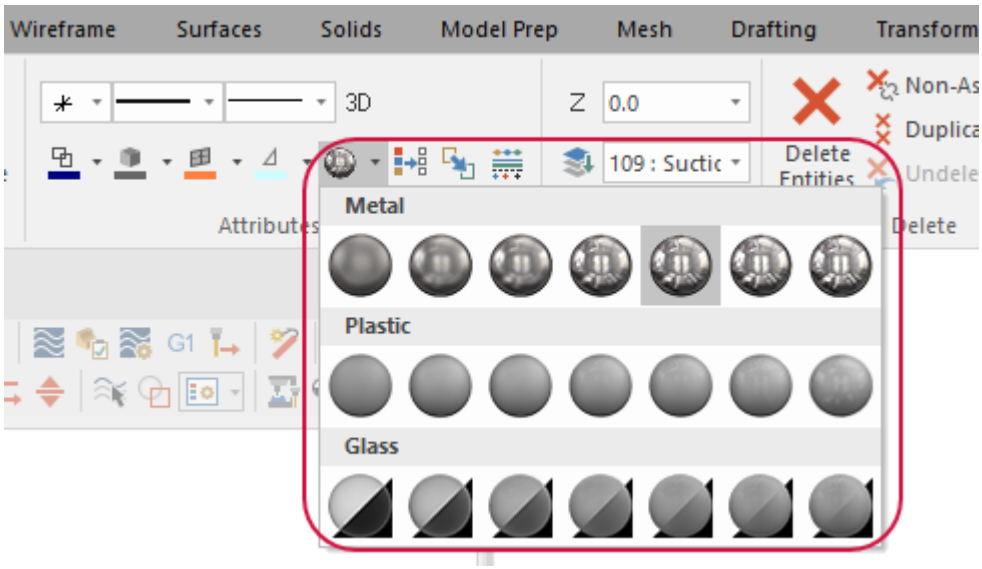
1. Open the GPS SUCTION MOUNT.mcam file, which is included with this tutorial. If prompted to save ANGLEBLOCK-MM.mcam choose **Don't Save**.
2. Press [**F9**] to hide the gnomon axes.
3. On the **View** tab, select **Material**.



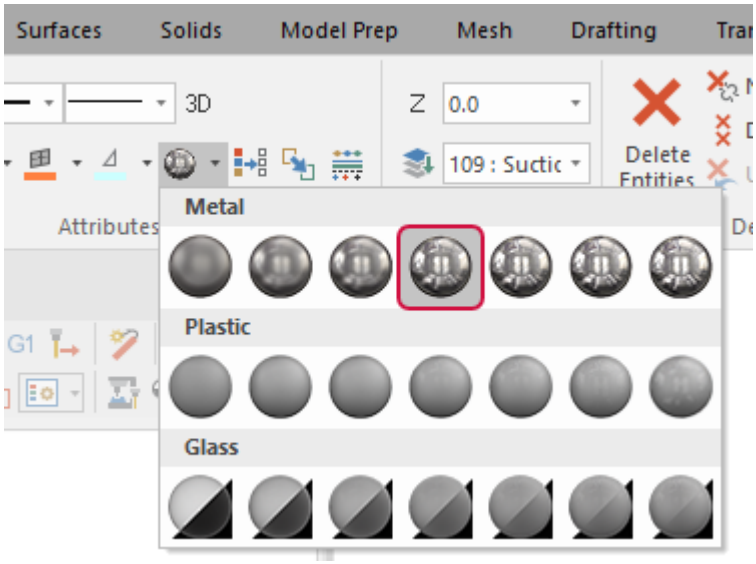
4. In the graphics window, click the base of the part to select it.



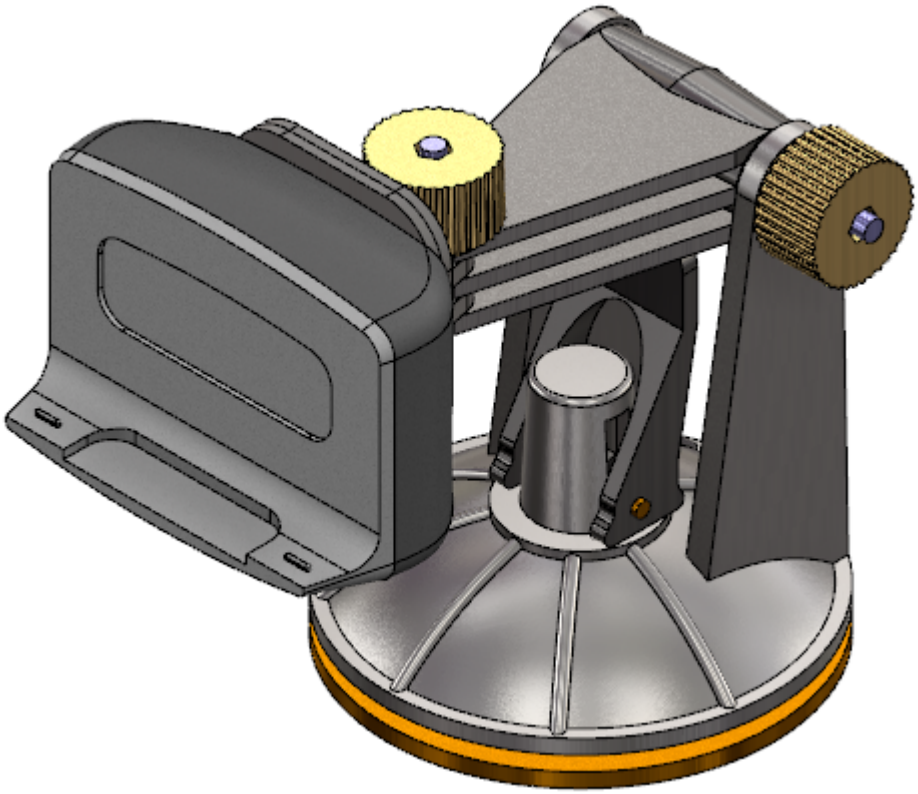
5. On the **Home** tab, select the **Set Material** drop-down to display the material type gallery.



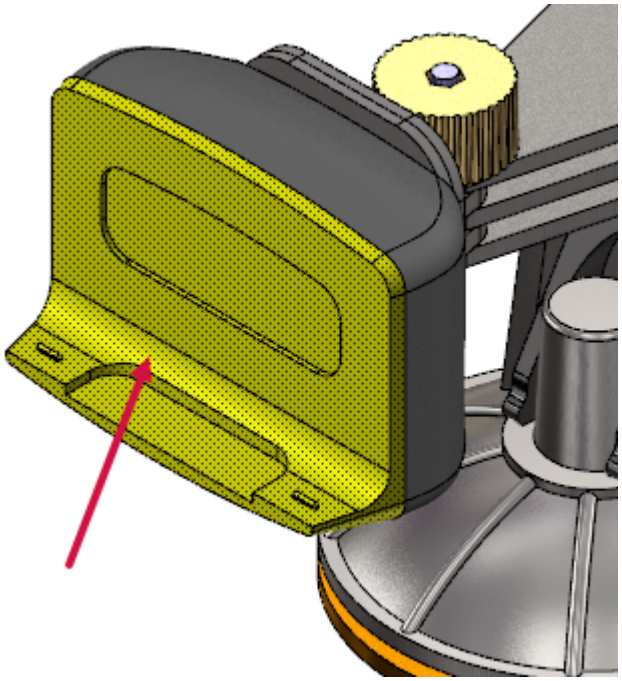
6. In the **Metal** group, select the **Metal Level 4** from the gallery.



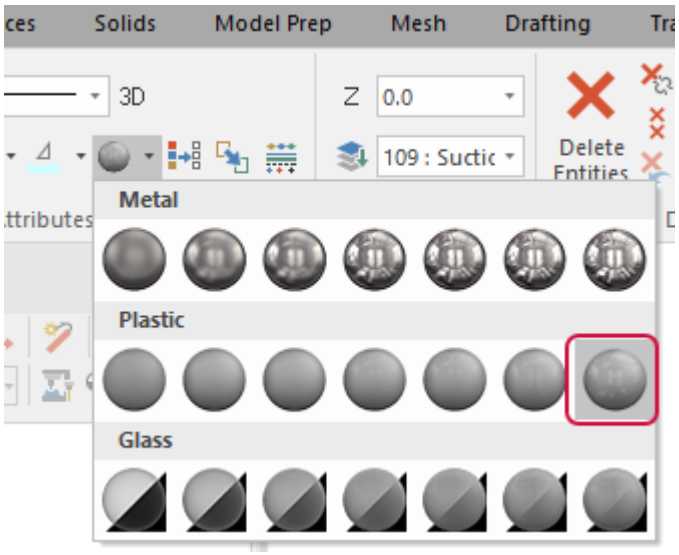
The part's base changes to the selected material, a semi-reflective metal.



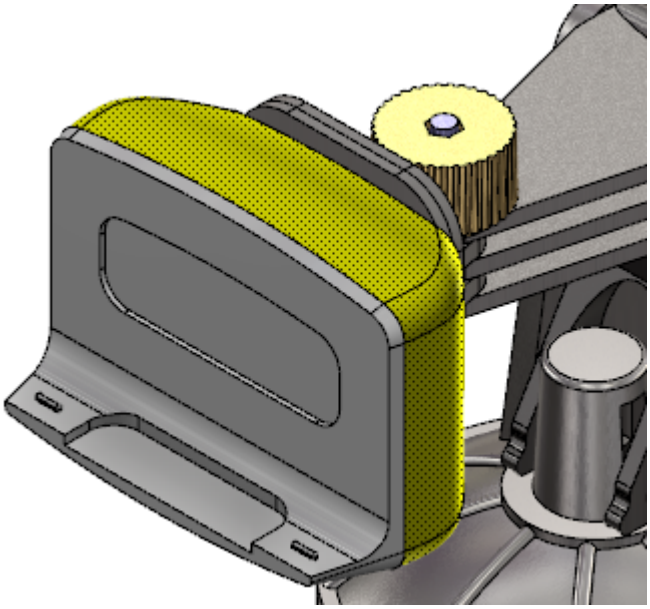
7. Select the face of the clamp, as shown below.



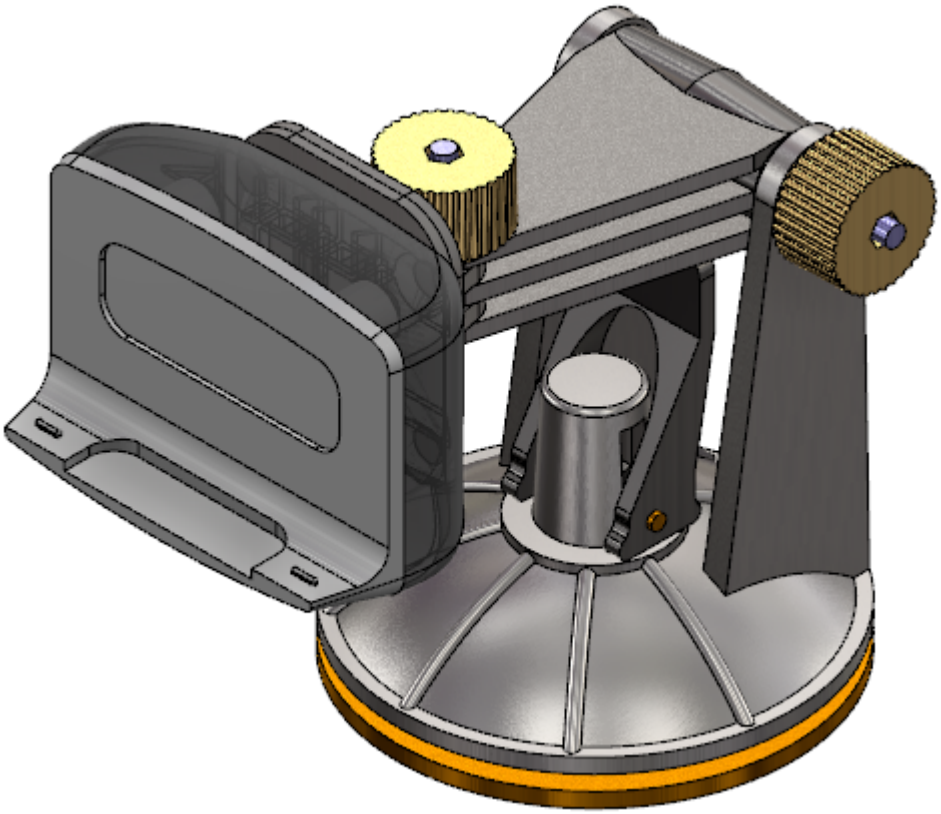
8. On the **Home** tab, select the **Set Material** drop-down and select **Plastic Level 7**.



9. Select the casing behind the clamp face, and choose **Glass Level 3** from the **Set Material** drop-down.



10. Rotate and zoom in on the part to view the different types of material.

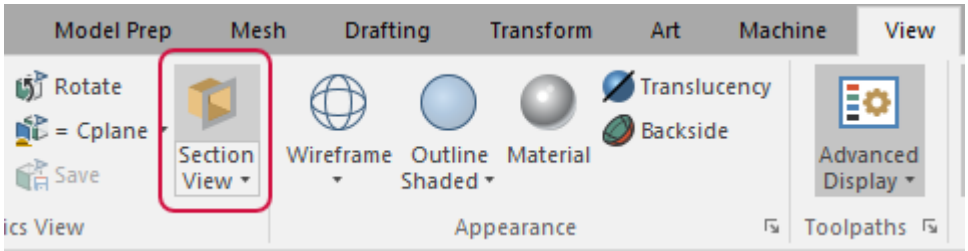


11. On the **View** tab, toggle **Material** on and off to see how the material options affect the part.
12. Before moving to the next exercise, ensure that **Material** is toggled off.

Exercise 8: Managing Section Views

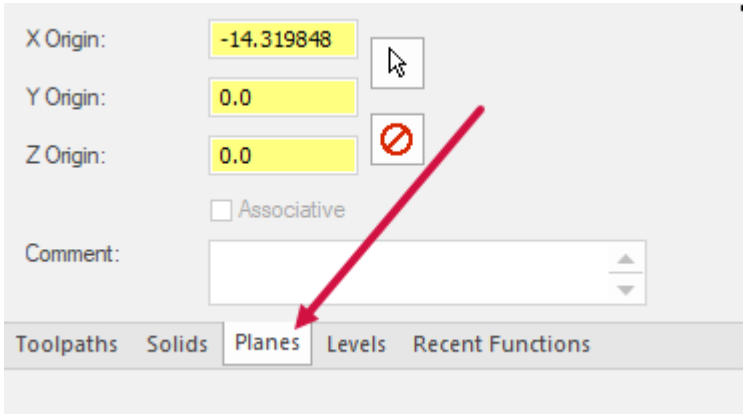
In this exercise, you use Mastercam's section views to show cross sections of a part.

1. Load the file SPOOL BODY.mcam, which is included with this tutorial. If prompted to save GPS SUCTION MOUNT.mcam choose **Don't Save**. If you see a message for switching units, click **OK** to allow the change.
2. On the **View** tab, select **Section View** to toggle section views on.

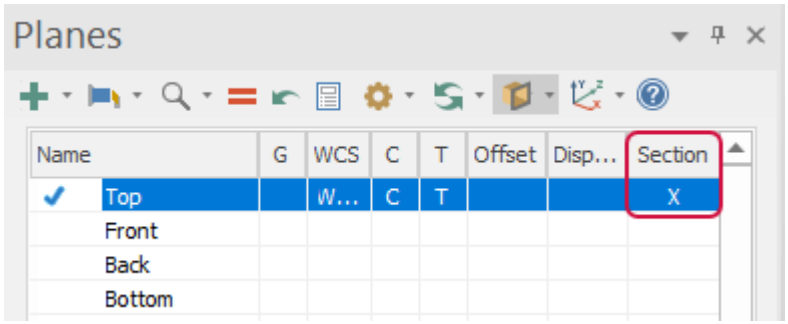


Section views only show in the graphics window when the feature is on.

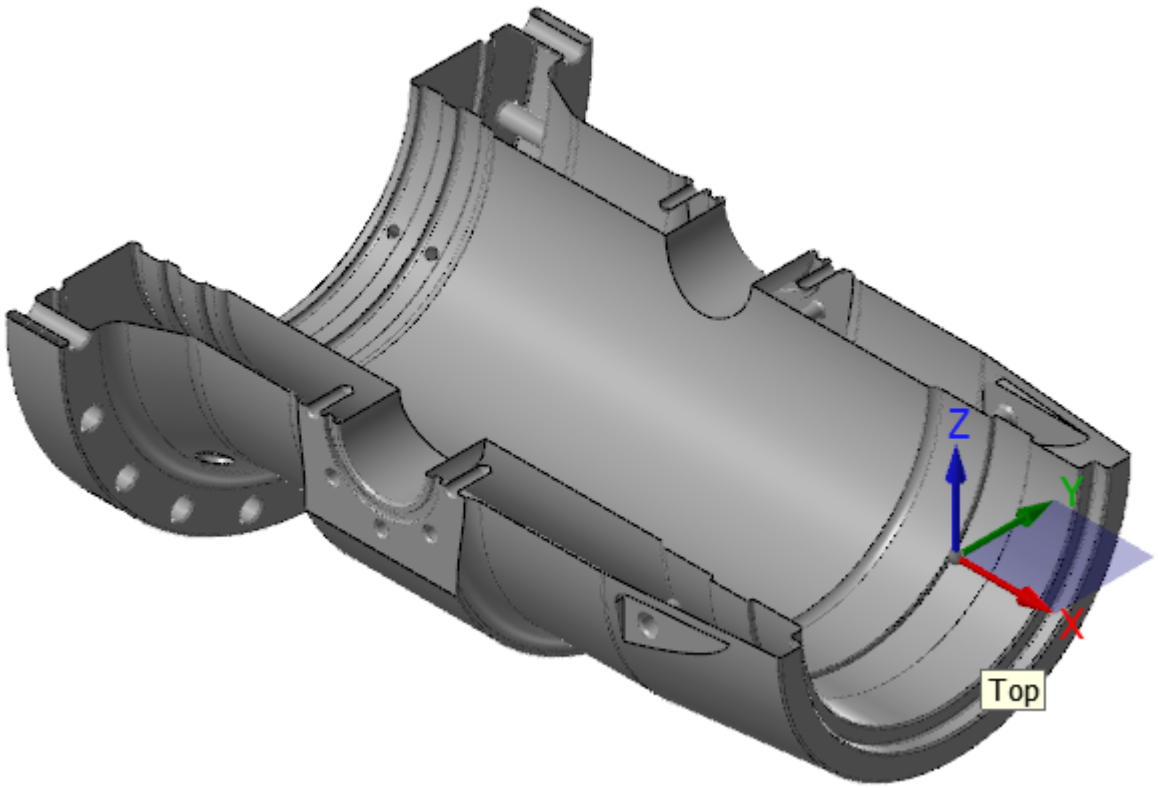
3. Select the **Planes** tab to bring the Planes Manager to the front.



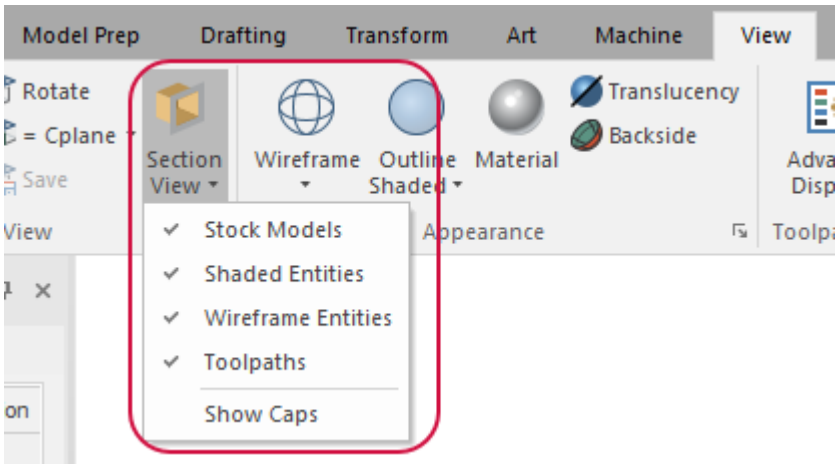
4. In the **Top** plane row, look at the **Section** column. An X indicates that a section view is created from the plane when the **Section View** is on.



Mastercam displays a cross section of the part, using **Top** as the clipping plane, as shown below.

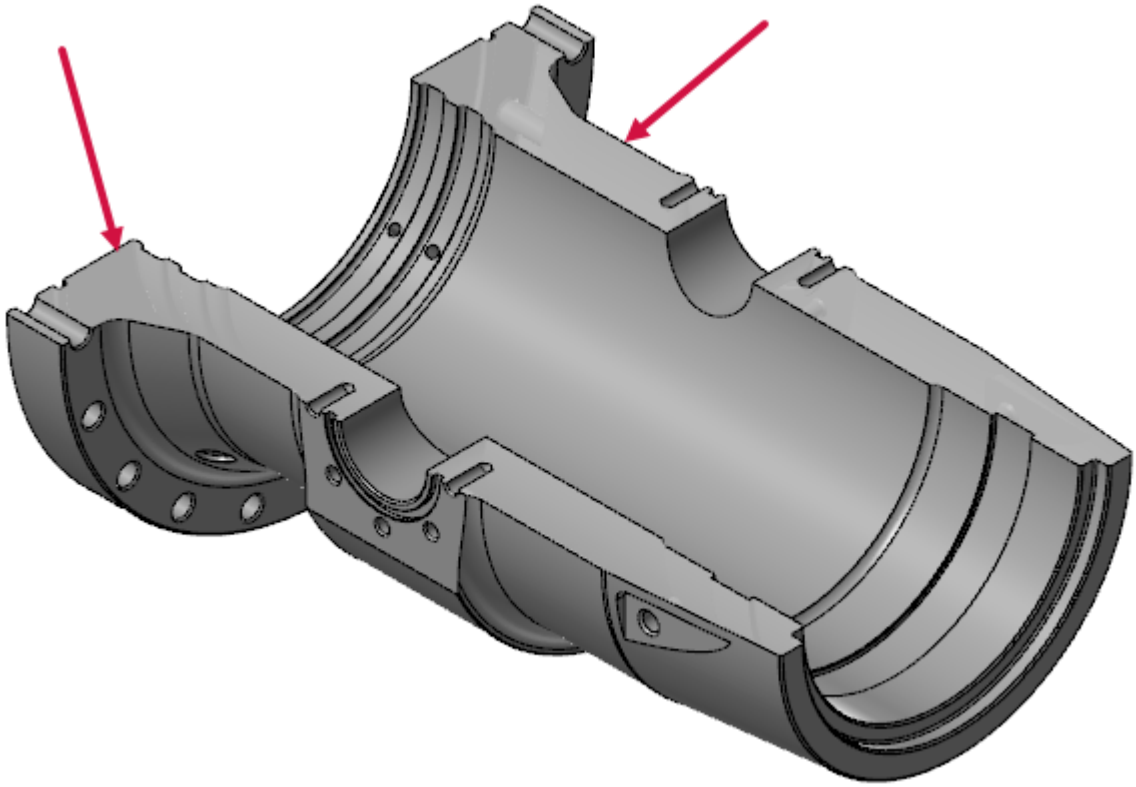


5. On the **View** tab, click the **Section View** drop-down.

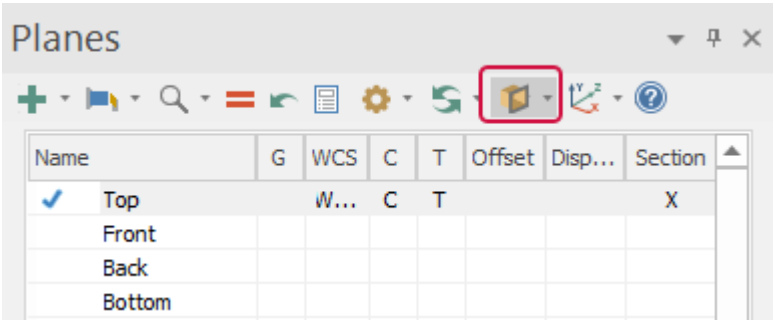


Use this menu to select what types of entities are affected when using section views.

6. From the **Section View** drop-down menu, select **Show Caps**. Mastercam adds solid caps to the sectioned areas, as shown below.



You can also access **Section View** from the Planes Manager's toolbar.



7. Close Mastercam without saving your changes or leave the file open for the next exercise.

You have now completed this chapter of the *Mastercam Basics Tutorial*. In the next chapter, you learn more about levels.

CHAPTER 6

WORKING WITH LEVELS

A Mastercam file can contain separate levels that include wireframe, solids, surfaces, drafting entities, and other part data. Organizing your geometry by levels lets you control which areas of the part are visible and which entities you can select in the graphics window.

This control makes it easier to work with the file and helps prevent you from affecting areas of the drawing you do not want to change.

You can create and name as many levels as you need and set any one to be the active, current working level. For each level you create, you assign a unique number and, optionally, a name. Mastercam's Levels Manager provides a central location where you can view and create levels, and set their properties.

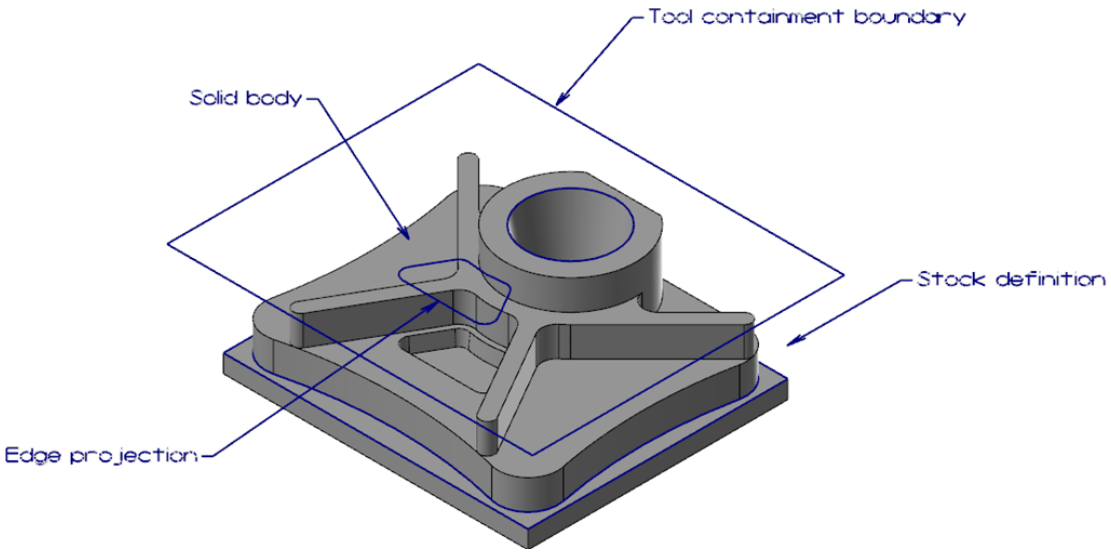
Goals

- Explore the Levels Manager.
- Modify the Levels Manager display options.
- Turn level displays off and on.
- Change the active level.
- Create a new level.
- Move selected entities to a different level.

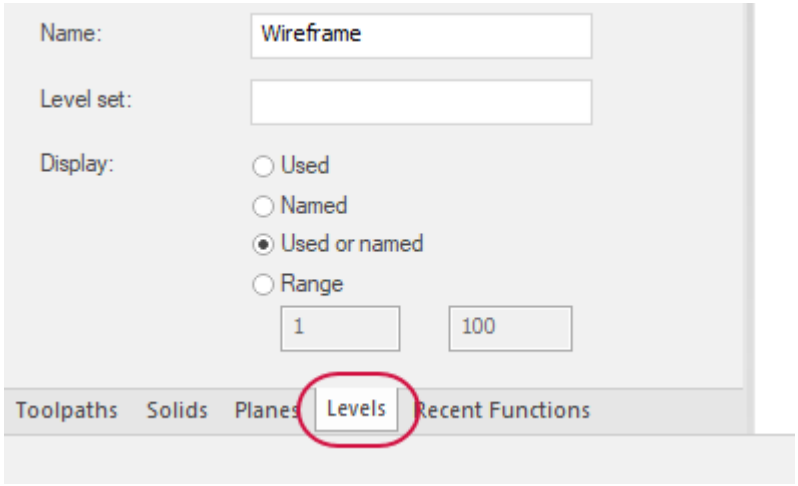
Exercise 1: Exploring the Levels Manager

In this exercise, you explore the Levels Manager and modify some display options.

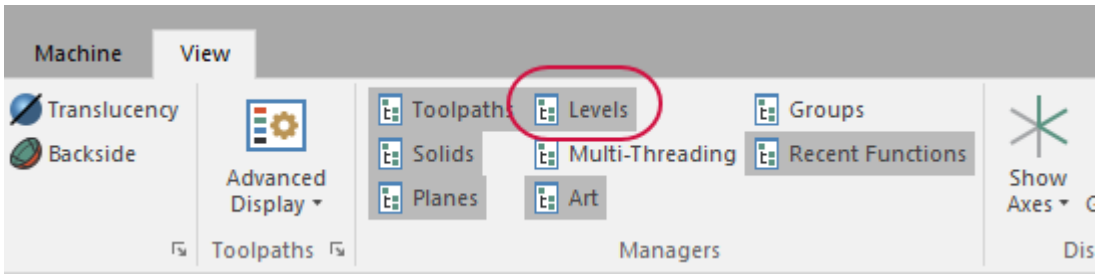
1. Open LEVELS_PART.mcam, which was provided with the tutorial, and save it with a new name. If prompted to save SPOOL BODY.mcam choose **Don't Save**. If you see a message for switching units, click **OK** to allow the change.



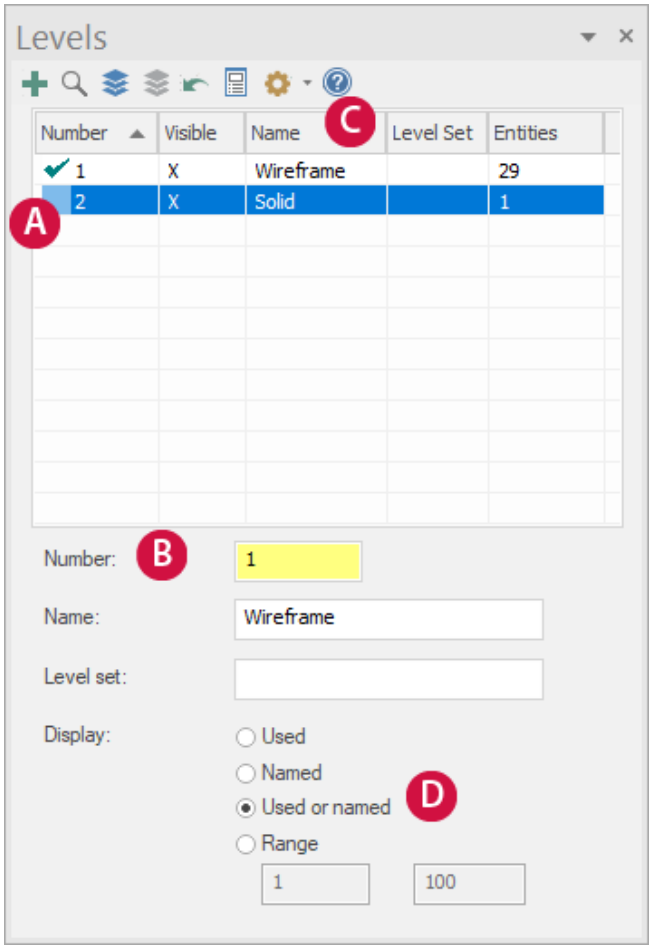
2. Click the **Levels** tab.



If the **Levels** tab is not displayed, select **Levels** on the **View** tab to toggle the display.



The Levels Manager shows all levels that are in the current part file, their display status, and the number of entities per level.



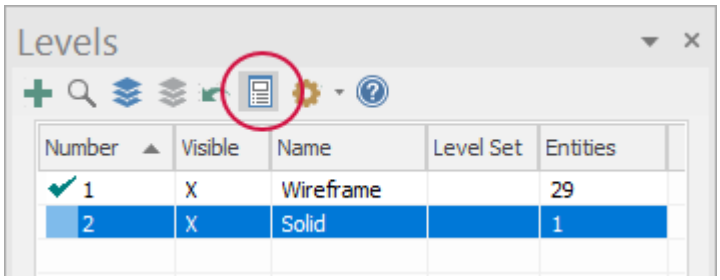
A. The checkmark in the **Number** column indicates the current active level.

B. Settings for the active level display in this section.

C. Click any column heading to sort the levels grid. Click again to change the sort order.

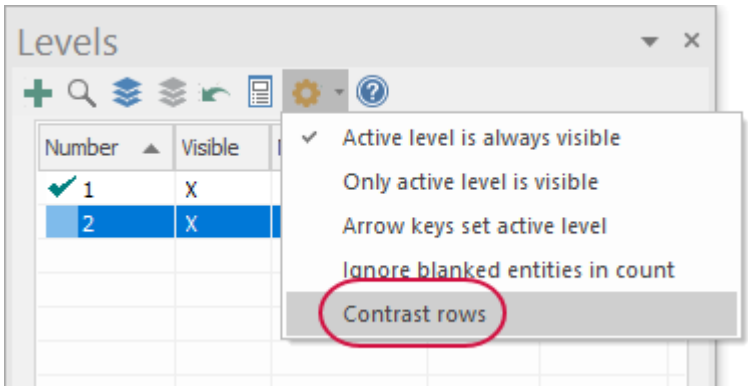
D. Use these options to choose which levels are displayed in the grid.

3. Select **Hide level properties** to hide the property controls in the lower section of the Levels Manager and maximize the space allotted to the levels grid.



4. Select **Hide level properties** again to show the controls at the bottom of the Levels Manager.

5. Select **Display options**, **Contrast rows** to apply shading to every other row used in the grid.



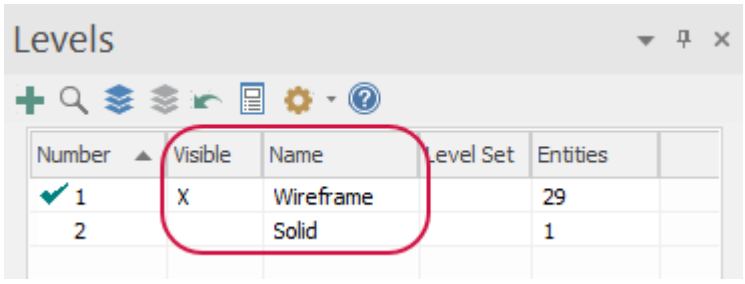
6. Select **Display options** and deselect **Contrast rows** to remove shading from the levels grid.

In the next exercise, you learn to toggle the display of different levels.

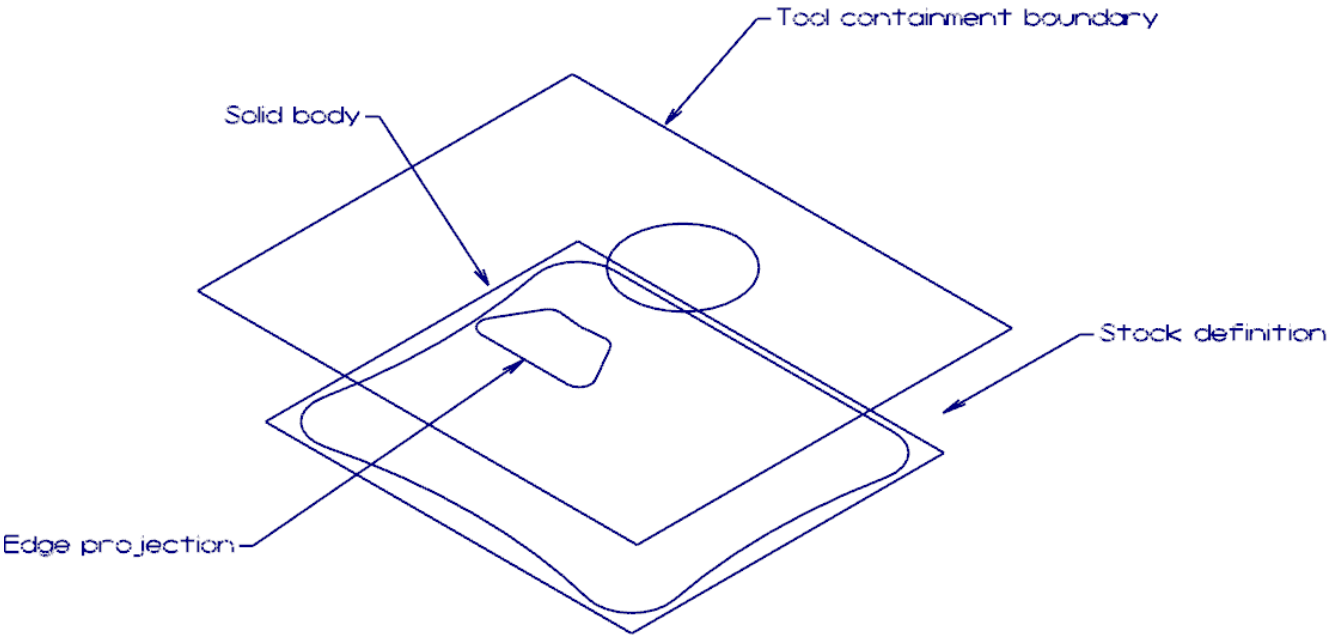
Exercise 2: Showing or Hiding Levels

In this exercise, you change the graphics window display by hiding and showing the levels. To select or otherwise work with an entity in the graphics window, the level it occupies must be visible.

1. In the **Visible** column for the **2: Solid** level, click once to hide the level.



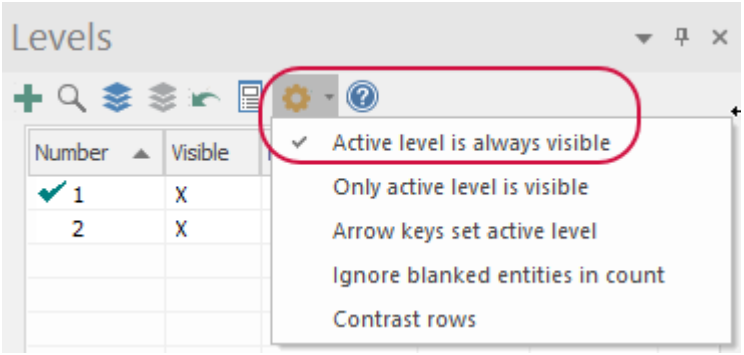
The part display in the graphics window updates accordingly.



2. Click again in the **Visible** column to display the solid.

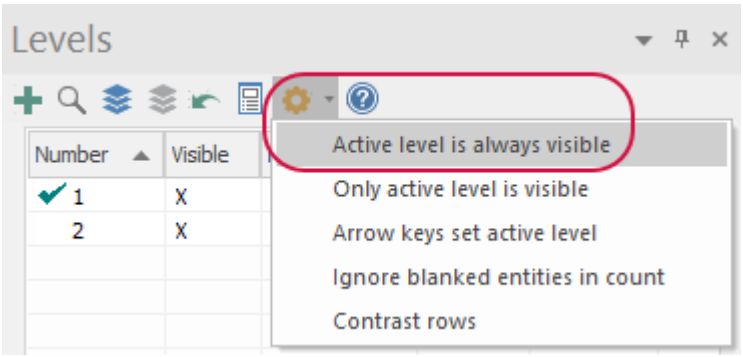
3. Try turning off the display of level **1: Wireframe**.

   When **Active level is always visible** is selected in **Display options**, you cannot hide the active level.

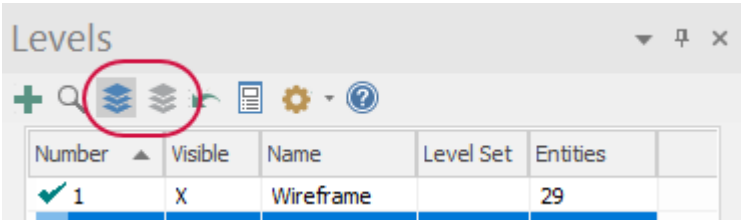


   Turning off the display of the active level while you are working in that level is not recommended. However, it may occasionally be necessary.

4. Select **Display options**, and deselect **Active level is always visible**.



5. Use **Turn all levels off** and **Turn all levels on** to toggle the display of all levels.

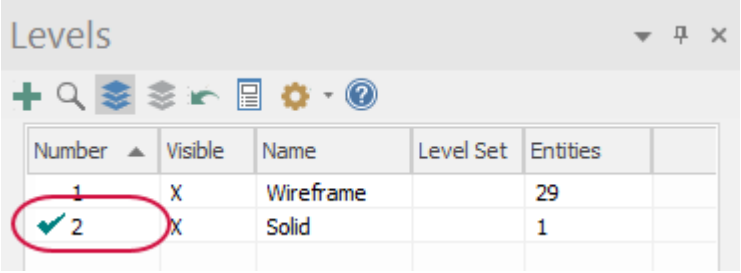


6. Select **Active level is always visible** again.
7. Make sure all levels are visible before continuing on to the next exercise.

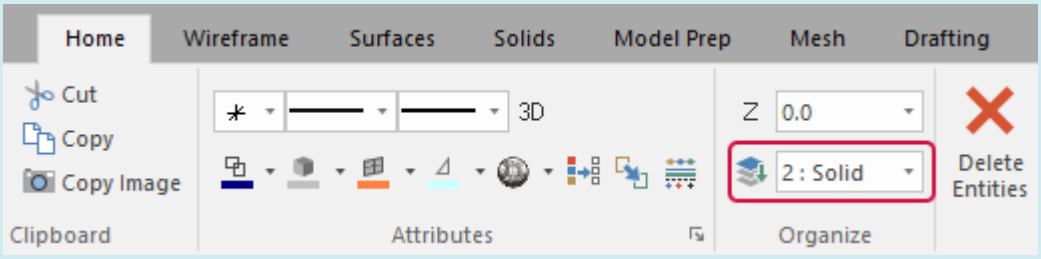
Exercise 3: Changing the Active Level

Any geometry that you create is always placed on the active level. There can only be one active level at a time, but you can change the active level as often as necessary to work with the part. In this exercise, you change the active level using several methods.

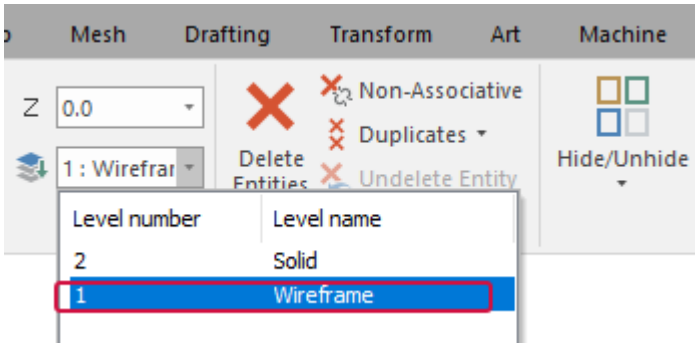
1. In the **Levels Manager**, click once in the **Number** column of level **2: Solid** to set it as the active level. A checkmark indicates that it is now the active level.



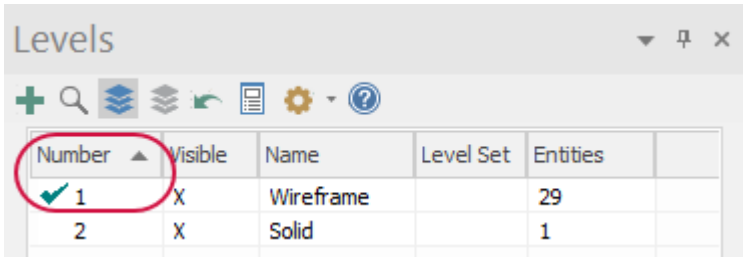
NOTE
The level field located on the **Home** tab also shows that level **2: Solid** is the active level.



2. On the **Home** tab, select the **Level** drop-down and select **1 : Wireframe** to make it the active level.

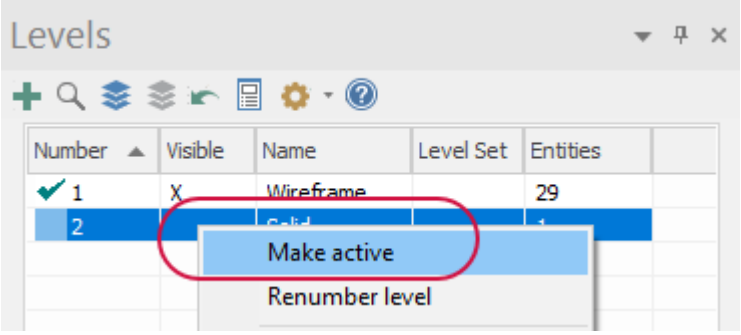


In the Levels Manager, a checkmark appears in the **Number** column for level **1**.



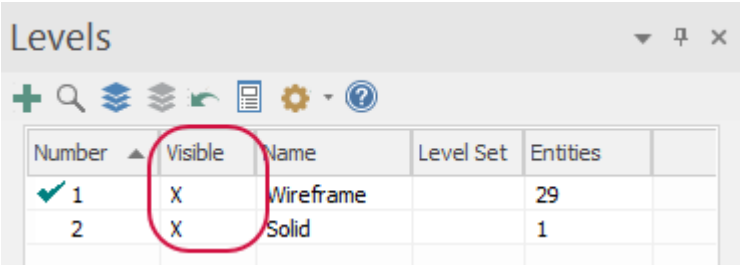
3. In the Levels Manager, turn off the visibility of level **2: Solid**.

4. Position the cursor anywhere in the row for level **2: Solid**, right-click and choose **Make active**.



A checkmark appears in the **Number** column for level **2**.

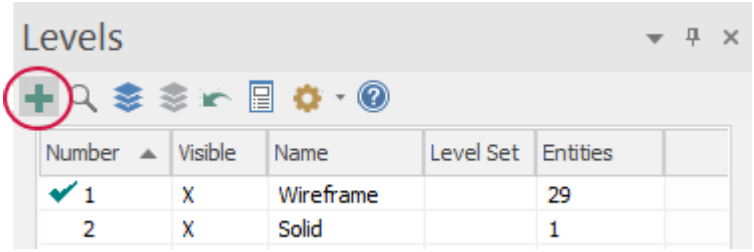
Level **2: Solid** is also visible because the Levels Manager display option **Active level is always visible** is selected.



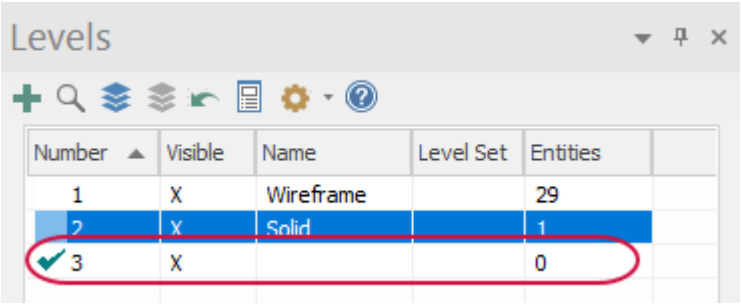
Exercise 4: Creating a Level and Assigning Entities

In this exercise, you create and name a level, and then assign entities to that level.

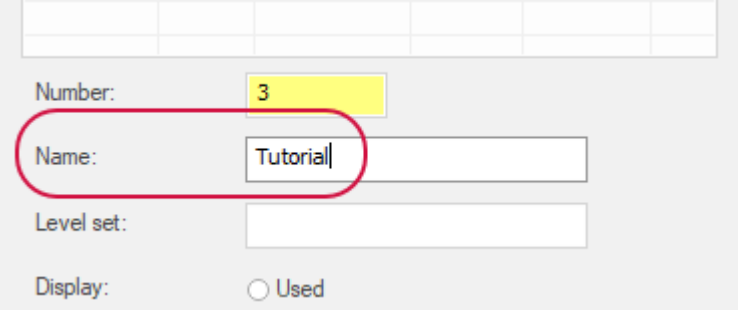
1. In the Levels Manager, select **Add a new level**.



Mastercam adds a new level to the Levels Manager grid and sets it as the active level.

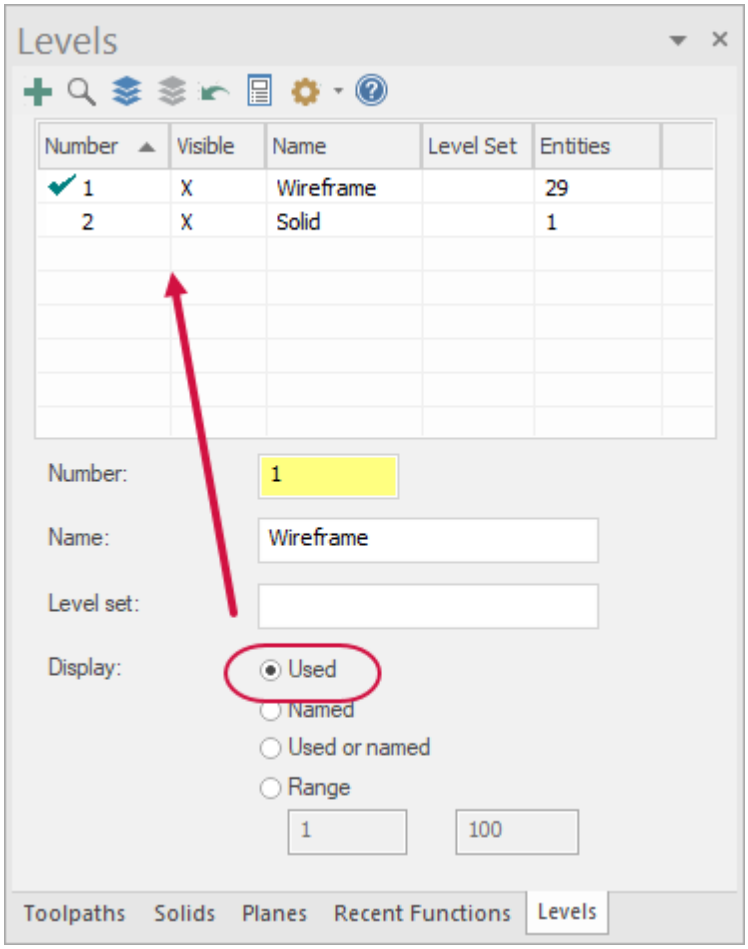


2. Enter **Tutorial** for the **Name**.



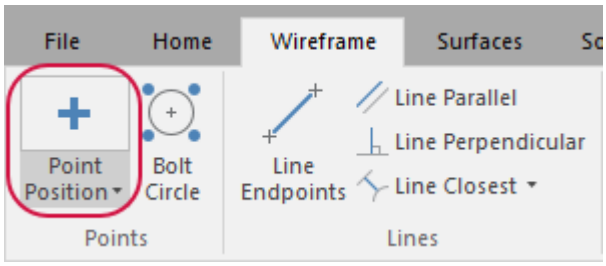
The Levels Manager grid updates to show the new name. Although level names are optional, descriptive names are helpful.

3. Make level **1: Wireframe** the active level.
4. Use the **Display** options in the lower section of the **Levels Manager** to control the visibility of levels.
    - Click **Used** and notice that level **3: Tutorial** is no longer visible because there are no entities assigned to it.



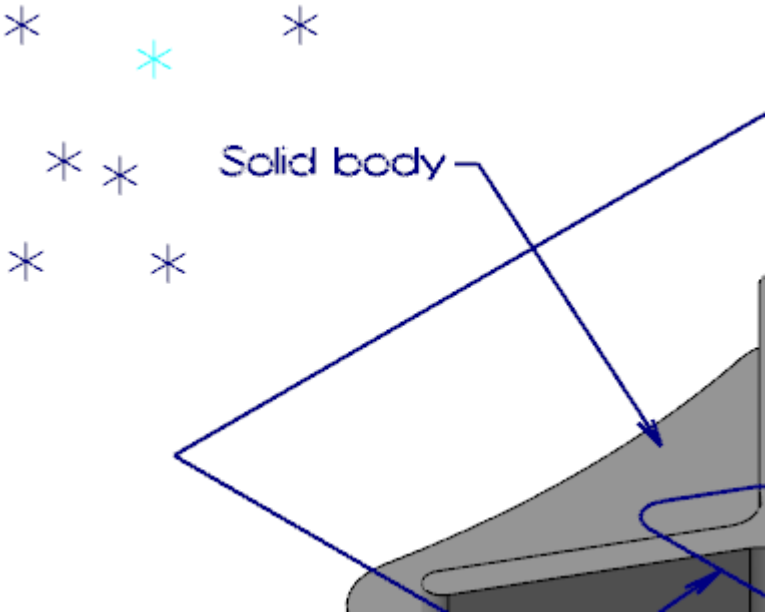
    - Click **Used or named** to restore visibility of all levels.
5. Make level **3: Tutorial** the active level.

6. On the **Wireframe** tab, select **Point Position**.

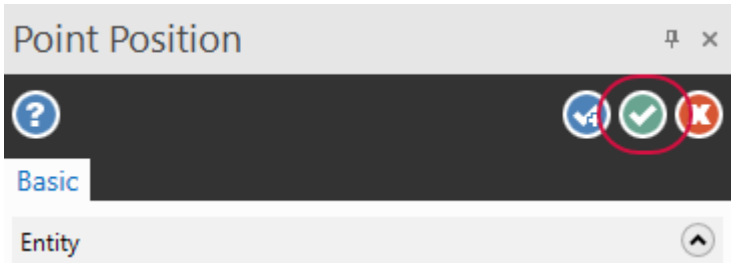


The **Point Position** function panel opens.

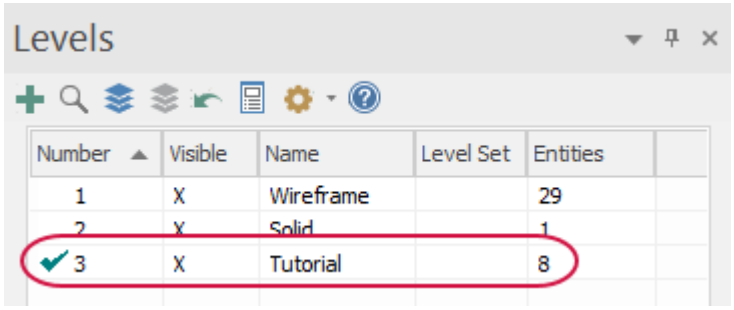
7. Click a few times in different places in the graphics window to create points.



8. Click **OK** in the **Point Position** function panel to exit the function.



9. In the Levels Manager, verify that the new geometry was created on level **3: Tutorial**. The number of entities is not **0**.



Your number of points may be different than what is shown.

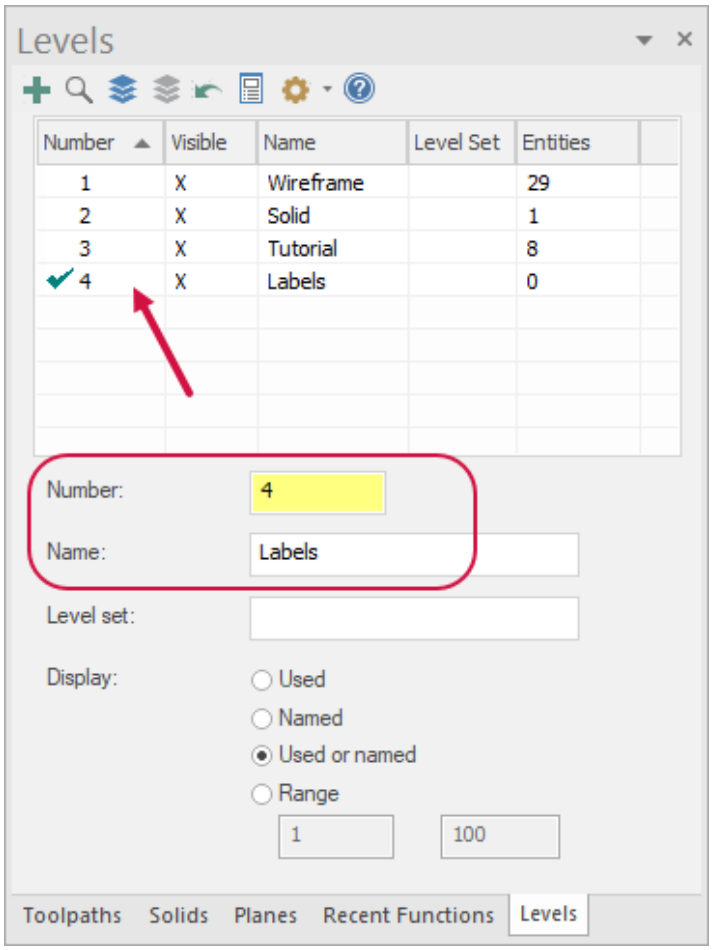
10. Practice turning level displays on and off and changing the main level.

    When you are ready, turn on the visibility for all levels and continue with the next exercise.

Exercise 5: Moving Entities to a Different Level

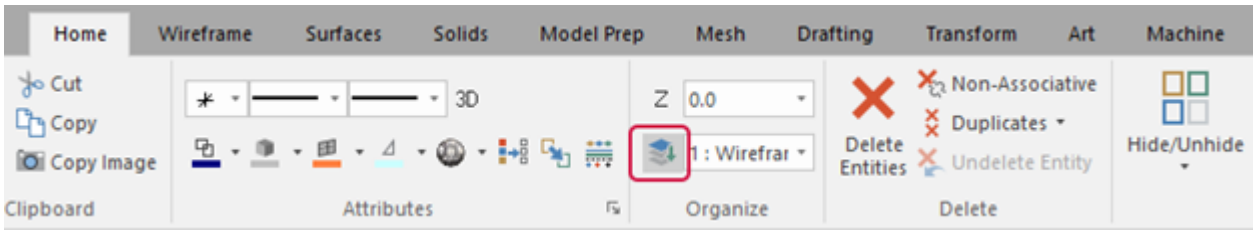
In this exercise, you create a new level by moving entities from one level to another. You can use the same basic procedure to copy entities to a different level.

1. In the Levels Manager, enter **4** in the **Number** field, and then enter **Labels** for the **Name**.



    The new level, **4: Labels** becomes the active level.

2. Make level **1: Wireframe** the active level.
3. On the **Home** tab, select **Change Level**.

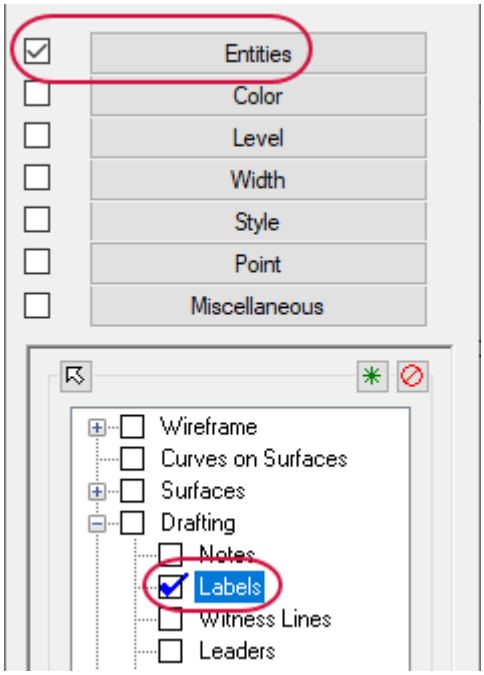


    Mastercam prompts you to select which entities to move.

4. In the graphics window, choose the **Select all advanced Quick Mask**.

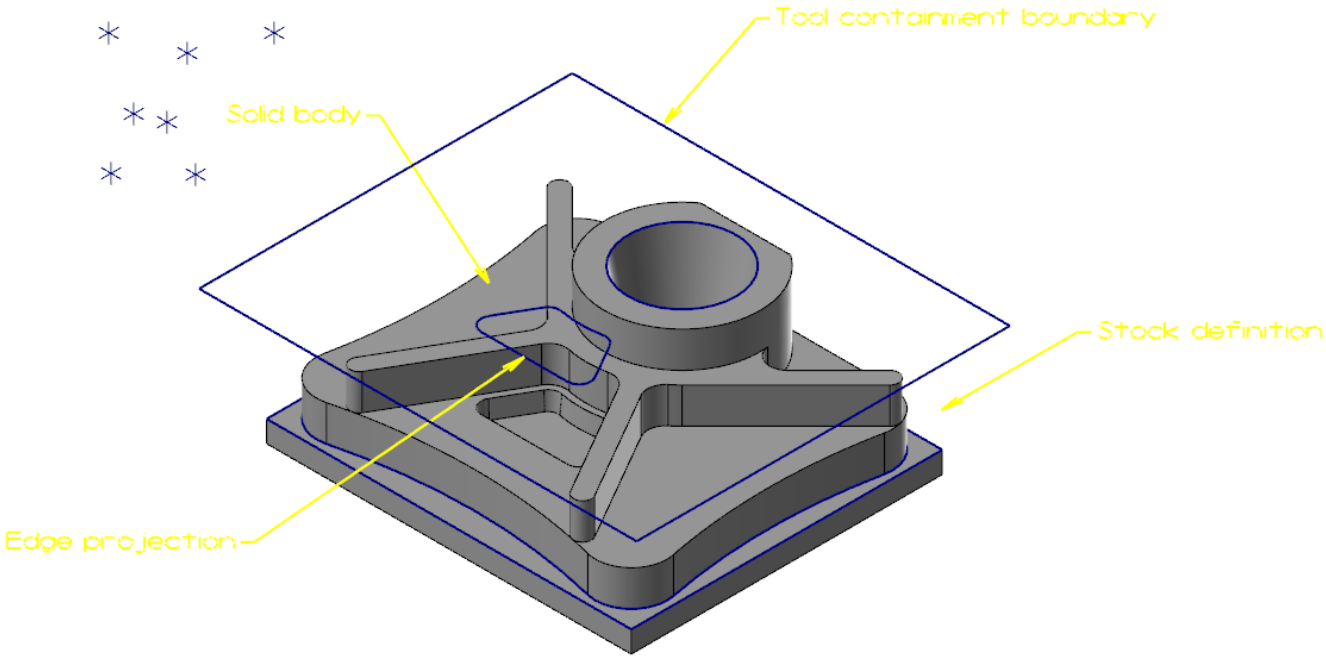
The **Select All** dialog box displays.

5. In the **Select All** dialog box, do the following:
    a. Select the **Entities** checkbox.
    b. Expand **Drafting** and select **Labels**.



6. Click **OK**.

   In the graphics window, Mastercam highlights all of the labels, indicating that they are selected.



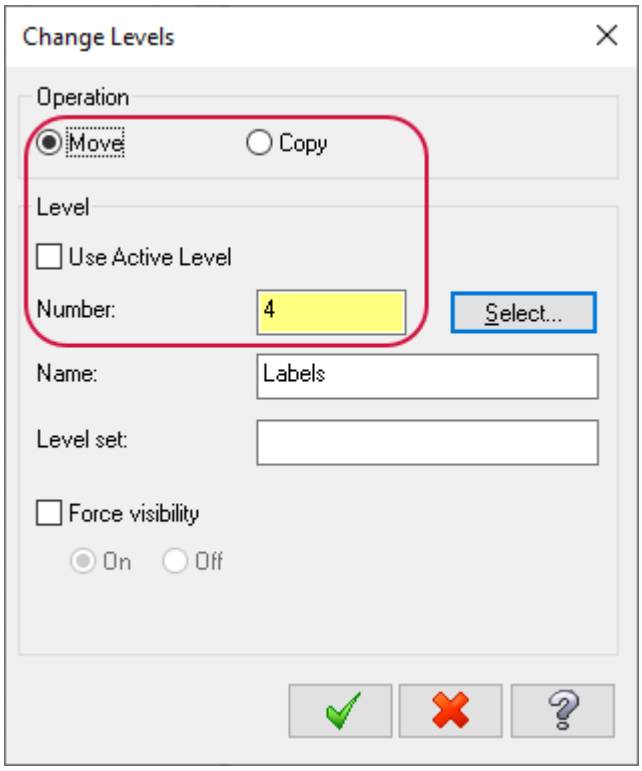
7. Press [**Enter**] or choose **End Selection**.

   The **Change Levels** dialog box opens.

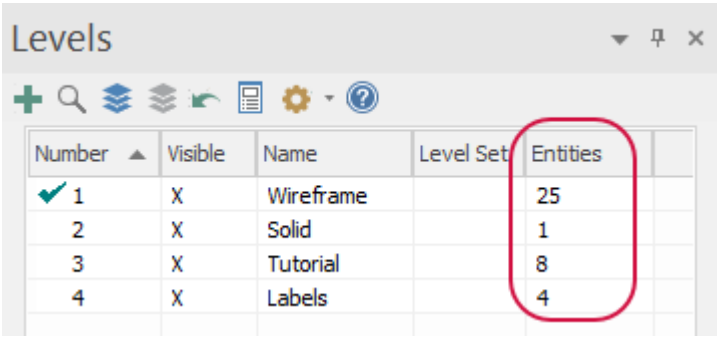
> **NOTE**
> You can also use general selection methods to pre-select entities before choosing the **Change Level** option. When you pre-select entities, Mastercam does not prompt you to select additional entities and you advance directly to the **Change Levels** dialog box.

8. In the **Change Levels** dialog box, do the following:
   a. Select **Move**.
   b. Deselect **Use Active Level** to allow the entities to be moved to any level.

c. Enter **4** for **Number**.



9. Press [**Enter**].
10. In the Levels Manager, verify that level **4: Labels** now contains the four label entities that were on level **1**. Level **1: Wireframe** contains four fewer entities.



11. Save your part.

You have now completed this chapter of the *Mastercam Basics Tutorial*. In the next chapter, you learn more about views and planes.

CHAPTER 7

UNDERSTANDING VIEWS AND PLANES

Mastercam locates 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 has a positive and negative direction.

In previous chapters, you learned about standard views and planes. In this chapter, you learn more about creating your own planes and using them for geometry creation.

Goals

- Understand views and planes.
- Customize the Planes Manager.
- Adjust follow rules.
- Create a new plane using the dynamic gnomon.
- Create a new plane using the Planes Manager.
- Use a new plane for geometry creation.

Views and Planes

You use views and planes in Mastercam for many drawing and machining purposes.

A **plane** is any two-dimensional slice of a 3D coordinate system. Examples of planes include the XY plane in a 3D coordinate system, or a face of your part, no matter its angle.

A **view** is a plane with an origin and orientation. For example, the Left side view and Right side view are both aligned with the same plane; but a clockwise arc in one view is a counterclockwise arc in the other view.

Mastercam includes a number of pre-defined views, which you explored in " Changing Standard Views" on page 87. These include Top, Front, Right, Isometric, and others.

The work coordinate system (WCS) refers to the alignment of the coordinate system itself. You can choose to align the coordinate system axes with any plane you wish. When this happens, you map the XY plane to the plane of the view. The origin of the view becomes your new (0,0,0) point. The view becomes the Top plane in the new working coordinate system.

This section introduces some ways planes are used and some ways to access them. You can use views and planes for three main functions:

- **Graphic views (Gviews)**— A Gview determines the angle from which you are looking at the part in the graphics window.
- **Construction planes (Cplanes)**—When you draw geometry, the Cplane is the plane in which the geometry is created. This does not have to be the same as the Gview. For example, you can look at your part in Isometric Gview, while drawing geometry in the Front Cplane.

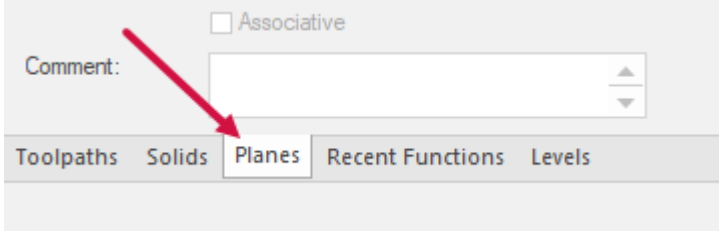
- **Tool planes (Tplanes)**— This is the plane that is typically normal to the tool axis. Tplanes are only used when creating toolpaths. Tool planes determine the tool orientation. In almost all cases, your Cplane will be the same as your Tplane (the most common exceptions are mill/turn operations).

You use the Planes Manager to select, edit, create, and manage planes.

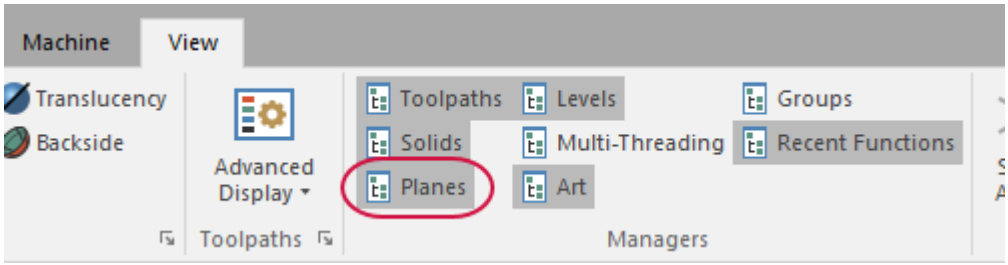
Exercise 1: Exploring the Planes Manager

In this exercise, you open the Planes Manager and experiment with display options.

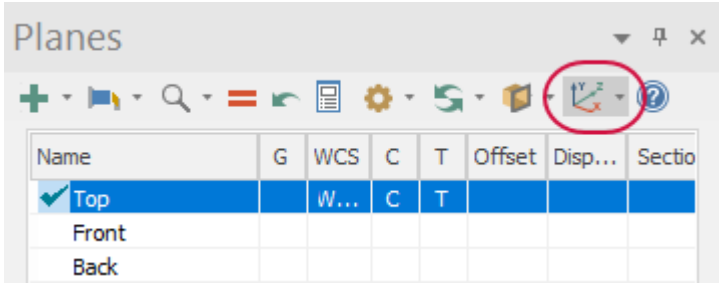
1. Open BRACE.mcam, which was provided with the tutorial. Save the file with a new name.
2. Select the **Planes** tab to bring the Planes Manager to the front.



If the **Planes** tab is not displayed, select **Planes** on the **View** tab to toggle the display.

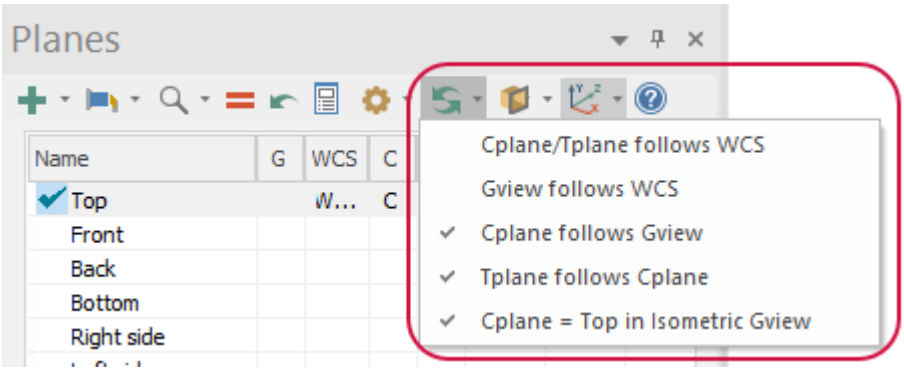


3. In the Planes Manager, select **Show Gnomons**.

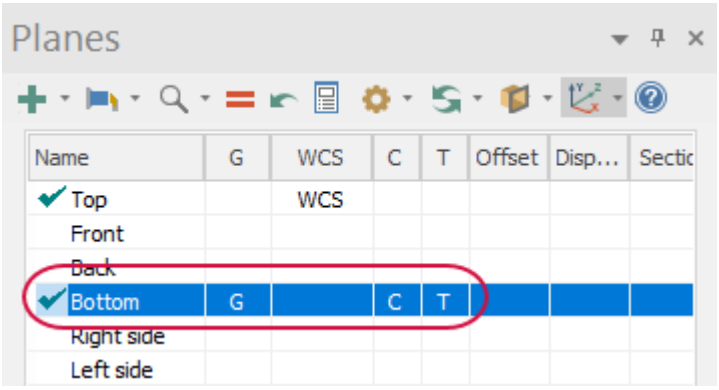


The gnomon for the currently selected plane displays in the graphics window whether or not the Planes Manager has focus. Additionally, if the **Display** column is selected for a particular plane, the gnomon will also display.

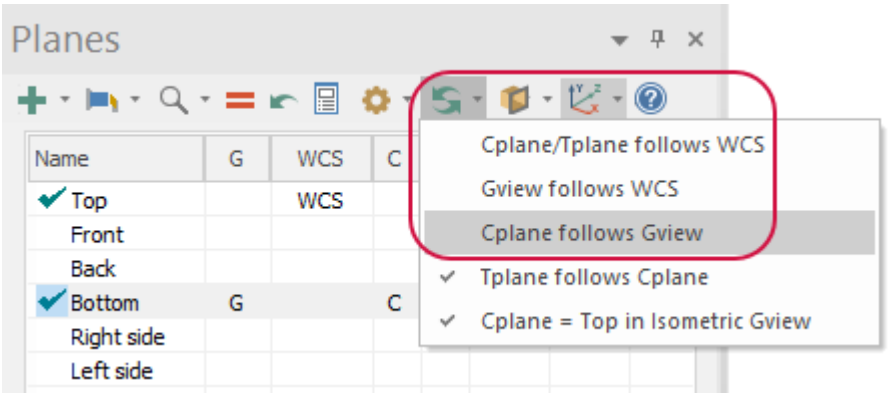
4. Click the **Follow rules** drop-down and select the options shown in the following image. With these settings, when you change the Gview of a plane, the Cplane and Tplane follows.



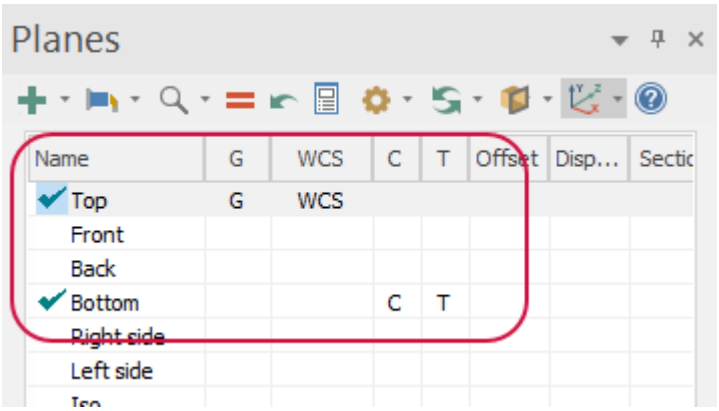
5. Click in the **G** column of the **Bottom** plane to set the Gview to bottom. Note that Cplane and Tplane follow the Gview as directed by the follow rules.



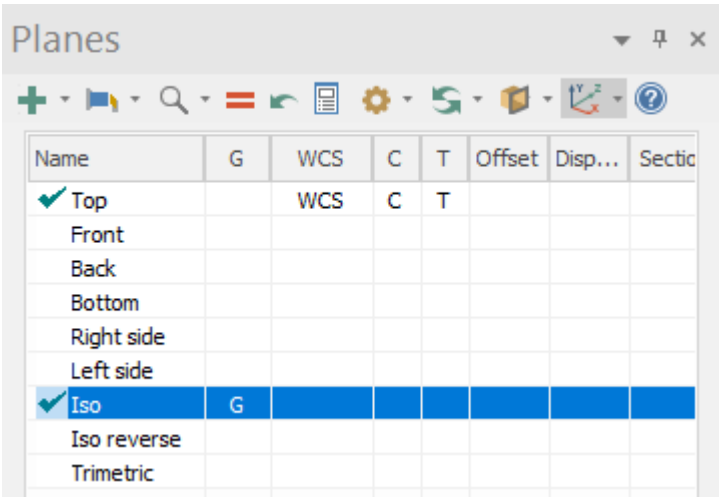
6. From the **Follow rules** drop-down, deselect **Cplane follows Gview**.



7. Click in the **G** column of the **Top** plane to set the Gview to Top. Note that Cplane and Tplane no longer follow the Gview, as directed by the new follow rules.



8. Click the **Follow rules** drop-down, and reselect **Cplane follows Gview**.
9. Click in the **G** column of the **Iso** plane to return the Gview to **Isometric**. Note that Cplane and Tplane remain in the Top plane as the follow rule, **Cplane = Top in Isometric Gview** is selected.

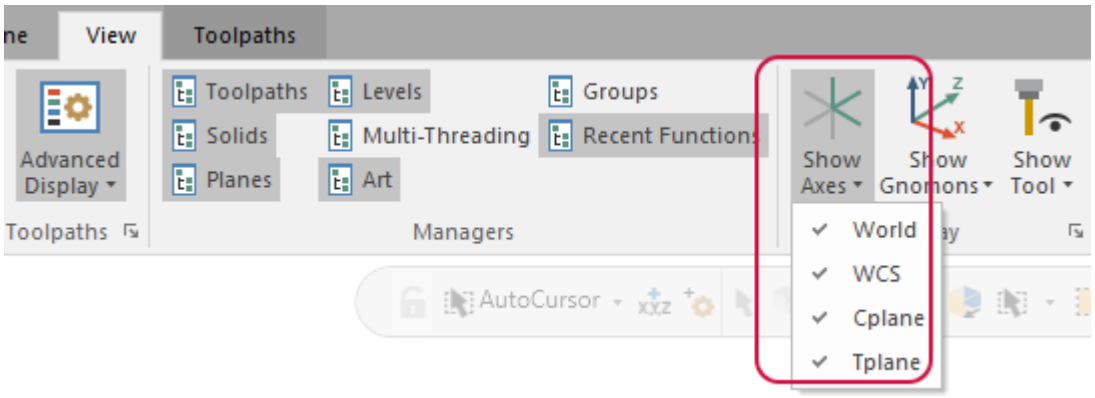


10. Select **Show Gnomons** to toggle it off.

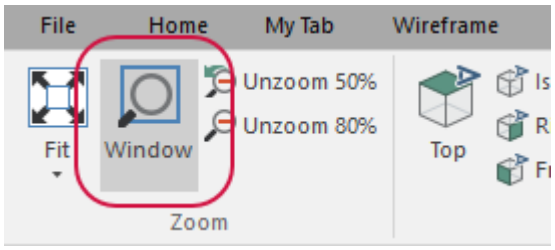
Exercise 2: Creating a New Plane Using the Dynamic Gnomon

In this exercise, you create a new plane using the dynamic gnomon. When creating a new plane using this method, the gnomon is initially oriented to the current Gview.

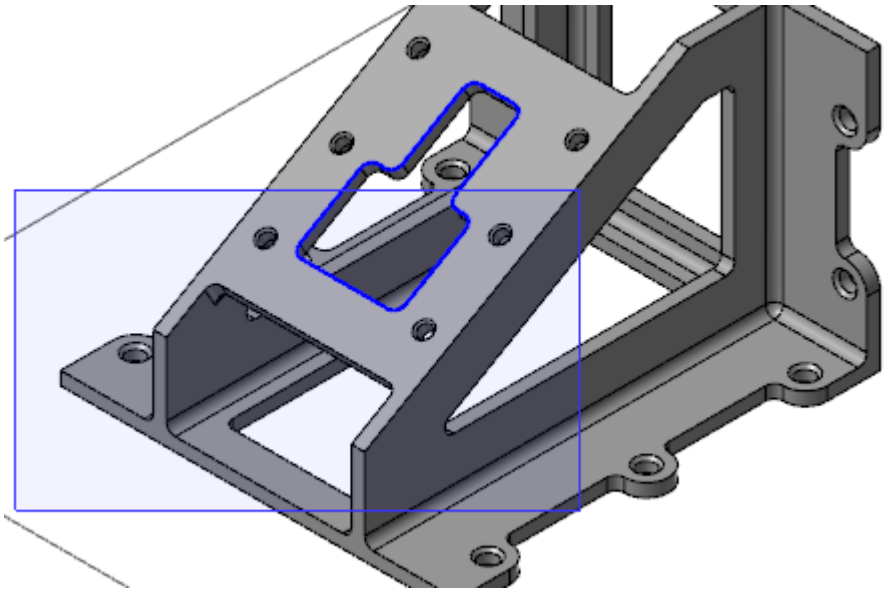
1. On the **View** tab, select the **Show Axes** drop-down and select **WCS**, **Cplane** and **Tplane**.



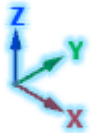
2. Toggle **Show Axes** on, if necessary.
3. On the **View** tab, select **Window**.



4. Draw a box around the area of the part shown below and click to zoom into the window.

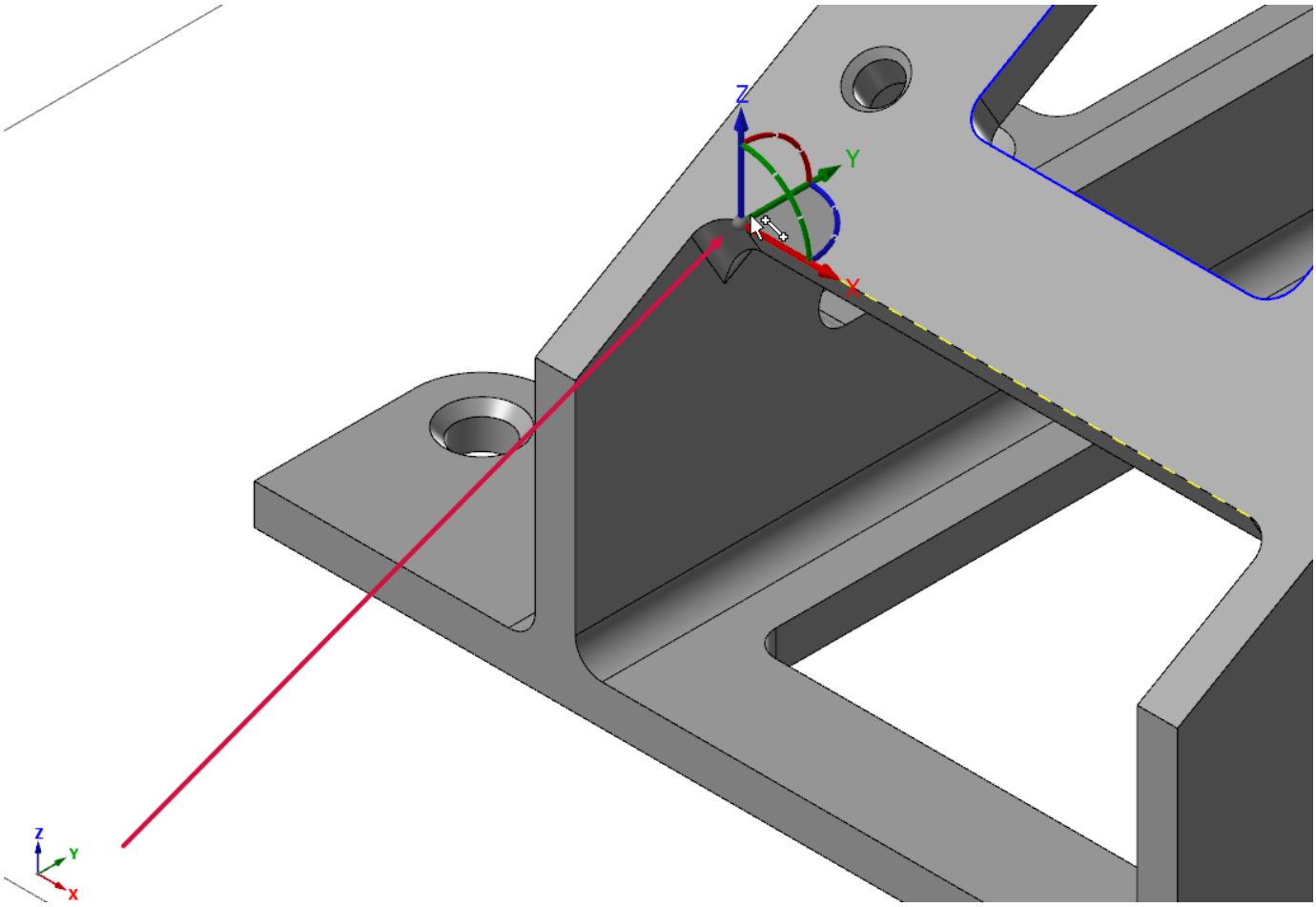


5. In the lower left corner of the graphics window, hover over the Dynamic Gnomon.



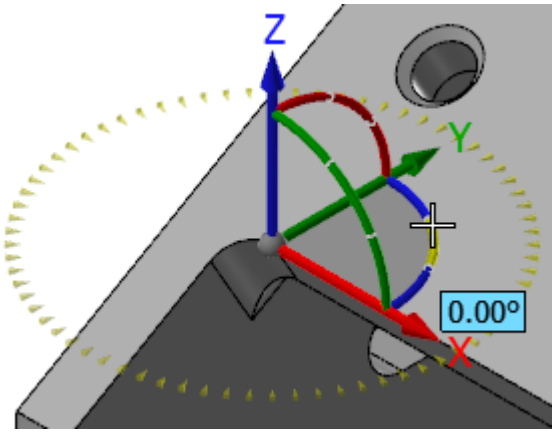
Iso

6. Click and drag the gnomon so that it matches the orientation shown below. Click to set the position.

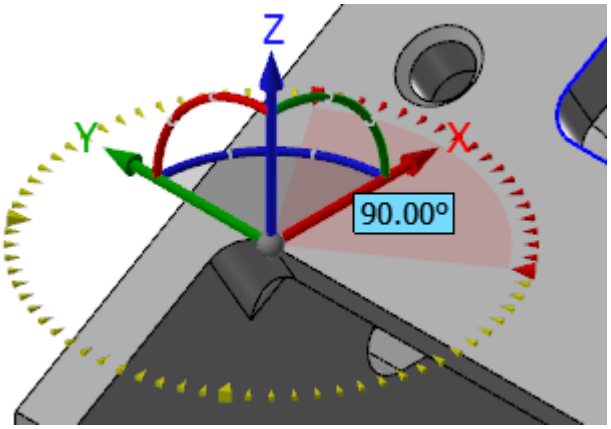


The **New Plane** function panel opens.

7. To change the orientation of the Dynamic Gnomon, hover over the blue arc until the dial appears.

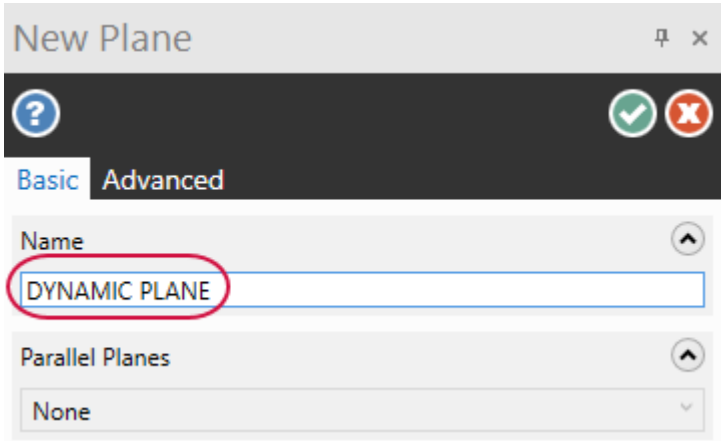


8. Click the blue arc and rotate the gnomon from within the dial until it is at **90** degrees. Click to set the location. (Rotating within the compass snaps in angular increments.)

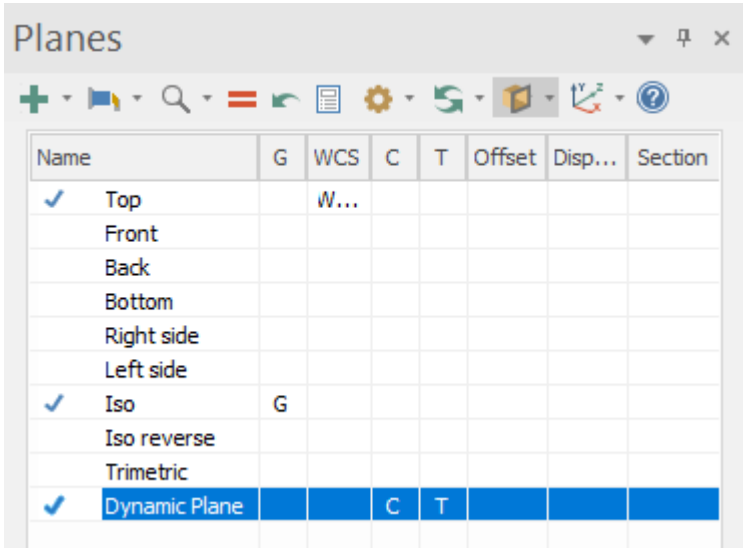


Or, you can enter **90** in the on-screen edit field, and press [**Enter**] twice.

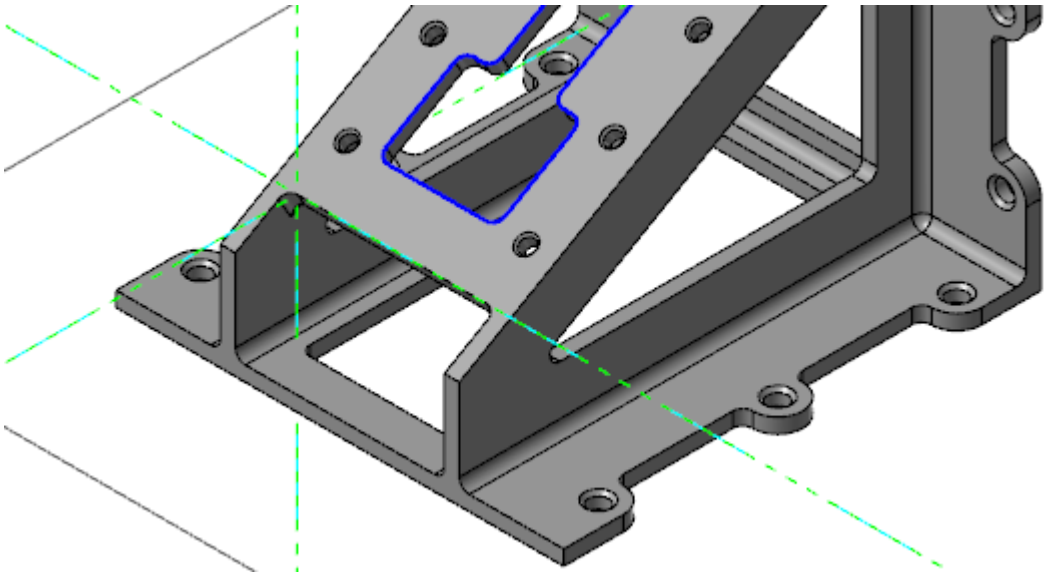
9. In the **New Plane** function panel, enter **DYNAMIC PLANE** for the **Name**.



10. Click **OK** to close the **New Plane** function panel.
11. **DYNAMIC PLANE** appears in the Planes Manager. Click in the CPlane column of **DYNAMIC PLANE**.



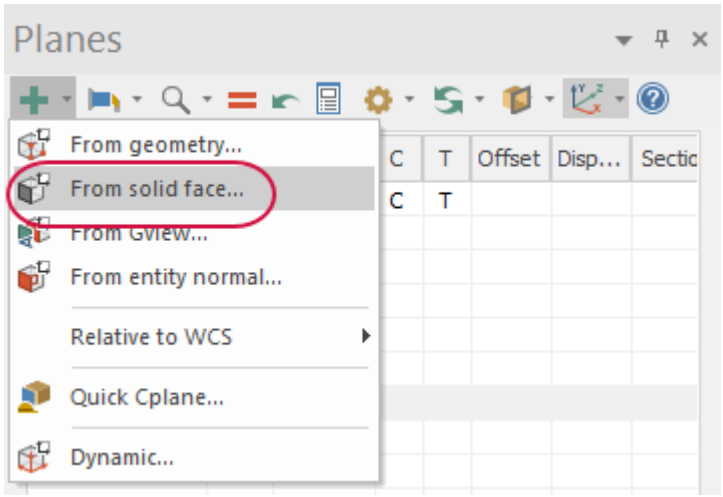
The Cplane and Tplane axes indicate the location of the new plane origin as shown below.



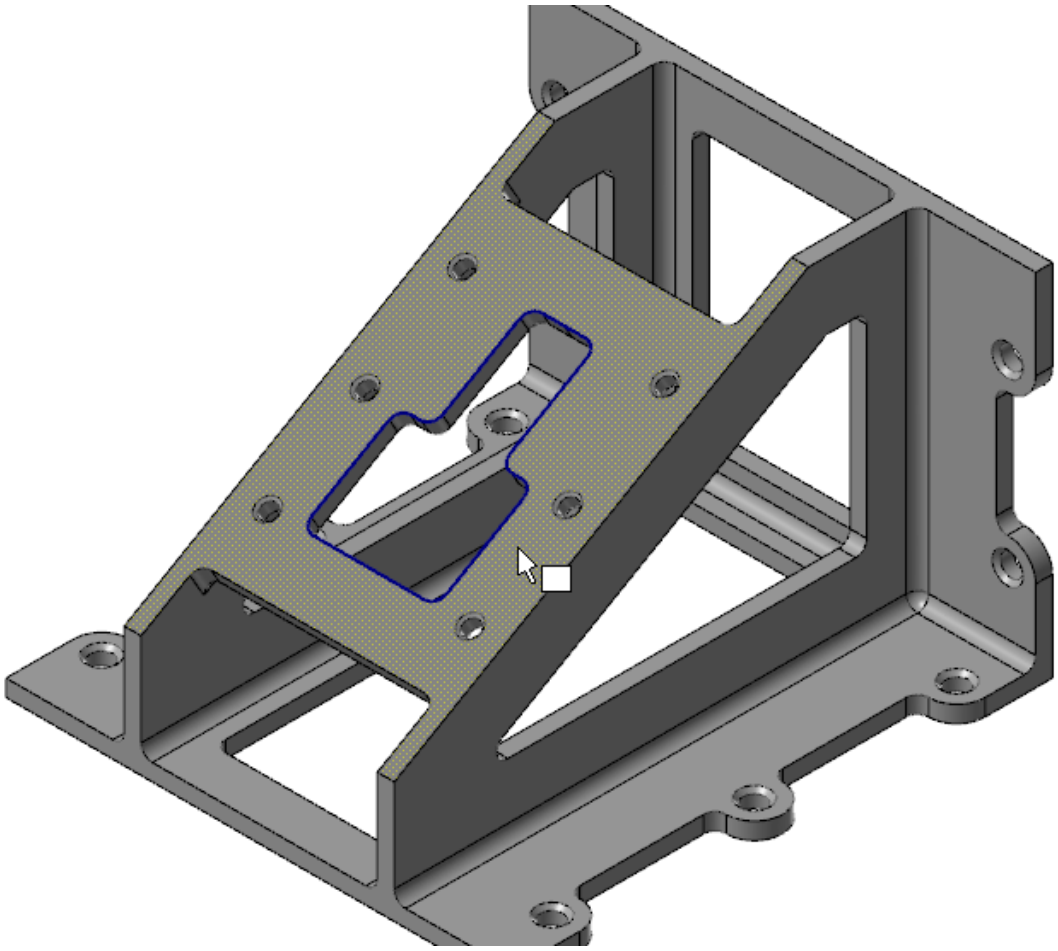
Exercise 3: Creating a New Plane Using Planes Manager

In this exercise, you use Planes Manager to create a new plane aligned with the slanted face of the part. You add it to the list of system planes.

1. Press [**F9**] to turn off the axes.
2. Press [**Alt + F1**] to fit your part to the graphics window.
3. In the Planes Manager, select **Create a new plane**, **From solid face**.

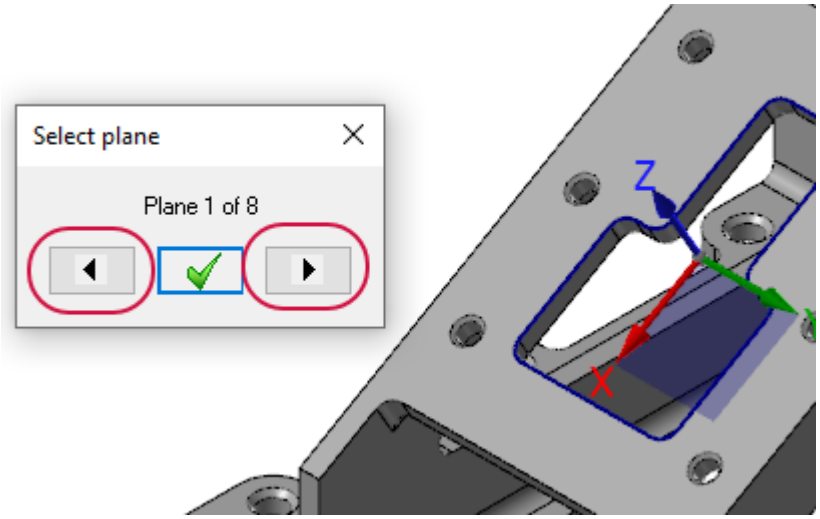


4. Select the slanted face of the part, shown below:

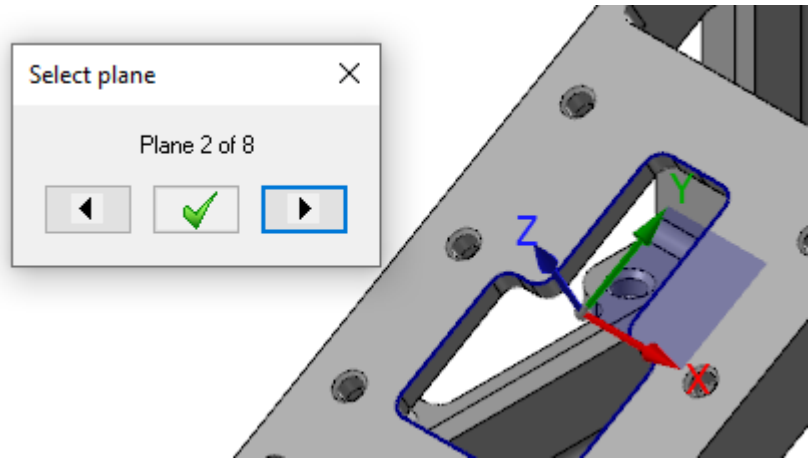


The **Select plane** dialog box opens.

5. In the **Select plane** dialog box, click the arrows to cycle through the possible planes. For most applications, it make sense to have +Z pointing away from the part.



6. Cycle to the plane shown below.



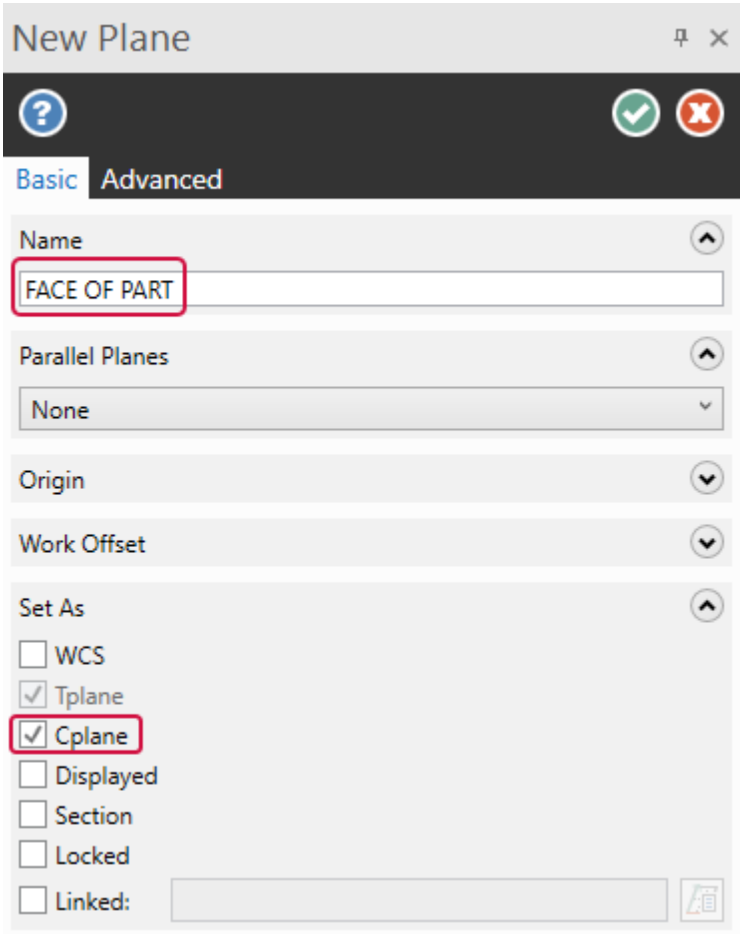
7. Click **Save this plane and exit** in the **Select plane** dialog box.

   The **New Plane** function panel opens.

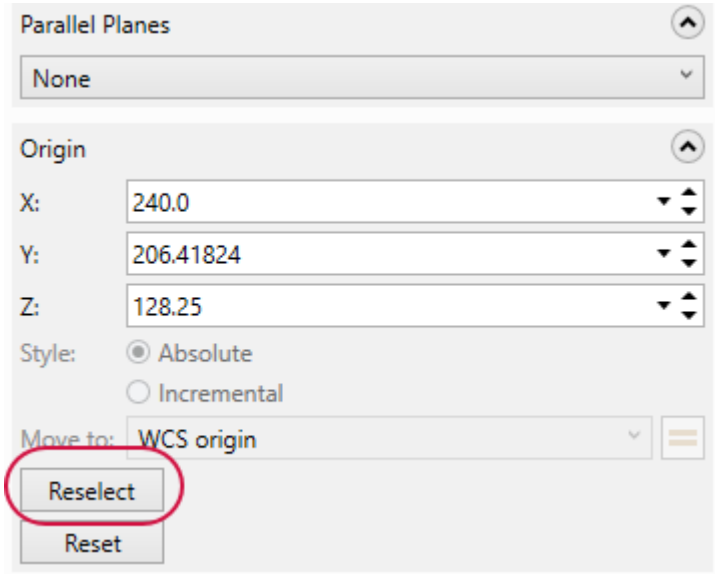
8. In the **New Plane** function panel, enter the following:

   a. Enter **FACE OF PART** for **Name**.

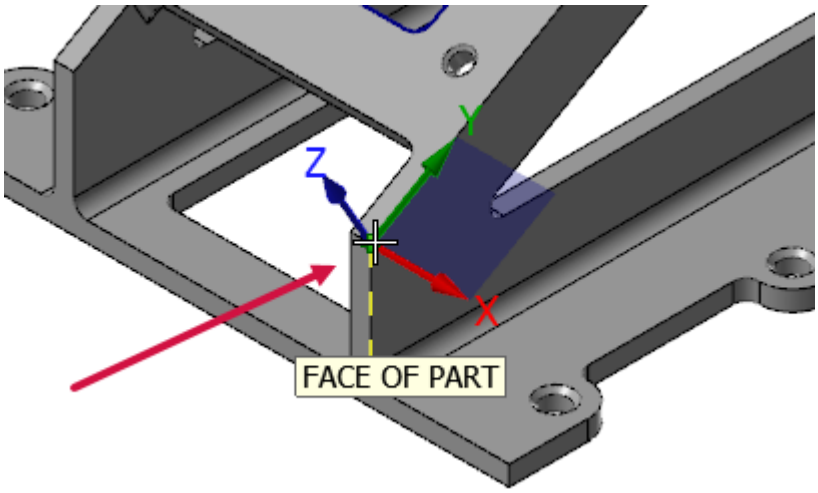
   b. Select **Cplane** for **Set As**.



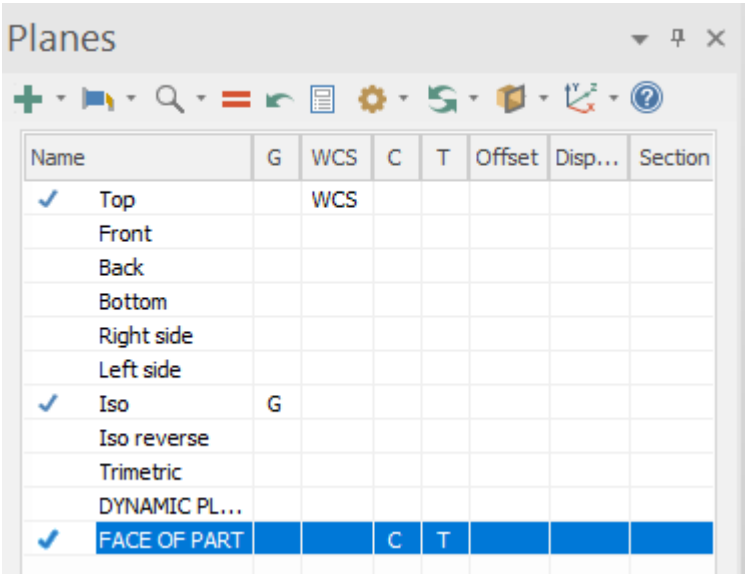
9. Under the **Origin** group, click **Reselect** to return to the graphics window to choose a new point for the origin.



10. Select the point at the lower-right corner of the face.



11. Click **OK** in the **New Plane** function panel to create a plane. Your new plane, **FACE OF PART**, displays in the planes grid.

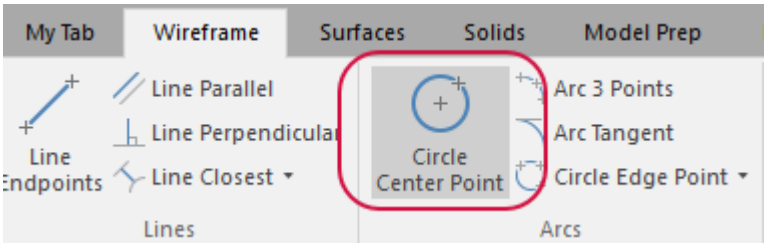


Exercise 4: Using a Custom Plane for Drawing Geometry

In this exercise, you create geometry using the new Cplane. You see that the new plane determines the orientation of the geometry. You then set the Gview to the new plane.

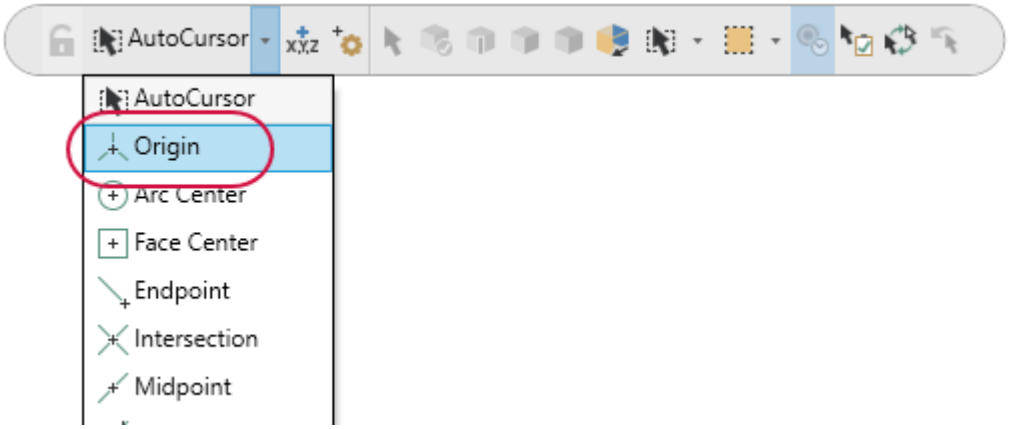
1. In the **Planes Manager**, click in the **WCS** column for the **FACE OF PART** plane.

2. On the **Wireframe** tab, select **Circle Center Point**.

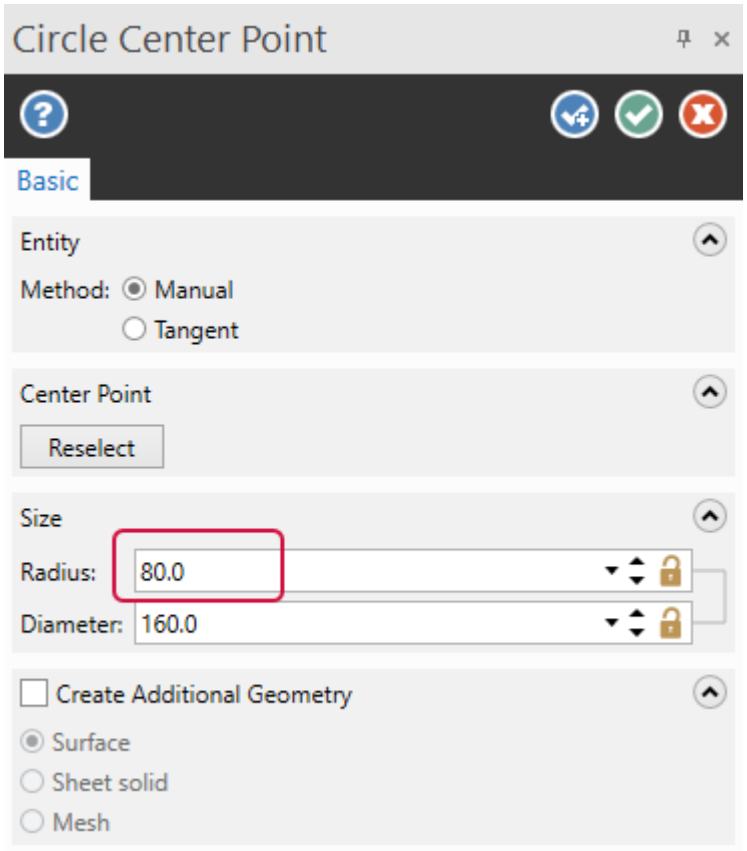


3. The **Circle Center Point** function panel opens.

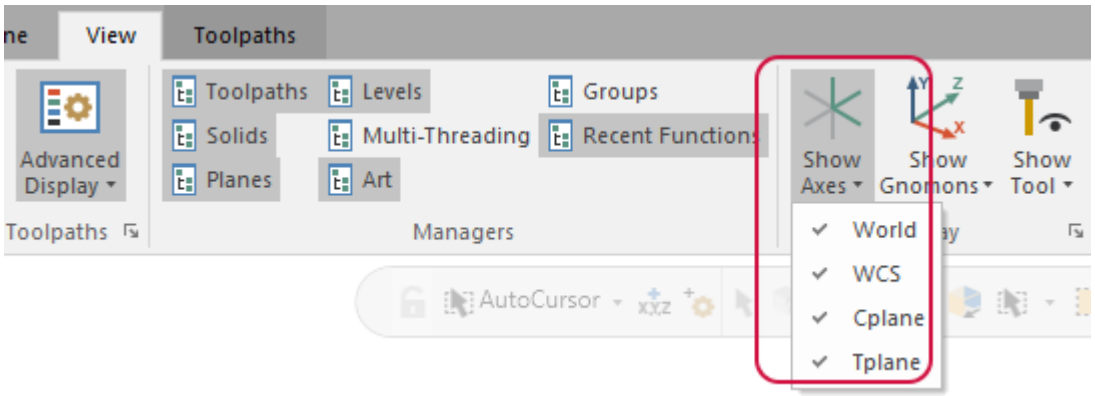
4. On the **Selection Bar**, select **Origin** from the **AutoCursor** drop-down.



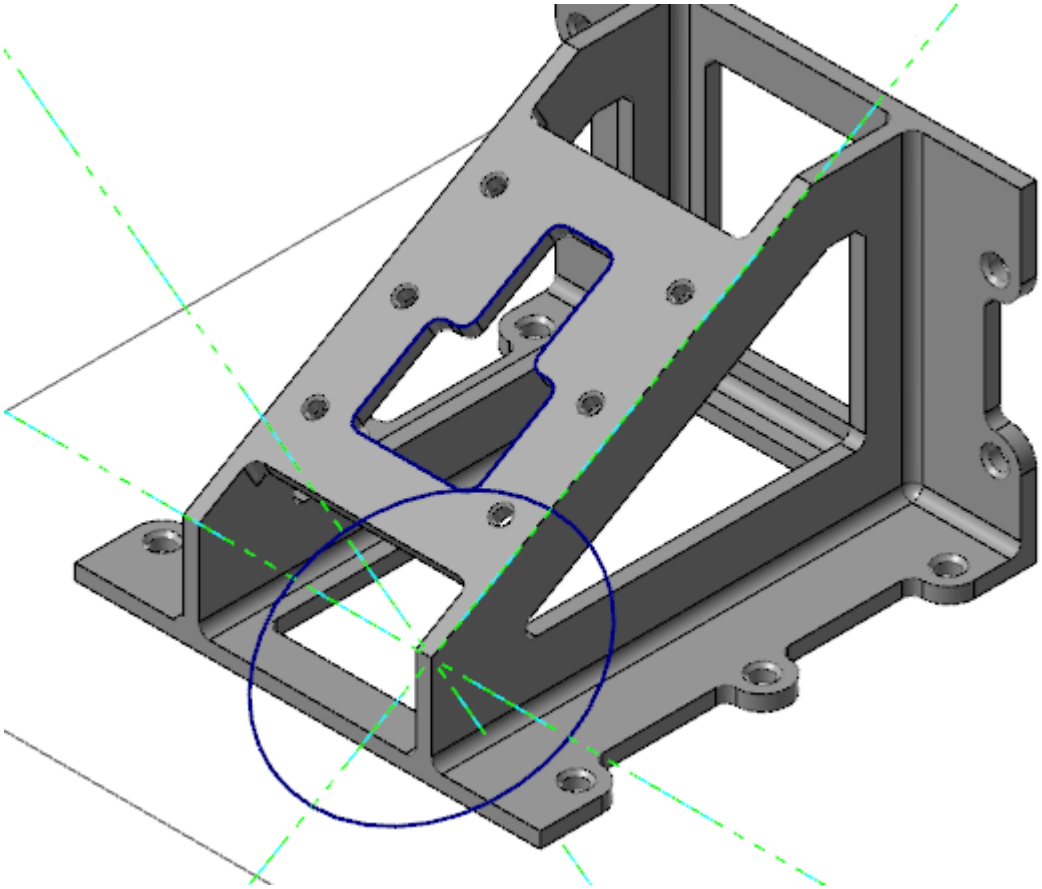
5. In the **Circle Center Point** function panel, enter **80.0** for **Radius** and then click **OK**.



6. On the **View** tab, select **Show Axes**. If necessary, click the drop-down and ensure that all options are selected.

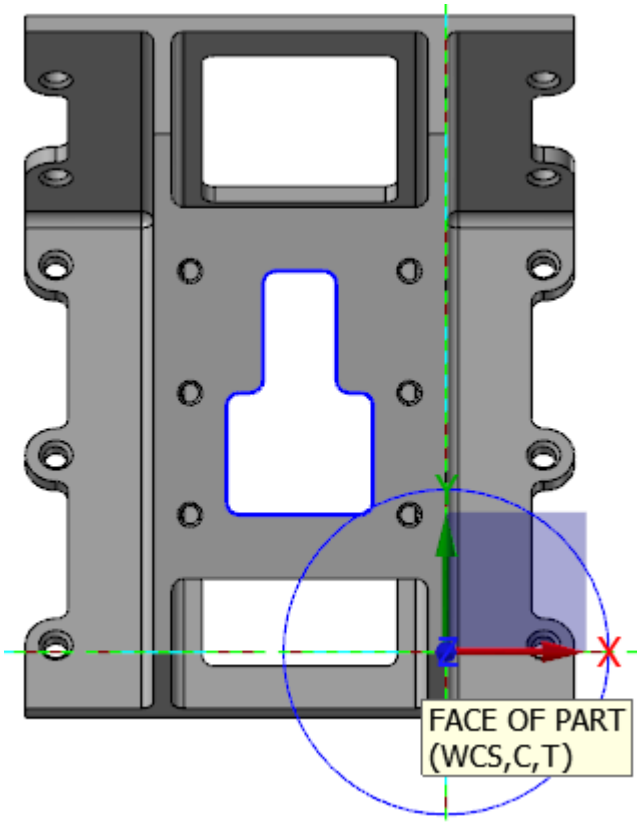


7. The axes that define each view display in a different color. Since Cplane and Tplane are the same, the axes display as dotted lines. Experiment by turning individual sets of axes on and off.



8. In the Planes Manager, click in the **G** column of the **FACE OF PART** plane to set the Gview.

   You are now looking straight down into the face of the part. The Gview is aligned with the custom plane that you just created.



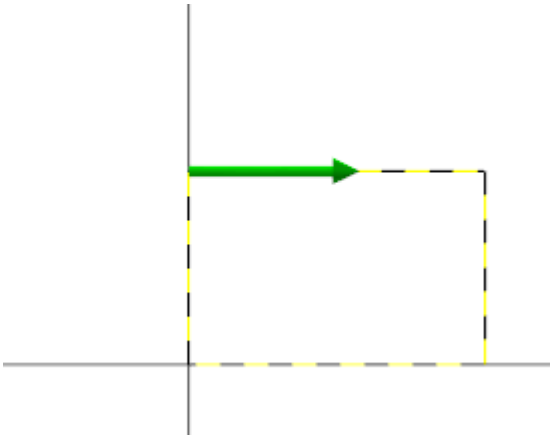
9. Press [**F9**] to turn off the axes.
10. Save your part.

You have now completed this chapter of the *Mastercam Basics Tutorial*. In the next chapter, you learn more about working with chain selection.

CHAPTER 8

WORKING WITH CHAIN SELECTION

Chaining is the process of selecting and linking pieces of geometry. Use chains to create a toolpath, surface, or solid. Chains are a subset of selection. However, chains are directional. Cutting tools follow the direction of the chain. The following image shows a clockwise wireframe chain. The tool will move in the direction of the green arrow.



There are two modes for chaining: wireframe and solid. The mode determines the selection options that are available to you. Wireframe chains are typically used when you wish to create a toolpath without first creating the associated solid. Solid chains are used on solid models.

Goals

- Use curve all edges to create wireframe from a solid.
- Explore wireframe chaining.
- Create and modify solid chains.

NOTE
This chapter requires a Mill machine. If your Mastercam license does not allow access to Mill, you can either:

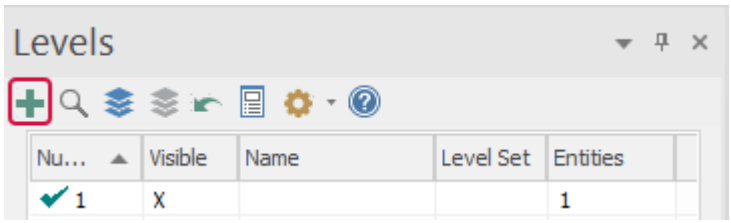
- Download Mastercam Learning Edition and use it to complete the exercise: https://signup.mastercam.com/demo-hle.
- Read through the exercise.

Exercise 1: Creating Wireframe from a Solid

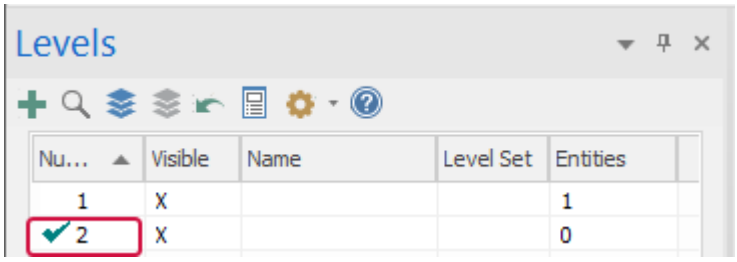
In this exercise, you use **Curve All Edges** to create wireframe for a solid.

1. Open the file SOLID CHAINS.mcam, which is included with this tutorial, and use **Save As** to save it with a new name. If you see a message for switching units, click **OK** to allow the change.

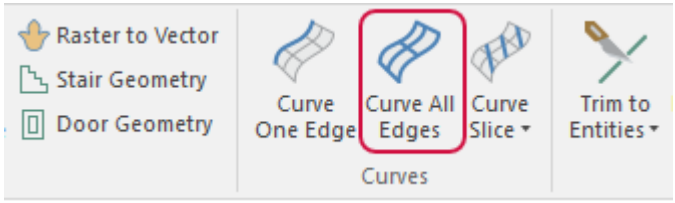
2. In the **Levels Manager**, select **Add a new level**.



Level **2** appears and becomes the active level. Note that it has 0 entities. The solid body has remained on level **1**.

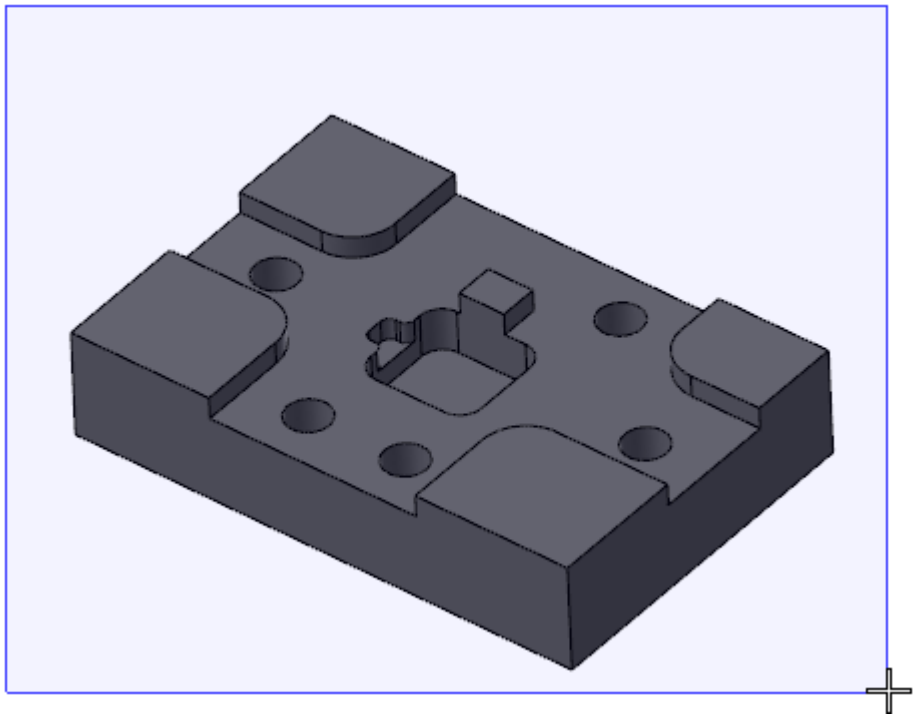


3. On the **Wireframe** tab, select **Curve All Edges**.



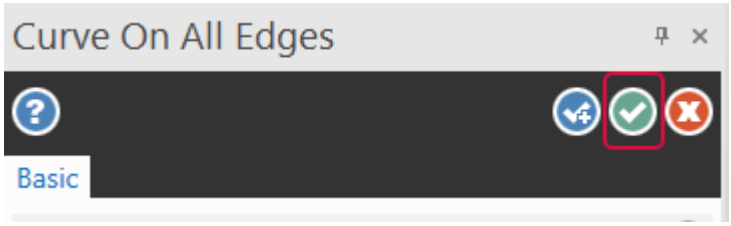
The **Curve On All Edges** function panel opens.

4. Window select your entire part.

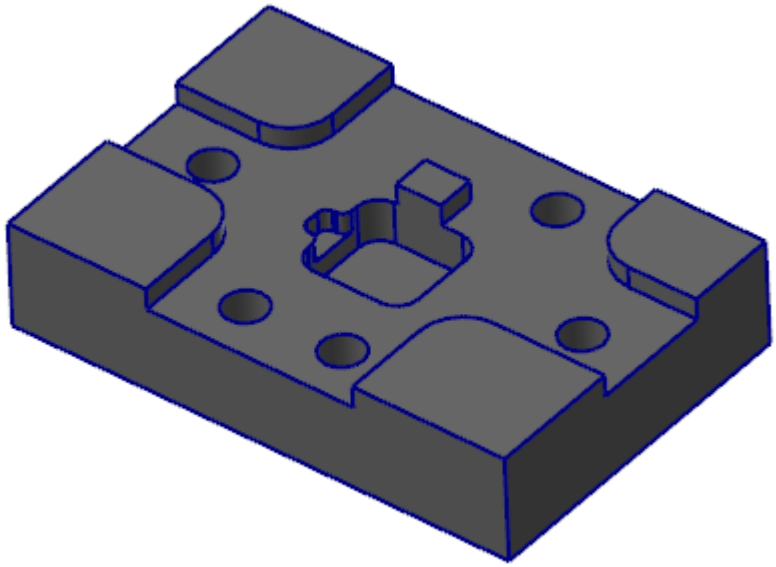


5. Click **End Selection**.

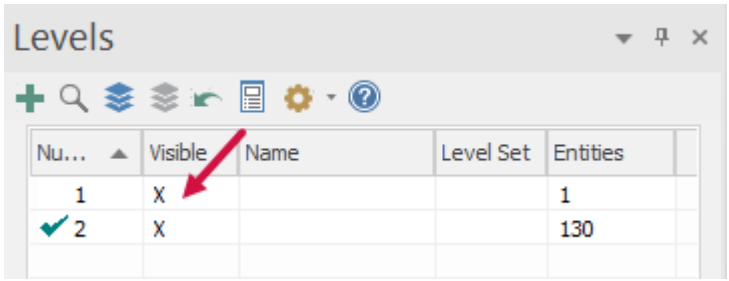
6. Click **OK** in the **Curve On All Edges** function panel.



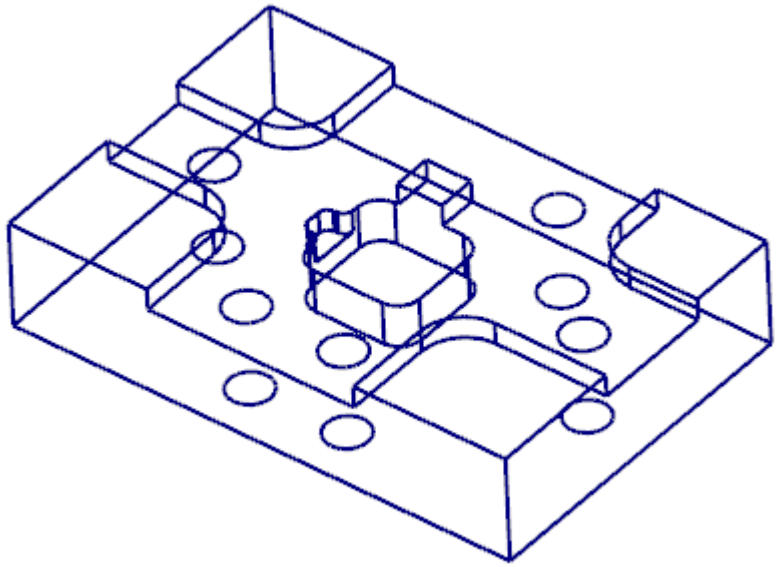
The solid body has a wireframe outline.



7. In the **Levels Manager**, hide level **1**.



The solid body is no longer visible. The wireframe remains.

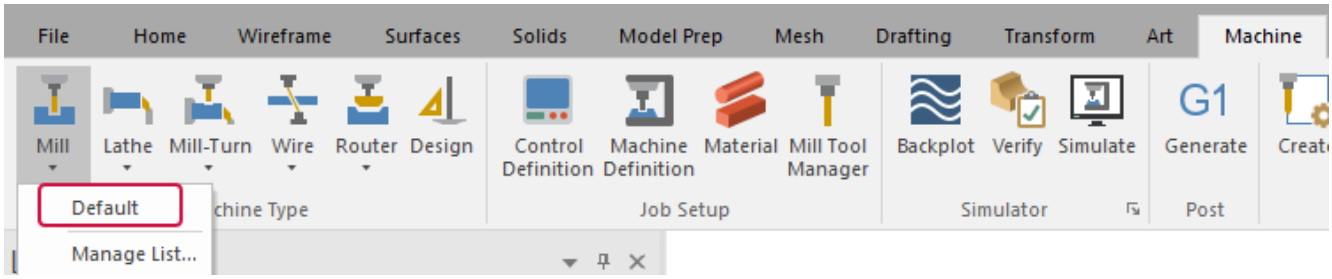


8. Save your part.

Exercise 2: Exploring Wireframe Chaining

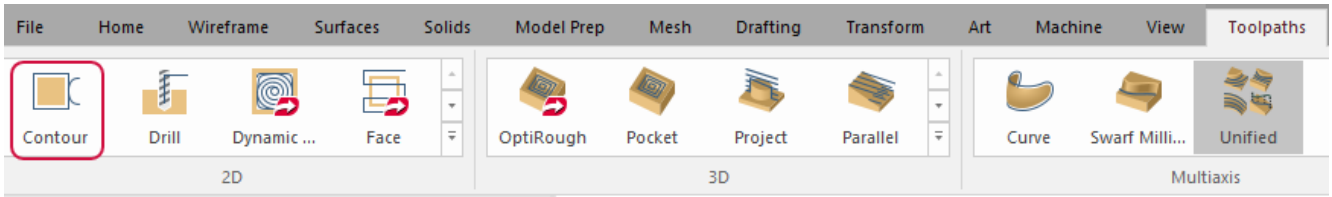
In this exercise, you explore wireframe chaining methods.

1. On the **Machine** tab, select **Mill**, **Default**.

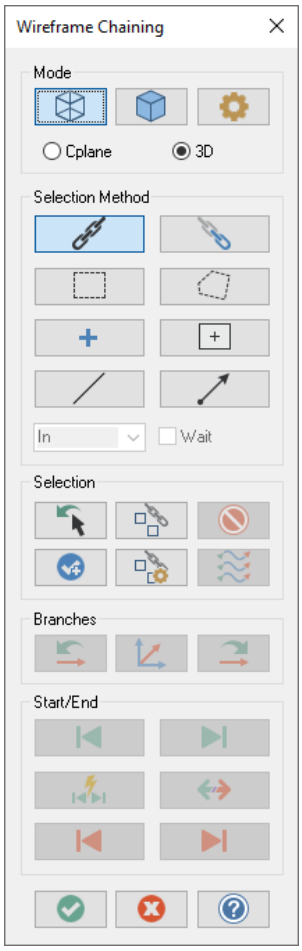


This sets your machine type, which is the first step in creating a toolpath. You will create a complete toolpath in the next chapter. For this chapter, you will follow the first steps of toolpath creation in order to practice chaining.

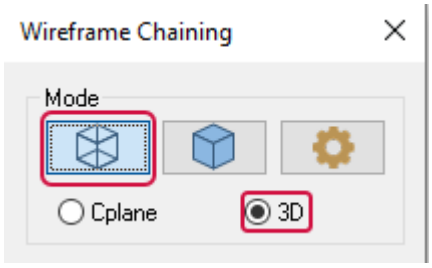
2. On the **Toolpaths** contextual tab, select **Contour** from the **2D** gallery. If you are not using a Mill, select any toolpath available to your machine that brings up the **Wireframe Chaining** dialog box.



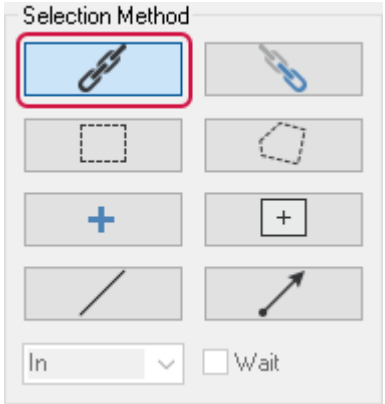
The **Wireframe Chaining** dialog box opens.



3. Check that **Wireframe** and **3D** are selected beneath **Mode**. These options allow you to select wireframe entities defined in the X,Y, and Z axes.

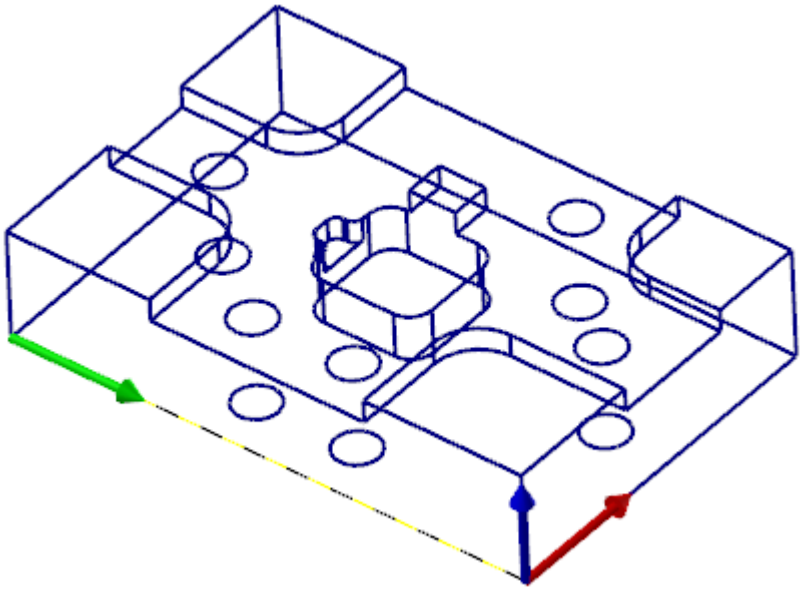


4. Be sure that **Chain** is selected under **Selection Method**.



Selection methods make it easier to select the desired geometry. For example, when **Chain** is selected in **3D** mode, Mastercam pauses chaining at all branches and prompts you to select the next entity.

5. Click the line shown below.

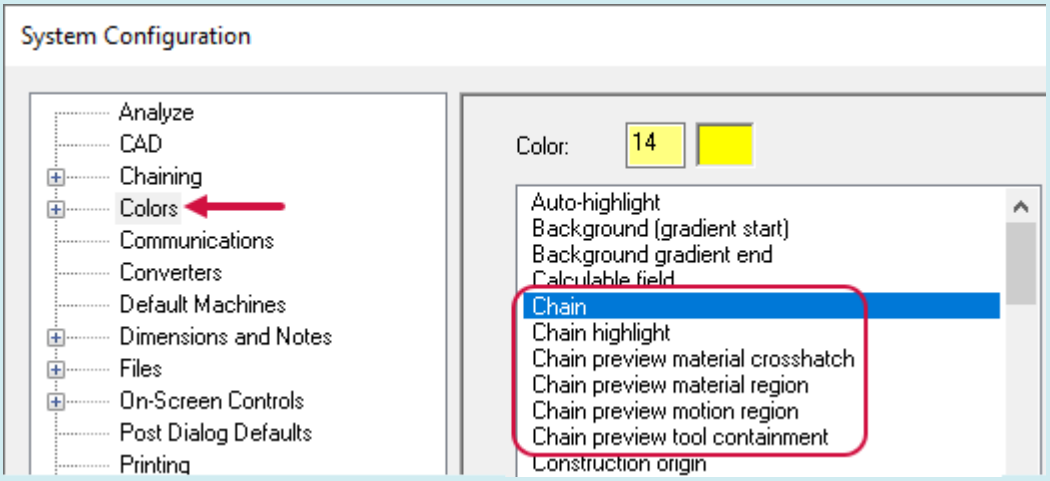


A chain appears with three arrows:

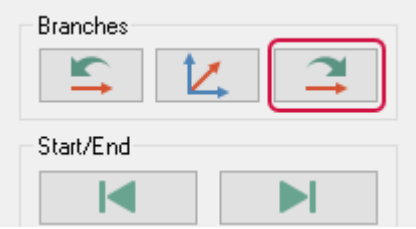
- Green appears at the start of the chain and shows the direction of the cutting tool.
- Red shows the default chaining direction.
- Blue shows alternate chaining directions.

These arrows allow for guided chaining. Guided chaining uses the **Next**, **Previous**, and **Adjust** buttons to control your chain.

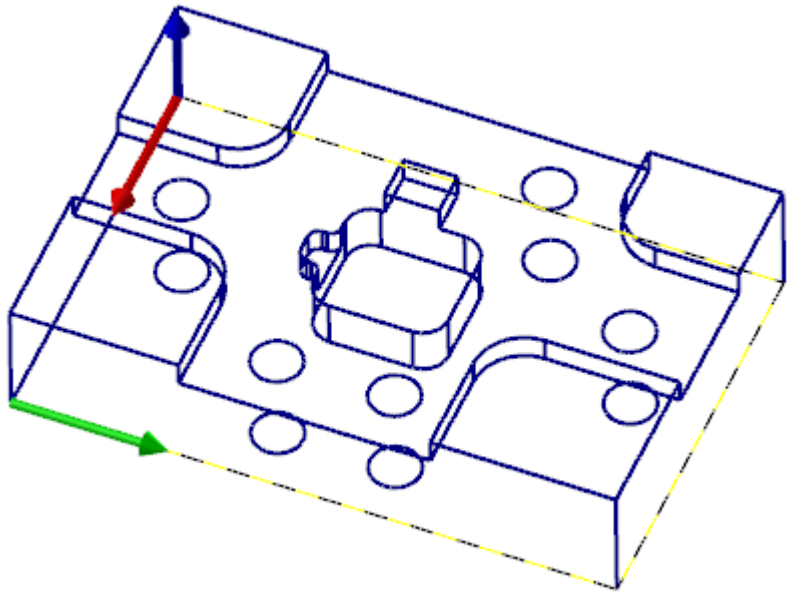
NOTE
You can customize your chain-related colors in the **Colors** page of the **System Configuration** dialog box.



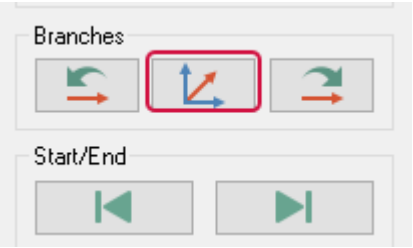
6. Click the **Next** button twice to continue your chain around the base of the wireframe.



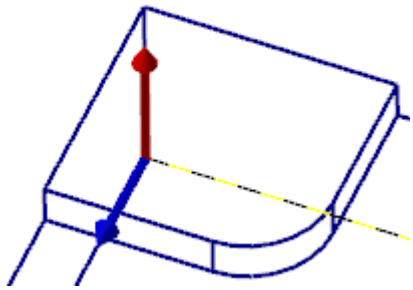
Each time you click **Next**, the chain continues in the default direction indicated by the red arrow.



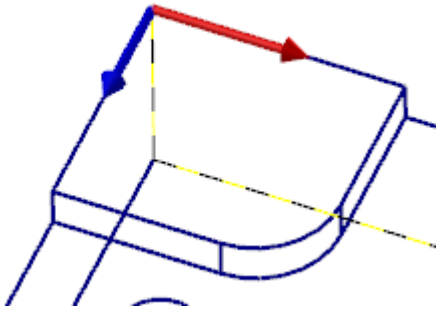
7. In the **Wireframe Chaining** dialog box, click **Adjust**.



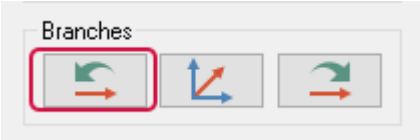
Adjust changes your default chaining direction. The red arrow of your chain switches direction.



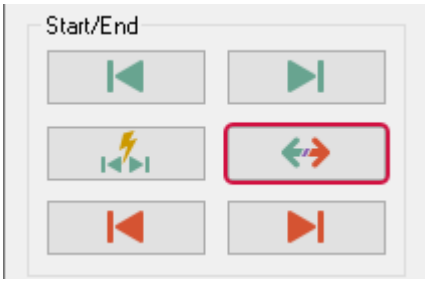
8. Click **Next**. Your chain continues up the corner edge of the wireframe.



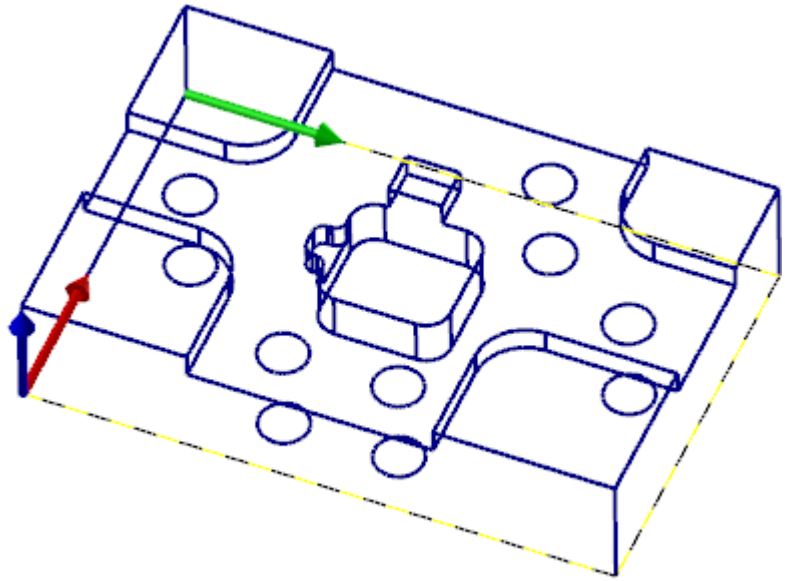
9. To undo the most recent addition to the chain, click **Previous**.



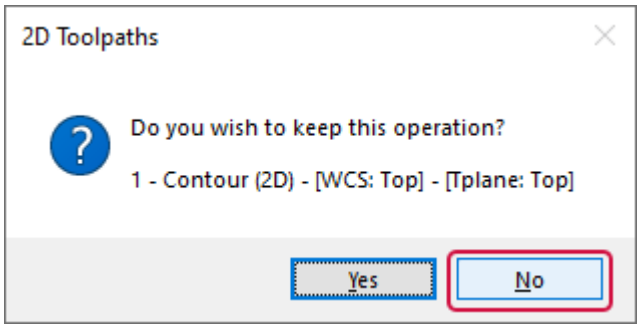
10. Click **Reverse** to switch the direction of your chain, as well as the start and end points.



NOTE
When chaining to create a toolpath, clicking **Reverse** changes the start point of your toolpath and direction of your cutting tool.



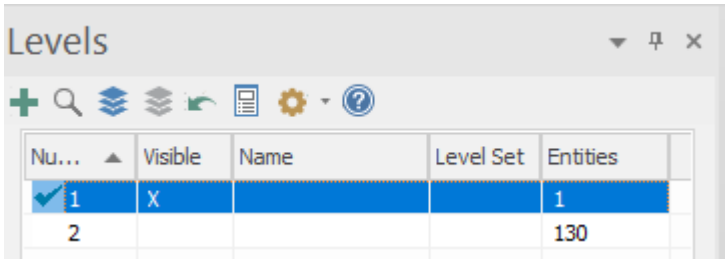
11. Press [**Esc**] and click **No** to exit the toolpath function.



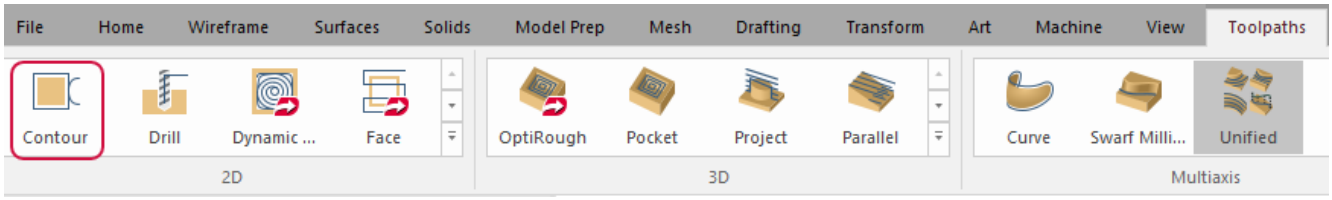
Exercise 3: Exploring Solid Chaining

In this exercise, you create and edit a solid chain.

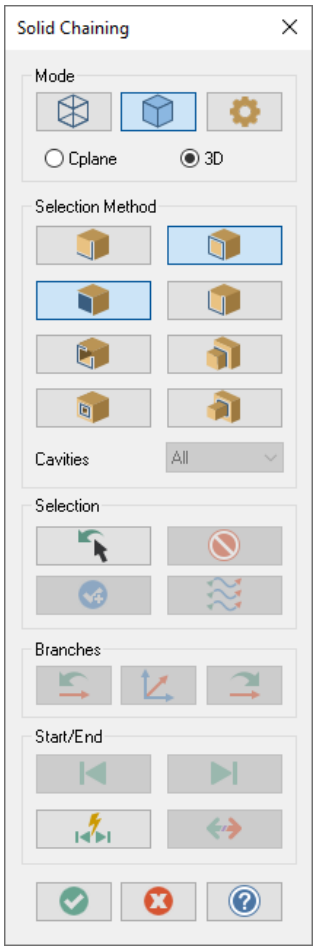
1. On the **Levels** tab, make level **1** active and visible. Click in the **Visible** column of **2** to make the level not visible.



2. On the **Toolpaths** contextual tab, select **Contour** from the **2D** gallery. If you are not using a Mill, select any toolpath available to your machine that brings up the **Solid Chaining** dialog box.

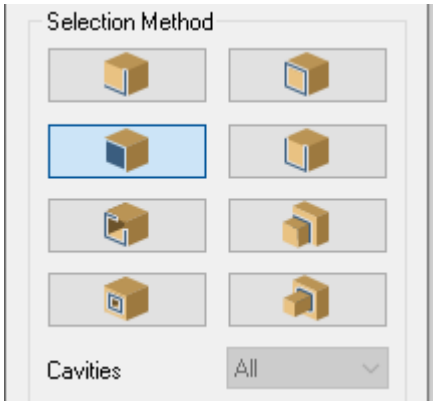


The **Solid Chaining** dialog box opens. It shares many similarities with the **Wireframe Chaining** dialog box.

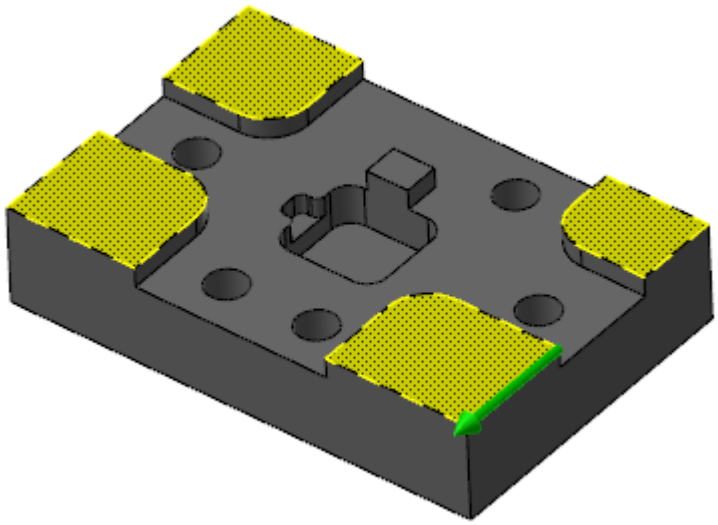


The default selection modes for solid chaining are **Loop** and **Face**.

3. Deselect **Loop** so that only **Face** is active.

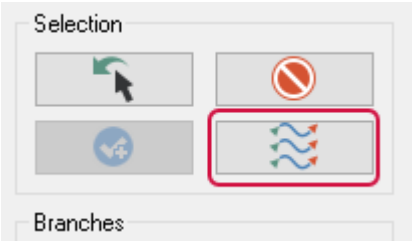


4. Select the four faces in the following image.

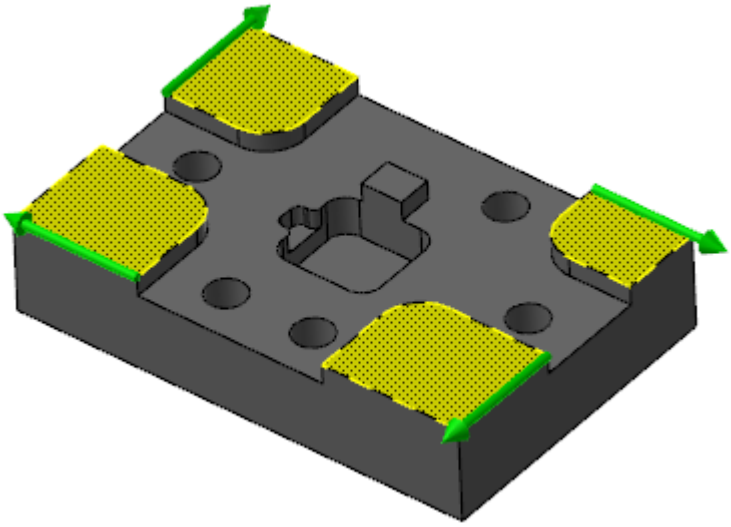


The green arrow appears on the last chain created.

5. Click **Display all selection arrows**.

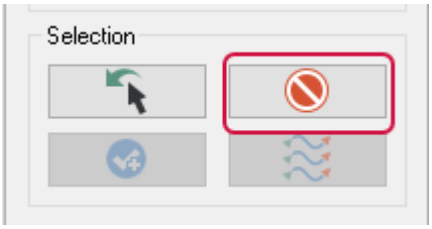


A green arrow appears to indicate the direction of each chain.



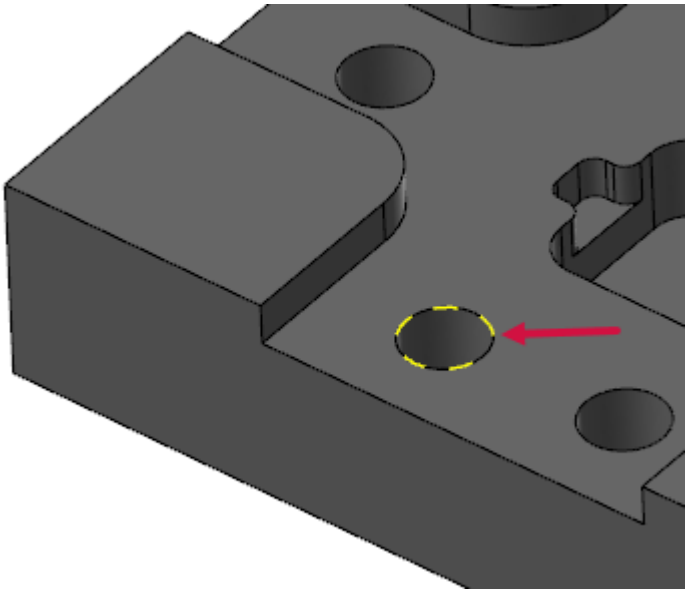
6. Select **Reverse** to reverse the direction of all selected chains.

7. Click **Unselect** to get rid of the selected chains.

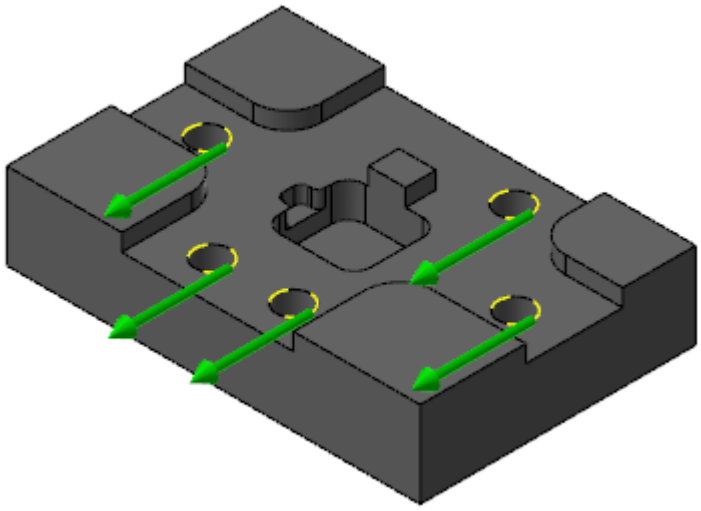


8. Change the selection method to **Loop**. Be sure to deselect **Face**.

9. Hold down [**Ctrl**] and click on the edge shown below.



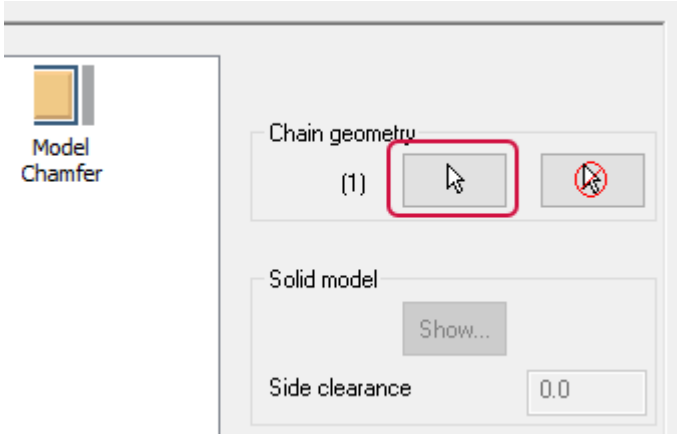
[**Ctrl + click**] allows you to select matching holes or fillets on a part. All circles on the same face as the original selection become chained.



10. Click **OK** to accept the chains.

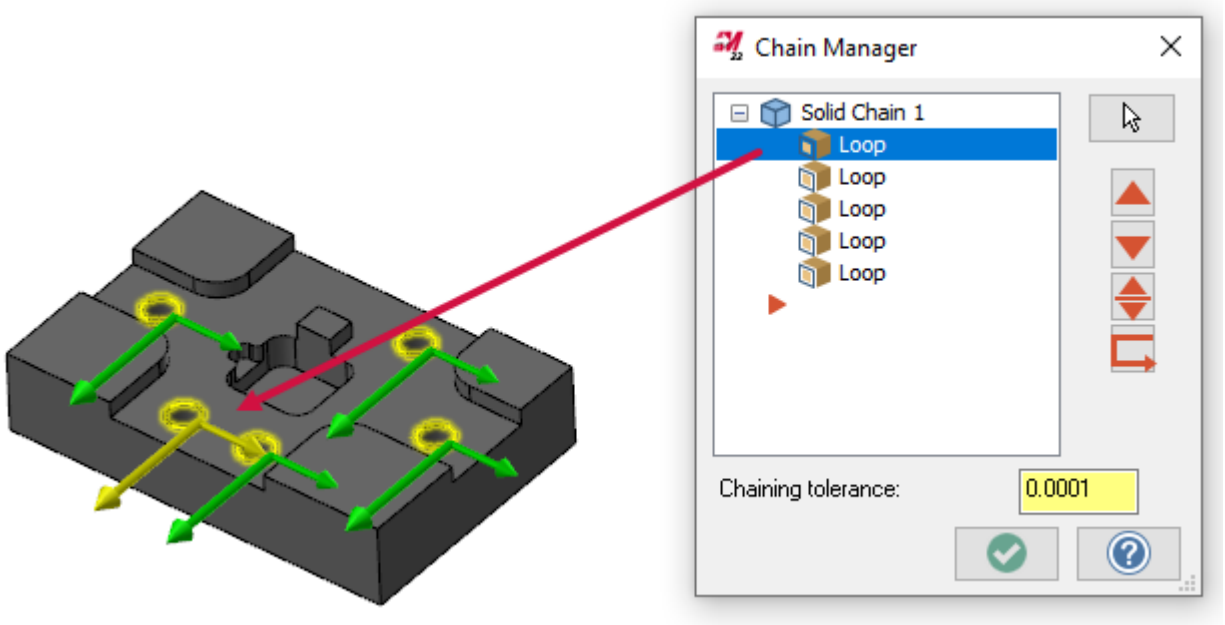
    The **2D Toolpaths - Contour** dialog box opens. This is where you program a toolpath. You can also edit chains here.

11. Click **Select chains** under **Chain geometry**.

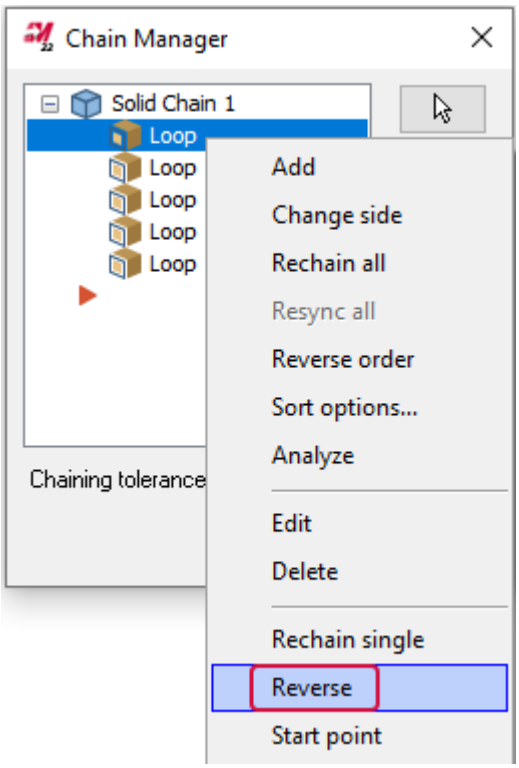


    Chain Manager opens.

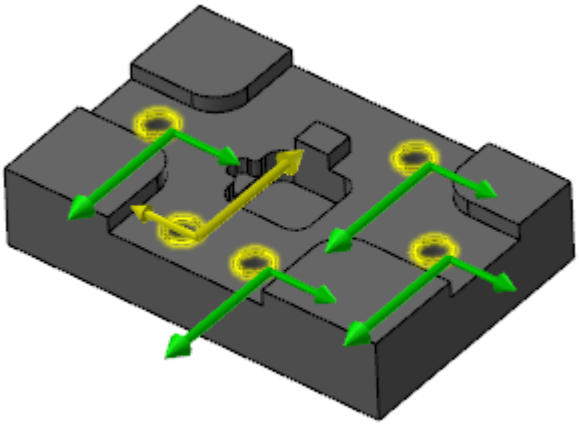
12. Expand **Solid Chain 1** and select the first **Loop**. Yellow arrows identify the loop on the part.



13. Right-click the first **Loop** and select **Reverse**.

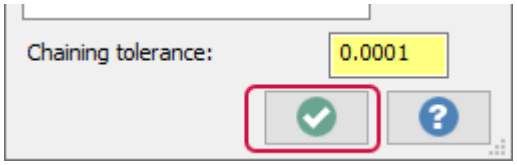


The first loop is reversed. The others maintain their original chaining direction.



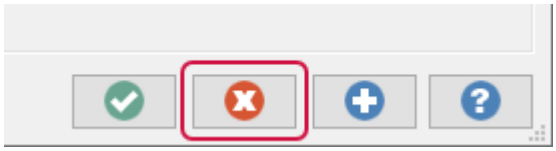
Use the right-click drop-down in **Chain Manager** when you want to analyze, edit, name, and make other changes to your chains.

14. Click **OK** to accept the new chains.

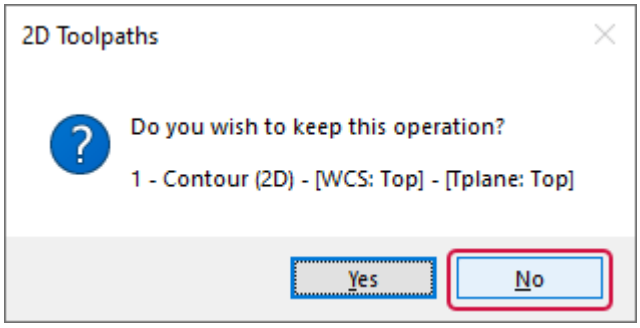


You return to the **2D Toolpaths - Contour** dialog box. After chaining, the next step in toolpath creation is to set parameters. You will do this in the next chapter.

15. Click **Cancel** to exit the dialog box.



16. Select **No** to discard your toolpath.



17. Save your part.

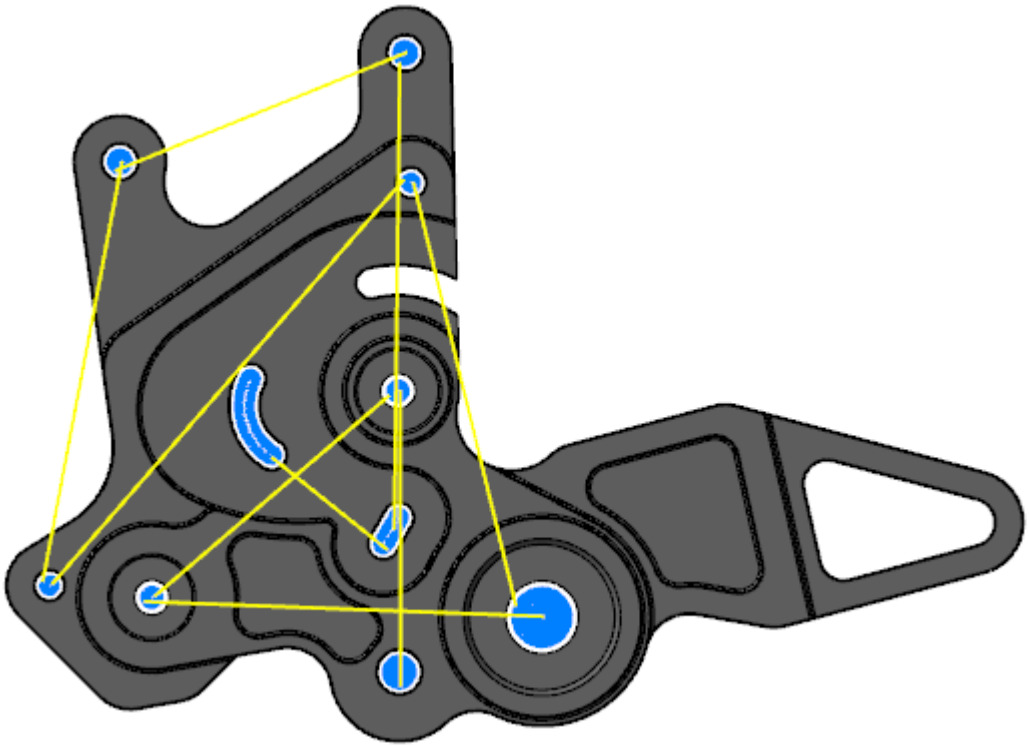
You have now completed this chapter of the *Mastercam Basics Tutorial*. In the next chapter, you learn more about the creating toolpaths.

CHAPTER 9

INTRODUCTION TO TOOLPATHS

As a CAD/CAM program, Mastercam allows you to create both the geometry and toolpaths needed to machine your part and achieve your project goals. Toolpaths are the routes along which the cutting tool is instructed to travel. To effectively meet the needs of users, Mastercam is capable of creating toolpaths that range from simple to complex.

For example, the following image shows a 2D milling toolpath.



One of the most important features of Mastercam is associativity. Associativity is the link between the part and the toolpath. When the part changes shape or moves, the associated toolpaths adapt and change as well. If there is a design change with the part, associativity makes it so the toolpaths just need to be recalculated, as opposed to completely reprogrammed.

To create toolpaths, input specific information such as the machine type, tool specifications, and size and shape of your material. You also select a toolpath type and set the feeds and speeds at which your part will be machined.

Mastercam takes this information and post processes it. This turns it into G-code (sometimes known as NC-code) to be read by a CNC machine. Depending on the type of machine and your part, this process may be entirely automated or may require occasional assistance from the machine operator.

Goals

- Choose a machine and set up stock.
- Select a toolpath and geometry.
- Set parameters.

- Backplot and verify.
- Post the part file.

NOTE
This chapter requires a Mill machine. If your Mastercam license does not allow access to Mill, you can either:

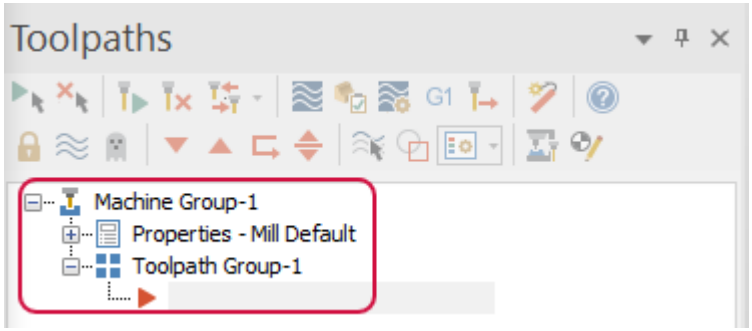
- Download Mastercam Learning Edition and use it to complete the exercise: https://signup.mastercam.com/demo-hle. Note that you will not be able to complete Exercise 5 with Mastercam Learning Edition.
- Read through the exercise.

Exercise 1: Selecting a Machine and Setting Up Stock

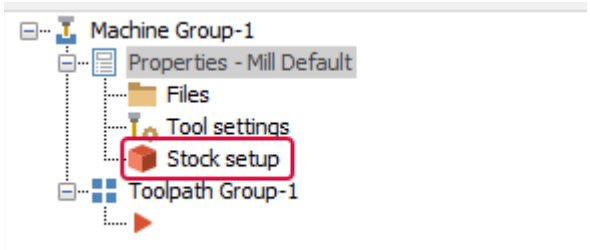
In this exercise, you select a machine type and set up your stock.

Each machine type allows access to toolpaths specific to that machine. Selecting a machine is the first step in setting up a toolpath. Once a machine is selected, you set up the stock material boundaries. Setting up stock enters the dimensions and location of your initial material.

1. If necessary, open the part you created in "Working With Chain Selection" on page 143.
2. Go to the **Toolpaths Manager**. **Machine Group-1** is present because you selected a Mill machine during the chapter on selection. Machine groups organize your cutting operations by machine.

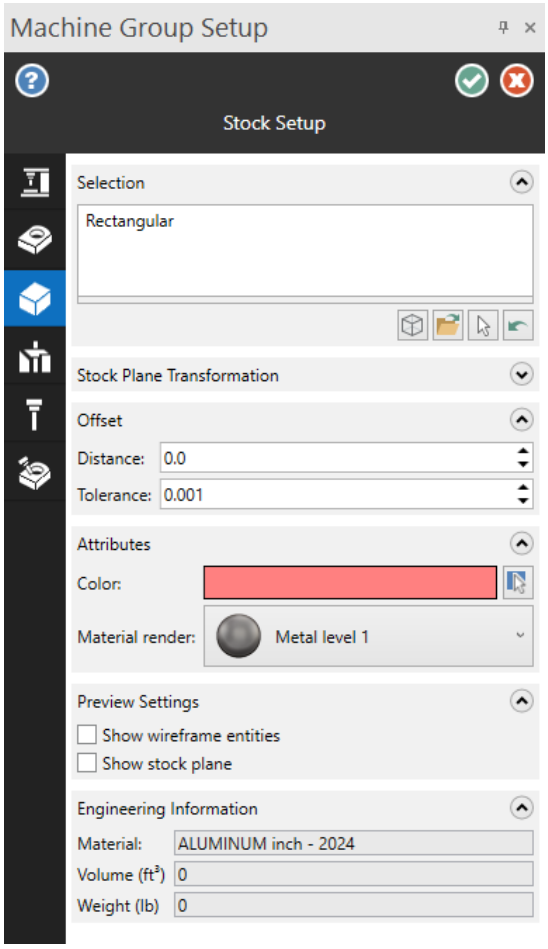


3. Expand **Properties - Mill Default**.
4. Click **Stock setup**.



Stock is the material from which your part will be cut.

The **Machine Group Setup** function panel opens to the **Stock Setup** page.

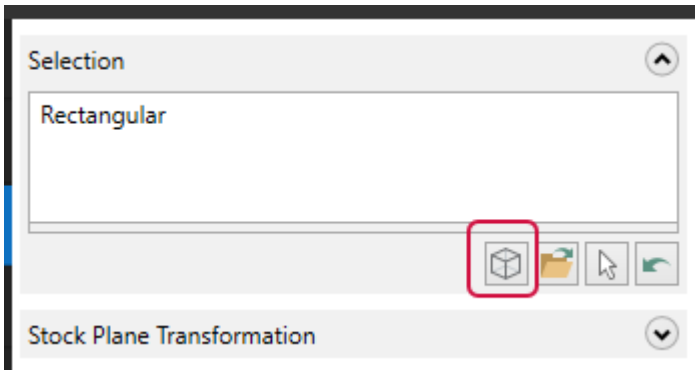


The Machine Group Setup function panel (accessed from the Toolpaths Manager) manages machine groups for Mill and Router machines. It features a top-down workflow, represented by the icons on the left side of the panel. When creating a toolpath, use the pages in this panel to:

- Select machine files, tool and operations libraries, and operation defaults.
- Define the final part (the workpiece) after all machining is complete.
- Configure stock shape, origin, and wireframe.
- Define workholding components, such as fixtures or vices.
- Set feed calculation, material type, and toolpath numbering.
- Set up simulation options without having to open Mastercam Simulator.

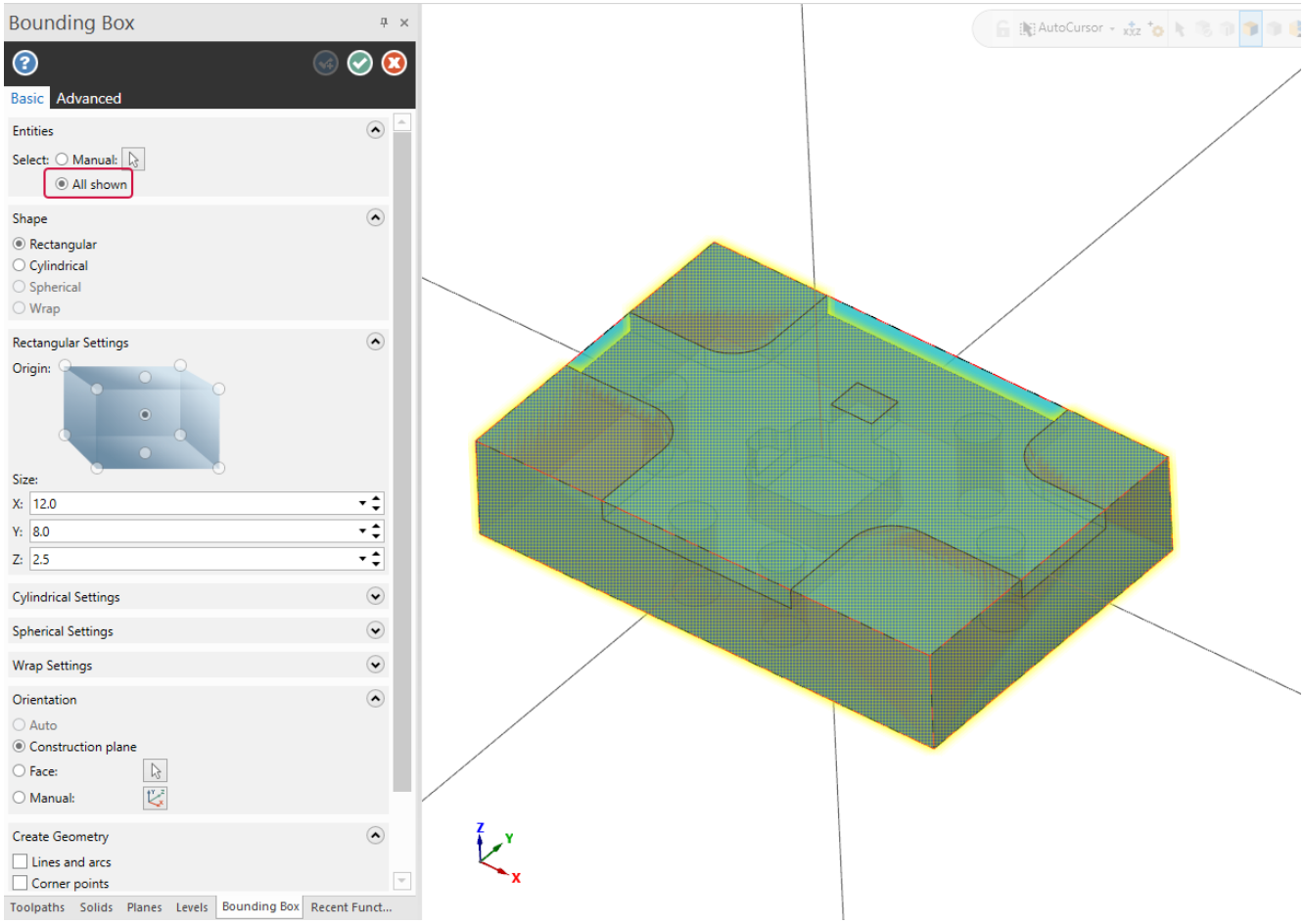
NOTE
Since we are using the default Mill machine in this exercise, we do not need to set up our machine files in the top (**Machine**) page. **Master Model** or workpiece setup, on the next page, is optional.

5. Click **Add from a bounding box**.



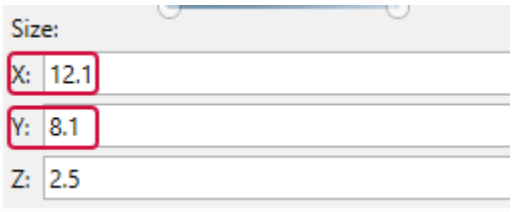
The **Bounding Box** function panel opens. A bounding box creates a boundary around selected entities in the graphics window.

6. Select the **All shown** option to select the part.



> **NOTE**
> To display the coordinate axes in the graphics window, press [**F9**] .

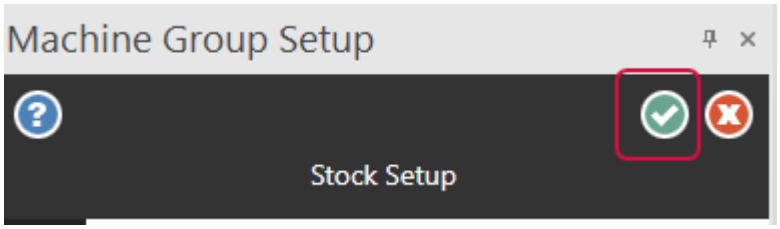
7. In the **Size** section of the **Bounding Box** function panel, change **X** to **12.1** and **Y** to **8.1**.



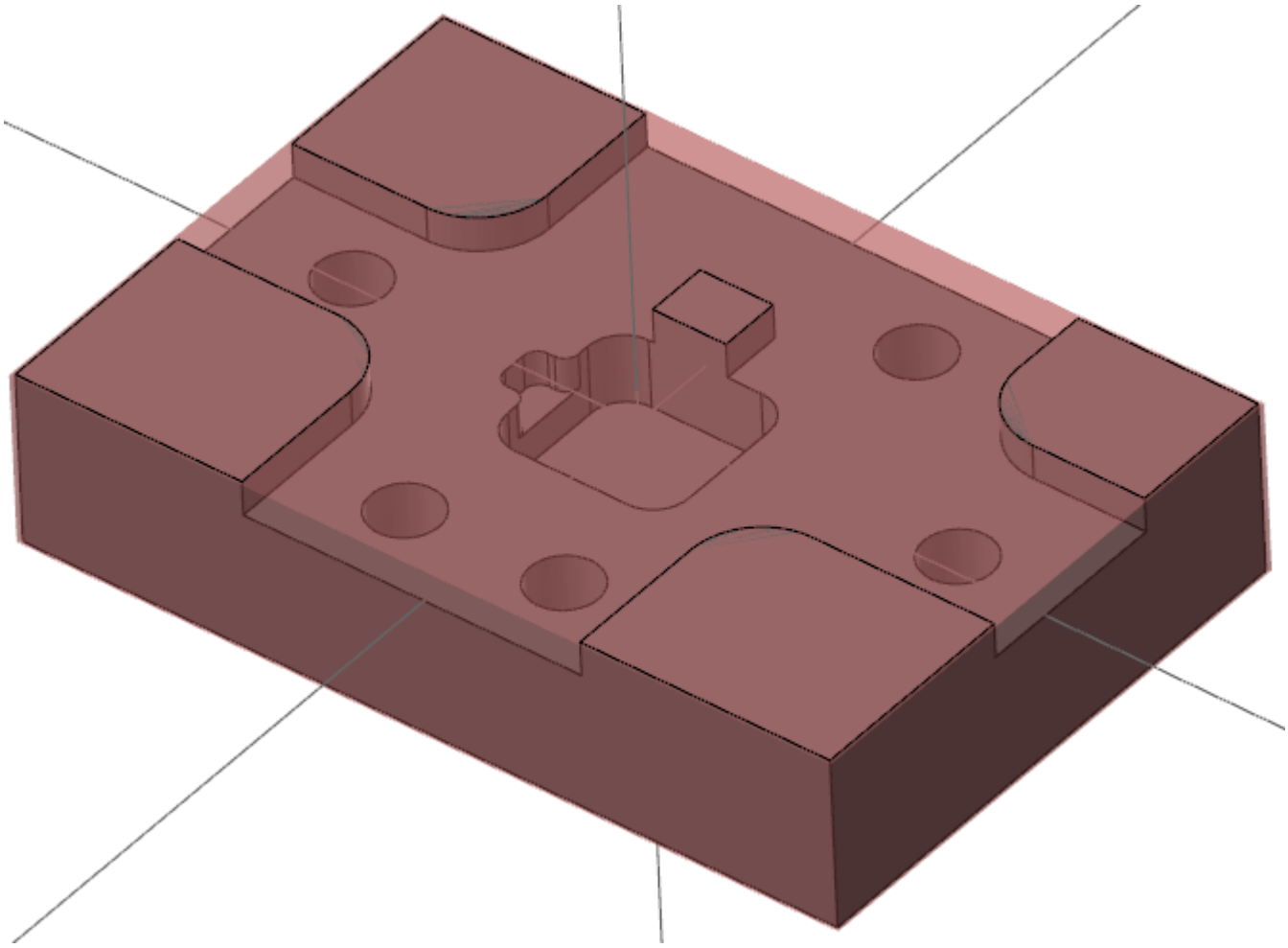
This adds a small amount of material to your stock in the X and Y directions. Although your stock values would ultimately be defined based on the actual stock that you have available, for the purposes of this exercise we only want to skim off a small amount of material.

8. Click **OK** in the **Bounding Box** function panel.

9. Click **OK** in the **Machine Group Setup** panel.



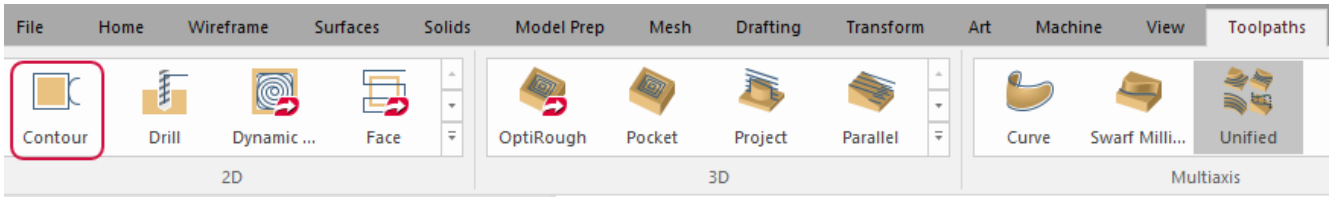
10. If necessary, toggle **Stock Display** and **Stock Shading** in the **Toolpaths** tab to see the stock in the graphics window.



Exercise 2: Choosing a Toolpath and Geometry

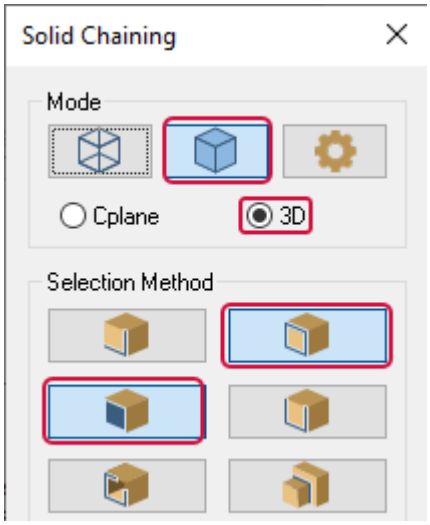
In this exercise, you choose a toolpath and chain associated geometry.

1. On the **Mill Toolpaths** contextual tab, select **Contour** from the **2D** group.

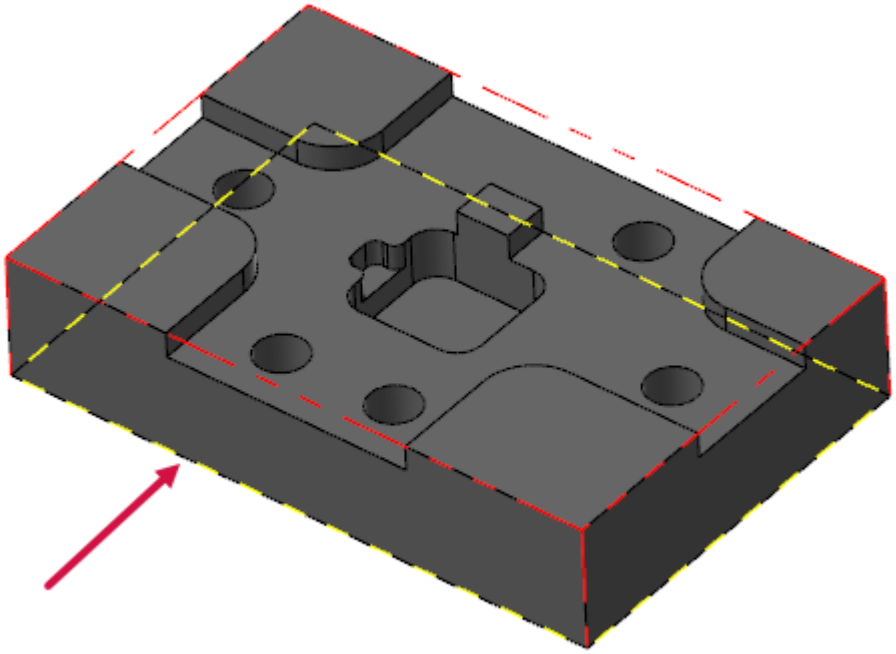


The **Solid Chaining** dialog box opens.

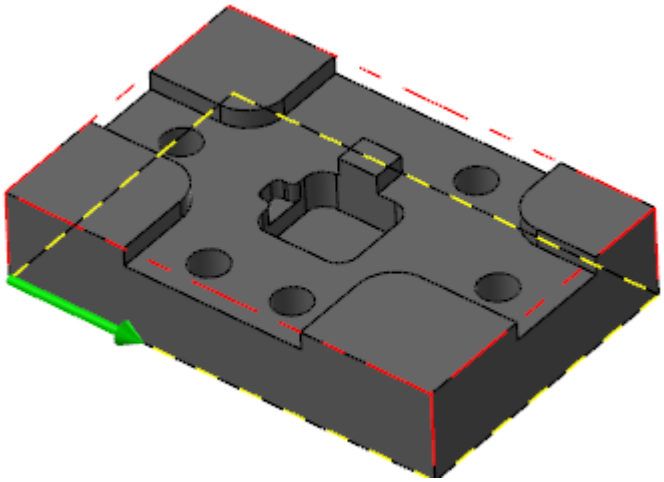
1. In the **Solid Chaining** dialog box, check that **Solids** and **3D** are selected in the **Mode** group. In the **Selection Method** group, check that **Loop** and **Face** are selected. These options allow you to chain the desired geometry.



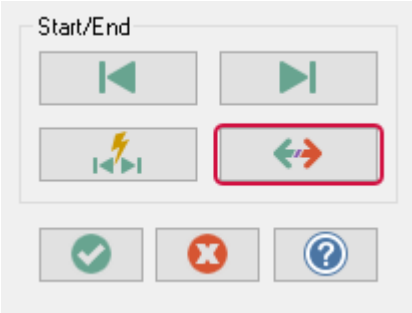
2. Select the chain shown in the following image.



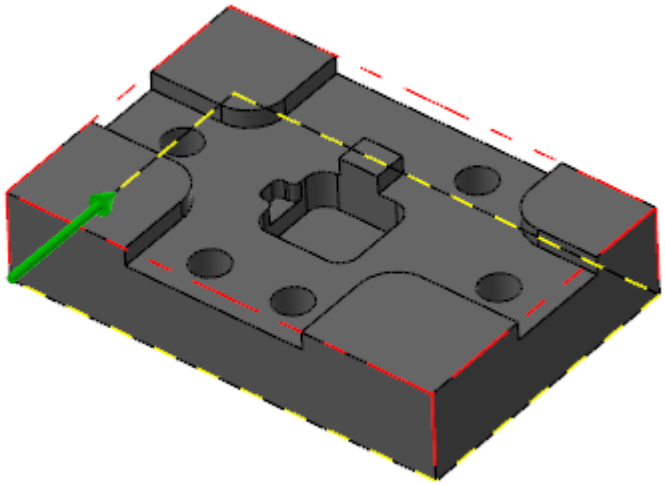
A green directional arrow appears. This arrow shows the direction the tool will follow. Typically, you want the cutter to move in a clockwise direction so that it will be on the left side of the chain. This is not the case in the following image.



3. If necessary, select **Reverse** to reverse the direction of the arrow so that the part is to the right side of the green arrow.

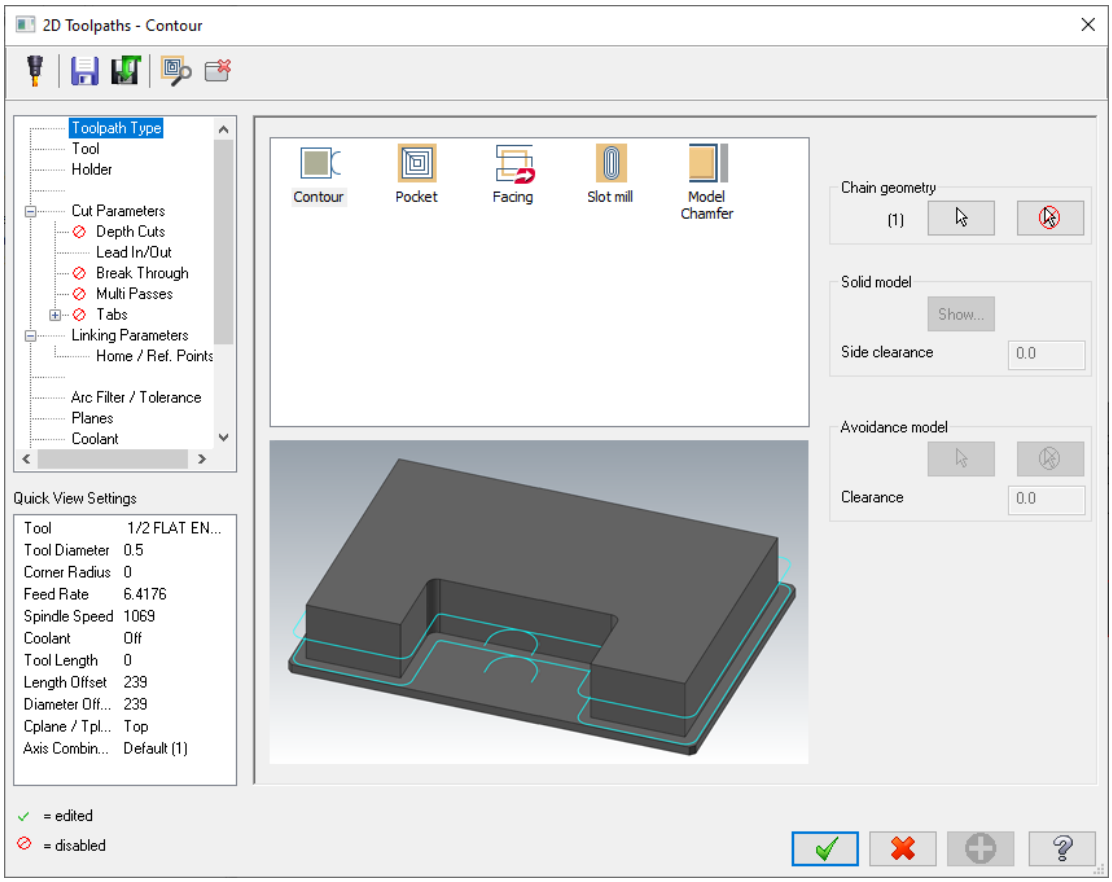


Your part is now to the right of the green arrow. Your cutter will be on the left side of the chain.



4. Click **OK** in the **Solid Chaining** dialog box.

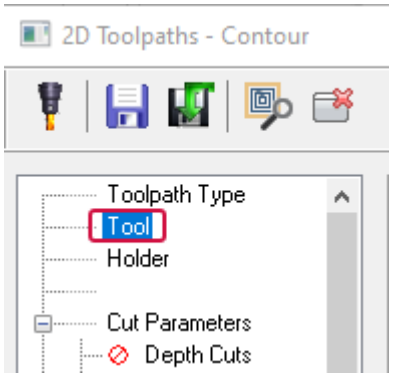
The **2D Toolpaths - Contour** dialog box opens. This is where you program your toolpath.



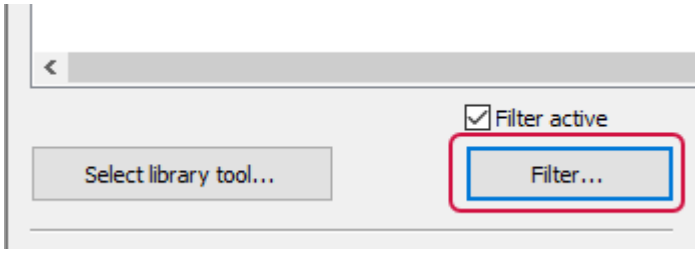
Exercise 3: Programming a Toolpath

In this exercise, you program your toolpath by selecting a tool and setting parameters.

1. Click the **Tool** page in the **2D Toolpaths - Contour** dialog box.

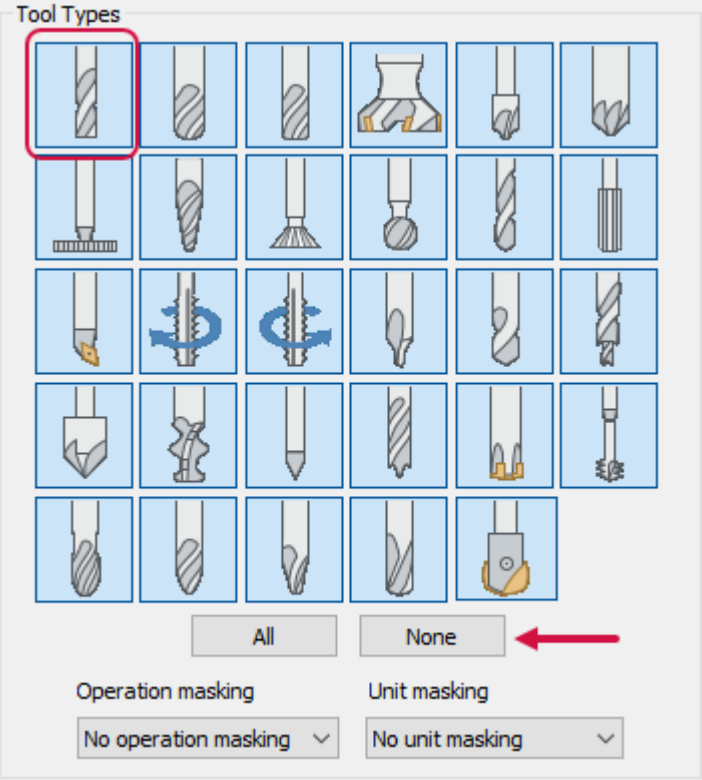


2. Click **Filter**.

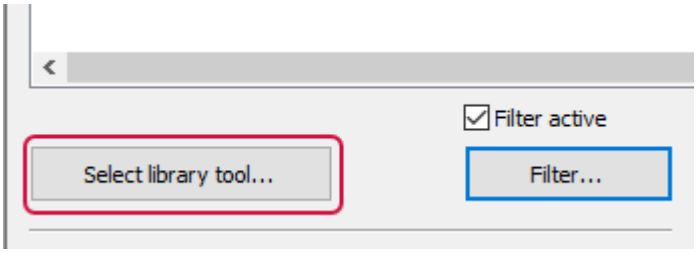


The **Tool List Filter** dialog box opens. Use this to filter your tools by type.

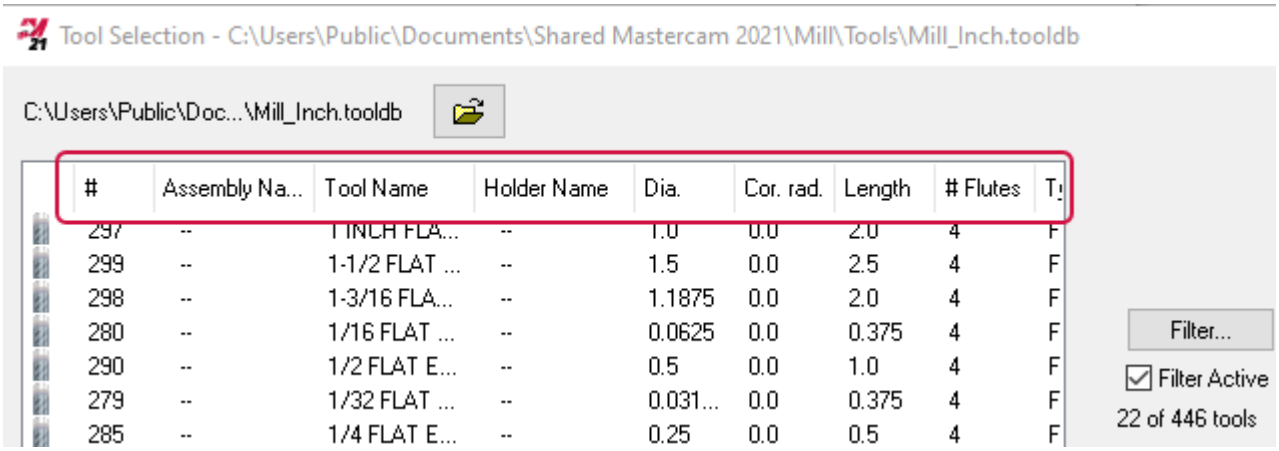
3. Select **None** to clear filters. Choose **Endmill1 Flat** from **Tool Types**.



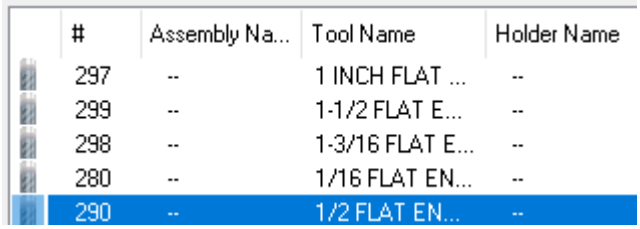
4. Click **OK**.
5. Click **Select library tool**.



The **Tool Selection** dialog box opens. All tools listed are endmills because they are filtered. Use the row headers to sort tools by characteristics such as name, diameter, or flutes.

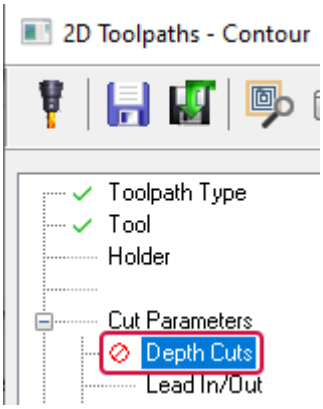


6. Select **1/2 Flat Endmill** and click **OK**.

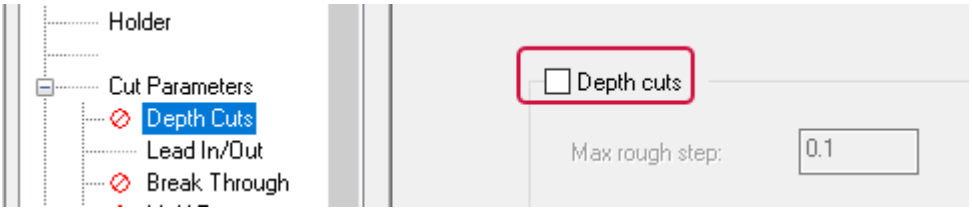


You return to the **2D Toolpaths - Contour** dialog box.

7. Select the **Depth Cuts** page.

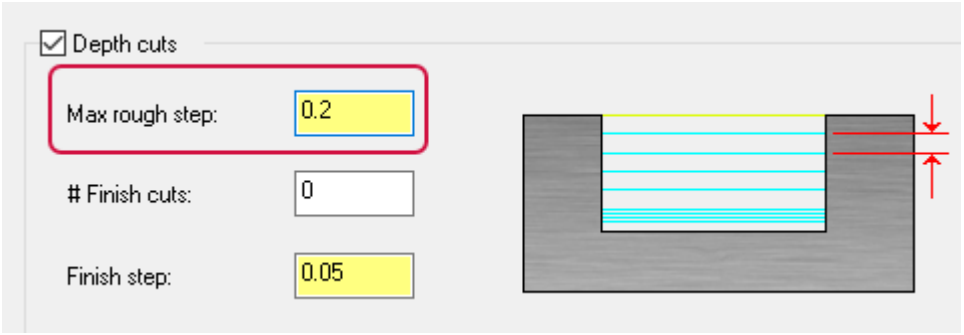


8. Select **Depth cuts**.

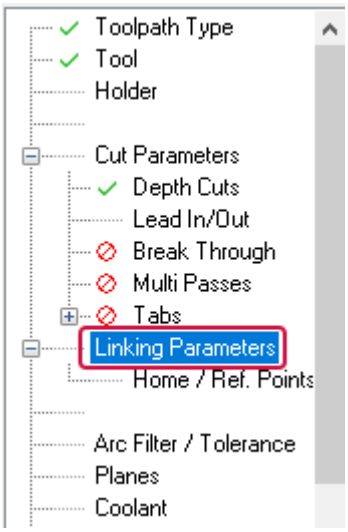


Depth cuts divide the total depth into separate depth cuts.

9. Change **Max rough step** to **0.2**. This is the maximum amount of material removed with each rough cut.



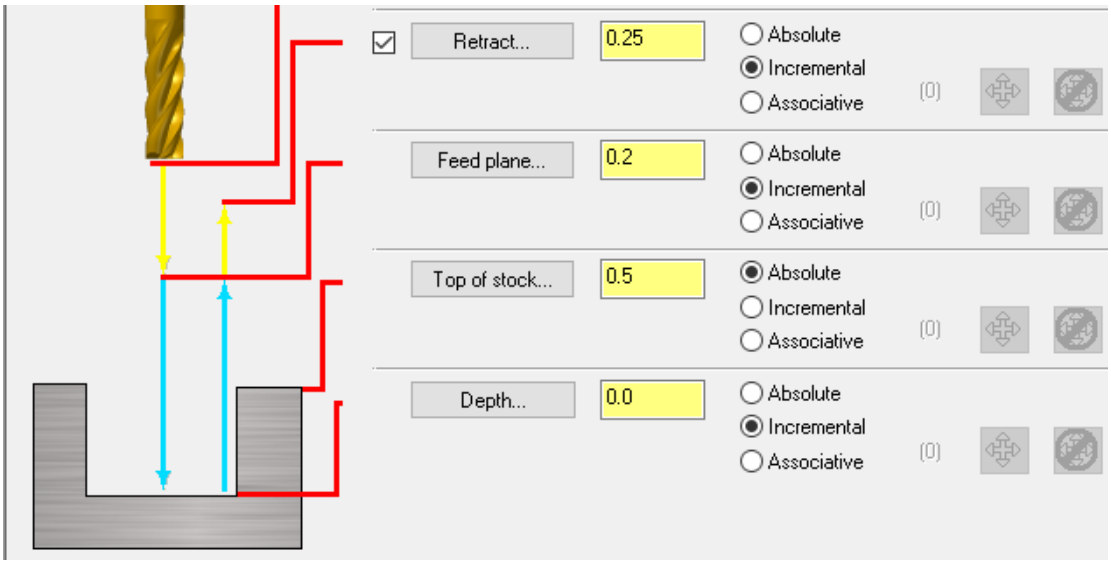
10. Select the **Linking Parameters** page.



The **Linking Parameters** page defines how Mastercam calculates heights and depths. These can be measured in absolute, incremental, or associative values.

11. Select the check box next to **Clearance**. Clearance sets the height at which the tool moves to and from the part.

12. Enter **2.00** and select **Absolute** for **Clearance**.
13. Enter the following, as necessary, on the **Linking Parameters** page.

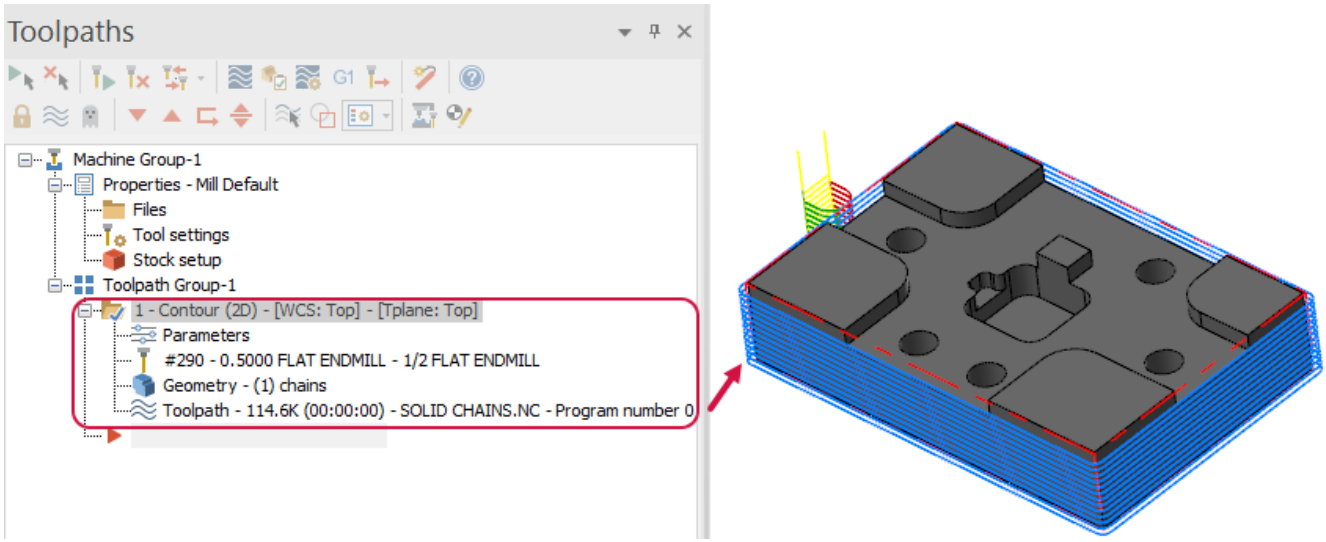


- **Retract**: Enter **0.25** and select **Incremental** to set the height that the tool moves up before the next tool pass. Incremental values are relative to other parameters of chained geometry.
- **Feed plane**: Enter **0.2** and select **Incremental** to set the height to which the tool rapids before changing to the plunge rate to enter the part.

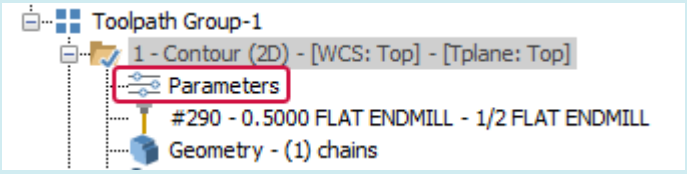
- **Top of stock**: The bounding box you created has pre-populated the **Top of stock** value to set the height of the material in the Z axis. Select **Absolute** to measure from the origin 0,0,0.
- **Depth**: Enter **0** and select **Incremental** to set the final machining depth and the lowest depth that the tool descends into the stock.

NOTE
You can explore the other pages, such as **Holder**, **Lead In/Out**, and **Break Through**. For this tutorial those settings remain at default.

14. Click **OK**. The toolpath information appears in the Toolpaths Manager. Your 2D Contour toolpath appears in the graphics window.



NOTE
To edit your toolpath, click on **Parameters**. The **2D Toolpaths - Contour** dialog box will re-open.

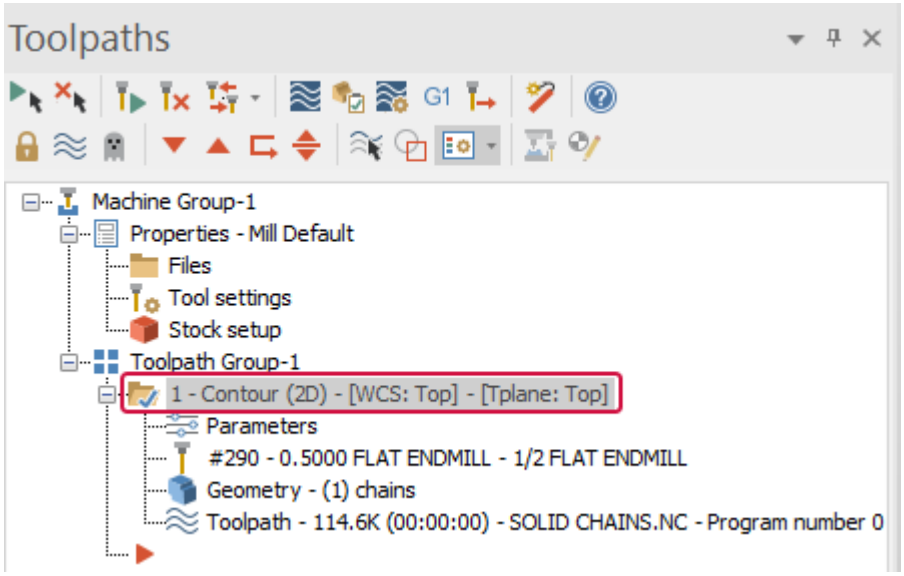


Exercise 4: Backplotting and Verifying

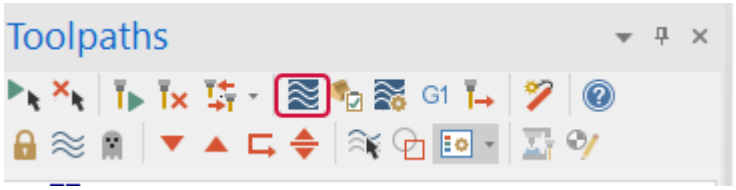
In this exercise, you use Classic Backplot to backplot a toolpath and use Mastercam Simulator to verify the toolpath.

Classic Backplot allows you to see a quick simulation of your toolpaths. Verify allows you to see a more detailed view of your operations, including simulated material removal.

1. In the **Toolpaths Manager**, check that your 2D Contour operation is selected.



2. Click **Backplot selected operations**.

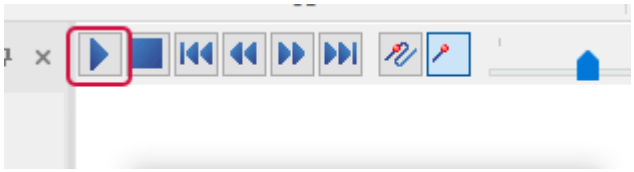


The following appear in the graphics window:

- Backplot dialog box: Controls the backplot display and allows you to save geometry or settings.
- Play bar: Adjusts the speed of the simulation and allows you to pause, play, and set stops.
- Tool and holder: Simulate the selected tool and holder.

> **NOTE**
> If you do not see the tool and holder, right-click in the graphics window and select **Fit**.

3. Click the **Play** button or press [**R**].

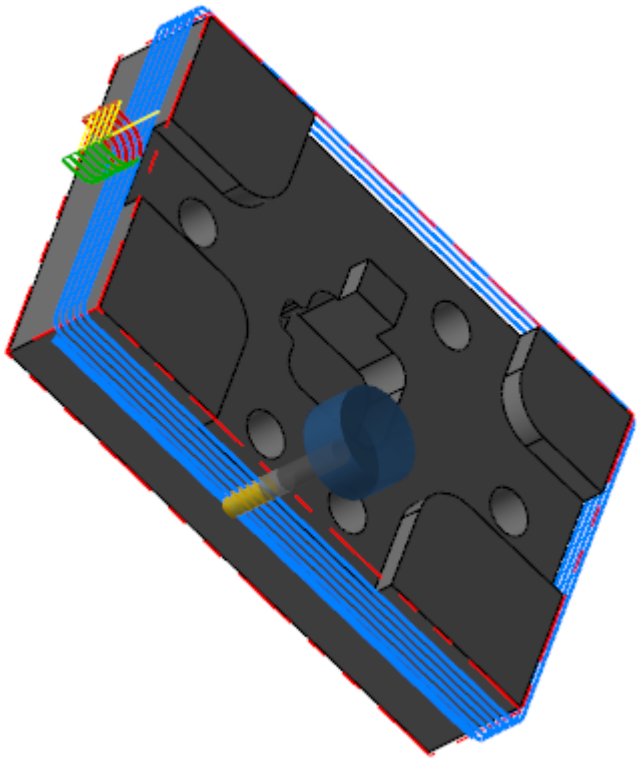


The tool and holder follow the toolpath.

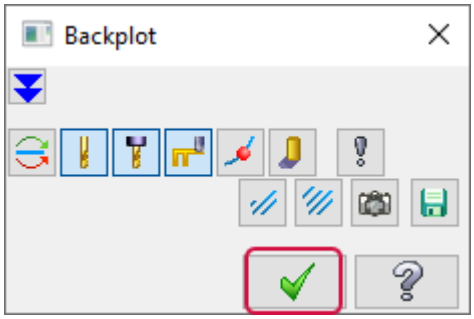
4. When the simulation has finished, change the simulation speed by adjusting the run speed slider.

5. Press [**R**] to view the adjusted simulation.

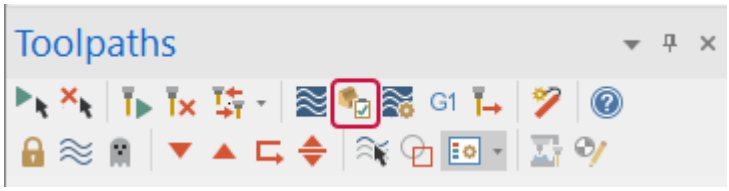
6. While the simulation is running, rotate your part. View the tool as it moves along the toolpath.



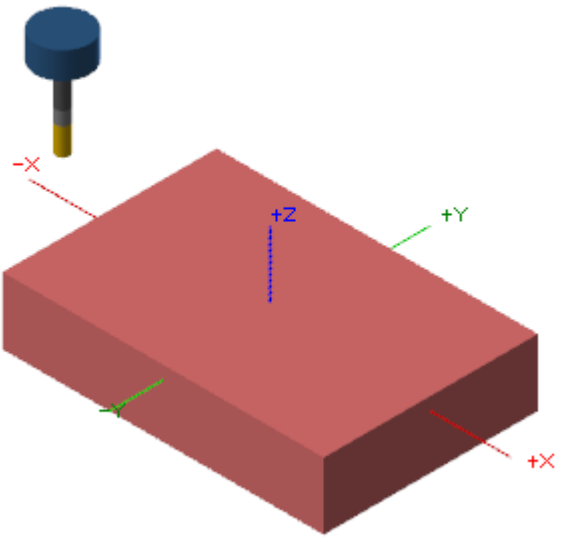
Click **OK** in the **Backplot** dialog box.



7. Reset your view to **Isometric** and **Fit**.
8. Select **Verify selected operations**.



Mastercam Simulator opens. Your tool, toolholder, and stock are visible in the graphics window.

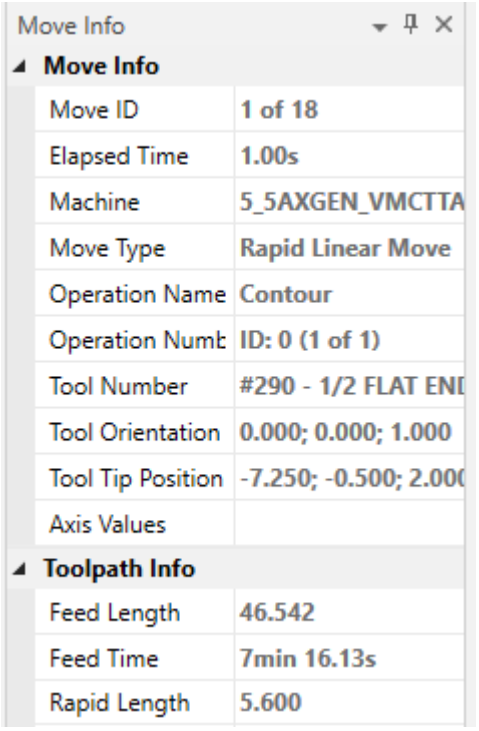


9. Click **Play**.

Mastercam Simulator shows the path of your tool and holder. The stock updates as material is removed from the part.

10. Reset the simulation by clicking **Start**.

11. Click **Play** and observe the information in the **Move Info** pane. One of the benefits of Mastercam Simulator is that it gives you more in-depth information about your operation.



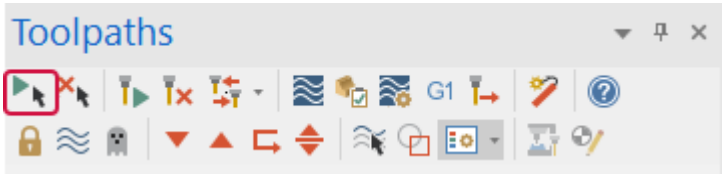
12. Exit Mastercam Simulator.

Exercise 5: Post Processing

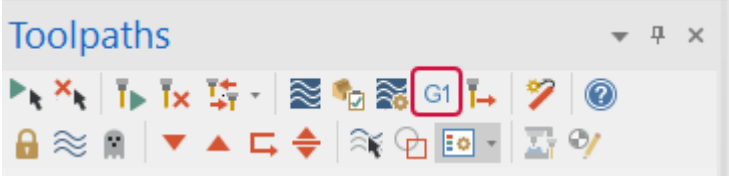
In this chapter you post process your part file.

Post processing, or posting, refers to the process by which the toolpaths in your part file are converted to a format that can be understood by your machine tool's control (for example, G-codes). A program called a post processor, or post, reads your part file and writes the appropriate NC code.

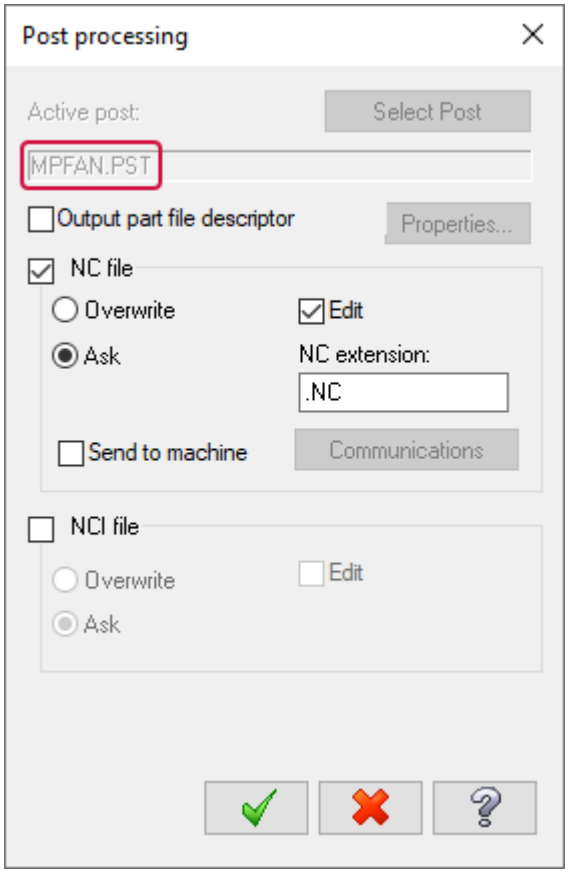
1. Click **Select all operations**.



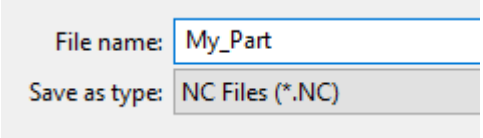
2. Click **Post selected operations** to post the toolpaths for the selected operations. The post processor creates a machine-readable NC program from the Mastercam file.



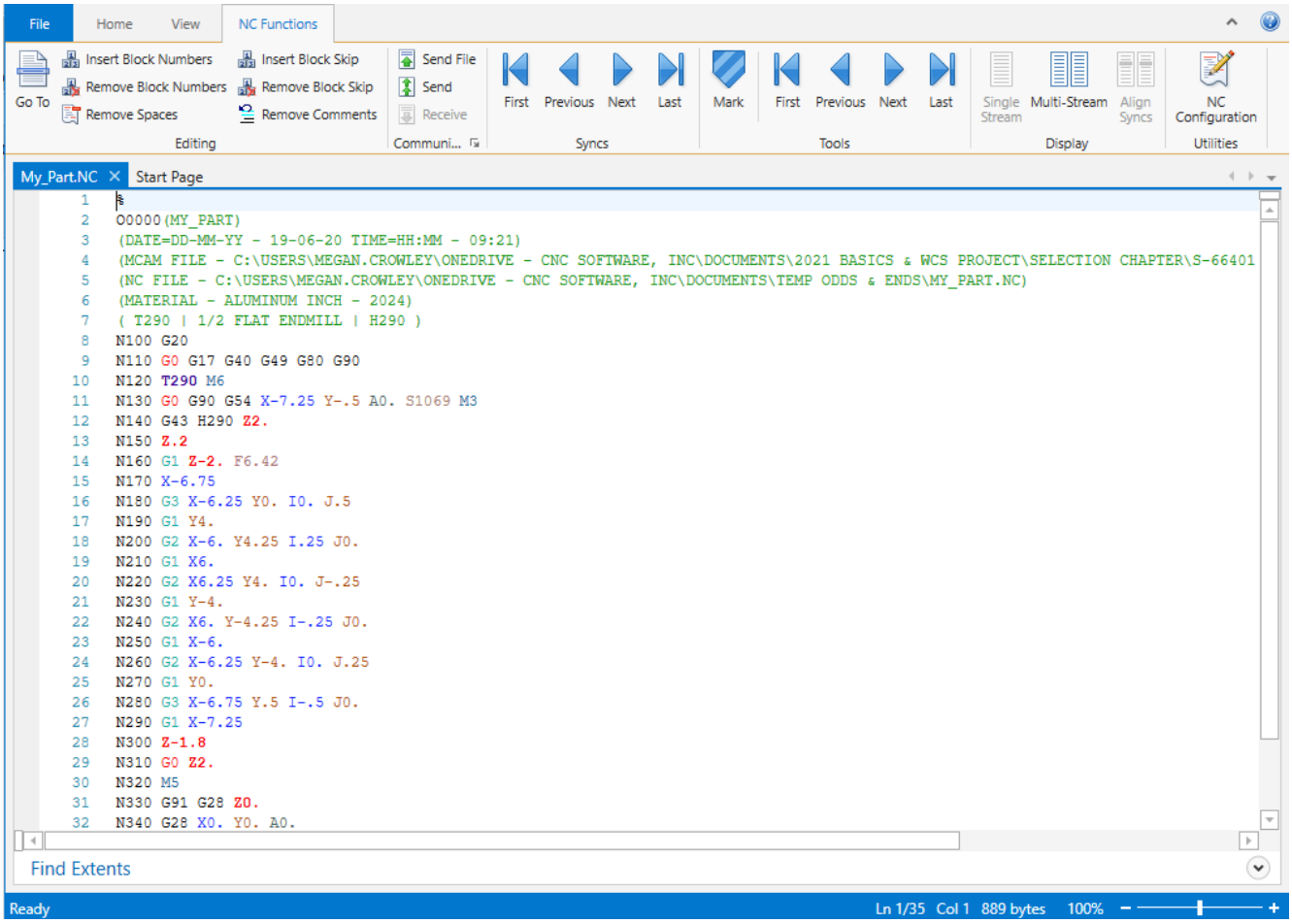
The **Post processing** dialog box displays. The name of the post processor for the machine you have chosen displays as the **Active post** field in read-only mode.



3. Click **OK**. The **Save As** dialog box opens.
4. Name your NC file and click **Save**.



The file opens in Mastercam Code Expert.



The NC code specifies information such as stops, cutting coordinates, and tool changes. Block numbers appear in front of each command. Code Expert allows you to sync operations, manually edit code, and switch NC Configurations.

5. Exit Code Expert.
6. Save your part.

CONCLUSION

Congratulations! You have completed the *Mastercam Basics Tutorial*! Now that you have mastered the skills in this tutorial, explore Mastercam's other features and functions.

You may be interested in other tutorials that we offer. Mastercam tutorials are being constantly developed, and we will add more as we complete them. Visit our website, or select **Help**, **Tutorials** from the **File** tab.

Mastercam Resources

Enhance your Mastercam 2023 experience by using the following resources:

- *Mastercam Documentation*—Mastercam installs a number of helpful documents for your version of software in the Documentation folder of your Mastercam installation.
- *Mastercam Help*—Access Mastercam Help by selecting **Help**, **Contents** from Mastercam's **File** tab or by pressing [**Alt+H**] on your keyboard.
- *Mastercam Reseller*—Your local Mastercam Reseller can help with most questions about Mastercam.
- *Technical Support*—Our Technical Support department (+1 860-875-5006 or support@mastercam.com) is open Monday through Friday from 8:00 a.m. to 5:30 p.m. USA Eastern Standard Time.
- *Mastercam Tutorials*—We offer a series of tutorials to help registered users become familiar with basic Mastercam features and functions. Visit our website, or select **Help**, **Tutorials** from Mastercam's **File** tab to see the latest publications.
- *Mastercam University*—Mastercam University, an affordable online learning platform, gives you 24/7 access to Mastercam training materials. Take advantage of our many videos to master skills at your own pace and help prepare for Mastercam Certification. For more information on Mastercam University, please contact your Authorized Mastercam Reseller, visit university.mastercam.com/, or email training@mastercam.com.
- *Online Communities*—You can find a wealth of information at www.mastercam.com.
    - Follow us on Facebook (www.facebook.com/Mastercam), Twitter (twitter.com/Mastercam), and Instagram (https://www.instagram.com/mastercam/) for the latest tech tips and Mastercam news.
    - See Mastercam in action on YouTube (www.youtube.com/user/MastercamCadCam).
    - For more information on CNC Software, LLC, to find and apply to jobs, and connect with people using Mastercam, visit us on LinkedIn (www.linkedin.com/company/cnc-software/).
    - Registered users can search for information or ask questions on the Mastercam Web forum, forum.mastercam.com, or use the Mastercam Knowledge Base at kb.mastercam.com.

Contact Us

For questions about this or other Mastercam documentation, contact the Technical Documentation Department by email at techdocs@mastercam.com.

Attention! Updates may be available. Please refer to my.mastercam.com/product-downloads for the latest downloads.

CNC Software, LLC
671 Old Post Road
Tolland, CT 06084 USA

www.mastercam.com

Mastercam® is a registered trademark of CNC Software, LLC. All other trademarks are property of their respective owners.
©1983-2022. All rights reserved.