CAMWorks – How To Create CNC G-Code for CO2 Dragsters

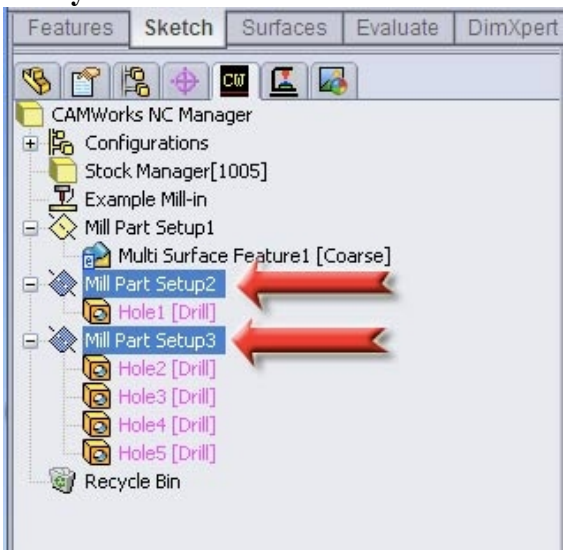
In this chapter we will create the smooth G-Code tool path for the bottom of our CO2 Dragster. The smooth tool path is necessary to create a finish that requires minimal work to for the designer to later complete the project. The smooth finish tool path is critical in alleviating unnecessary mistakes that later have to be corrected and redrawn/modeled in the model and drawing files. This smooth process is vital because it is hard to get down into the pockets for the wheels on a shell dragster to sand and fix tolerance mistakes that occur in the design process.

We will build on the previous tool path we created in the roughing G-Code tool path. We will reuse the Mill Part parameters and Strategies for the tool paths. What we will do is to modify the tool settings in those strategies.

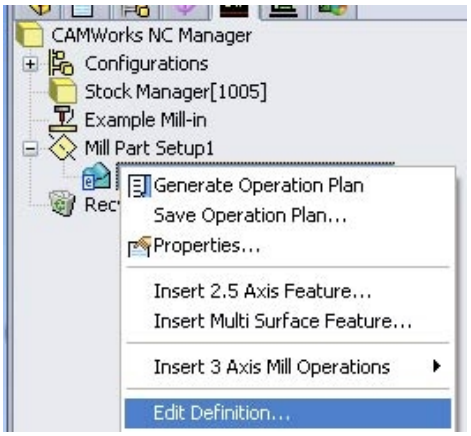
III.1.
Save the rough tool path for the bottom of the CO2 Dragster as "Dragster bottom 001 rough".

Immediately, save the open file, "Dragster bottom 001 rough" as "Dragster bottom 001 smooth". The rough file has useful setups we can reuse for the smooth tool path operation.

III.2.
Click on the CW tab.
Highlight and delete any unused Mill part setups. These will be the Mill Part Setup2 and Mill Part Setup3 CAMWorks will attempt to use Automatic Feature Recognition to create possible features for our considerations. **Empty the recycle bin.**



III.3.

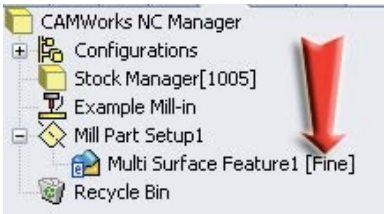


Select Multi Surface Feature1 [Coarse] and right click to edit the "Edit Definition". We will go back and redefine the strategy for this mill part setup. We have already defined the direction of the cutter from the previous rough tool path and since we are milling in the same direction, we are reusing this Mill Part Setup.

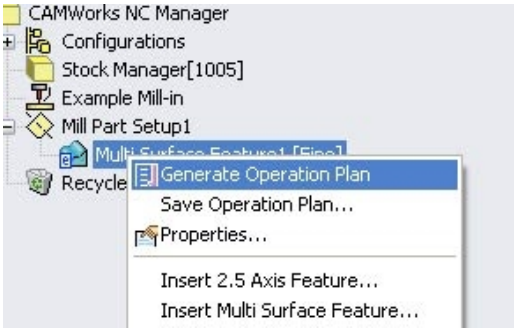
III.4.

The first arrow shows from the previous rough tool path, this strategy is set to Coarse. **As the second arrow shows, select the fine Strategy.**

III.5.
Note the Mill Part Setup1[Fine] has changed to "Fine".

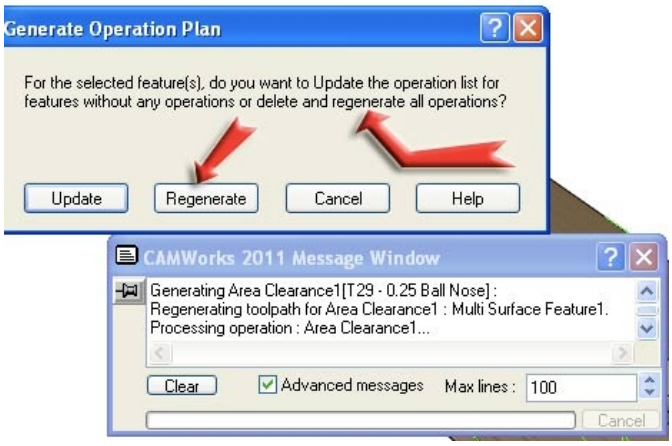


III.6.

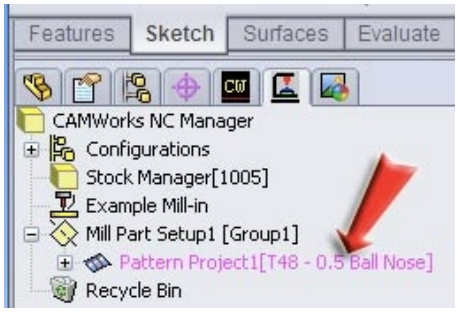


Select and right click and then click on the Generate Operation Plan.

III.7.
Select the Regenerate button, because as the arrow indicates we need to regenerate all operations for this strategy.



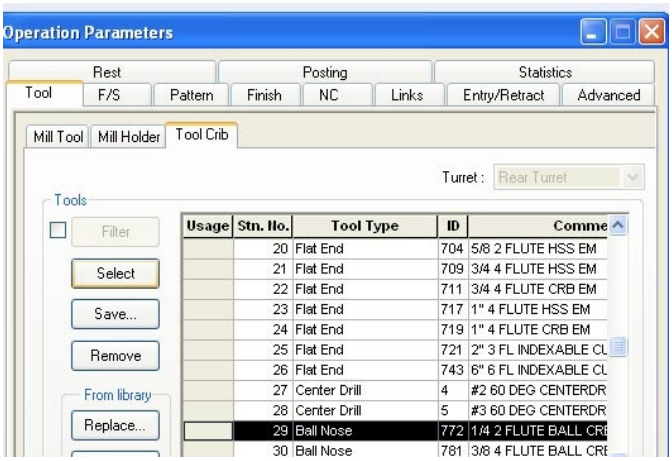
III.8.



Here you see the new "Regenerated" strategy in the Operations Tabs. We do no have to change the Mill Part Setup1 [Group1]. It still has the off set up, the X-Axis cut direction and the NC planes set properly.
Note, that Pattern Project is set to [T48 – 0.5 Ball Nose]. This will have to be changed, because our tool is a .25" Ball Nose.

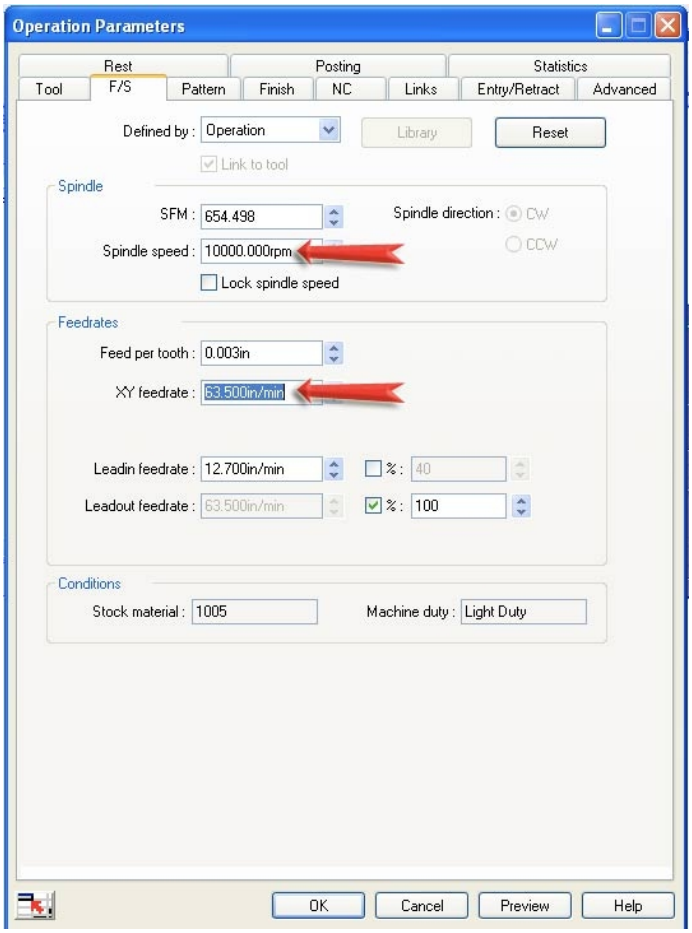
III.9.
Select Pattern Project1[T48 – 0.5 Ball Nose]. **Right click to Edit Definition...** From here we will change the tool and other definitions of this strategy.

III.10.



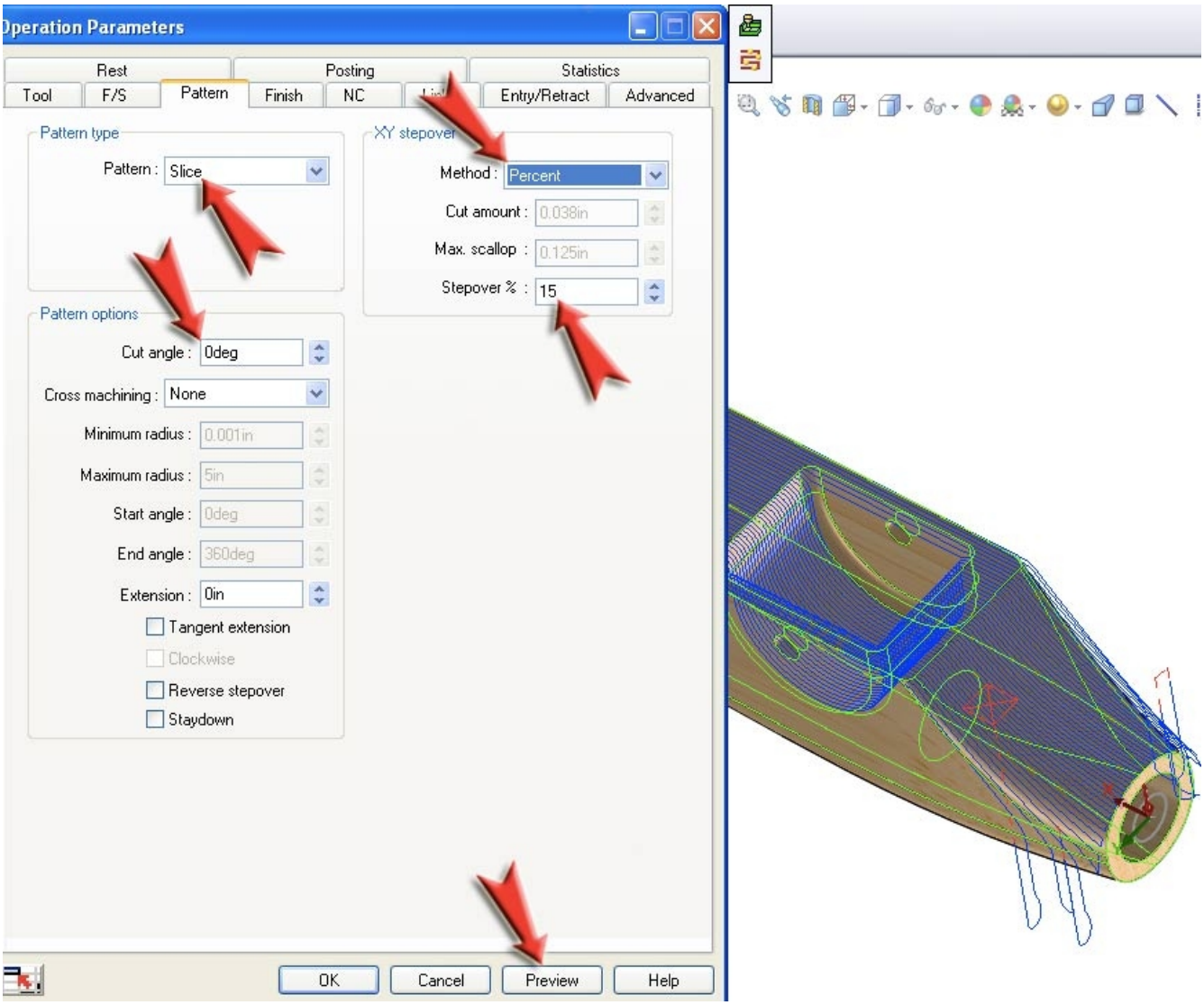
Select the Tool Crib and select Stn. No. 29. This more represents the tool install in our machine.

III.11.



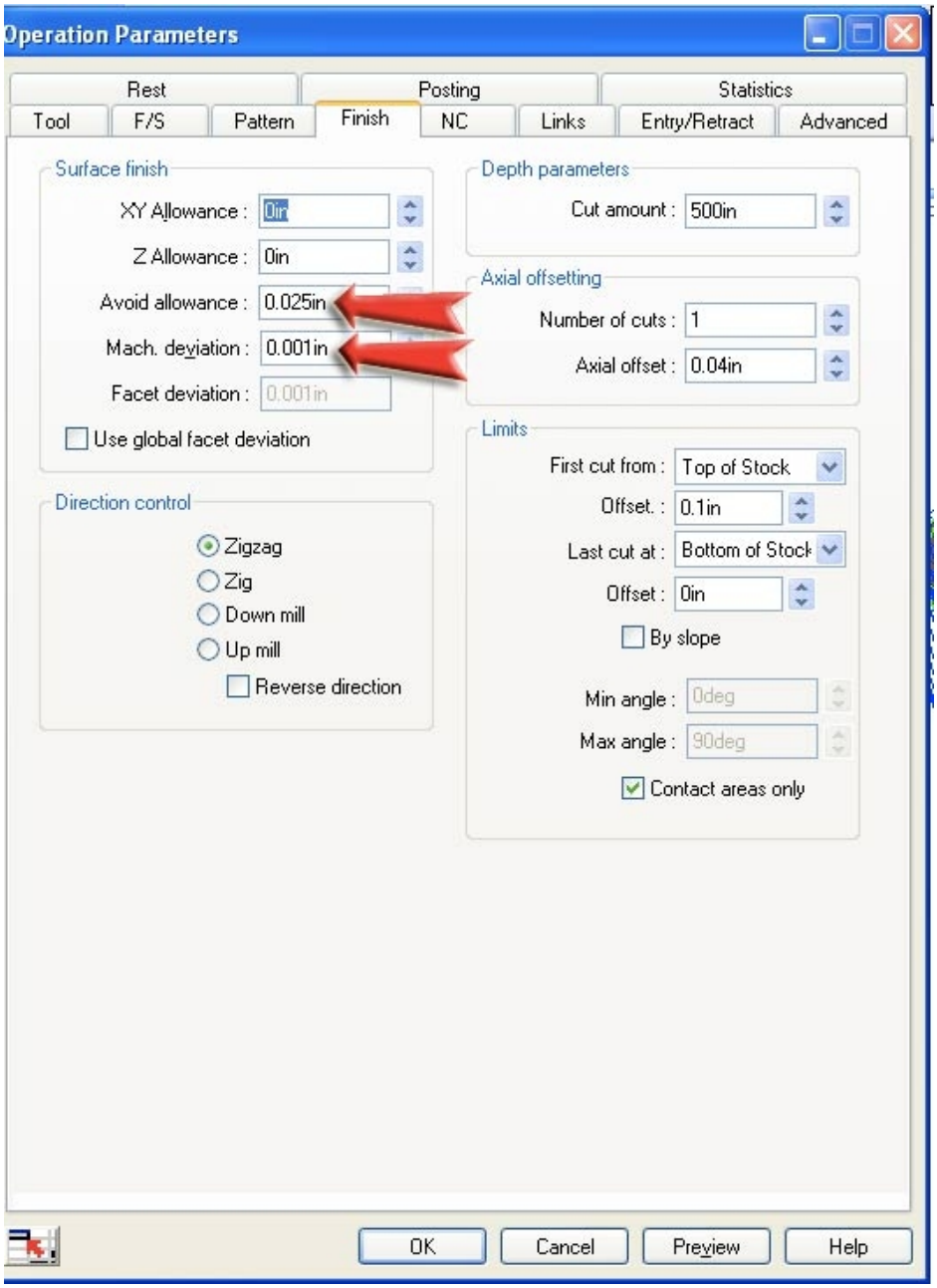
Select the F/S tab and then set the Spindle Speed: 10000. Notice it will change the XY feedrate: 63.500in/min automatically. The XY feedrate setting is important because it dictates the milling speed of you CNC Mill and CNC Router. Remember you have feed rate control on your CNC Router and CNC Mill. As such, the 63.500in/min rate is fast and you can increase the speed based on your billet density.

III.12.



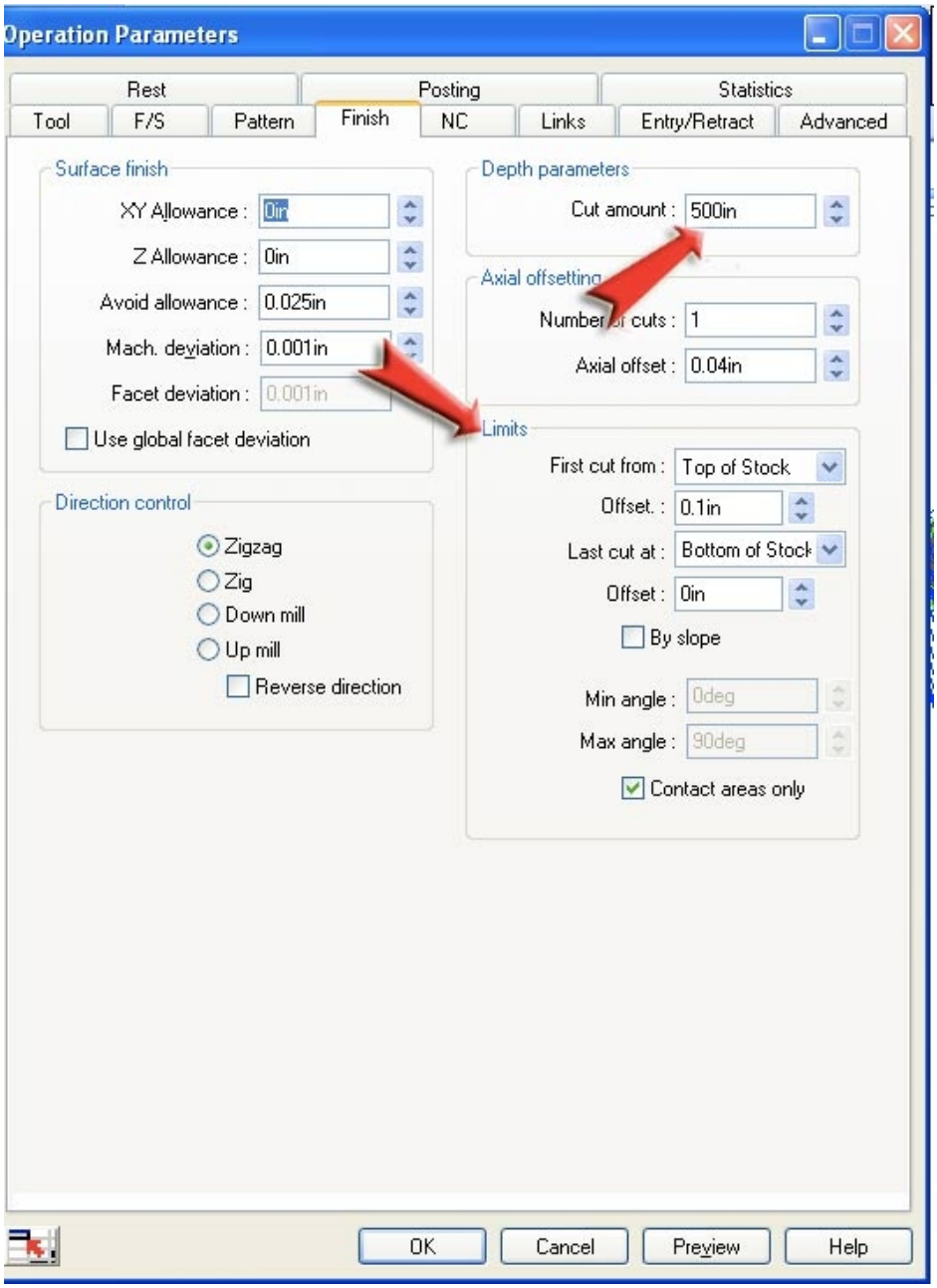
Select Pattern field and make sure it is set to Slice.
Make sure the Cut angle: is set to 0 degrees.
On the XY step over field Method: to Percent.
Set Stepover % to 15%.
Click on the Preview button to process the tool file and then to see the tool pattern as it appears to the right. From the tool path you can see there are changes that need to be made to the bottom of the car.

III.13.



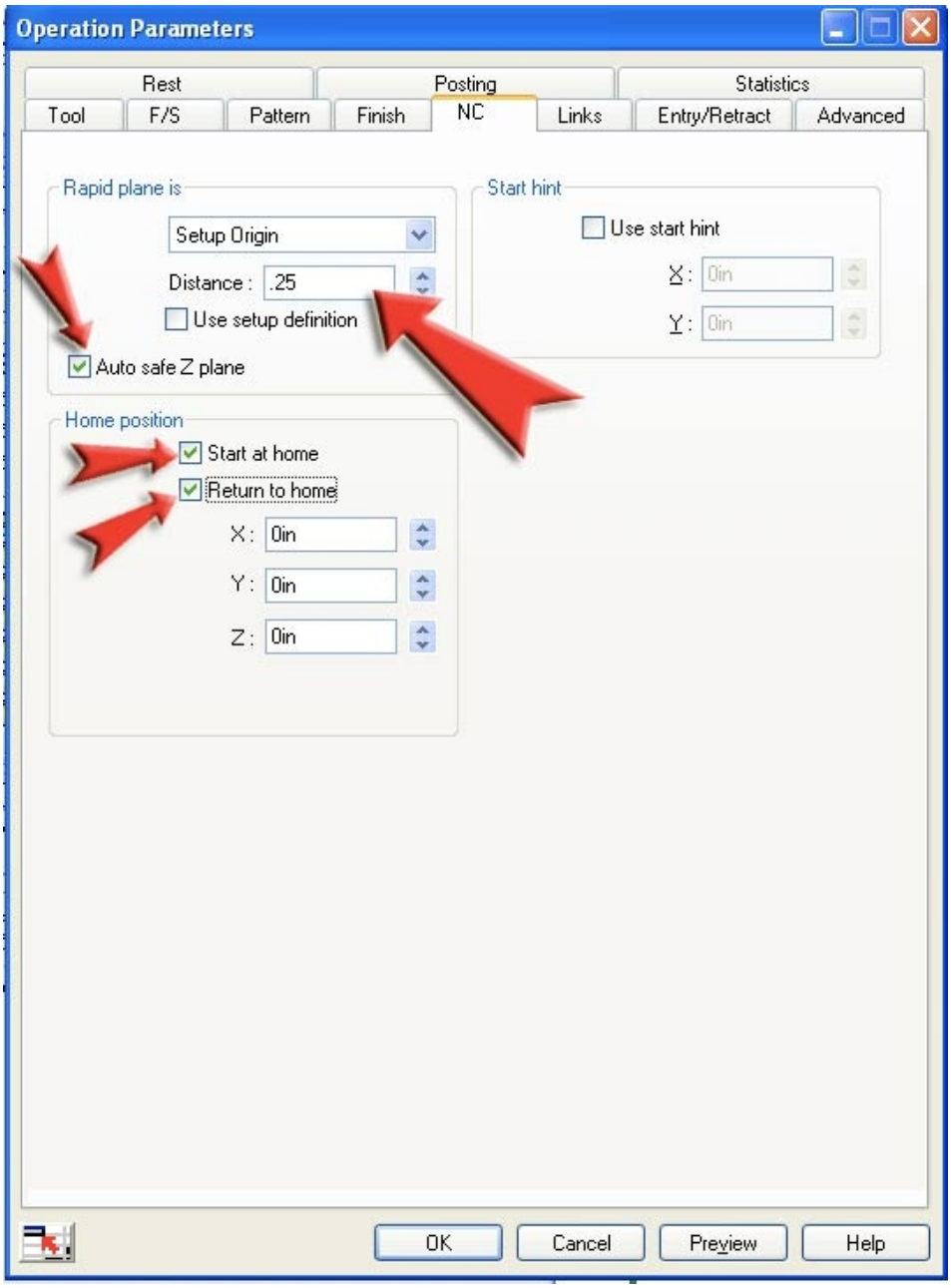
Make sure your Avoid allowance is set to .025in. Make sure Mach. (Machine) deviation is set to .001in. This creates closer tolerances for the machine to cut down inside the pockets for your wheels and especially if your CO2 Dragster uses any kind of Computational Fluid Dynamics (CFD) design techniques you may have designed into your CO2 Dragster. This is common on more sophisticated and competitive CO2 Dragsters designers will later create.

III.14.



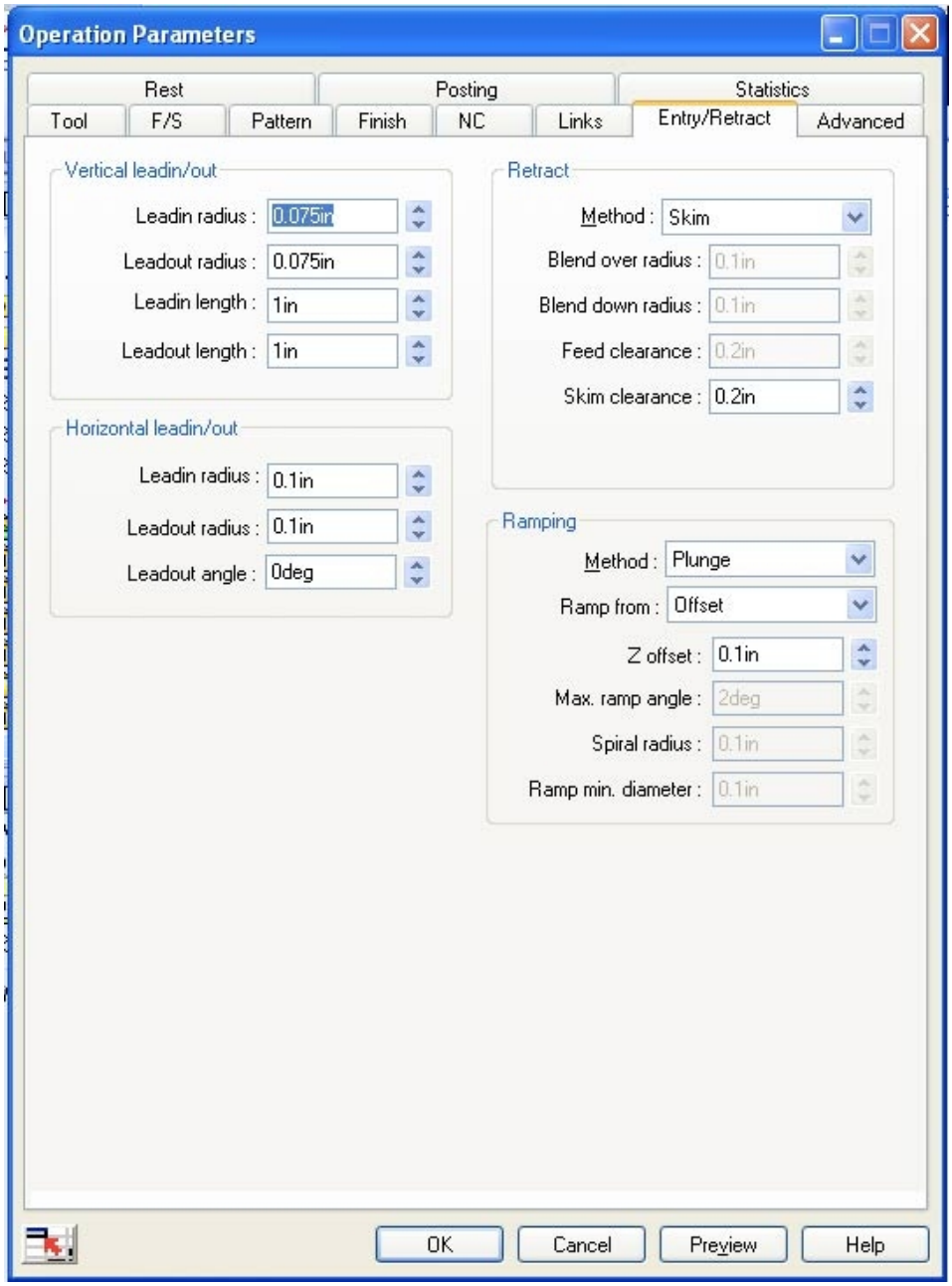
NOT Be concerned.
Depth parameters are not a concern on this tool path for the bottom for this tool strategy.
Limits are not a concern on this tool path, however later on the side cuts they will be a factor and we will address those parameters at that time. The Limits are not a concern at this time, because later we will set contain areas on the wheel wells, as such it is not a concern at this time.

III.15.



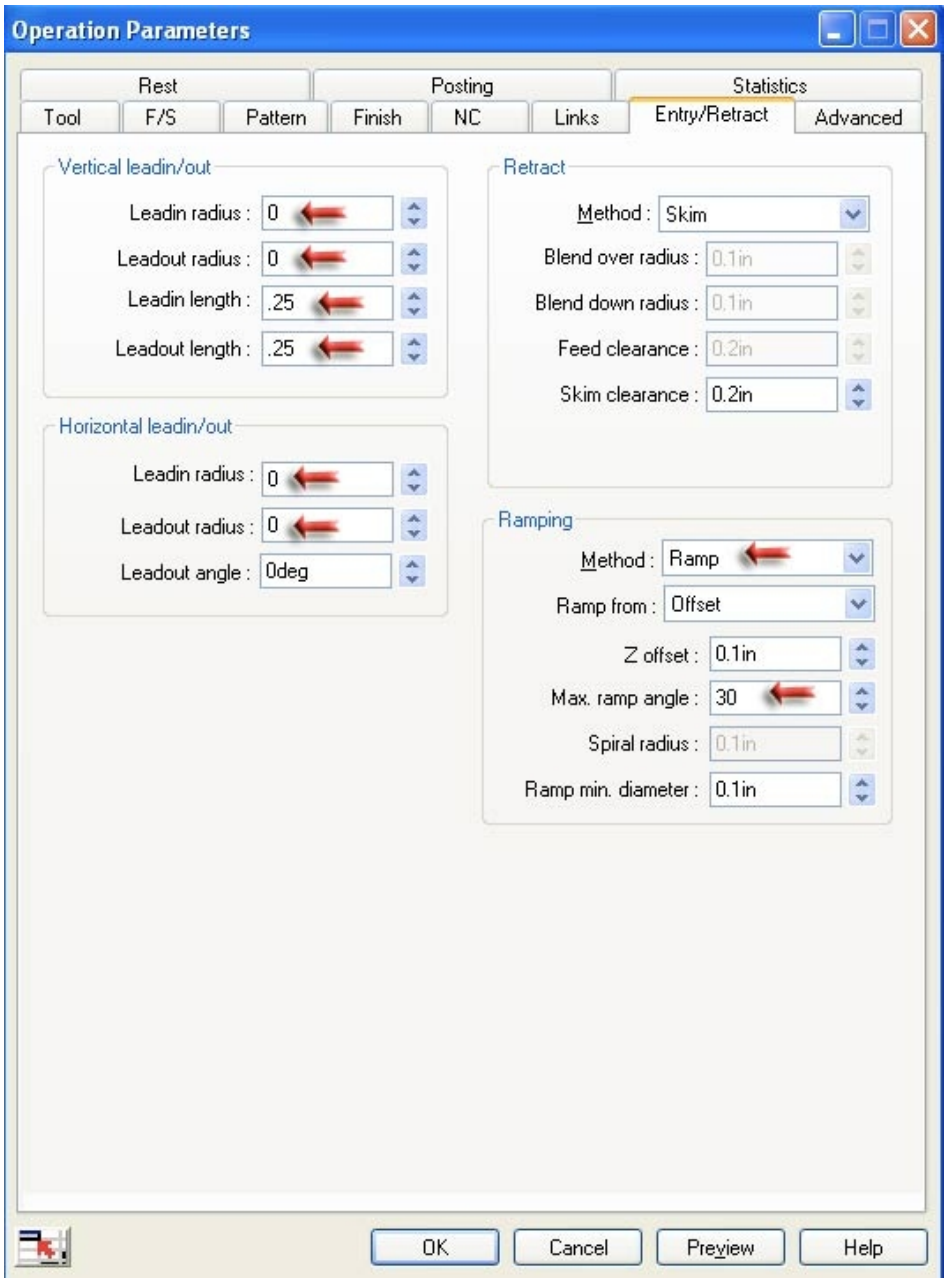
Make sure your settings are as indicated in the image above. These settings will alleviate a lot of problems with avoiding Z-Axis limit errors and it will dramatically improve the speed of your milling operation due the fact you are not raising and lowering the Spindle or the Router in Z-Axis with the "Lead-in feedrate" of 12.70in a minute. Remember this is set in the F/S tab. You have set the NC Planes in the Mill Part Setup, however you can override the settings here and you do not want to do that based on the nature of our types of CNC Routers and CNC Mills manufactured by TechnoCNC and Denford. These machines have limited Z-Axis travel.

III.16.



These are the default settings and these will be changed in the next image below. The settings for this panel are important, because here you can significantly increase your milling speed and operations by making the changes as show in the next image.

III.17.

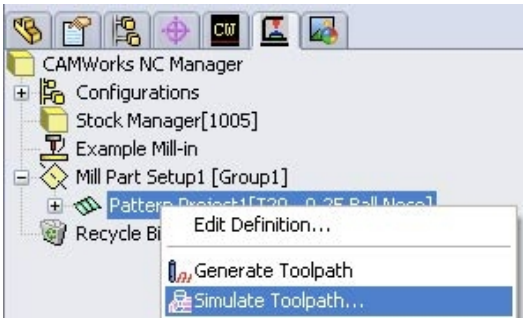


Make the changes to the settings you see above. These settings are more conducive to the types of CNC machines we have in our labs. The setting of Ramp at 30 degrees is aggressive due to the density and weight of the CO2 Dragster blocks you use and you may need to slow down the machine feed rate over ride on your CNC Router, or you can change this setting to 20 degrees and you should be fine.

II.18.

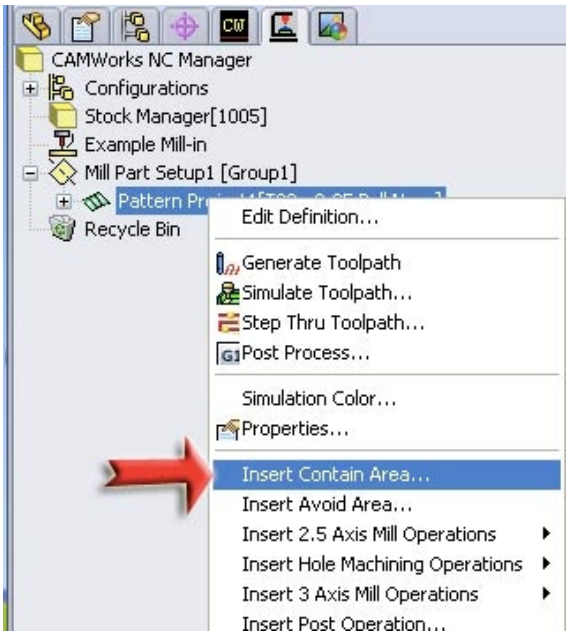
After you have finished Entry/Retract click OK and CAMWorks will ask you to recalculate and Regenerate your toolpaths now? Go ahead and select yes to reprocess your toolpaths. At that point you can accurately simulate your toolpath using the CAMWorks simulation tool.

II.19.



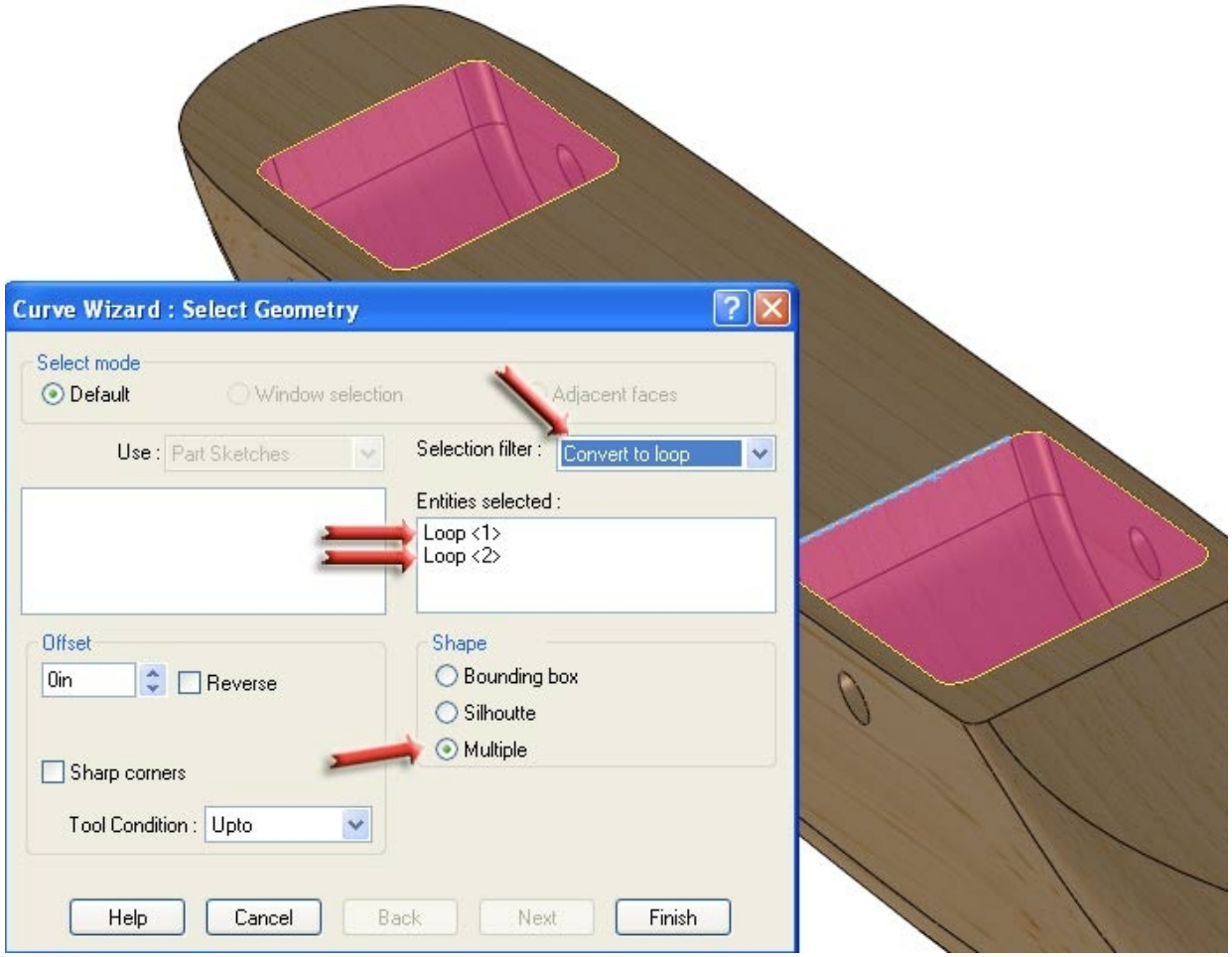
Select Pattern Project1[T29 – 0.25 Ball Nose]and then Right Click your mouse and then select Simulate Toolpath... to see the simulation of your settings to date.

II.20.



Select Pattern Project1[T29 – 0.25 Ball Nose]and then Right Click your mouse and then scroll down and select Insert Contain Area...this is where we will isolate just the wheels to mill and no other areas will be affected.

II.21.

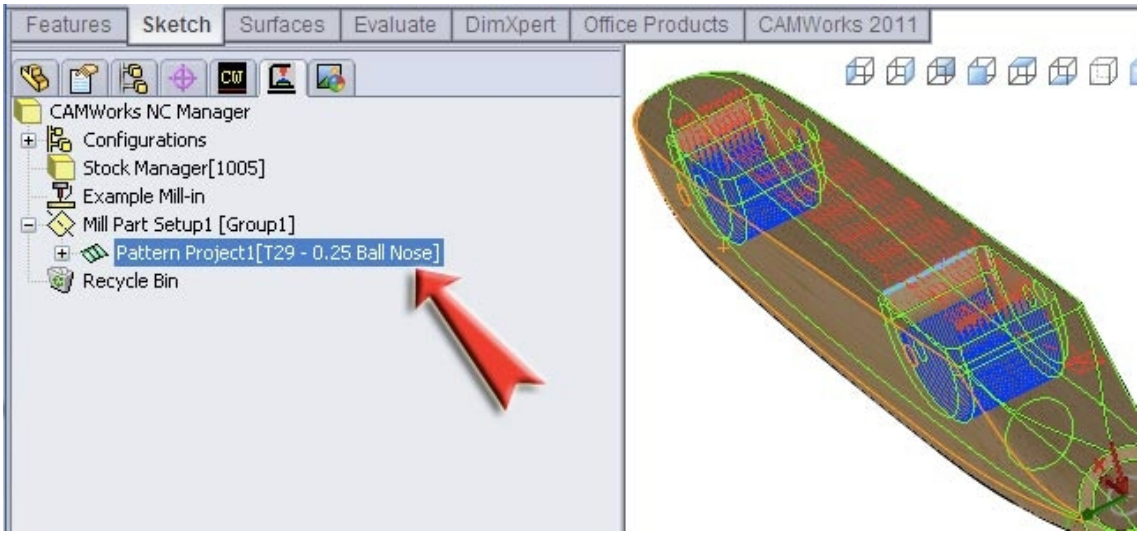


By selecting the edge of the opening on the wheel and having the Selection filter: set to Convert to loop, CAMWorks will follow around the edge of the wheel well opening and highlight it with a pink selection. Immediately select the edge of the next wheel well opening and it will do the same as the first. This is why you must **select Multiple** to select multiple loops.

II.22..

After select Finish, CAMWorks will display the above warning. Select Yes and it will regenerate your toolpaths. The purpose of regenerating your toolpaths is to account for the contain loops you just selected and thus eliminating all of the areas of around the wheel wells to mill.

II.23.



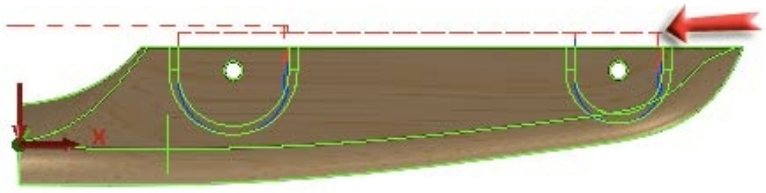
Select the Pattern Project1[T29 – 0.25 Ball Nose] tool path and it will highlight the newly regenerated tool path showing the effects of the contain areas. Notice that none of the areas outside of the contain areas have blue milling lines. The red lines are traverse lines from one area to another. No milling occurs on those areas.

IMPORTANT CONSIDERATION:
By having a rough pattern that has already "roughed" out the content of the wheel wells, your milling operation would most likely NOT crash your CNC Mill or CNC Router. Crashing results from binding up the spindle or router to deep with to much material, thus forcing the machine to loose it's off set. When this happens, the machine will start to mill in different directions destroying your work up to that point. Some CNC machines will just stop and the mill computer will show either a limits exceed error or just stop. Some machines, by losing it's off set will crash the cutter into the fixture. This usually damages the fixture and breaks the tool. If the tool is not broken, then is typically galled (coated with the aluminum or plastic from the fixture). When you create a new tool path, it is important to stand by your machine, in close proximity to the Emergency Stop (E-Stop) button while the machine mills out the part for the first time the tool path is run. If you see an abnormal operation immediately press the

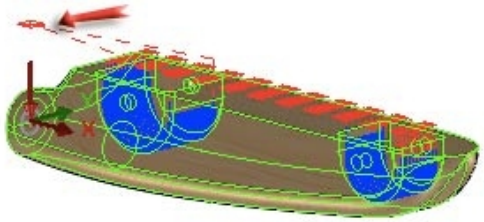
E-Stop button.

After you stop your machine, go back to CAMWorks and re-evaluate your tool path. See the areas where you might have made a potential mistake. Look for areas like to deep of depth of cut, excessive speed on your feedrate and that if your ramp angle is aggressive on the Entry/Retract tab. Make corrections and retry.

II.24.
VISUALLY CHECK YOUR TOOL PATHS

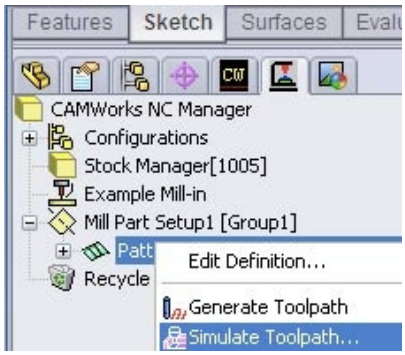


Here you see that the toolpath is .25" above the CO2 Dragster...this is what we setup and this is what we need.



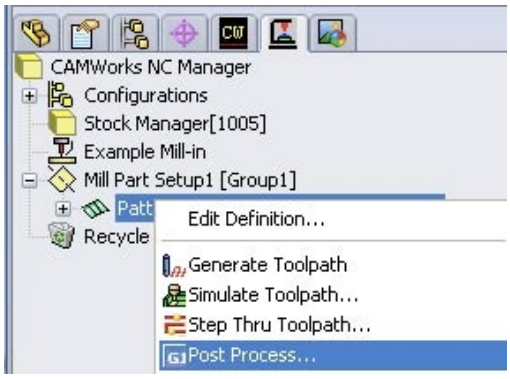
Here you see the start and return to start settings on the origin of the toolpath. Remember this was set in the NC tab. Look for this in your tool path review.

II.25.
Simulate your toolpath on your new contain areas.



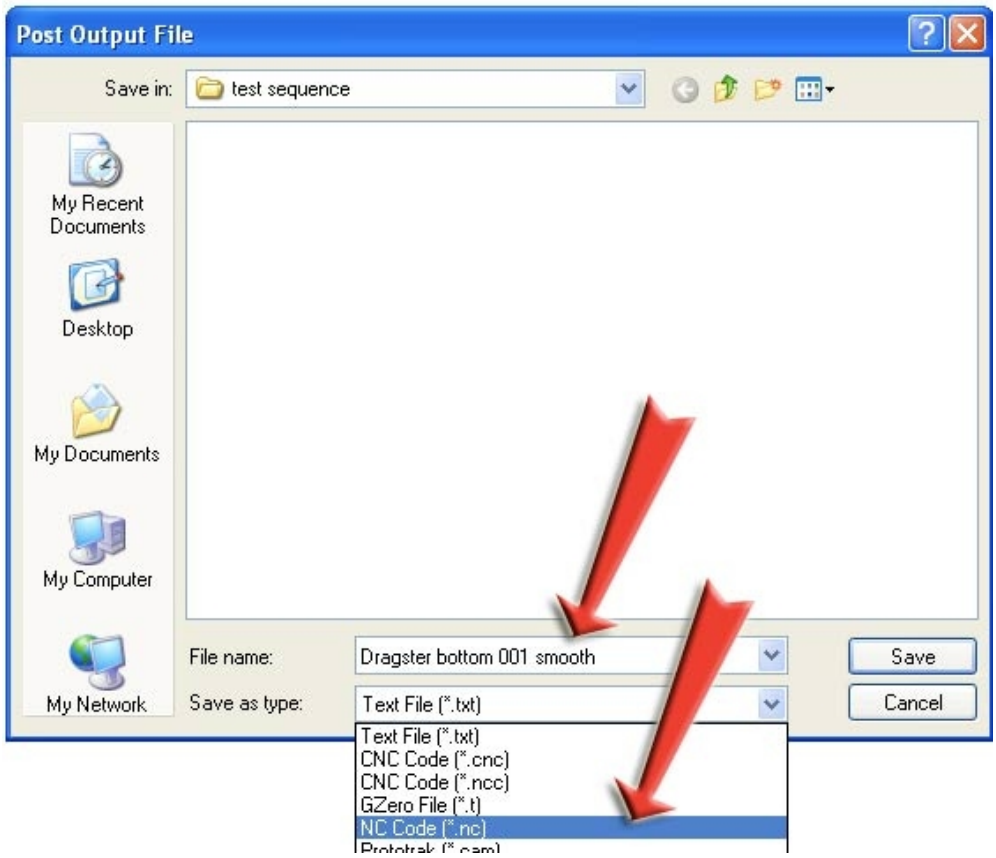
Check to make sure it simulates the way you want it run.

II.26.



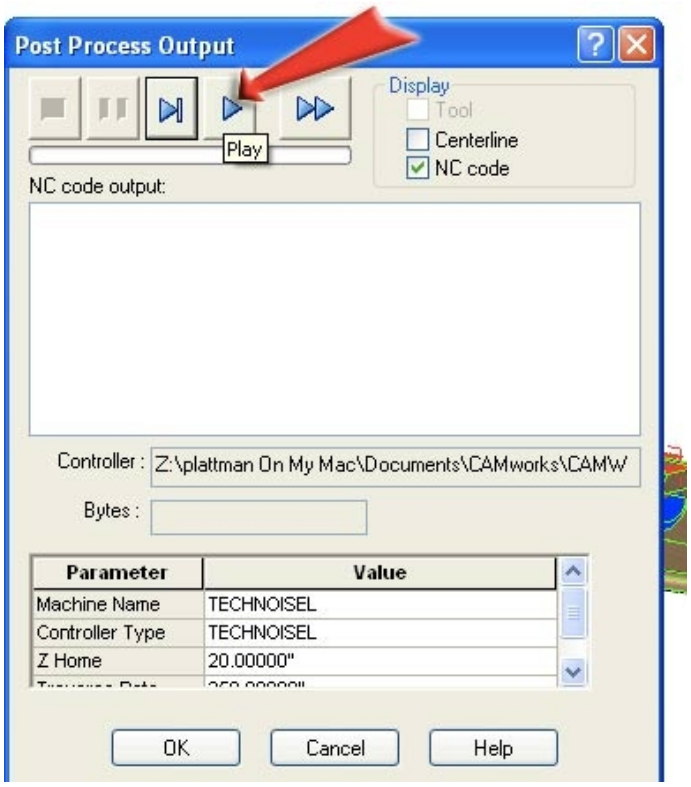
Create the toolpath by Select, then Right Click, and then Select Post Process.. to create your toolpath.

II.27.



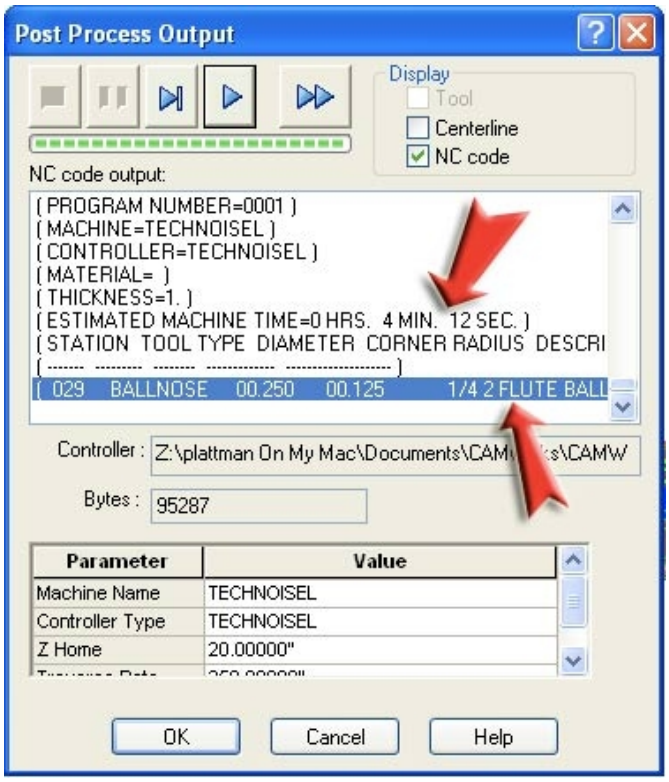
As you see above on the top arrow, save the file to a name you will remember and makes sense to the location of which side you plan to mill, then select the NC Code [*.nc] file type. The Denford uses .fnc file type, but the VR Milling software will read .nc files. The Techno machines use .nc file the Techno software interface software.

II.28.



Select the Play button and CAMWorks will write the G-Code x-y-z code to the file you just named. CAMWorks will start scrolling the x-y-z code it is writing to your file. Once it is complete, note the processing time and number of lines of code. You can scroll back up through the code to see the processing.

II.29.



Notice items. 1) The processing time. 2) the Tool number and the type of tool you are using. The estimated machine time can be affected by the manual feed/rate override on each machine you are using. You can slow the machine down or speed the machine up. In a diverse manufacturing operation, you are constantly switching the cutters for the different products you are manufacturing. This is not a big problem if you have a tool changer, however if you are using a CNC Router, this is a problem. So before you mill, make sure you are using the right tool. This can be confusing if you are using a 3/16" cutter and you need to switch to a 1/4" cutter. If you switch cutter make sure your length of cutter and the shoulder height is the same as you specified in the Tool tab in your Operation Parameters menu. The last thing to be aware of is the off set you have the machine currentlyset to use. Is this the right off set. Make you are using the correct off set to the product you are making. More of this will be covered in the CNC Router and CNC Mill setup portion of this guide.

SUMMARY:
This portion of the guide took you through the setup of the smooth tool path for your CO2 Dragster. The bottom tool paths must be run in order of "ROUGH" to "SMOOTH".

The methods we used for the bottom smooth path will be similar but different for the sides of the CO2 Dragsters. Remember the tool paths have differences that can be confusing. As such refer back to this guide to create your toolpaths. Note and write down the tool path differences in the settings tabs.