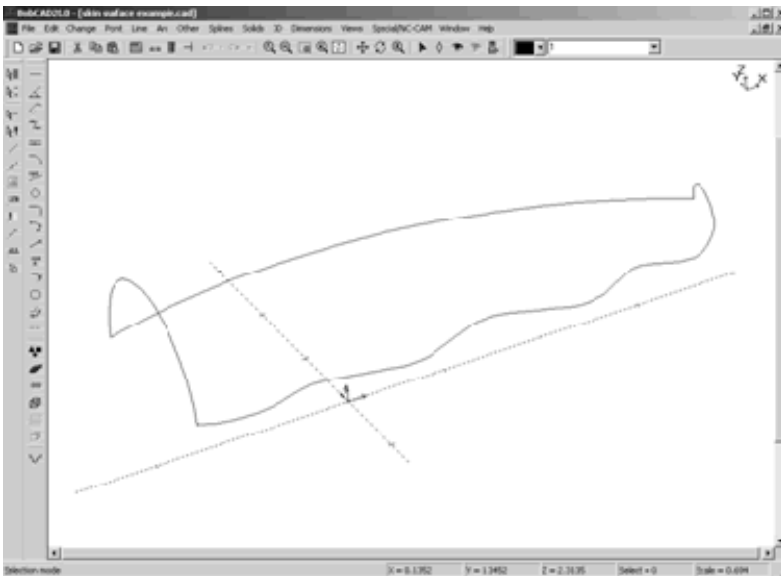


# Skin Surface & Toolpath Generation with G-Code

Version 20 offers a series of surface creation functions. One of them is the SKIN function. This lesson will teach you how to use this function using an existing 3D wireframe file that is located in the Version 20 Samples Folder.

## STEP 1

With a NEW drawing screen go to the FILE menu of the CAD side and select OPEN. Locate the BobCAD-CAM Version 20 Folder and double-click on it. Now locate the Samples folder and double click on it as well. Locate the file called, "**Skin surface example.cad**" and double click on it to open it in the CAD window.

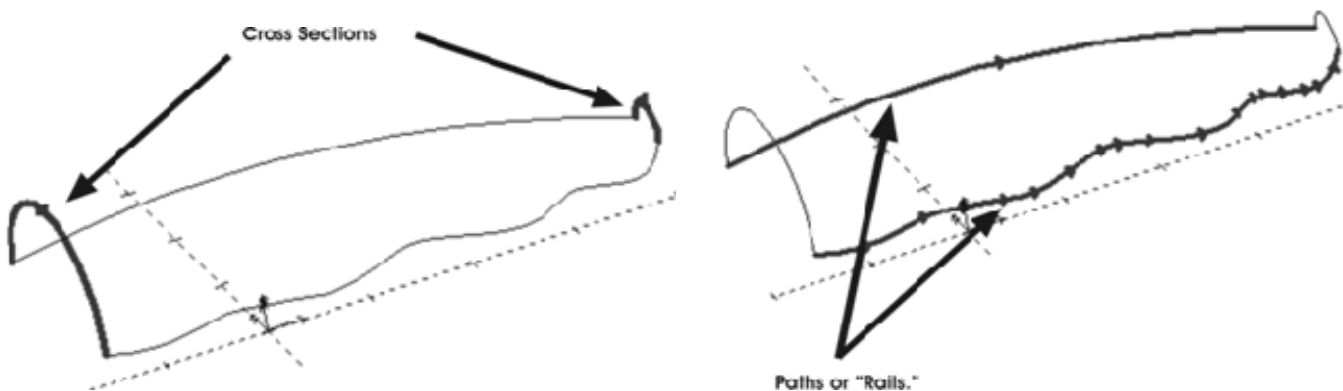


Notice that the 2 cross sectional ends are black and the 2 paths are blue. The paths are also called "rails."

## STEP 2

Because this is an Action-Object only function you must go to the SOLIDS menu, choose Surface Creation, Skin and then click NEXT to begin the chain selection on the cross section or construction geometry first.

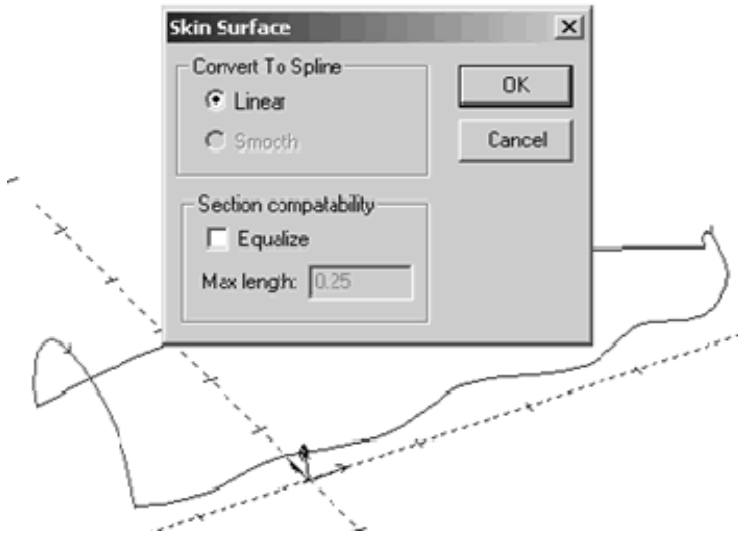
You need to select the two (end) cross sectional geometry first. Then the second part will be to select the path (rails) geometry.



### STEP 3

After selecting the skin option and the NEXT button from the Surface Creation box, you need to **chain-select** the left end first by clicking on it one time, pointing the directional arrow toward the front end of the arc and then clicking your left mouse button one more time to set the direction and then **Right-click** your mouse to complete the selection.

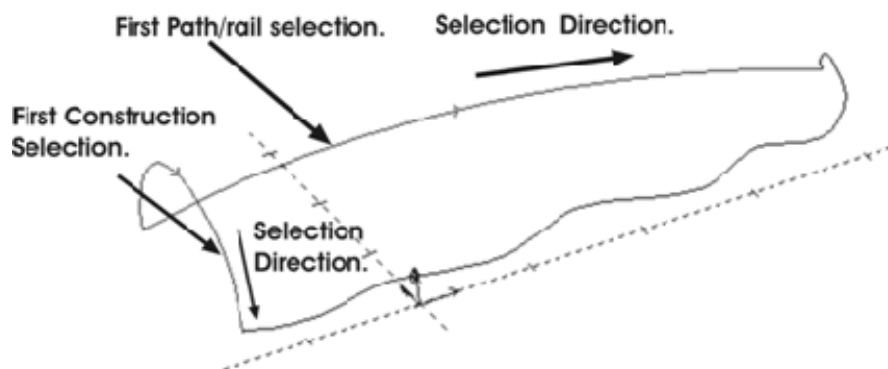
Now chain-select the second cross section in the same way with the selection directional arrow in the same direction as the first side. After right-clicking to set the direction and complete the second chain selection, **RIGHT-click your mouse once more** to open the Skin Surface box.



### STEP 4

You will now have 2 options: Convert to Spline allows you to create a “Linear” surface or “Smooth” which creates a surface based off of converted spline geometry. In most cases “Smooth” will not be available unless you are working with converted spline cross sections. By clicking OK you will be able to move on with the path/rail selections.

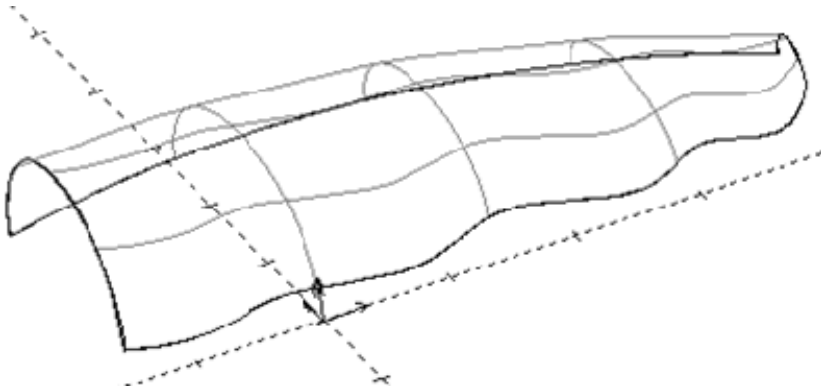
It is important to select these in the correct sequence. You need to select the path that connects to the first construction cross section with the path selection direction going AWAY from the start of that construction entity or chain.



### STEP 5

After clicking OK in the Skin Surface box, place your cursor on the first path entity or first entity starting that chain. In the above example you would click on the back “Path” arc, point your directional arrow going away from the left construction arc and click your left mouse button again and then right-click your mouse to complete that chain selection.

Now place your cursor on the far LEFT end of the FRONT chain and left-click on it one time. Point your directional arrow going toward the Right end of the chain and then take your cursor over to the last entity of that chain and click on it. Make sure that you select the last entity of that chain or hit your F3 key on the keyboard. This automatically generates the surface.



### STEP 6

Now place your cursor on an interior line of the surface and click on it to select it (*it may already be selected if your auto Preselect option is checked in the environment settings of the CAD File menu*). Now go to the Solids menu and select RENDER.



### STEP 7

Now, select the CLOSE button in the render window. If the CAM side of the software is open, this is fine. If not, go to the Special/NC CAM menu at the top menu bar and select INSERT NC. Locate the **HAAS VF Series** post processor file and double click on it to open the CAM. Now go to the CAD side of the screen and select the View All icon from the main icon bar at the top of the software.



Now, have a look at the CAM side. You will see the CAM icon bar that runs down the LEFT wall of the CAM. Toward the bottom of this CAM icon bar you will see a button that has 3D on it. Click that 3D button to turn the CAM 3D mode on for machining.



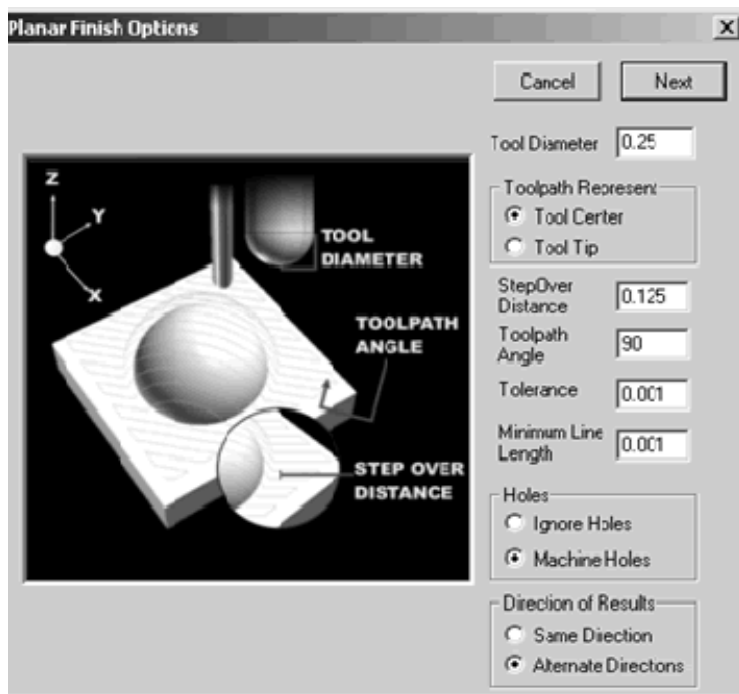
Now go to the TOOL menu on the CAM side and select TOOL CHANGE. In the tool change box enter 1 under Tool Number and enter .25 under Tool Diameter. Enter 2000 under Spindle Speed, 3 under Spindle Direction which represents Clockwise and 8 under Coolant.



**NOTE:** The HAAS VF Series post configuration will be different from other posts. If you have a HAAS CNC machine you can use this post configuration. If your post is not listed in the V20 demo, we either have it or can create it at no additional cost to the customer.

### STEP 8

With the surface selected (if it isn't selected, select it now) go to the SOLIDS menu and select GENERATE TOOLPATH. (If it is not available for selection, then you didn't select the surface and need to select the surface first.) This will open the Planar Cut Options box.



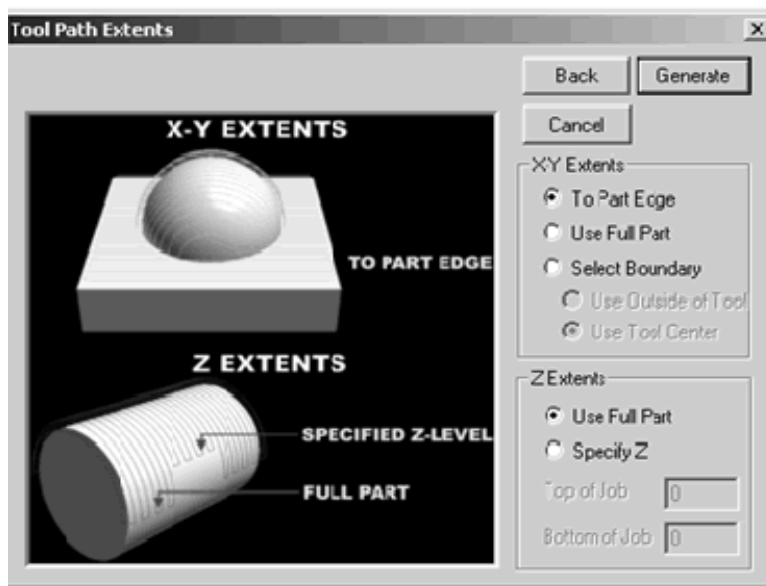
Enter the following:

- .25 for the Tool Diameter.
- Check the Tool Center option.
- Enter .125 for the Step Over distance.
- Enter 90 for the toolpath angle. This will set the toolpath going from right to left across the surface.
- Enter .001 for the Tolerance.
- Enter .001 for the Max Line Length so that we get a fine toolpath that follows the surface contour closely.
- Select Machine Holes even though there are none.
- Select the Alternate Direction option as we do not want to climb mill this surface.

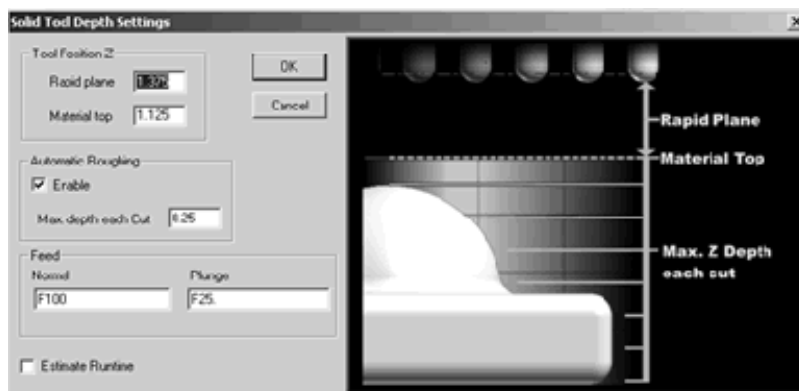
Click the NEXT button to advance in the wizard.

## STEP 9

This opens the Toolpath Extents box.



Select the “To Part Edge” cutting option as well as “Use Full Part” under the Z Extents section and click GENERATE to produce the toolpath across the surface as directed and open the Solid Tool Depth Settings box when finished.



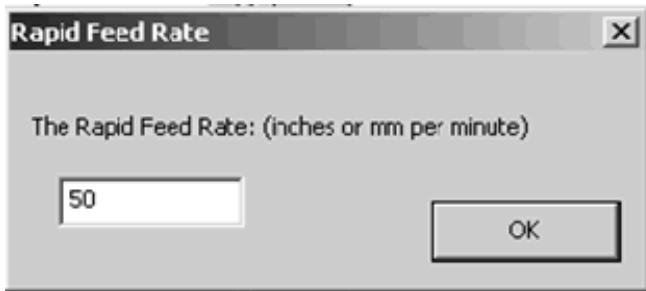
The Rapid Plane is the clearance and this box will have a value in it as the top of this surface is not at Z0. You will also have 1.125 as the top of material. This is fine as this is the material top for this example.

Check the **ENABLE** button under Automatic Roughing and enter .25 as the max depth of each cut. Go ahead and click OK. This will automatically generate the G-Code program for a HAAS VF Series control. Other post processors are available as needed.

### STEP 10

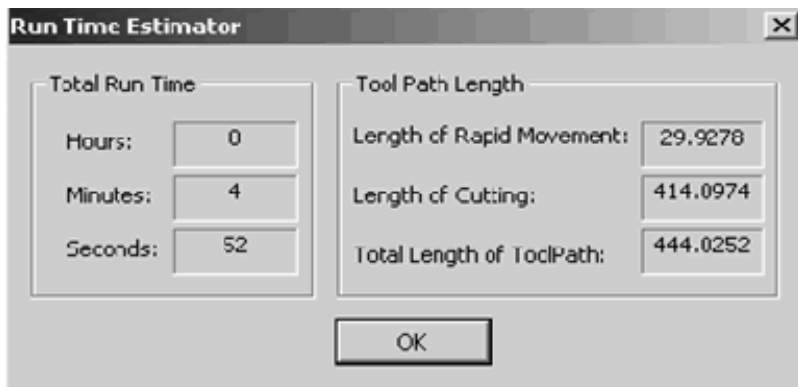
Go to the CAM EDIT menu and choose SELECT ALL. This will highlight all of the code. Now go back to the CAM EDIT menu and click on **Run Time Estimate**.

The new box will prompt you for a feed rate. You can enter in the feed rate that you will be using.



This will be in inches or millimeters per minute depending on what you have the software set to in the software environment settings (*located in the CAD side edit menu under environment and coordinates tab*). Click OK.

This will produce your run time.



Click OK to exit the run time box.

