

MFG464722

Adding Machining Operations to Your Inventor Designs Using Inventor CAM

Jennifer MacMillan
ASCENT, Center for Technical Knowledge

Learning Objectives

- Create the machining Setup for an Inventor CAM model.
- Create toolpaths in an Inventor CAM model
- Simulate created toolpaths in an Inventor CAM model.
- Generate the CNC code required to machine a model.

Description

This course focuses on instructing new users on how to use the Inventor CAM add-on to create milling toolpaths. You will learn the key skills and knowledge required to take a 3D model, set it up in the CAM environment, and assign the toolpaths needed to generate the CNC code required by milling machines. The demonstration in this class will focus on a simple design that requires 2D Milling toolpaths. This will allow us to keep the focus on the workflow and learning how to create the toolpaths. Specifically, the Face, 2D Adaptive, 2D Pocket, 2D Contour, and Drill toolpaths will be generated for the demonstration model. To complete the course, you will also learn how to use the Simulation option to visualize the new toolpaths and how to output the CNC code required to machine a model.

Speaker

Jennifer MacMillan holds a mechanical engineering degree from Dalhousie University, Nova Scotia, Canada. For the past 25 years, Jennifer has been teaching and developing CAD training material. She is currently working with the Rand Worldwide company, ASCENT-Center for Technical Knowledge, where she develops and manages the training curriculum used by Autodesk Authorized Training Centers. She also has her Autodesk Certified Instructor (ACI) certification. Jennifer has always enjoyed teaching and passing her knowledge of and interest in CAD software on to her students. This is Jennifer's 11th year attending Autodesk University and her 6th time speaking at Autodesk University.



jmacmillan@rand.com



[@ASCENT_CTK](https://twitter.com/ASCENT_CTK)



ASCENTed.com (Blog)

This handout is provided as a supplement to the Autodesk University 2020 class *Adding Machining Operations to Your Inventor Designs Using Inventor CAM (MFG464722)*. To get the full benefit out of this handout, consider also watching my recorded session on the Autodesk University website. This handout is broken into five main sections. Four of which align directly with the course objectives. The first section is provided as an introduction to the Inventor CAM interface. My course and this handout focus on Milling; however, the Inventor CAM interface and layout will be similar for Turning.

Inventor CAM Interface

Once Autodesk Inventor CAM is installed, the CAM tab is available in the ribbon. The options on the ribbon enable you to add operations to your Inventor models. The following discusses this tab along with other unique interface items that you should understand before getting started.

CAM Tab

The CAM tab provides access to all of the tools that you will need to set up your manufacturing model, define and simulate your toolpaths, and post process the data to generate the required NC code for your CNC machines. Figure 1 shows the options that are available when the CAM tab is selected. The options displayed are for the Inventor CAM Ultimate add-on. The panels that are displayed on this ribbon vary depending on the CAM add-on package that is installed.



Figure 1

CAM Browser

The CAM browser is where the CAM operations are listed and organized according to each defined setup. This browser is only available once a setup has been created in the model. Figure 2 shows a CAM browser that has two setups, with five toolpaths in each.

*If the CAM browser is not displayed once a setup has been assigned, click **+** to expand the list of available tabs and select CAM.*

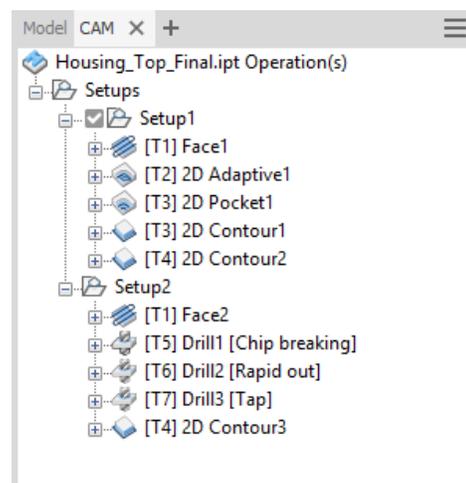


Figure 2

Consider the following when reviewing the CAM browser:

- The active setup is identified with the icon adjacent to the setup name. All new toolpaths that are assigned get added automatically to the setup that is identified as the default folder. To assign the default folder, right-click on the setup name and select **Default Folder**. When a new setup is created, it is automatically set as the default folder.
- Folders can be created in a setup to help group toolpaths.
- Toolpaths can be dragged and dropped between setups and folders, if required.
- Expand a toolpath node to display information on its tool, WCS, and size, as shown in Figure 3.

*To obtain a list of feeds, speeds, movement types, and coordinates, right-click on the toolpath size value and select **View Toolpath**. This can be used as an alternative to playing a simulation.*

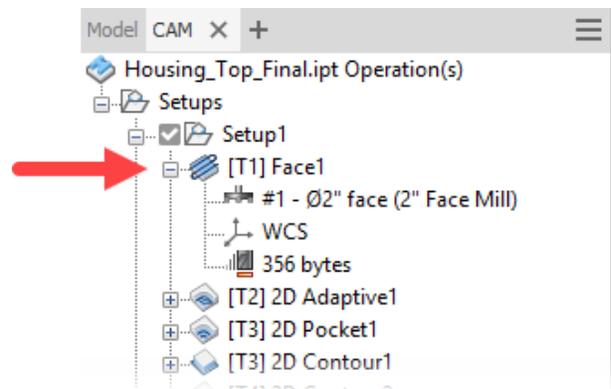


Figure 3

- Contextual right-click menus are available in the CAM browser by selecting on the file's Operation(s) node (top of the list), the setup nodes, or the toolpaths.
- Double-clicking any node name will open the palette used to create it. You can edit the options within the palette to modify the item.
- All node names (setup or toolpath) can be renamed. Single-click to select it and then slowly select it a second time to rename it. Consider using names that will help identify or describe the item.

Palettes

Inventor CAM uses a palette interface for machining setups, creating toolpaths, and running simulations. These palettes are shown in Figure 4. Each of these palettes has different tabs that are used to define the options available.

Depending on the width of your palette and the number of tabs, the tab name may or may not be displayed.

In the Toolpath palette, there are five tabs so only the icons are displayed at this width.

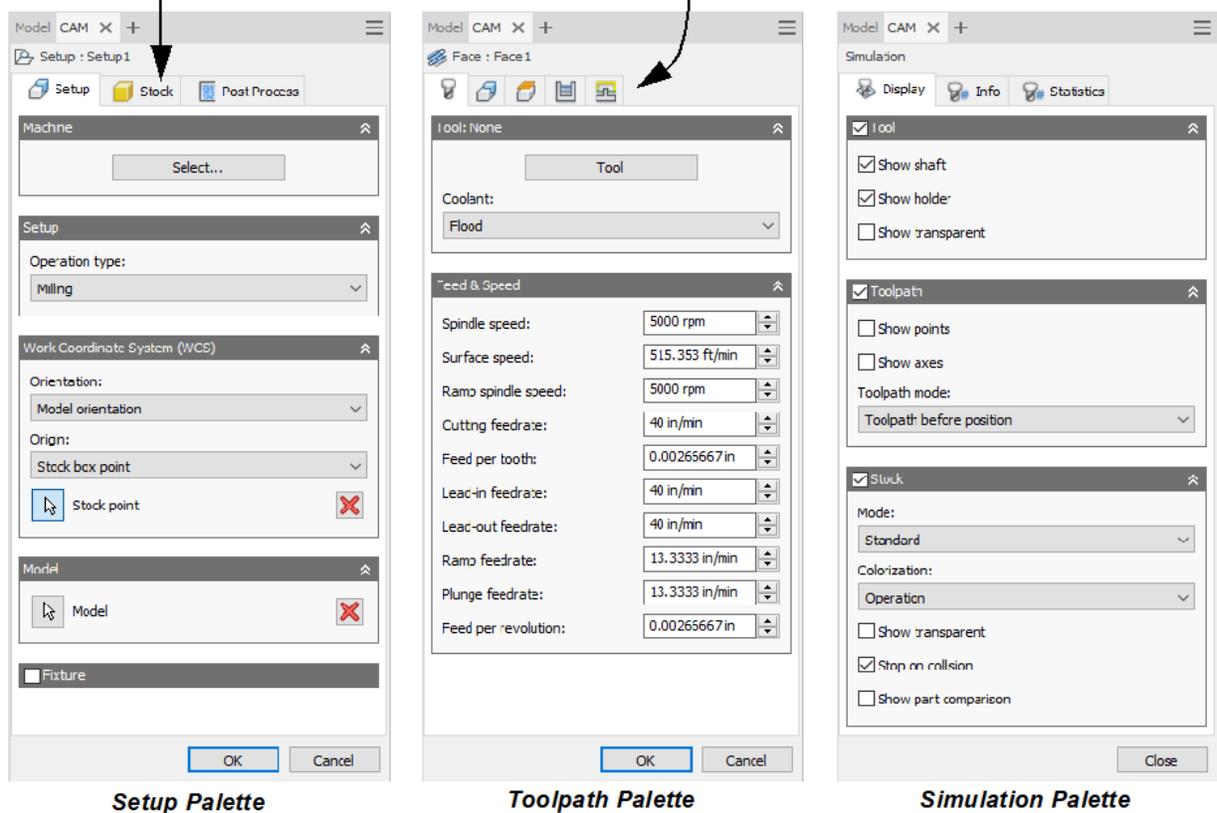


Figure 4

Graphics Window

The graphics window is the main working area of the model. In addition to displaying the 3D model, it displays the stock, work coordinate system (WCS), and toolpath movements that have been created in the model.

- To display the stock and WCS for a setup, select the setup name in the CAM browser, as shown in Figure 5. This allows you to quickly preview the setup and verify that the stock and WCS have been assigned properly.

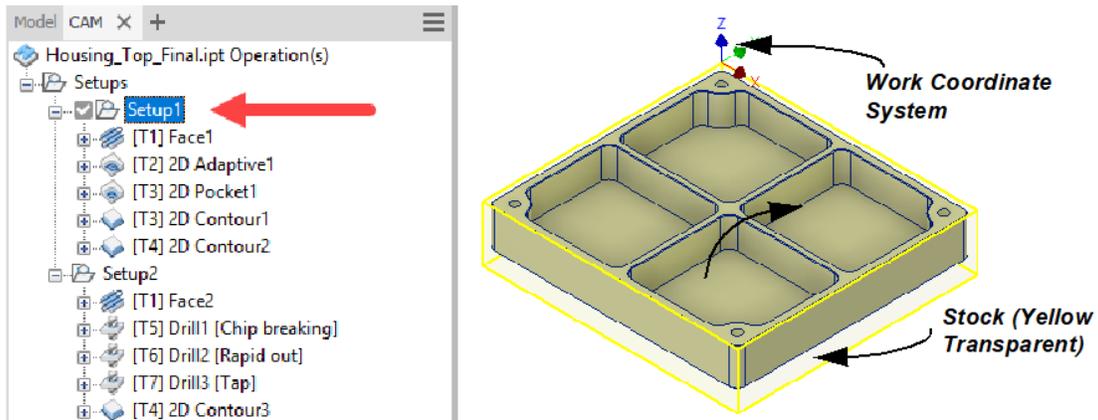


Figure 5

- To display a toolpath in the graphics window, select the toolpath name in the CAM browser. The toolpath movements are color coded as shown in Figure 6.

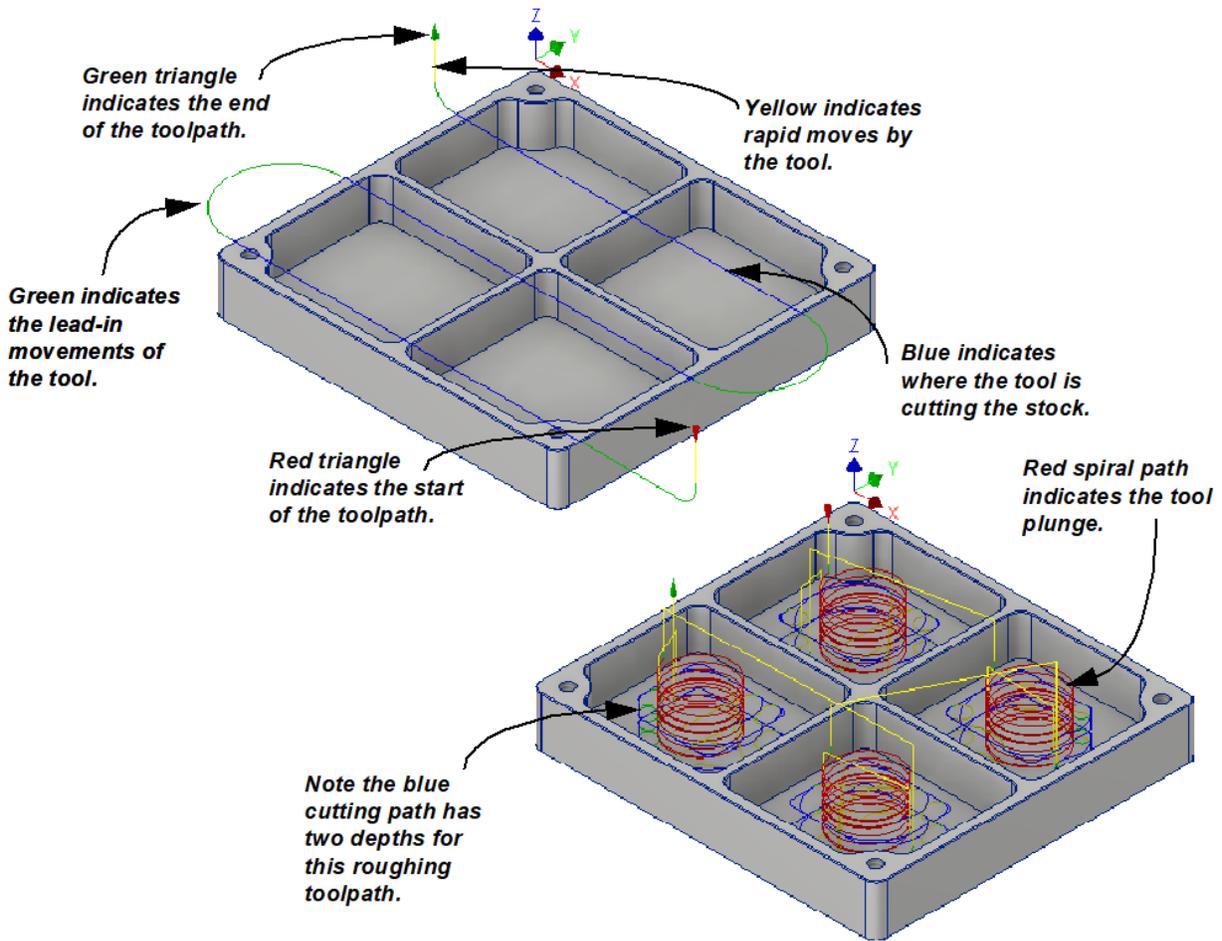


Figure 6

Tool Library

A default tool library is available with Inventor CAM. This library provides access to a large database of tools that you can use. Although tools are assigned during toolpath creation, you can also access the Tool Library directly to create and manage your own custom tools. In either situation, the Tool Library functions the same once it is opened. To open the Tool Library, in the

CAM tab>Manage panel, click  (Tool Library). Figure 7 shows the Tool library dialog box and the tools that have been defined for the Housing_Top_Final open document (model).

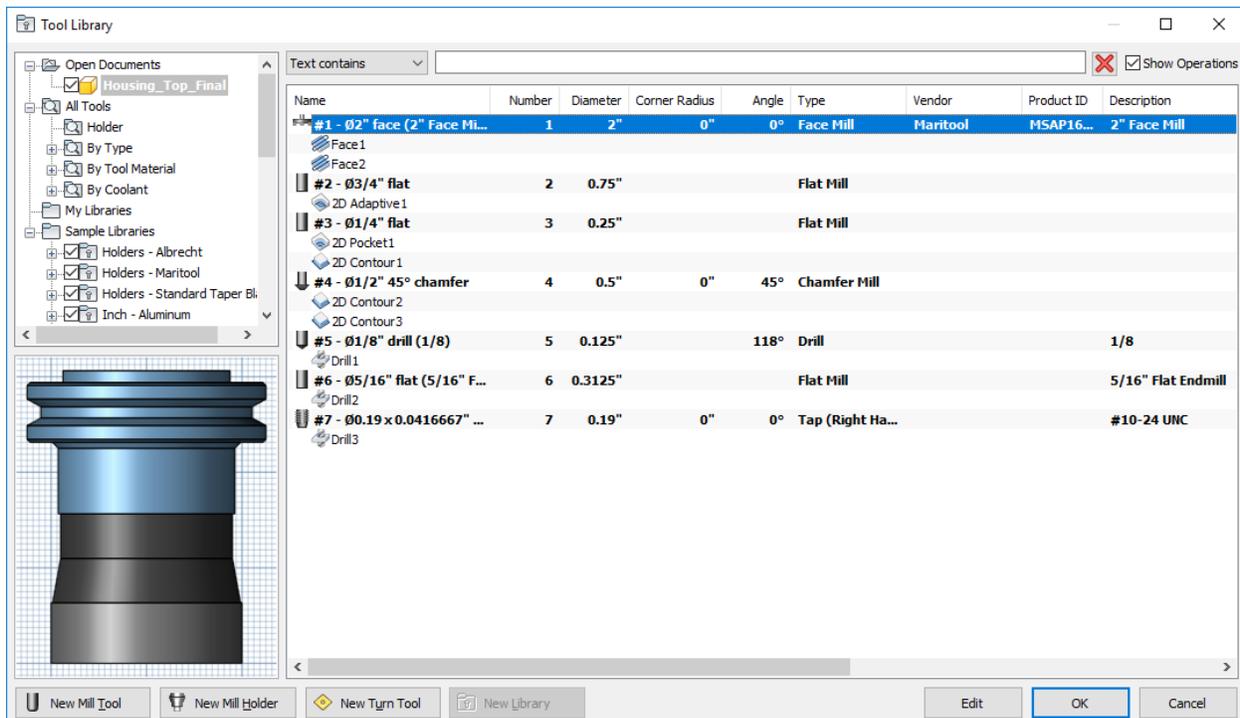


Figure 7

Consider the following regarding the Tool Library dialog box:

- Open models are listed in the Open Documents node. For models that already have tools assigned, the Tool Library appears similar to that shown in Figure 7; otherwise, the right-hand pane will be empty until a tool is assigned.
- The default tools are organized by category in the top-left pane. The All Tools and Sample Libraries categories list the default tools. The My Libraries category lists any custom tool libraries that you create.
- A preview of the selected tool is displayed in the lower-left pane. This provides a visual preview as to the shape and size of the tool,

- To locate a tool, you can navigate through the category folder structures or you can use the filter and sorting tools that are available in the dialog box to refine the list.
 - Select a search option from the drop-down list to best describe the tool you are looking for. Enter the supporting text in the search field adjacent to the list.
 - Select a column header to sort in ascending/descending order.
 - Drag and drop the column headers to reorder the display order of the columns. This can help you more easily see the columns that are important to you.
 - In the Sample Libraries and My Libraries categories, you can clear the activation boxes adjacent to the tool folders to clear them from the search.

Custom tools are stored in custom libraries in the **My Libraries** category. To populate a custom library, you can copy from the provided standard library or create new tools. To create a library, right-click on the **My Libraries** category and select **New Library** or click **New Library** at the bottom of the dialog box. Enter a descriptive name to accurately describe the new custom library.

How to reuse an Existing Standard Tool

The default tools can be used as a reference in creating new tools. This simplifies the creation process because you can locate a similar tool, copy it to your custom library, and edit it as needed. To copy a tool, right-click on the parent tool that will be copied, select **Copy**, then navigate to the custom library folder and right-click in the list and select **Paste**. To edit the tool once copied, select the tool name and click **Edit**.

How to create a New Tool

Use the following steps to create a new custom tool.

1. Select a custom library name in the My Libraries category.
2. Select a new tool option from the bottom of the dialog box. The options enable you to create a new milling tool, mill holder, and turning tool.
3. The Tool dialog box opens, enabling you to define the tool. There are six tabs available for you to define the tool. These include general parameters, as well as cutter, shaft, holder, and feed and speed parameters for the new tool.
4. Click **OK** to complete the tool and add it to the custom library.

Accessing the Tool Library through its option on the Manage tab is used to manage and create tools. All tool assignment is done during toolpath creation. At that time, the same Tool Library dialog box is accessed. All functionality discussed in this topic can also be used while assigning tools to a toolpath.

Defining the Machine Setup

The first step in Inventor CAM, prior to creating any machining toolpaths, is to define the setup for the operations that will machine the model. Multiple setups can exist in each model. The setup defines the following:

- Operation type being used to machine the model
- Work coordinate system of the model
- Stock that the model will be machined from
- Post process settings

To create a setup, in the CAM tab>Job panel, click  (Setup). The Setup palette and model display. The yellow box around the model identifies the default stock. The Setup palette has three tabs that are used to fully define the setup.

Setup Tab

The  (Setup) tab is the first tab in the palette. It enables you to define the machine, operation type, work coordinate system (WCS), model, and whether a fixture is to be assigned. The options are shown in Figure 8.

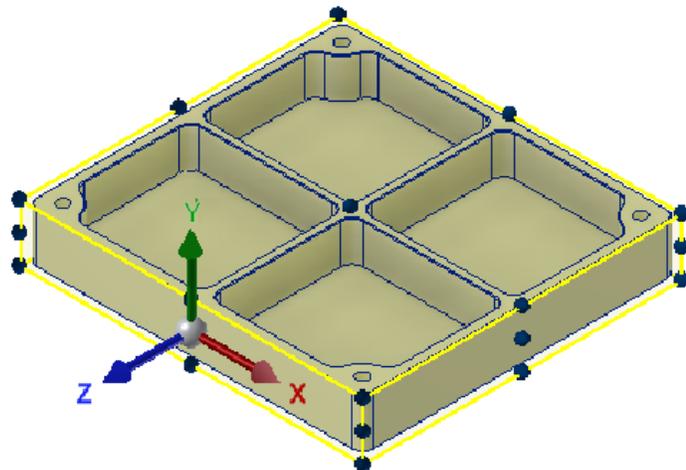
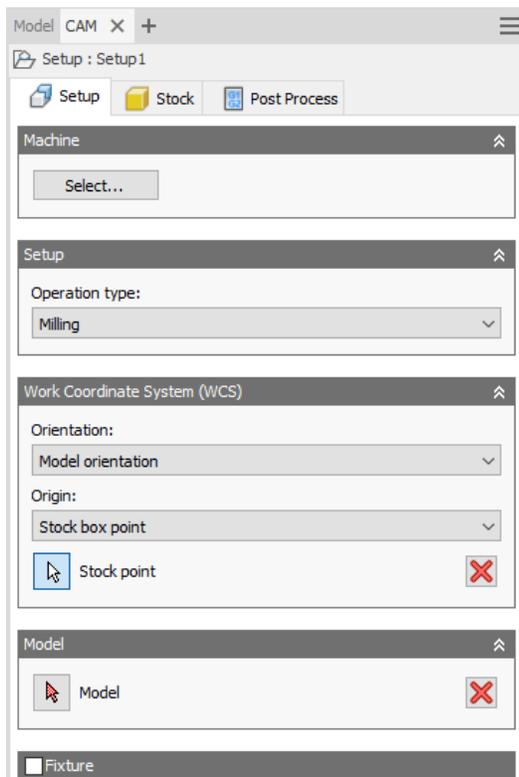


Figure 8

- The Machine area of the palette enables you to optionally define the type of machine that will be used. The available types include milling, turning, and cutting machines. Select from the list, or if your machine is not available, select a type that closely matches your machine. Click **OK** to assign the machine to the setup. Depending on the machine chosen, it will help pre-populate some of the parameters required.
- Select the Operation type option from the drop-down list. The options include **Milling**, **Turning or mill/turn**, or **Cutting**.
- The Work Coordinate System (WCS) area of the setup is used to set the orientation for the cutting plane of the machine. By default, the WCS is displayed in the same orientation as the model's coordinate system. For milling machines, the Z-axis should point perpendicular to the surface being machined. For turning, it should be oriented along the centerline of the model.
 - The Orientation drop-down list enables you to define a new orientation for the WCS. You are only required to reorient the WCS if it does not match the orientation required for your machine. Select an option and select edges or vertices in the model to define the WCS's orientation in the X-, Y-, and Z-axis. Use the **Flip** settings, as needed, to flip the orientation along an axis.
 - The Origin drop-down list enables you to define where the WCS is located on the stock. The options enable you to define the WCS on the model origin, at a selected vertex or edge, or on a stock or model point.
- The Model field enables you to select the model being machined. By default, the entire design model is assigned. If the design model is an assembly or multi-body design, where a portion of the model is representing non-machined geometry, use this field to refine the selection. Click  to clear the selection and select the geometry to be machined. If the design model is a single part file that represents the model being machined, no changes are required.
- The Fixture area at the bottom of the Setup tab can be used to assign geometry from the design model as belonging to the machine fixture or its attachment. By default, this field is disabled, but it can be enabled if required. In general, this would be used only if an assembly design model is being used and you want portions of the assembly recognized as fixturing geometry to check toolpaths for collisions.

Setup Tab

The toolpaths that are generated are based on the stock that is being removed; therefore, proper stock definition is a key step in setting up your operation. By default, a 0.04 in. (1 mm) side and top offset stock value is assigned. These values can be modified to match your actual stock by selecting the  (Stock) tab at the top of the Setup palette. The Stock tab is shown in Figure 9.

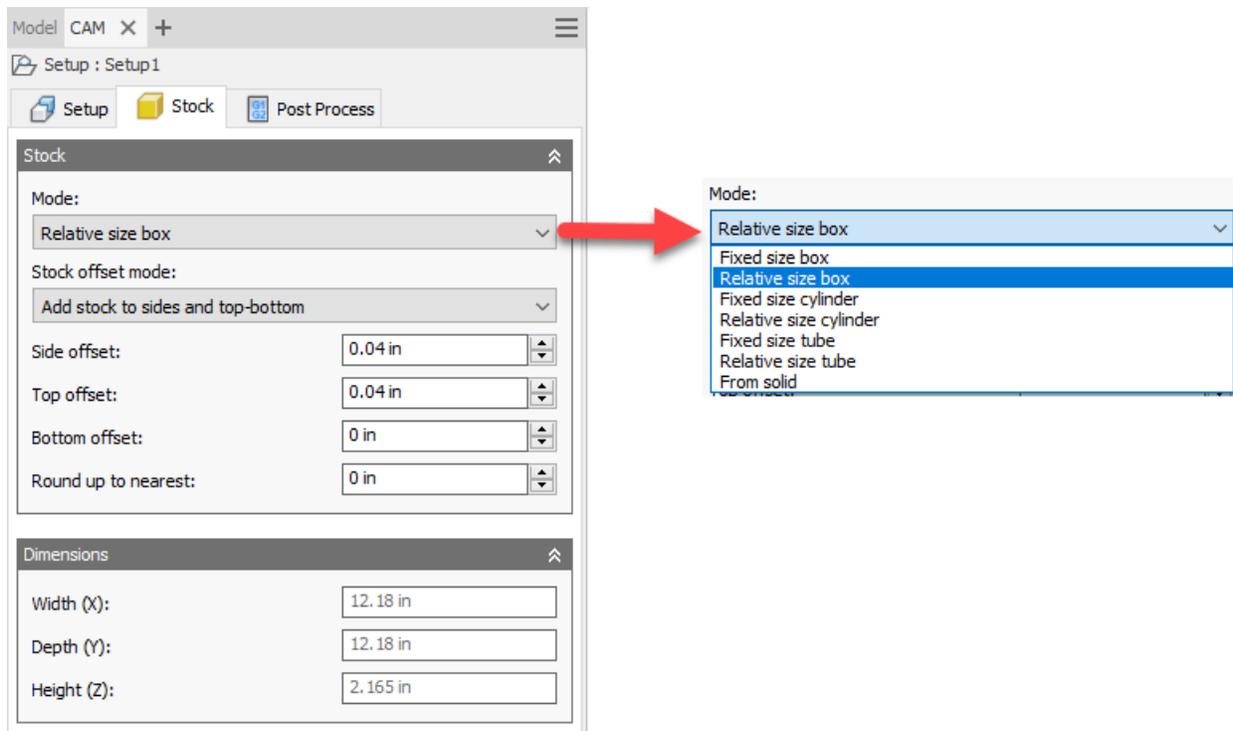


Figure 9

- The Mode options enable you to define the shape of the stock as either box, cylinder, tube, or a selected solid body from the model. In the case of the box, cylinder, and tube stock shapes, you can define the size as fixed or relative. For fixed, you define the exact size in the x, y, and z directions, whereas for relative you define the size by entering offset values that are measured from the model geometry. The Stock offset mode options can also be used to refine the offset fields if the stock is being created relative to the model.
- The Dimensions area of the Stock tab is a non-editable field that simply provides the overall size of the stock being used. As changes are made, these values automatically update.

Ensure that the stock size you use considers operations that may be required on the top, bottom, and sides.

Post Process Tab

The final tab in the Setup palette is the  (Post Process) tab. This tab enables you to define the post processing parameters for the machine. It includes the program number and comments, as well as the WCS offset. Post processing will be discussed in more detail later in this handout.

To complete the setup, click **OK** at the bottom of the Setup palette. The Setup appears listed in the CAM browser as shown in Figure 10. It is recommended to rename each setup to help identify its intent. To rename, slowly select <Setup#> twice and enter a name.

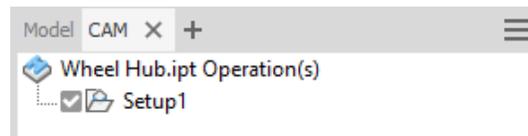


Figure 10

Creating Toolpaths in an Inventor CAM Model

In general, 2D toolpaths are used to machine models with vertical and horizontal surfaces. In 2D Milling, the tool moves in the Z axis, cutting is done parallel to the XY plane and is feature based. 3D Milling allows for machining models with curvature (i.e., contoured surfaces), and therefore reference surfaces, faces, and edges on the model. Examples are shown in Figure 11.

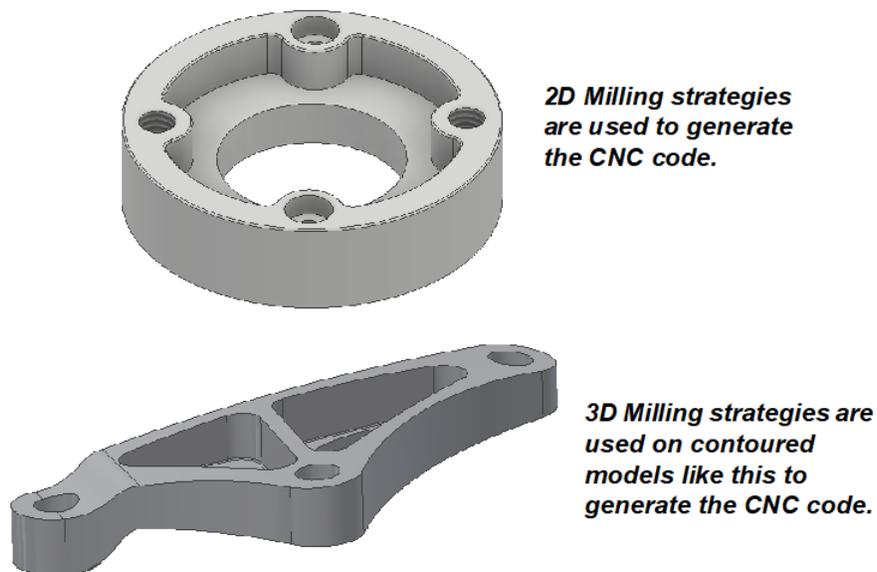


Figure 11

In milling contoured surfaces, specific roughing and finishing toolpaths exist, as compared to in 2D milling where there are fewer toolpaths and the same type of toolpath can be used for both roughing and finishing. In general, for 3D milling the majority of the stock is removed with a roughing toolpath which leaves a poorer quality stepped surface. To finish the surface, more refined toolpaths are used that remove less material while leaving a higher quality contoured surface. It is common that multiple toolpaths with smaller and smaller tools may be required to mill the desired surface finish.

Using Inventor CAM you can assign 2D Milling, 3D Milling, Drilling, Turning, and Multi-Axis Milling toolpaths to your models. The expanded 2D and 3D Milling menu options are shown in Figure 12. Turning and Multi-Axis Milling toolpaths are not discussed in this handout.

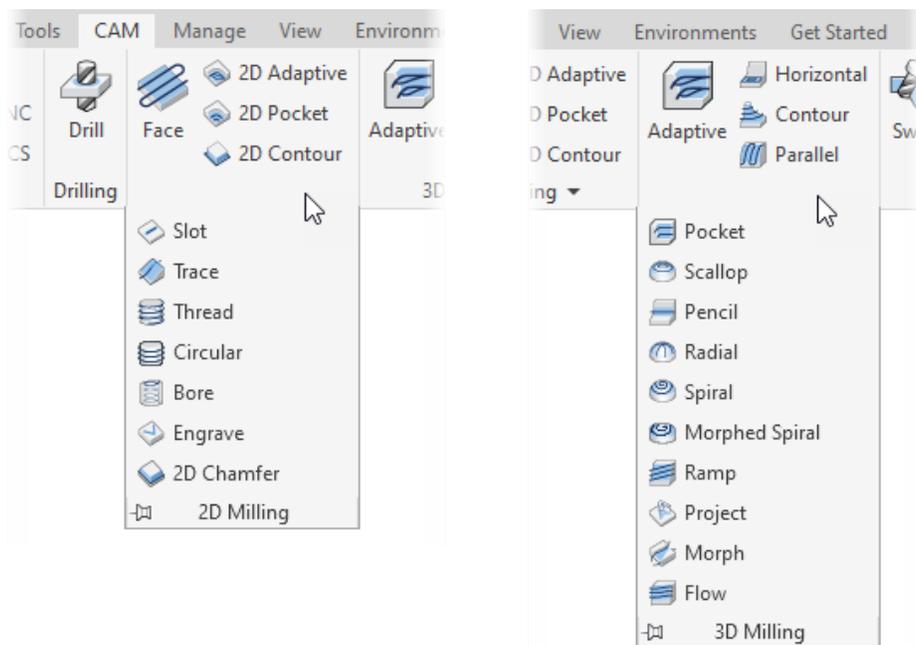


Figure 12

The interface for all toolpaths involves the use of the toolpath palette. The palette contains multiple tabs that provide access to all the options/parameters to define the specific toolpath. Many options are the same; however, they do vary depending on the toolpath that is selected for creation.

*In general, the default settings for the toolpaths may produce an acceptable result. When starting the creation of a new toolpath, it is recommended that you begin by defining the tool and geometry, review the heights, and then click **OK**. Simulate the result and make the required changes. Consider changing only one parameter at a time to refine the path.*

The workflow to add a toolpath to your model is explained as follows for the Facing toolpath; however, is similar for all toolpaths. The default values assigned in the fields are based on the tool and type of operation. To create a toolpath, begin by selecting the toolpath type in the CAM ribbon. The <toolpath> palette opens as shown in Figure 13.

1. In the  (Tool) tab, select **Tool** to access the Tool Library to assign the tool. Locate and select the required tool for the toolpath and click **Select**. Once the tool is selected, it is added to the list of tools in the Open Documents section of the Tool Library.

 - Once the tool is selected, the parameters update in the <toolpath> palette to that of the selected tool. Changes to the tool parameters can be made in the palette or you can edit the tool in the Tool Library dialog box. Changes made are listed for the toolpath and will not update the parent tool.

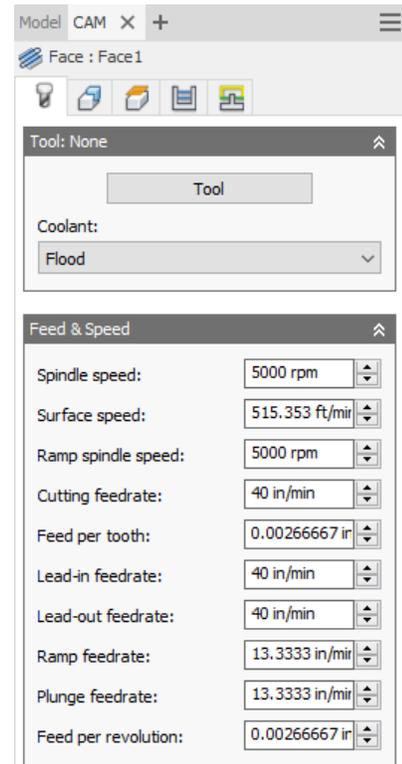


Figure 13

2. Select the  (Geometry) tab on the palette. The Face palette displays as shown in Figure 14.

 - By default, the stock's outside boundary is preselected (yellow bounding box). If the boundary for the toolpath is not displayed as required, you can select  (Stock selections) and select edges or a sketch to define the area. Geometry selection options will vary for the difference toolpaths.
 - The top of the model is based on how the WCS was set up. Enable the Tool Orientation option to change the face that is being cut. The orientation options are the same as those used to define the WCS orientation during setup.

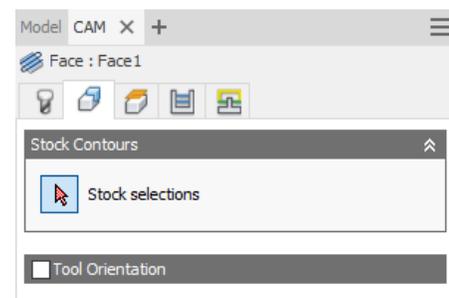
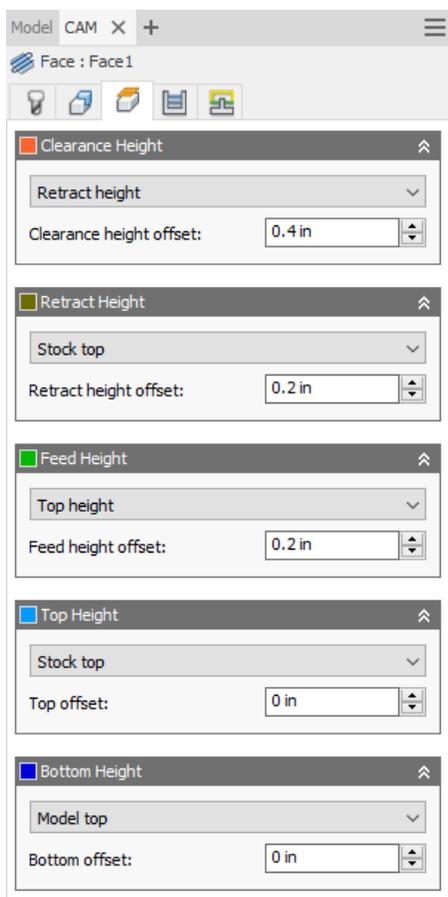
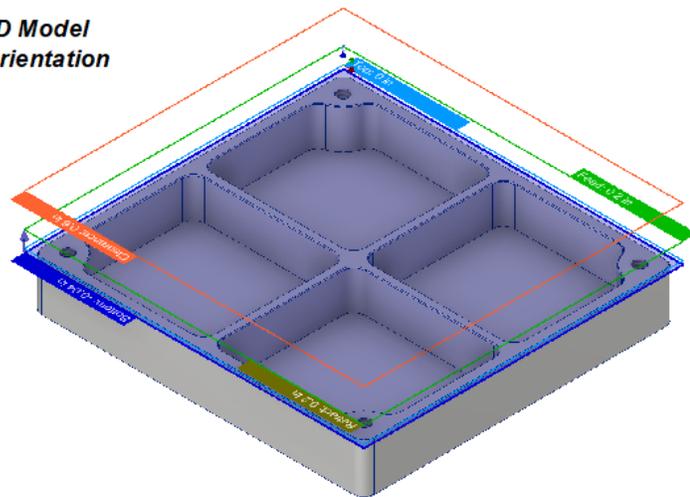


Figure 14

3. Select the  (Heights) tab. The planes available on this tab enable you to define the required heights for the tool for clearance, retract, etc. (as shown in Figure 15). Each plane is color coded and gives you a visual representation of the planes. Edit any of the values, as needed, directly in the palette or in the mini-toolbar.
- The red plane indicates the Clearance Height for the tool.
 - The Retract Height and Feed Height (different shades of green) are at the same height in this model. The Retract Height sets the height that the tool will move to before a pass. The Feed Height sets the height that the tool rapids to before changing to the cutting rates defined to cut the stock.
 - The Top Height is set at the top of the stock. This is where cutting begins.
 - Because facing is a finishing pass, by default, the Bottom Height is set as the Model top with no offset value.



3D Model Orientation



2D Model Orientation



The mini-toolbar can be used as an alternative to entering values in the Face palette. Select a height option from the drop-down list to toggle between the various settings.

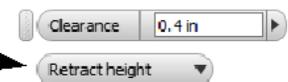


Figure 15

4. Select the  (Passes) tab. The options on this tab set how the toolpath passes are defined, in terms of tolerance, pass direction, extensions, offsets, and stepovers (Figure 16).
 - The Multiple Depths area is not enabled by default for Facing toolpath. This indicates that the stock depth is being removed in a single pass. If a depth is to remain after the facing toolpath, this option can be enabled and values set.
 - The Stock to Leave area is not enabled by default. If enabled, you can set values for stock that will remain for future toolpath operations.

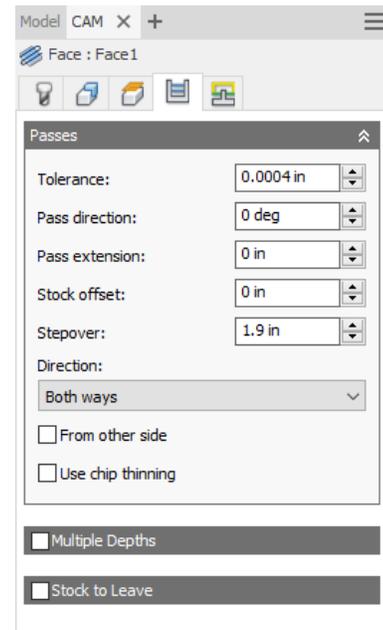


Figure 16

5. Select the  (Linking) tab. This tab's options (Figure 17) define how each toolpath pass should be linked to others and how many leads and transitions will be generated.
 - In the Linking area, the High feedrate mode drop-down list provides multiple options to determine if rapid movements will be output as G0 (true rapids) or G1 (high feedrate movements). Additional options in this area enable you to further customize how the path is generated.
 - The Leads & Transitions area contains options to define if the leads and transitions are to be generated in the final toolpath and their values, as well as define a transition type.

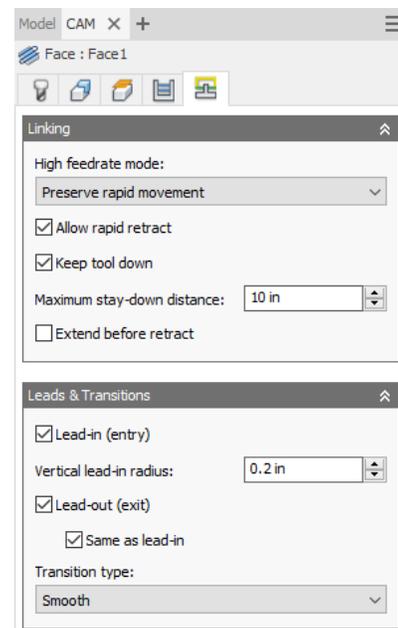


Figure 17

6. Click **OK** at the bottom of the palette to complete the toolpath. Once the toolpath is completed, it is added to the active setup and is listed in the CAM browser, as shown in Figure 18.

- The icon adjacent to the toolpath name indicates the type of toolpath.
- The [T#] entry indicates the tool number.
- Expanding the toolpath node reveals more details about the toolpath, as shown in Figure 18, including the tool name and size, WCS, and toolpath size.
- To rename a toolpath, slowly select it twice and enter a new name. Once renamed, the tool number remains as the prefix in the node to easily identify the tool that is used.

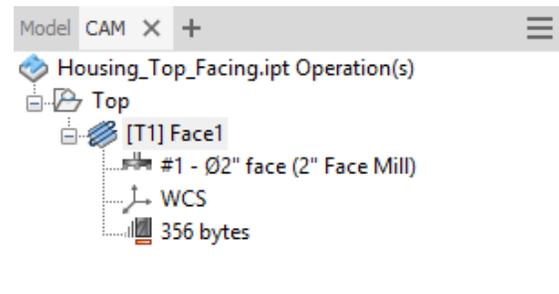


Figure 18

The tabs available when creating a drilling toolpath are slightly different. The Cycle tab replaces the Passes tab. Use this tab to define the type of drilling cycle that is to be generated, as shown in Figure 19. Depending on the option, additional parameters may or may not become available.

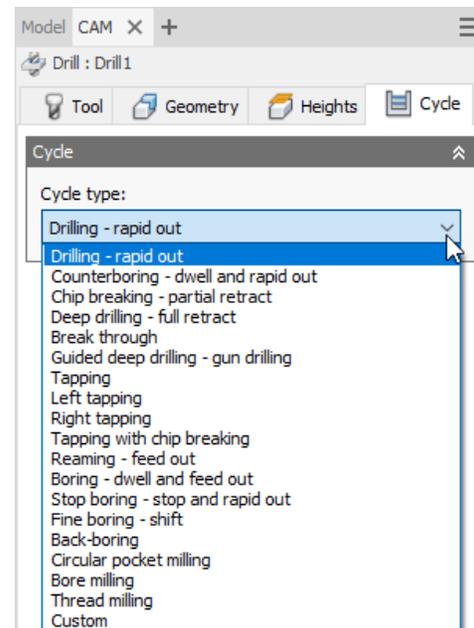


Figure 19

Inventor CAM has valuable help tooltips that can help you understand each of the options and parameters. Hovering your cursor over the fields provides access to descriptive help tooltips, similar to that shown in Figure 20. This can be done on all tabs in the palette to help learn more about how the option or parameter can be used to control the cutting path.

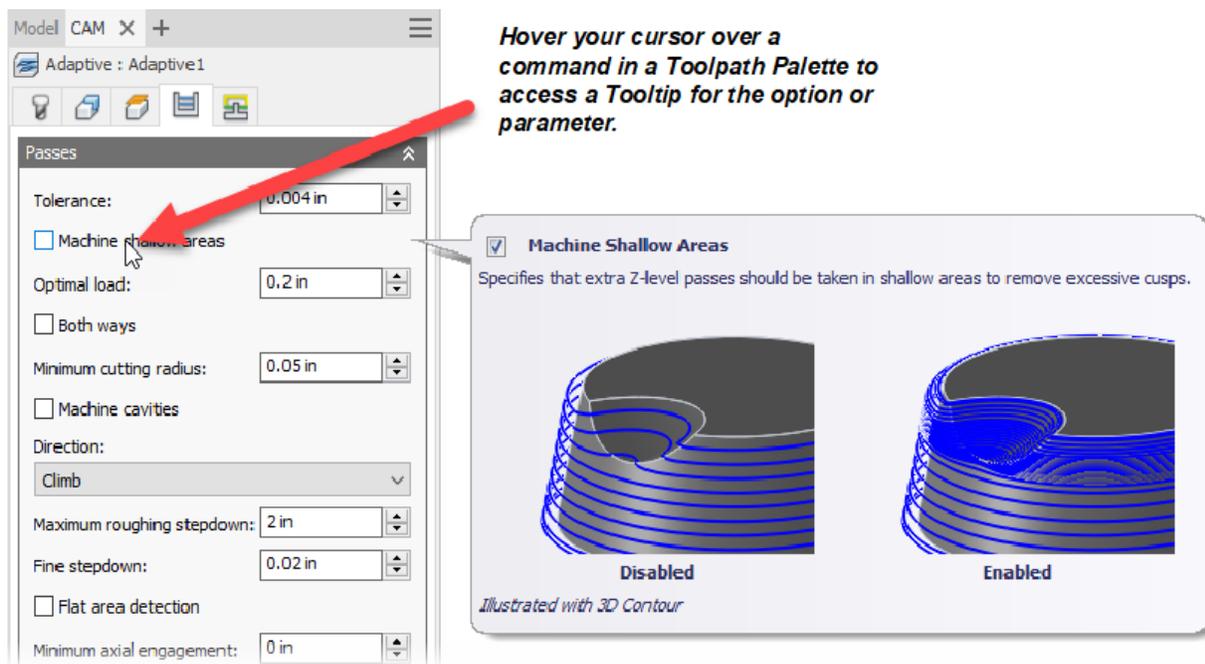


Figure 20

Simulating Inventor CAM Toolpaths

The Simulate option enables you to visually preview toolpaths in the graphics window to help confirm that they have been created properly. This section discusses the options in this environment which can control the display as well as the playback and speed of the simulation. A simulation can be run on any of the following:

- **Operation** - Select the file's Operation(s) node at the top of the CAM browser to simulate the entire operation.
- **Setup** - Select a setup node in the CAM browser to simulate all the toolpaths in the selected setup.
- **Toolpath** - Select a toolpath node in the CAM browser to simulate the single selected toolpath. Multiple toolpaths can be simulated at the same time by pressing and holding <Ctrl> prior to activating the simulation option.

To enable this environment, in the CAM tab>Toolpath panel, click  (Simulate). Once active, the Simulation Palette, Player, and Timeline appear.

Simulation Palette

The Simulation palette opens as shown in Figure 21. The  (Display) tab is active by default. This tab controls the display of the Tool, Toolpath, and Stock in the graphics window during the simulation.

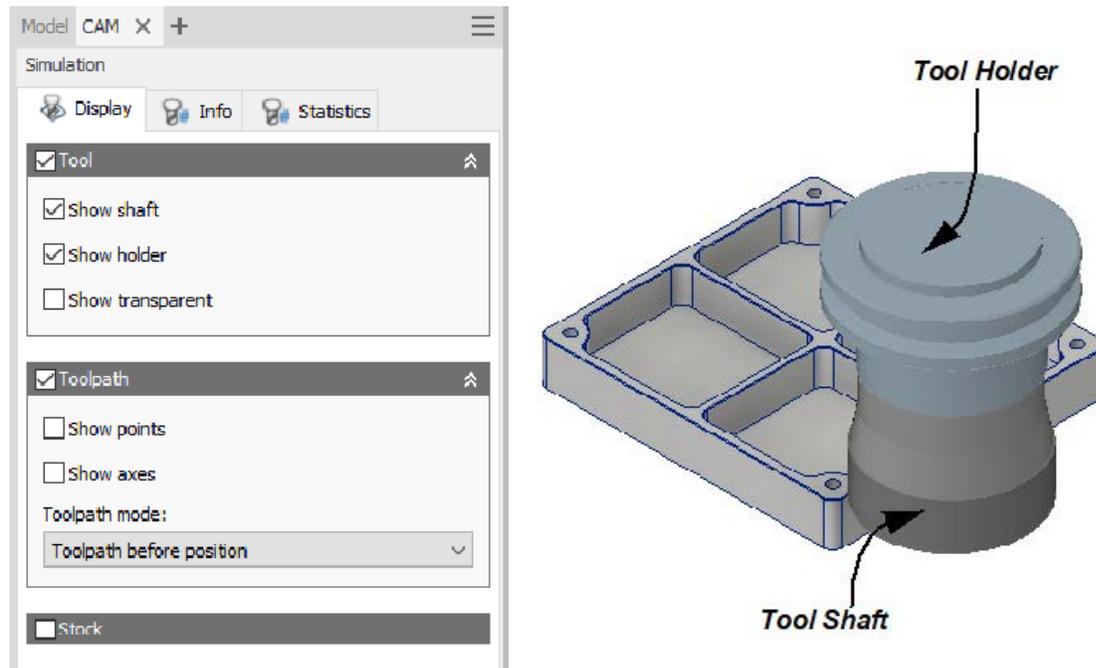


Figure 21

The Display tab is where you customize whether the tool, toolpath, and stock are displayed during the simulation. Enable each respective area and its options as required.

- By default, the Tool is set to display in a simulation. The shaft and holder for the tool are displayed as solid; however, the **Show Transparent** option can also be enabled to help with visualization. The tool's shaft and holder can be toggled on and off as required.
- Toolpaths can be displayed to help visualize the path of the tool. They display using the same color scheme as if you selected a toolpath in the CAM browser. The Toolpath mode drop-down list can be used to control the path's display before, during, and after the tool traces the path.
- The Stock can be enabled so that it is displayed during simulation. If enabled, additional stock options are available (i.e. to control color and transparency).

*The **Stop on collision** option enables you to test that the selected toolpaths do not produce any collision between the shaft, holder, and fixture. If a collision occurs, playback stops.*

The  (Info) tab provides general information about the position, operation type, and verification data (as shown in Figure 22). The specific values depend on where the playback marker is located and thus the values constantly change during the simulation.

The  (Statistics) tab provides you statistics on the completed simulation. For example, to display the overall statistics for multiple setups that contain multiple toolpaths, you must have preselected the top-level operation to be simulated. The statistics that are included are the machining time, distance, number of selected toolpaths being simulated, and number of tool changes required, similar to that shown in Figure 22.

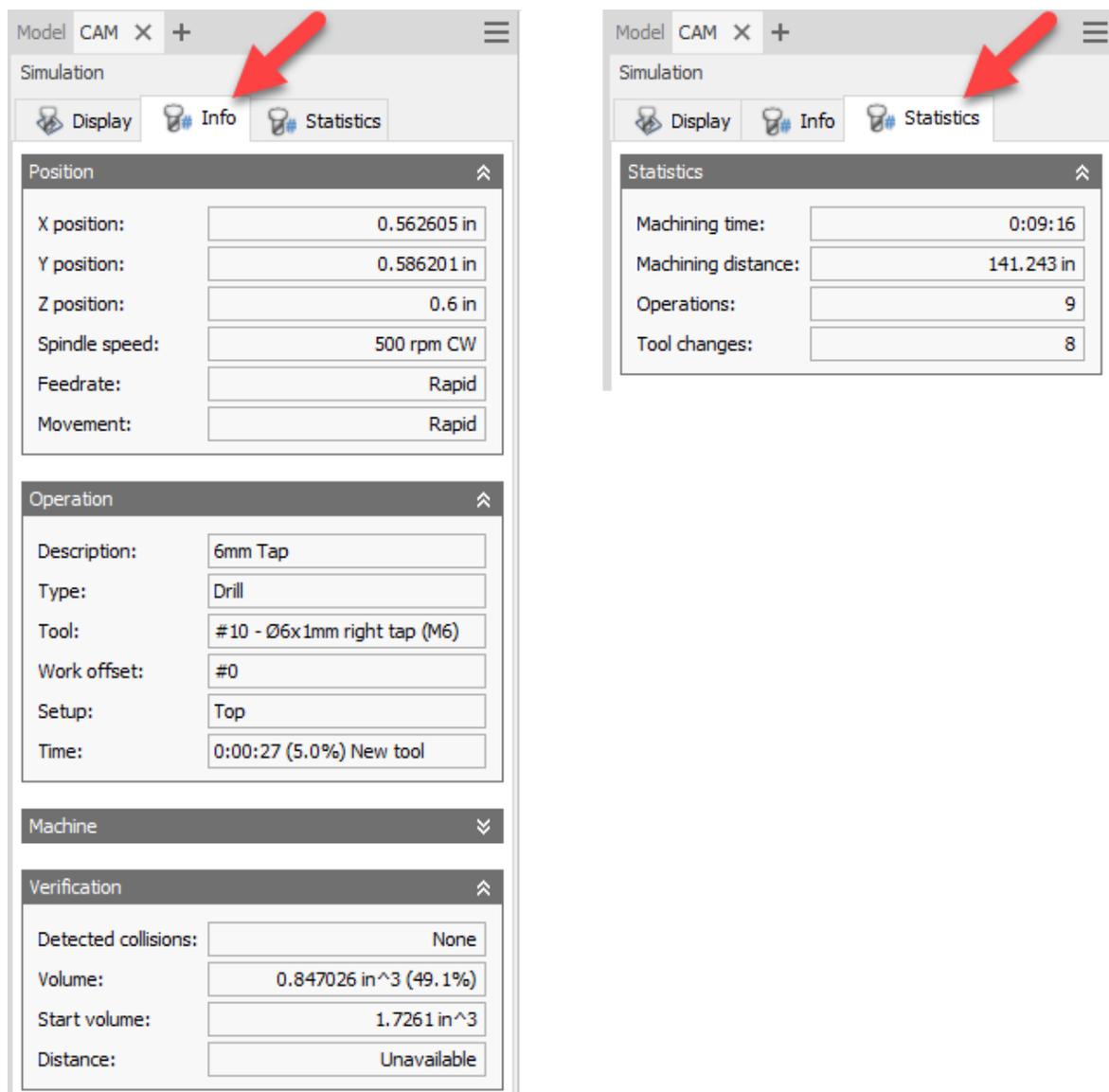


Figure 22

Simulation Player and Timeline

The simulation player and simulation player timeline (as shown in Figure 23) appear at the bottom of the graphics window when you activate the Simulation environment.



Figure 23

- The simulation player timeline displays the operations that will be played in the simulation. The setups/operations displayed are dependent on what was selected in the CAM browser prior to initiating the simulation. The current operation is displayed in green, while the remaining are displayed in alternating shades of gray to distinguish each change in operation.
- If an operation encounters a collision, a red line appears through the operation in which it occurs.
- Information on an operation can be displayed by hovering the cursor over an operation.
- Click  (Play) in the simulation player to play all selected operations/setups/toolpaths. As they are played, material is removed from the stock/solid model to simulate the machining of the model. The additional playback controls are described in Figure 24.
- To control the speed of the playback, select the circular control under the simulation player (as shown in Figure 24) and drag it to the right to increase the speed or to the left to reduce the speed. The playback reverses once you drag to the left of the midpoint. Speed control can be manipulated while the simulation is in progress.

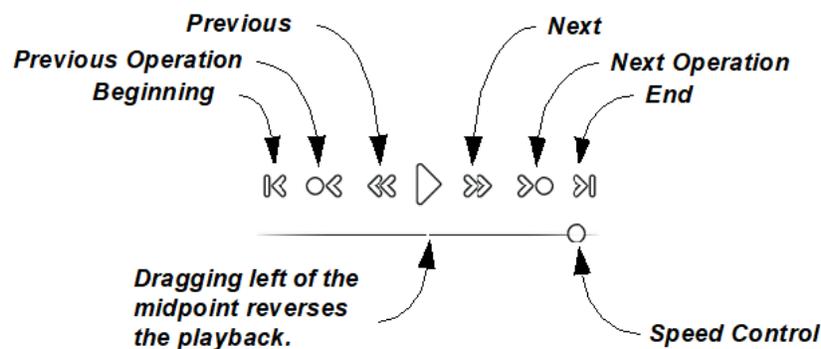


Figure 24

Once the simulation has completed, you will notice that all selected operations/setup/toolpaths are visible in the graphics window. For example, in Figure 25 the file's Operation(s) node was selected for simulation, so all toolpaths in both Setup1 (top) and Setup2 (bottom) are shown.

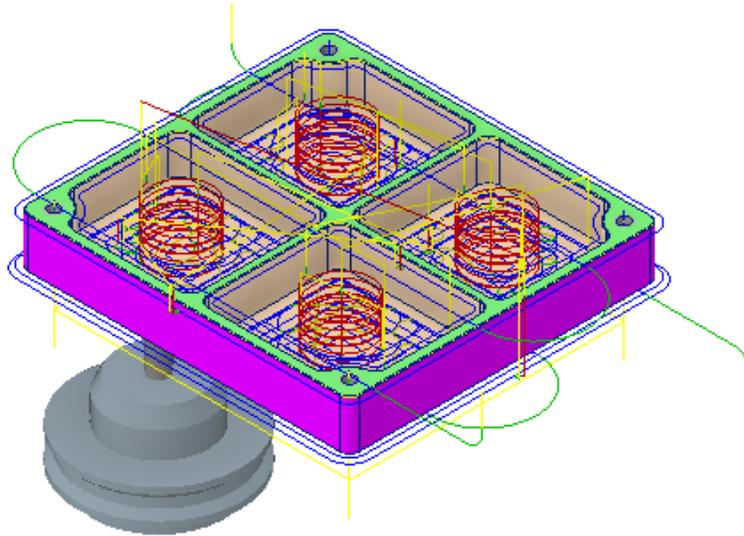


Figure 25

Click **Close** at the bottom of the Simulation palette to close the palette.

Generating the CNC Code

Once all the setups and toolpaths have been created, the CNC code can be generated through Post Processing the data into code so that it can be read by the machine. In the Toolpath panel, click  (Post Process). The Post Process dialog box opens similar to that shown in Figure 26.

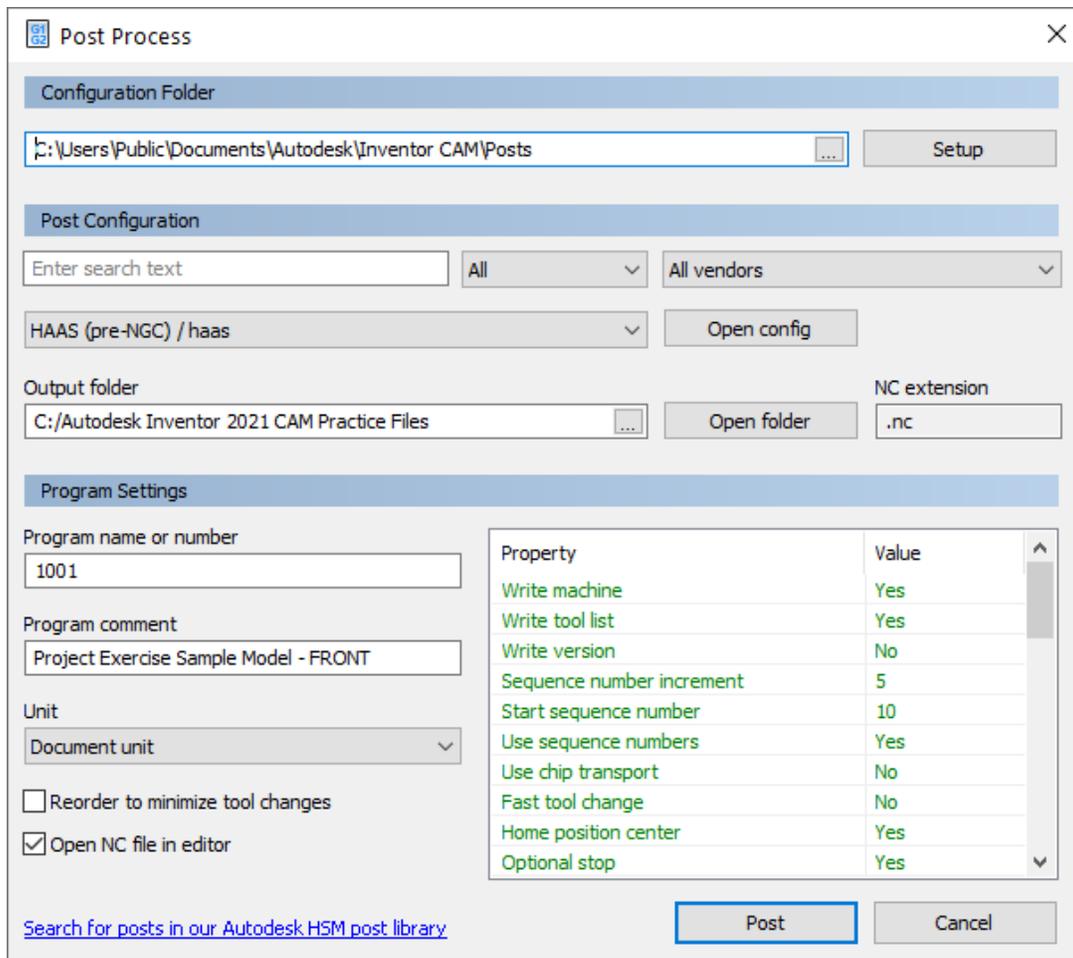


Figure 26

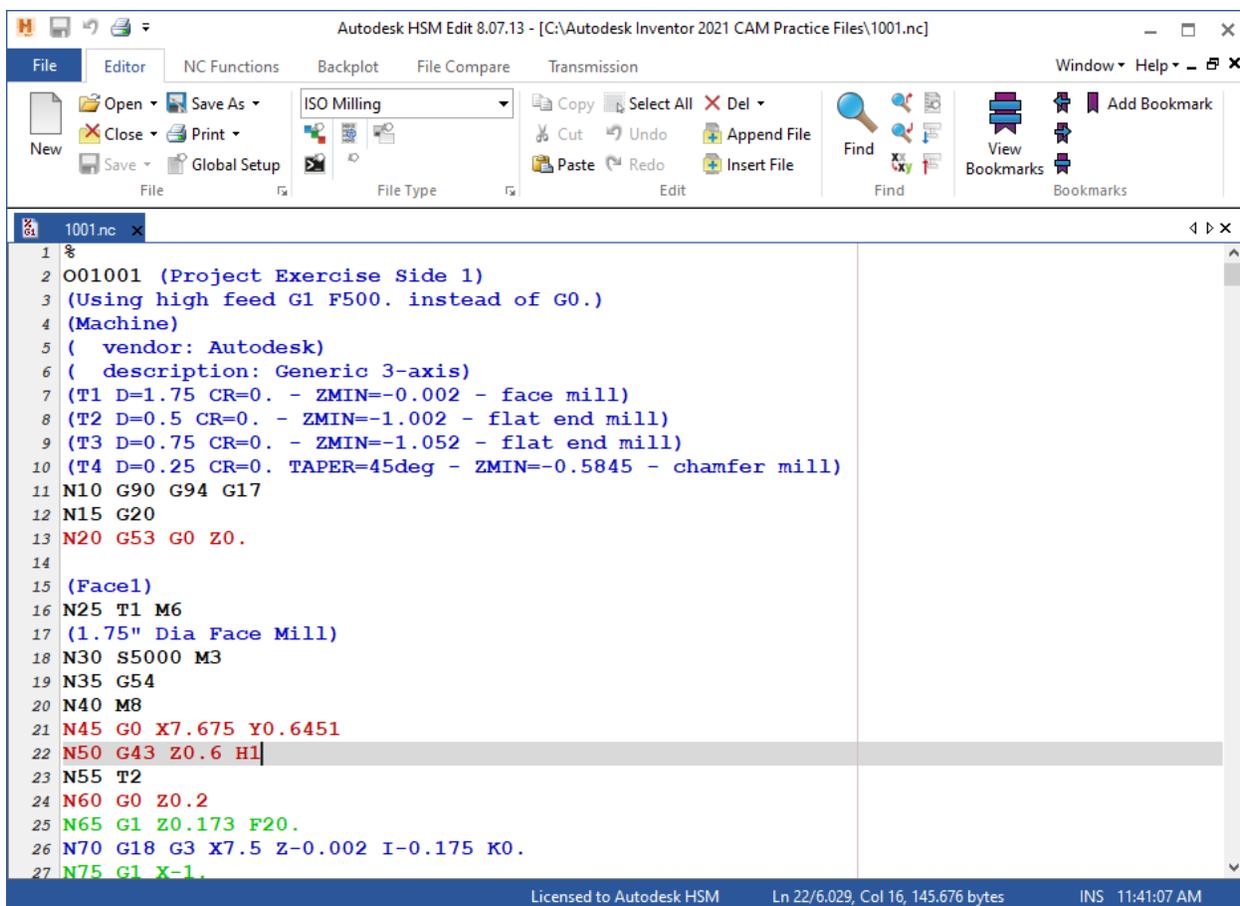
There are several settings that can be defined to customize the generated code. The main settings include:

- Select the required machine type from the configuration drop-down list.
- Change the Output folder to a location where you can easily locate the output file.
- Change the default Program name or number, if required. Notice that the Program name or number is set by default to 1001. This is the value that was assigned by default when the setup was created.
- Enter a Program comment value to help identify the program.

- Select **Reorder to minimize tool changes**, if required.
- Ensure that **Open NC file in editor** is selected so that once the *.nc file is created, it will be displayed in the editor.
- Customize the values for the properties listed in the bottom right-hand corner of the dialog box, as needed.

Consider using the **Setup** option at the top of the Post Process dialog box to access the Autodesk HSM Post library to download pre-created post files for select machines.

Click **Post**, then click **Save** to accept the default *.nc file name and begin post processing. Once complete, the post processed machine code will open in an editor, similar to that shown in Figure 27. In this case, the resulting code is N-code that is readable by a HAAS (pre-NGC) / haas machine. Scroll through the N-code to review it prior to sending to your machine. Further edits can be made, as needed to this file.



```

Autodesk HSM Edit 8.07.13 - [C:\Autodesk Inventor 2021 CAM Practice Files\1001.nc]
File Editor NC Functions Backplot File Compare Transmission Window Help
New Open Save As ISO Milling Copy Select All Del Append File
Close Print Cut Undo Append File
Save Global Setup Paste Redo Insert File Find Find View Bookmarks Add Bookmark
File File Type Edit Find Bookmarks Bookmarks

1001.nc x
1 %
2 O01001 (Project Exercise Side 1)
3 (Using high feed G1 F500. instead of G0.)
4 (Machine)
5 ( vendor: Autodesk)
6 ( description: Generic 3-axis)
7 (T1 D=1.75 CR=0. - ZMIN=-0.002 - face mill)
8 (T2 D=0.5 CR=0. - ZMIN=-1.002 - flat end mill)
9 (T3 D=0.75 CR=0. - ZMIN=-1.052 - flat end mill)
10 (T4 D=0.25 CR=0. TAPER=45deg - ZMIN=-0.5845 - chamfer mill)
11 N10 G90 G94 G17
12 N15 G20
13 N20 G53 G0 Z0.
14
15 (Facel)
16 N25 T1 M6
17 (1.75" Dia Face Mill)
18 N30 S5000 M3
19 N35 G54
20 N40 M8
21 N45 G0 X7.675 Y0.6451
22 N50 G43 Z0.6 H1
23 N55 T2
24 N60 G0 Z0.2
25 N65 G1 Z0.173 F20.
26 N70 G18 G3 X7.5 Z-0.002 I-0.175 K0.
27 N75 G1 X-1.

Licensed to Autodesk HSM Ln 22/6.029, Col 16, 145,676 bytes INS 11:41:07 AM
  
```

Figure 27