F1 in Schools™ Car Design Inventor HSM CAM Tutorial

Abstract: Gain a basic understanding of the CAD-CAM-CNC work flow. The F1 in Schools Car CAD model has been generated in previous F1 in Schools Car Design Inventor tutorials. In this tutorial, you apply the CAM toolpaths and the NC-code generated to send the card model to a CNC milling machine using Inventor HSM for manufacturing. Learn how to prepare the model to cut, create the stock model, orient the model, pick the surfaces to cut, CAM and generate the toolpaths. Create the toolpath strategies to cut the car, and create the NC-Code to send to the machine.

Table of Contents

Getting Started with Inventor HSM 4

Downloading Autodesk Inventor HSM 4
Datasets 4
Video Tutorials 4
F1 in Schools Rules and Technical Regulations 4

Key Terms 5

Activity 1: Understanding the CAD-CAM-CNC Process for an F1 in Schools Car Body 5

Manufacturing an F1 CAR using a CNC Machine, Subtractive MFG 5
Design it with CAD 6
Creating a Toolpath 6
Cut it on a CNC Machine 6
Setup, Orientation and Stock 7
Stock 7
Orientation of the XYZ Axis Program Reference Zero 7
Left Car Side Orientation 8
Right Care Side Orientation 8
Vertical Milling Center (VMC) Machine Terminology 8
Work Coordinate System Terminology 8

Activity 2: Open the F1 In Schools Car Model in Inventor, Prep Model for CAM 10

Step 1: Load the F1 in Schools Model 10
Step 2: Turn off the Wings 12
Step 3: Save the File with a New Name 13

Activity 3: Prepare the F1 in Schools Model to Toolpath 14

Step 1: Create Sketched Lines to Define Setup Points 14

Activity 4: Open CAM, Create the Setup for the Left Side of the Car 16

Step 1: WCS Point for the First Setup 17
Step 2: Modify the Stock Model to Represent the F1 Model Block Blank 23

Activity 5: Creating a Toolpath and Defining a Tool 25

Step 1: Select 3D Toolpath 25
Step 2: Understanding the Toolpath Tabs 26
Tool Tab 26
Geometry Tab 26
Heights Tab 26
Passes Tab 26
Linking Tab 27
Step 3: Select the Cutter and customize it to cut an F1 Car 27
Step 4: Define Machining Parameters for the Denford CNC Router 30
Step 5: Geometry Tab Containment of the Toolpath 31
Step 6: Heights Tab set the machining depth 31

Step 7: Passes Tab Set the Space between Passes and Toolpath Orientation 32
Parallel Cutting Strategies: 33
Step 8: Linking Tab 35
Activity 6: Create the Right Side Toolpath 36
Step 1: Copy and Paste the existing Setup and Toolpath 36
Step 2: Change the Z Axis Orientation of the Second Setup 37
Activity 7: Simulate the Toolpath 39
Step 1: Select the toolpaths and Simulate 40
Activity 8: Select the Post Processor Generate the NC Code 41
Step 1: Select the Post Process and Generate the NC-code for the first toolpath. 43
Step 2: Generate the NC-code for the second toolpath Right Side. 45
Activity 9: Generate the Setup Sheet 46
Activity 10: Enrichment Optimization-Cut the Car Faster 48
Credits 50

Getting Started with Inventor HSM

Inventor HSM™ provides CAD/CAM solutions for Inventor users. The CAM module allows the user to create the toolpaths, that is, the path the CNC machine will take to cut out the model by cutting material away from a block. This method of manufacturing is called **Subtractive Manufacturing.** The opposite would be **Additive Manufacturing**. 3D printing is the best known application of Additive Manufacturing.

Downloading Autodesk Inventor HSM

Autodesk Inventor HSM Pro integrated CAD/CAM solution helps designers, engineers and CNC programmers rapidly turn designs into machined parts directly inside the Autodesk Inventor design environment. As a student or educator, you can download a free license of Autodesk Inventor HSM software for personal or education purposes at www.autodesk.com/education/free-software.

Datasets

All project files that are required for the tutorial are provided in a dataset .zip file. Download the dataset file and extract the files to your computer.

Video Tutorials

Download the video tutorials supporting this tutorial. The video tutorials offer the same step-by-step software instruction for learners that prefer guidance through video format.

F1 in Schools Rules and Technical Regulations

It is extremely important to that you design your car to comply with the current F1 in Schools rules and regulations. Each country has slightly different specifications that may change from year to year. It is critical that you download and review your country's F1 in Schools Rules and Regulations documentation and design your car to the outlined specifications.

Activities in this tutorial leverage F1 in Schools rules and regulations for 2015-2016 and may not apply to your country. Please confirm the F1 in Schools rules and regulations for your country and competition before getting started. Go to your local F1 website and download the F1 in Schools rules and regulations documentation http://www.f1inschools.com/international-sites/.

Key Terms

CAD: Computer Aided Design; software to design 2D and 3D models.

CAM: Computer Aided Manufacturing\Machining; software to generate toolpaths, an NC-Code to drive.

CNC: Computer Numerical Control; different machines use CNC controls: mills, lathes, lasers. To cut an F1 car, a 3 axis CNC mill will be used.

Toolpath: The path that the tool will follow in the Milling Machine to cut away material.

Post processor: In the CAM module the toolpath is automatically turned into NC-code specific to the CNC machine needs, through a post processor.

NC-Code: Instructions/language the machine can understand (i.e. where to cut when).

Fixture: What holds the material block in place on the machine table.

Work Coordinate System (WCS): Calibrating the model position, so the computer knows where to match a point on the CAD model to the corresponding point on the material block.

Activity 1: Understanding the CAD-CAM-CNC Process for an F1 in Schools Car Body.

In the previous tutorial, you designed the F1 In Schools car body. The next phase of the design (CAD) and manufacturing (CAM) process is to orient the CAD file so that the CNC milling machine can understand it. This document will cover how to create the CNC toolpaths to cut an F1 in Schools car body on a CNC machine.

Note: Before starting this tutorial, it is recommended to orient the F1 In Schools car model so that the XY zero is placed at the center of the CO_2 hole and the Zero plane is set to the back of the car. This tutorial assumes that this has not been done.

Manufacturing an F1 CAR using a CNC Machine, Subtractive MFG

Understanding the process the car will go through to be machined is extremely helpful before starting the step-by-step process. Understanding the whys and the definitions of a few terms makes this easier to learn, repeat, and apply in different circumstances.

Design it with CAD

In the first stage, the F1 in Schools car is designed to meet the design specifications as outlined in the F1 in Schools Rules and Regulations and the designer's functional aesthetics. This will be called "the model".

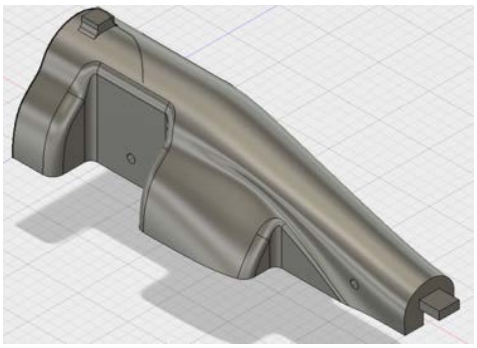


Figure 1-Finished Model

Creating a Toolpath

For basic designs, the car could be cut in one toolpath from the top. As car designs have increased in complexity, due to competition, the ability to cut the car from all 4 sides was developed, Left-Right-Top-Bottom. For most designs, all of the details that need to be machined can be reached in two setups, left side and right side, and for more complicated designs we add top and bottom.

In the first toolpath graphic, (see figure 2), the car is rotated with the left side up and a toolpath applied. In the second, (see figure 3), graphic the car has been rotated to the right side and the toolpath applied.

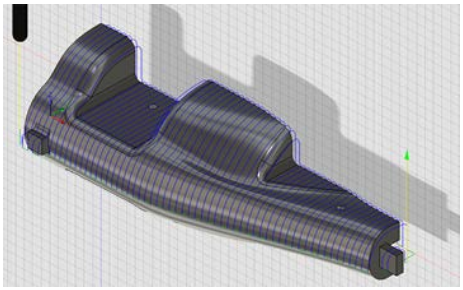


Figure 2-Toolpath Left Side

Note*: This process is for a fixture that is manually rotated between toolpaths. For an automatic rotary axis, a code can be inserted which allows the program to be mirrored and the fixture to rotate*.

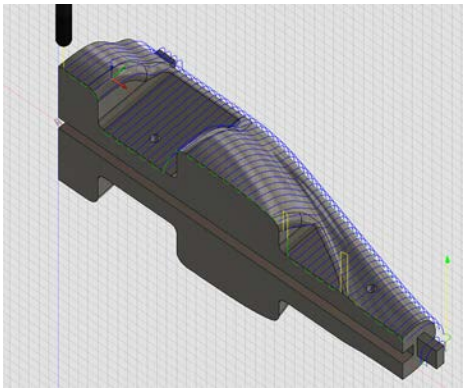


Figure 3-Toolpath Right Side

Cut it on a CNC Machine

To toolpath the car, we must know how it will be held in the manufacturing process. Figure 4 shows the model rotated to the left side with the machining complete. Focus on the fixture; note how the stock is held. On one end, a shaft is inserted into the CO_2 cartridge. On the opposite end, the stock is supported so the model can rotate around the shaft allowing all sides to be cut while keeping the stock aligned. The center of this hole on the back of the F1 Model Block will be our XYZ orientation point. This fixture is manually indexed, (moved from one position to another). Our model is basic in detail and would only require two setups; one with the model rotated to cut the right side and one to cut the left side. The cutter would come down from the top Z axis.

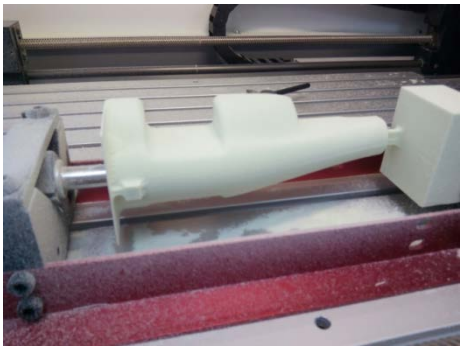


Figure 4-F1 Car and Fixture

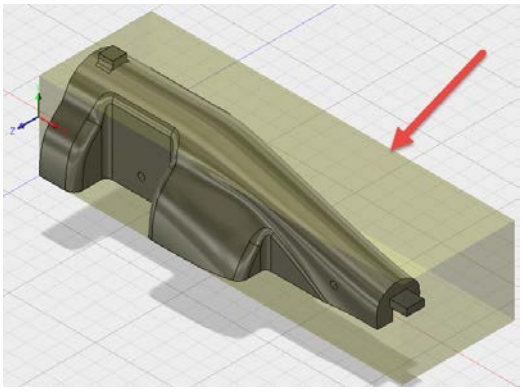
Setup, Orientation and Stock

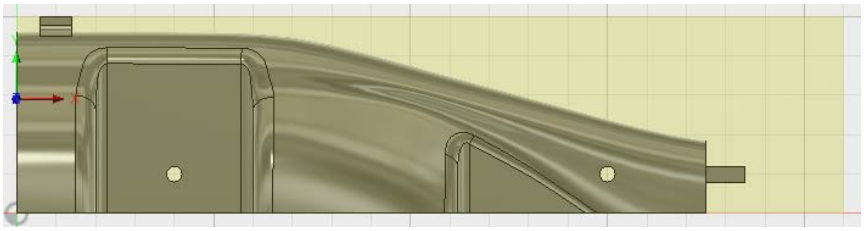
Next, it's important to understand how you calibrate the machine so that when the "go" button is pushed on the CNC machine, everything lines up and the F1 car is perfectly cut.

First, we will need to define the Setup. "Setup" comprises of defining the raw stock size and defining where the coordinate (0,0) will be in relation to the machine. This may sound a bit complicated, but as it is done, it is easy to comprehend. The model sits in space on the computer screen. It needs to be aligned to how it will be held and cut in the CNC machine.

Stock

Here we see the model with the stock (F1 Model Block) defined. The stock is displayed as a translucent box. The dimensional size of the F1 Model Block is 65mm wide, by 50mm high by 210mm, (it is actually longer, but for this application we will actually use 210, giving a safety margin to avoid hitting the fixture with the cutting tool.



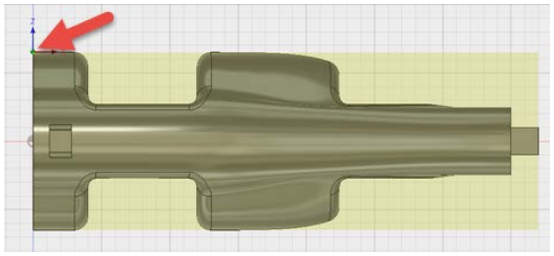


Orientation of the XYZ Axis Program Reference Zero

Setting the orientation of the coordinate system is vital to the success of cutting out the F1 in Schools car. A point called the Work Coordinate System/Origin has to be defined on the machine bed so the computer can match the WCS on the CAD model to the WCS on the machine bed. The cutter on the CNC machine will be set to the Program Reference Zero, before cutting starts.

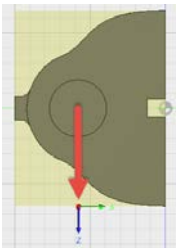
Left Car Side Orientation

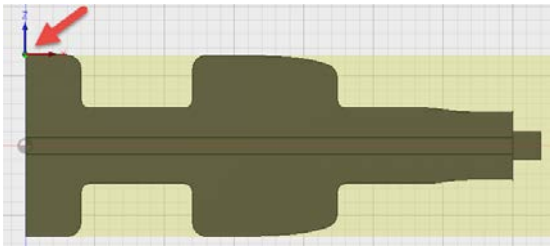
The correct orientation of the X, Y and Z axis when cutting the Left side of the car is illustrated here. The X and Y axis are the center of the CO_2 canister hole, and the Z axis is set to the top of the stock.



Right Care Side Orientation

Here the car is rotated in the fixture to cut the right side so in the CAM system it must change as well.





Vertical Milling Center (VMC) Machine Terminology

The material to be machined is fastened to the machine table. This table moves in the XY-Plane. As the operator faces the machine, the X-Axis moves the table left- right. The Y-Axis moves the table forward-backward. (See figure 2)

The machine column grips and spins the tool. The column controls the Z-axis and moves up-down.

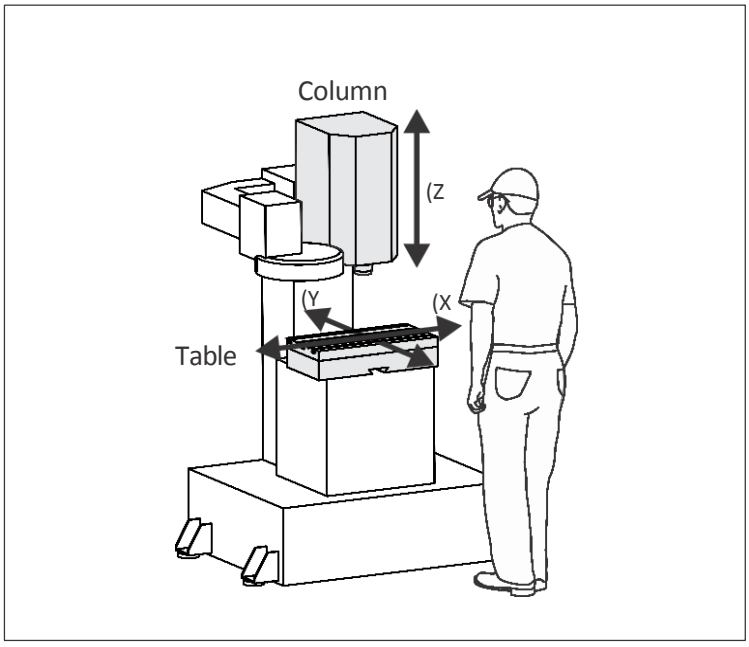


Figure 2: Vertical CNC Machining Center

Work Coordinate System Terminology

To make programming and setting up the CNC easier, a Work Coordinate System (WCS) is established for each CNC program.

The WCS is a point selected by the CNC programmer on the part, stock or fixture from which all toolpaths will be generated. While the WCS can be the same as the part origin in CAD, it does not have to be. While it can be located anywhere in the machine envelope, its selection requires careful consideration.

Best Practice: *Place your WCS against the fixed jaw, by doing this the stock can vary in size, but the part will not.*

- The WCS location must be able to be found by mechanical means such as an edge finder, coaxial indicator or part probe.
- It must be located with high precision: typically plus or minus .001 inches 0.03mm.
- It must be repeatable: parts must be placed in exactly the same position every time.
- It should take into account how the part will be rotated and moved as different sides of the part are machined.

For example: The illustration below shows a part gripped in a vice. The outside dimensions of the part have already been milled to size on a manual machine before being set on the CNC machine.
The CNC is used to make the holes, pockets, and slot in this part. The WCS is located in the upper-left corner of the stock. This is common practice as the WCS is against the hard jaw. Select a position/corner that is easily found using an Edge Finder or Probe.

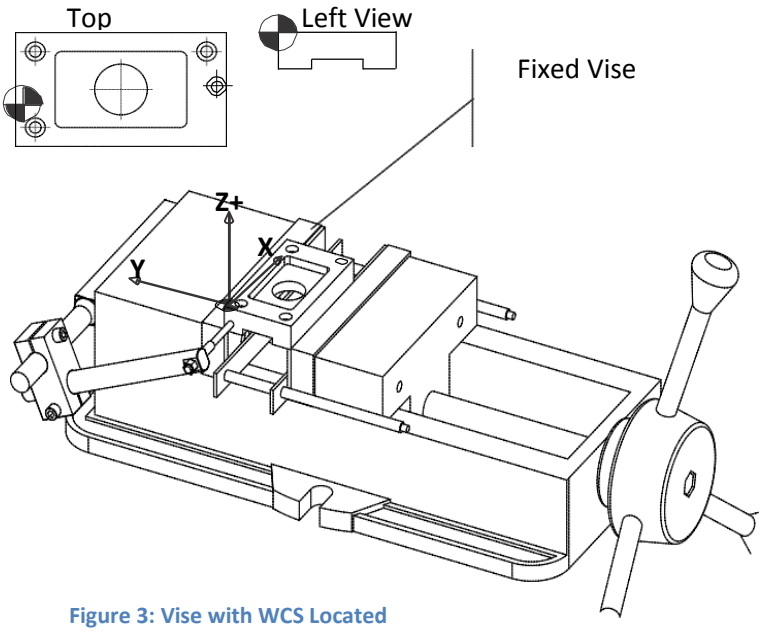


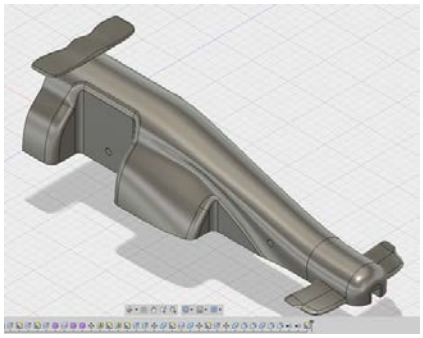
Figure 3: Vise with WCS Located

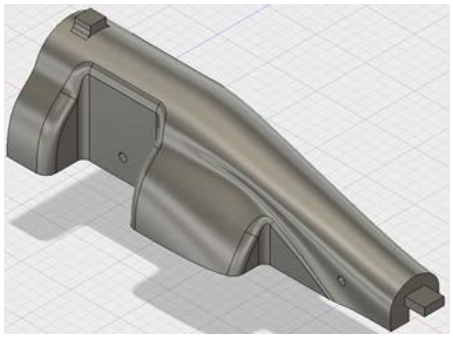
Activity 2: Open the F1 In Schools Car Model in Inventor, Prep Model for CAM

In this activity, you will open the F1-CAM-Car file. Then you will prepare it to apply the toolpath, as the rear wing and the nose and wing will be 3D printed. These features will be turned off. Finally you will save the file with a unique name.

You will:

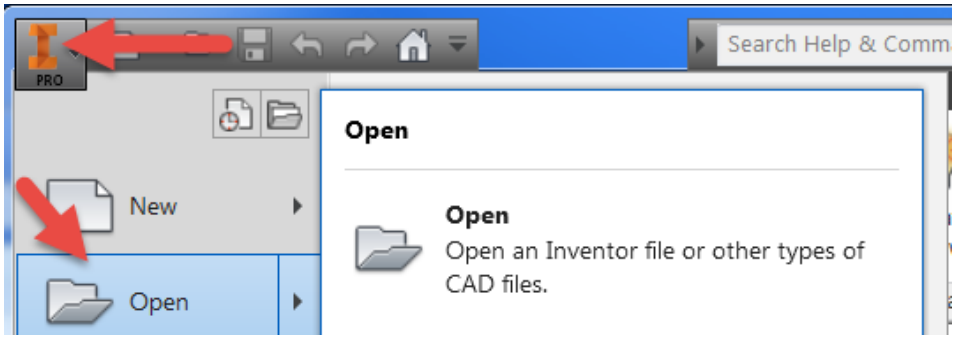
- Open file
- Turn off bodies
- Turn off sketches
- Save file with new name





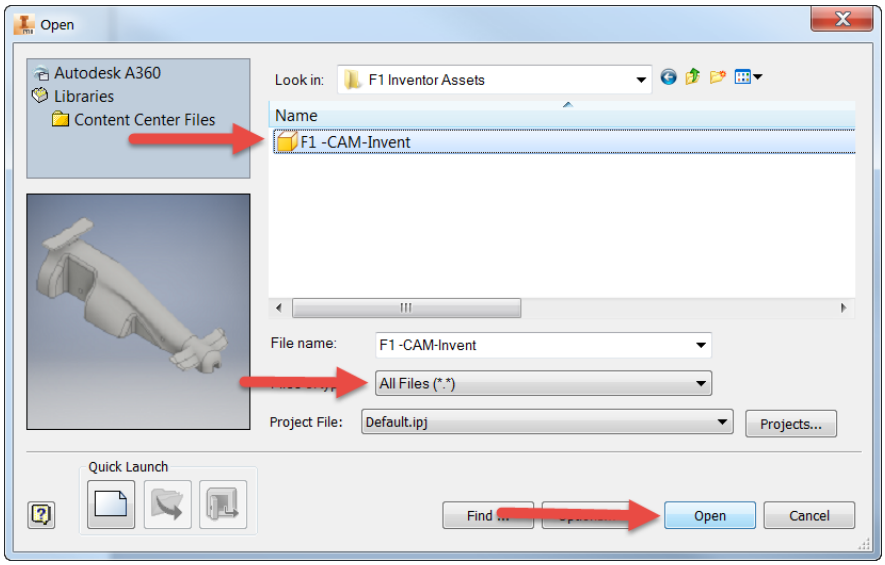
Step 1: Load the F1 in Schools Model

1. If you have previously completed the F1 in schools car design lesson, simply load the file and skip to Step 2.
2. **Open Inventor** then **Select the Inventor Icon,** upper left.
3. **Select Open** on the left panel.

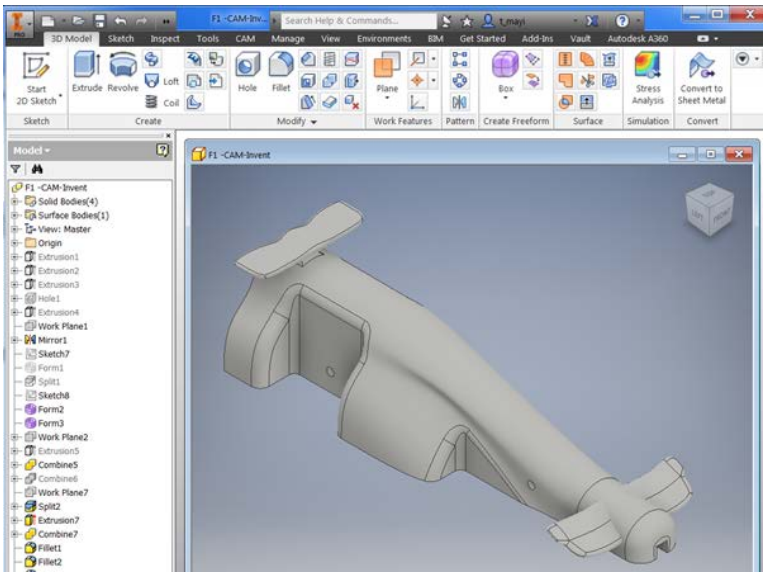


4. **Path to** where the F1-CAM-Invent assets were saved.
5. Select **All Files**, if no files are displayed.
6. Select **F1-CAM-Invent.itp.**

7. Select **Open.**



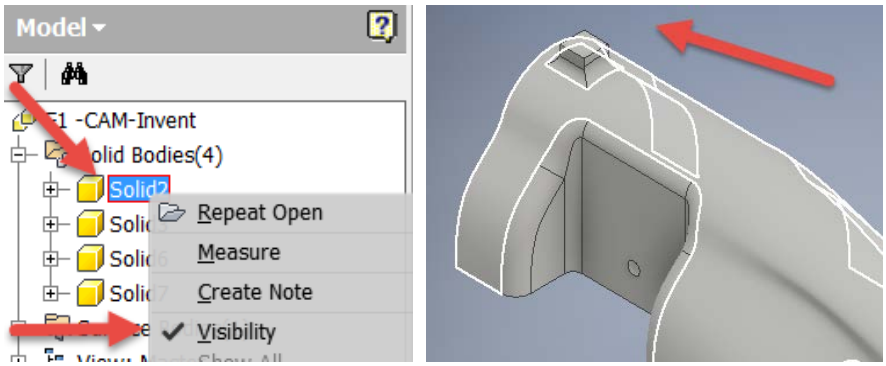
8. The F1-model is displayed.



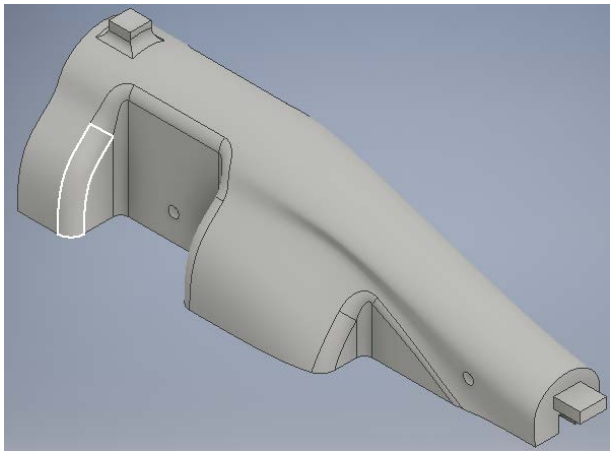
Step 2: Turn off the Wings

As the nose and wings will be 3D printed, and not machined so we will turn them off.

1. **Open Bodies list from the browser, click on the + to expand the list.**
2. **Right click on the first body in the list.**
3. **Uncheck Visibility.**
4. The back spoiler will disappear.

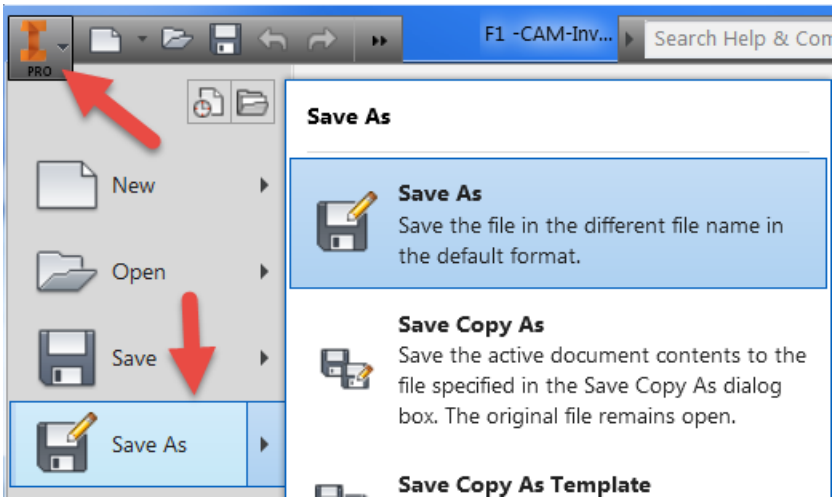


5. **Turn off Visibility for the first, second and third solids** to remove the back wing, front wing and nose, using the above procedure.
6. Your model should now look like this, just the body to be machined with the nose and the wings removed.

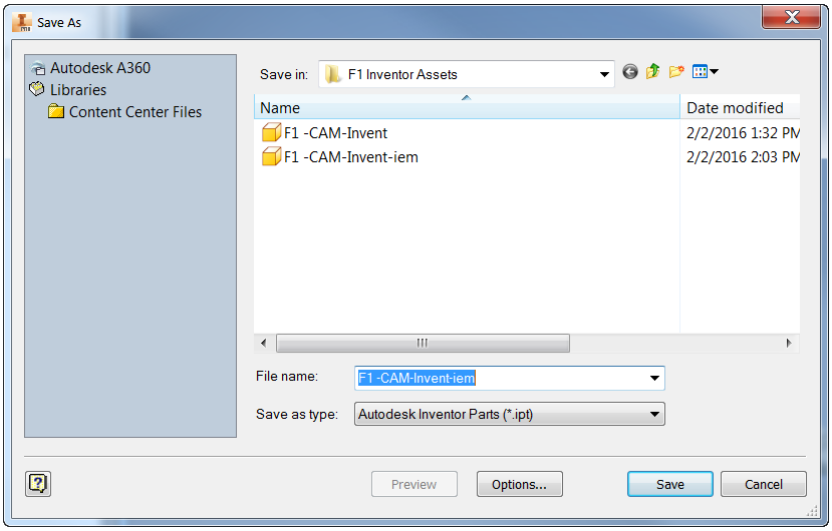


Step 3: Save the File with a New Name

1. **Click the Inventor Icon.**
2. **Select Save As, path the directory you want to save the file too.**



3. **Save file with new name (record the path and name), suggestion: F1-CAM-Inventor-(your initials).**



Activity 3: Prepare the F1 in Schools Model to Toolpath

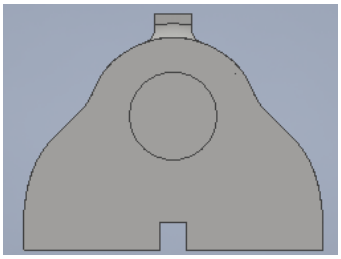
In this section, you will add some additional geometry to prepare the model for the application of toolpaths. Creating this geometry will allow you to set the orientation of the model easily.
You:

- Create a new sketch
- Sketch line to the center of the CO_2 hole
- Sketch line to the edge of the blank

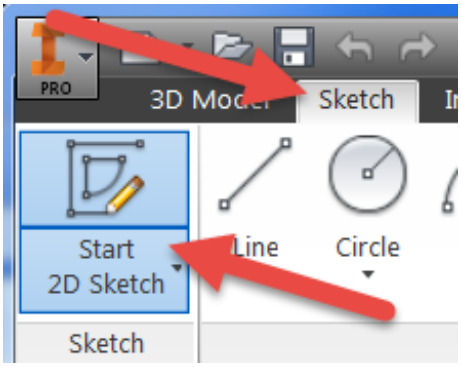
Step 1: Create Sketched Lines to Define Setup Points

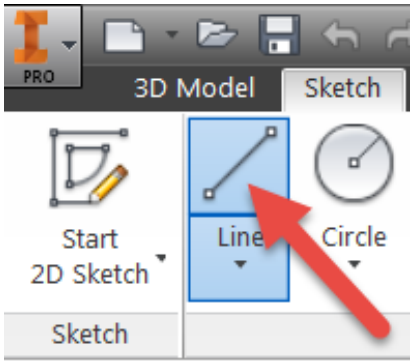
To toolpath a model, the XYZ coordinate system needs to be defined by sketching a few lines. You will be able to accomplish this easily with a simple selection.

1. **Use the View Cube to Orient the model to the Back**, so you are looking at it from the back side where the CO_2 Hole is situated.



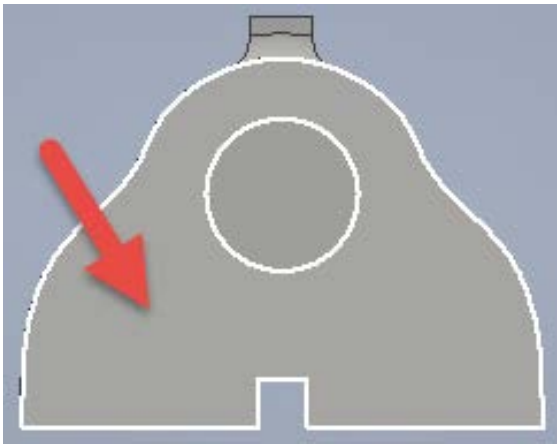
2. **Select the Sketch Tab, then Start 2D Sketch** from the menu, **Select Line.**

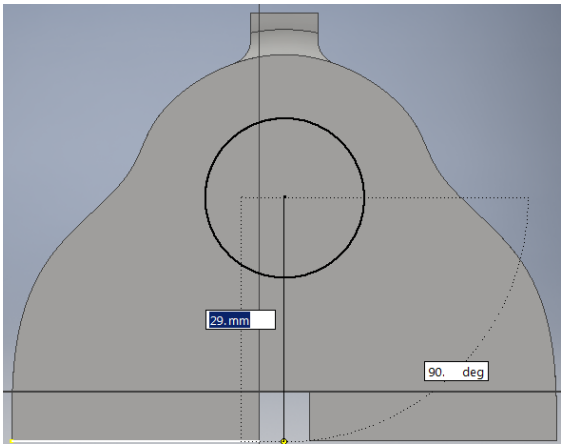




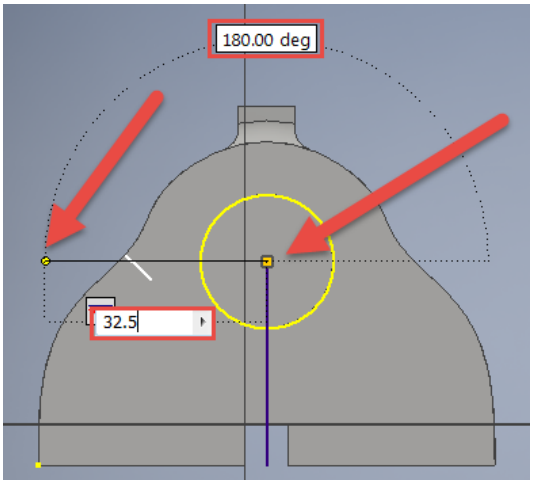
3. **Select the sketch plane, Select the center of the CO_2 Circle.**
4. **Move your mouse over the circle edge and the center point of the hole will be displayed.**

5. **Select the center point and drag it straight down to 29mm, 90 deg, click.**

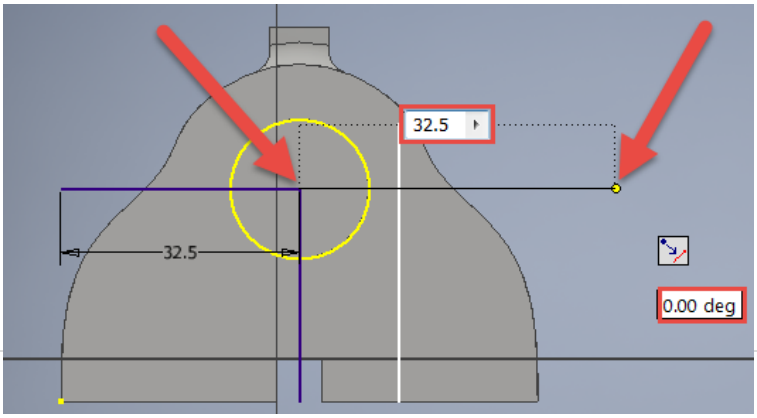




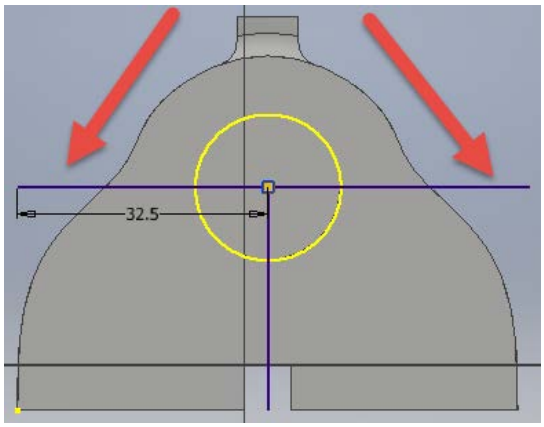
6. **Hit Esc key.**
7. **Select Line.**
8. **Sketch a horizontal line** from the **center of the CO_2 hole** out to the **Left=32.5mm**, **at 180 degrees** this will define the edge of the material.
9. **Hit Esc key.**



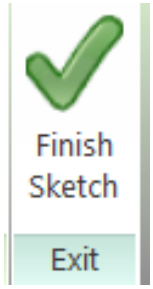
10. **Select Sketch** from the menu and **Select line.**
11. **Sketch a horizontal line from the center of the CO_2 hole out to the right**, this will define the edge of the material on the right side. **Select the center of the CO_2 hole** and then drag it to the right, **Input=32.5mm** at **0.00 degrees.**



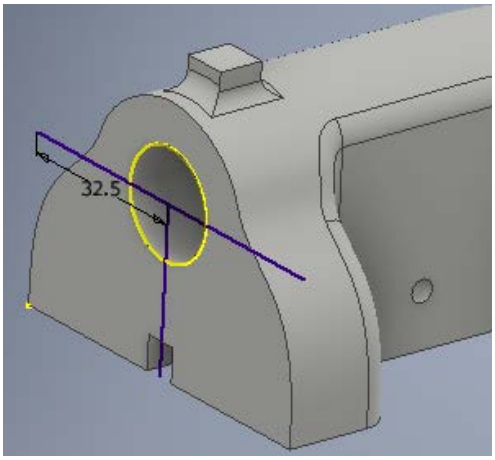
12. Your sketch should look like this graphic.



13. **Select Finish Sketch,** and leave the sketch visible.



14. This sketch should remain visible. The model should look like this when you rotate it

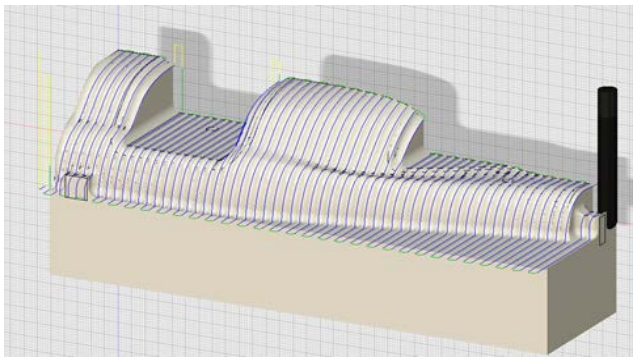


15. Save your file F1-CAM-Invent-(your initials).

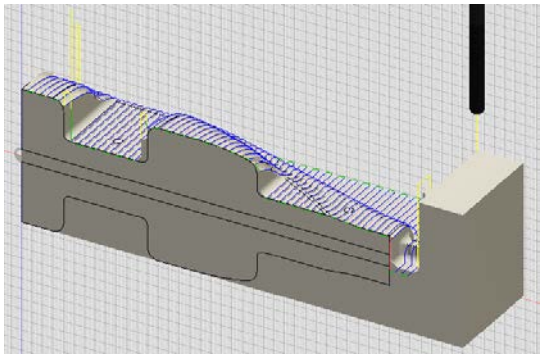
Activity 4: Open CAM, Create the Setup for the Left Side of the Car

Before we create the CAM toolpaths to drive the CNC machine, we must establish the Work Coordinate System, (WCS). We set the WCS in the software according to how the part will be set up on the CNC Milling machine. On any 3 Axis vertical mill the Z+ moves the cutter up, Z- move the cutter down. On the CNC milling machine, we will set the car up to cut the left side first. After it is complete, the car will be manually rotated in the fixture so the right side is up. Because we have two setups on the CNC machine, we will need two setups for the CAM toolpaths, one left and one right.

Setup 1 =Left Side



Setup 2=Right Side



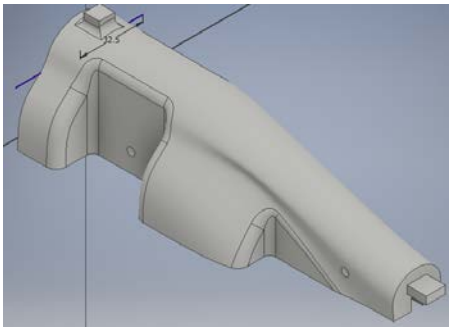
You will:

- Set Units to millimeters
- Create setup for Left side of Car
- Set the Work Coordinate System WCS
- Create F1 Model Block Stock Model
- Select the Model to be machined

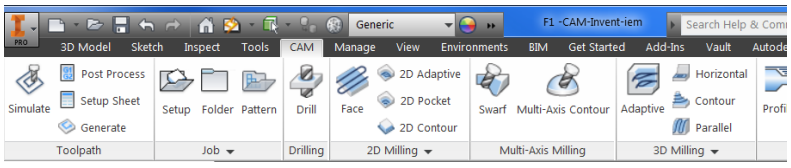
Step 1: WCS Point for the First Setup

You will open the CAM workspace, select the WCS point for the left side of the car.

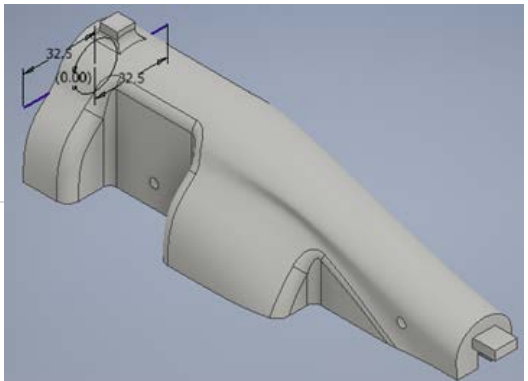
1. **Place the model into and isometric view by selecting the home icon.**



2. **Select the Cam Tab,** to open the Inventor HSM Workspace, the CAM menu bar is displayed across the top. The workflow in the CAM menu is left to right, for most applications. The Menu will be defined as it is used

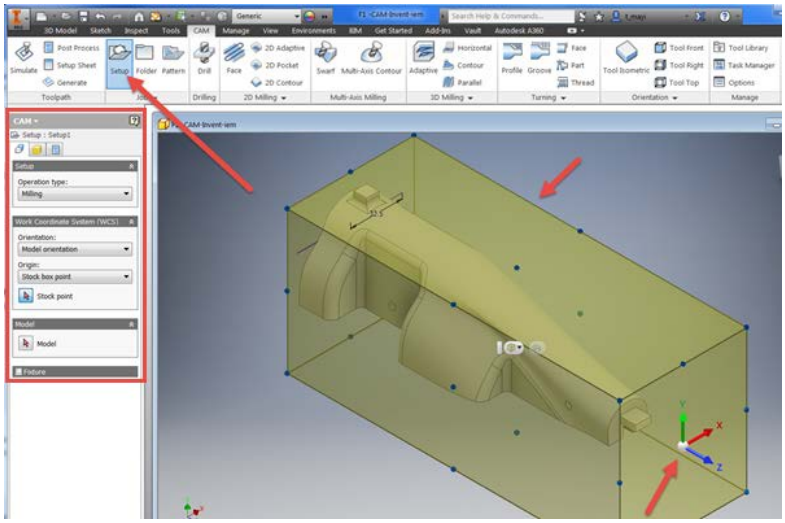


3. **Move the model into approximately this position** use the view cube to select home.



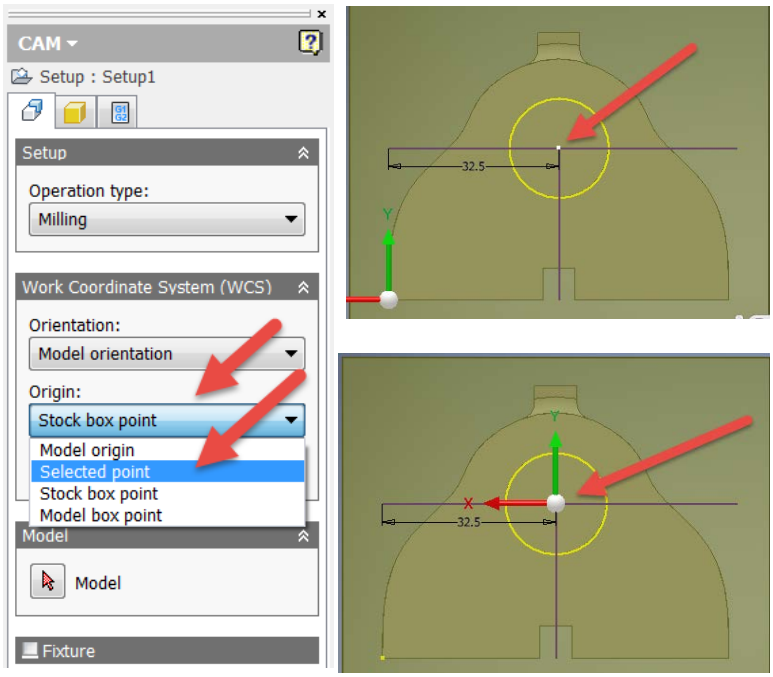
4. **Select Setup** from the CAM toolbar, and the CAM browser will show up in the browser pane. Also the stock model is displayed and the WCS gnomon. We will need to adjust both for our F1 car application.

5. The next step is to move the WCS gnomon to the correct position to cut the left side of the car. The white points on the stock model are called “Stock Box Points” and represent the most common positions for WCS points. The F1 car is held in a special fixture and requires a specific WCS. We will need to select either the center point of the CO_2 cartridge or the endpoint of one of the lines we created, depending on what CNC machine we are using.



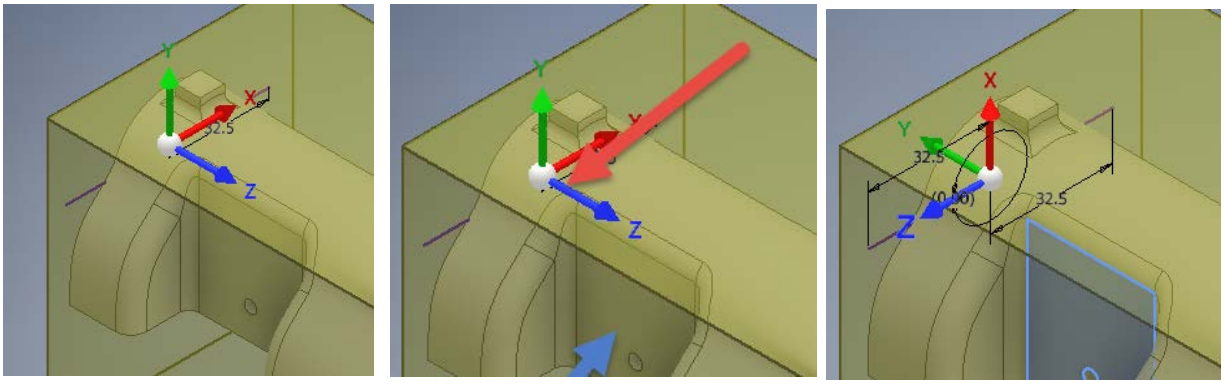
6. **Important:** Check with your instructor on which setup position to use:
    - For a Denford CNC Mill with their custom F1 in Schools fixture, the center of the CO_2 hole will be selected. Follow instructions 9-13.
    - If you are using a different CNC machine and fixture, it may use the center or the edge of the stock. For the edge of the stock skip to instruction number (14) in this this section.

7. **Setup at CO_2 hole center XYZ –Denford Mill.**
   Select the Stock Box Point dropdown arrow and pick **Selected Point, Select the center of the CO_2 hole,** rotate and/or zoom the model as needed, and the WCS gnomon will move to the center of the CO_2 hole, rotate the model as needed to get a good view to select the point.

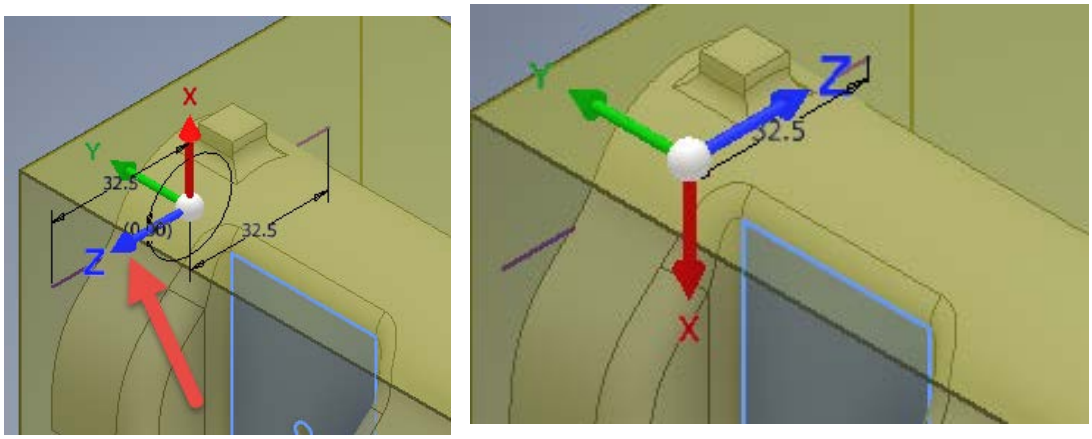


8. Now that we have the WCS in the right place we need it in the right orientation
9. Move the model into the home/isometric position.
10. **To orient the Z axis select the bottom of the Z shaft, it will highlight, then select the plane**, this will orient the Z pointer to the side of the stock.
11. The Z axis is now oriented to the side of the stock, next we will make it point to the left side of the stock as we will cut that side first.
12. The Z axis needs to point to the Left side of the Car, **Select the cone on the Z axis of the WCS gnomon**, and the Z axis will flip to the opposite side, to the left side.

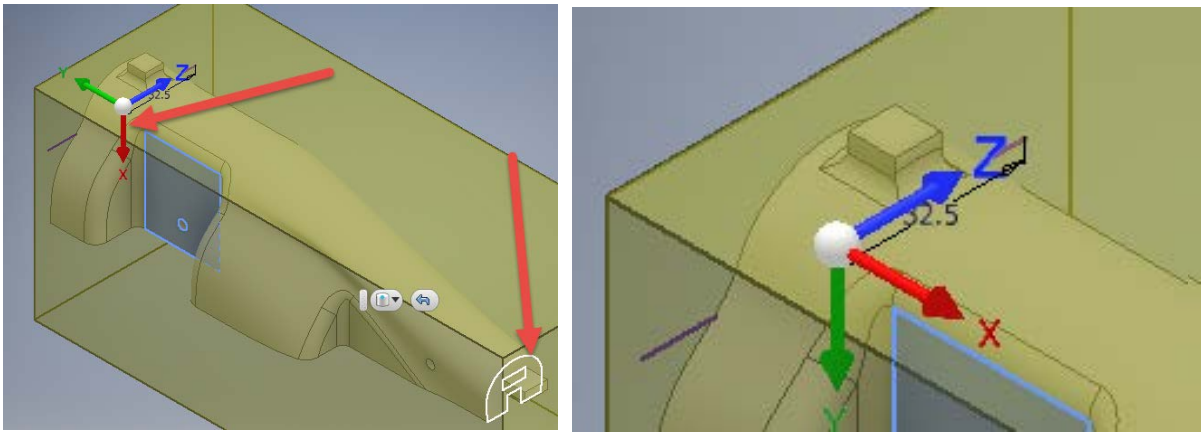
13. The Z axis is now oriented to the right side of the stock, next we will make it point to the left side of the stock as we will cut that side first.



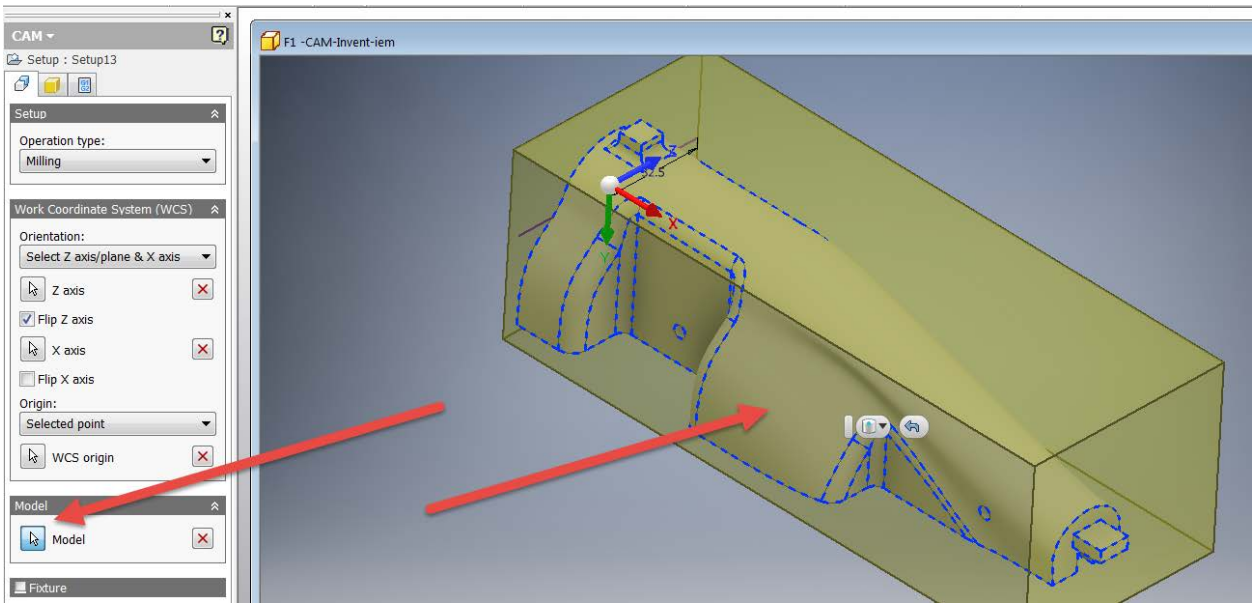
14. **Select the cone on the Z axis of the WCS gnomon**, and the Z axis will flip to the opposite side, to the left.



15. Now we need to orient the X axis, **Select the bottom shaft and of the X arrow and select the front plane.**
16. The WCS should be all setup for the cutting the Left side of the car.

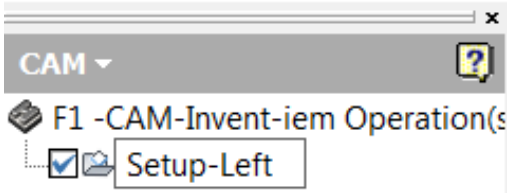


17. The WCS is now set, make sure all of the axes are oriented like the graphic
18. Next we select the model to be machined, **Select Model**, **then Select the body of the F1 Car**.



19. **Select OK** from the bottom of the Setup dialog box in the CAM browser.

20. Now Setup shows up in the CAM browser, **Click on setup wait a second click again** and rename setup to **Setup Left**



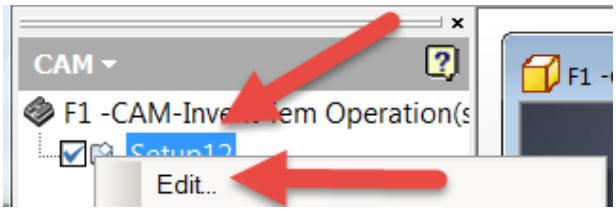
21. **Click the Save icon**

Step 2: Modify the Stock Model to Represent the F1 Model Block Blank.

Making sure the Stock Model is correctly represented is vital to the successful machining of the model. The Stock Model is used by the CAM software to determine where stock material is on the machine bed.

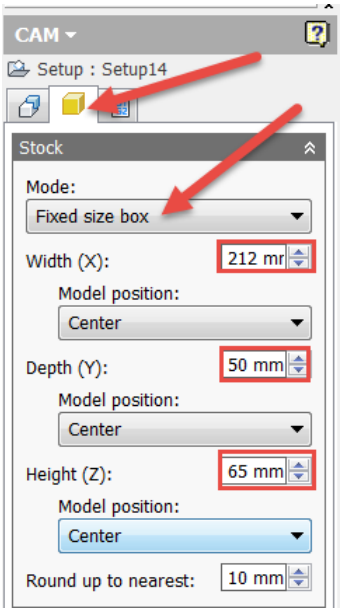
Note: ***This is very important as you don't want the cutter going through the stock at the programmed federate, (slow) and fast through air, not the other way around.***

1. **Place the model into isometric view by selecting the home icon.**
2. **Right Click on Setup** and **Select Edit** from the top of the list and the Setup dialog is displayed.

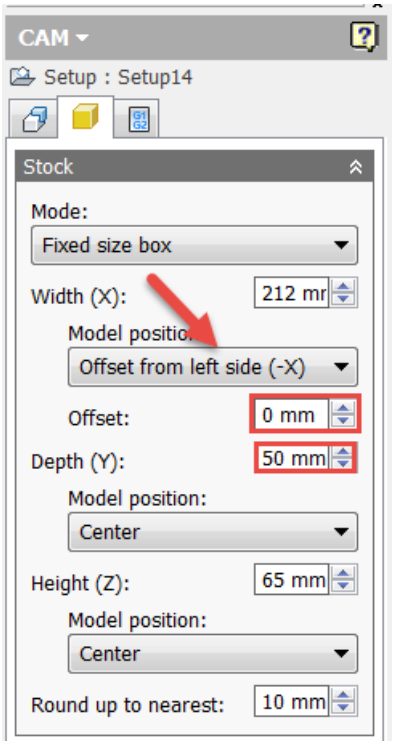


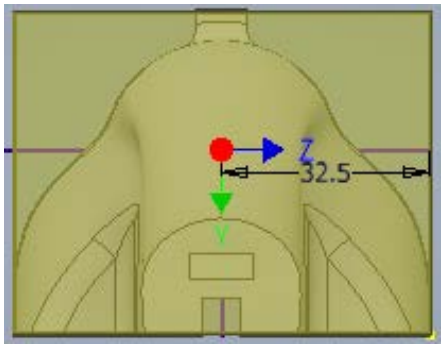
3. **Select the Stock Tab** and **Select Fixed size box**,
4. I**nput the following:**
5. **X=210mm long, Y=50, Z= 65mm**. (The blank is actually longer, but we are printing the nose and wing, so what we need to use of the blank is quite a bit less.)

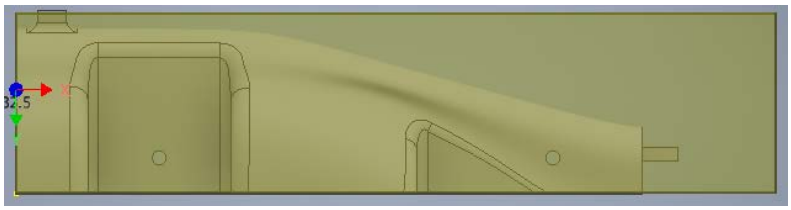
   Note: Do not hit OK, as there is one more thing to do in stock setup, if you did hit it, then just right click on setup and select edit..



6. Offset the stock on the X Axis to be flush with the back of the car, go **to Model position under Width X, Select the down arrow. Select Offset from the left side, and in the Offset box Enter=0.**
7. **Select OK**, rotate the model and check to make sure the F1 in Schools model is fully enclosed in the Stock Model and the block is flush with the back of the car .

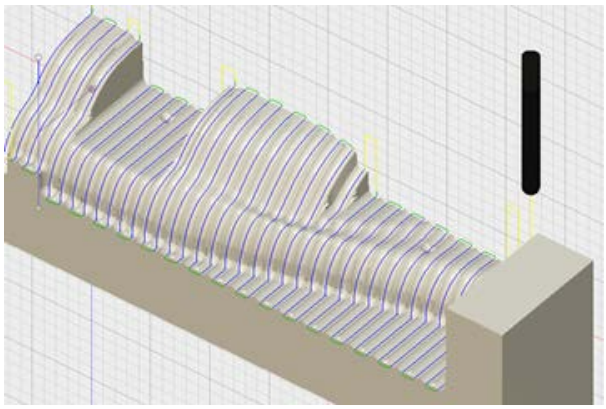






Activity 5: Creating a Toolpath and Defining a Tool

You have now successfully defined; F1 in Schools car setup for the left side, the Stock Model/Model Block and selected the model to be cut. We are now ready to apply the 3D toolpath. We will also need to define a cutter and set the machining parameters. You will:

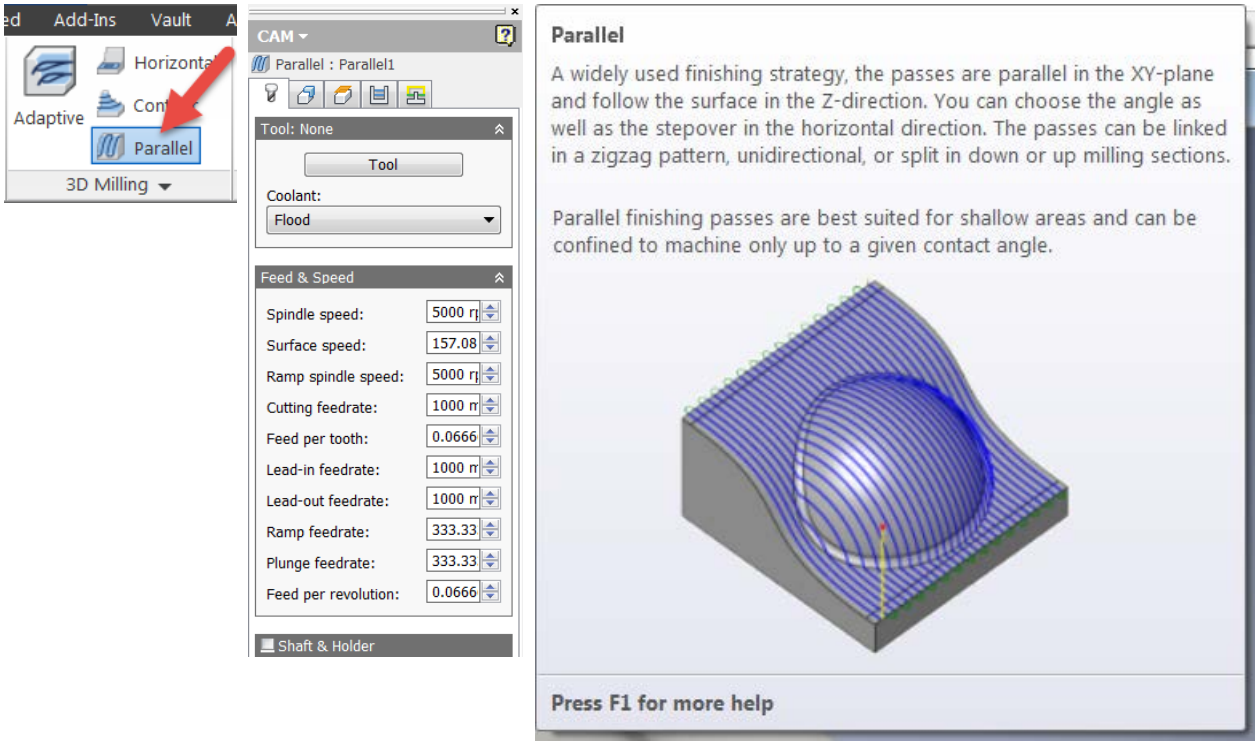


- Select the 3D toolpath strategy parallel
- Define the Cutting tool and select it
- Understand the toolpath dialog
- Set the machining parameters
- Select the geometry to cut
- Set how deep the tool will cut
- Set the cutter stepover
- Set the linking parameters

Step 1: Select 3D Toolpath

The workflow is left to right. We have established the Setup and since we are using a soft material like foam, we do not need to rough out the model with 2D toolpaths. We can just skip right to 3D toolpaths. This F1 in Schools model is made up of complex 3D curvy surfaces that require a 3D toolpath. There are many different toolpath strategies to make many different shaped parts. We will use a single 3D toolpath strategy to machine the F1 in Schools model.

1. **Select Parallel** from the 3Dmilling tab Parallel Toolpath dialog will display, in the CAM browser

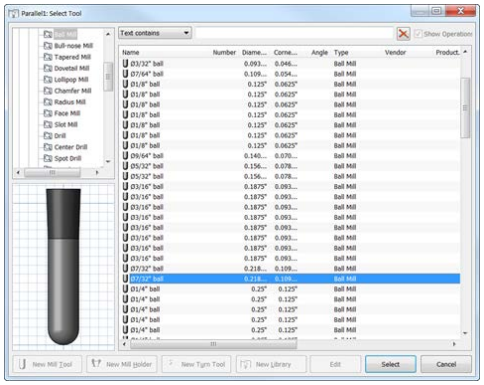


Step 2: Understanding the Toolpath Tabs

Just like the main CAM toolbar menu, all of the toolpath dialogs are set up with Tabs at the top with a workflow from left to right.

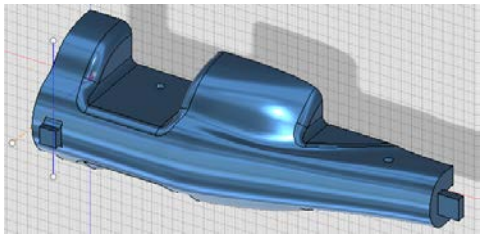
Tool Tab

In the Tool Tab, the tool for the operation is selected, and the machining parameters are set. Here tool libraries can be built, tools created, customized, and saved.



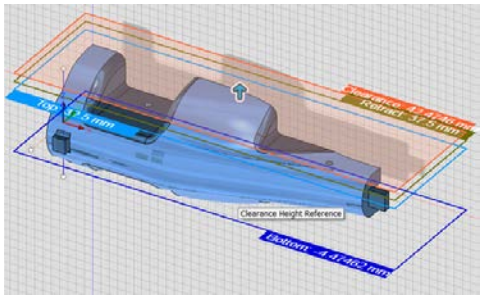
Geometry Tab

Selected the geometry to contain the toolpath.



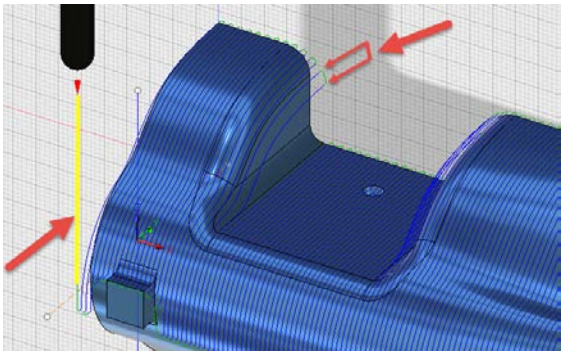
Heights Tab

The position of the standard planes are set, Top of Stock, Bottom of Stock, Rapid retract and more.



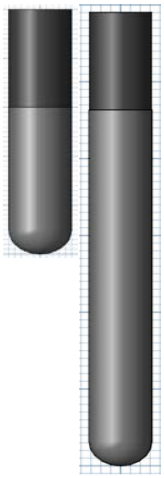
Passes Tab

In the Passes Tab, the spacing between the toolpaths is set along with tolerance of the toolpath. Tolerance is how closely the toolpath actually adheres to the true surface.



Linking Tab

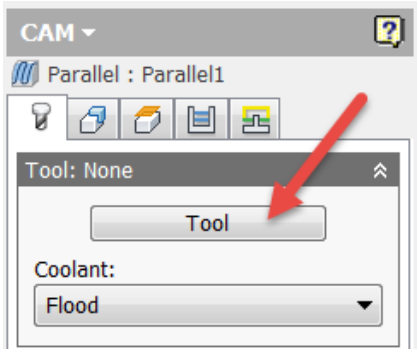
In the Linking Tab, how the cutting tool moves from one operation to another is set, the yellow line represents a rapid move.



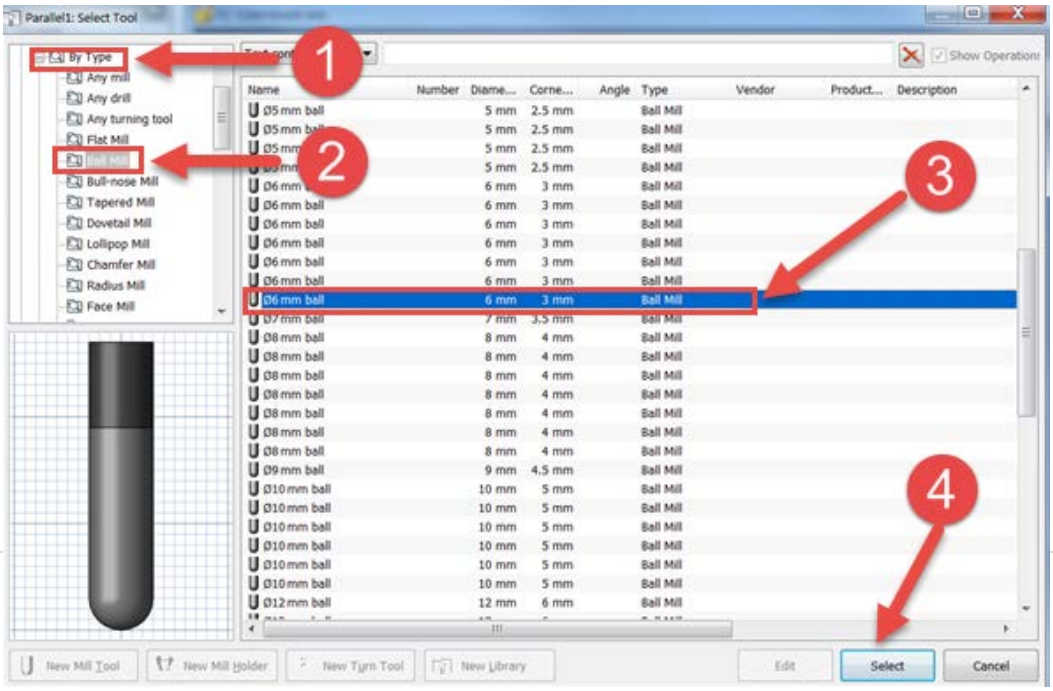
Step 3: Select the Cutter and customize it to cut an F1 Car

When the F1 model block is made out of foam, it can be cut with a single finishing pass, a roughing pass is not necessary. However, it requires an extra-long cutting flute on a ball nose cutter as it will cut though all the material at once. In the graphic, the standard end mill is placed next to the custom F1 in Schools Cutter, (the longer one). As the F1 in Schools cutter has a non-standard cutting length, you will need to create a new cutter in the system to use. Your instructor may have already defined the cutter. If so, you may select it from the list.

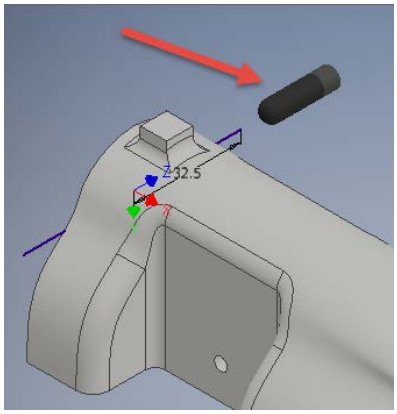
1. **Click the Tool button** and the tool library will appear.



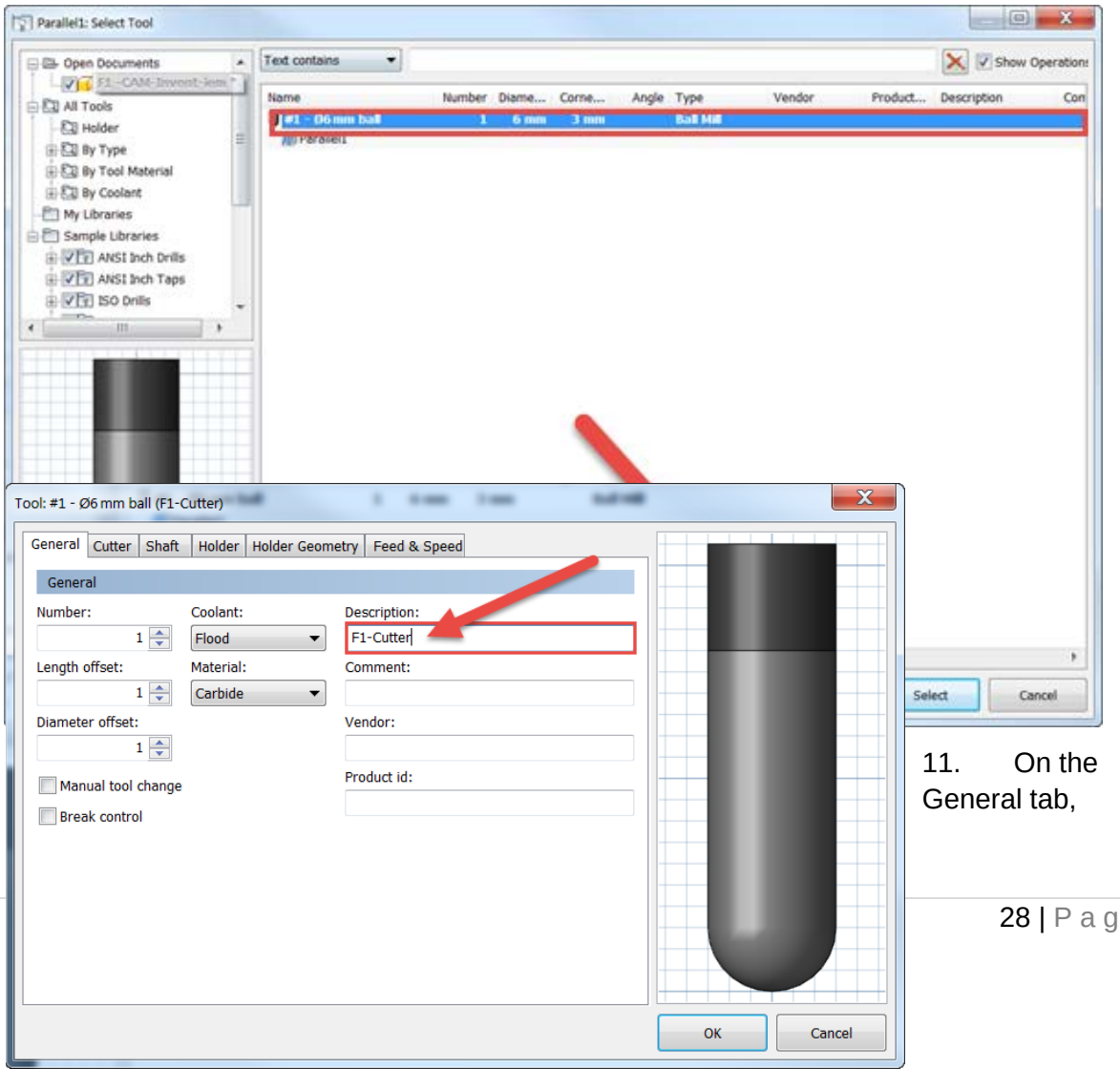
2. Next we will select a ball endmill that is close to what we need then we will modify it, and create a custom tool.
3. **Expand Tool by Type.**
4. **Select Ball Mill.**
5. Scroll down the list and select the last **06mm ball** tool.
6. **Pick Select** at the bottom and now a tool is selected.



7. The tool is now displayed in the correct orientation to cut the car from the Left side.



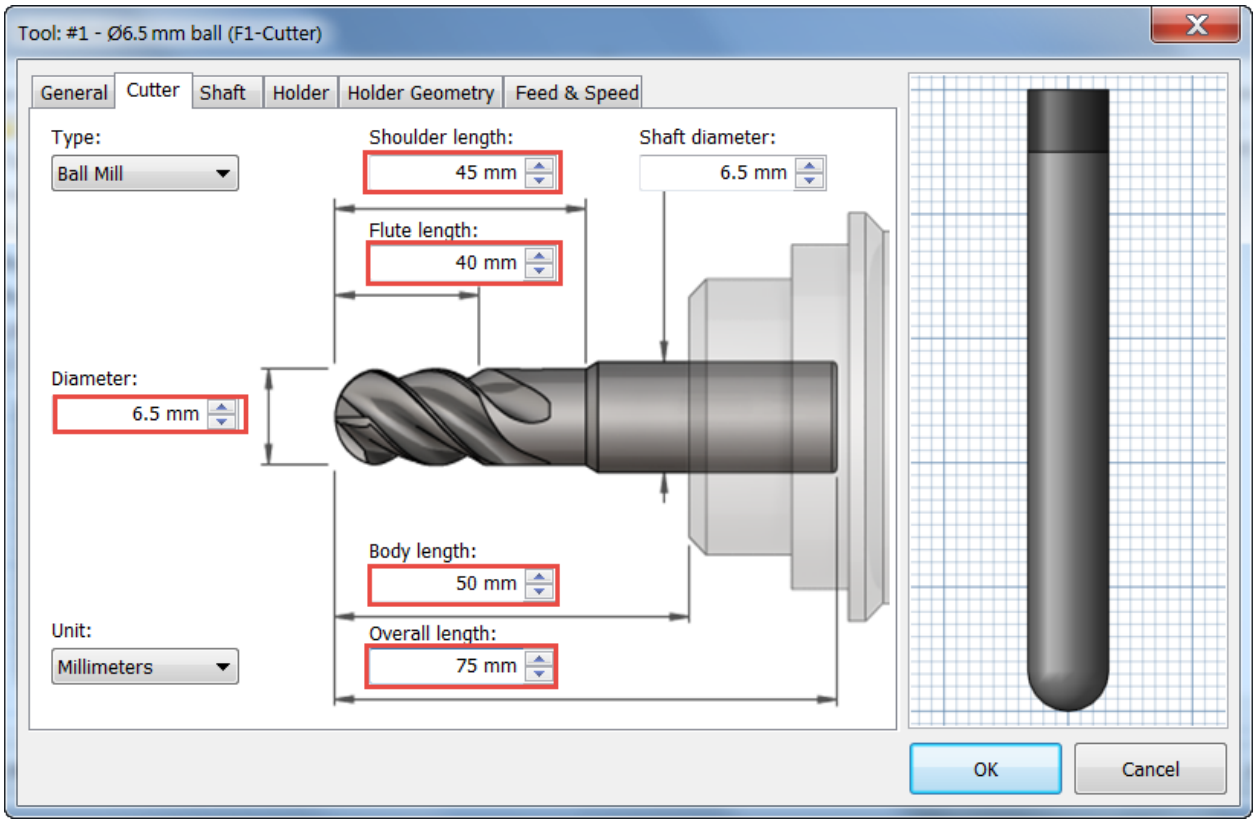
8. Next we need to customize the cutter to cut the F1 blank all in one pass, to accomplish this, the cutting surface needs to be 40mm long.
9. Edit the tool by **selecting the Tool button,** the tool associated to the parallel toolpath is highlighted
10. Click Edit



11. On the General tab,

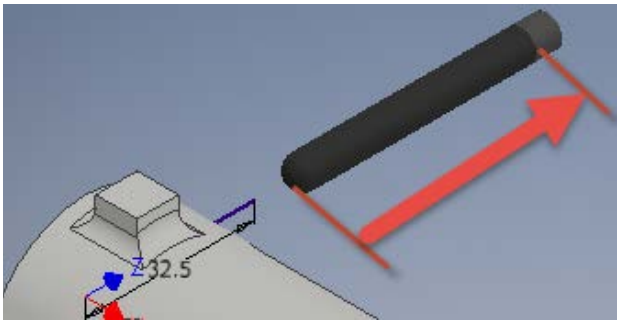
Description=F1-Cutter

12. **Select the Cutter tab, enter the following parameter fields**



13. **Select OK, then pick Select at the bottom of the dialog**

14. The new tool is displayed with a longer cutting length.

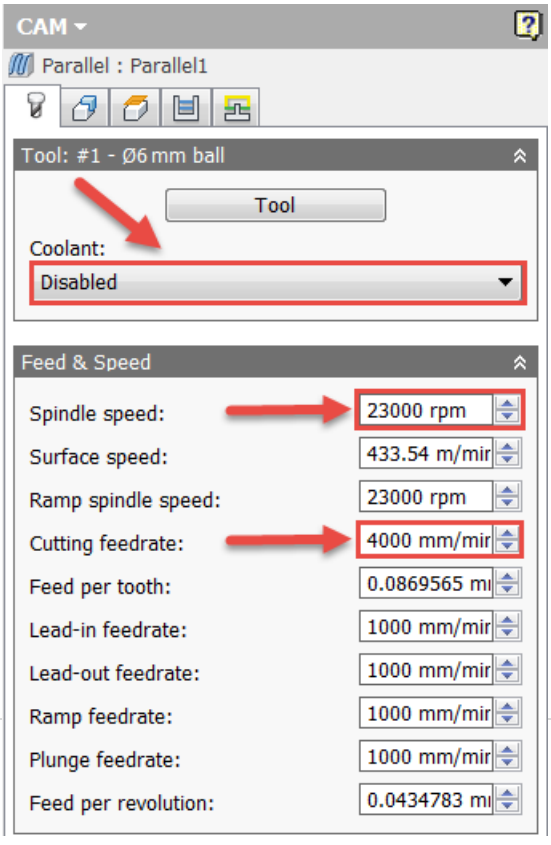


15. Click the Save Icon

Step 4: Define Machining Parameters for the Denford CNC Router

Machining and toolpath parameters can seem a little overwhelming as there are enough parameters to cut pretty much anything out there with any number of strategies. For our project, there are just a few parameters to change to give us an excellent toolpath. These parameters will be for a Denford Machine. If you have a different CNC machine, have your instructor write in the appropriate parameters.

1. Now back to the to the Parallel toolpath dialog, we will set the machining parameters for this tool and material.
2. **Change the following three fields for the Denford Router, then select the Geometry Tab**, *do not hit OK at the bottom of the dialog, until we complete filling out all of the tabs.*



Turn coolant off, coolant is not needed for wood or foam.

How fast the cutter will rotate

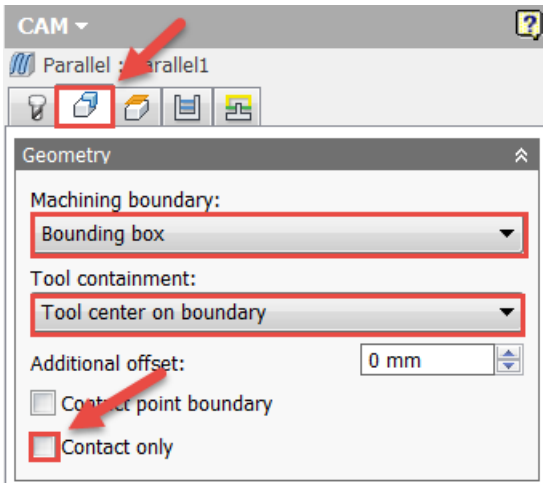
How fast the cutter will move through the stock

Step 5: Geometry Tab Containment of the Toolpath

On this tab we will contain the toolpath.

1. **Select the Geometry Tab.**
2. Three things to change here:
3. **Set the Machining boundary=Bounding Box.**
4. **Tool Containment=Tool center on boundary**

5. **Contact only=Uncheck.**



Bounding box includes the all of the stock.

Containment, Tool center means that the tool tip will move out to the center line of the bounding box or stock edge.

Note: This very important and must be selected as other options could allow for the cutter to pass down the sides of the Model Block blank and clash with the fixture, damaging the CNC machine.

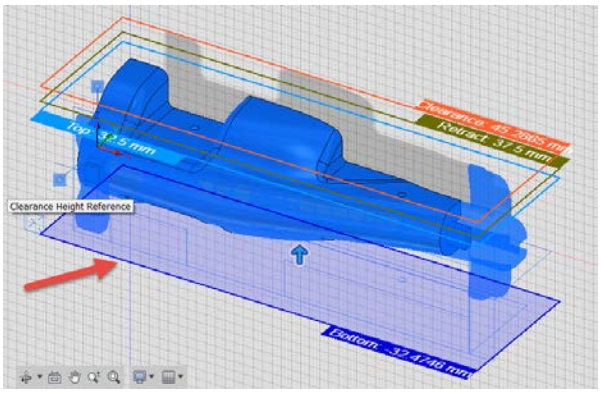
Contact only checked means, just cut the model surface, we want to machine away all of the stock, so uncheck

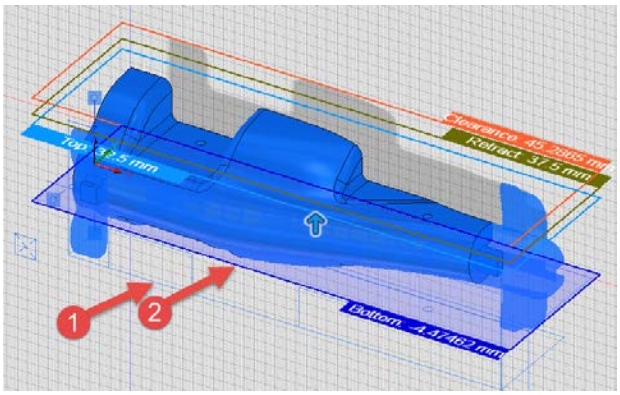
6. **Select the Heights Tab**

Step 6: Heights Tab set the machining depth

When we cut the left side of the car, we want to make sure that the ball end mill move past the exact center of the car by the diameter of the cutter. If it does not, then there would be a ridge of material that would need to be removed along the length of the car running down the center axis. The system defaults to the bottom of the stock. It needs to be changed to a little past the center.

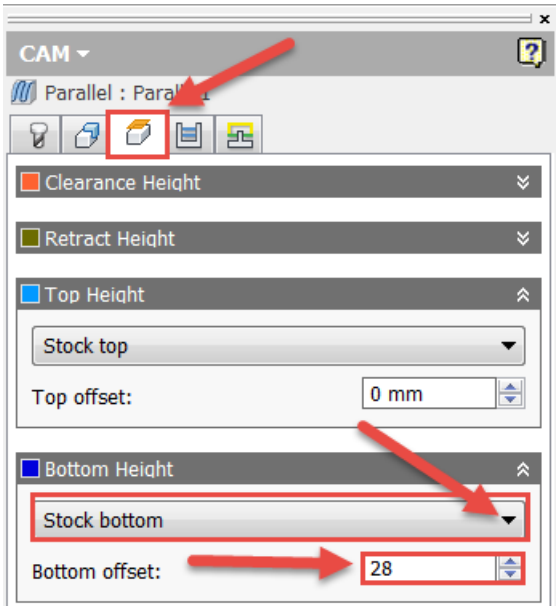
1. With the Heights Tab selected, the Heights dialog opens, **Select the Bottom Height Dropdown and select Stock Bottom** from the list and **Enter 28 for the offset.** This





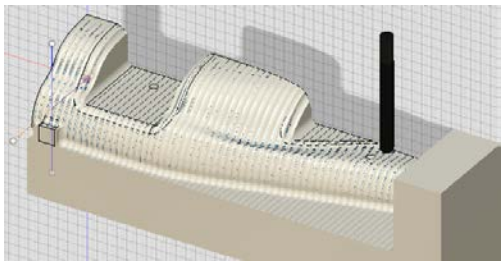
contains the toolpath, so that it cuts just a bit more than halfway through the stock.

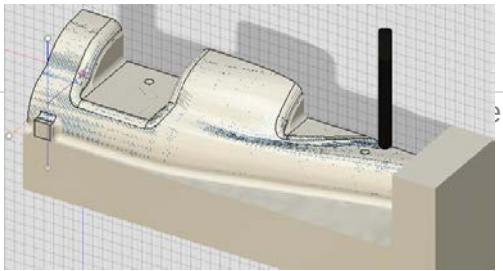
2. **Select the Passes Tab.**



Step 7: Passes Tab Set the Space between Passes and Toolpath Orientation

The spacing between cutter passes controls the surface finish of your F1 in Schools Car, so the closer together the step over, the smoother the surface, and the longer it will take to cut. Inversely, the wider the step over, the rougher the surface, and the faster it will cut. For a super smooth surface, a step over of 10% of the cutter diameter would be used. For a 6.5mm cutter, a tool stepover of 0.65 would produce a good

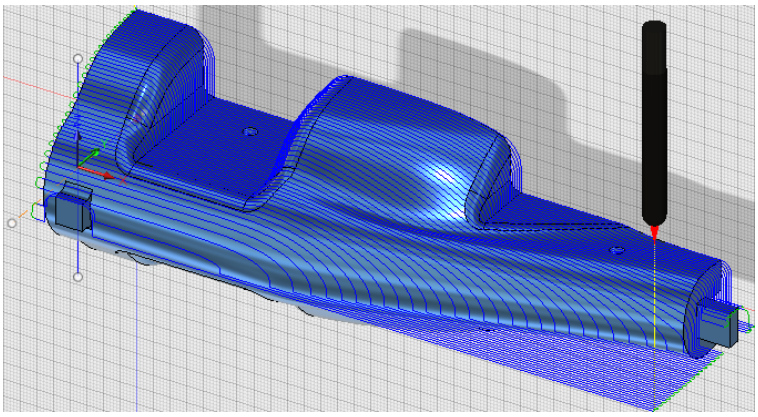




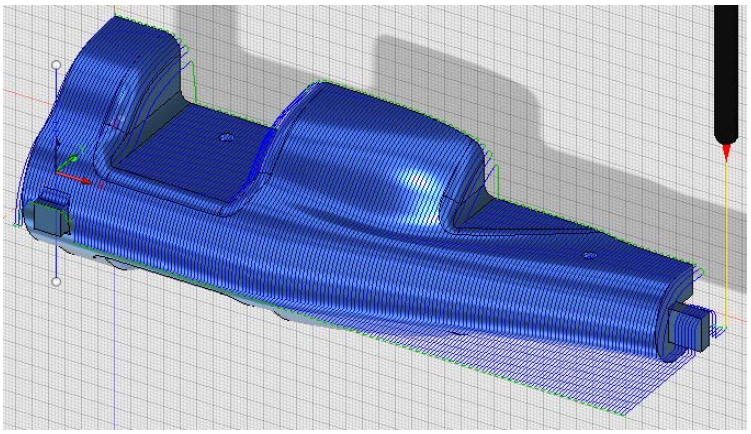
finish for competitive cars. On the Denford CNC Router l, it would take about 7-10 minutes to cut. For a quick but acceptable surface, one that would take only a little sanding, 15% of the cutter diameter would be selected.

Parallel Cutting Strategies: There are two ways F1 in Schools cars are cut, with the parallel toolpath running down the length of the car, or across the car. Balsa and foam across the car is chosen to prevent splitting. We would also want to cut from the back of the car to the front of the car, as the front of the car is only supported by a little peg.

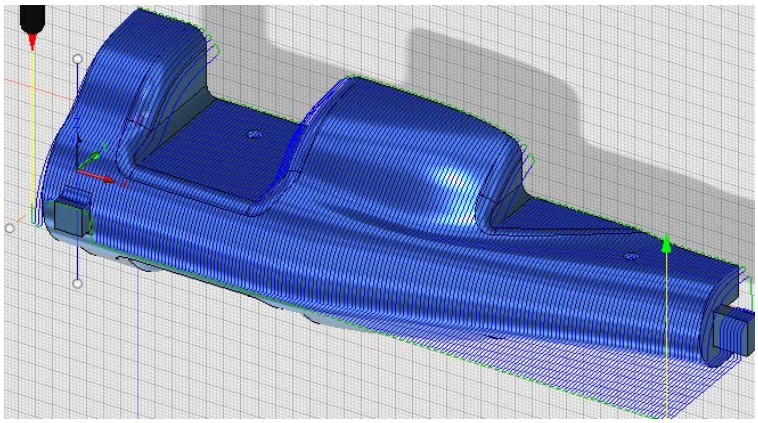
Pass Direction=0 degrees-cuts along the length, start at front



Pass Direction= +90 degrees cuts across car starts at front



Direction=minus 90 starts at back of car



FYI: ROUGHING AND FINISHING PASSES

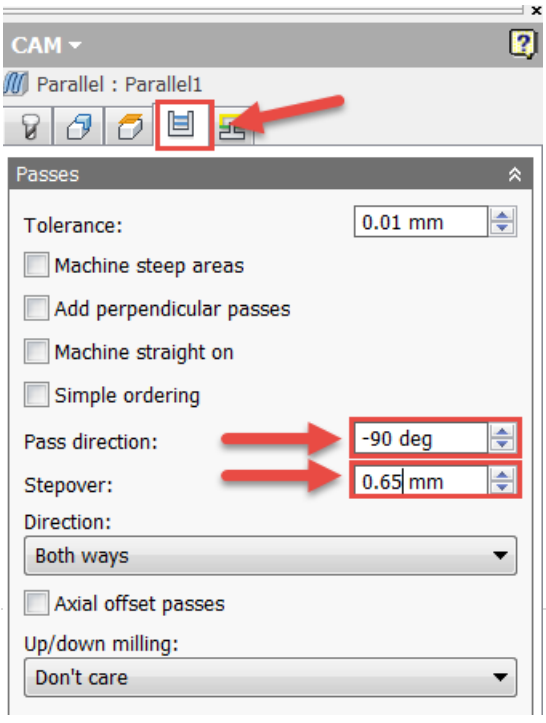
ON FOAM, SOME SCHOOLS ROUGH MACHINE ALL CARS FIRST IN THE 0 DIRECTION, AT 50% STEPOVER TO QUICKLY REMOVE THE MAJORITY OF THE MATERIAL AND THEN GO BACK IN AT 90 ON A FINISHING CUT AT A 10% STEPOVER. THIS TAKES LONGER BUT REDUCES THE CHANCE OF BREAKING OFF THE JIG.

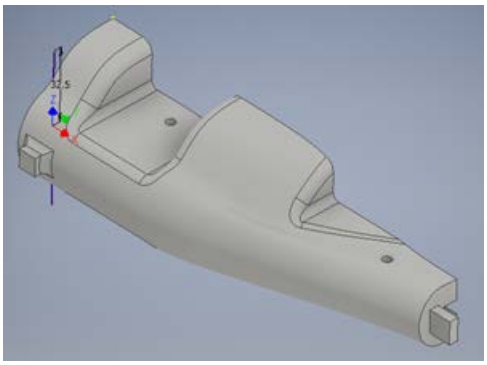
IN ADDITION, MORE ADVANCED SCHOOLS ALSO LIKE TO USE A 4MM BALL NOSE CUTTER

IF THERE IS A CUTTER CHANGE RADIUS INTERNAL PART JOINTS (WHERE PRINTED PARTS MEET THE BODY) TO HALF THE DIAM OF THE NEW TOOL.

There are just two parameters to change:

1. **Pass Direction= -90 deg**
2. **Stepover= 0.65mm**
3. **Select the Linking Tab**

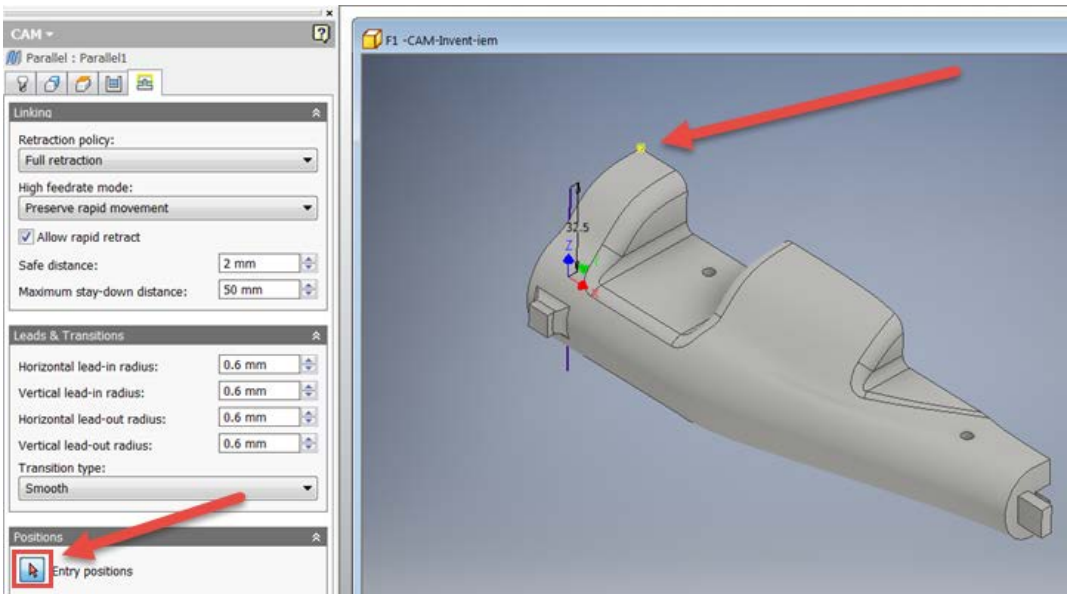




Step 8: Linking Tab

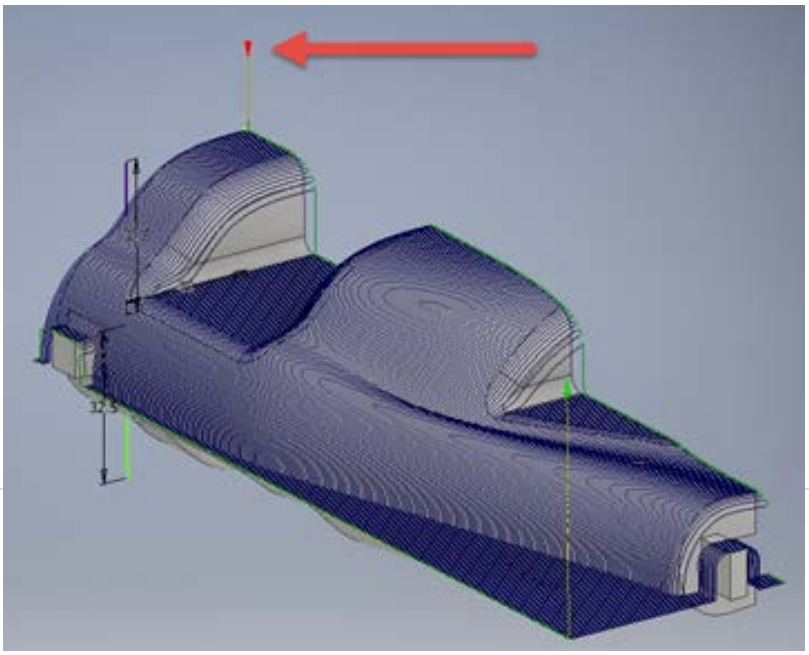
We want the toolpath to start at the back bottom corner of the car

1. **Select Entry Position button**, then **Select the back top corner of the model**



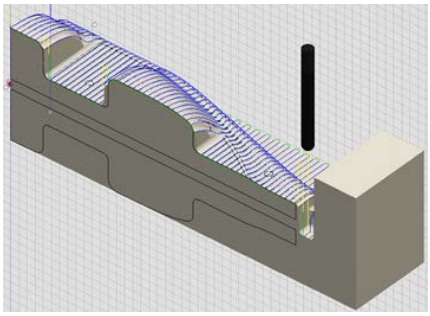
2. **Select OK** at the bottom of the dialog. All of the parameters for the Left side toolpath have been set. The toolpath will generate. This will take a few seconds and your toolpath will be displayed. It should look like the graphic below
3. **Rename the toolpath to F1-Left**

Note: If your toolpath does not look like this, recheck your parameter fields in each tab by right clicking on the toolpath and selecting Edit



Activity 6: Create the Right Side Toolpath

Applying the toolpath to the right side will be very easy as the car is bilateral. In Inventor HSM, you can capture all the information that was input for the left side of the car and apply it to the right side. We will copy the current toolpath and paste it back in the CAM Browser as a new toolpath. Then, simply change the Z axis orientation and the toolpath is complete.

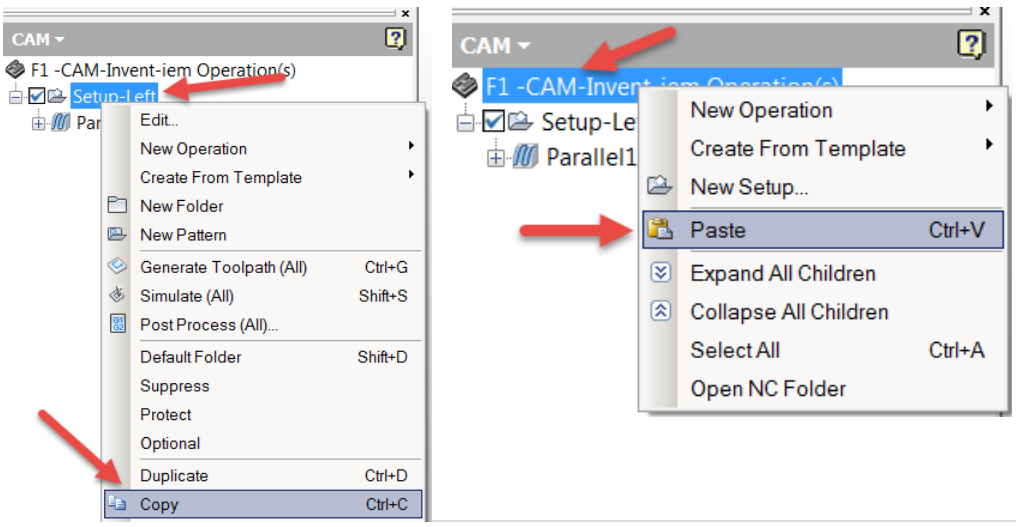


You will:

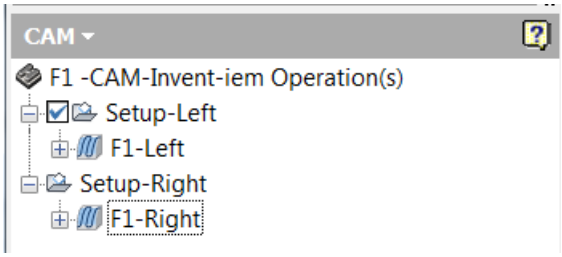
- Copy and paste the setup and toolpath for the Left side of the car to create the Right side of the toolpath.
- Change the orientation of the WCS to cut the Right side of the car by flipping the Z axis.

Step 1: Copy and Paste the existing Setup and Toolpath

1. **Right Click on Setup Left** in the Browser
2. **Select Copy**
3. **Right Click on F1-CAM Invent-Operations and select Paste**

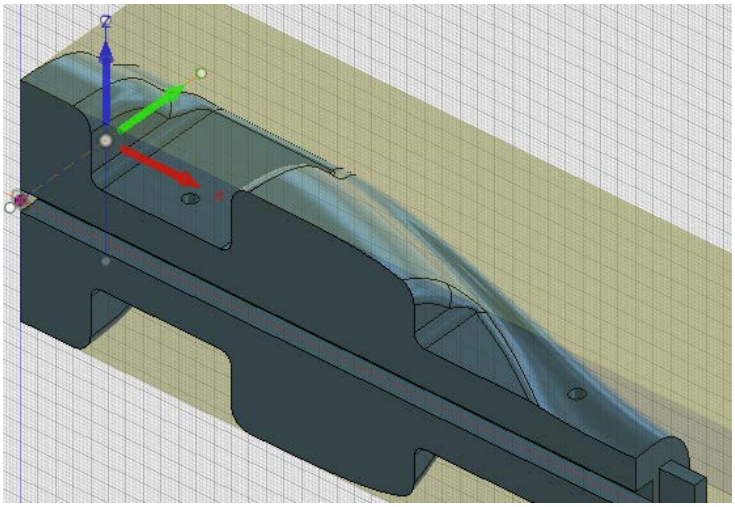


4. Now there are two Setups in the Browser, with one toolpath each
5. **Rename the second setup to Setup-Right**
6. **Rename the second toolpath to F1-Right**

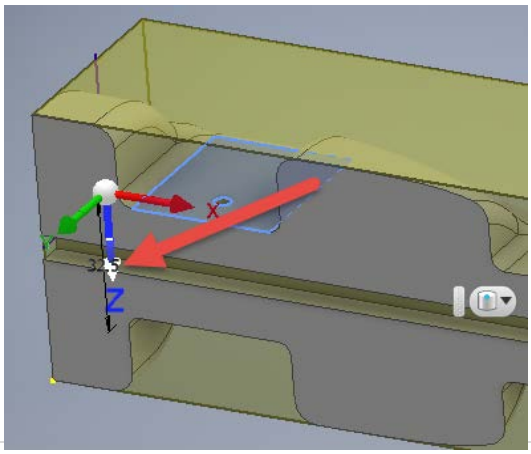


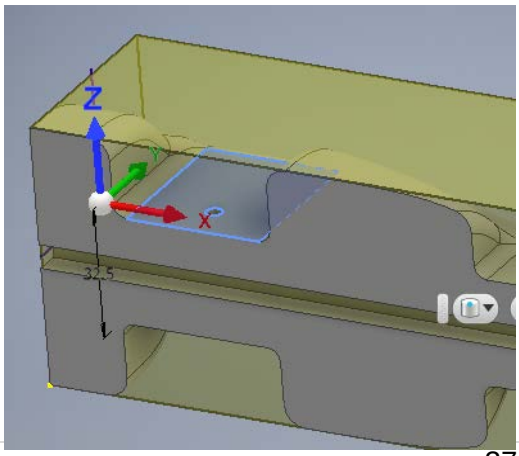
Step 2: Change the Z Axis Orientation of the Second Setup

When the toolpath was copied, so was the orientation, since we need to flip the car in the fixture 180 degrees to cut the other side. The Z axis also needs to be flipped 180 degrees for the toolpath to be in the correct orientation.

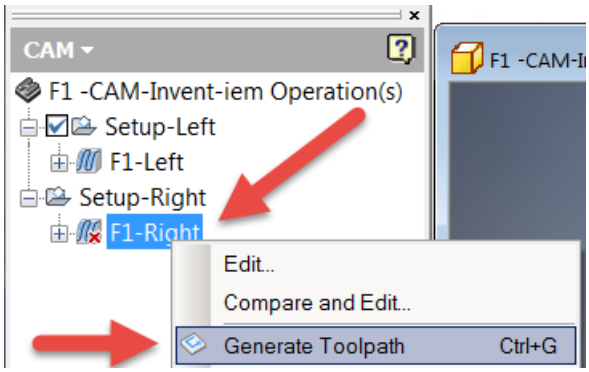


1. Rotate the car to this orientation.
2. **Right Click on Setup-Right and select Edit.**
3. **Click on the Z axis arrow tip to flip the Z axis, it should now be pointing down.**
4. **Click OK.**

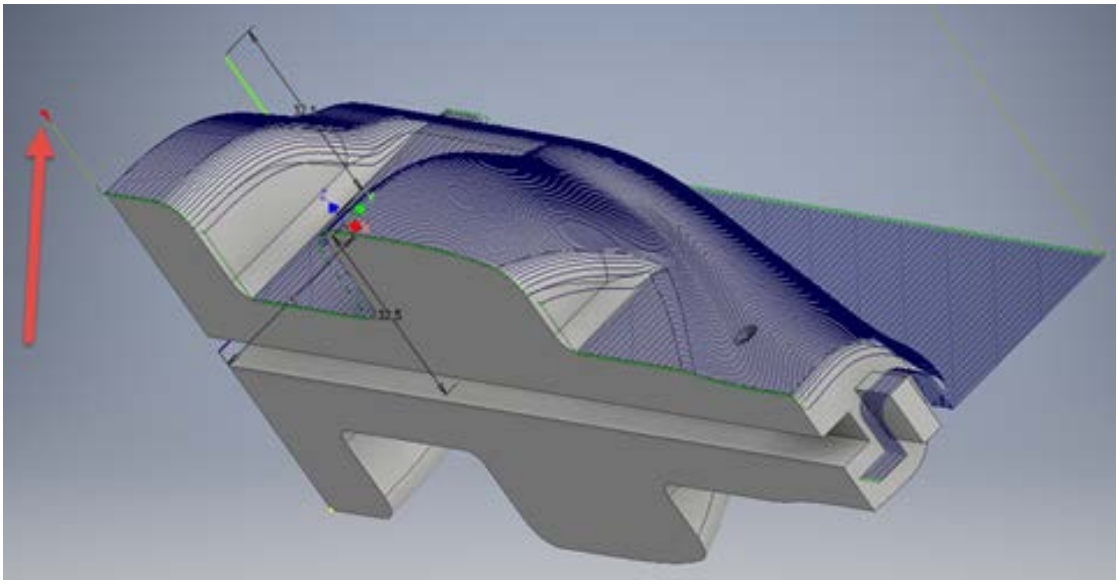




5. The Setup has changed so the toolpath is now shown as dirty; **right click on the second toolpath F1-Right.**
6. **Select Generate Toolpath** to regenerate the toolpath, the toolpath is now on the right, side of the car.

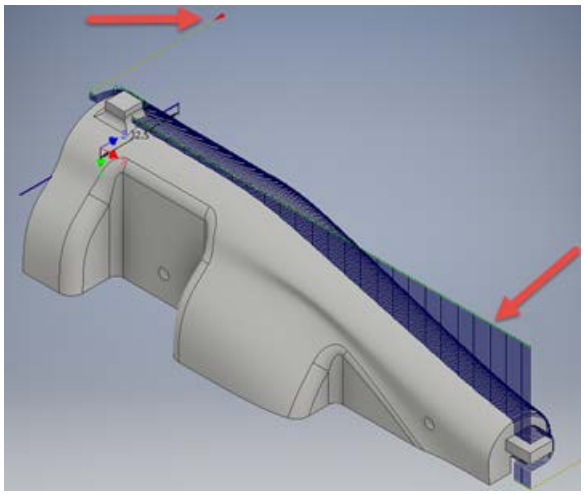


7. This is how the second toolpath would look.

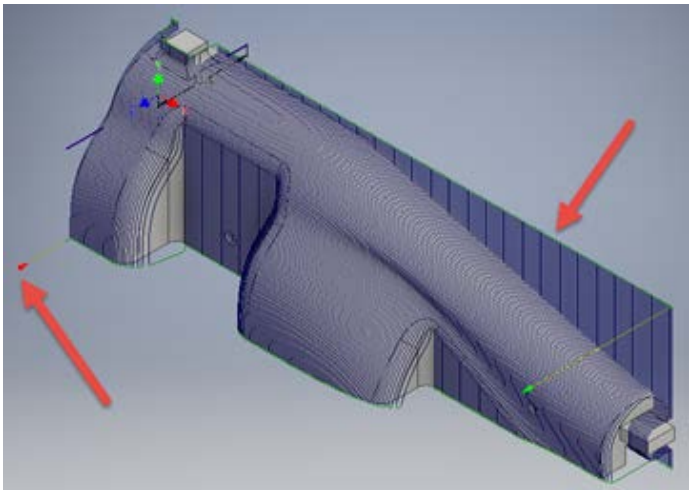


8. **Check the toolpaths against each other.**
9. **Rotate into the Home/Isometric view.**
10. **Select the first toolpath Left side**. Is the tool on the top of the model? Does it start on the top back corner? Does it cut all of the stock?
11. For the Right Side, is the tool coming from the bottom? Does it start on the back corner? Does it cut all of the stock? If the toolpaths match these criteria, move to the next step. If not check with your instructor.

Toolpath 1 Left Side, on top



Toolpath 2 Right Side, on bottom



12. **Save your file**

Activity 7: Simulate the Toolpath

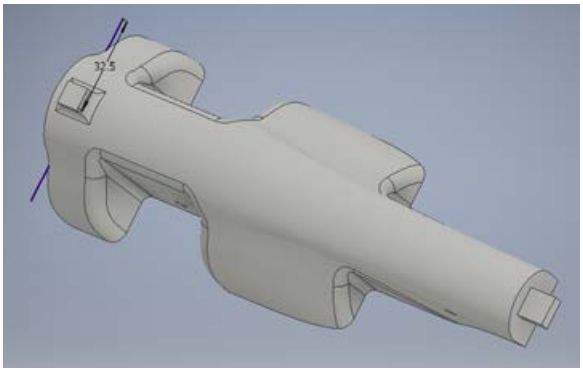
Inventor has the ability to virtually machine the model, use this functionality to check to see if your toolpath is doing everything correctly.

You will:

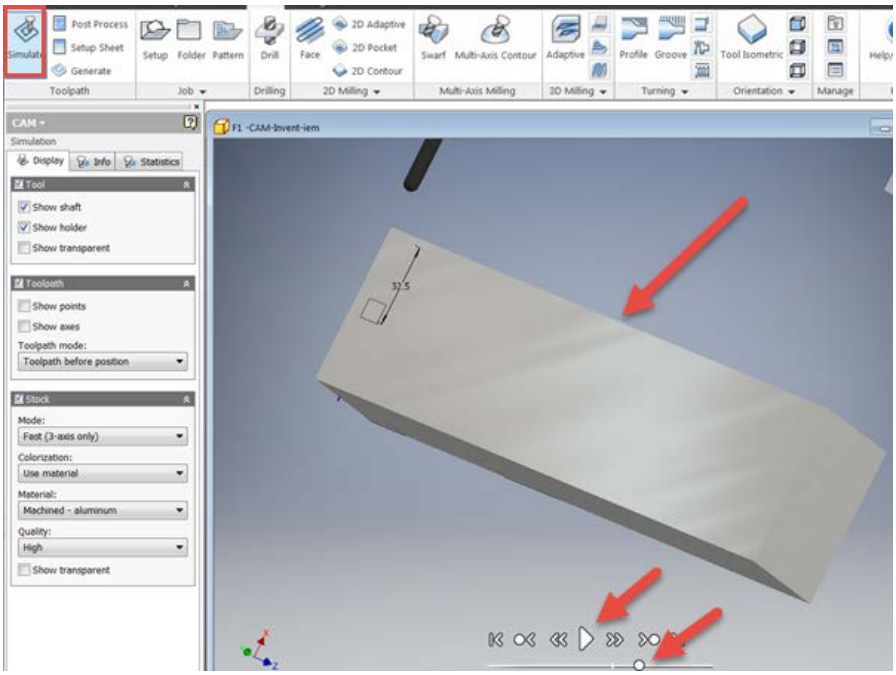
- Setup the simulation tool options
- View the car being cut from both sides.

Step 1: Select the toolpaths and Simulate

1. **Rotate the model into this view**

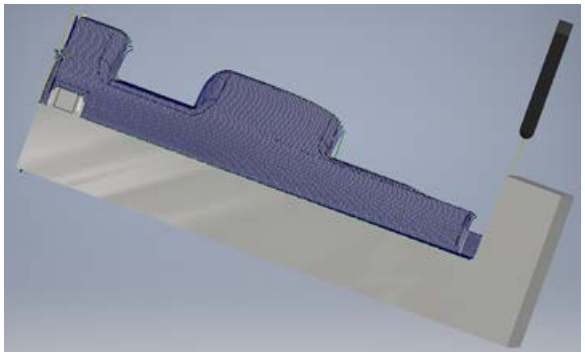


2. **Select the first toolpath F1 Left**
3. **Select Simulate from the CAM menu bar**
4. The Simulate menu will open and the simulation controls will be displayed at the bottom center of the screen
5. **Select the play Arrow**, the playback speed is the large white dot.
6. Rotate and Zoom in an out while the model is verifying to see what is really happening.

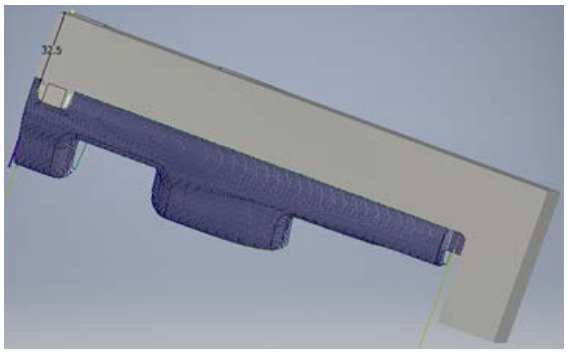


7. **Click the Close button at the bottom of the Simulation dialog**
8. **Select the second toolpath and simulate, select the Start Simulation Arrow**
9. **Select both toolpaths, by holding down the Ctrl Key and select both toolpaths and simulate**, **use the round dot lower left in the control to speed up the playback.** While the simulation is playing rotate, zoom in and out to see what is really happening.
10. Experiment with different materials.
11. **Click Close from the dialog when finished.**

Left Side Toolpath



Right Side Toolpath



Activity 8: Select the Post Processor Generate the NC Code

The toolpath is complete, checked, and simulated - no shortcuts here! A bad toolpath can cause damage to the tool, stock, and machine, and also can be dangerous for the operator.

Next, we will generate the NC-Code. This is the easiest of all our activities. Simply select the post processor and this automatically generates the code. The CAM system generates neutral moves, and the post processor converts them to machine specific code. In our case, we will select a post processor that will generate code specifically for a Denford CNC Router. The main thing to remember here is where you save the NC-code, as you will need to retrieve it to load onto the CNC machine.

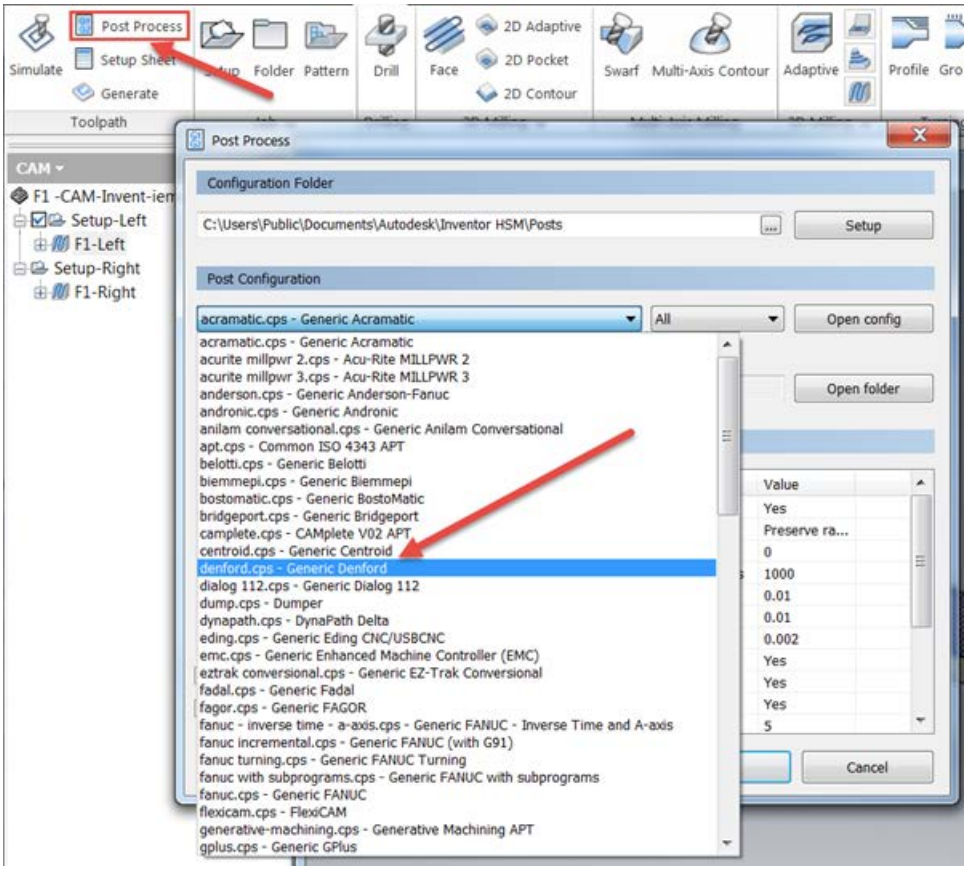
You will:

- Select the Denford Post Processor
- Generate the NC-Code For each toolpath

```
(1001)
(left side up)
(T1  D=6.35 CR=3.175 - ZMIN=-4.475 - ball end mill)
[Billet X212 Y50 Z65]
N10 G90 G94
N15 G17
N20 G21
N25 G28 G91 Z0
N30 G90
(F1-Left)
N35 M9
N40 T1 M6
N45 S2300 M3
N50 G54
N55 M8
N65 G0 X0.003 Y29.621
N70 G43 Z47.5 H1
N75 Z34.437
N80 G1 Z33.109 F1000
N85 G19 G2 Y29 Z32.475 J-0.635
N90 G1 Y27.005 Z32.432 F4000
N95 Y25.445 Z32.378
N100 Y23.904 Z32.303
N105 Y22.378 Z32.201
N110 Y20.855 Z32.068
N115 Y19.715 Z31.943
N120 Y18.2 Z31.742
N125 Y17.445 Z31.621
N130 Y16.691 Z31.493
N135 Y15.567 Z31.274
N140 Y14.448 Z31.022
N145 Y13.708 Z30.836
```

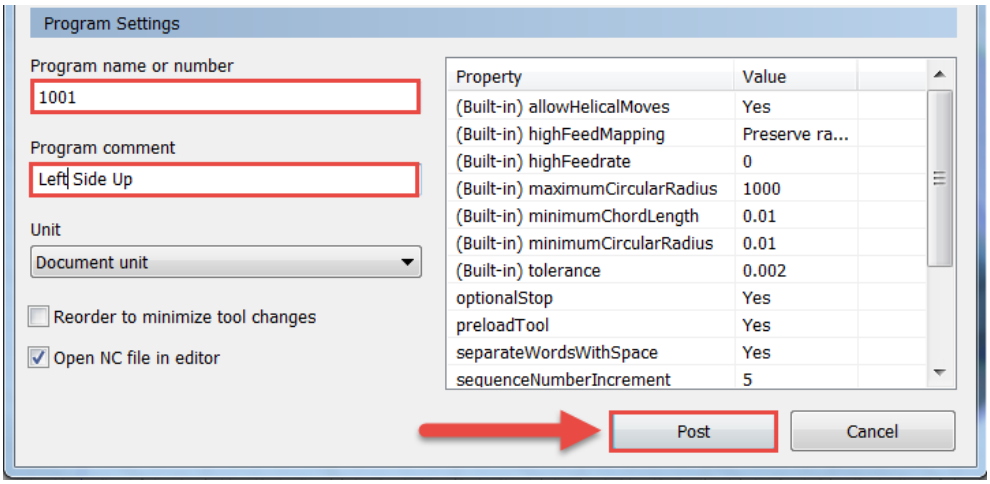
Step 1: Select the Post Process and Generate the NC-code for the first toolpath.

1. **Select the First toolpath F1 Left.**
2. **Select Post Process Icon from the CAM Toolbar.**
3. The Post Processor dialog will open.
4. **Select the down arrow and the post list will display. Scroll down and select the Denford.cps post.**



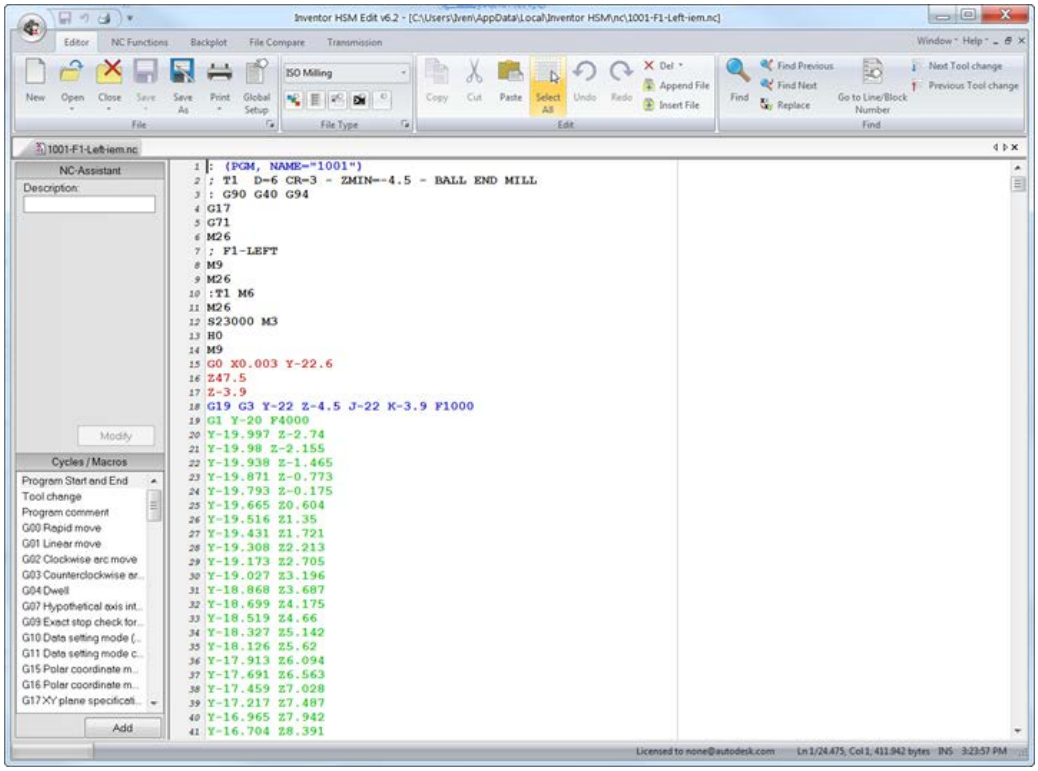
5. **Under the Output Folder header, Select the square with three dots and create a path to where you want to store the toolpath, write this down.**

7. **Enter the progam number=1001, and Program Comment=Left Side up, select Post.**



8. **Enter 1001-F1-Left (your initinals) as the file name, write down where you sent the file, its extension will be (.fnc)** if you need to search for it.

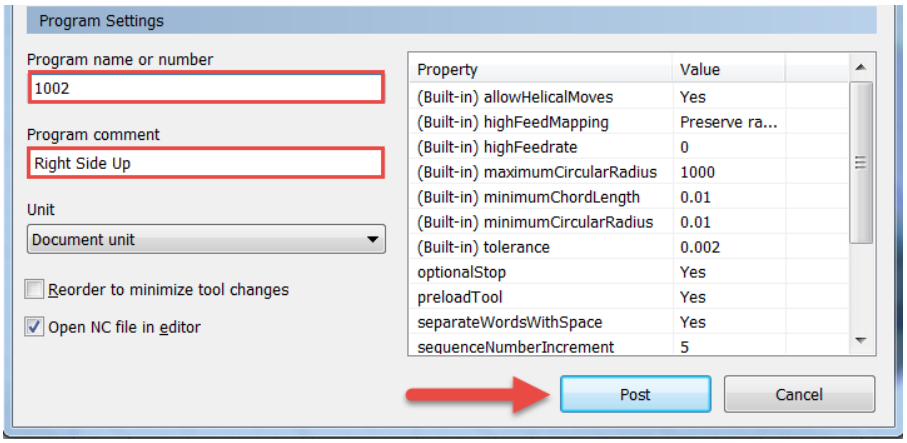
9. The NC-code editor will open and the NC-Code wil be displayed for your review.



10. **After reviewing the code select the X at the top right corner of the NC-Code editor to dismiss the editor.**

Step 2: Generate the NC-code for the second toolpath Right Side.

1. Generating the second toolpath will be very quick as we have set up all of the parameters.
2. **Select the second toolpath F1-Right.**
3. **Click the Post Process Icon.**
4. **Change the program number to 1002.**
5. **Program Comment to Right Side Up. Click Post.**
6. **Click Post.**



7. **File Name=1002-F1-Right(your initials) (leave no open spaces in the name)**
8. The NC-Code editor will open, check out your code
9. **Exit the NC-Code Editor**

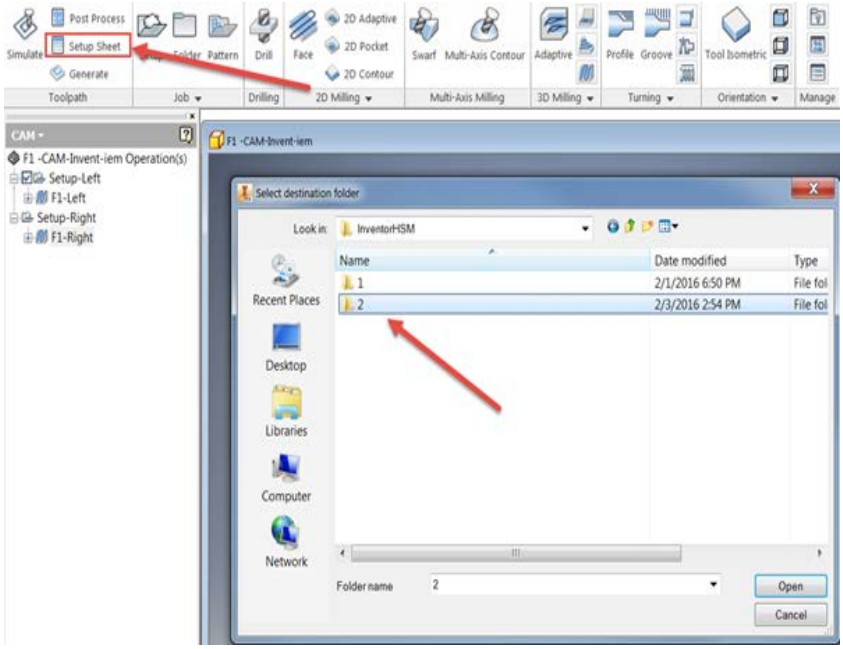
Activity 9: Generate the Setup Sheet

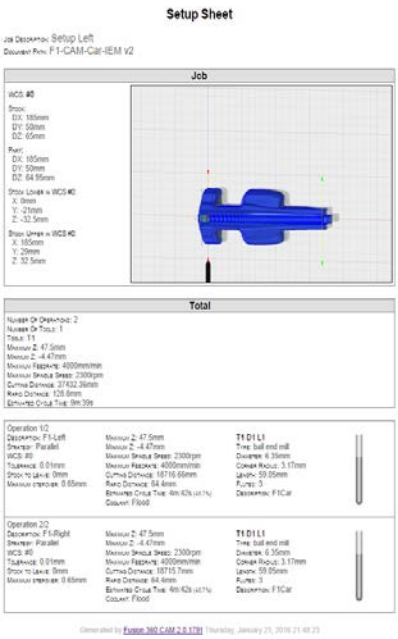
Inventor HSM can automatically document your setup, the setup sheet will help you and your instructor setup the CNC machine to cut your car.

You will:

- Create a setup sheet, and save it.
- Print the setup sheet.

1. **Select Setup Left**
2. *Note: The setup sheet will capture the screen, best practice is to create a view that represents the appropriate setup with toolpaths before entering setup.*
3. **Select Setup Sheet from the CAM toolbar**
4. **Select the folder to save the file to, yours will be different, pick Select Folder.**





5. The Setup sheet will be generated.
6. The Setup Sheet will be displayed.
7. Print the setup sheet.
8. Repeat process for Setup-Right

Setup Sheet

Job Description: Setup Left
Document Path: F1-CAM-Car-IEM v2

Job

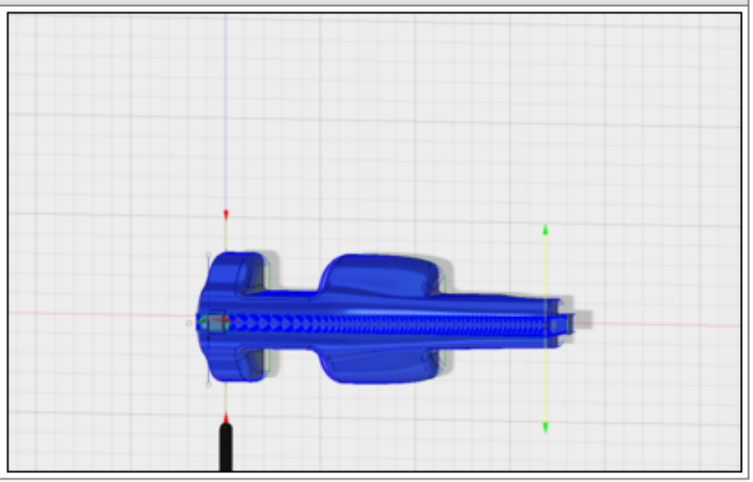
WCS: #0

Stock:
DX: 185mm
DY: 50mm
DZ: 65mm

Part:
DX: 185mm
DY: 50mm
DZ: 64.95mm

Stock Lower in WCS #0:
X: 0mm
Y: -21mm
Z: -32.5mm

Stock Upper in WCS #0:
X: 185mm
Y: 29mm
Z: 32.5mm



Total

Number Of Operations: 2
Number Of Tools: 1
Tools: T1
Maximum Z: 47.5mm
Minimum Z: -4.47mm
Maximum Feedrate: 4000mm/min
Maximum Spindle Speed: 2300rpm
Cutting Distance: 37432.36mm
Rapid Distance: 128.8mm
Estimated Cycle Time: 9m:39s

Operation 1/2
Description: F1-Left
Strategy: Parallel
WCS: #0
Tolerance: 0.01mm
Stock to Leave: 0mm
Maximum stepover: 0.65mm

Maximum Z: 47.5mm
Minimum Z: -4.47mm
Maximum Spindle Speed: 2300rpm
Maximum Feedrate: 4000mm/min
Cutting Distance: 18716.66mm
Rapid Distance: 64.4mm
Estimated Cycle Time: 4m:42s (48.7%)
Coolant: Flood

T1 D1 L1
Type: ball end mill
Diameter: 6.35mm
Corner Radius: 3.17mm
Length: 59.05mm
Flutes: 3
Description: F1Car

Operation 2/2
Description: F1-Right
Strategy: Parallel
WCS: #0
Tolerance: 0.01mm
Stock to Leave: 0mm
Maximum stepover: 0.65mm

Maximum Z: 47.5mm
Minimum Z: -4.47mm
Maximum Spindle Speed: 2300rpm
Maximum Feedrate: 4000mm/min
Cutting Distance: 18715.7mm
Rapid Distance: 64.4mm
Estimated Cycle Time: 4m:42s (48.7%)
Coolant: Flood

T1 D1 L1
Type: ball end mill
Diameter: 6.35mm
Corner Radius: 3.17mm
Length: 59.05mm
Flutes: 3
Description: F1Car

Generated by Fusion 360 CAM 2.0.1791 Thursday, January 21, 2016 21:48:23

Activity 10: Enrichment Optimization-Cut the Car Faster

In this activity, you will learn how to smooth a toolpath so it will run faster on some machines.

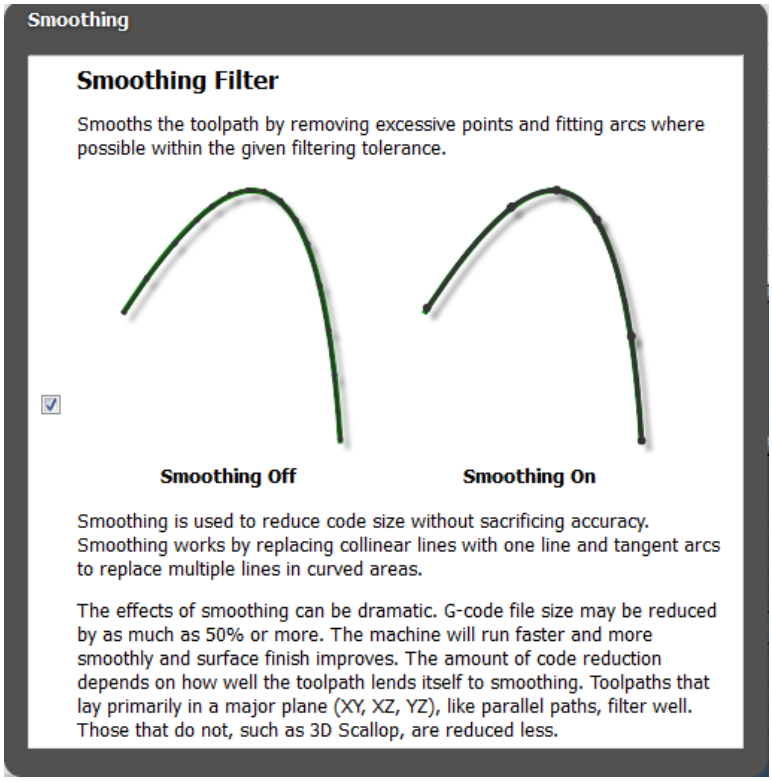
You will:

- Understand how toolpaths are created
- Smooth a toolpath to run faster

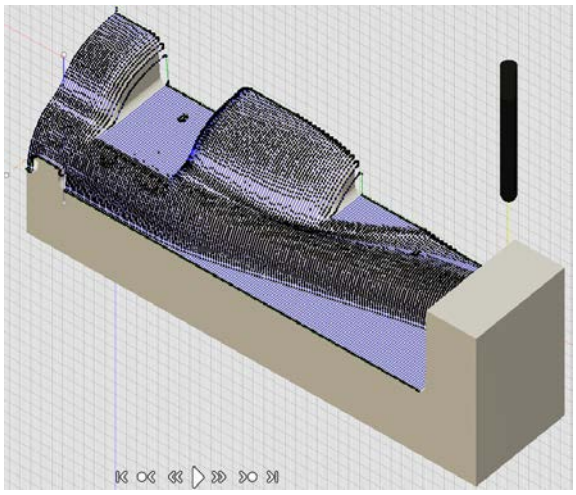
The CAM system resolution is set very high to tackle high precision parts in manufacturing. If we adjust this resolution down, it will allow us to cut cars faster. For more in-depth information on smoothing, see diagram (right).

One can increase the smoothing parameter to a point where the surface finish degrades.

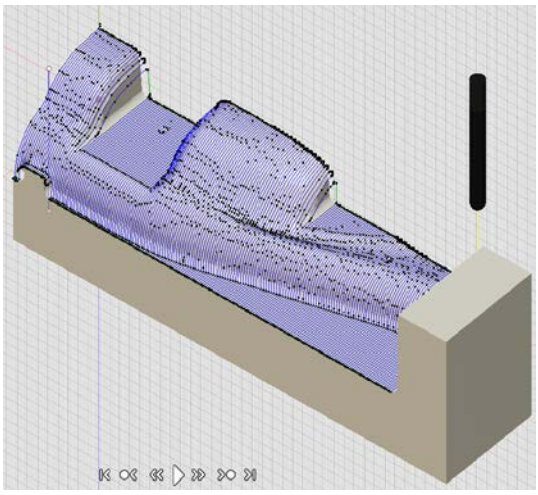
This concept is best understood when explained with a practical example and graphics. In the below graphics, the first graphic uses the default tolerance of 0.01mm, the second toolpath uses a large one of 0.04mm and a smoothing of 0.03mm. Notice the difference in point density. Each point defines a single line of code.



Tolerance set to 0.01mm

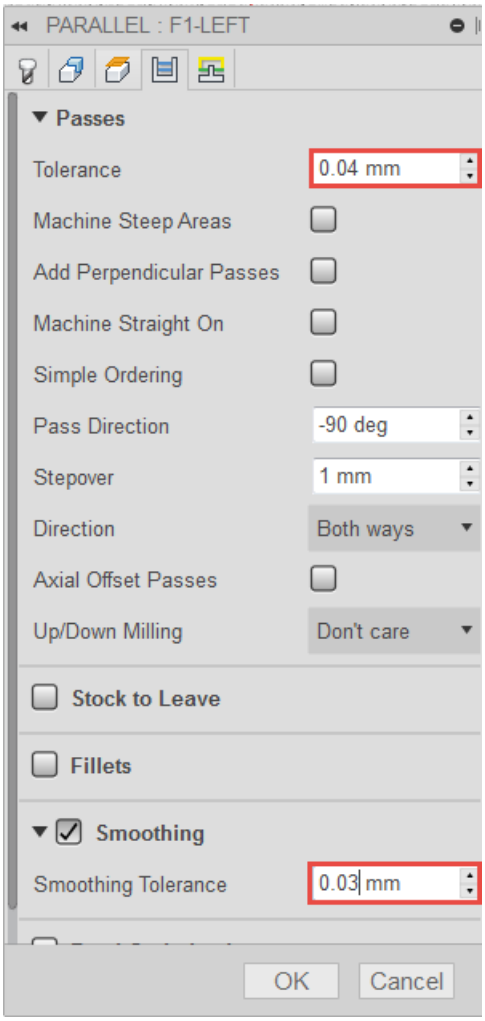


Tolerance 0.03mm



Note: *The Denford machines have Microstepping drives fitted to the axes and have a move buffering feature which allows moves to be read in advance and then changes the axis speed in accordance with the requirement of acceleration and deceleration to get the optimum performance, thus there will be little to no time savings on a Denford CNC Mill.*

1. To change the smoothing tolerance, simply right click on the toolpath and select the passes tab and change these parameters. The smoothing tolerance should be less than the tolerance parameter.
2. Don't forget to change the second toolpath as well, these would be good numbers to start with. Experiment to find the optimal tolerance. Good luck!



Credits

Tutorial Author: Iven May, Autodesk CAM Education Program Manager, Autodesk.

Outline Author/Review Paul Bray, Schools Projects Manager and F1 in Schools Specialist, Yas Marina Circuit

Technical Lead/Reviewer: Anton Fedoseyev, Premium Support Technical Lead, Autodesk

Partner Manager: Lynn Austin, F1 in Schools Partner Manager, Autodesk

Content Development: Jessica Bendy, Content Development Manager, Autodesk