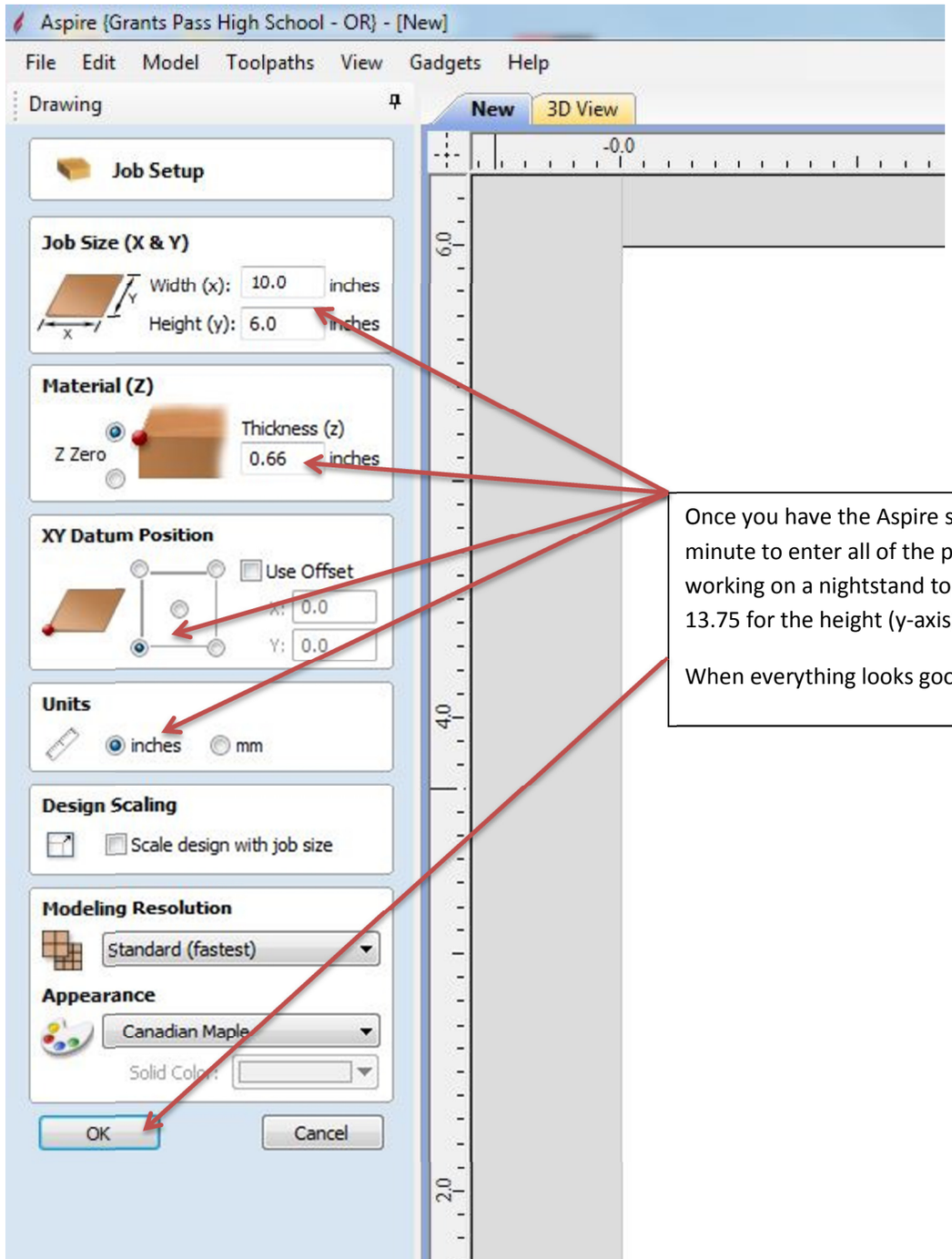
Welcome to CNC and laser design tutorial. The first item on the agenda is to open the CAD software you will be using today. One of the most user friendly (easiest) software packages we have available to us is called “Aspire”. Quite simple to get up and running and it’s a lot of fun to use.

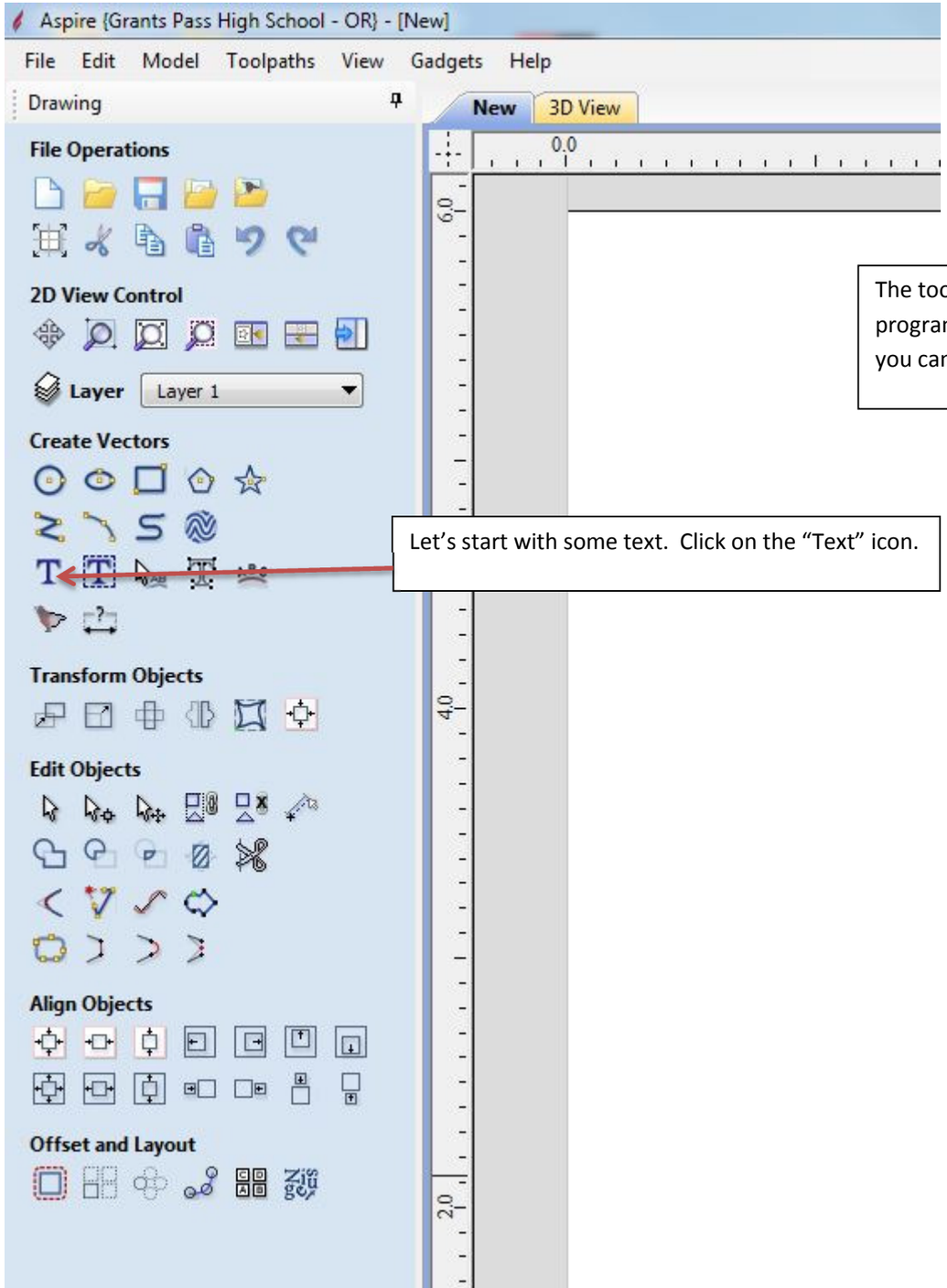
So…the first thing you need to do is find this program, Aspire. It may not be an icon on your desktop so you may need to look for it in the programs list on your PC.

Just a quick bit of info for you up front. We are going to be designing a very simple project that you will take over to the shop and machine. Although it is simple, there are opportunities for you to make alterations in order to create something more complex than is described in this tutorial. We encourage you to ask questions and explore the software to your hearts content.



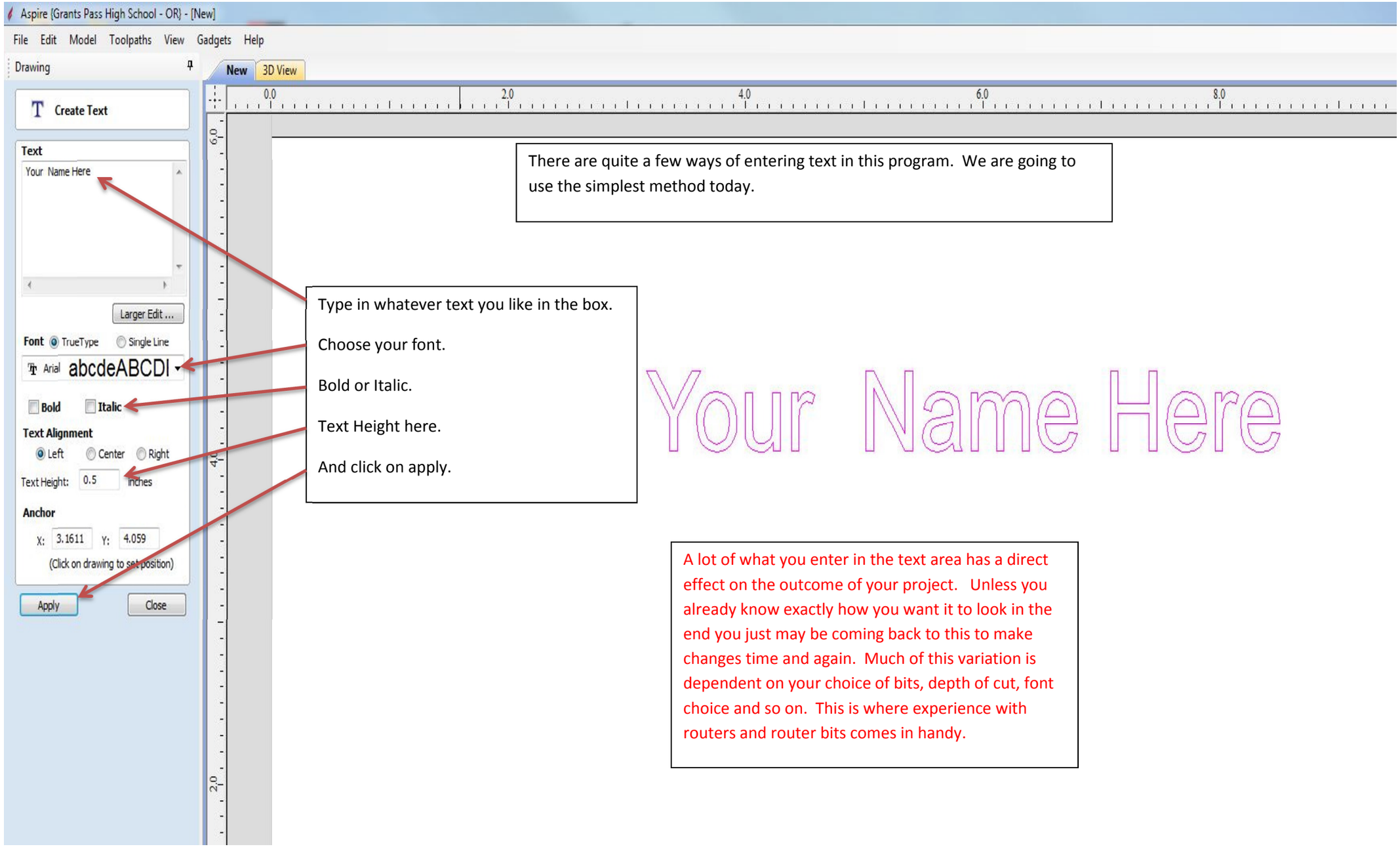
Once you have the Aspire software open, we need to set the job up. Take a minute to enter all of the parameters you will be using. For example, if you are working on a nightstand top you would enter 16.5 for the width (x-axis) and 13.75 for the height (y-axis) and 0.75 for the Thickness.

When everything looks good, click “OK”



The tool bar seen on the left is where you do all of your design work in this program. We are going to do a simple name sign although if you would like to, you can get a bit more creative than that. Just ask for help if you need it.

Let's start with some text. Click on the "Text" icon.



There are quite a few ways of entering text in this program. We are going to use the simplest method today.

Type in whatever text you like in the box.

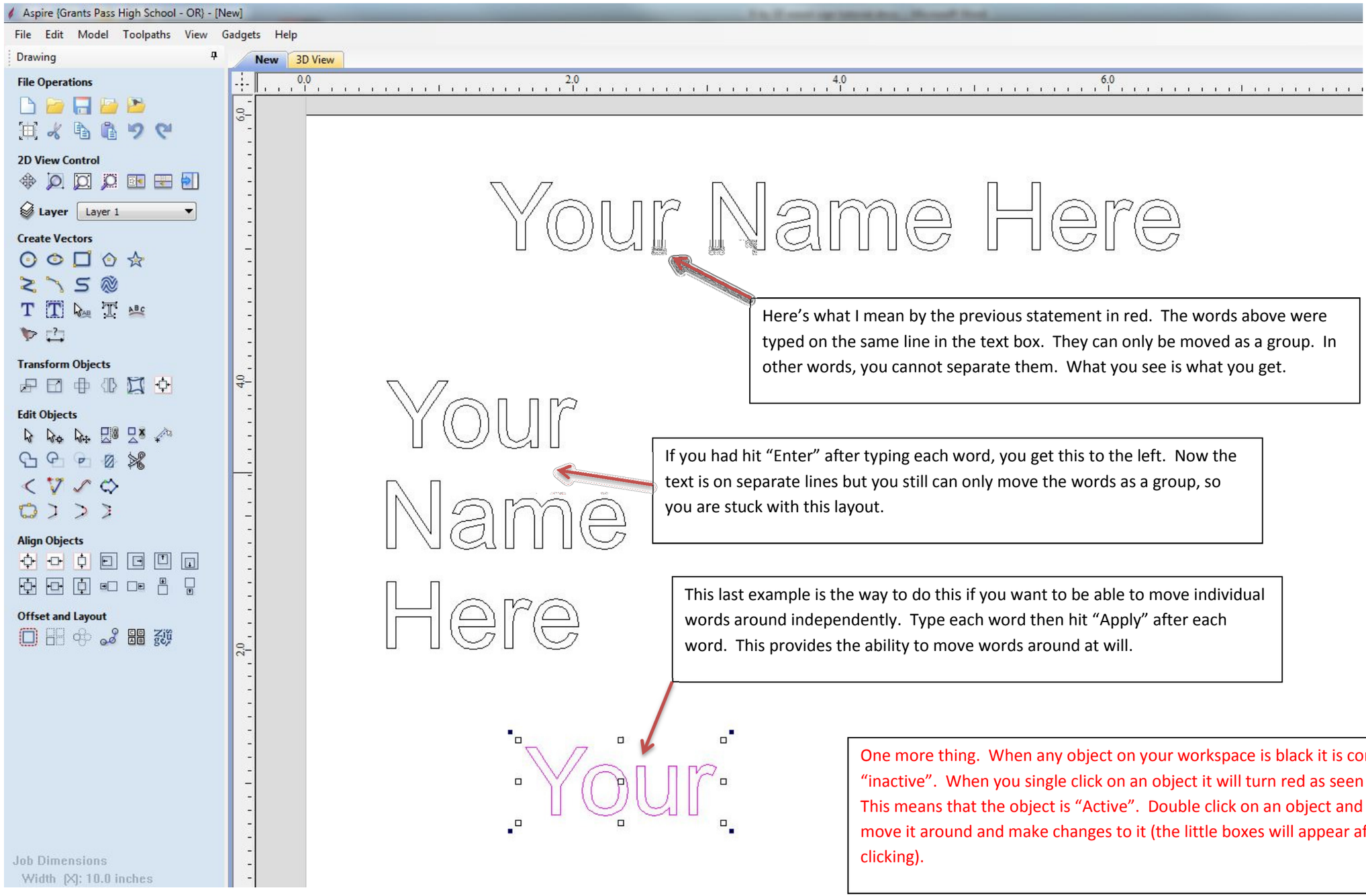
Choose your font.

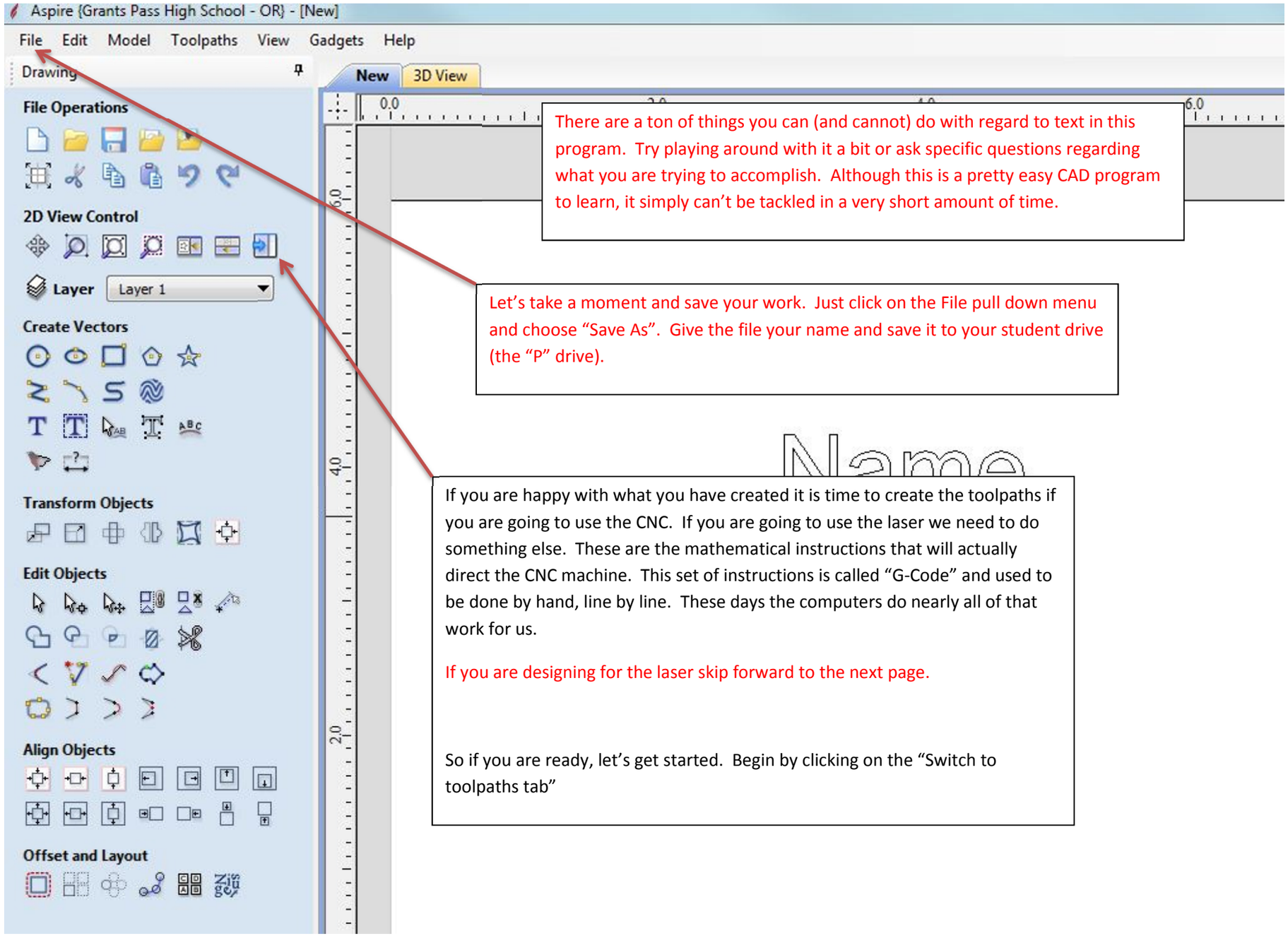
Bold or Italic.

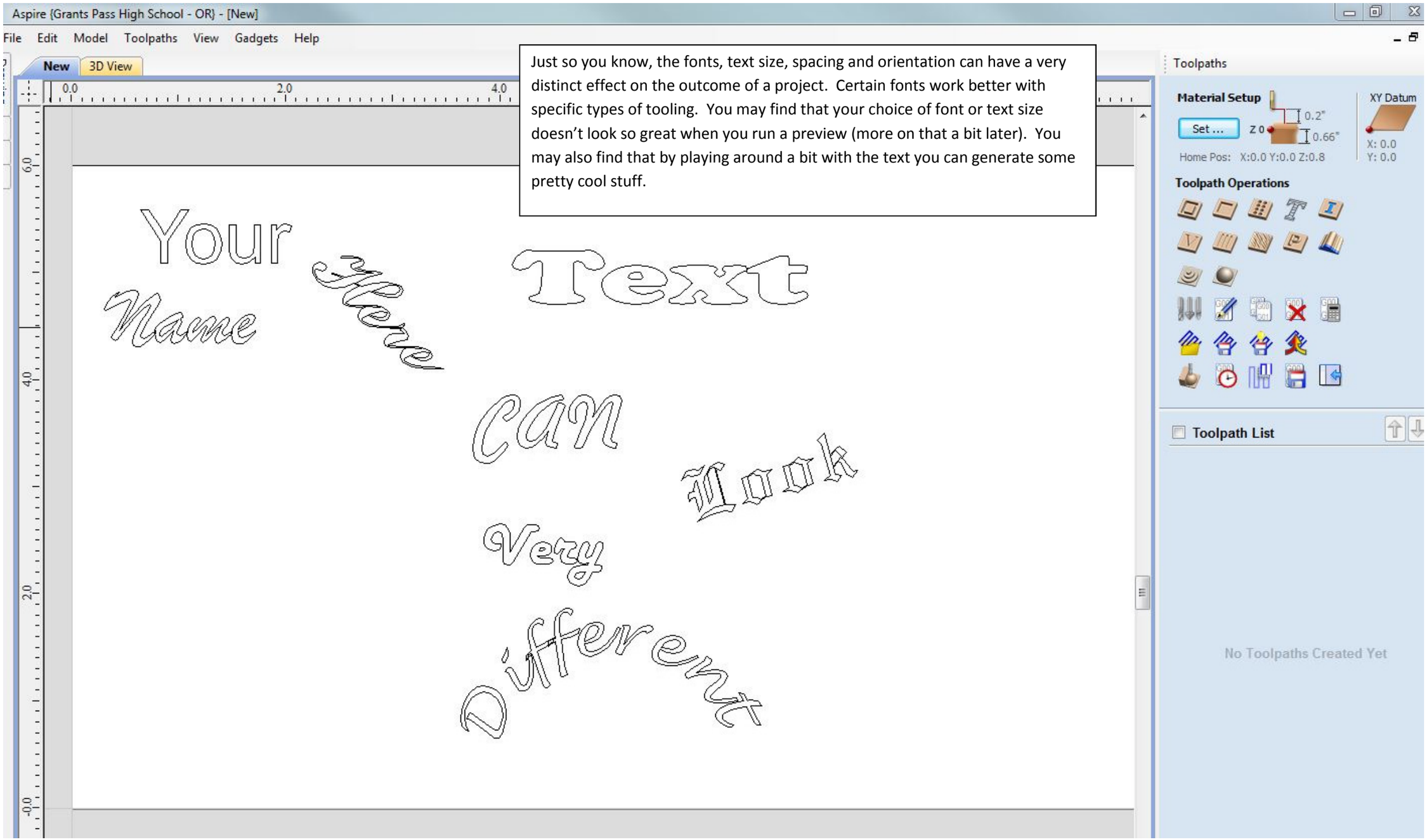
Text Height here.

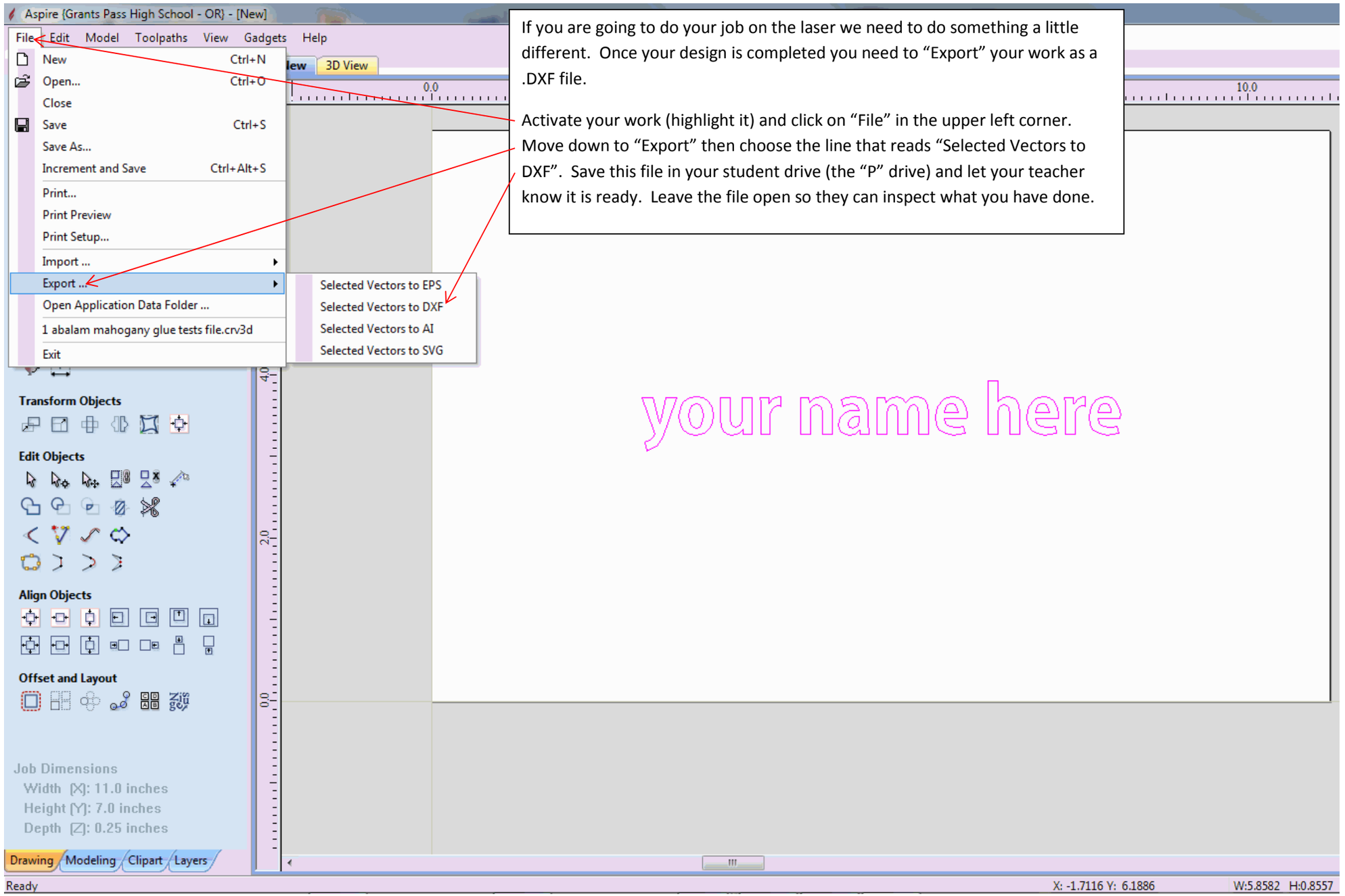
And click on apply.

A lot of what you enter in the text area has a direct effect on the outcome of your project. Unless you already know exactly how you want it to look in the end you just may be coming back to this to make changes time and again. Much of this variation is dependent on your choice of bits, depth of cut, font choice and so on. This is where experience with routers and router bits comes in handy.


Aspire {Grants Pass High School - OR} - [New]
File Edit Model Toolpaths View Gadgets Help
Drawing
New 3D View
File Operations
2D View Control
Layer Layer 1
Create Vectors
Transform Objects
Edit Objects
Align Objects
Offset and Layout
Job Dimensions
Width [X]: 10.0 inches
Your Name Here
Here's what I mean by the previous statement in red. The words above were typed on the same line in the text box. They can only be moved as a group. In other words, you cannot separate them. What you see is what you get.
Your
Name
Here
If you had hit "Enter" after typing each word, you get this to the left. Now the text is on separate lines but you still can only move the words as a group, so you are stuck with this layout.
This last example is the way to do this if you want to be able to move individual words around independently. Type each word then hit "Apply" after each word. This provides the ability to move words around at will.
Your
One more thing. When any object on your workspace is black it is considered "inactive". When you single click on an object it will turn red as seen to the left. This means that the object is "Active". Double click on an object and you can move it around and make changes to it (the little boxes will appear after double clicking).


Aspire {Grants Pass High School - OR} - [New]
File Edit Model Toolpaths View Gadgets Help
Drawing
New
3D View
File Operations
2D View Control
Layer
Layer 1
Create Vectors
Transform Objects
Edit Objects
Align Objects
Offset and Layout
0.0
6.0
6.0
4.0
2.0
There are a ton of things you can (and cannot) do with regard to text in this program. Try playing around with it a bit or ask specific questions regarding what you are trying to accomplish. Although this is a pretty easy CAD program to learn, it simply can't be tackled in a very short amount of time.
Let's take a moment and save your work. Just click on the File pull down menu and choose "Save As". Give the file your name and save it to your student drive (the "P" drive).
Name
If you are happy with what you have created it is time to create the toolpaths if you are going to use the CNC. If you are going to use the laser we need to do something else. These are the mathematical instructions that will actually direct the CNC machine. This set of instructions is called "G-Code" and used to be done by hand, line by line. These days the computers do nearly all of that work for us.
If you are designing for the laser skip forward to the next page.
So if you are ready, let's get started. Begin by clicking on the "Switch to toolpaths tab"


Aspire {Grants Pass High School - OR} - [New]
File Edit Model Toolpaths View Gadgets Help
New 3D View
Just so you know, the fonts, text size, spacing and orientation can have a very distinct effect on the outcome of a project. Certain fonts work better with specific types of tooling. You may find that your choice of font or text size doesn't look so great when you run a preview (more on that a bit later). You may also find that by playing around a bit with the text you can generate some pretty cool stuff.
Your Name Here
Text
can
Look
Very
Different
Toolpaths
Material Setup
Set ...
Z 0
0.2"
0.66"
XY Datum
X: 0.0
Y: 0.0
Home Pos: X:0.0 Y:0.0 Z:0.8
Toolpath Operations
Toolpath List
No Toolpaths Created Yet



If you are going to do your job on the laser we need to do something a little different. Once your design is completed you need to "Export" your work as a .DXF file.

Activate your work (highlight it) and click on "File" in the upper left corner. Move down to "Export" then choose the line that reads "Selected Vectors to DXF". Save this file in your student drive (the "P" drive) and let your teacher know it is ready. Leave the file open so they can inspect what you have done.

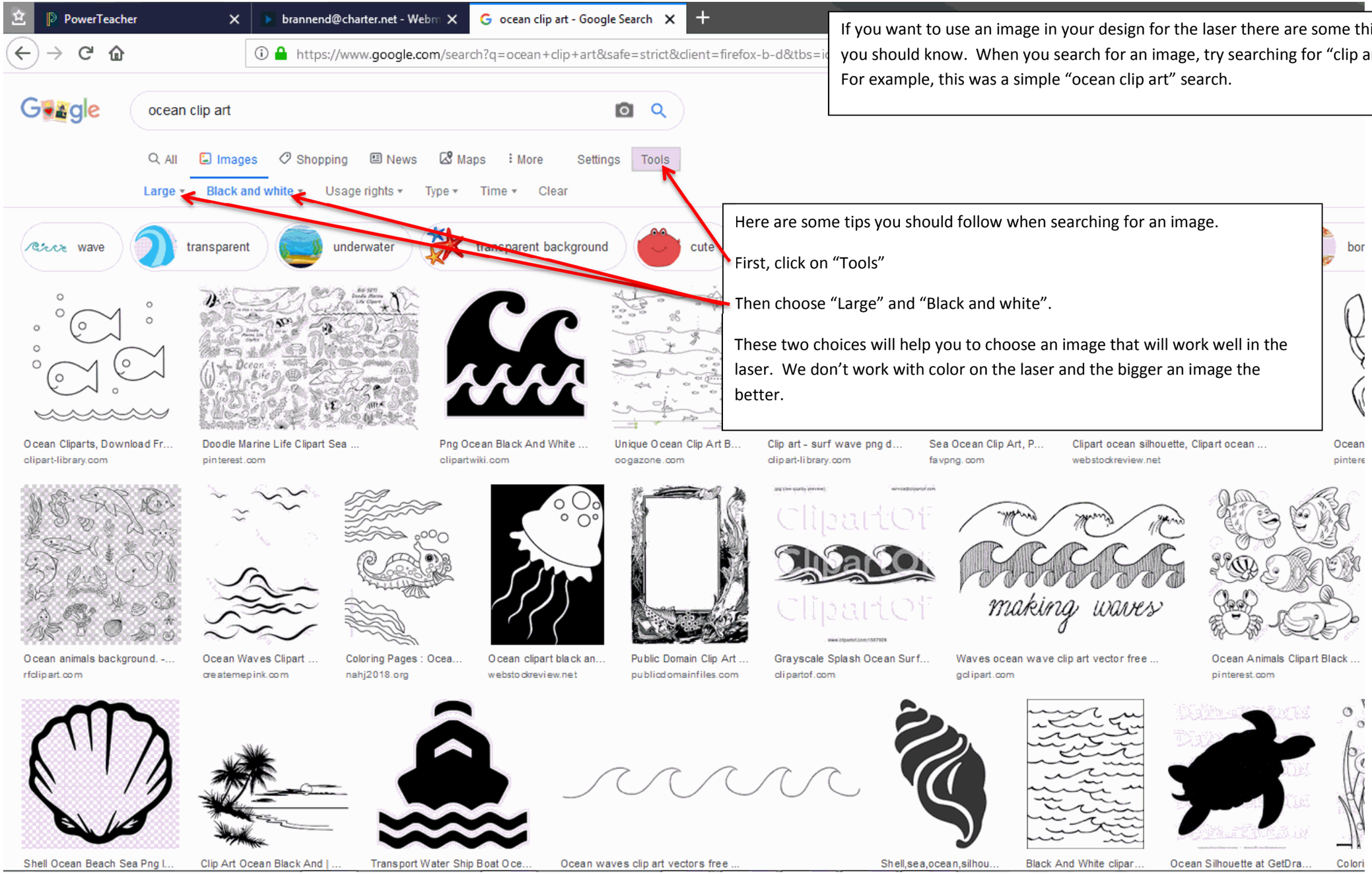
If you want to use an image in your design for the laser there are some things you should know. When you search for an image, try searching for “clip art”. For example, this was a simple “ocean clip art” search.

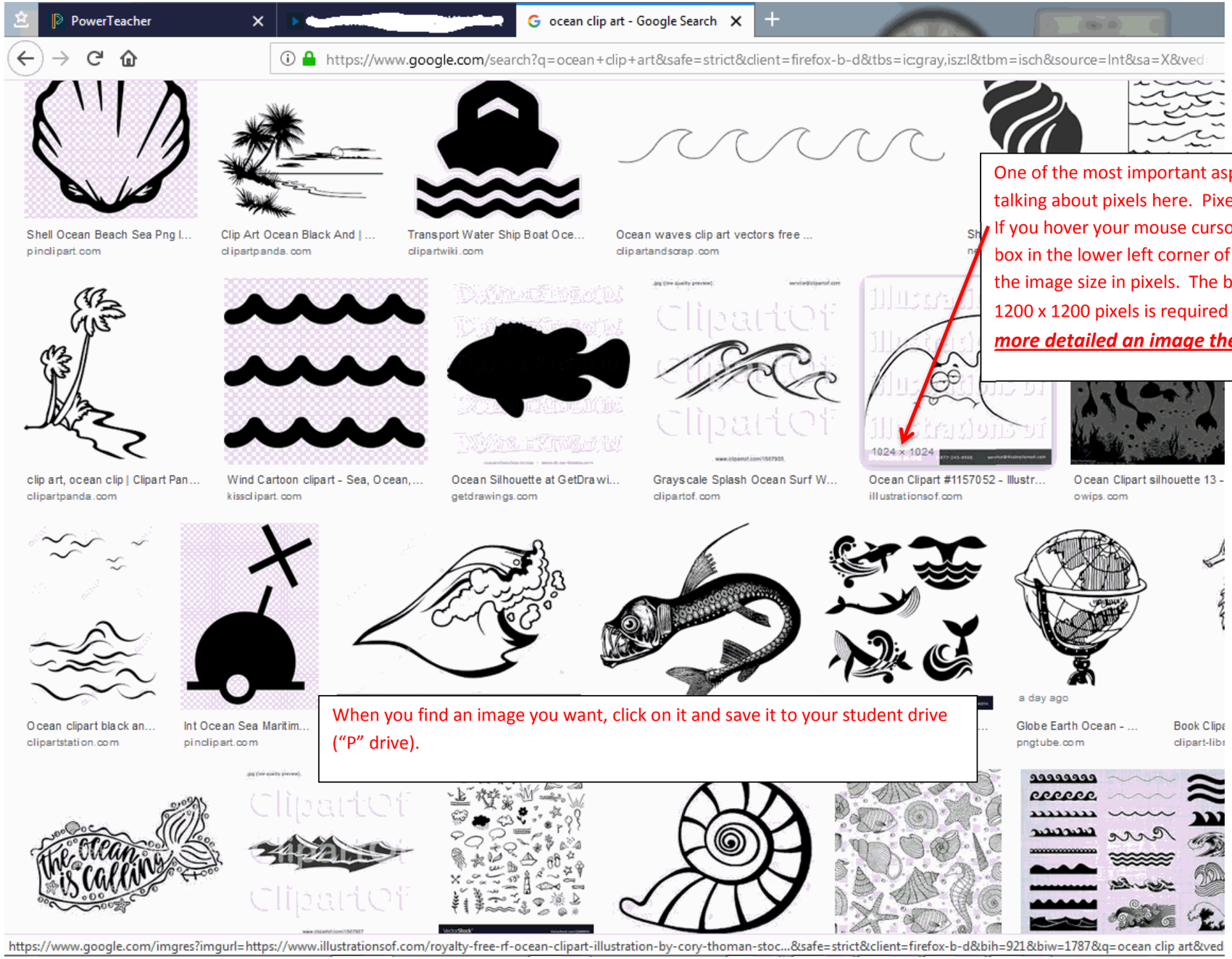
Here are some tips you should follow when searching for an image.

First, click on “Tools”

Then choose “Large” and “Black and white”.

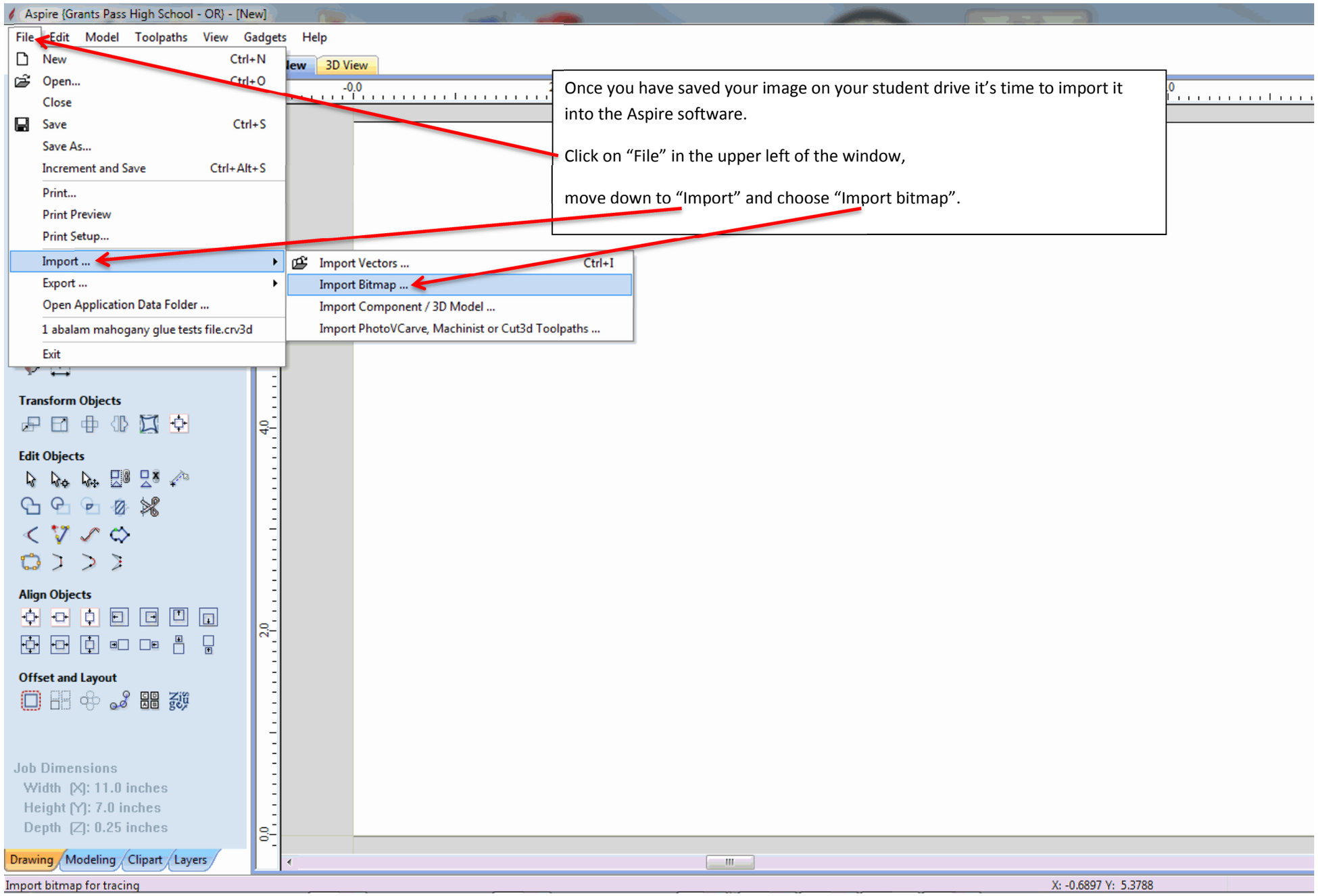
These two choices will help you to choose an image that will work well in the laser. We don’t work with color on the laser and the bigger an image the better.





One of the most important aspects of the image you choose is its size. We are talking about pixels here. Pixels are tiny little squares that make up an image. If you hover your mouse cursor over one of the images you should see a small box in the lower left corner of the image with a couple of numbers in it. That is the image size in pixels. The bigger the better is the rule here. A minimum of 1200 x 1200 pixels is required for a very basic image (not a lot of detail). ***The more detailed an image the larger it needs to be.***

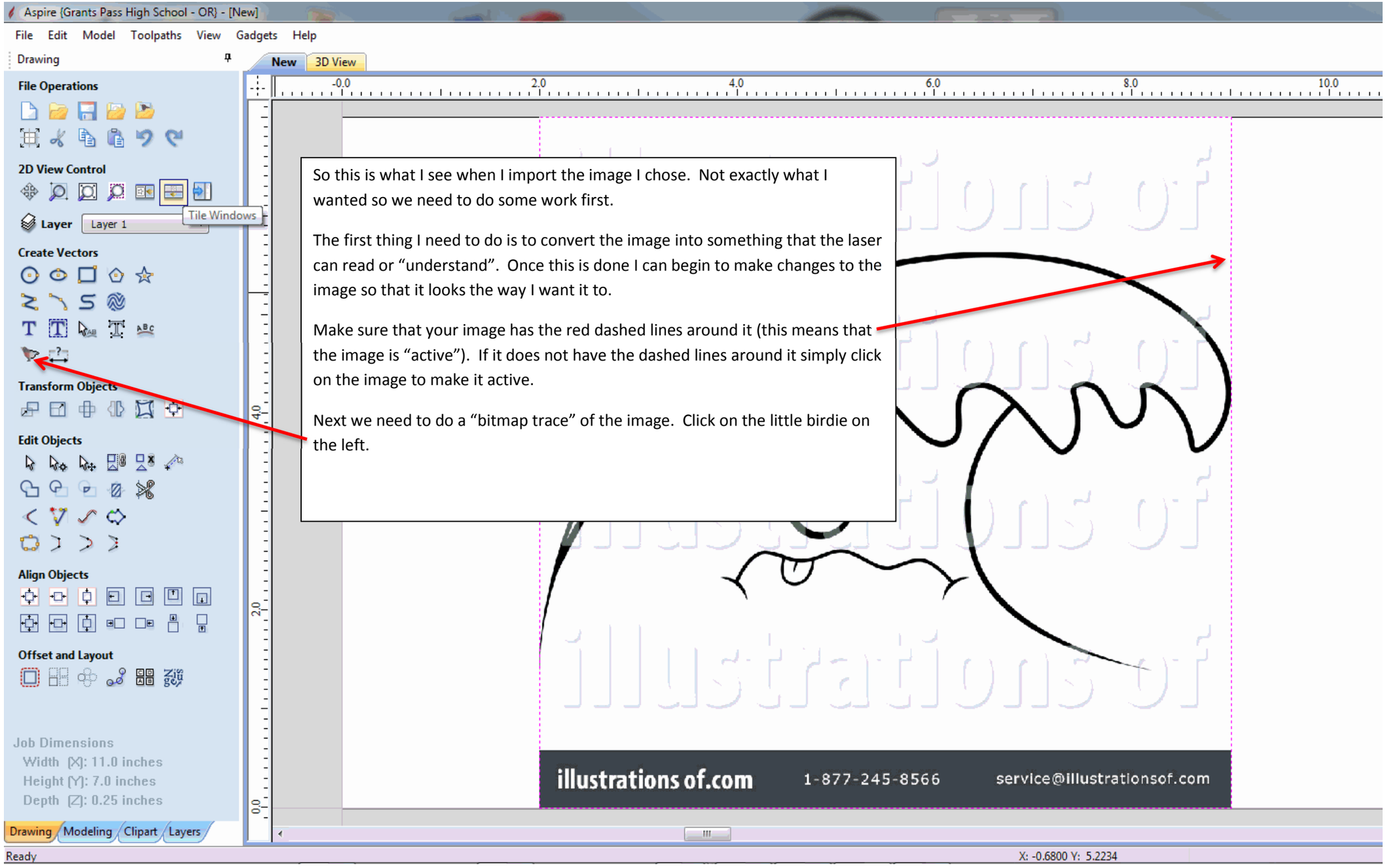
When you find an image you want, click on it and save it to your student drive ("P" drive).



Once you have saved your image on your student drive it's time to import it into the Aspire software.

Click on "File" in the upper left of the window,

move down to "Import" and choose "Import bitmap".

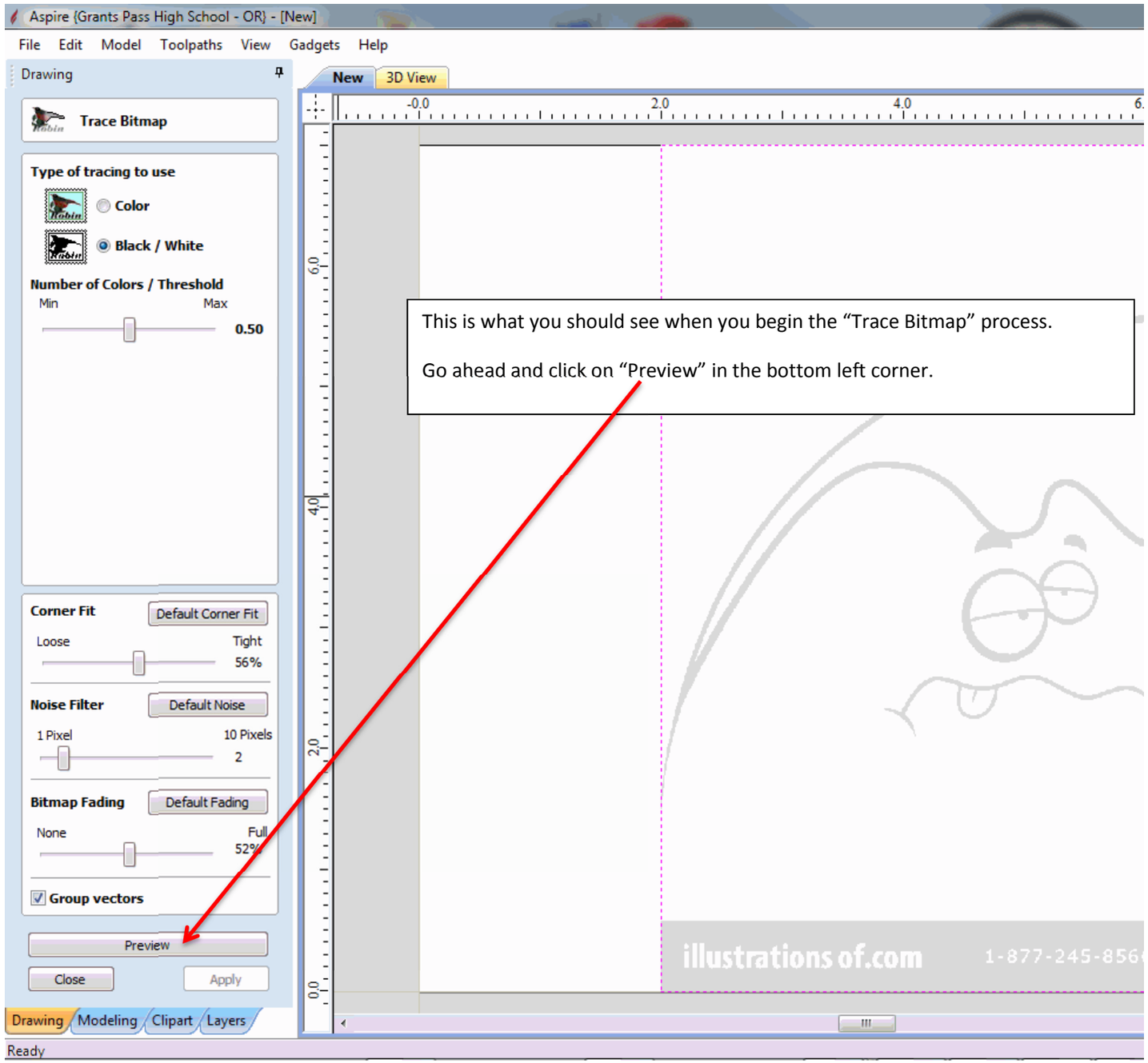


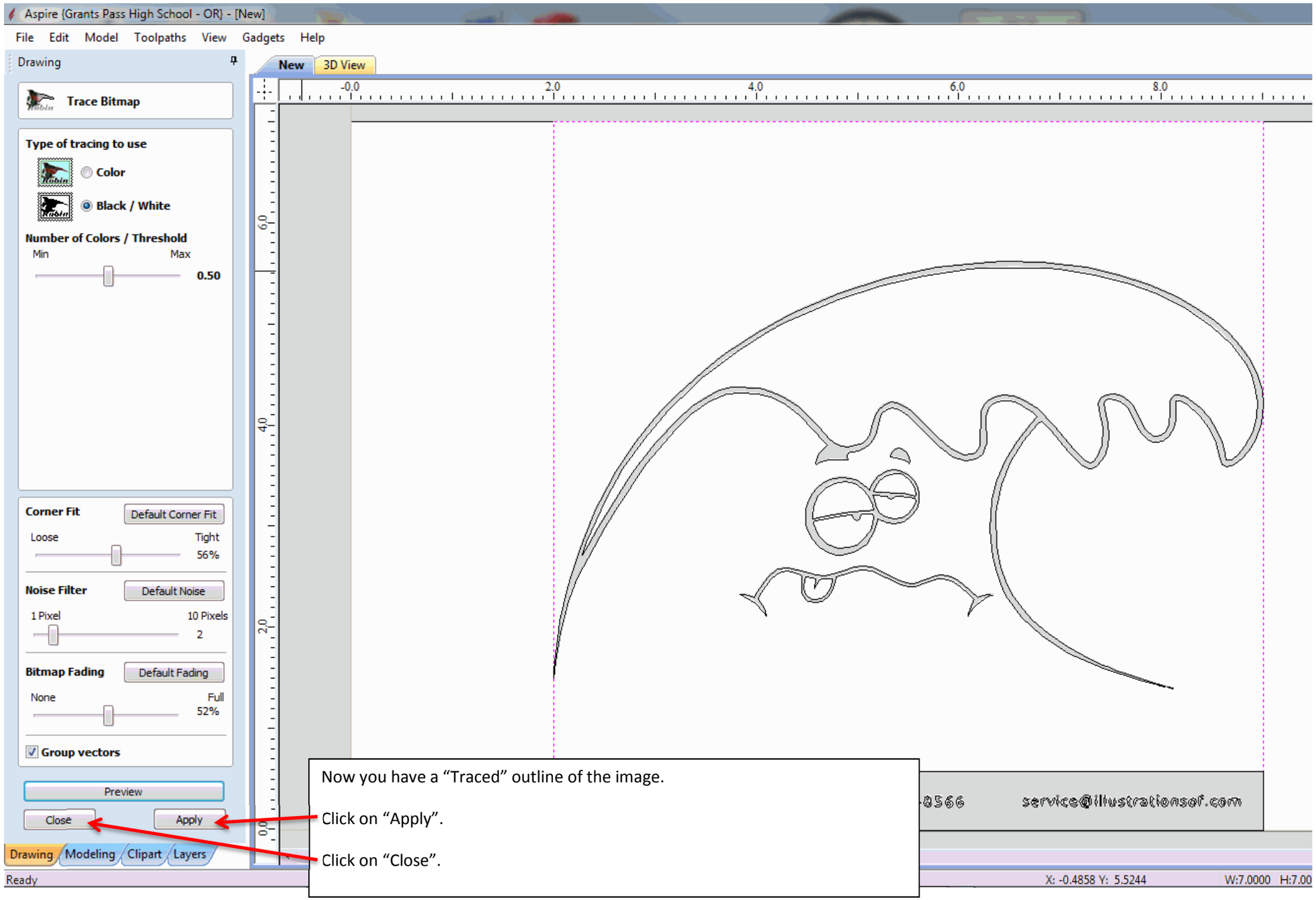
So this is what I see when I import the image I chose. Not exactly what I wanted so we need to do some work first.

The first thing I need to do is to convert the image into something that the laser can read or "understand". Once this is done I can begin to make changes to the image so that it looks the way I want it to.

Make sure that your image has the red dashed lines around it (this means that the image is "active"). If it does not have the dashed lines around it simply click on the image to make it active.

Next we need to do a "bitmap trace" of the image. Click on the little birdie on the left.

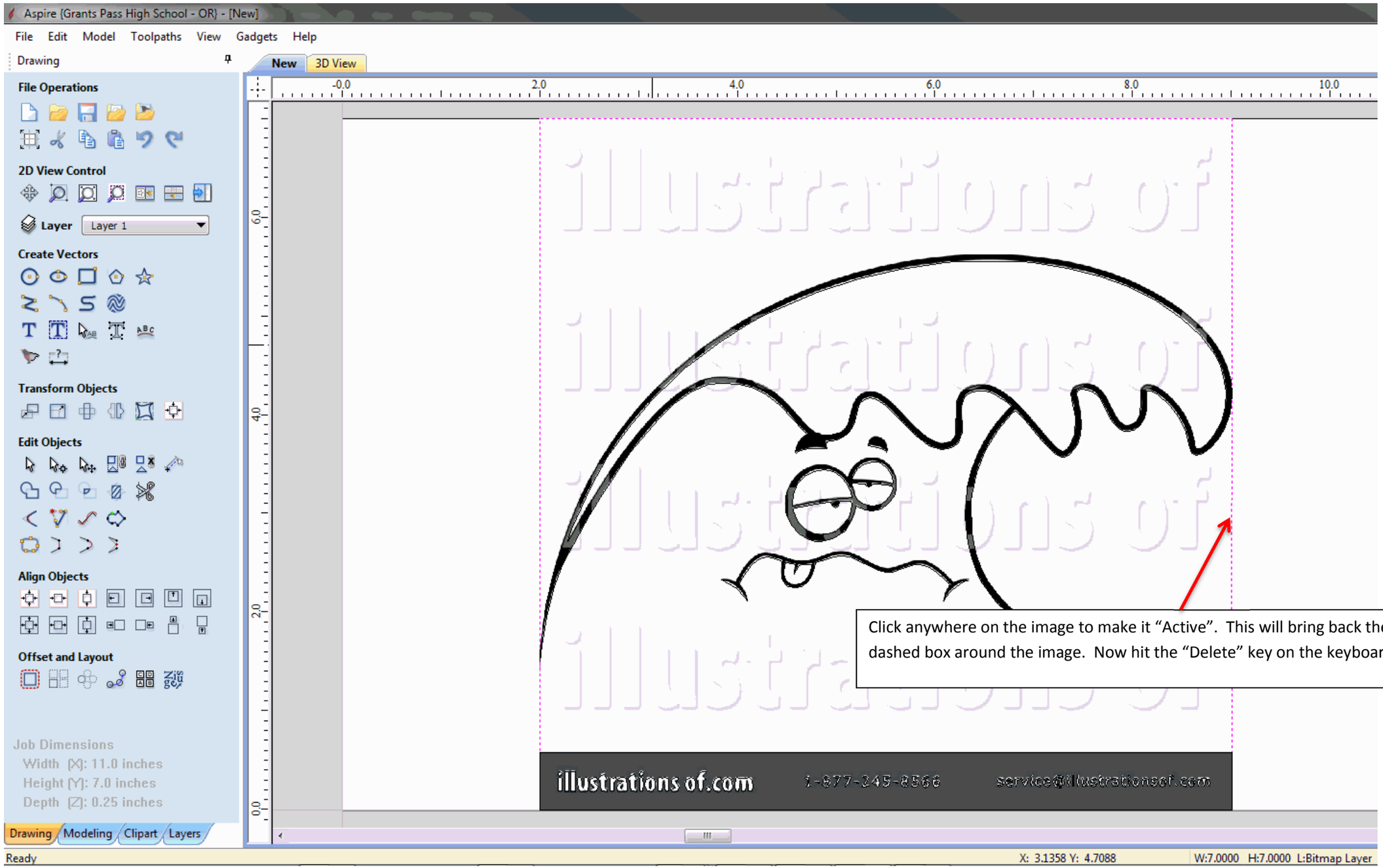

Aspire {Grants Pass High School - OR} - [New]
File Edit Model Toolpaths View Gadgets Help
Drawing
Trace Bitmap
Type of tracing to use
Color
Black / White
Number of Colors / Threshold
Min Max
0.50
Corner Fit
Default Corner Fit
Loose Tight
56%
Noise Filter
Default Noise
1 Pixel 10 Pixels
2
Bitmap Fading
Default Fading
None Full
52%
Group vectors
Preview
Close
Apply
Drawing Modeling Clipart Layers
New
3D View
This is what you should see when you begin the “Trace Bitmap” process.
Go ahead and click on “Preview” in the bottom left corner.
illustrations of.com
1-877-245-856
Ready

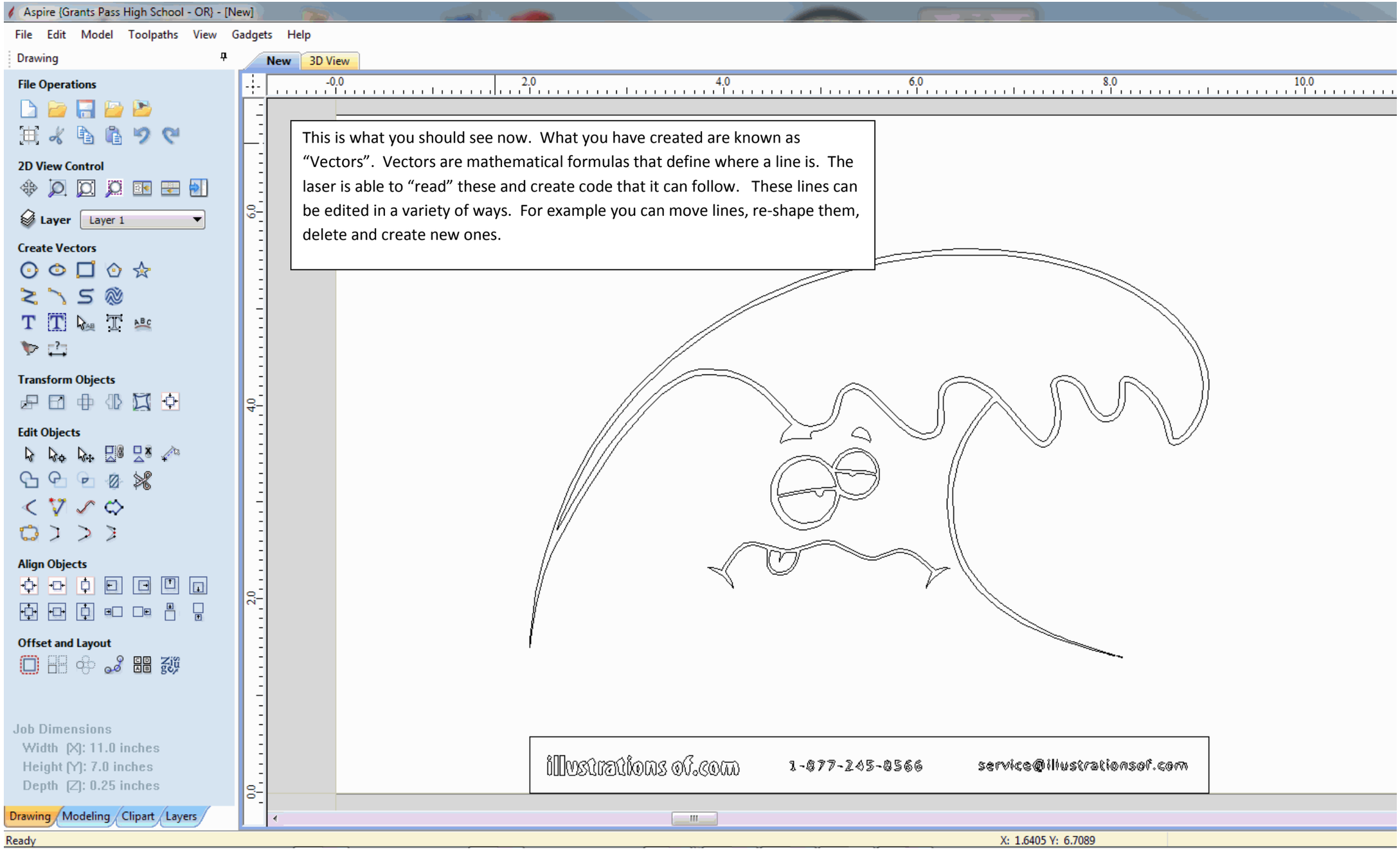


Now you have a “Traced” outline of the image.

Click on “Apply”.

Click on “Close”.


Aspire {Grants Pass High School - OR} - [New]
File Edit Model Toolpaths View Gadgets Help
Drawing
New
3D View
File Operations
2D View Control
Layer
Layer 1
Create Vectors
Transform Objects
Edit Objects
Align Objects
Offset and Layout
Job Dimensions
Width [X]: 11.0 inches
Height [Y]: 7.0 inches
Depth [Z]: 0.25 inches
Drawing
Modeling
Clipart
Layers
illustrations of.com
1-877-245-8566
service@illustrationsof.com
Click anywhere on the image to make it “Active”. This will bring back the red dashed box around the image. Now hit the “Delete” key on the keyboard.
Ready
X: 3.1358 Y: 4.7088
W:7.0000 H:7.0000 L:Bitmap Layer


Aspire {Grants Pass High School - OR} - [New]
File Edit Model Toolpaths View Gadgets Help
Drawing
New
3D View
File Operations
2D View Control
Layer
Layer 1
Create Vectors
Transform Objects
Edit Objects
Align Objects
Offset and Layout
Job Dimensions
Width [X]: 11.0 inches
Height [Y]: 7.0 inches
Depth [Z]: 0.25 inches
Drawing
Modeling
Clipart
Layers
Ready
X: 1.6405 Y: 6.7089
This is what you should see now. What you have created are known as "Vectors". Vectors are mathematical formulas that define where a line is. The laser is able to "read" these and create code that it can follow. These lines can be edited in a variety of ways. For example you can move lines, re-shape them, delete and create new ones.
illustrations of.com
1-877-245-8566
service@illustrationsof.com

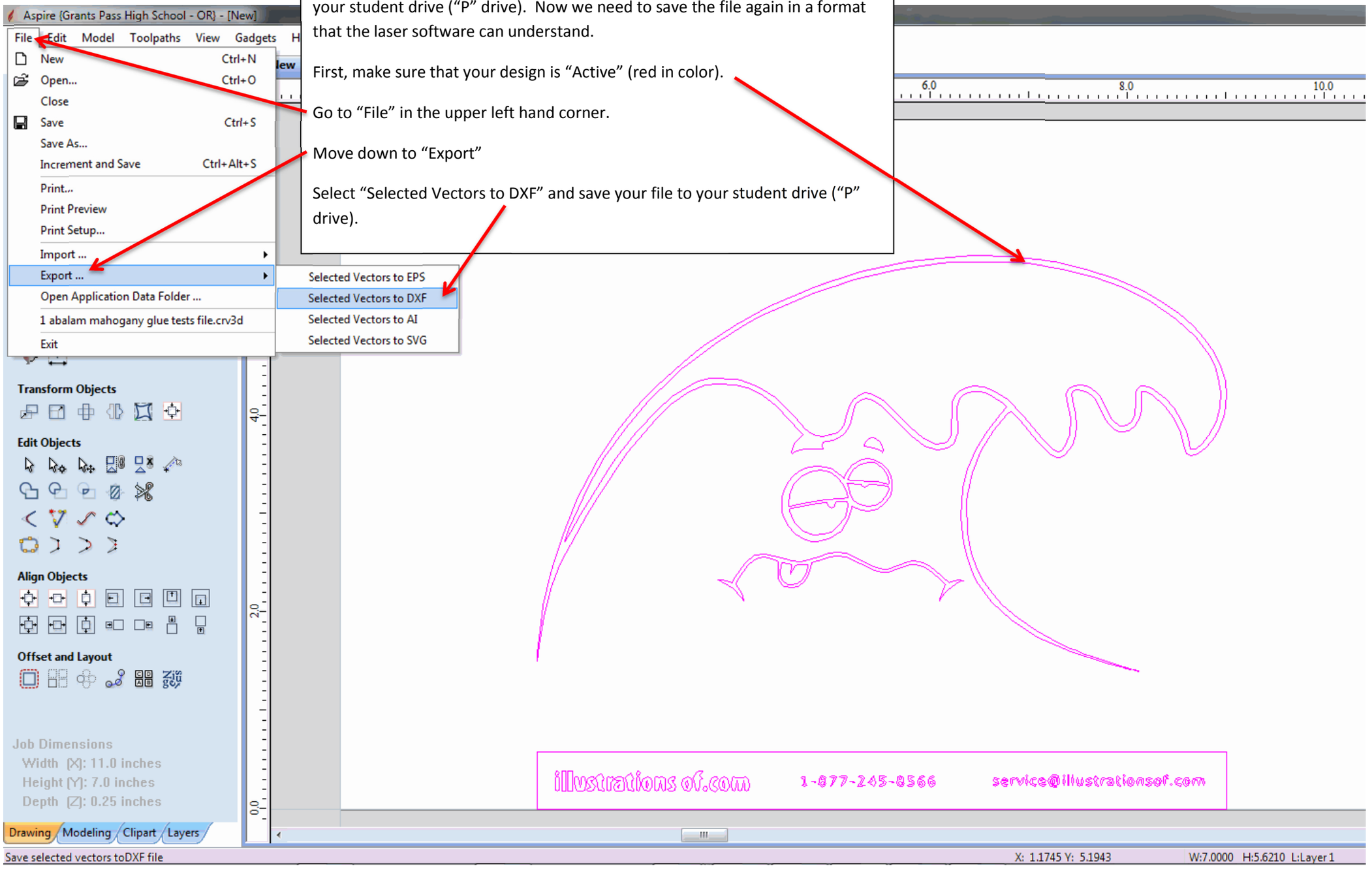
Once you have your design just the way that you want it you should save it to your student drive ("P" drive). Now we need to save the file again in a format that the laser software can understand.

First, make sure that your design is "Active" (red in color).

Go to "File" in the upper left hand corner.

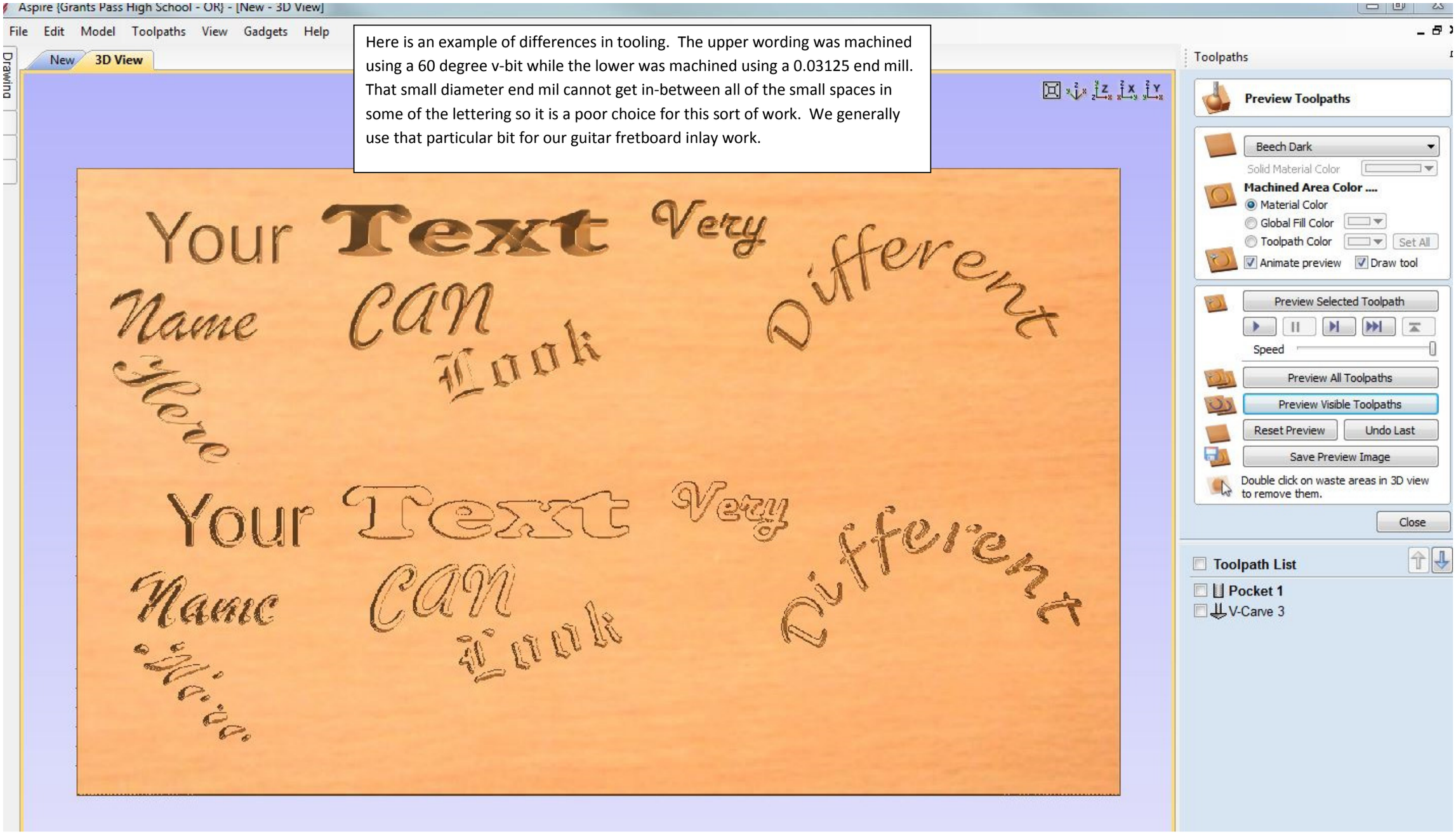
Move down to "Export"

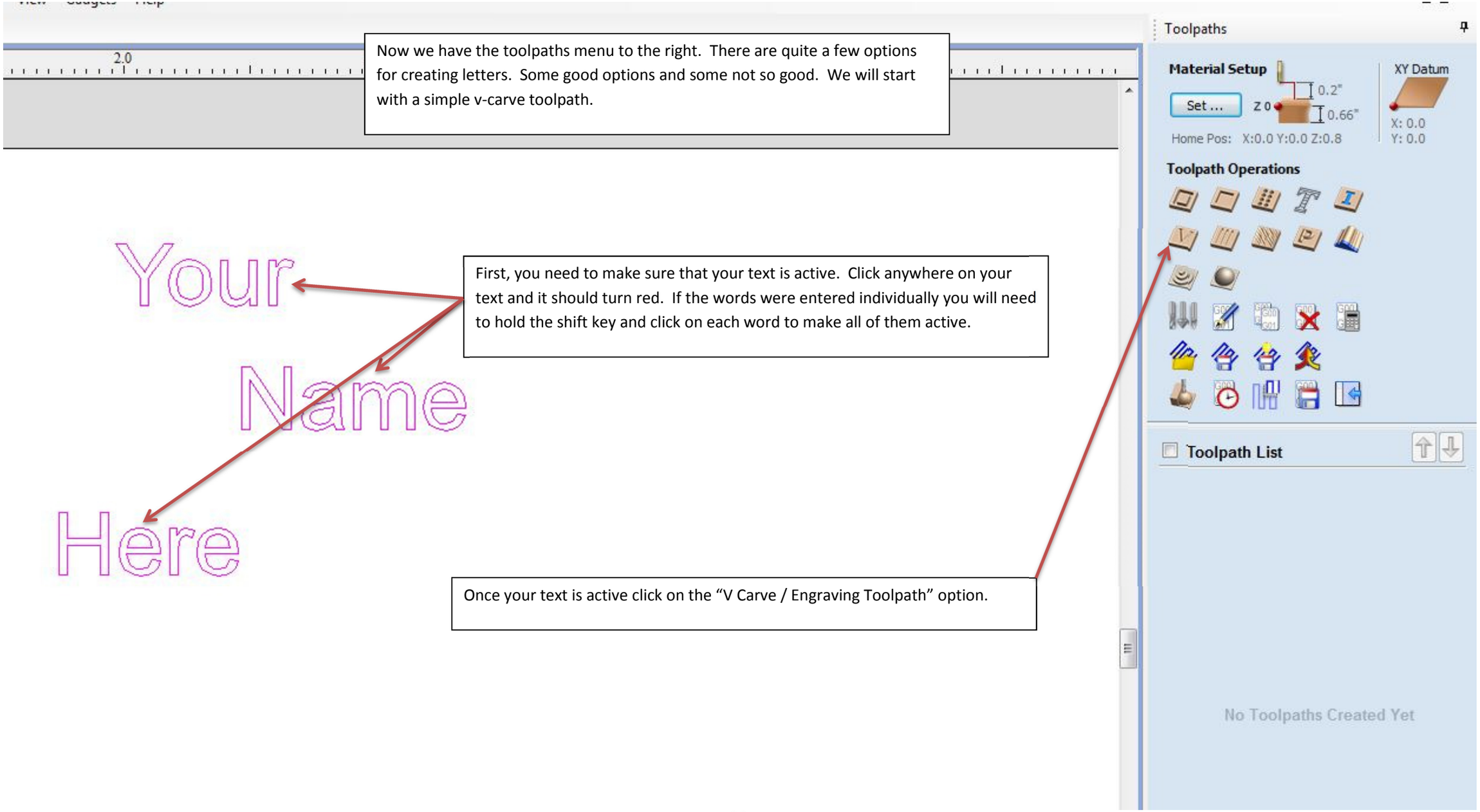
Select "Selected Vectors to DXF" and save your file to your student drive ("P" drive).

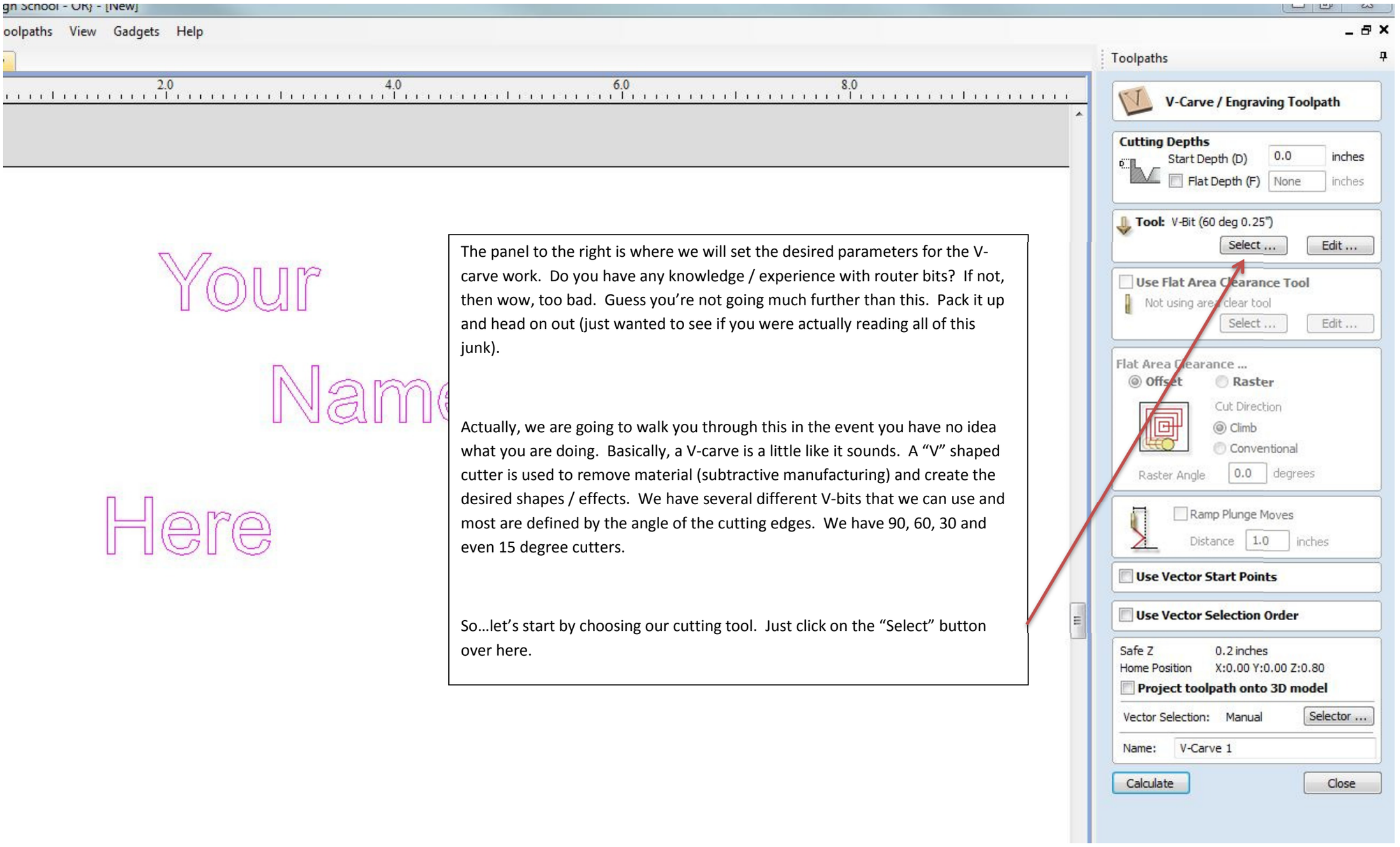


Now you can let your teacher know that your file is ready. You should be handed a flash drive that you can copy your file to and then we will move to the laser for the final preparation of your job.

If you are planning on creating your project using the CNC router, continue reading.


Aspire {Grants Pass High School - OR} - [New - 3D View]
File Edit Model Toolpaths View Gadgets Help
New 3D View
Here is an example of differences in tooling. The upper wording was machined using a 60 degree v-bit while the lower was machined using a 0.03125 end mill. That small diameter end mil cannot get in-between all of the small spaces in some of the lettering so it is a poor choice for this sort of work. We generally use that particular bit for our guitar fretboard inlay work.
Your Text Very Different Name can Look Here
Your Text Very Different Name can Look Here
Toolpaths
Preview Toolpaths
Beech Dark
Solid Material Color
Machined Area Color ....
Material Color
Global Fill Color
Toolpath Color
Set All
Animate preview
Draw tool
Preview Selected Toolpath
Speed
Preview All Toolpaths
Preview Visible Toolpaths
Reset Preview
Undo Last
Save Preview Image
Double click on waste areas in 3D view to remove them.
Close
Toolpath List
Pocket 1
V-Carve 3

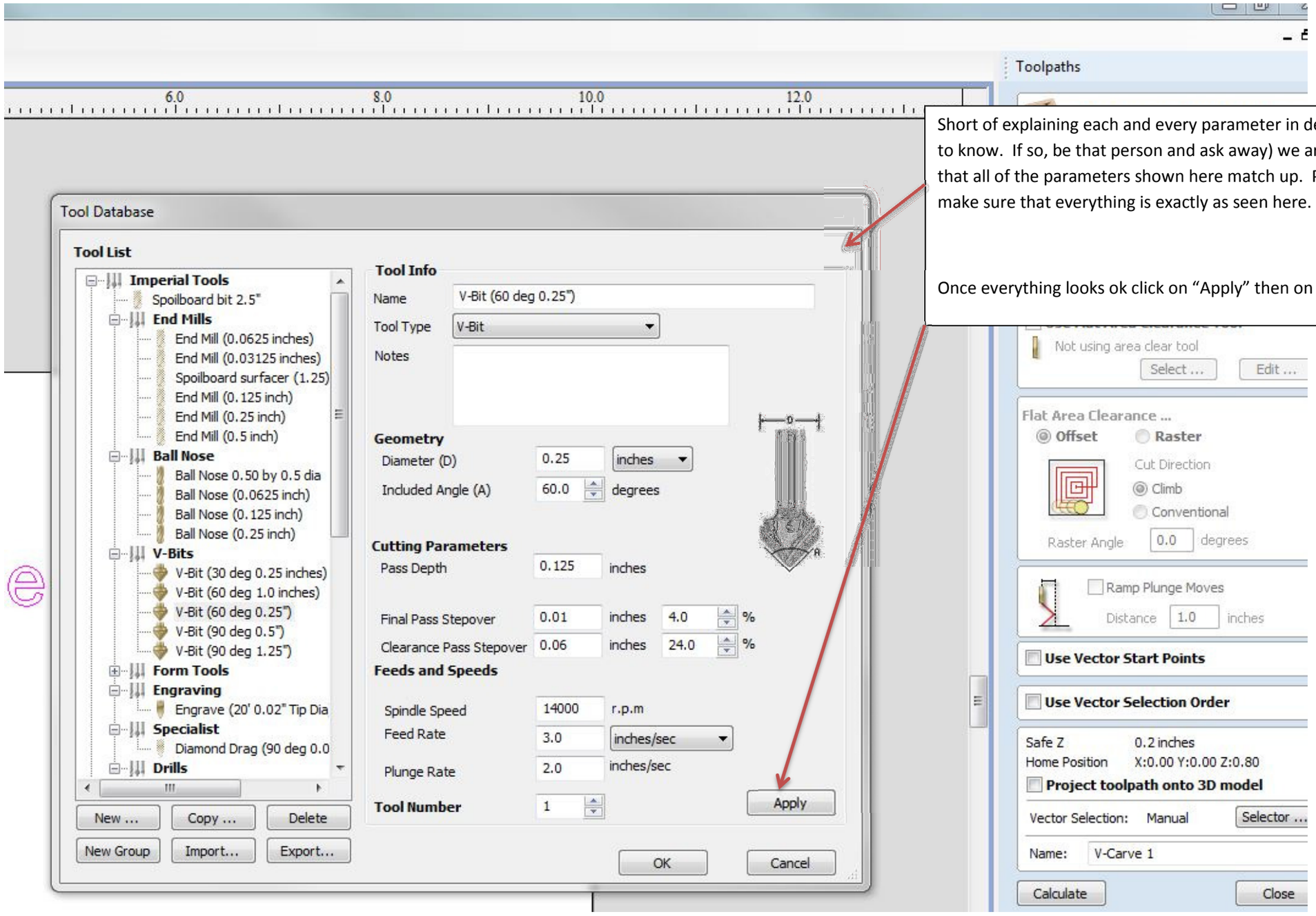

Now we have the toolpaths menu to the right. There are quite a few options for creating letters. Some good options and some not so good. We will start with a simple v-carve toolpath.
Your
Name
Here
First, you need to make sure that your text is active. Click anywhere on your text and it should turn red. If the words were entered individually you will need to hold the shift key and click on each word to make all of them active.
Once your text is active click on the “V Carve / Engraving Toolpath” option.
Toolpaths
Material Setup
Set ...
Z 0
0.2"
0.66"
XY Datum
X: 0.0
Y: 0.0
Home Pos: X:0.0 Y:0.0 Z:0.8
Toolpath Operations
Toolpath List
No Toolpaths Created Yet



The panel to the right is where we will set the desired parameters for the V-carve work. Do you have any knowledge / experience with router bits? If not, then wow, too bad. Guess you're not going much further than this. Pack it up and head on out (just wanted to see if you were actually reading all of this junk).

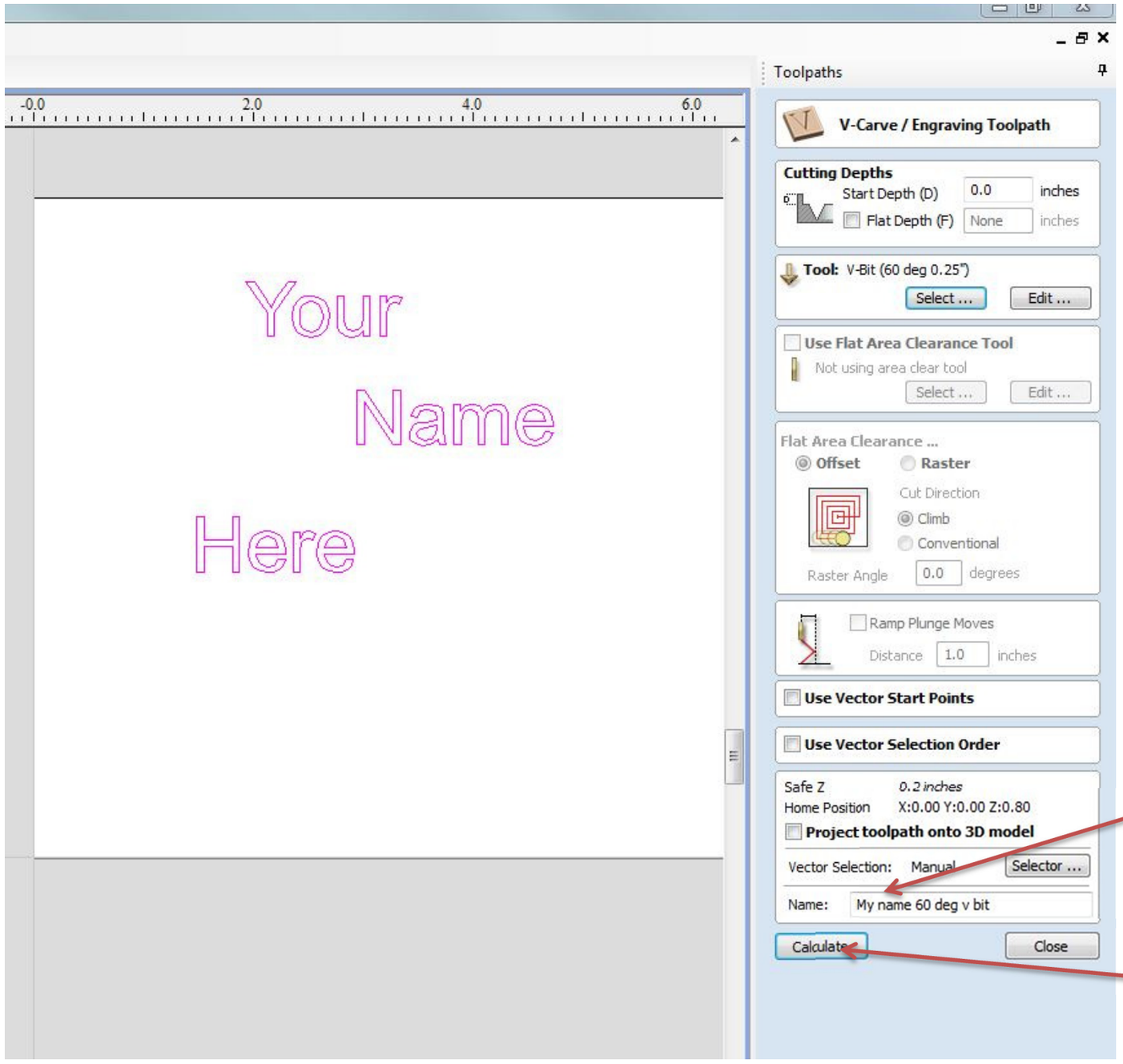
Actually, we are going to walk you through this in the event you have no idea what you are doing. Basically, a V-carve is a little like it sounds. A "V" shaped cutter is used to remove material (subtractive manufacturing) and create the desired shapes / effects. We have several different V-bits that we can use and most are defined by the angle of the cutting edges. We have 90, 60, 30 and even 15 degree cutters.

So…let's start by choosing our cutting tool. Just click on the "Select" button over here.



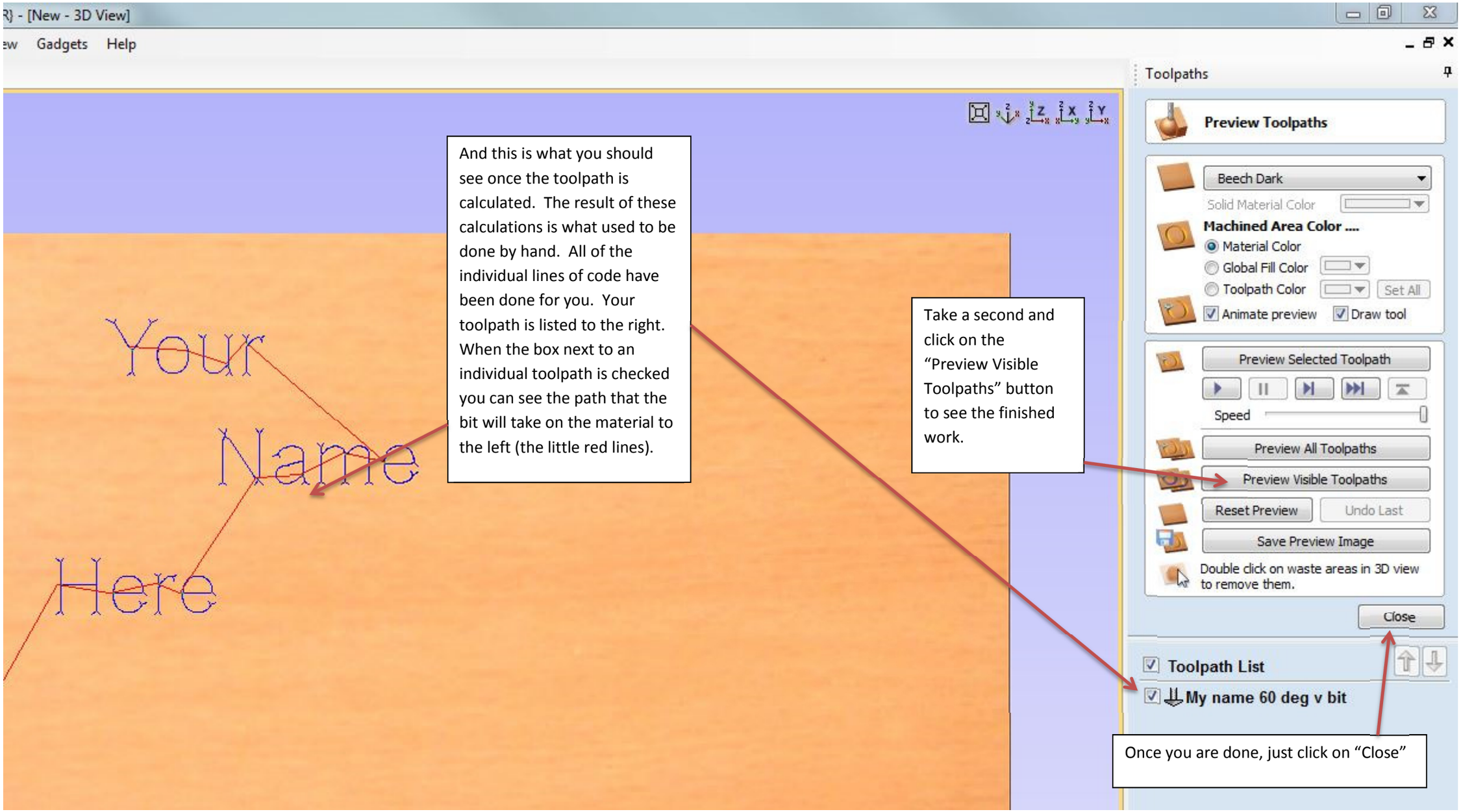
Short of explaining each and every parameter in detail (unless you really want to know. If so, be that person and ask away) we are simply going to make sure that all of the parameters shown here match up. Please take your time and make sure that everything is exactly as seen here.

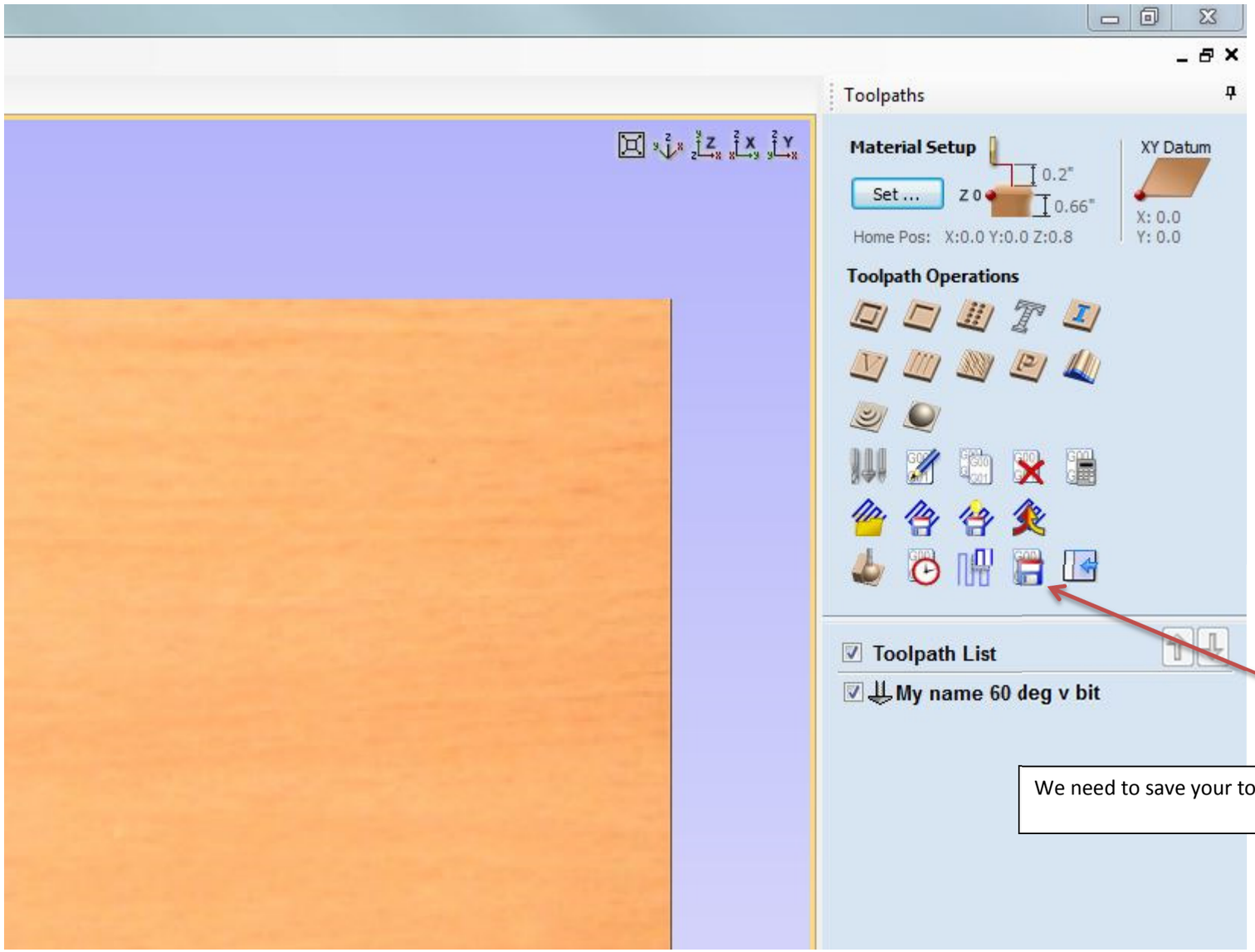
Once everything looks ok click on “Apply” then on “OK”.



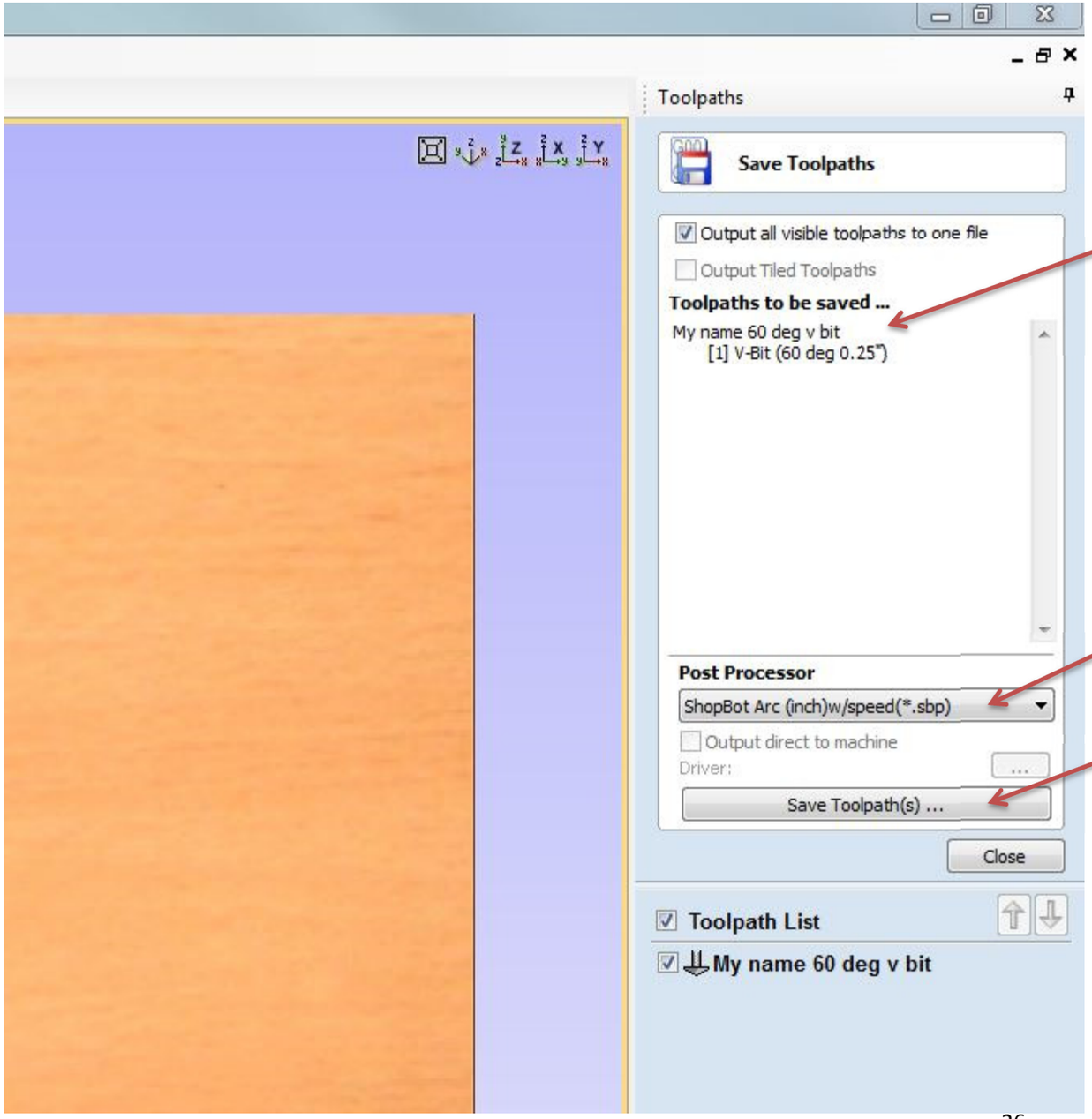
Make sure that everything to the left matches what you have on your project and give the toolpath a name (your name is preferable) and add the tool that will be used for that particular work.

Once you have done this just click on “Calculate”.


R} - [New - 3D View]
ew Gadgets Help
Your
Name
Here
And this is what you should see once the toolpath is calculated. The result of these calculations is what used to be done by hand. All of the individual lines of code have been done for you. Your toolpath is listed to the right. When the box next to an individual toolpath is checked you can see the path that the bit will take on the material to the left (the little red lines).
Take a second and click on the "Preview Visible Toolpaths" button to see the finished work.
Toolpaths
Preview Toolpaths
Beech Dark
Solid Material Color
Machined Area Color ....
Material Color
Global Fill Color
Toolpath Color
Set All
Animate preview
Draw tool
Preview Selected Toolpath
Speed
Preview All Toolpaths
Preview Visible Toolpaths
Reset Preview
Undo Last
Save Preview Image
Double click on waste areas in 3D view to remove them.
Close
Toolpath List
My name 60 deg v bit
Once you are done, just click on "Close"



We need to save your toolpaths, so click on the “Save Toolpath” icon.



You should see your named toolpath here.

Double check that the Post Processor being used is the same as the one here.

Now click on the “Save Toolpath” button and save the file to your desktop.

My name 60 deg v bit.sbp - ShopBot Edit

File Edit View Tools Help

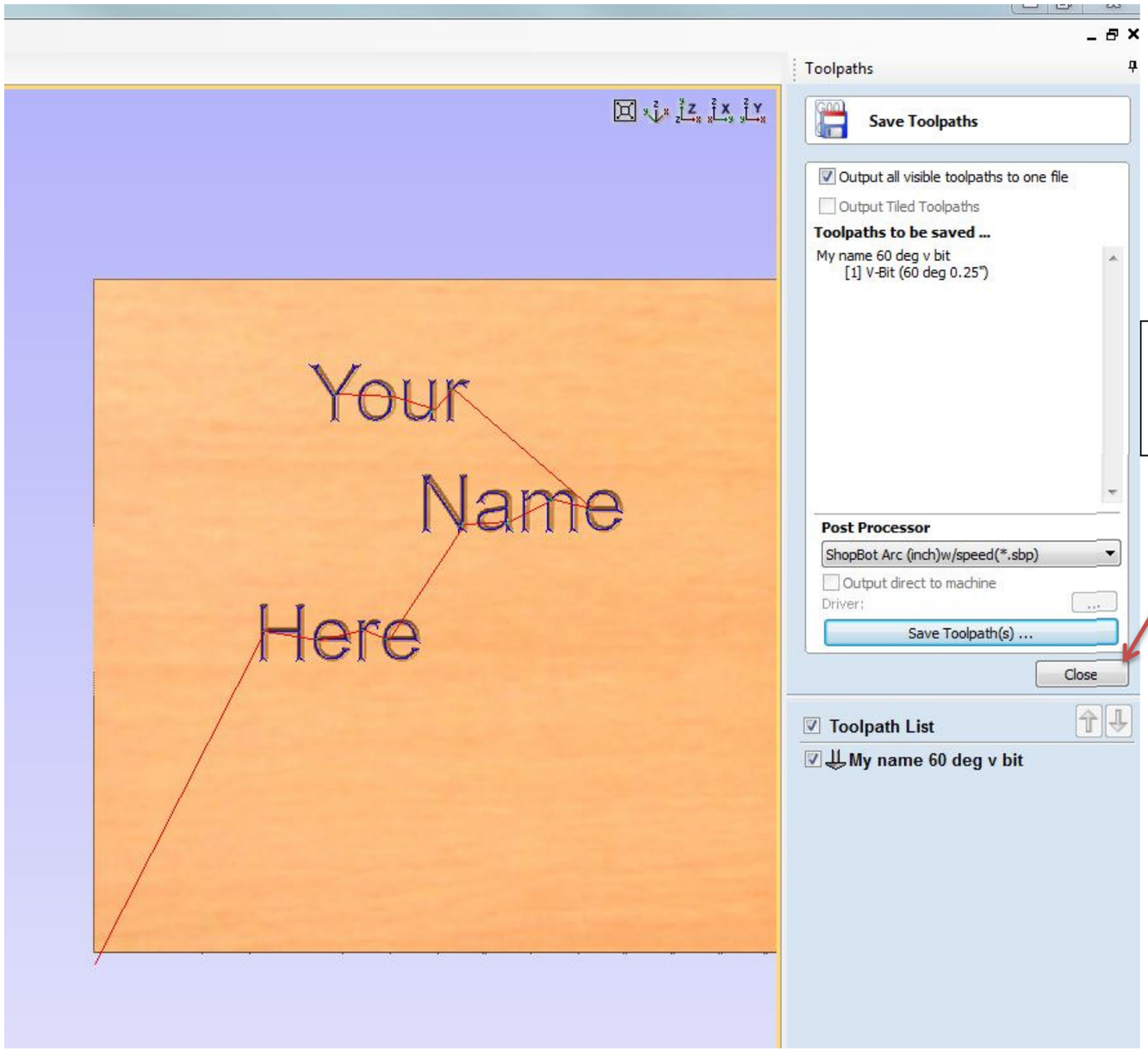
SB-Preview | SB-Cut

```
'My name 60 deg v bit
'File created: Wednesday March 14 2018 - 01:30 PM
'From Vectric
'
'SHOPBOT FILE IN INCHES
IF %(25)=1 THEN GOTO UNIT_ERROR 'check to see software is set to standard
C#,90                    'Lookup offset values
'
TR,14000,1
'
'
'Turning router ON
SO,1,1
PAUSE 2
'
'Toolpath Name = My name 60 deg v bit
'Tool Name    = V-Bit (60 deg 0.25")
MS,2.99,1.99
JZ,0.800000
J2,0.000000,0.000000
J3,1.518034,2.861266,0.200000
M3,1.518034,2.861266,-0.068694
M3,1.514745,2.852625,-0.062998
M3,1.513101,2.846515,-0.060150
M3,1.511457,2.831764,-0.057302
M3,1.511457,2.629172,-0.057302
M3,1.478374,2.596088,0.000000
M3,1.511457,2.629172,-0.057302
M3,1.544540,2.596088,0.000000
M3,1.511457,2.629172,-0.057302
M3,1.511457,2.831764,-0.057302
M3,1.513101,2.846515,-0.060150
M3,1.514745,2.852625,-0.062998
M3,1.518034,2.861266,-0.068694
M3,1.526499,2.861266,-0.059896
M3,1.535792,2.861266,-0.053298
M3,1.544540,2.861266,-0.051099
M3,1.804431,2.861266,-0.051099
M3,1.813179,2.861266,-0.053298
M3,1.822472,2.861266,-0.059896
M3,1.830937,2.861266,-0.068694
M3,1.834226,2.852625,-0.062998
M3,1.835870,2.846515,-0.060150
M3,1.837514,2.831764,-0.057302
M3,1.837514,2.629172,-0.057302
M3,1.804431,2.596088,0.000000
M3,1.837514,2.629172,-0.057302
M3,1.870597,2.596088,0.000000
M3,1.837514,2.629172,-0.057302
M3,1.837514,2.831764,-0.057302
M3,1.835870,2.846515,-0.060150
M3,1.834226,2.852625,-0.062998
```

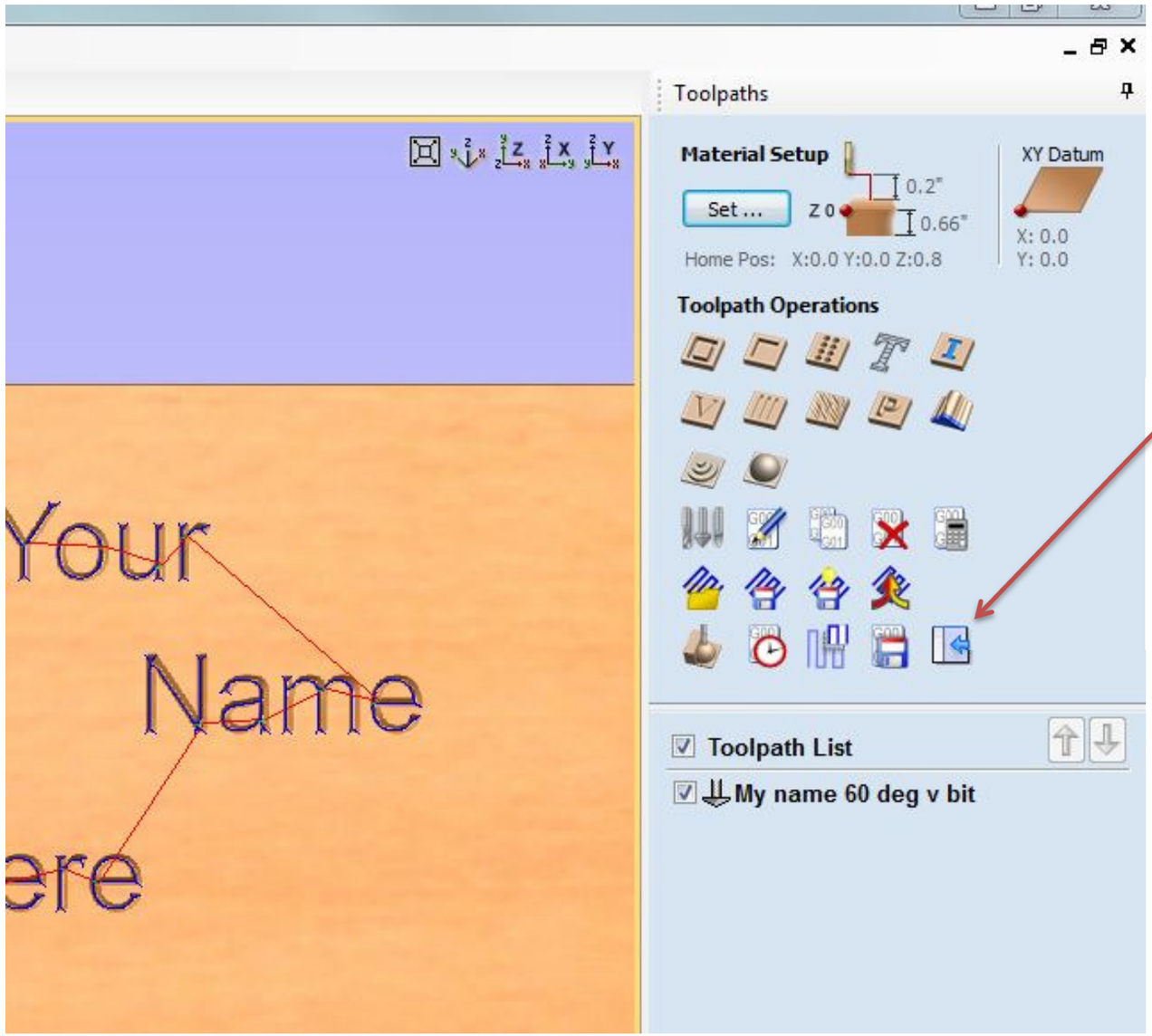
What you see to the left are all of the instructions that the “Calculate” button generated. This is known as “G-Code” and for the most part, it is all mathematical instructions for the machine based on the Cartesian coordinate system (x, y and z-axis movement). The cool thing is that you can physically edit this code or any of the other instructions contained in the file and these changes will be carried over to the machine when you run the file.

What really blows most kids minds is this. That simple “Your Name Here” carving that is represented by the code seen here? It takes well over a thousand lines of code to instruct the machine how to cut this simple job. We have run files in the shop that have over a million lines of code before. The file for just the seat on one of our Sam Maloof rocking chairs has over 156000 lines of code. One of our guitar necks takes over 15000 lines of code. Imagine having to write all of that by hand, line by line.

```
1330  M3,2.148743,4.971967,-0.059998
1331  M3,2.153031,4.968952,-0.063833
1332  J3,2.153031,4.968952,0.200000
1333  JZ,0.800000
1334  J2,0.000000,0.000000
1335
1336  'Turning router OFF
1337  SO,1,0
1338  END
1339  UNIT_ERROR:
1340  C#,91                    'Run file explaining unit error
1341  END
1342
```



Let's complete a couple more things before we move over to the shop to machine our project. First, click on "Close" to get out of the Save Toolpaths window.



Click on the “Switch to Drawing” tab and save your file just as you did earlier by clicking on “File in the upper left corner and saving your work to the desktop.

At this time, try to get the attention of the so called “Teachers” in the room and ask for a flash drive. We are going to copy your file to said drive so we can take it over to the shop and run it on the CNC.

That is pretty much all we are going to do with this particular tutorial. We will work with you on an individual basis if you want to do anything more complex. Feel free to ask if there is anything more that you would like to do or know.