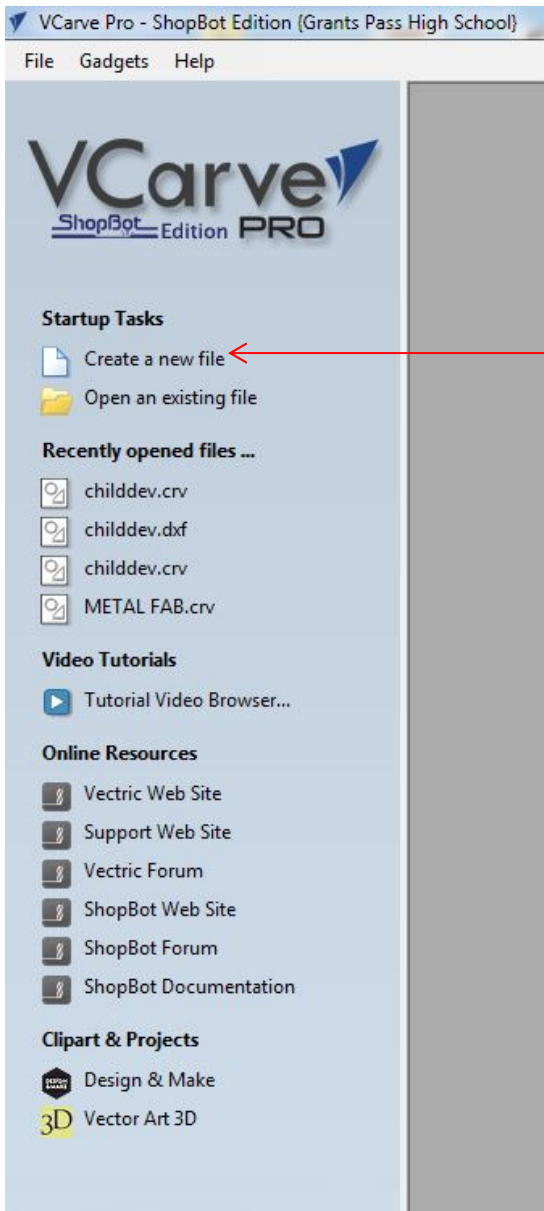


## Summer STEM Academy

### VCarve Name Sign Instructions

Follow these instructions to draw your personalized sign on the CNC router in the Woodshop / Manufacturing shop.

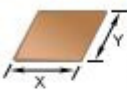
1. Log on to your computer and double click on the “VCarve Pro” icon on the desktop. Your screen should look like the image below.



Use the left mouse button to click on the “Create a new file” icon.

### Job Setup

#### Job Size (X & Y)



Width (x): 6.0 inches

Height (y): 6.0 inches

#### Material (Z)



Thickness (z)

0.75 inches

#### XY Datum Position



☐ Use Offset

X: 0.0

Y: 0.0

#### Units



☒ inches

☐ mm

#### Modeling Resolution



Standard (fastest)

#### Appearance



Blue Plastic

Solid Color:

OK

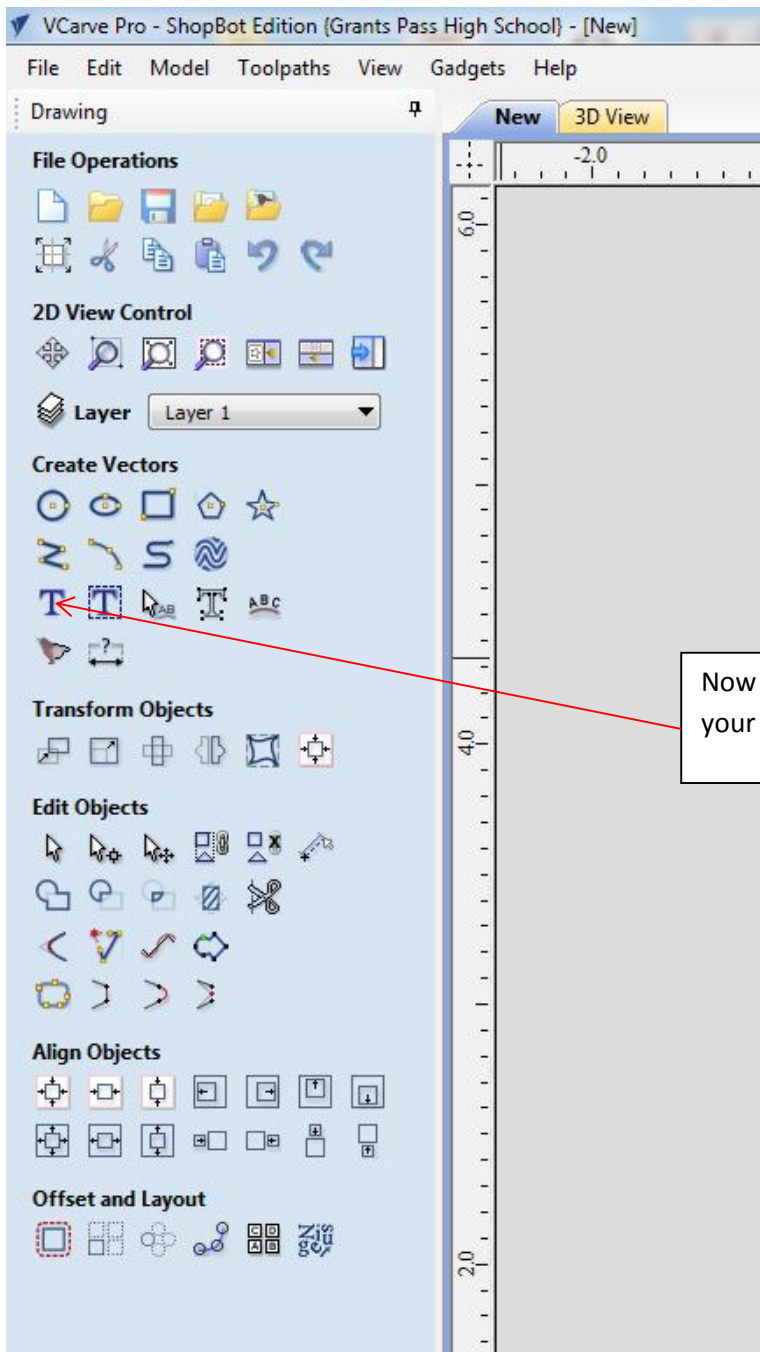
Cancel

Here is where you will set your material dimensions. Click in the "Width" box and enter 6.0

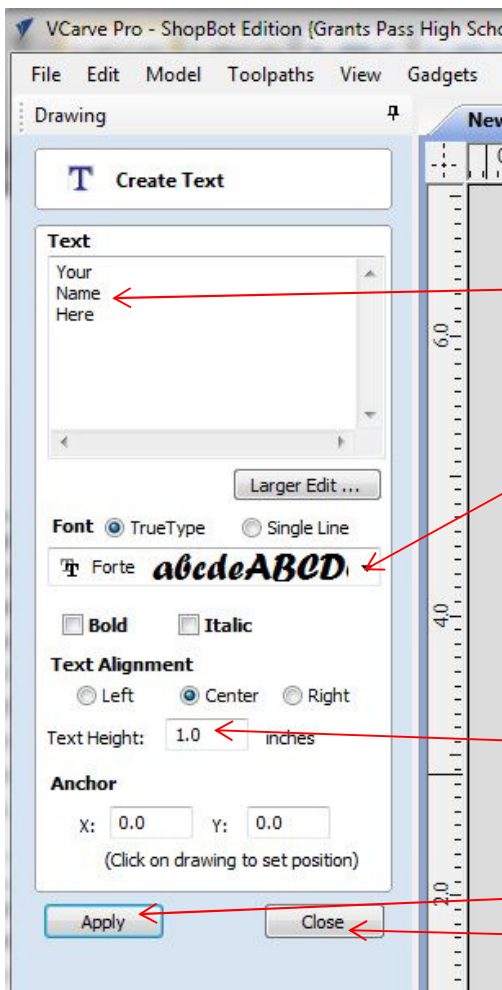
Now click on the "Height" box and enter a value of 6.0

Click in the "Thickness" box and enter 0.75

All of the remaining values should look just like they do in the example on the left. Double check everything and make sure your settings match what you see on the left and then click on the "OK" button at the bottom.



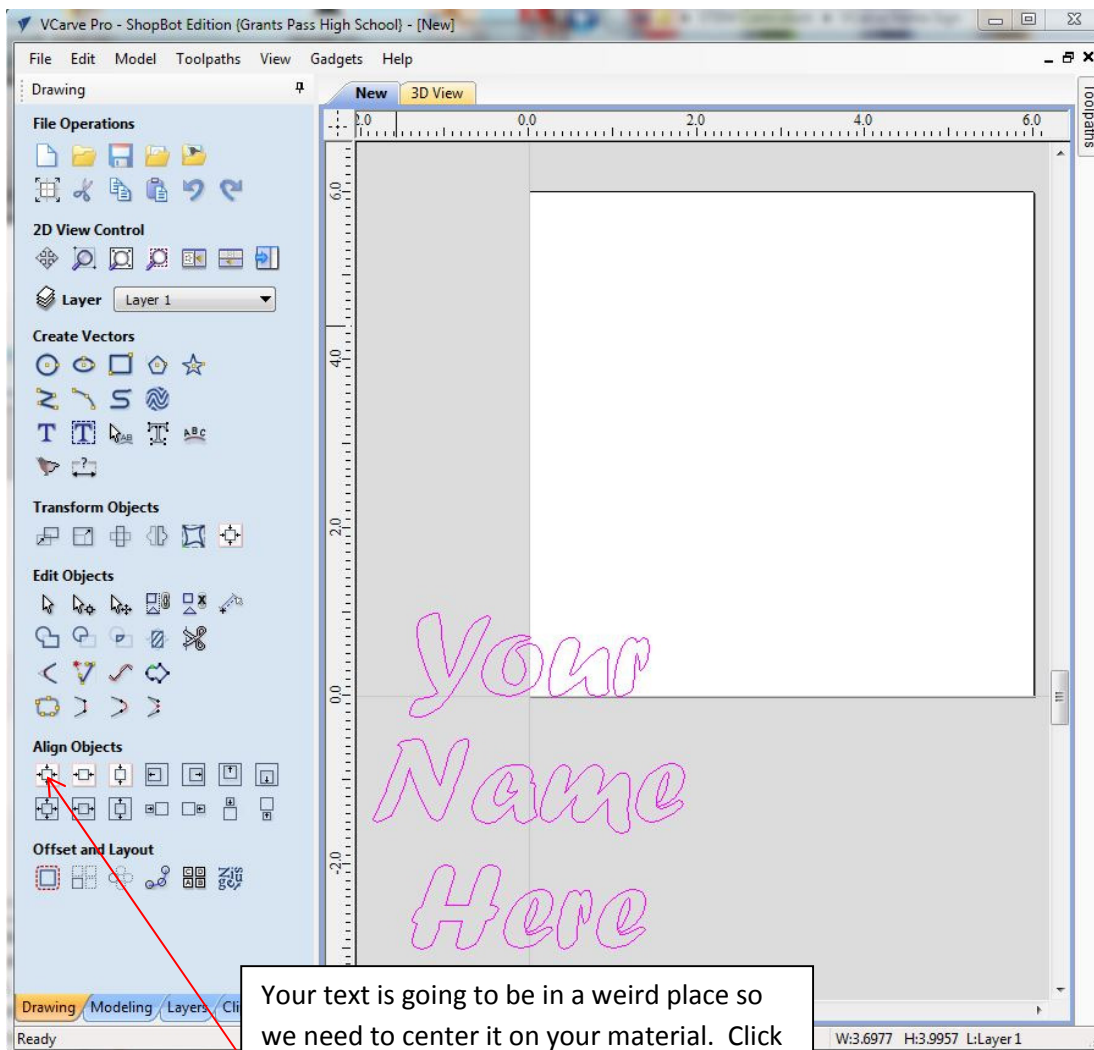
Now we are going to enter some text on to your material. Click on the "Text" icon.



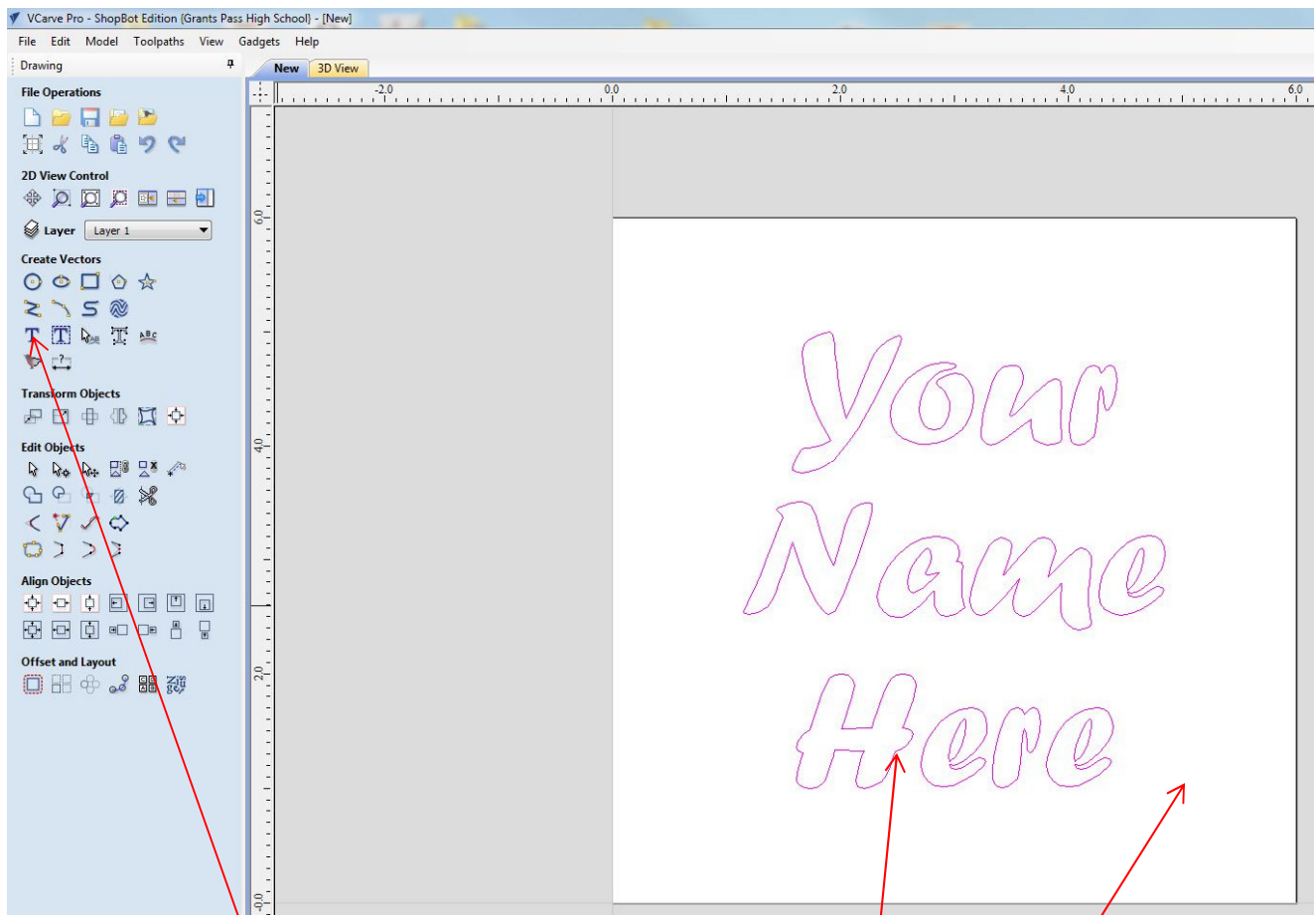
You need to type your name (or whatever you would like to cut out) in the Text box. Use the enter key when entering your text so that every word is on its own line. You can change the font to whatever you would like to use but there are some fonts that won't work too well with the V-carving you will be doing so check with your instructor if you aren't sure.

We will start out with a "Text Height" of 1.0 inch. If your name is too big we will need to make it smaller so it fits on your material.

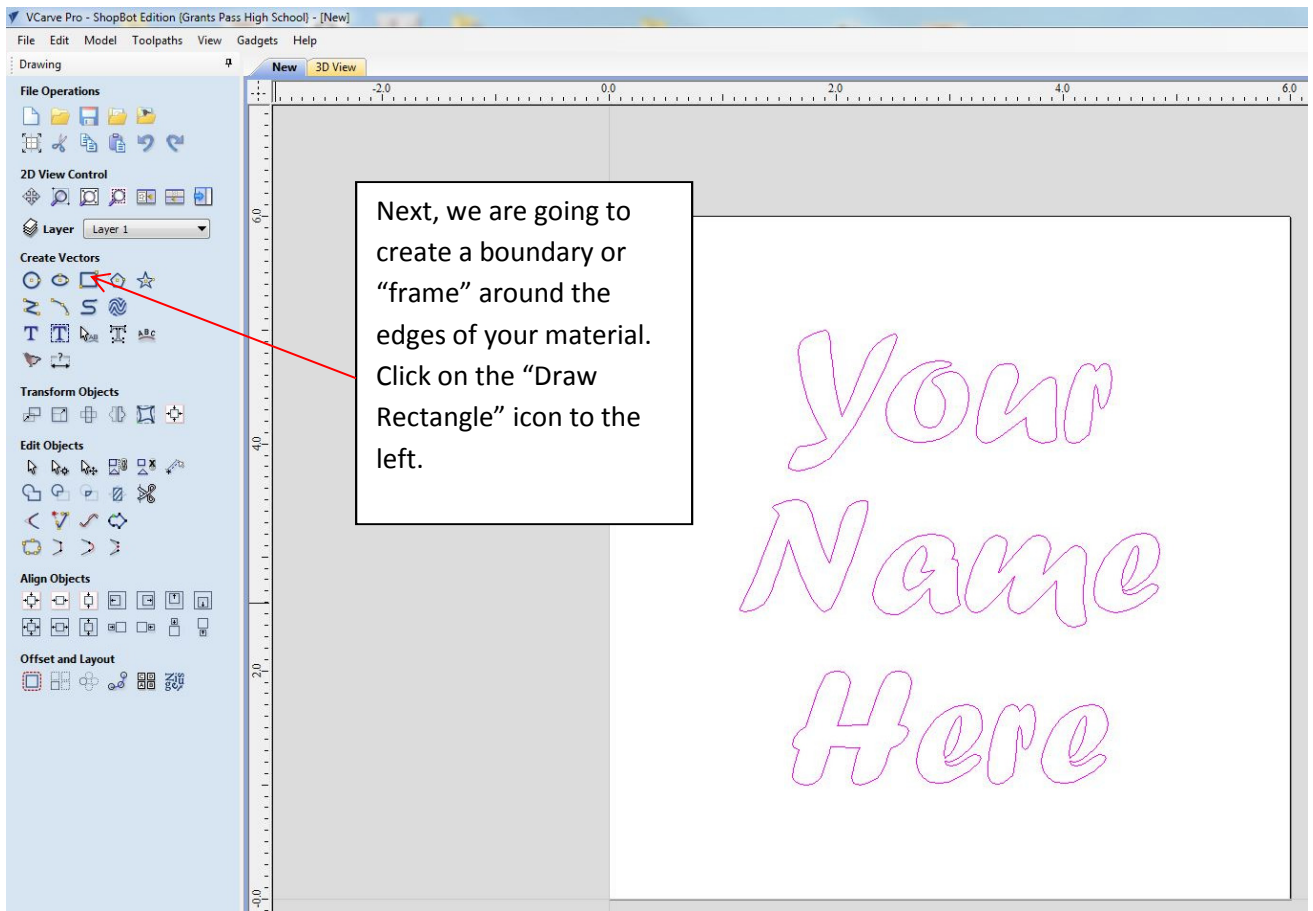
Now you can click on the "Apply" button, then click on the "Close" button.



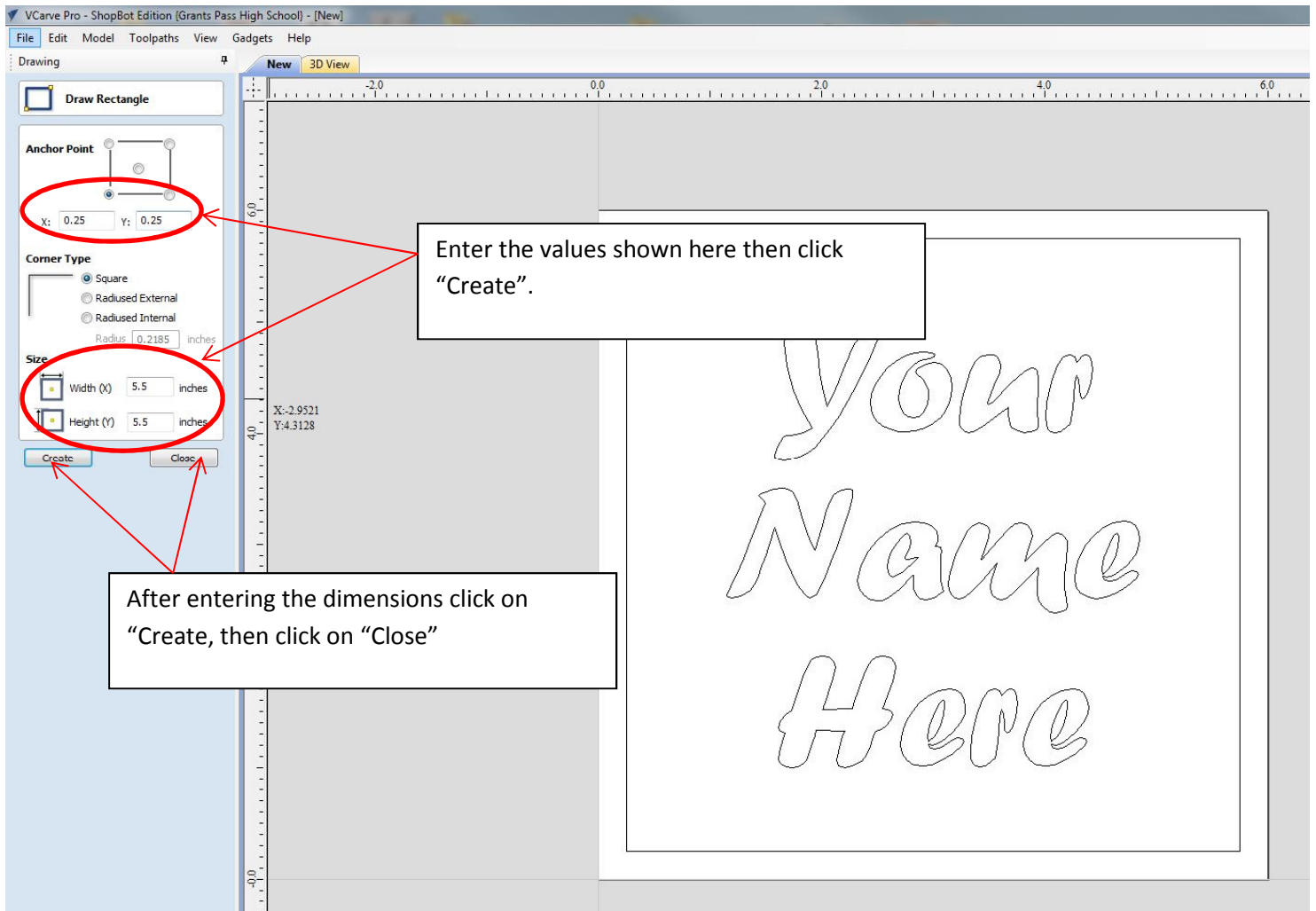
Your text is going to be in a weird place so we need to center it on your material. Click on the "Center on Material" button under "Align Objects" to place your text in the middle of the material.



Your screen and work space should look like this now. You can click on the text that you created and it will turn red, meaning that it is active. If you want to make changes, just click on the text so it turns red then click on the "T" (text icon) and you can make any changes you like such as making the text size smaller so that your name fits on the material. Clicking anywhere on your material outside of the text turns it black and makes it inactive.

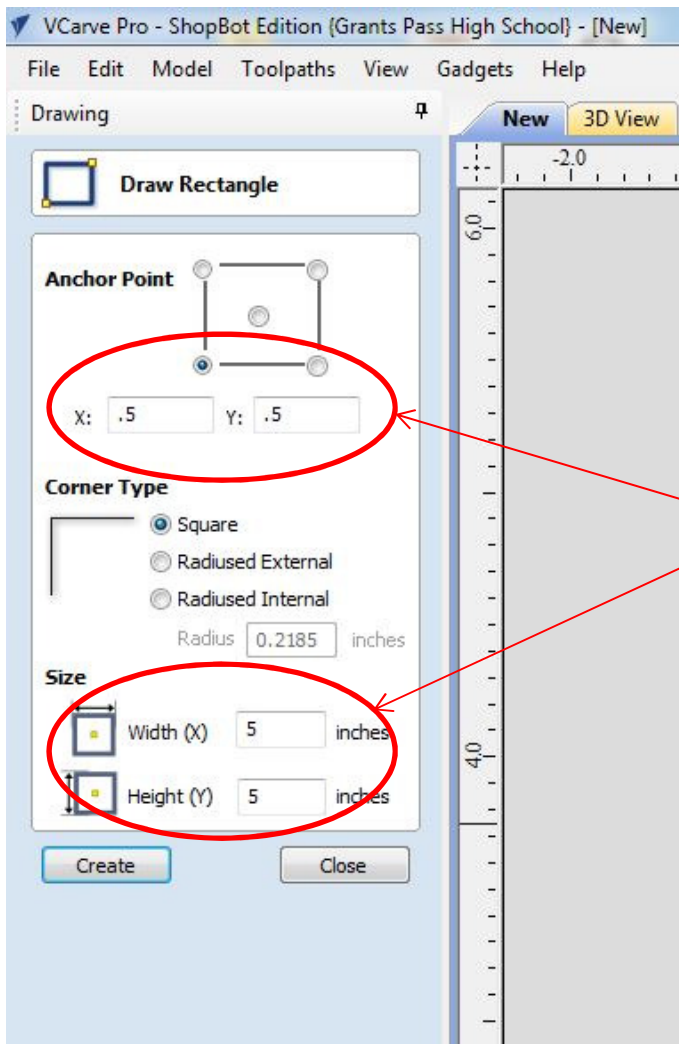




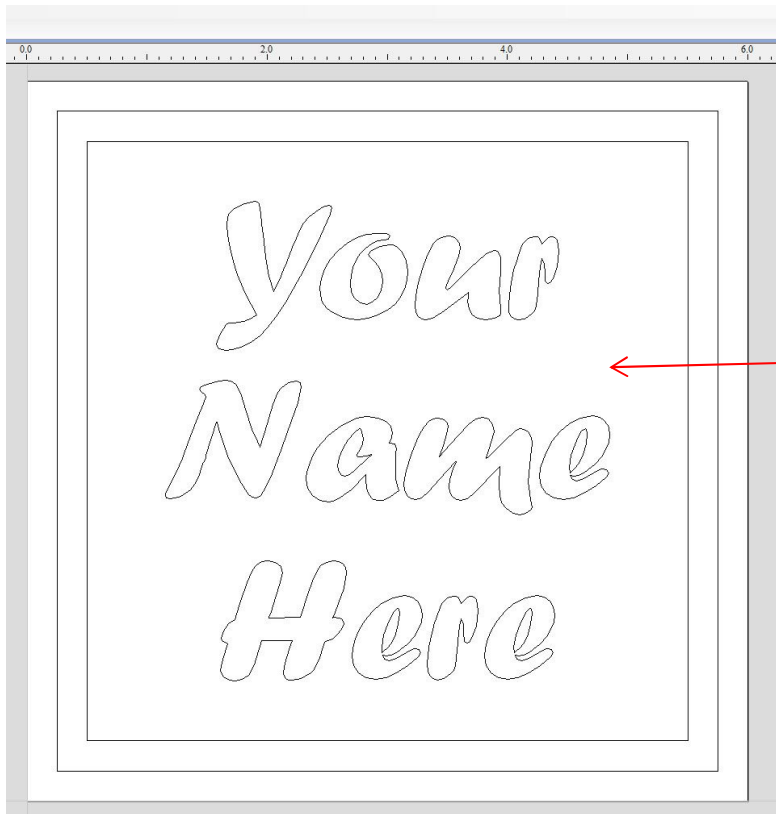


Your drawing should look like this one (but it will have your text in it)



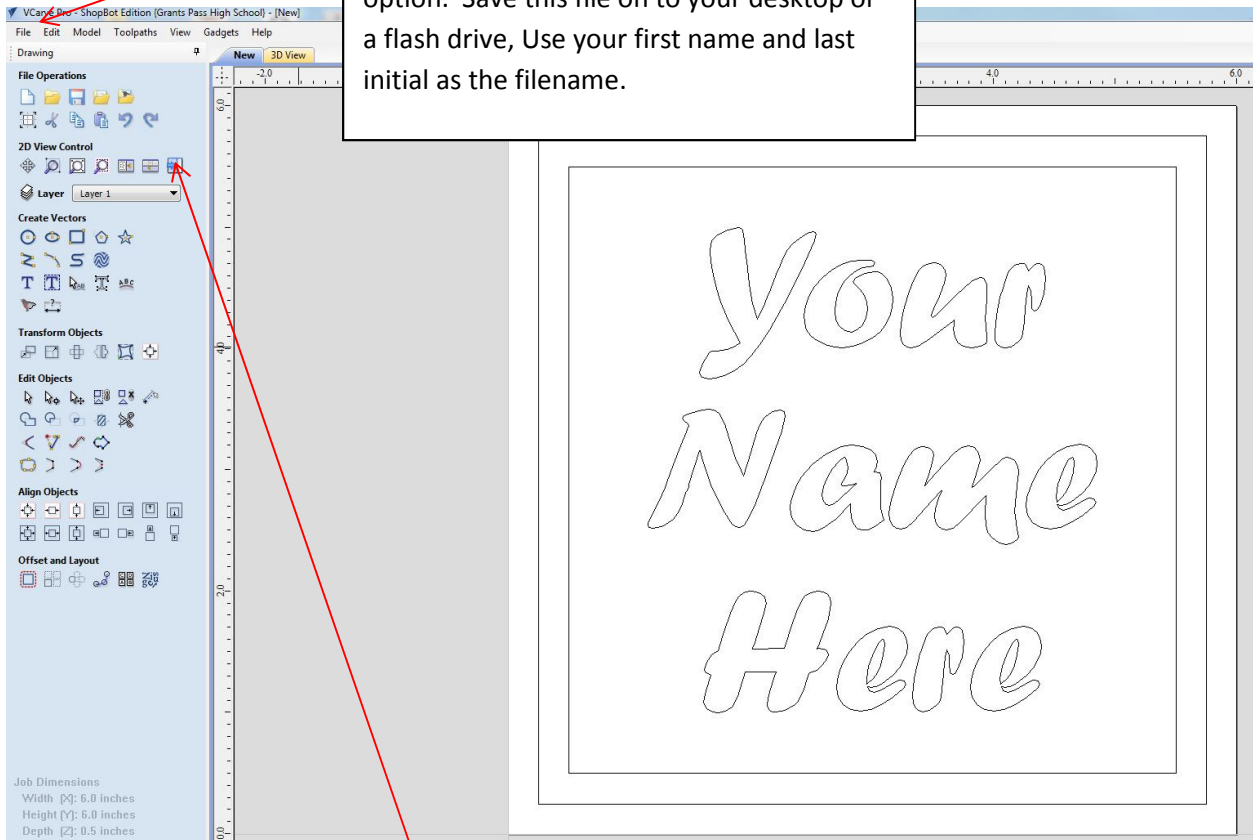


We are going to create a second box that is a quarter of an inch smaller than the first one all the way around. This will define the vcarve "frame" that we will be cutting. Enter all of the values shown here then click "Create" then "Close".

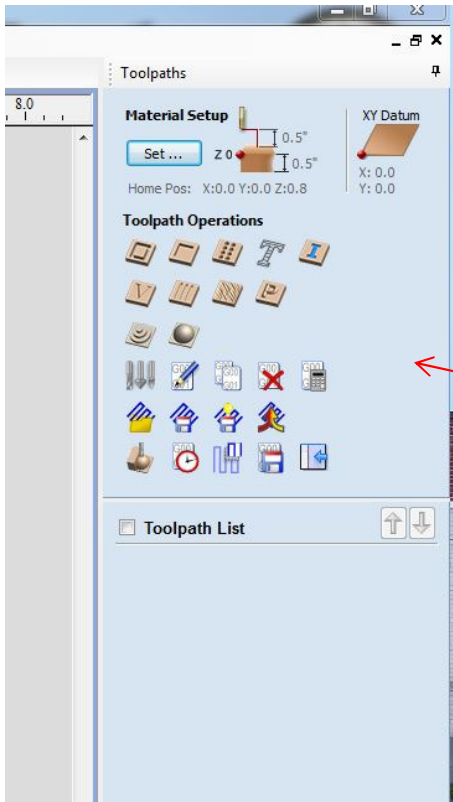


Your project should look like the one here (except that it has your text in it).

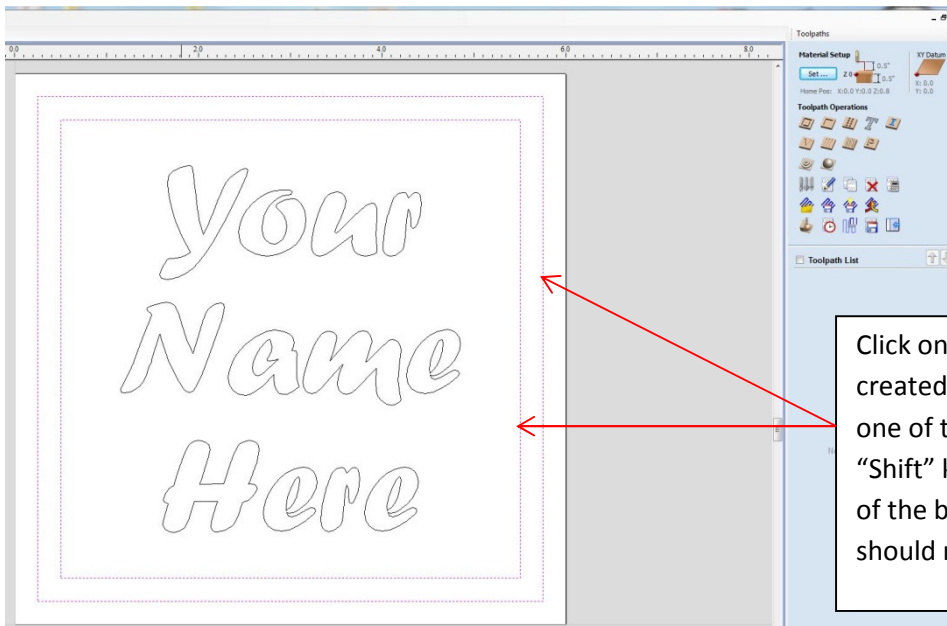
Before going any further you should save your drawing. Click on the “File” button in the upper left corner of the window then move down and click on the “Save As” option. Save this file on to your desktop or a flash drive, Use your first name and last initial as the filename.



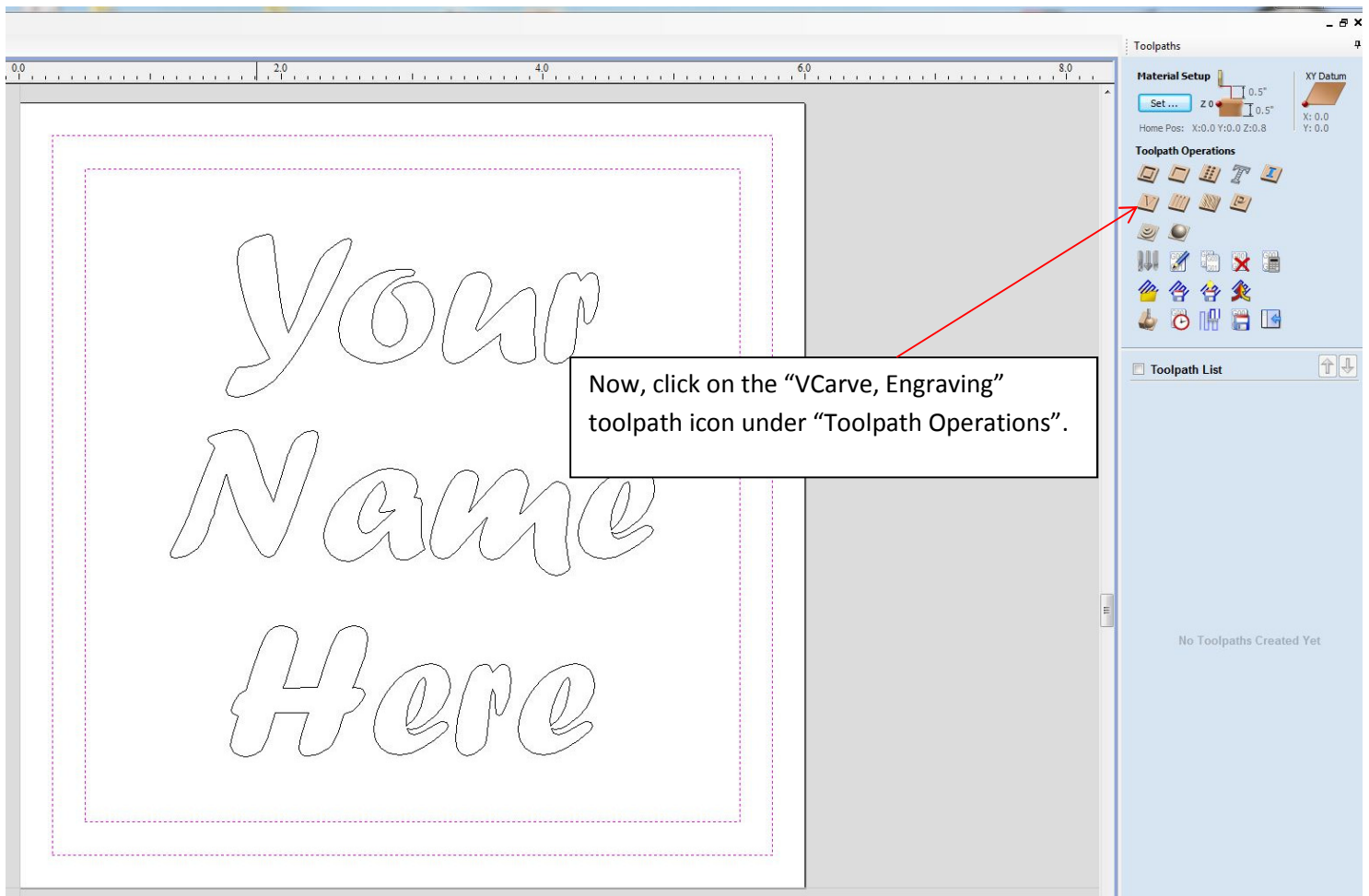
Now we are going to work on generating the toolpaths that will eventually make your sign. Click on the “Switch to Toolpaths” icon.



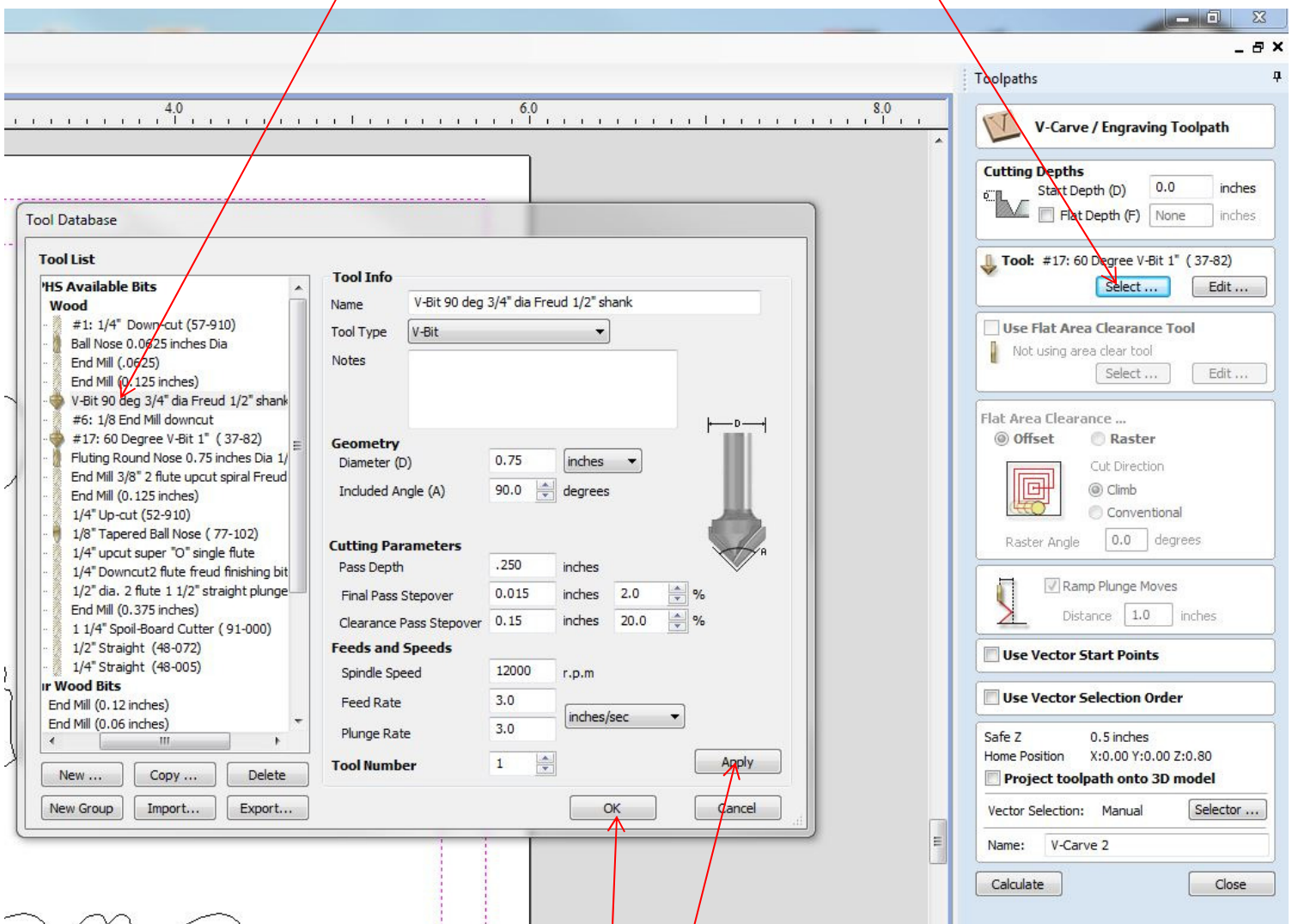
The toolbar that was on the left side of the screen has now been replaced with the “Toolpaths” toolbar on the right side. This is where we will generate the “G Code” that will instruct the machine just how to go about cutting out your sign.



Click on one of the two boxes that you created earlier and it should turn red. Once one of them is activated, hold down the “Shift” key and click on the other box. Both of the boxes that define your “Frame” should now be red.



Now we need to choose the bit and all of the parameters that go along with your bit choice. Click on the Tool “Select” button and the window seen below will pop-up. Look down in the “Tool List” on the left and click on the 90 deg V-Bit.



Once you have all of the numbers in the Tool Database the same as you see here, click on the “Apply” button to save all of your settings then click on “OK”.

**Tool Database**

**Tool List**

**Available Bits**

- #1: 1/4" Down-cut (57-910)
- Ball Nose 0.0625 inches Dia
- End Mill (.0625)
- End Mill (0.125 inches)
- V-Bit 90 deg 3/4" dia Freud 1/2" shank
- #6: 1/8 End Mill downcut
- #17: 60 Degree V-Bit 1" ( 37-82)
- Fluting Round Nose 0.75 inches Dia 1/2" sh
- End Mill 3/8" 2 flute upcut spiral Freud 75-1
- End Mill (0.125 inches)
- 1/4" Up-cut (52-910)
- 1/8" Tapered Ball Nose ( 77-102)
- 1/4" upcut super "O" single flute
- 1/4" Downcut 2 flute freud finishing bit
- 1/2" dia. 2 flute 1 1/2" straight plunge
- End Mill (0.375 inches)
- 1 1/4" Spoil-Board Cutter ( 91-000)
- 1/2" Straight (48-072)
- 1/4" Straight (48-005)

**Good Bits**

- Mill (0.12 inches)
- Mill (0.06 inches)

**Tool Info**

Name: V-Bit 90 deg 3/4" dia Freud 1/2" shank

Tool Type: V-Bit

Notes:

**Geometry**

Diameter (D): 0.75 inches

Included Angle (A): 90.0 degrees

**Cutting Parameters**

Pass Depth: 0.25 inches

Final Pass Stepover: 0.015 inches 2.0 %

Clearance Pass Stepover: 0.15 inches 20.0 %

**Feeds and Speeds**


Spindle Speed: 12000 r.p.m

Feed Rate: 3.0 inches/sec

Plunge Rate: 3.0

**Tool Number**: 1

Buttons: New ..., Copy ..., Delete, New Group, Import..., Export..., Apply, OK, Cancel



We need to tell the machine how fast we want it to work. Set all of the numbers to exactly what you see here.



Tool Database

**Tool List**

**Available Bits**

- od
- #1: 1/4" Down-cut (57-910)
- Ball Nose 0.0625 inches Dia
- End Mill (.0625)
- End Mill (0.125 inches)
- V-Bit 90 deg 3/4" dia Freud 1/2" shank
- #6: 1/8 End Mill downcut
- #17: 60 Degree V-Bit 1" ( 37-82)
- Fluting Round Nose 0.75 inches Dia 1/2" sh
- End Mill 3/8" 2 flute upcut spiral Freud 75-1
- End Mill (0.125 inches)
- 1/4" Up-cut (52-910)
- 1/8" Tapered Ball Nose ( 77-102)
- 1/4" upcut super "O" single flute
- 1/4" Downcut2 flute freud finishing bit
- 1/2" dia. 2 flute 1 1/2" straight plunge
- End Mill (0.375 inches)
- 1 1/4" Spoil-Board Cutter ( 91-000)
- 1/2" Straight (48-072)
- 1/4" Straight (48-005)

**ood Bits**

- Mill (0.12 inches)
- Mill (0.06 inches)

New ... Copy ... Delete

New Group Import... Export...

**Tool Info**

Name V-Bit 90 deg 3/4" dia Freud 1/2" shank

Tool Type V-Bit

Notes

**Geometry**

Diameter (D) 0.75 inches

Included Angle (A) 90.0 degrees

**Cutting Parameters**

Pass Depth 0.25 inches

Final Pass Stepover 0.015 inches 2.0 %

Clearance Pass Stepover 0.15 inches 20.0 %

**Feeds and Speeds**

Spindle Speed 12000 r.p.m


Feed Rate 3.0 inches/sec

Plunge Rate 3.0

**Tool Number** 1

Apply

OK Cancel



Once all of the numbers match what is shown here, click on the "Apply" button, then click on "Ok".



Toolpaths

### V-Curve / Engraving Toolpath

**Cutting Depths**

Start Depth (D)  inches

☐ Flat Depth (F)  inches


**Tool:** V-Bit 90 deg 3/4" dia Freud 1/2" shank

☐ **Use Flat Area Clearance Tool**

Not using area clear tool.

**Flat Area Clearance ...**

☒ **Offset** ☐ **Raster**



Cut Direction

☒ **Climb** ☐ **Conventional**

Raster Angle  degrees

☒ **Ramp Plunge Moves**

Distance  inches

☐ **Use Vector Start Points**

☐ **Use Vector Selection Order**

Safe Z  inches

Home Position X:0.00 Y:0.00 Z:0.80

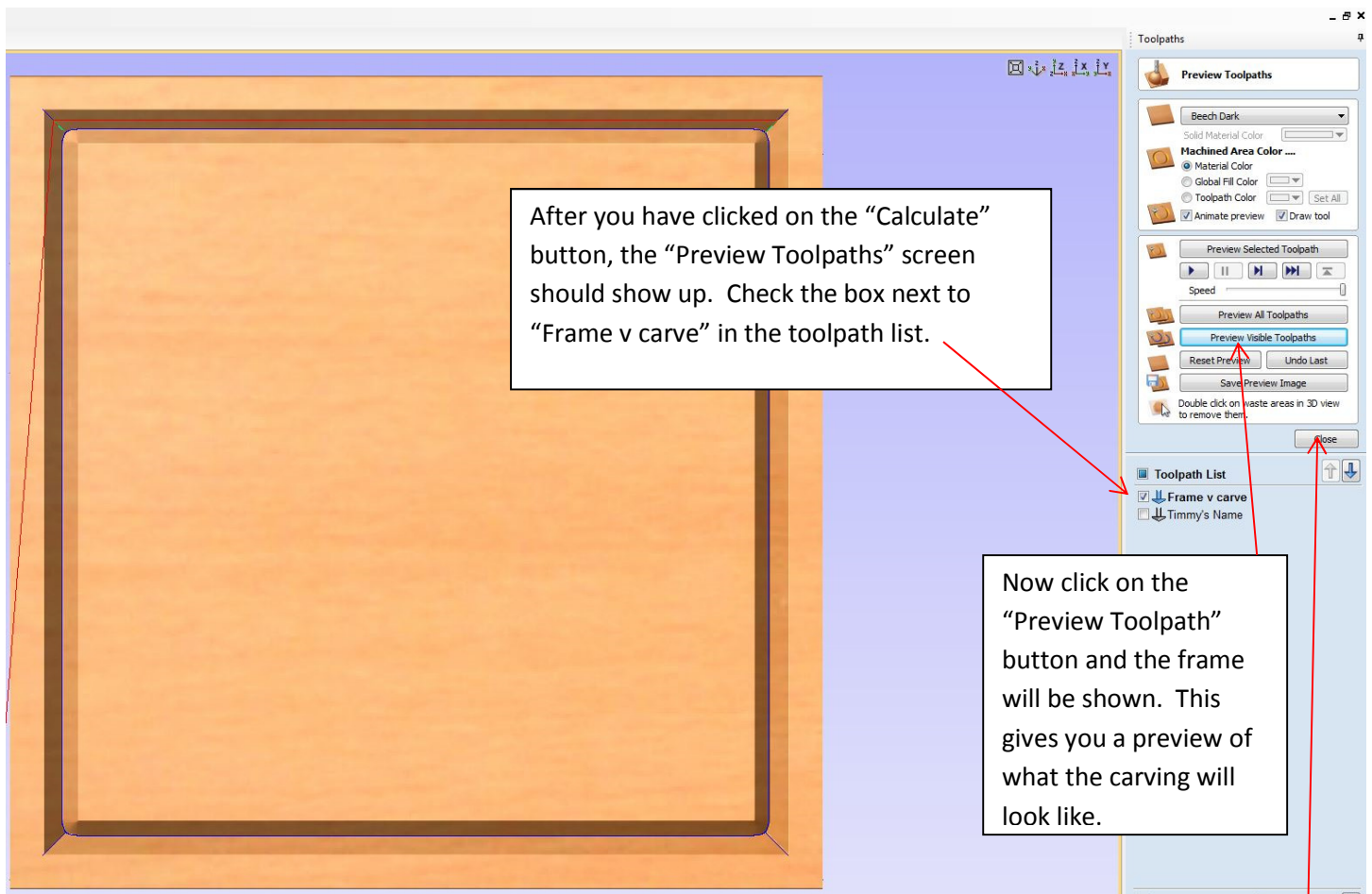
☐ **Project toolpath onto 3D model**

Vector Selection:

Name:

Once you closed the "Tool Database" window, check that the "V\_Curve / Engraving Toolpath" window has the same settings as you see here.

Give the file the name you see here and click on the "Calculate" button.

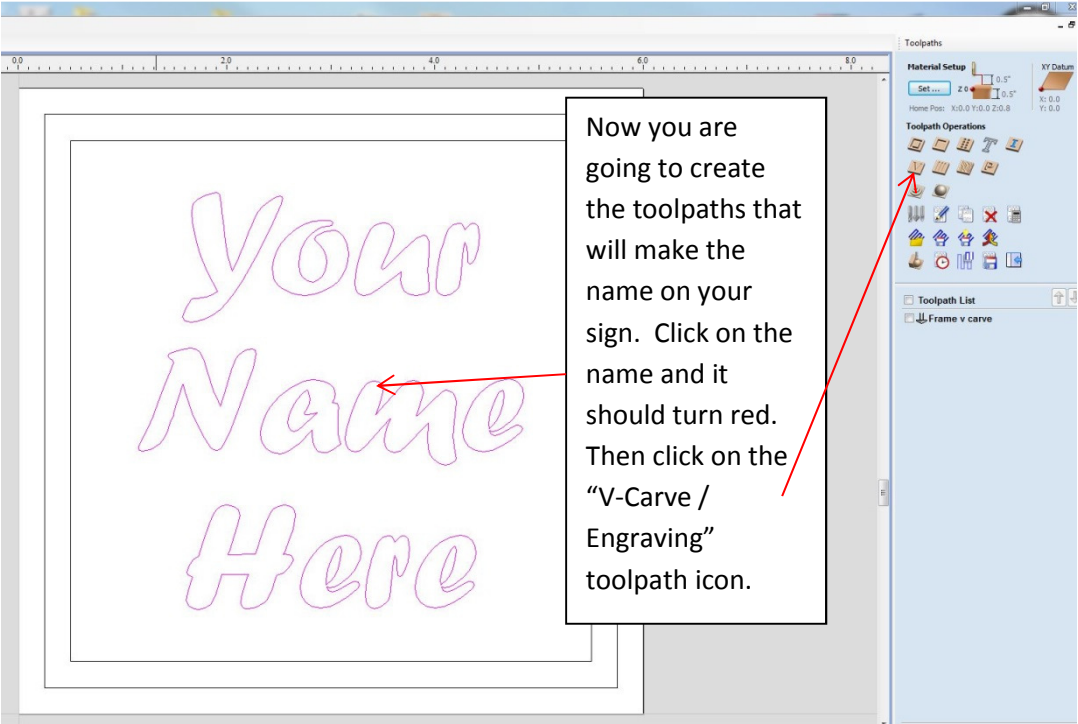
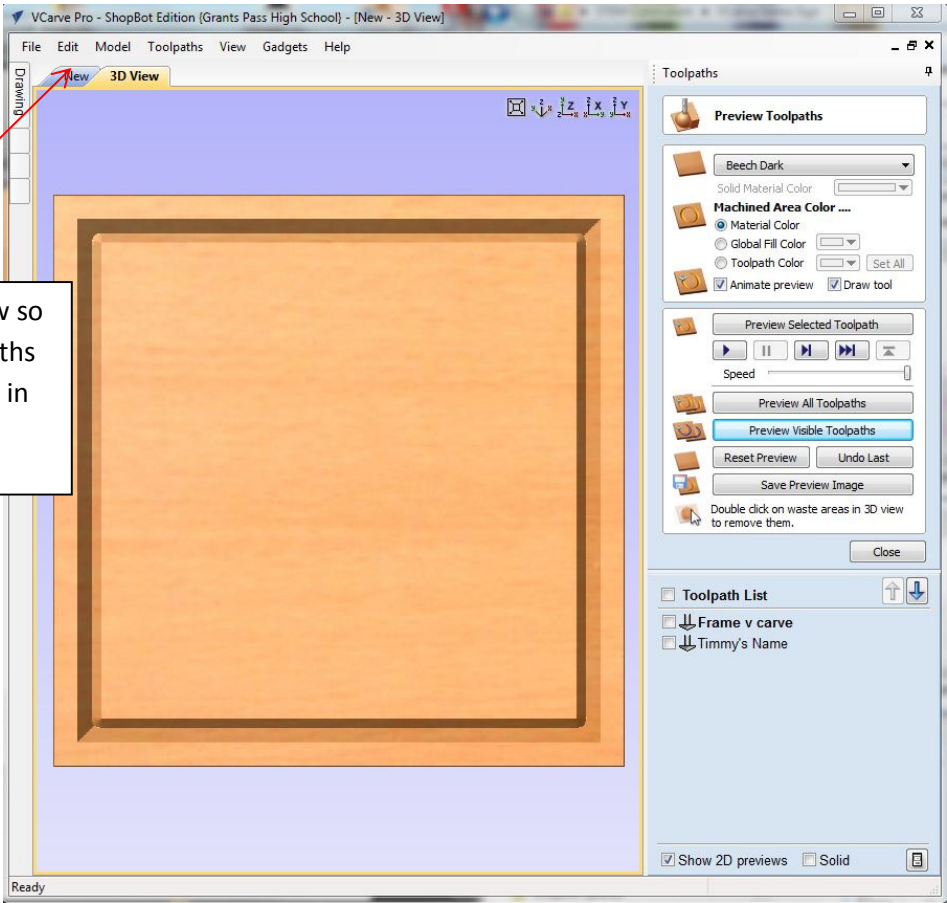


After you have clicked on the “Calculate” button, the “Preview Toolpaths” screen should show up. Check the box next to “Frame v carve” in the toolpath list.

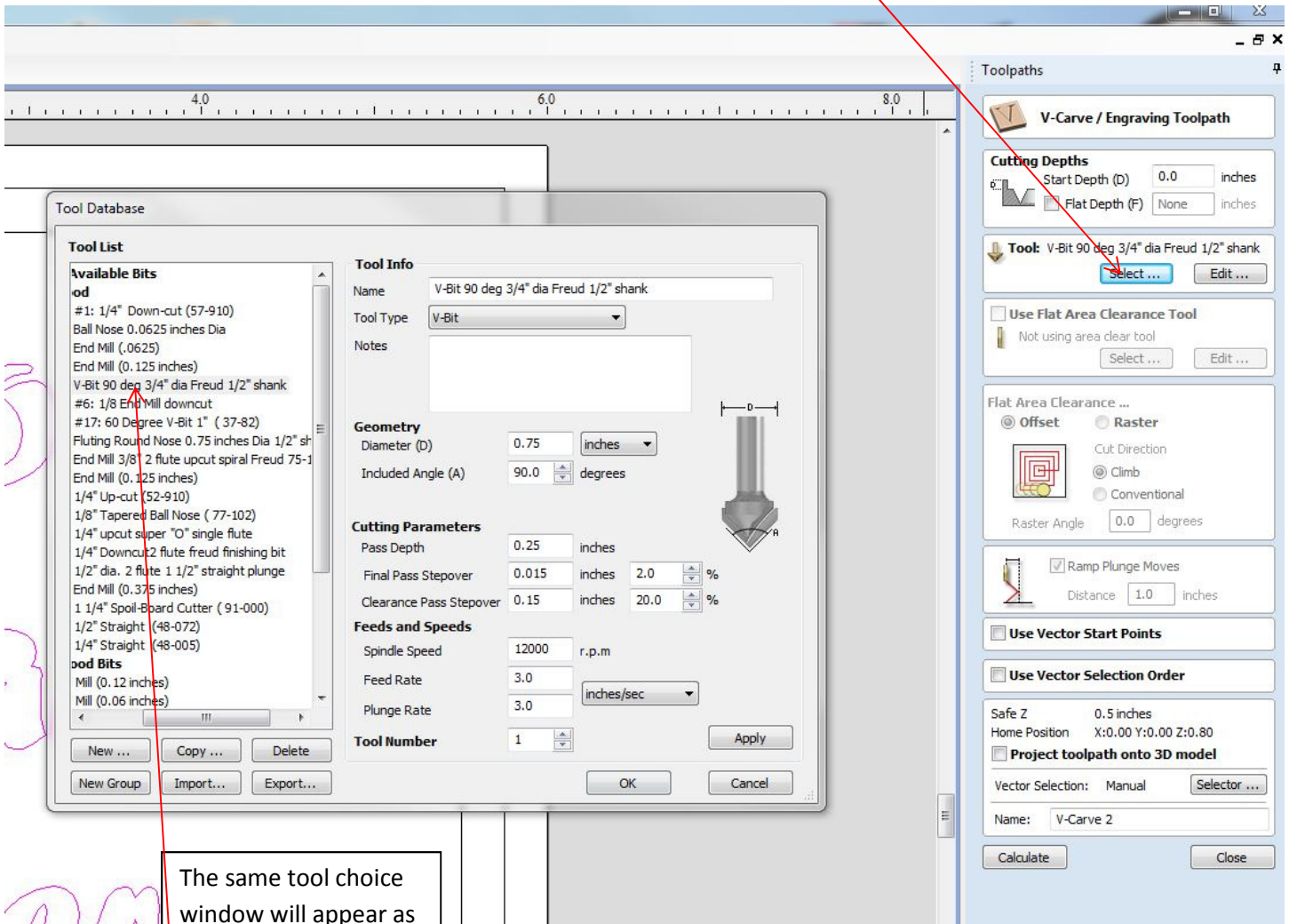
Now click on the “Preview Toolpath” button and the frame will be shown. This gives you a preview of what the carving will look like.

Now click on the “Close” button to exit this screen.

We need to get back to the drawing view so that we can begin to work on the toolpaths for the text. Just click on the “NEW” tab in the upper left corner of your screen.



We are going to follow the same steps as we did to make the frame border. Click on the "Select" button in the tool window.



The same tool choice window will appear as before. Choose the "90 deg v-bit again and check to make sure that all of the parameters are set to

**Tool Database**

**Tool List**

**Available Bits**

od

#1: 1/4" Down-cut (57-910)

Ball Nose 0.0625 inches Dia

End Mill (.0625)

End Mill (0.125 inches)

V-Bit 90 deg 3/4" dia Freud 1/2" shank

#6: 1/8 End Mill downcut

#17: 60 Degree V-Bit 1" ( 37-82)

Fluting Round Nose 0.75 inches Dia 1/2" sh

End Mill 3/8" 2 flute upcut spiral Freud 75-1

End Mill (0.125 inches)

1/4" Up-cut (52-910)

1/8" Tapered Ball Nose ( 77-102)

1/4" upcut super "O" single flute

1/4" Downcut 2 flute freud finishing bit

1/2" dia. 2 flute 1 1/2" straight plunge

End Mill (0.375 inches)

1 1/4" Spoil-Board Cutter ( 91-000)

1/2" Straight (48-072)

1/4" Straight (48-005)

**Good Bits**

Mill (0.12 inches)

Mill (0.06 inches)

**Tool Info**

Name V-Bit 90 deg 3/4" dia Freud 1/2" shank

Tool Type V-Bit

Notes

**Geometry**

Diameter (D) 0.75 inches

Included Angle (A) 90.0 degrees

**Cutting Parameters**

Pass Depth 0.25 inches

Final Pass Stepover 0.015 inches 2.0 %

Clearance Pass Stepover 0.15 inches 20.0 %


**Feeds and Speeds**

Spindle Speed 12000 r.p.m

Feed Rate 3.0 inches/sec

Plunge Rate 3.0

**Tool Number** 1



New ... Copy ... Delete

New Group Import... Export...

Apply

OK Cancel

Make sure that all of your settings match what you see here.

Once you have checked all of the settings, click on "Apply" then click on "Ok".



Toolpaths

**V-Carve / Engraving Toolpath**

**Cutting Depths**

Start Depth (D)  inches

☐ Flat Depth (F)  inches



**Tool:** V-Bit 90 deg 3/4" dia Freud 1/2" shank

☐ **Use Flat Area Clearance Tool**

Not using area clear tool

**Flat Area Clearance ...**


☒ **Offset** ☐ **Raster**

Cut Direction

☒ Climb ☐ Conventional

Raster Angle  degrees

 ☒ Ramp Plunge Moves

Distance  inches

☐ **Use Vector Start Points**

☐ **Use Vector Selection Order**

Safe Z  inches

Home Position X:0.00 Y:0.00 Z:0.80

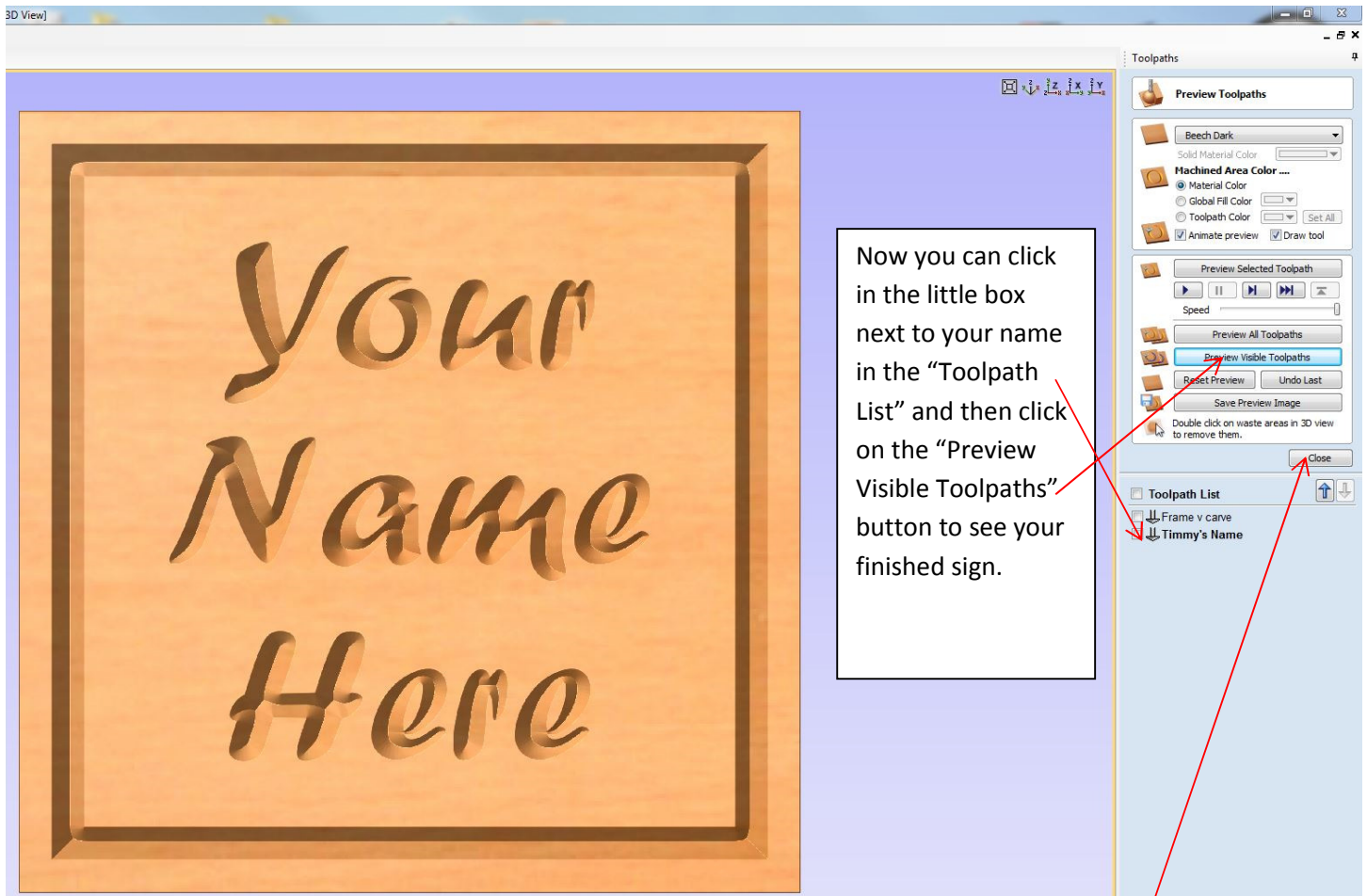
☐ **Project toolpath onto 3D model**

Vector Selection: Manual

Name:

Everything should look the same as it does here. The only thing you need to do is to give the toolpath a name. Give it your own name so we can find it later when we go out into the shop to cut your sign.

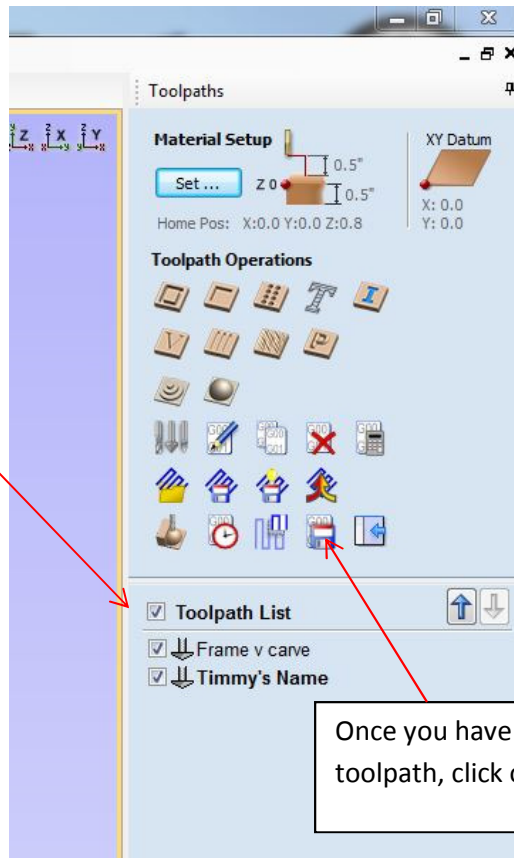
Now you can go ahead and click on the "Calculate" button then click on the "Close" button.



If everything looks good, click on the "Close" button to exit out of this window.

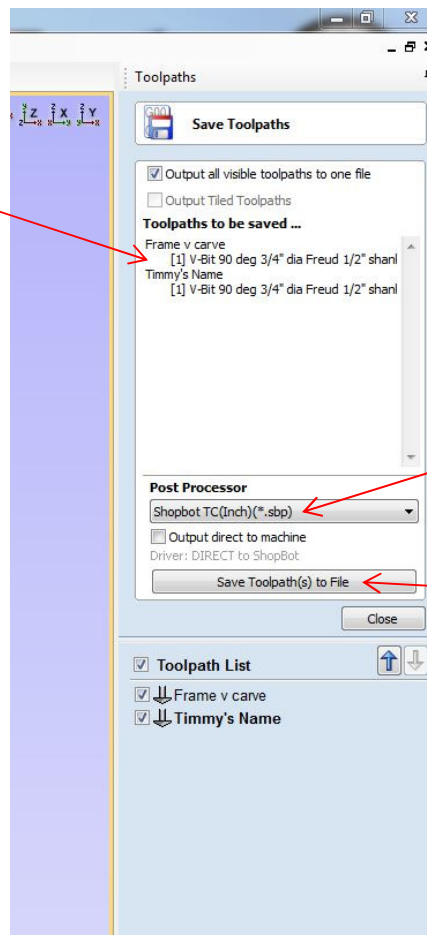


We need to save your toolpaths next so make sure the boxes are checked in the “Toolpath List”.

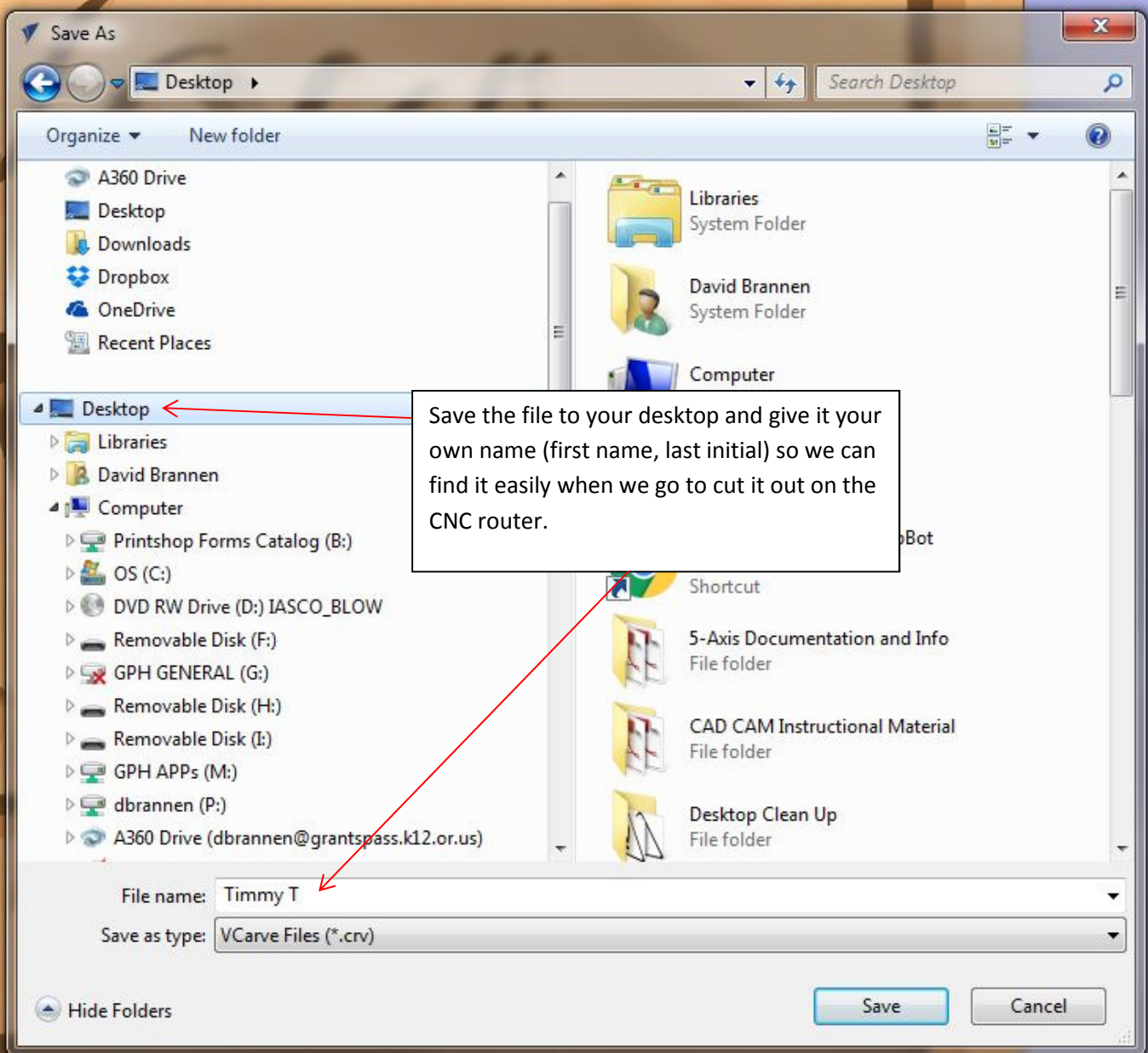


Once you have checked the boxes for each toolpath, click on the “Save Toolpath” icon.

You should see the two toolpaths that you are saving in the window. The Frame VCarve and the Name toolpath.



Make sure that the “Post Processor” information matches what you see here and then click on the “Save Toolpaths to File” button.



**Toolpaths**

**Material Setup**

Set ... Z 0 0.5" 0.75"

Home Pos: X:0.0 Y:0.0 Z:0.8

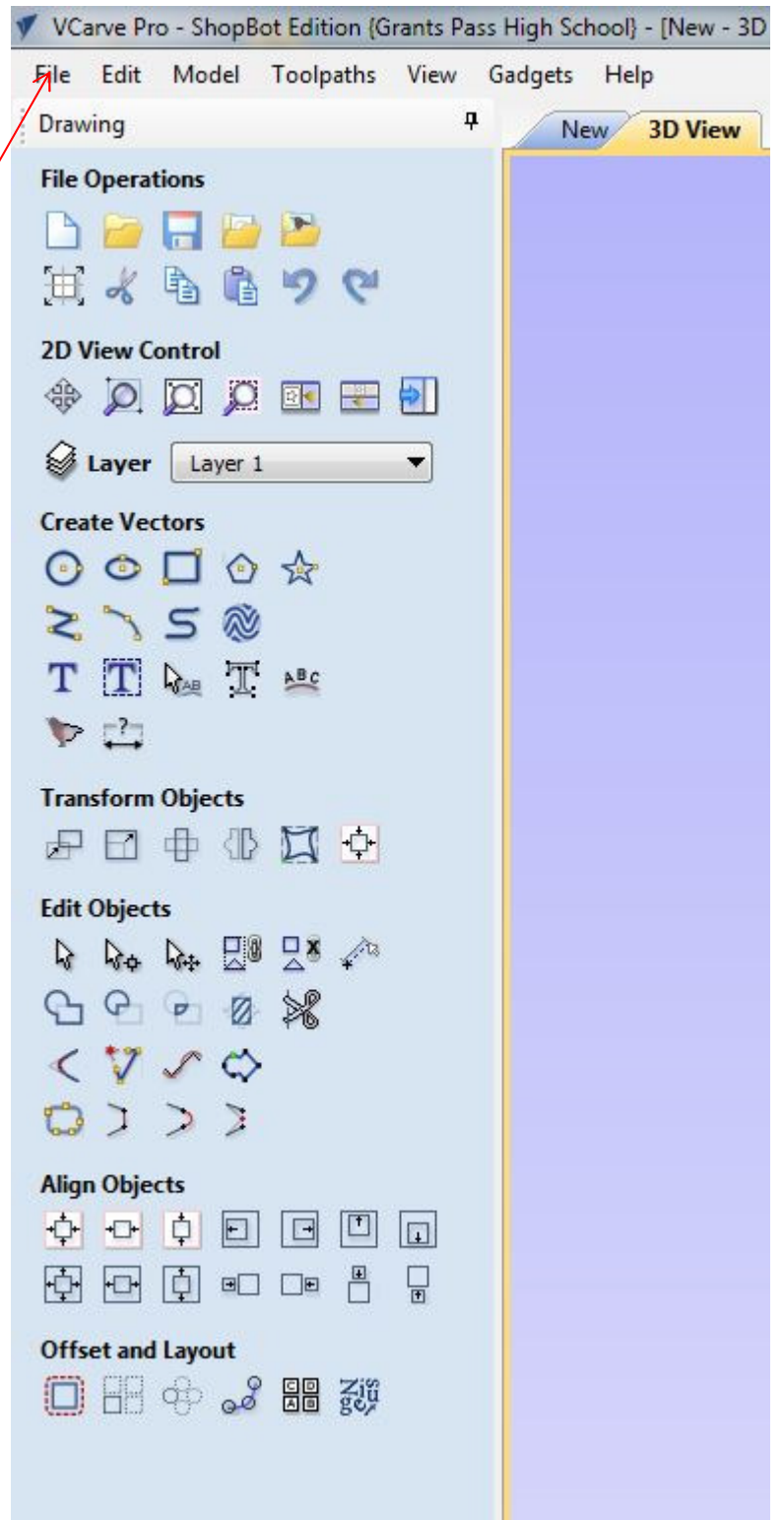
**Toolpath Operations**

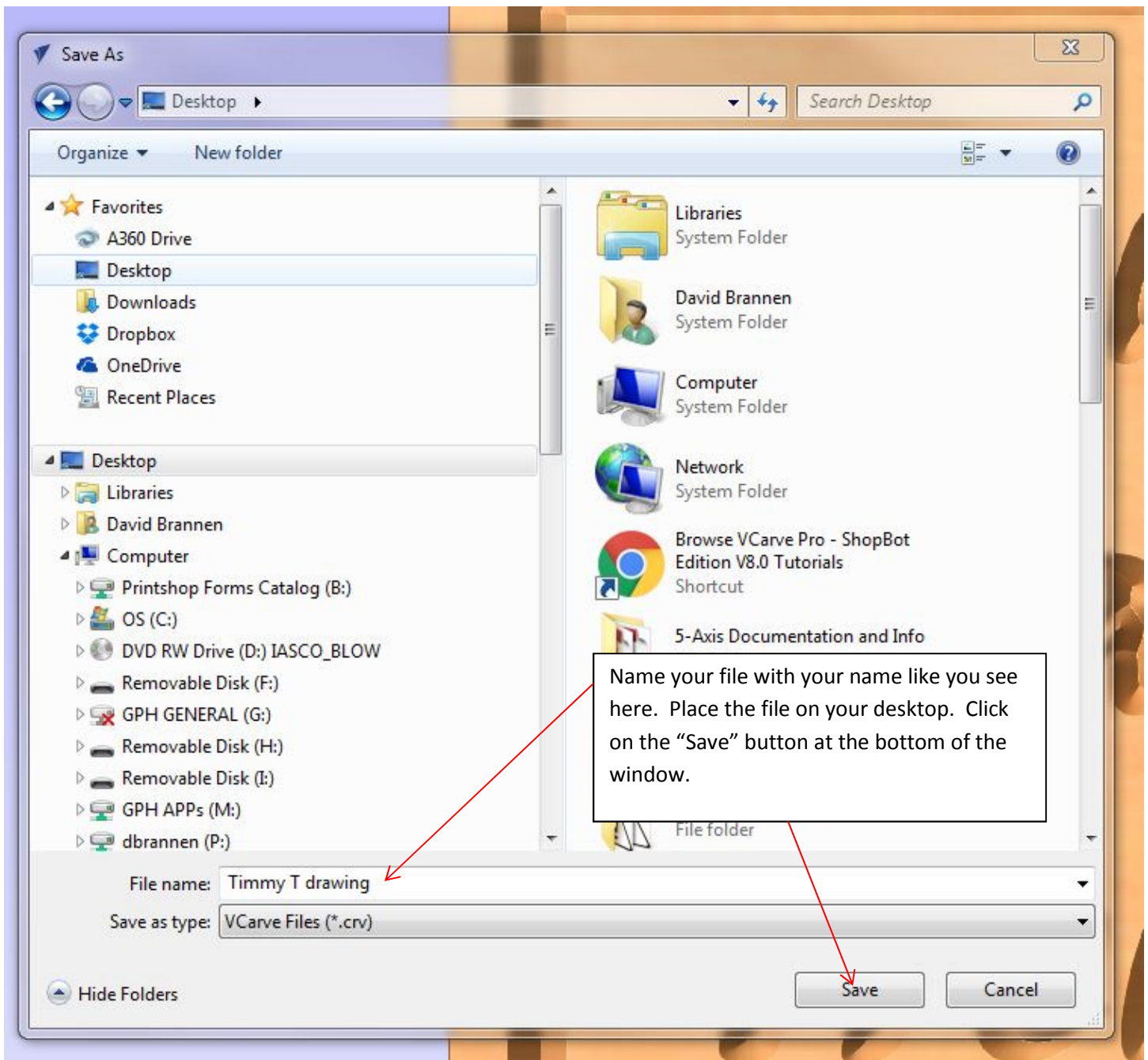
**Toolpath List**

- V-Carve 1
- Timmy's name

We have saved the toolpaths but your drawing needs to be saved too. Click on the "Switch to Drawing Tab" icon to take you back to the drawing panel.

Click on "File" then from the drop down menu click on "Save As".





That's it! You are ready to go out to the shop and create your custom sign on the CNC router!