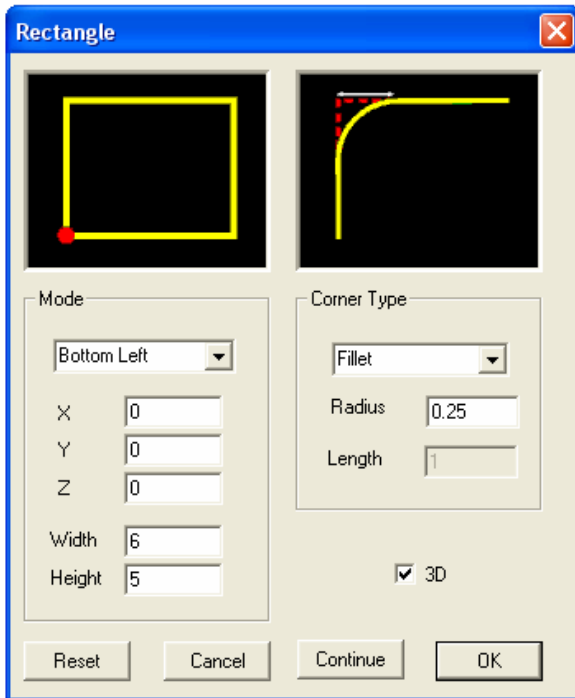


QUICK SPIRAL POCKET LESSON:

IMPORTANT: V21 has global environmental option that is called, "Auto Pre-select." This allows you to draw geometry whether freehand or by using coordinate values and have the geometry appear on the CAD screen "selected" and ready to be used for functions that are considered Object/Action. This simply allows you to draw shapes and not have to select them first to use a function. For advanced operators, this can help save time. In this lesson, we would like you to turn this option OFF to prevent any discrepancies executing the steps of this lesson. To do this, go to the main FILE menu and select ENVIRONMENT. Choose the DEFAULTS TAB and UN-CHECK the Auto Pre-select button if it is checked. Now click OK to exit the dialog.

STEP1

Select the OTHER menu and then RECTANGLE. This will bring up the rectangle dialog.



- Select Bottom Left under Mode. Use the pull-down as needed for your selection.
- Enter 0 for X, Y and the Z values. If the Z entry is grayed out, check the 3D option button as needed to access Z.
- Enter 6 for the Width and 5 for the Height.
- Under Corner Type choose Fillet from the pull-down and enter .25 as the radius value.
- Click OK to draw the shape.

STEP 2

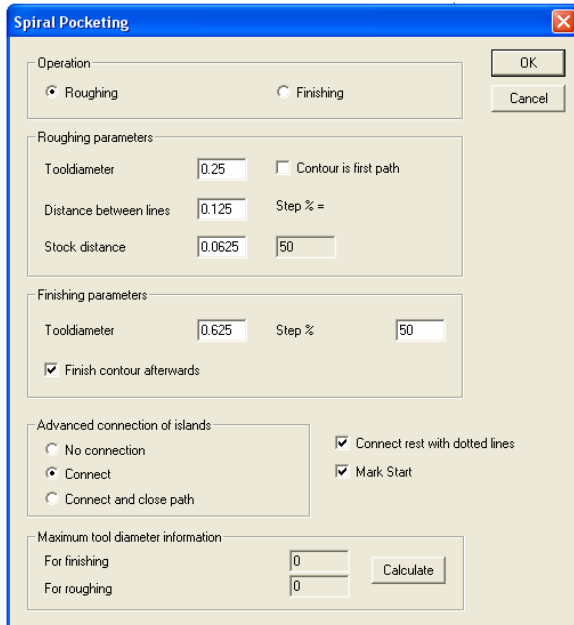
Go to the ARC menu and choose Coordinate Center. Enter the following values:

- Enter 3 for X and 2.5 for Y. Leave Z at 0.
- Enter 1 for the radius.
- The Start Angle should be 0 and the End Angle should be 360 to draw a full circle.
- Click OK to draw the circle.

Now that we have created our profiles we are ready to create our toolpath.

STEP 3

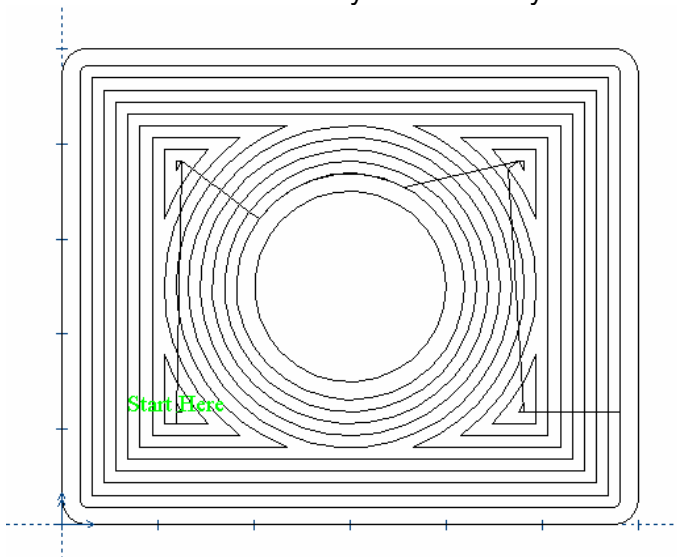
Go to the OTHER menu and choose POCKET. Now click the Spiral operation button. This opens the Spiral Pocket dialog box.



- In the Operation section, select Roughing.
- In the Roughing Operations section enter .25 for the tool diameter.
- Un-check the “Contour is first Path” option box if it is checked.
- Enter .125 as the Distance Between Lines. This is our tool step-over.
- Enter .0625 for the Stock Distance value. This will be the amount left around our profile walls allowing a finishing operation.
- Under the section, “Advanced Connection of Islands” check the Connect option.
- Check the Connect Rest with Dotted Lines as well as the Mark Start options.
- Click OK.

STEP 4

Place your cursor on the outside profile and click your left mouse button to select it. Now click on both circles one at a time the same way. Then click your RIGHT mouse button to create the toolpath.

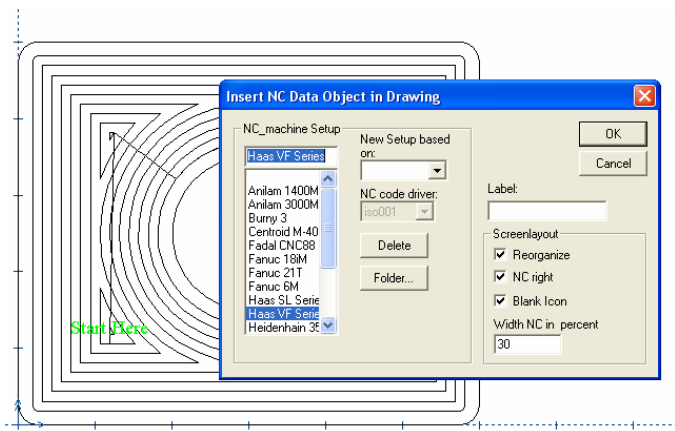


The toolpath that we created is basically editable line and arc geometry at this point. Now let's machine it.

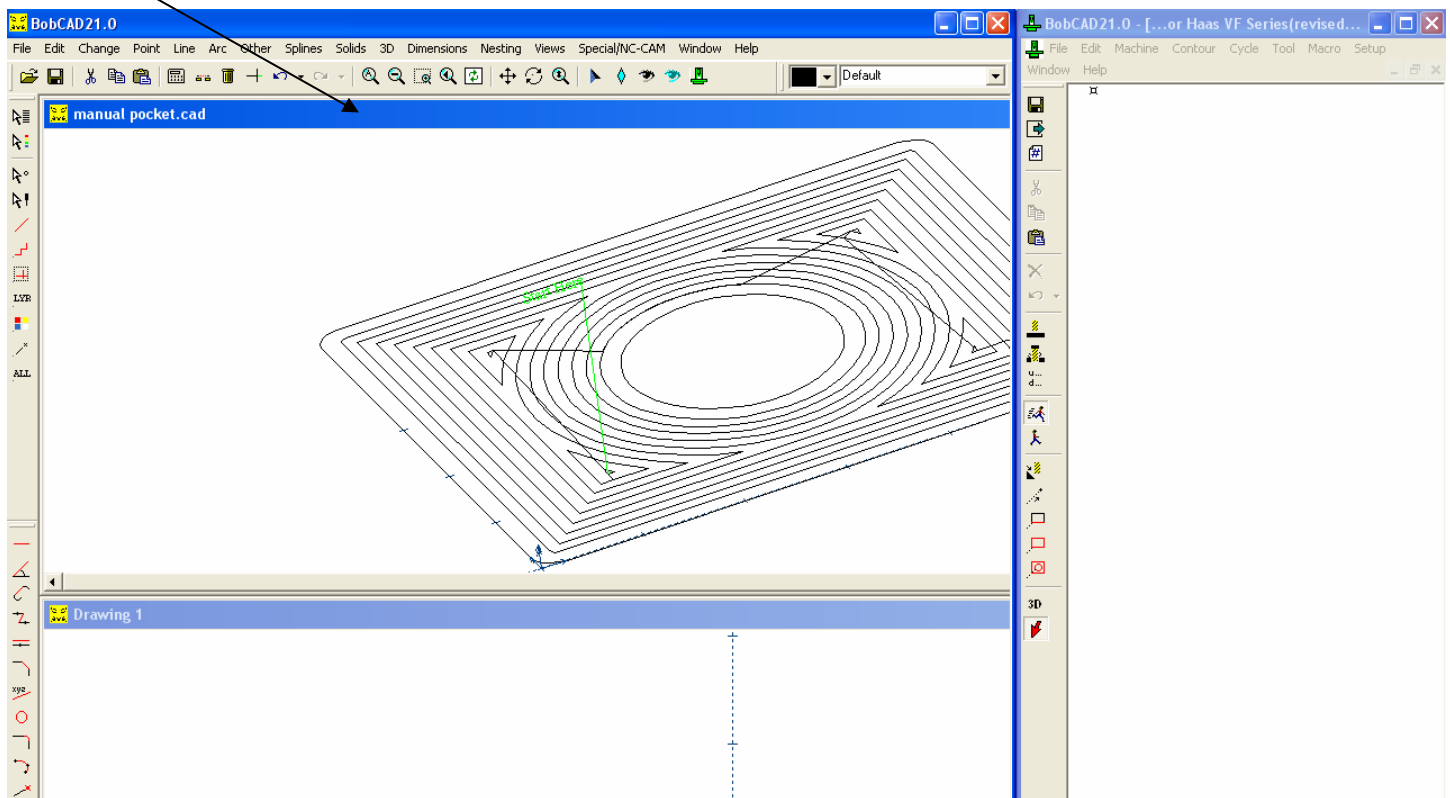
STEP 5

Go to the Special NC CAM main menu and choose Insert NC.

Choose the HAAS VF Series post processor from the list of configurations in the box.



Click OK to open the CAM in the right hand side of the software. When the CAM opens your CAD screen may split into multiple screens if you have multiple drawing screens open. You can place your cursor on the **window bar** and double click your mouse.

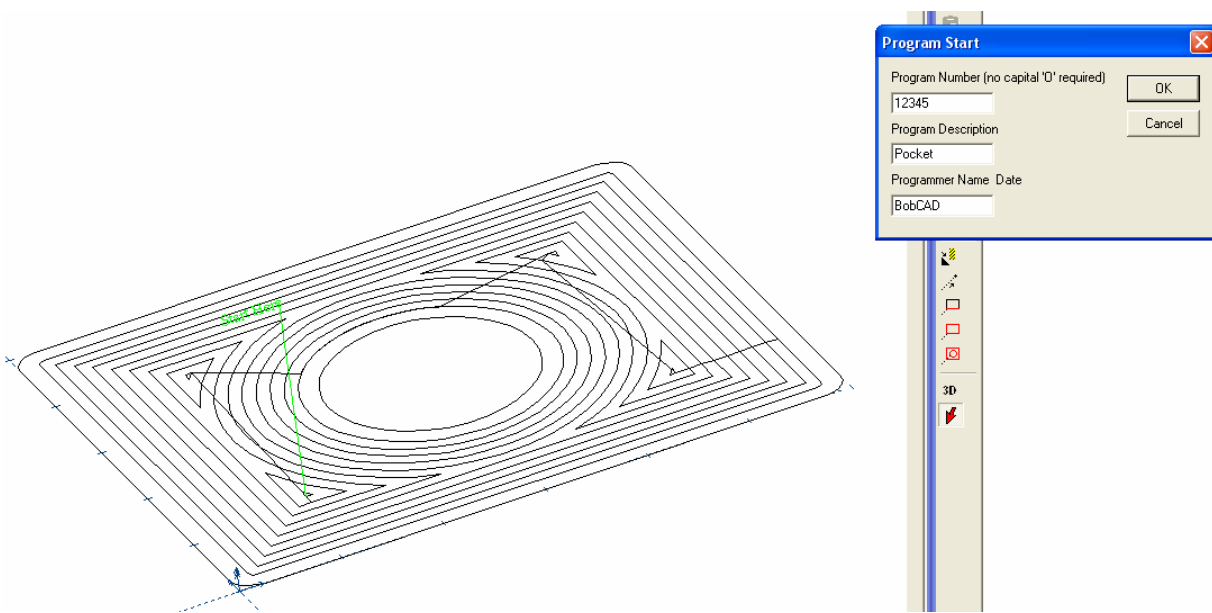


Before moving on, go to the CAD 3D main menu and turn 3D ON. This will be your first option in the 3D menu. Now select the View ALL button from the main CAD toolbar so that the part zooms into view better. This is the small function icon that has a magnifying glass with the letter A in it.



STEP 6

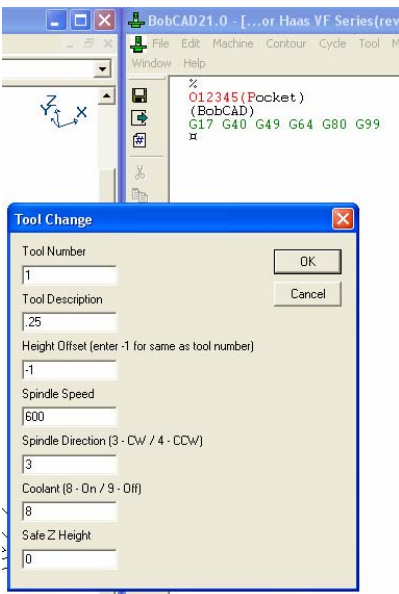
At this stage in the process you will want to build the header or start block of the program. Do this by going to the CAM Macro menu and choosing Program and then Start.



The current HAAS VF Series script will then ask for a program number, description and allow for a programmer name and date as needed. Enter 12345 for the number (HAAS Requirement), call the description: Pocket and click OK to advance.

STEP 7

Now the script will ask for more information.



- Enter 1 for the tool number.
- Enter .25 for the tool diameter.
- The height offset can be set at -1 because this will make the H & T numbers match. The height and tool numbers need to match for HAAS.
- Spindle Speed can be 600 as this is a roughing cycle.

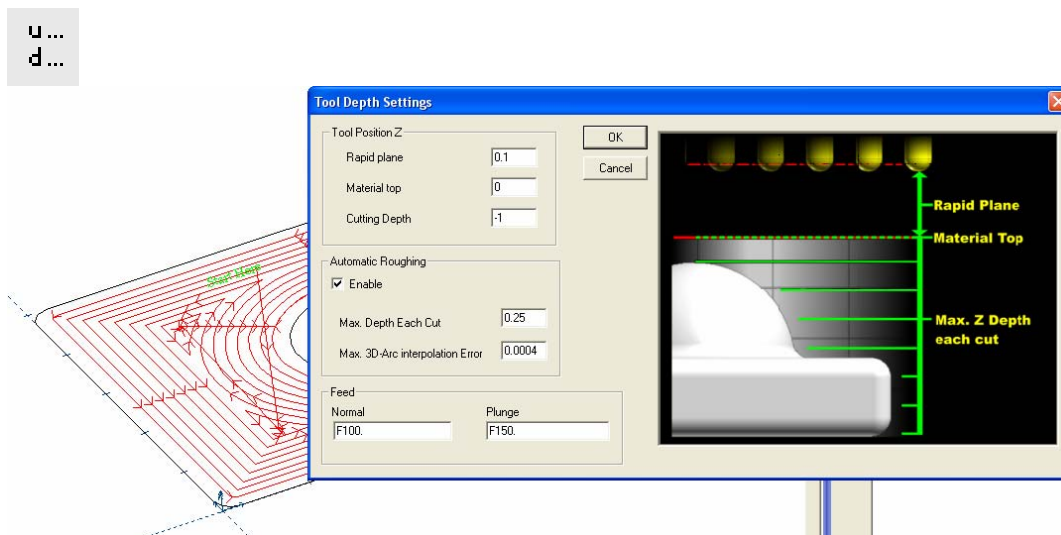
- Initial Spindle Direction is 3. This is clockwise.
- Coolant is 8. This means ON.
- Safe Z height refers to the amount of height you want the tool to retract to during rapid moves so that you can avoid clamps or fixtures. Leaving this at 0 would make the tool rapid to the start point of the pocket toolpath at the height used in the tool depth settings dialog called, "Rapid Plane." This value in the script does not overwrite the value entered in the Tool Depth Settings Parameters box under Rapid Plane. You can enter .1 as the safe Z-Height so that the tool rapids just above the part.
- Click OK. This brings up a Rapid Location box. Leave both values at 0 and click OK. Once again, this is a HAAS script being used for machining this example. Your machine control may be different and thus, requiring a custom script for you.
- Click OK to insert the start block into the program.

STEP 8

Now you want to select the toolpath. Go to the EDIT menu on the CAD side, choose Select Entities and then CHAIN. Now place your cursor on the green vertical starting line of the toolpath and click your left mouse button to select it. Move your cursor lower than the mid-point of this line to make the directional arrow point DOWN. When you have done this click your left mouse button again to secure the direction and then hit the F3 key on your keyboard to select the rest of the chain of toolpath.

STEP 9

With your toolpath selected go to the CAM side and select the Tool Depth Settings icon from the main CAM toolbar. This is the function icon that has a small U & D on it.



- Enter .1 for the Rapid Plane. This is your clearance.
- Set the material top to 0.
- Enter -1 for the Cutting Depth.
- Check the Enable button under Automatic Roughing.
- Enter .25 for the Max Depth Each Cut.

We will basically be doing 4 roughing passes here. Now click OK.

STEP 10

With the toolpath selected, go to the Machine menu on the CAM side and choose AUTO. This will automatically create the roughing toolpath on the screen as well as your G-Code program for it.

You have completed this lesson.