



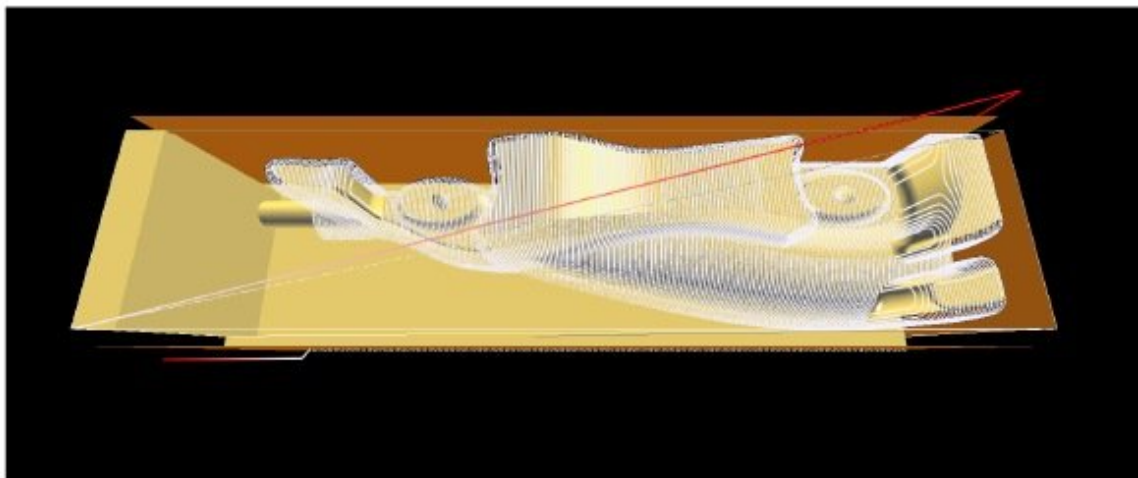
4th Axis - Side Machining - QuickCAM Pro - F1 in Schools Car Tutorial

Tutorial to use QuickCAM Pro to generate a toolpath for an F1 in Schools Car.

Written By: REA



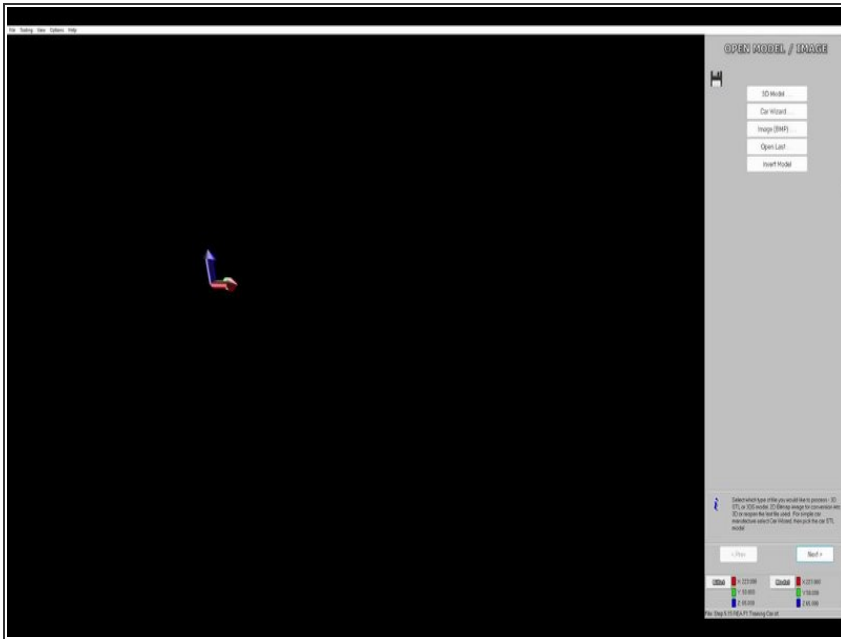
QuickCAM Pro F1 in Schools Car Machining Tutorial 4th Axis - Side Machining



INTRODUCTION

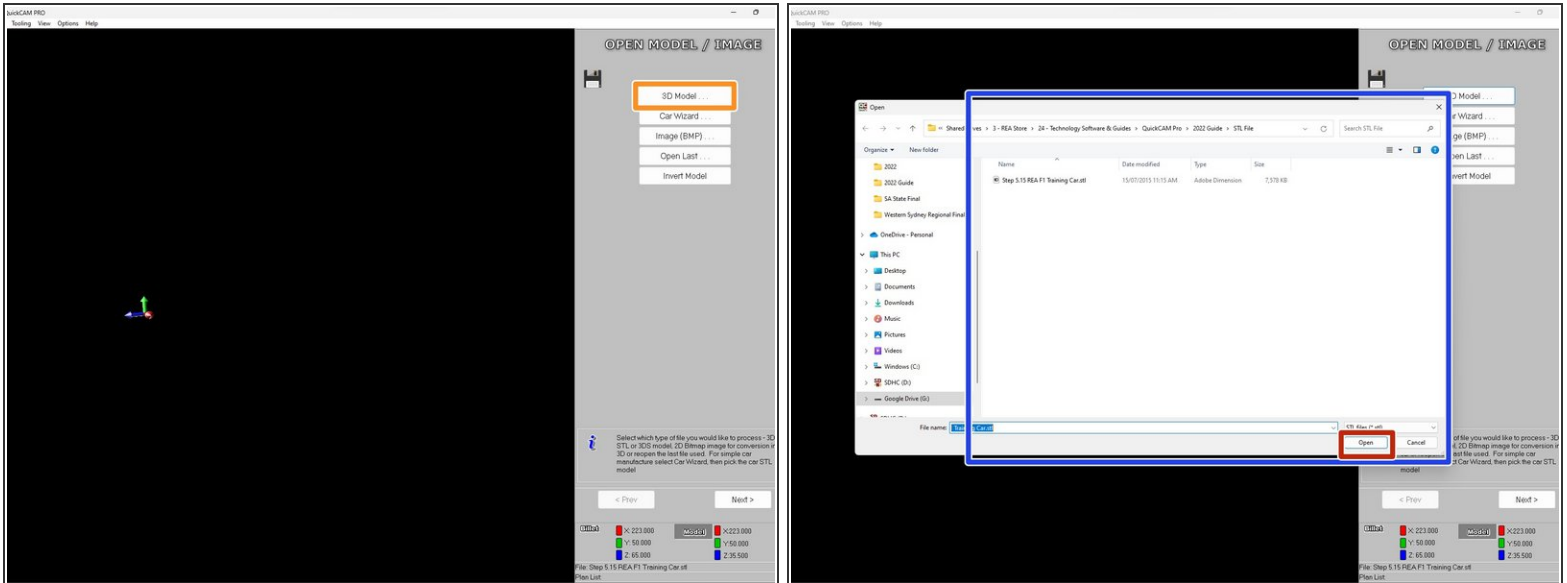
This guide is showing the procedure for the producing a code for an F1 in Schools car on a Denford Compact 1000 or 2600 with 4th Axis Fixture fitted.

Step 1 — Video Run Through



- ❗ Here is a screen recording going through the whole process.
- ❗ This may help if something didn't make sense in the steps below.

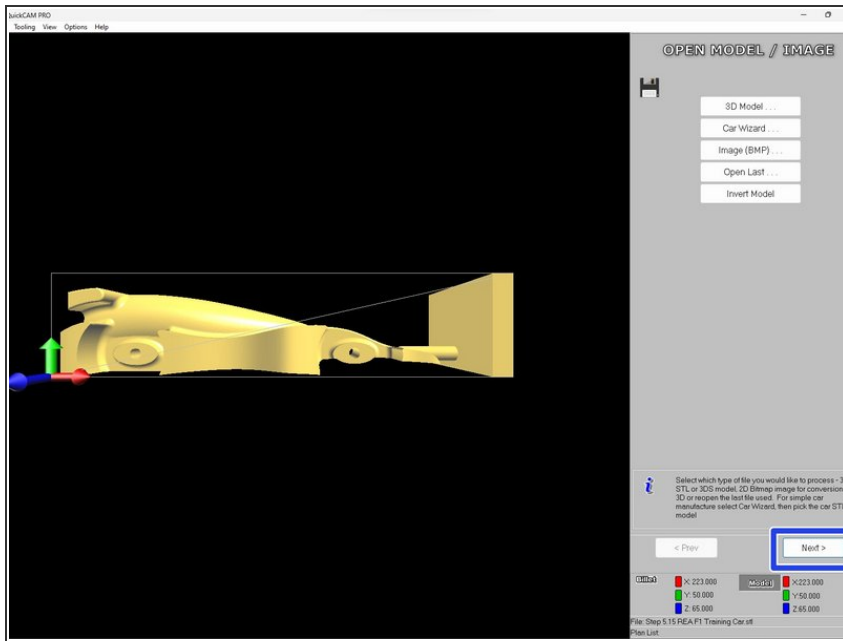
Step 2 — Opening Model



Open QuickCAM Pro

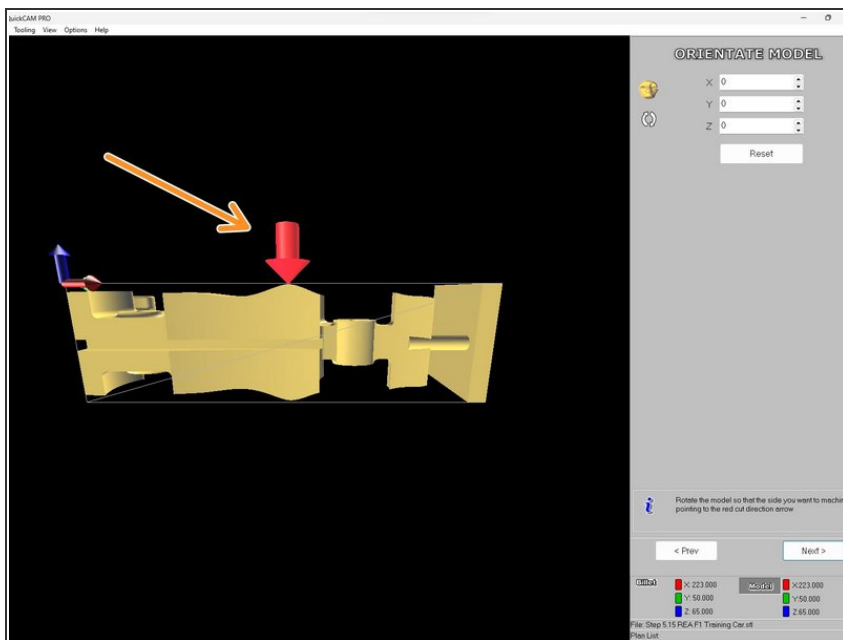
- You will see this screen once it has opened.
- We now want to open the STL file of the car we want to cut. Select the 3D model button.
- A window will pop-up asking you to select the STL file you want to open.
- Once you have selected the STL file press the open button.

Step 3 — Confirming Model



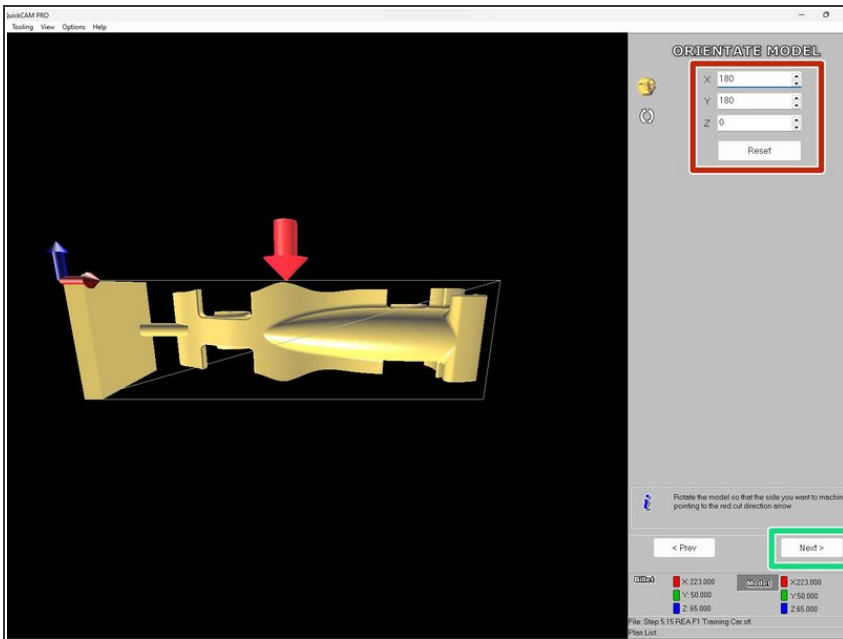
- The STL model of the Car you selected will open.
- You can click and hold the left mouse button down and move the mouse around to rotate the model.
- Confirm this is the car you want to create the .fnc file for and click on next

Step 4 — Orientate Model



- ⚠ This step is one of the most important steps to set correctly.
- Rotate the view around until you get the red arrow pointing down as shown.
- ⓘ The arrow represents the cutter in the CNC Router, so orientating the view like this means it is like us standing in front of the machine and looking through the door of the machine.
- We now need to orientate the car to match how it is orientated inside the CNC Router.

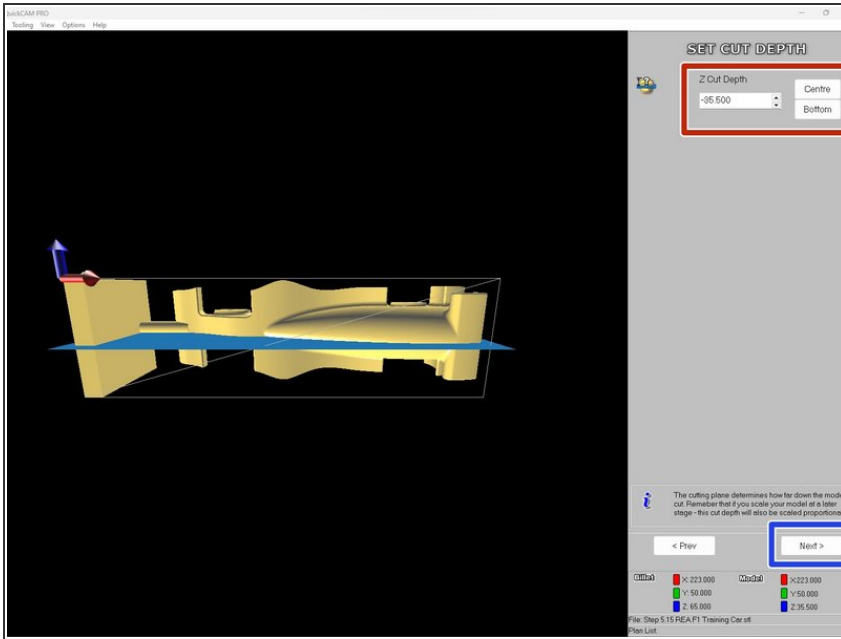
Step 5 — Orientate Model



⚠ This guide is showing you the settings for cutting the car using a 4th axis fixture. If you are using the manual fixture please go back and select the guide for the manual fixture.

- Using the up and down arrows next to the X, Y and Z values on the top left of the screen.
- ① You need to get your car to be the same orientation as shown in the screenshot. We need the canister hole facing the right side and the bottom of the car facing away from us.
- Once you have the correct orientation of the car select the next button

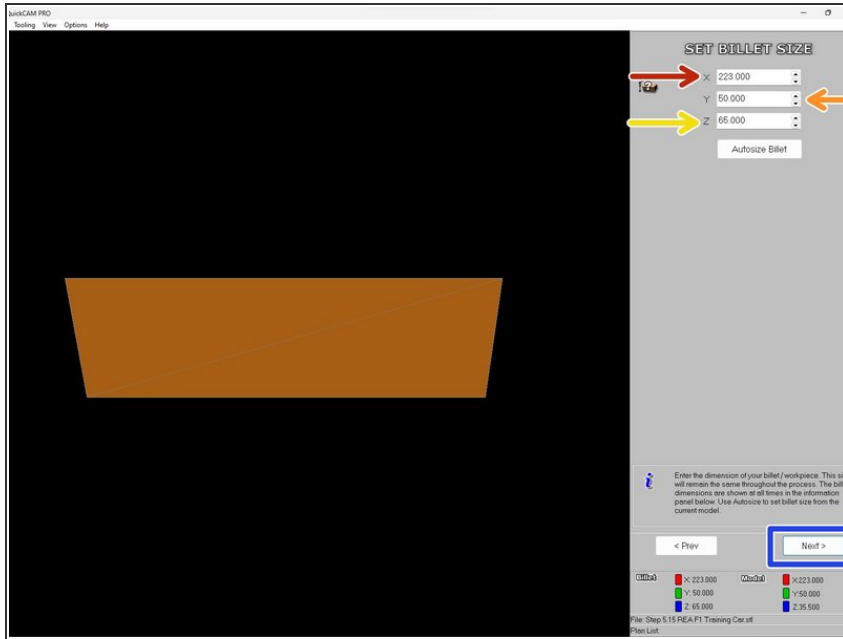
Step 6 — Set Cut Depth



i The next step is to set the Depth of Cut. By default this value will be 32.5mm (1/2 the width of the balsa blank). However you must change this value so that it takes into account the radius of the 6mm ball nose cutter we are using, which is 3mm.

- Change the Depth of cut value to - 35.5mm. Please note that this is **NEGATIVE!**
- You will see the depth of cut represented as a blue plane cutting through your model.
- Once the cut plane depth is set click on next.

Step 7 — Set Billet Size



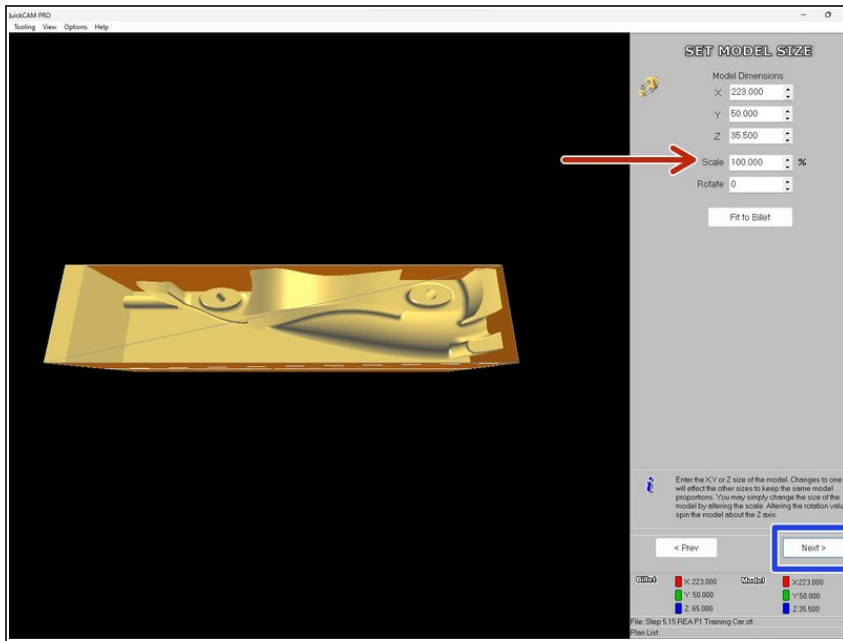
i You will now be asked to Set Billet Size. This **MUST** be set to the exact size of the Balsa Blank. Set the X, Y & Z values to the following:

- X = 223.000
- Y = 50.000
- Z = 65.000

⚠ These values are correct for machining the side of the car. For Top and Bottom Machining the values are different. [Click here for the Top and Bottom Machining guide.](#)

- Now click Next.

Step 8 — Set Model Size



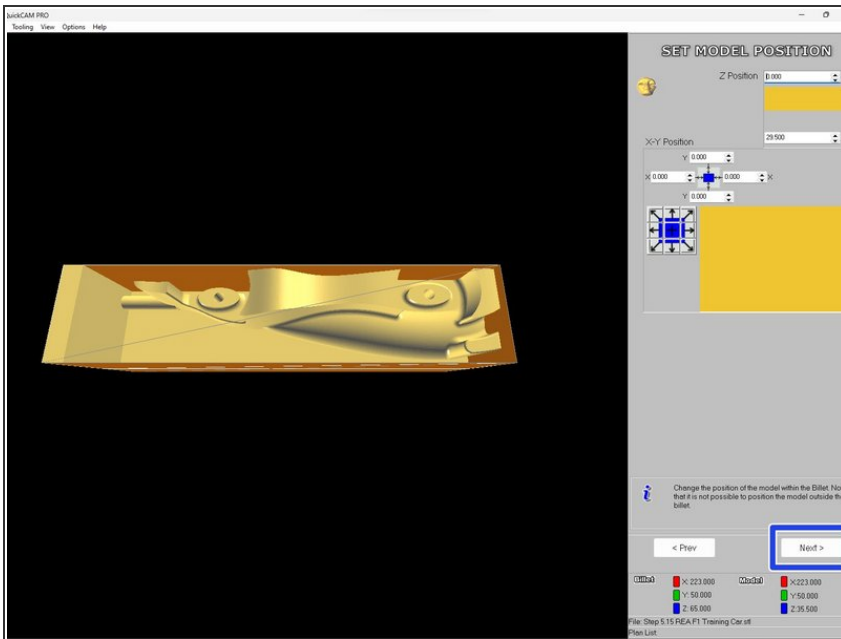
⚠ At this point it is important to check that the Scale value is set to 100%. If this is not set to 100% you are running the risk of machining your fixture!

★ QuickCAM Pro will automatically scale the model down to fit within the billet if it is too large. If this happens you need to go back to your CAD package and modify the car body and bring the new STL file in and run it through again.

i Most issues come from the billet locator at the front of the car being the incorrect size or not in the correct position.

- Once you have the scale at 100% click on Next.

Step 9 — Set Model Position

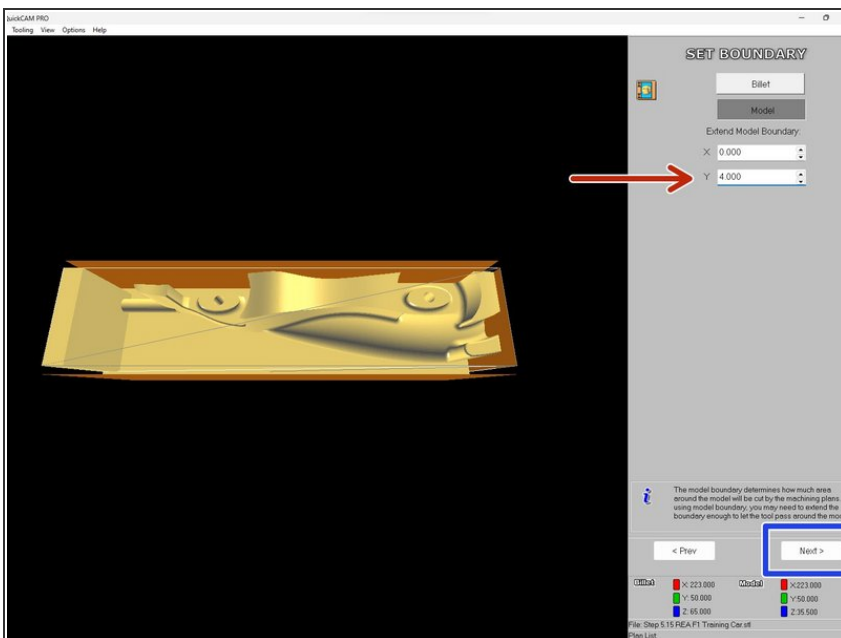


i You will now be asked to set the Model Position. This is not required because of the fact that the QuickCAM Billet Locator has ensured that the position of your car is in the correct position relative to the Balsa Blank.

! One thing to check is that the Z Position is zero in the top cell.

● Now click Next.

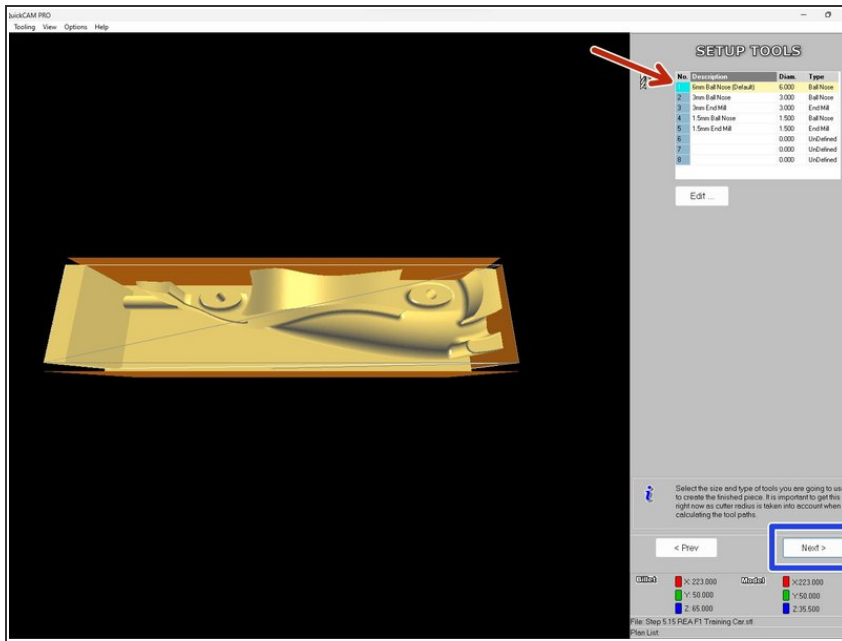
Step 10 — Set Boundary



● You are now asked to Set Boundary. You must extend the model boundary in the Y value from 0 to 4. This is to allow the cutter to cut to the outer extremities of the model. This is to be set to 4 due to the fact that the 6mm Ball nose cutter has a radius of 3mm and we want to give it an extra 1mm to ensure it can pass around all features of the car.

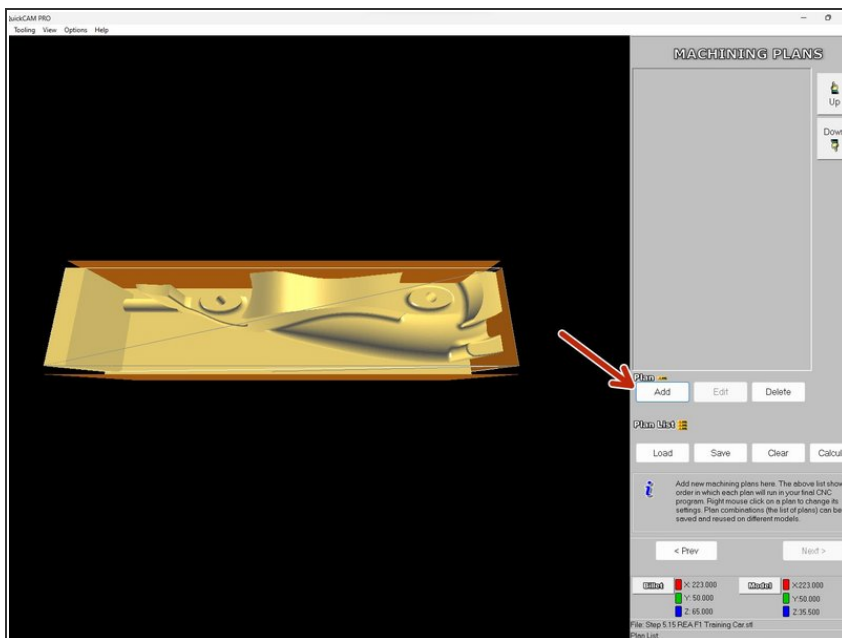
● Now click Next.

Step 11 — Setup Tools



- You will now be asked to Setup Tools. By default you will be presented with a 6mm Ball Nose cutter in slot 1. Ensure the 6mm Ball nose cutter is selected.
- ❗ Your 6mm Ball nose cutter may not be the first tool as it is shown here but that is ok.
- Click on Next.

Step 12 — Machining Plans



- You now need to add a Machining Plan. A machining plan dictates how the car will be machined. To add a

Machining Plan, simply click on the Add button.

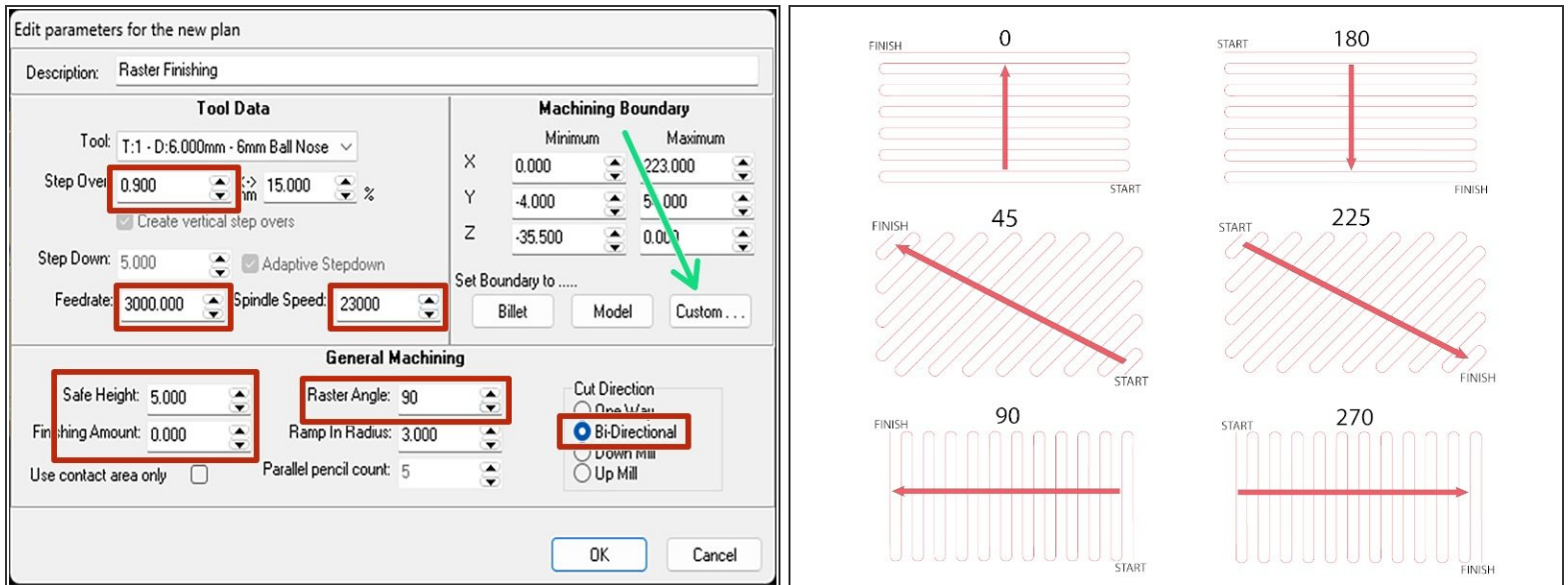
Step 13 — Machining Plan Selection



- You will now be presented with the window shown below. The machining plan we will be using for the car is Raster Finishing. Simply click on the Raster Finishing button to add this to the Machining Plan.

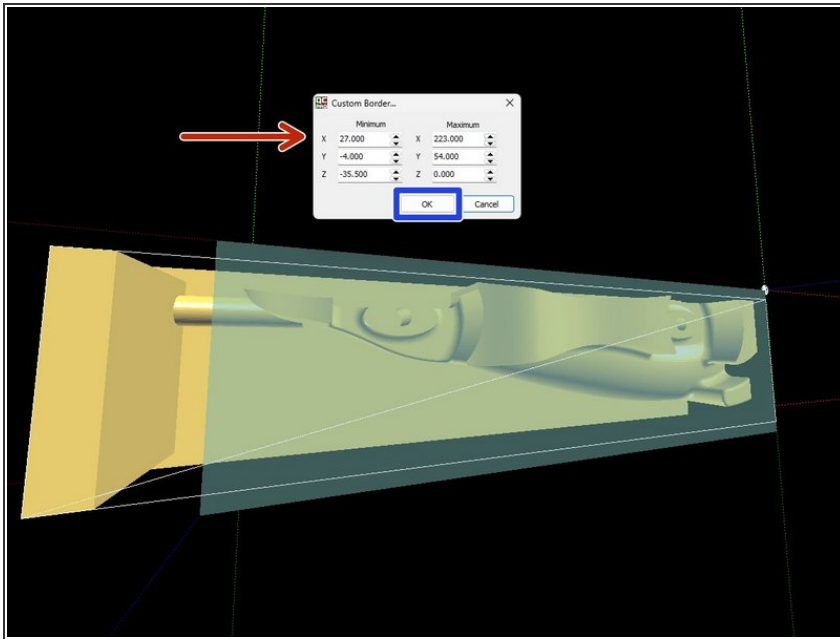
ⓘ Note also – you can RIGHT CLICK on a plan to get additional information and help!

Step 14 — Machining Plan - Parameters



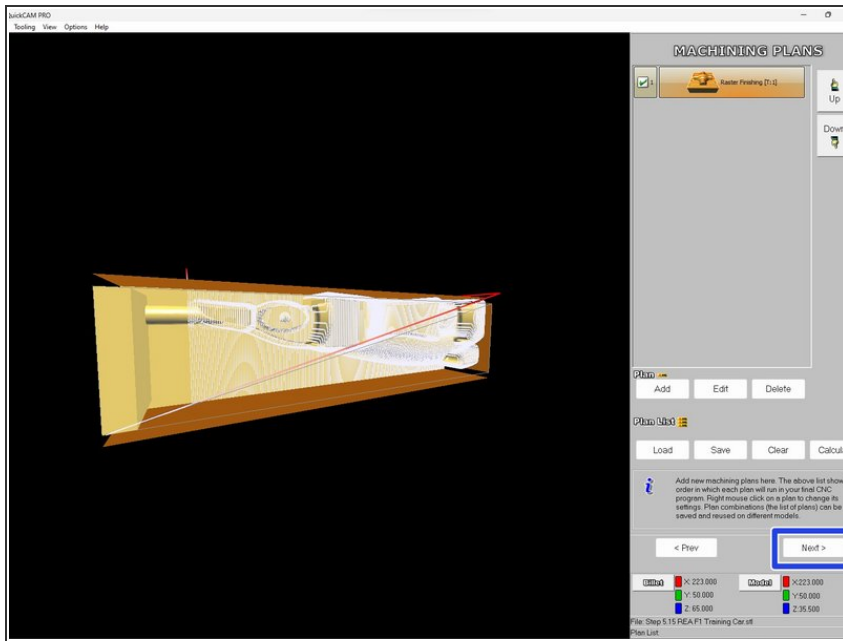
- You will now be presented with the window shown below. Make your settings common with the values highlighted.
- RASTER ANGLES** The diagrams show a variety of different raster angles that can be used. Although for an F1 car with the tether line guide to the back and canister housing on the right hand side the raster angles 45 and 90 are the most popular. This will ensure that the block is machined from the back of the car to the front of the car.
- Then click on the Set Boundary to....Custom button which is shown by the arrow above on the right.

Step 15 — Machining Plan - Custom Boundary



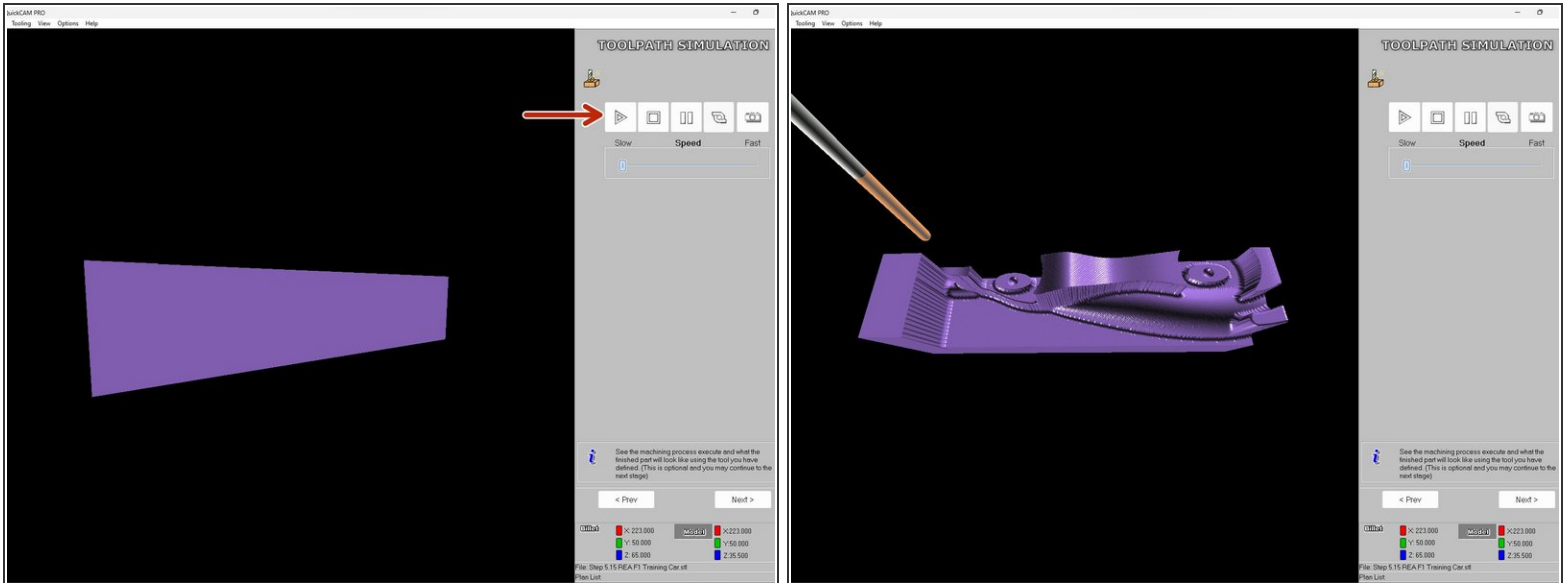
- We now need to tell the machine where it is safe to cut up to. This value is determined using the Custom Boundary Rules
- You need to adjust the X minimum value to reduce the machining boundary back towards the front of the car. If you increase the value you will see the blue transparent bounding box shrink in length.
- ① We want to bring this bounding box to the front of the car while leaving a 3-4mm gap to allow the cutter to cut around the front of the car.
- ⚠ 13mm is the minimum value you can use. If you use a value less than that you will machine into the Front Clamp Fixture.
- Now click OK.

Step 16 — Machining Plan - Generating Toolpath



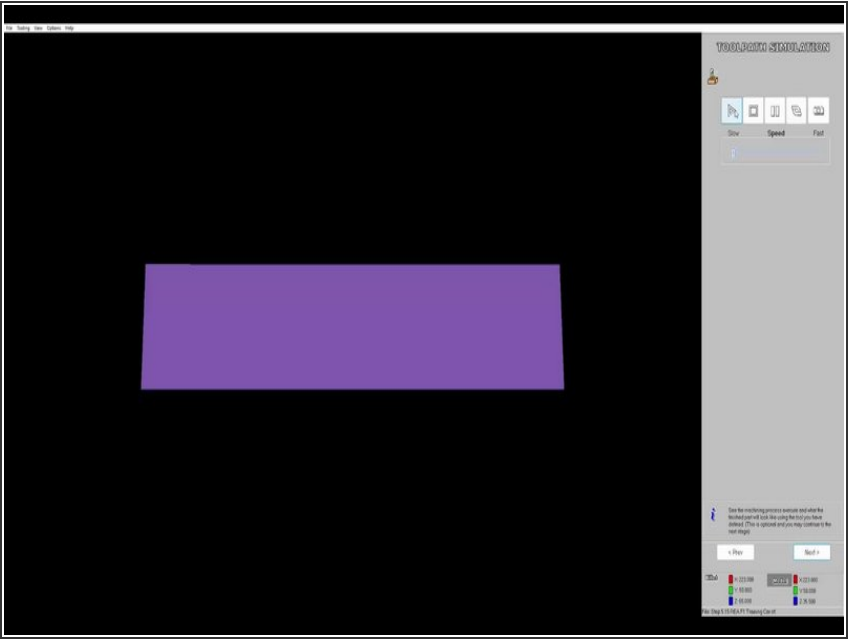
- QuickCAM Pro will now generate the tool path for your car and you will see the toolpath appear over the car in white.
- You can now click next.


Step 17 — Toolpath Simulation



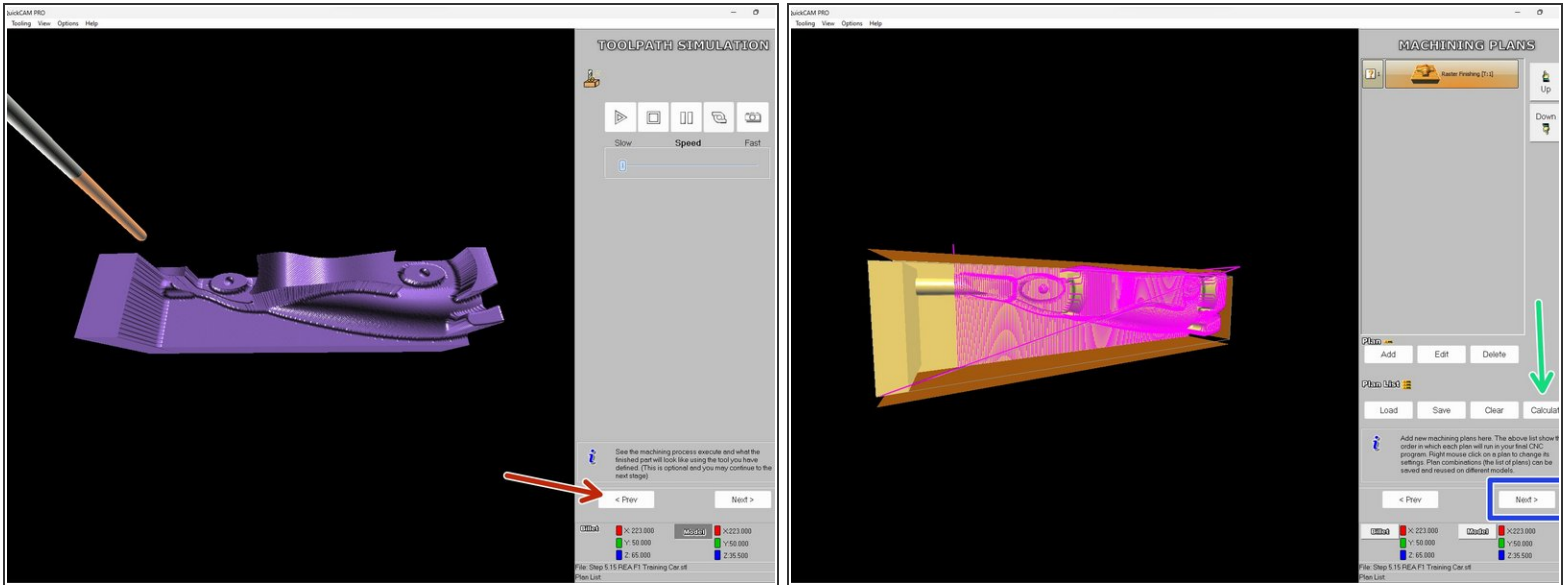
- You will now be presented with the Toolpath Simulation window. The purple block represents the balsa blank.
- To start the simulation, simply press the 'Play' button
- ❗ The toolpath will be simulated and you can use this to check that it is cutting the car as you had setup.
- ⚠ You want to check 2 things during the simulation the first is the it started cutting from the rear of the car and cut towards the front of the car. You also want to check that it cut around the front of the car.

Step 18 — Toolpath Simulation Video



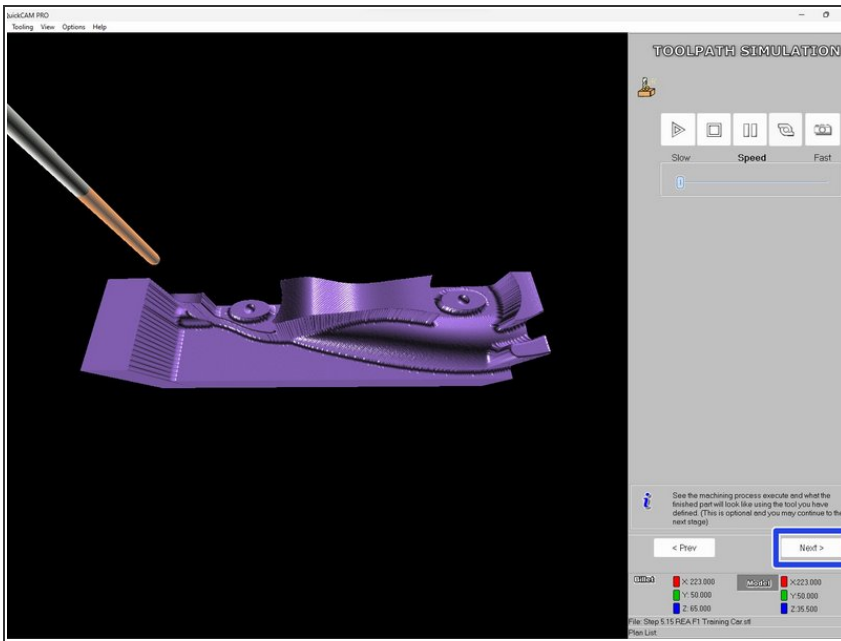
 Here is a video showing the toolpath simulation

Step 19 — Modifying settings



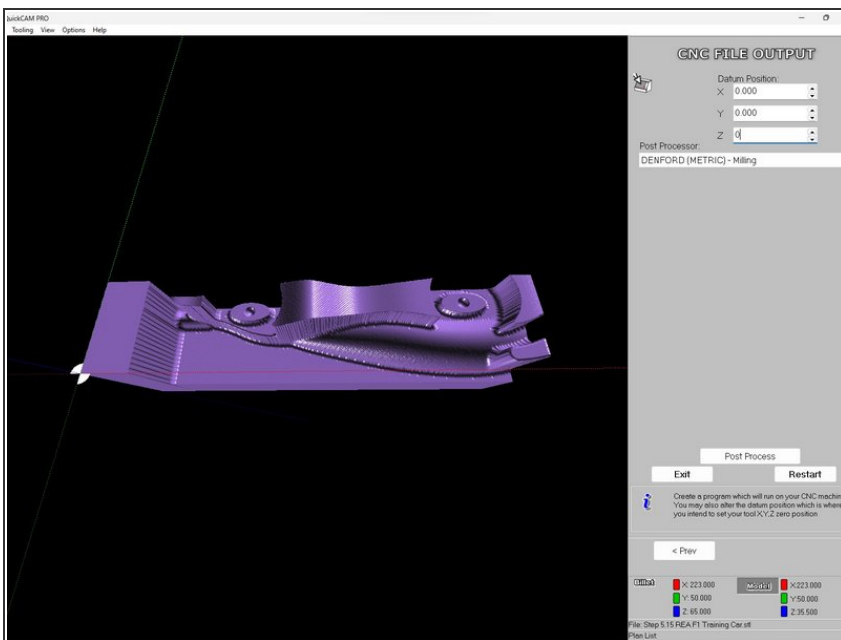
- If the simulated toolpath wasn't correct you can click on previous and go back to the machining plan page and double click the plan and make the required changes.
- ❗ If the car was being cut from the front and working to the rear you need to change the raster angle to 45 or 90.
- ❗ If it didn't cut around the front of the car then you need to adjust the custom boundary to give it more space to cut around the front of the car.
- Once you make these changes the toolpath will go purple and you need to click calculate to recalculate the plan.
- Now click Next.

Step 20 — Re-run Simulation



- Run the simulation again to confirm the settings are correct
- If everything looks good click Next.

Step 21 — Datum Position

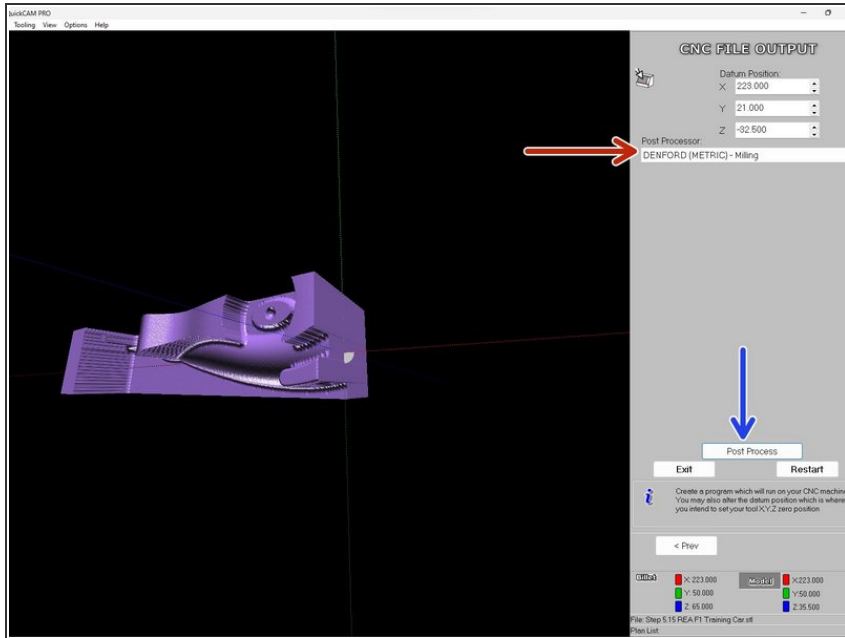


- ① You now need to set the Datum Position for side machining. This must match the origin of your car

model. The origin is set at the centre of the CO2 hole on the back face of the Balsa block.

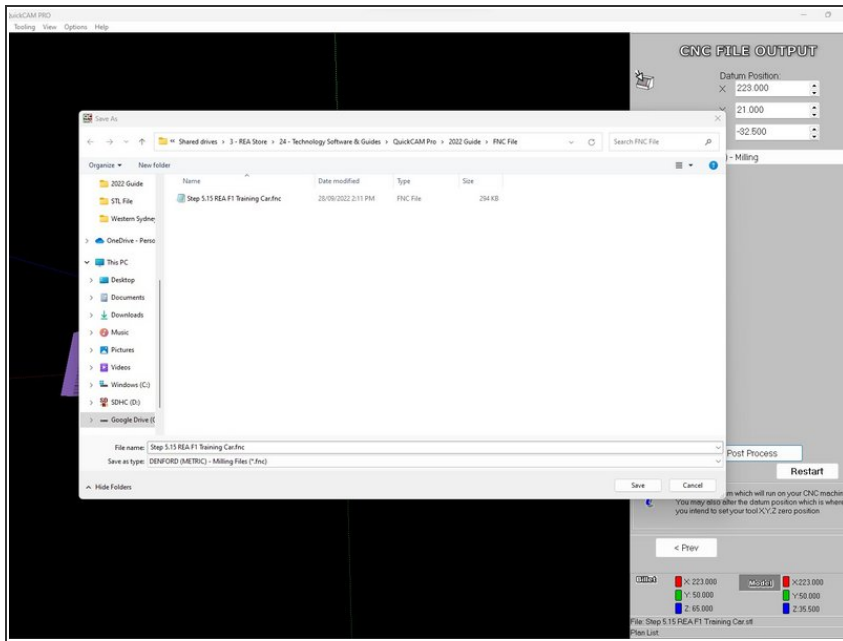
- ❗ By default the datum position will be the front left top of the balsa block.

Step 22 — Datum Position - Values



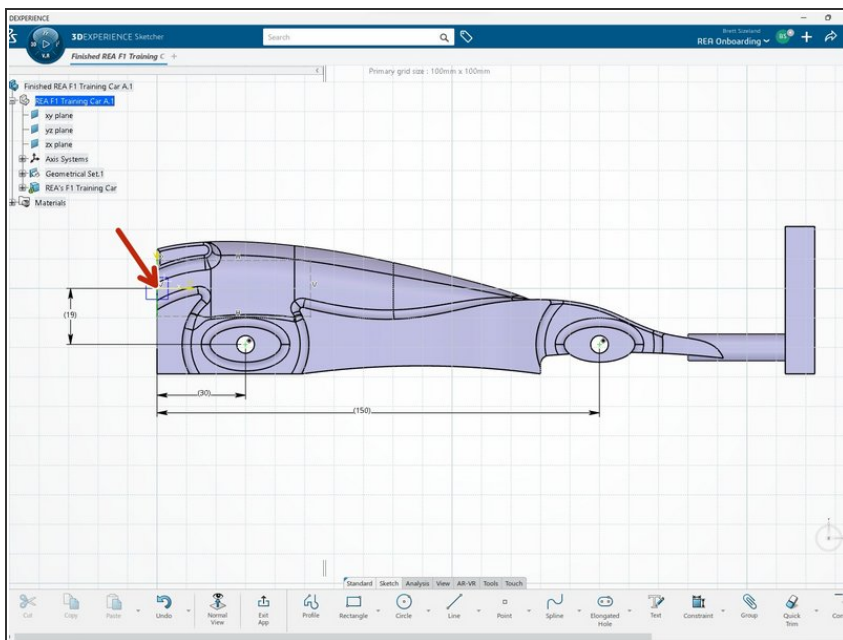
- Set the X, Y and Z values to the following.
- X = 223.000
- Y = 21.000
- Z = -32.500
- ❗ Rotate the view around to see the rear of the car to confirm that the datum position is located in the correct position.
- Now ensure the post processor is Denford Metric Milling.
- Now click Post Process

Step 23 — Saving CNC File



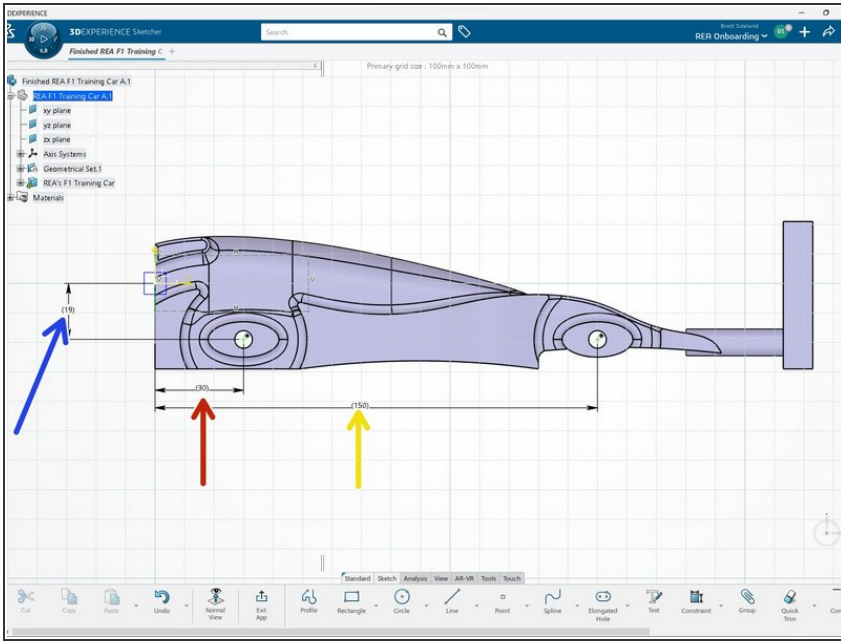
- You will now be asked to save your cnc file. Simply give the file a name, choose your save destination and click save.

Step 24 — Axle Holes



- QuickCAM Pro is not capable of adding the axle holes to the CNC file so we need to add it ourselves
- Using the CAD file of the car body we need to find the locations of the axle holes.
- We need the distances from our datum position which we have set to the back face of the car at the centre of the hole. The red arrow is showing that location.

Step 25 — Axle Hole Locations



- We need 3 dimensions. The first is from the back face of the car to the centre of the rear axle hole. For this example car it is 30mm.
- The next dimension we need is from the centreline of the canister hole down to the centre of the axle holes. For this example car it is 19mm
- The last dimension we need is from the back face of the car to the centre of the front axle hole. For this example car it is 150mm.

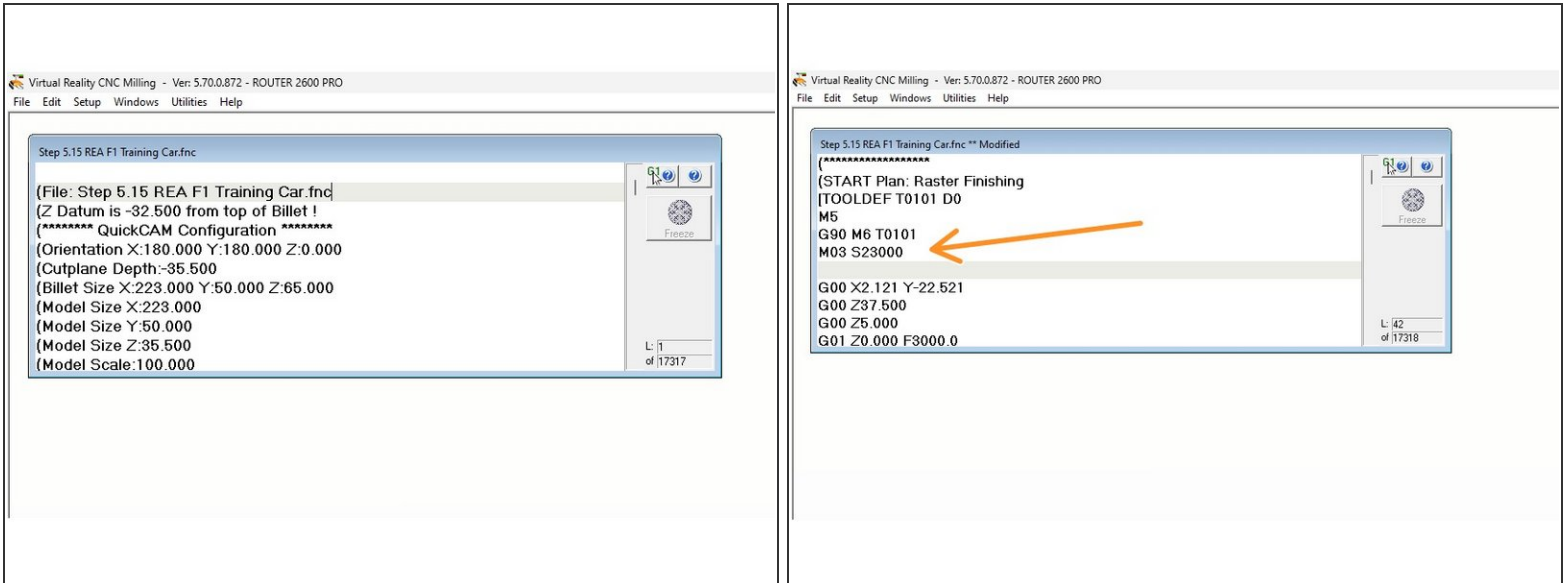
Step 26 — Values from CAD Model







i These are the 3 values that we have found.

- Rear X value = 30
- Rear Y value = 19
- Front X value = 150

Step 27 — Opening Code in VR Milling



-  We now want to open the code we created in QuickCAM Pro in VR Milling V5
-  Once opened you should see the code in the editor window as shown.
 -  We want to scroll down through the beginning of the code and find the M03 S23000 line of code it should be at line 41.
-  Click at the end of that line and press enter/return to create an empty line underneath it. This is where we will enter the code to cut the axles.

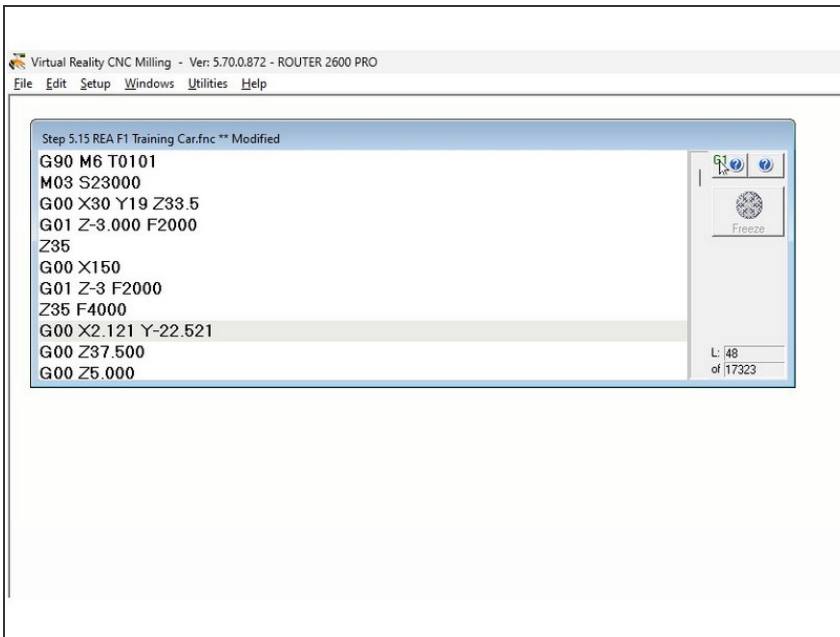
Step 28 — Axle Hole Code

```
G00 X(insert your REAR X value) Y(insert your REAR Y value) Z33.5
G01 Z-3 F2000
Z35 F4000
G00 X(insert your FRONT X value)
G01 Z-3 F2000
Z35 F4000
```

⚠ This is the code that we would enter for this example car, the code for your car will be different. Take note of the image as it shows the values in red where you enter your own values from your car to the code.

- G00 X30 Y19 Z33.5
- G01 Z-3 F2000
- Z35 F4000
- G00 X150
- G01 Z-3 F2000
- Z35 F4000

Step 29 — Adding Axle Hole Code



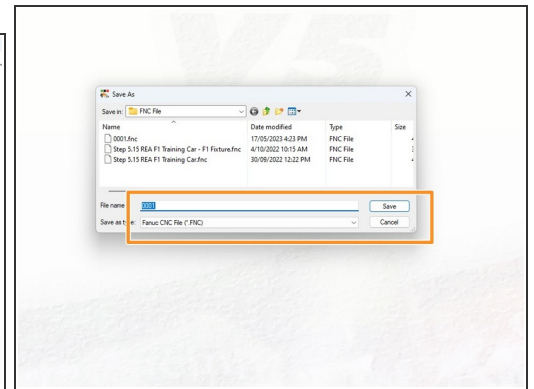
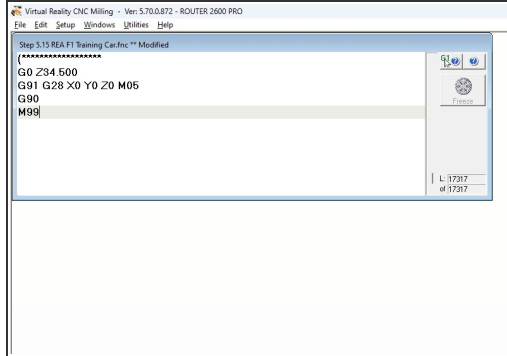
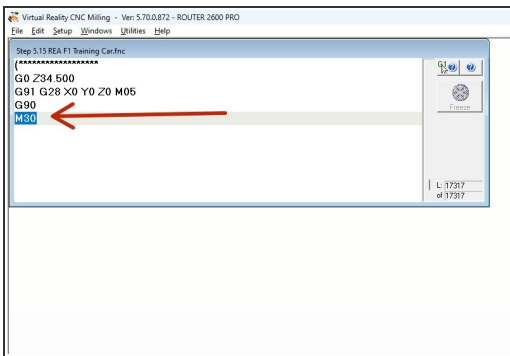
- Type the code into your code in VR Milling. It should look similar to the screenshot showing the code for the example car.
- Now click File and then Save to save the changes.

Step 30



- With the inclusion of the 4th axis, we can now configure the code to perform the following sequence automatically: first, it will cut the initial side of the car, then rotate 180 degrees, and finally proceed to cut the second side.

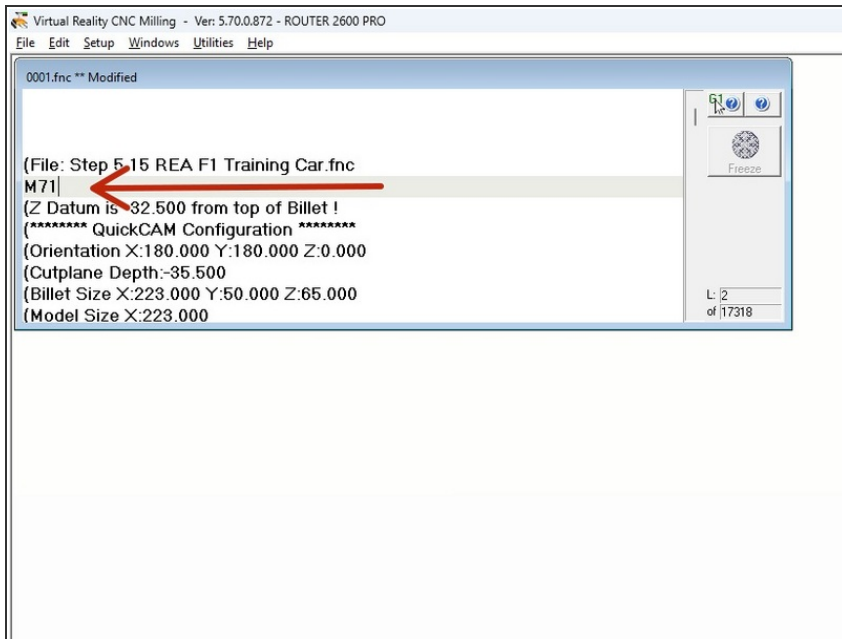
Step 31



- We now need to modify the code for the first side of the car to set it up for use with the Sub Call Program
- Scroll to the very bottom of the code and change **M30** to **M99**
- Now save the file as **0001.fnc**

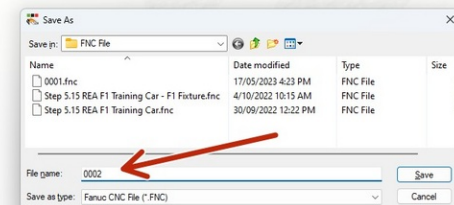
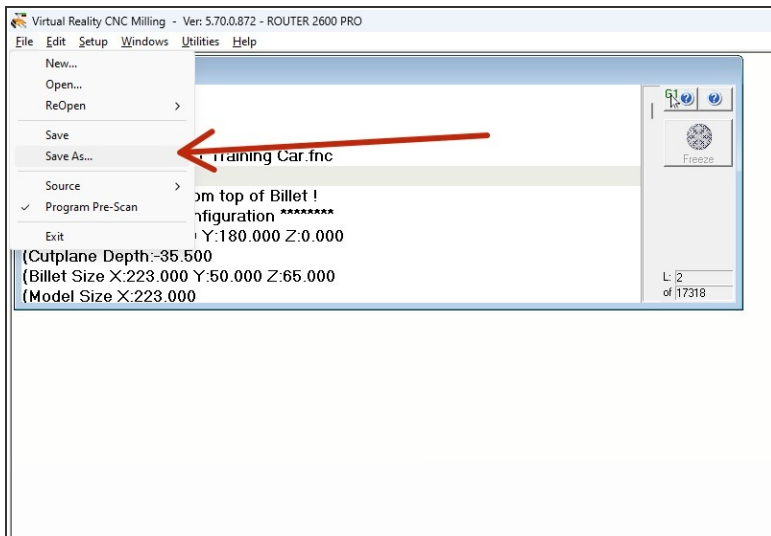
⚠ When creating files for new cars ensure that you have a folder for each car, otherwise the 0001.fnc files will overwrite each other

Step 32



- i** Now that we have created the 0001.fnc which will be the first side of the car we need to create a code for the second side of the car.
- This is done by adding a mirror code to the 0001.fnc and saving it as 0002.fnc
 - Scroll to the top of the code and add a new line by pressing enter. Now type in **M71**

Step 33



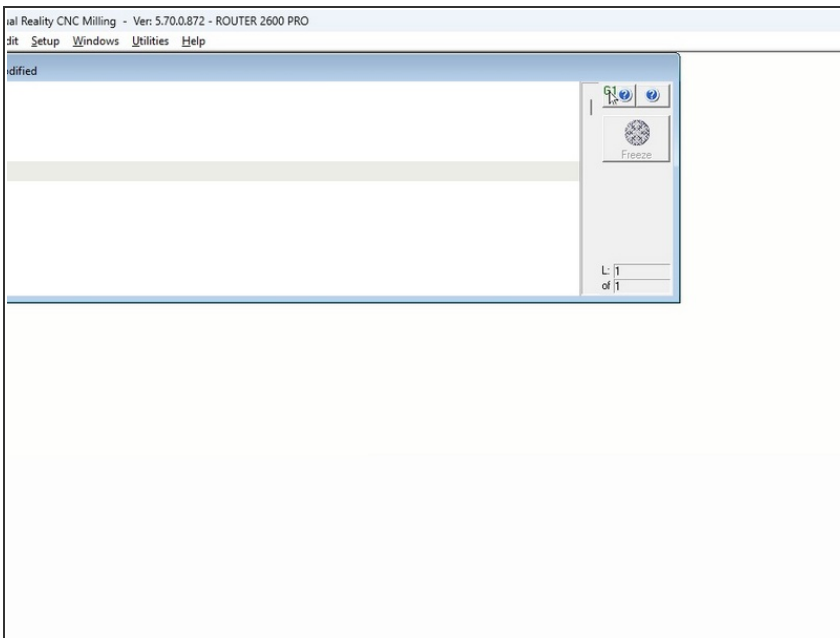
- Now go to Save as and change the name to **0002** and hit save

Step 34



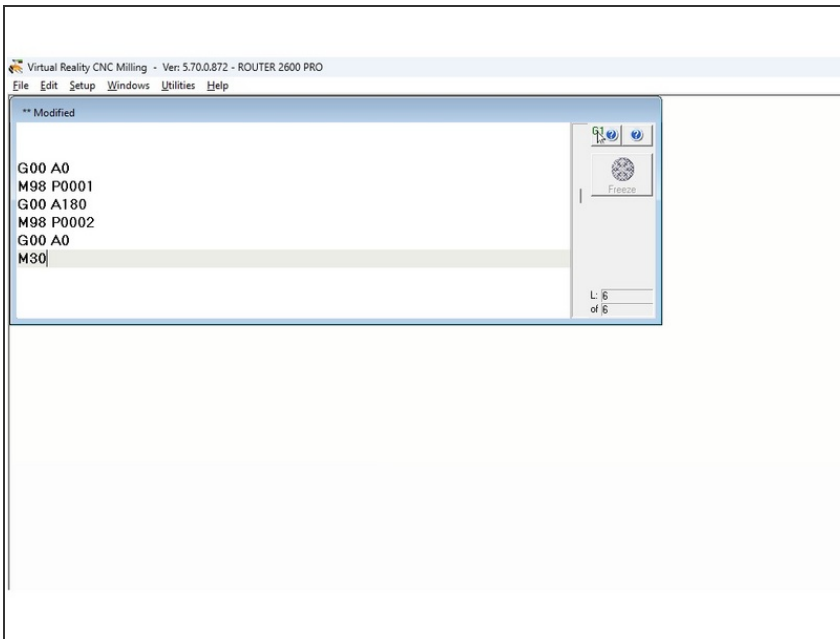
- We now have 0001.fnc to cut the first side of the car and 0002.fnc to cut the second side of the car.
- What we need to do now is create a program called sub call.fnc which is basically a set of instructions to tell the machine to cut the first side of the car then rotate to 180 degrees and cut the second side of the car.

Step 35



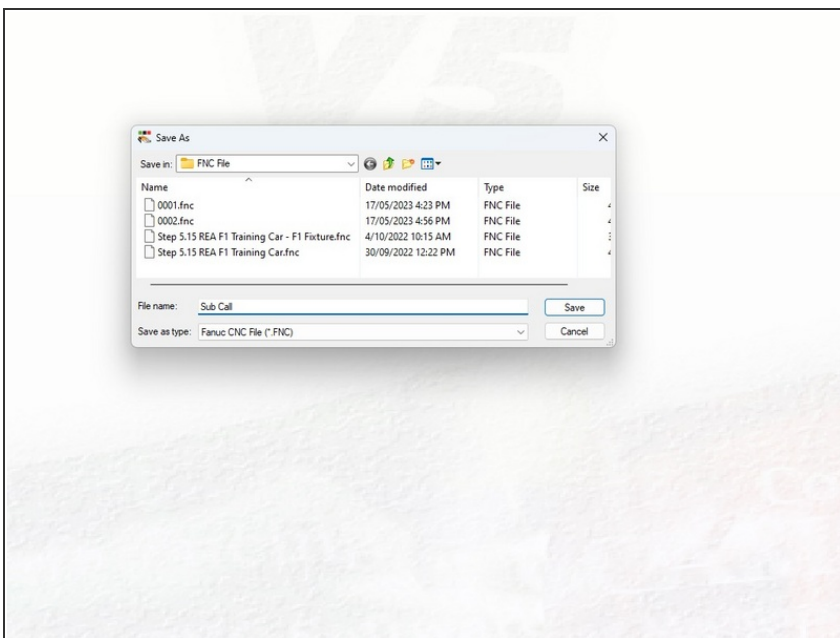
- Open New File in VR CNC Milling
- ① We want to create a file that will instruct the machine to cut the first side of the car and then rotate to the second side and cut that side of the car.

Step 36



- Enter the following code into the new file.
- G00 A0
- M98 P0001
- G00 A180
- M98 P0002
- G00 A0
- M30

Step 37



- Now save the file as **Sub Call.fnc**

Step 38



⚠ You need to ensure that the sub call.fnc and the 0001.fnc and 0002.fnc are in the same folder.

- You can now open the sub call.fnc in VR Milling and run the program to cut the car out.

Step 39



- There is a zip folder below that has the Sub Call.fnc and an axle code.fnc that you can use to copy the code from.

Congratulations you have created the .fnc file that contains the G-code to cut the side of your car.