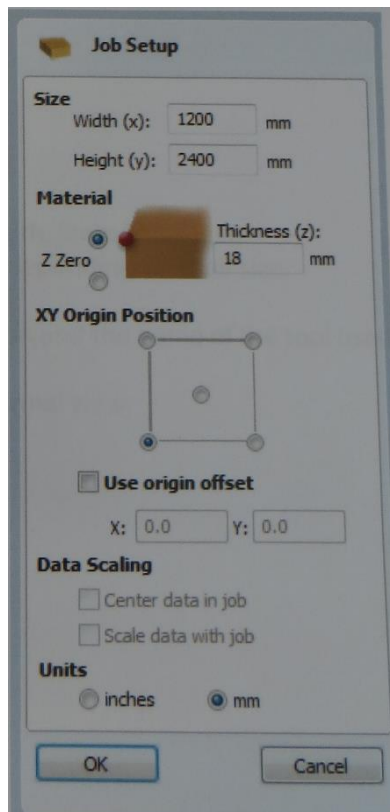


vCarve - convert 3D models into gCode for CNC Router

Step 1) Setting up the sheet size, thickness and Origin

Set size of sheet 1220mm by 2440mm or whatever the side of material you are using as well as its thickness. I used 18mm OSB. Make sure origin in bottom right.



2) Import DXF files

Open up drop down File menu. Then select import and select import vectors. If importing several files, Select M to move them otherwise they'll import on top of each other.

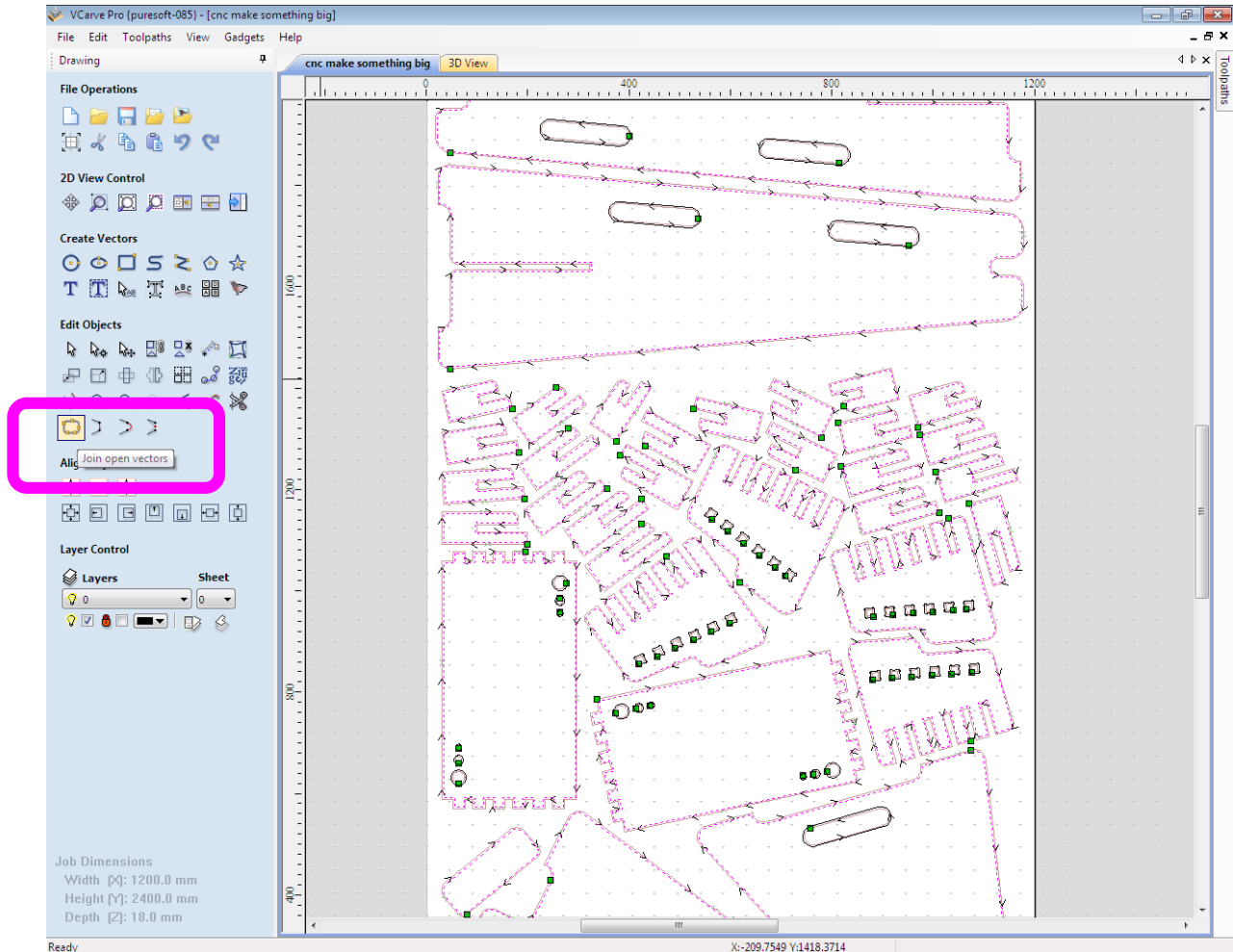
Before you copy and paste all the pieces you want to cut on the CNC. Import one of each piece. When you have joined an unjoined vectors, filleted the edges and grouped the vectors together then copy and paste each piece.

Trimmed away any unwanted lines.

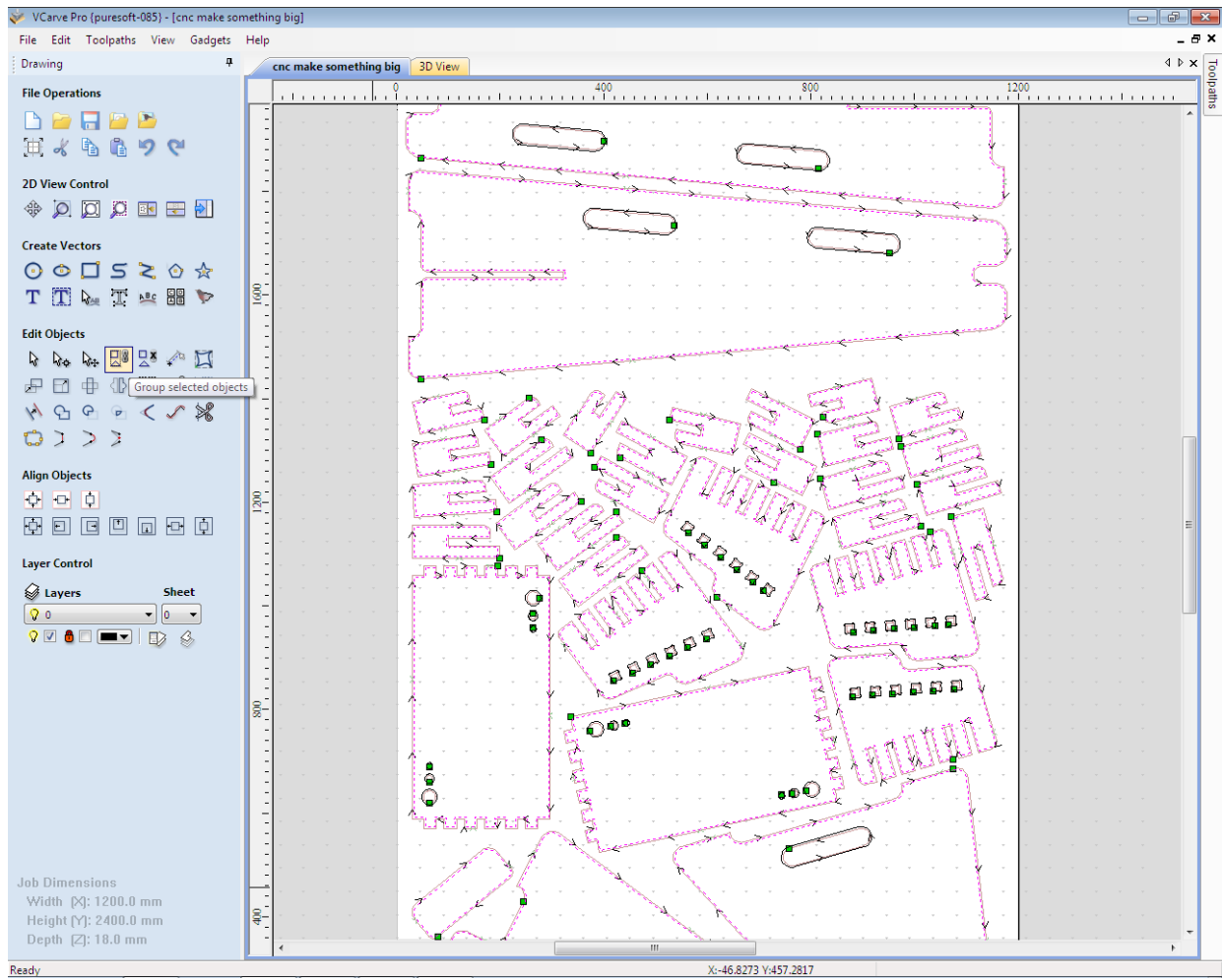
3) Joining Lines

Sometimes there can be lines that aren't joined even though they visually looked joined. You need to select everything and then join these. (As long as you want them joined and they are closed shapes.)

make sure all the pieces have no open lines (use the joining Tool)



4) Group the lines together

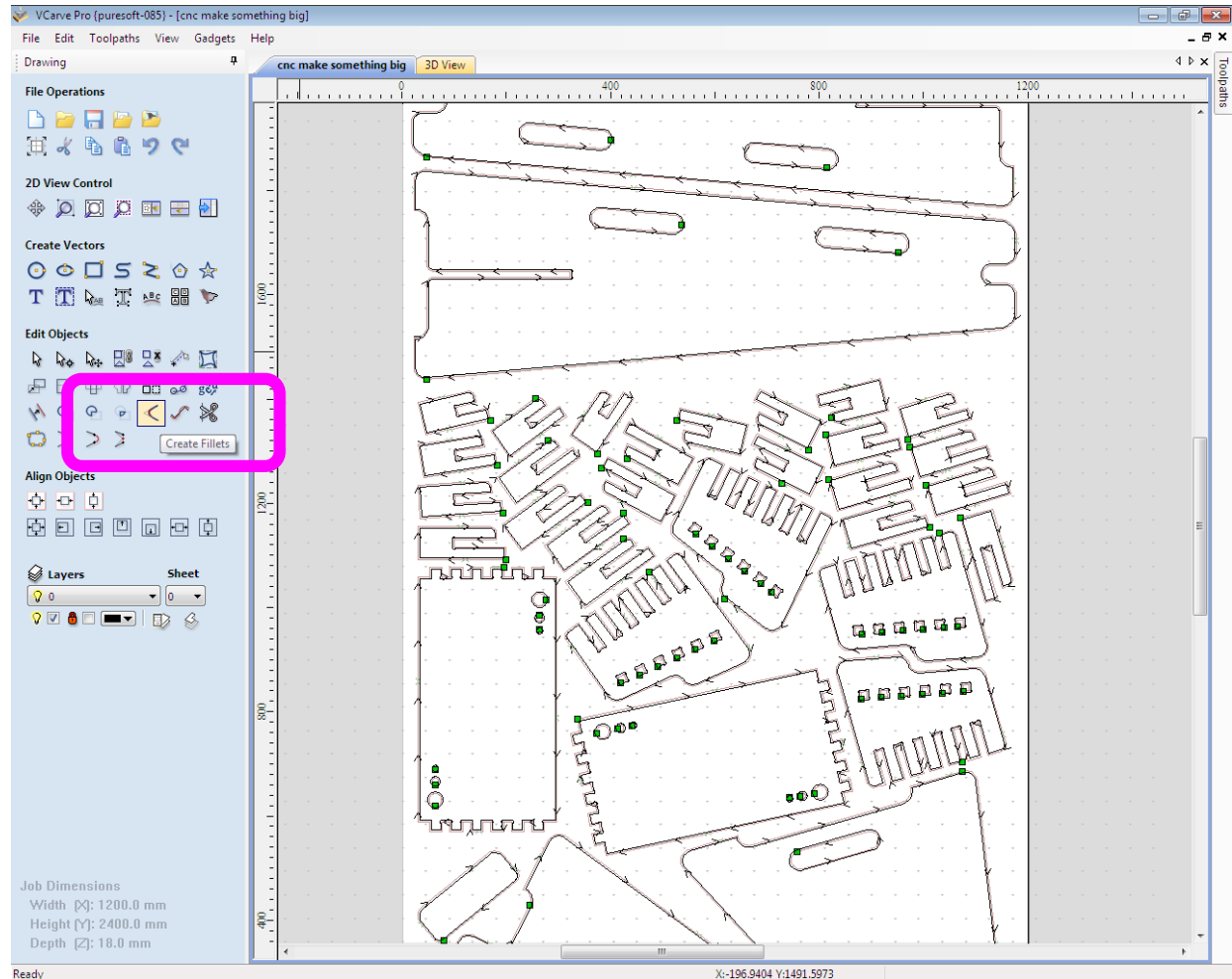


5) Fillet the Corners:

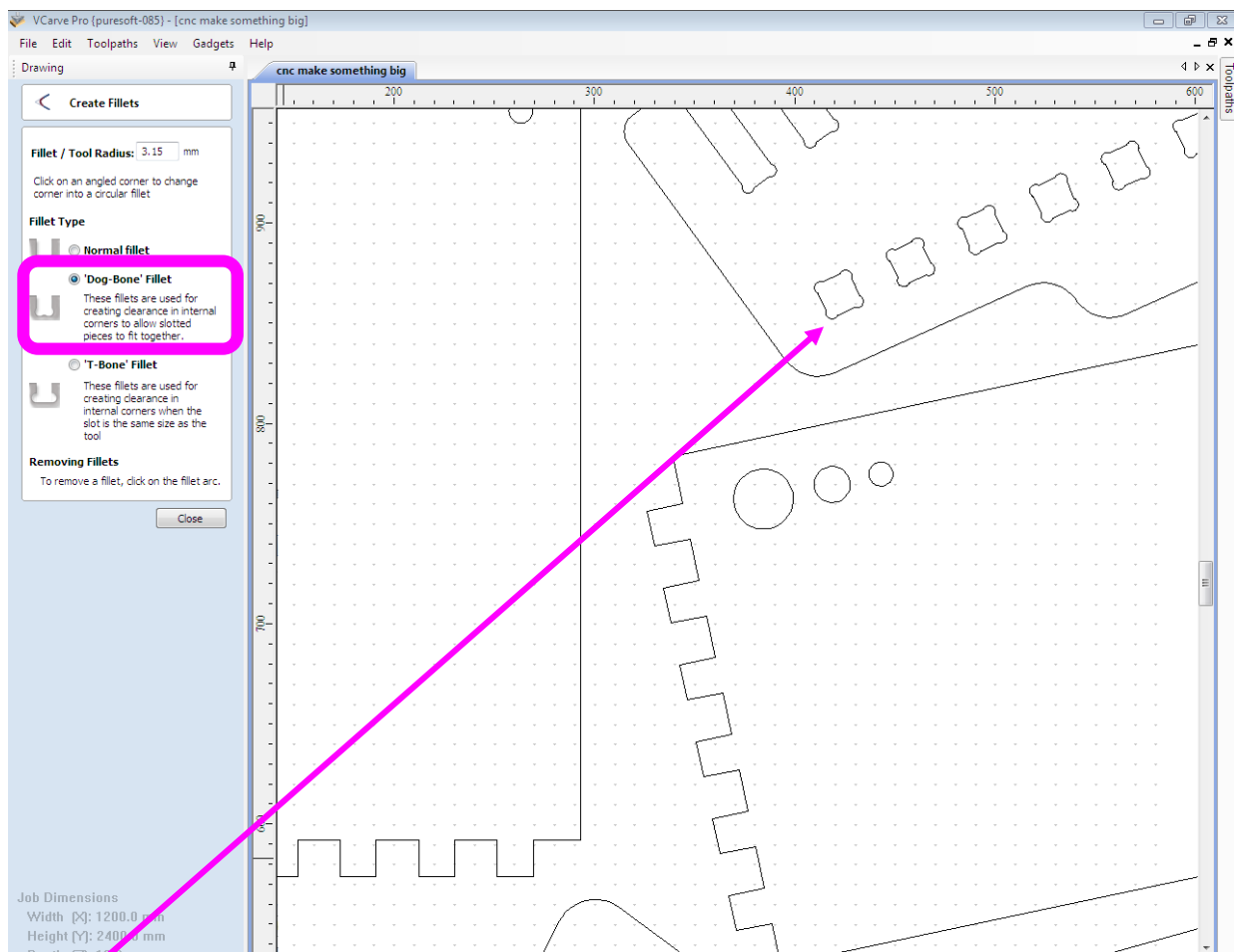
Add the fillets so the tool piece will cut into the Corners (internal corners that join together. Otherwise if you have piece locking together like I do in the stool they won't slot together.)

Make sure the fillets are half the size of the tool bit.

Fillet the corners of the pieces to match the size of the tool bit

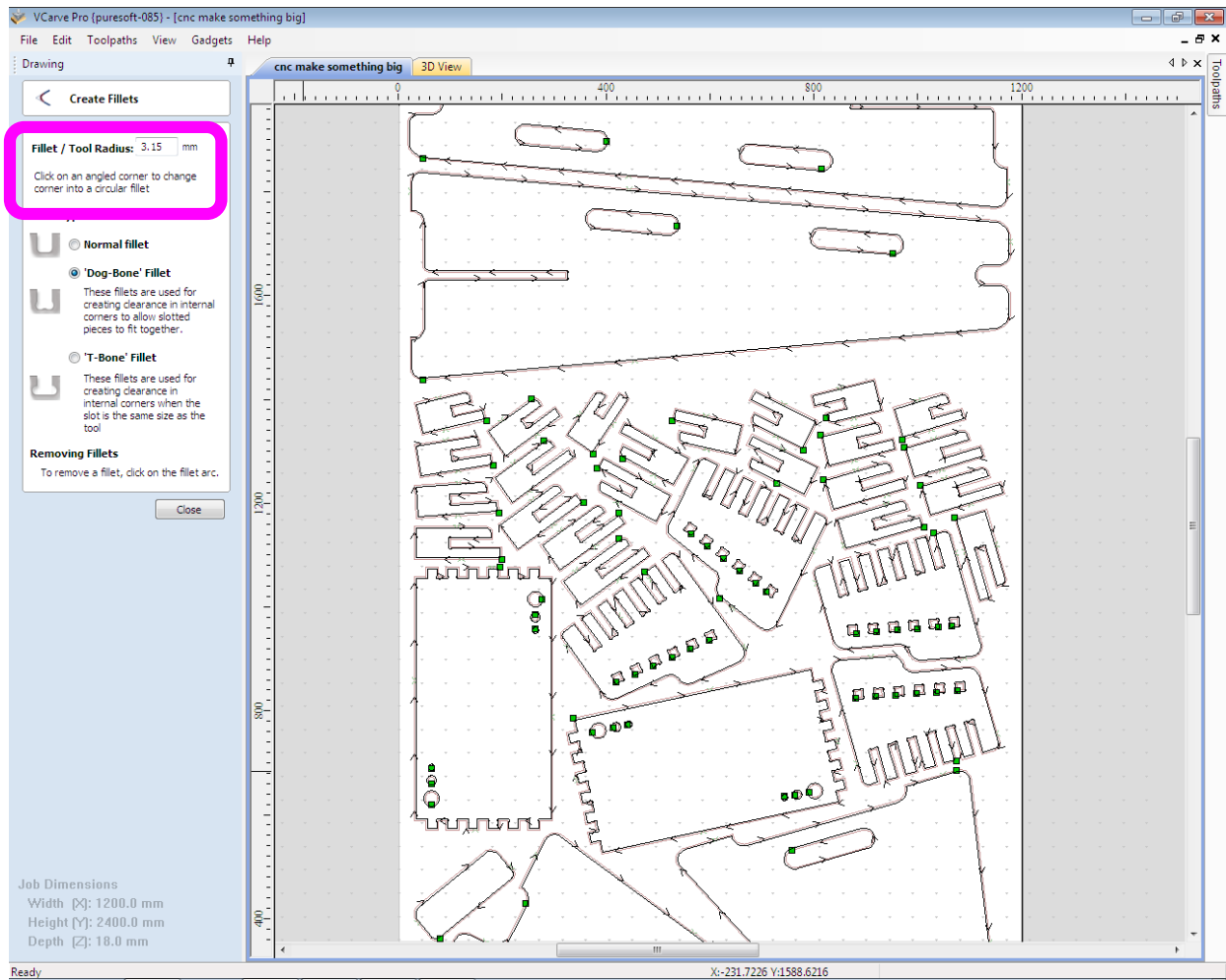


Use dog bite



Notice after filleting the tool bit will now cut into the wood more at the corners where I have joints that fit together.

Match radius of fillet to the size of the toolbit

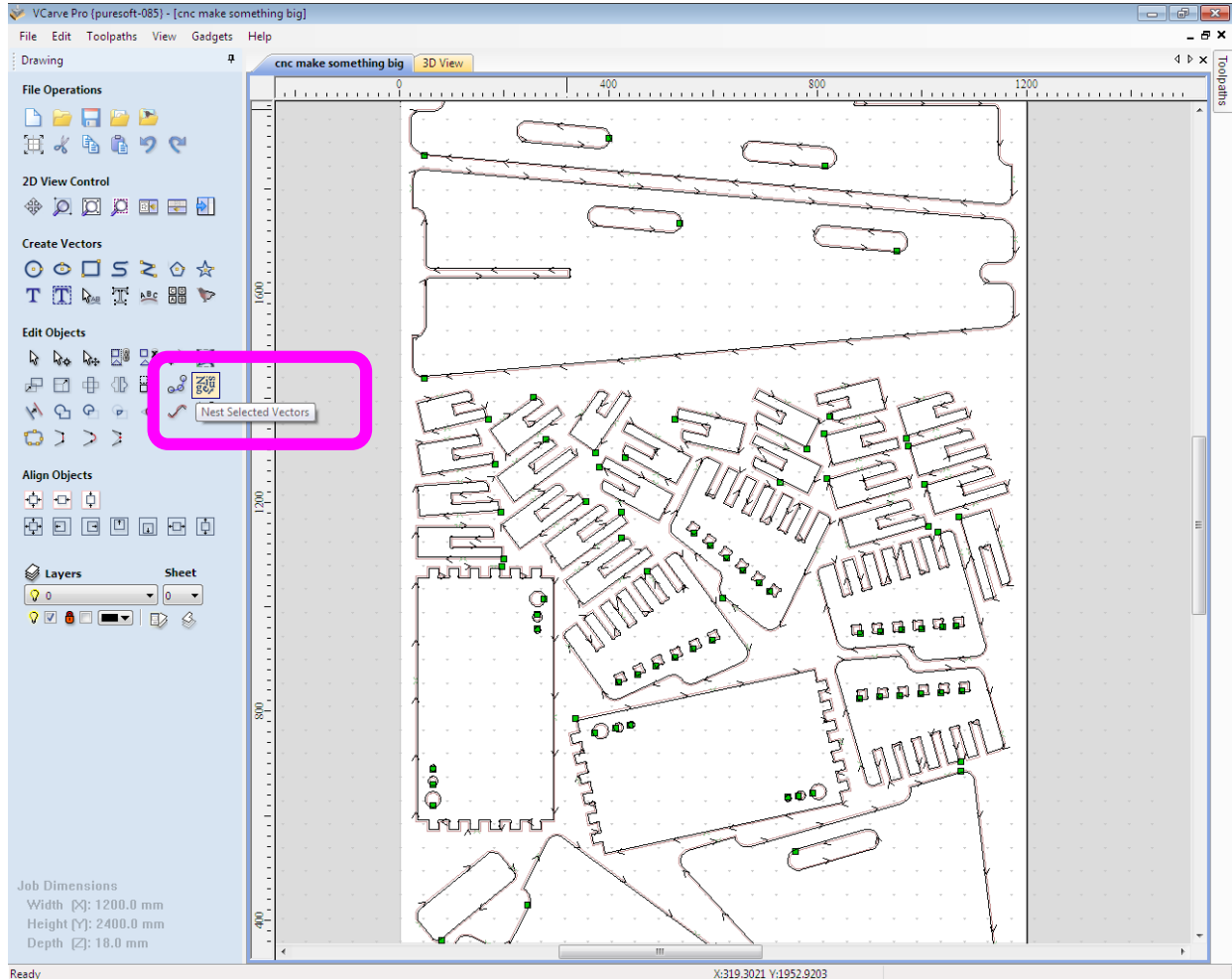


5) Nest:

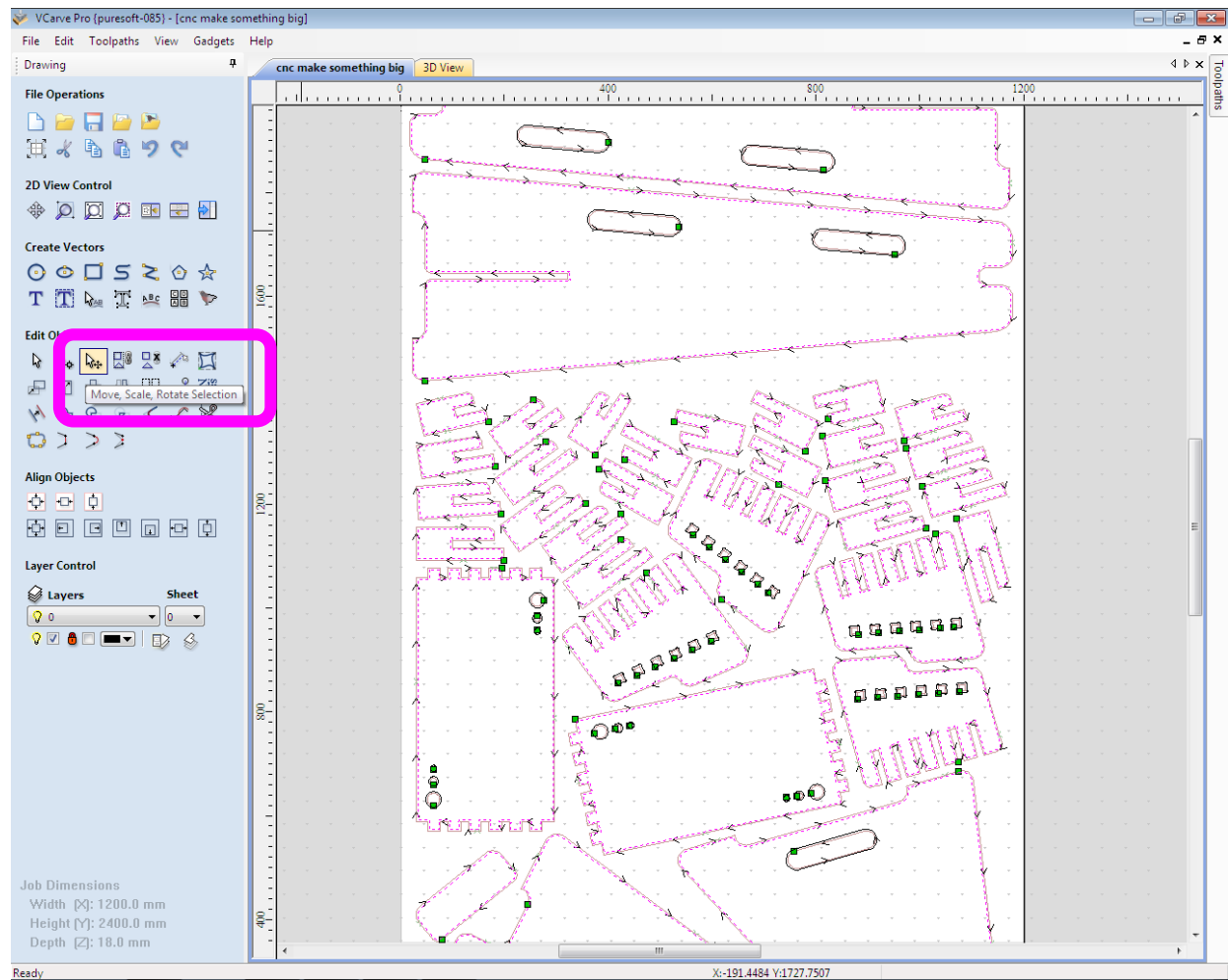
You can use Nest to automatically orientate the piece to cut to best use the size of the sheet you have giving clearance between parts and around the edge.

Once you have made all the fillets and are sure you have all the pieces to be cut on the screen.

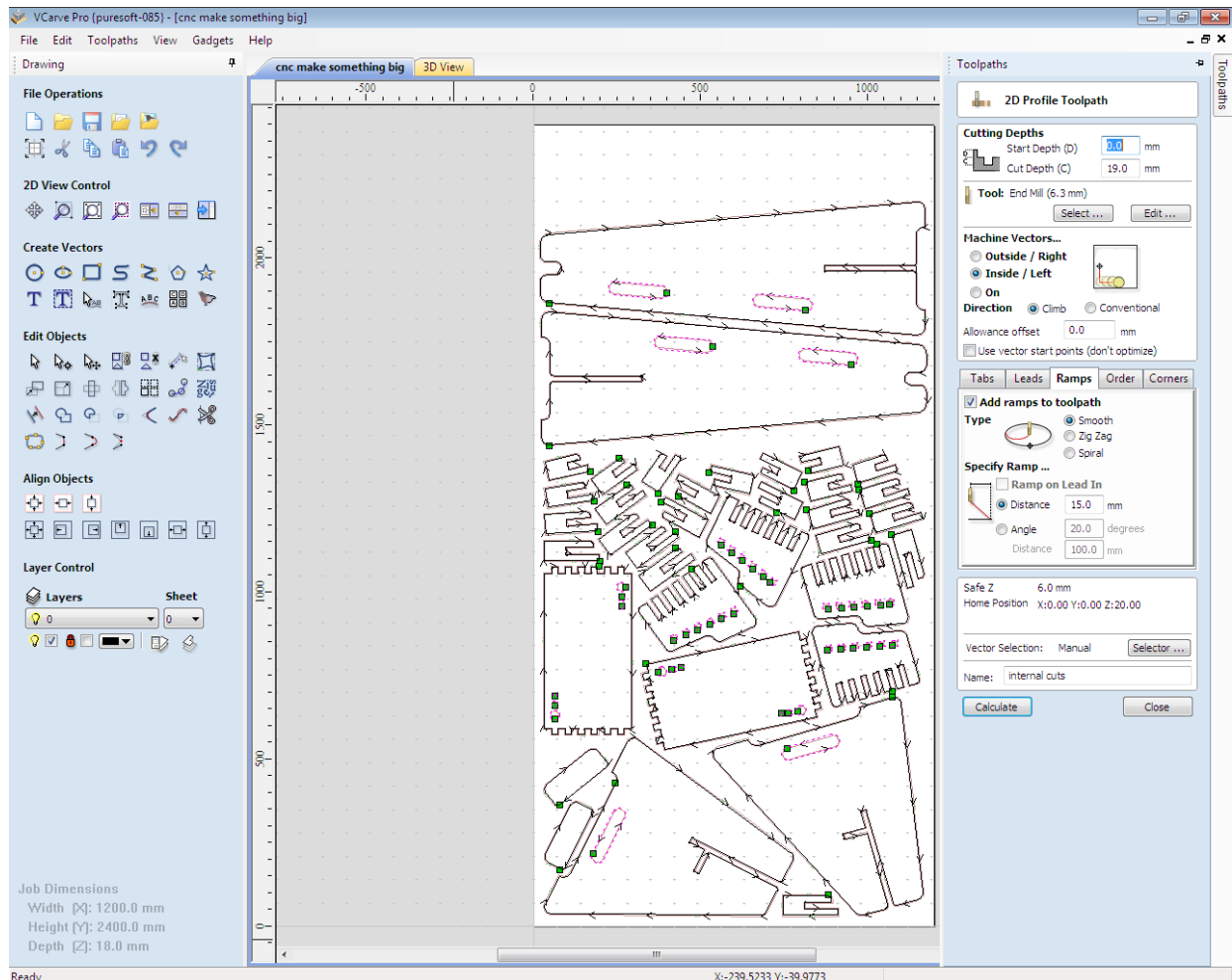
We nest them. This asks the software to automatically layout the pieces to be cut on the material in the best way that uses the material. It takes some time for the software to generate this so be patient.



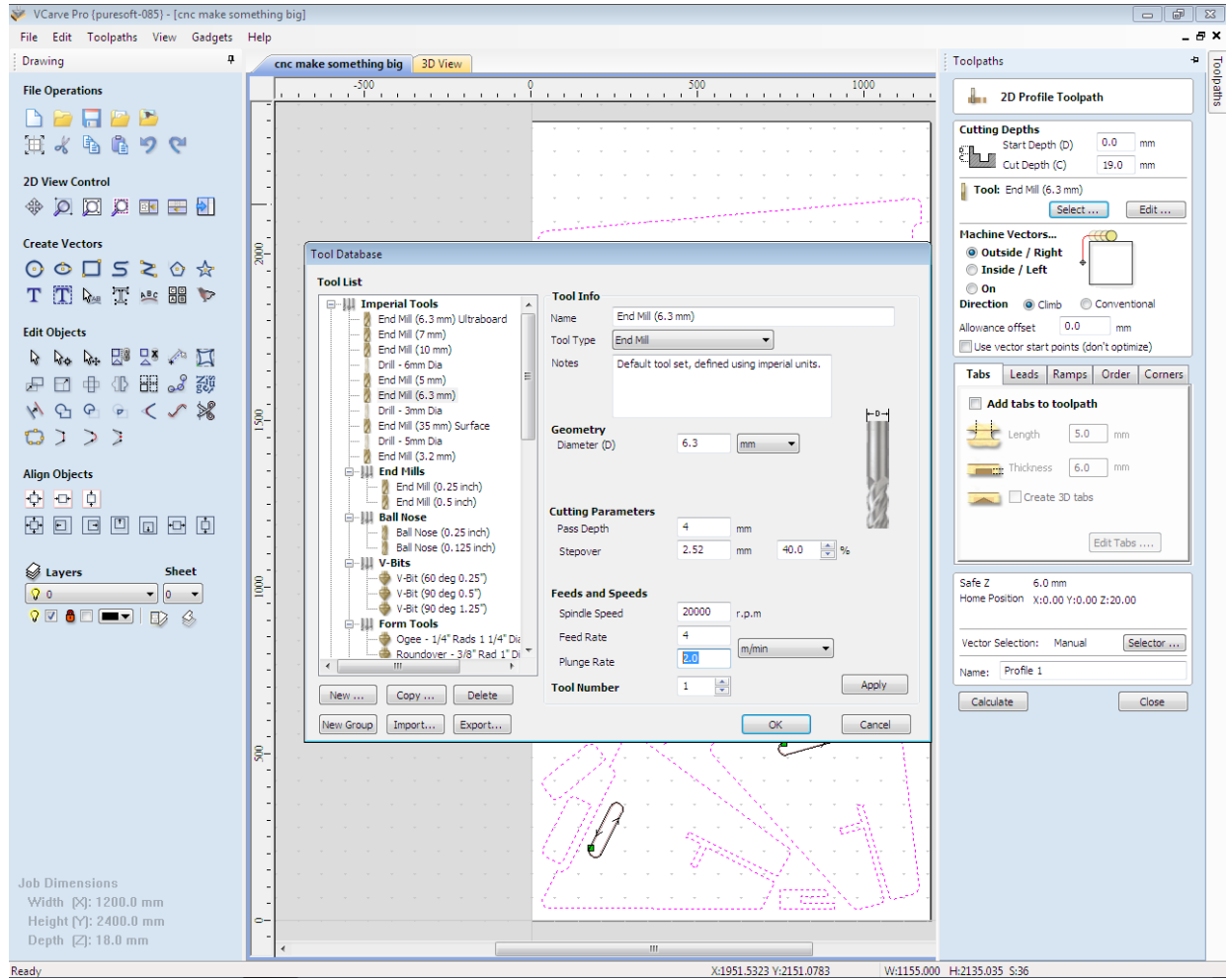
The software is normally much better than you at layout out the pieces to make the best use of the material but if you do want to move pieces around use the Move, Scale, Rotate tool shown below.



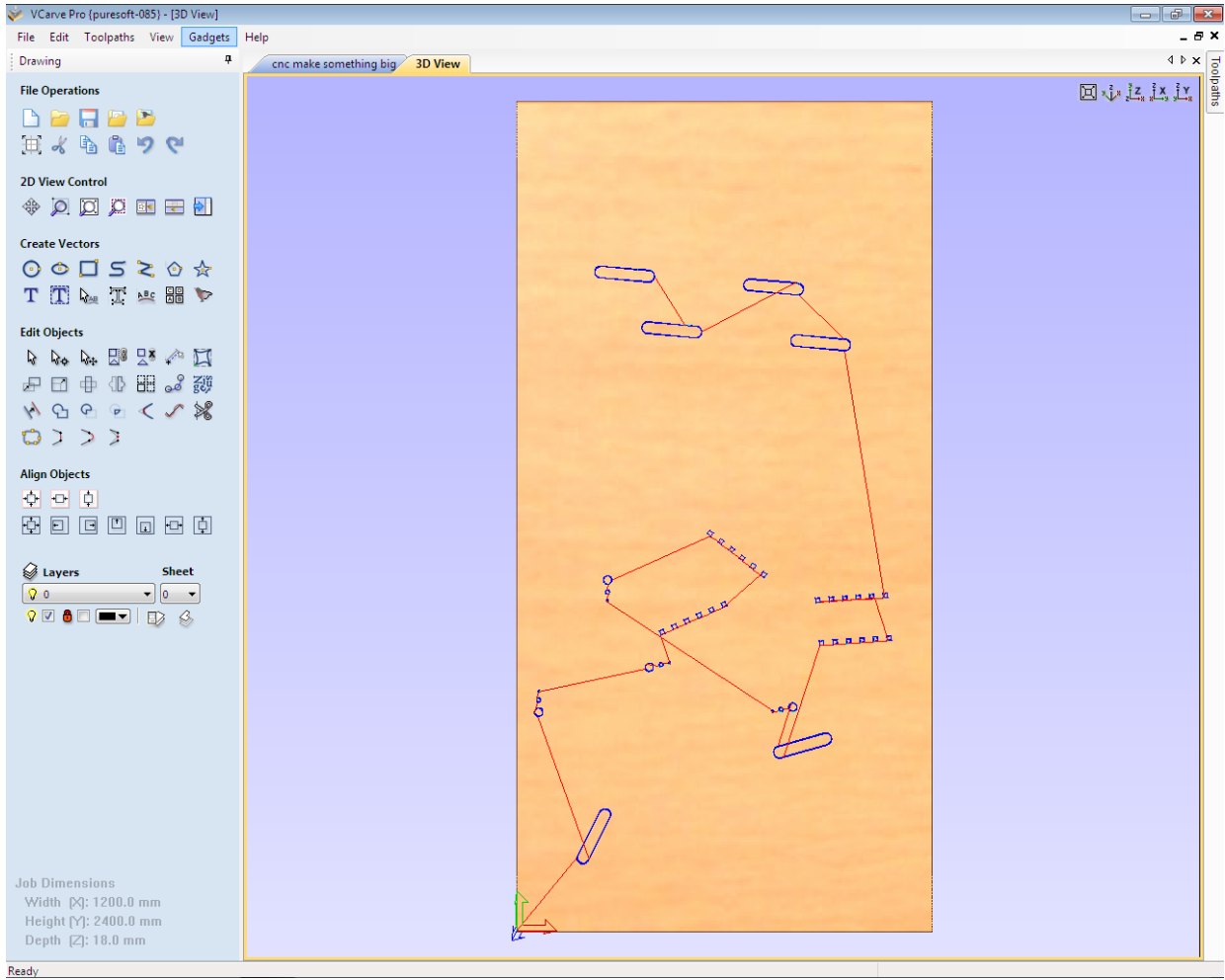
Set tool piece to correct one for the machine making sure the depth, feed rate etc is correct. I used 19mm depth even though the wood is 18mm. This is to make sure it cuts right through the wood. The start depth is 0mm. I used a 6.3mm End Mill. For routing the internal piece I used the 'inside/Left' option under Machine Vectors to make sure it cut inside the shapes and not into the remaining piece of material I want to use for the stool or bike stand. We can name this tool path as well. I've named it internal cuts. Make sure the inner or outer is selected properly for what you are trying to do. For inner cuts you want the inner. Then select the vectors you want to cut, holding down shift as you go to select more than one. Once all of them are selected then select calculate on the bottom right of the toolpaths panel.



Select the tool bit as well shown below.

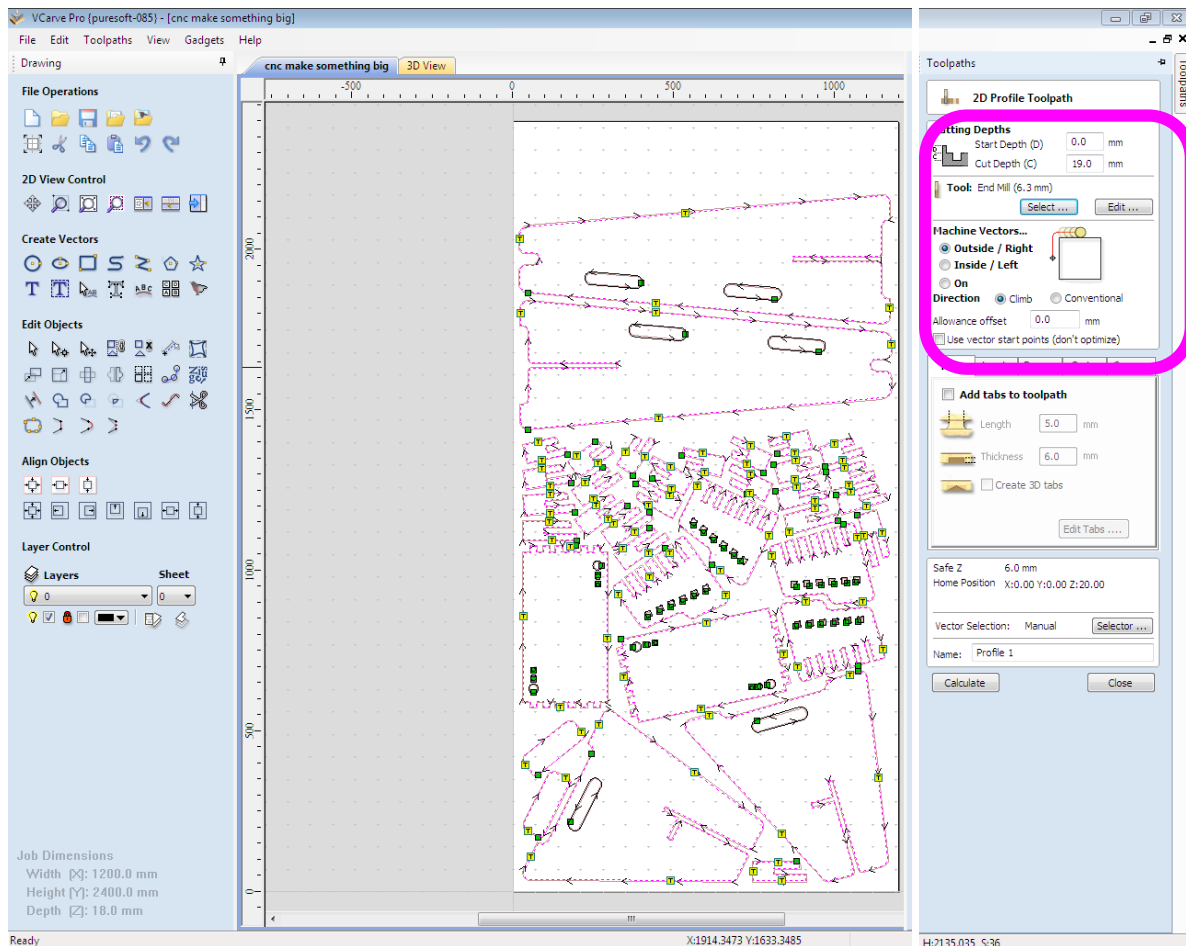


Once you've selected calculate you'll get a simulated view of what the bit will mill, shown below.



) Set tool paths for outer shapes:

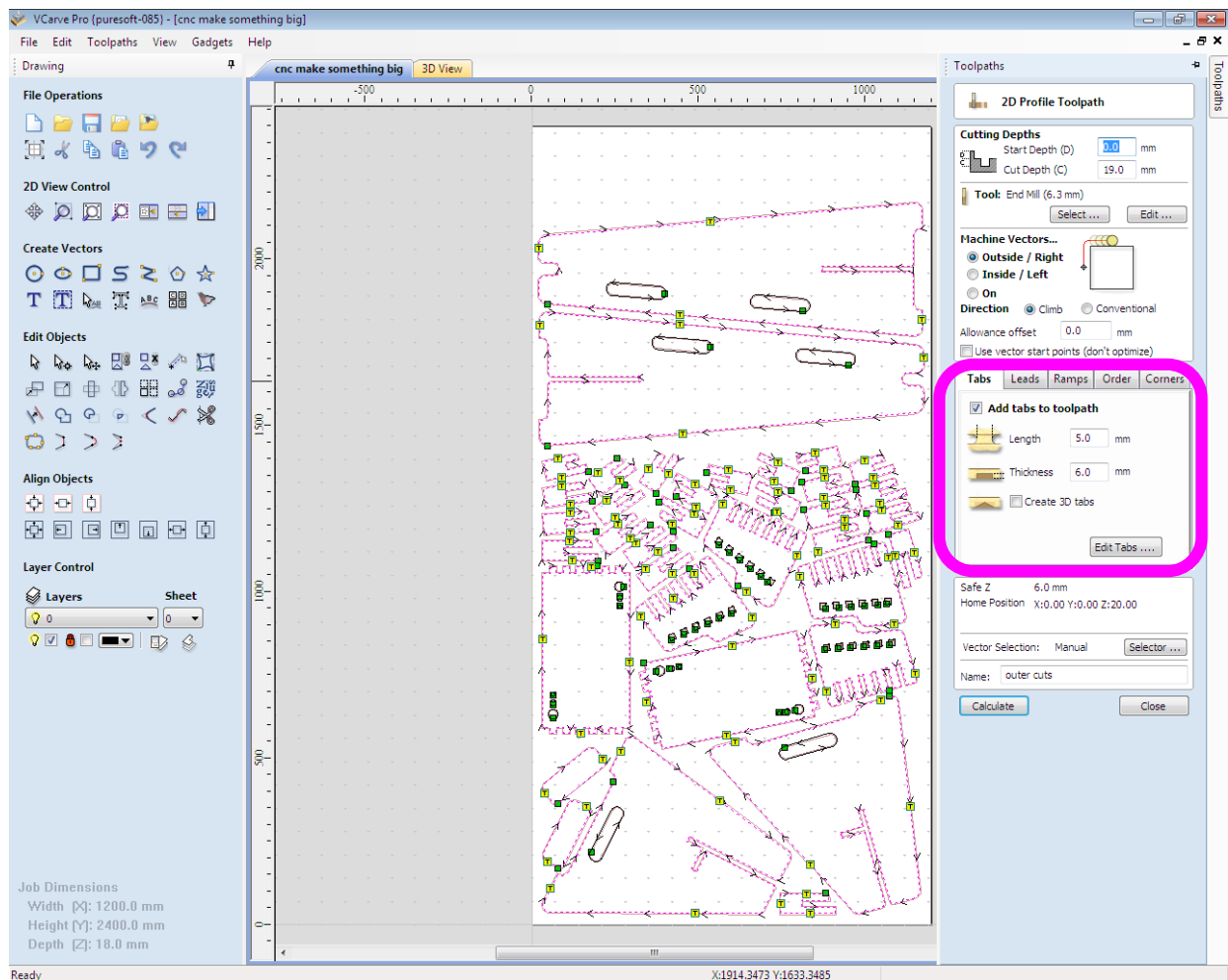
Now we're set the tool paths for the internal cuts we need to do the same for the exterior cuts (the one's that will actually cut out the final shapes). Cutting the outside of the shapes. Select a Profile cut. Make sure the Machine Vectors is set to Outside.



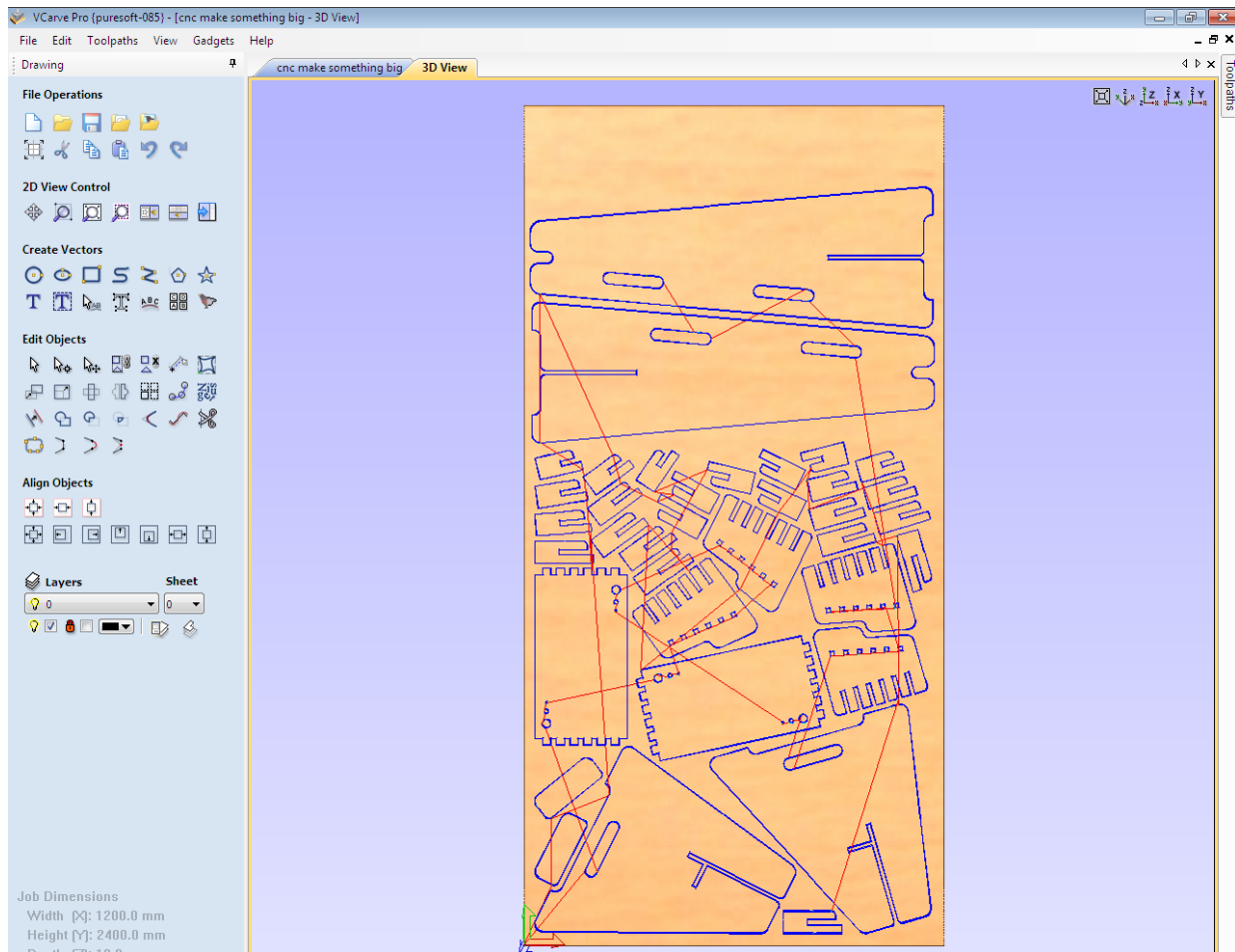
Make sure you select the same tool as was set before for the inner cut unless you want to use a different tool bit. I used the same one.

Because these are the external cuts we don't want to cut away all the material from the main board otherwise we'd have loose pieces of wood. So we add tabs that keep the internal pieces that are cut held to the main board. We do this in tool path settings for the outer mills, shown below. Afterwards you'll see these pop up all over the wood along the cuts as little 'T' signs. Adding tabs - if you think pieces are very close to each other, then align the tabs of the pieces that are close to each other so the tabs overlap and hold the pieces in place. The tabs will probably have to be sanded off so try not to put them anywhere that might be very difficult to sand them off. Select Edits tabs option and move the tabs to where you want them based on 1) holding the wood securely and 2) how easy it would be to sand off the tab from that area.

Once you've done this then calculate the tool path.



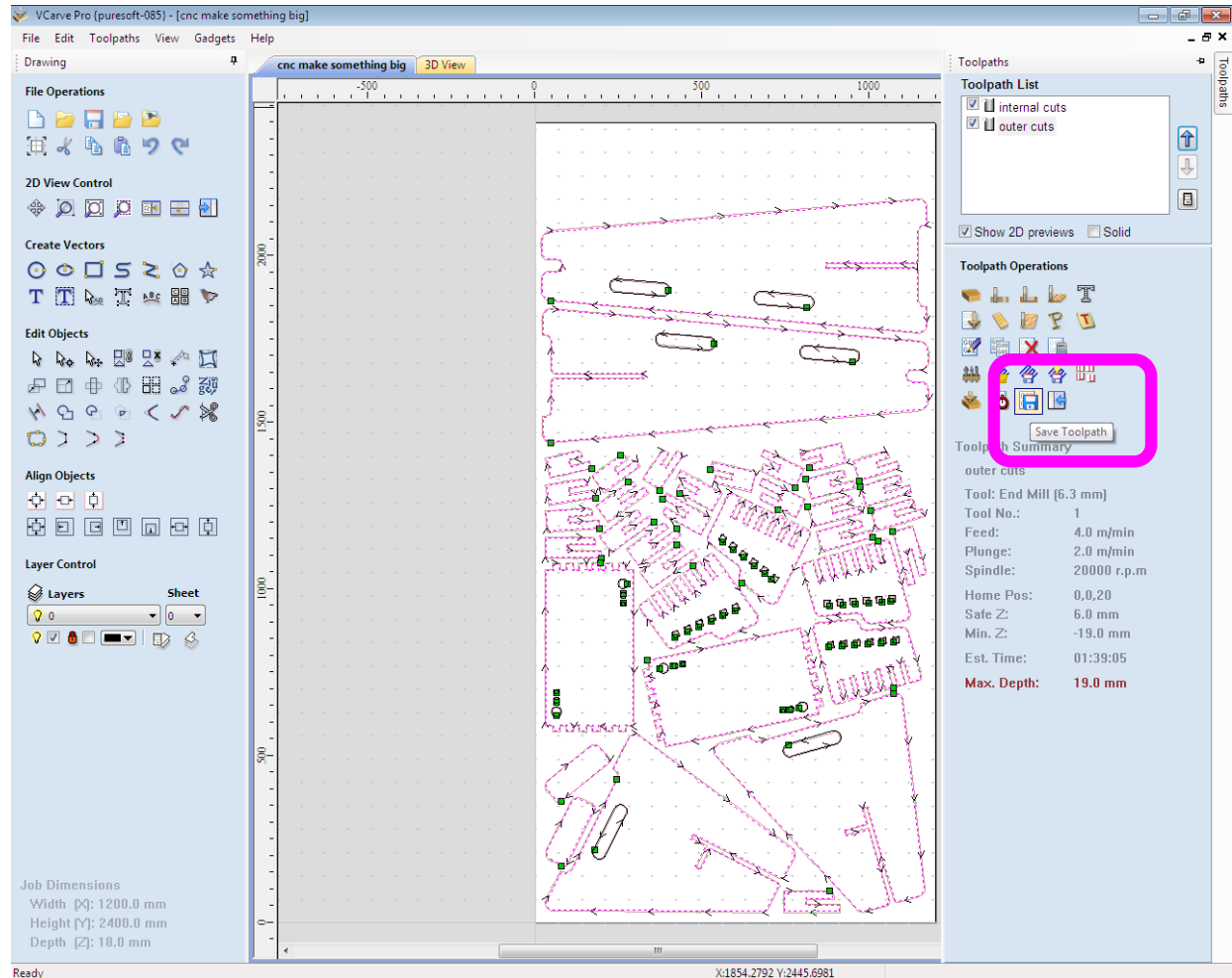
Again after calculating the tool path it'll show you what it is going to cut, shown below.



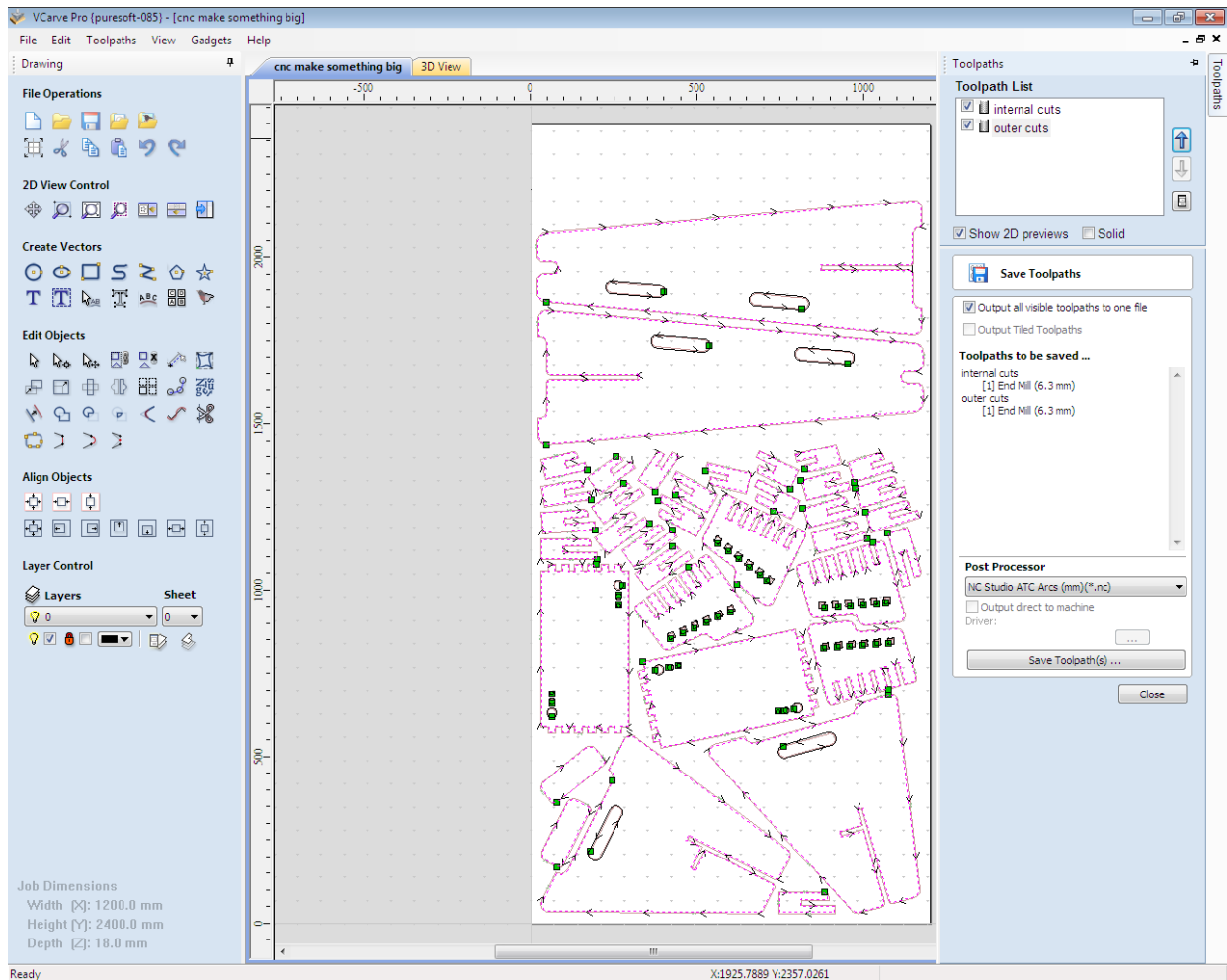
) Creating a file to send to the CNC machine

Once the pieces to be cut are laid out properly and the tool paths are set including the tool settings (speed, federate, size, plunge depth, etc.) then we need to output the file for the cnc machine.

In the toolpath bar on the right of vcarve. Select 'Save Toolpath' icon. Note, you should have 2 tool paths in the toolpath list. One for the internal cuts and one for outer cuts.



A window will open with a drop down menu called ' Post Processor'.



Select 'NC Studio ATC Arcs (mm)(*.nc)' from the drop down menu. This will be different for types, models and manufacturers of machines.

Once you've selected 'NC Studio ATC Arcs (mm)(*.nc)' then select 'Save Toolpath(s)' button. This generates gCode that talks to the machine to tell it instructions on what to route.

Next enter you're file name and save.

And it's ready for outputting to the cnc machine.

vCarve is the software we use to set up the layout of the routing relative to the billet (piece of material), set the feed rates, speeds, depth and tool bits. Then vCarve exports gCode which communicates with the CNC route. The software that communicates with the Router itself is different from vCarve. The one we used is called 'NcStudio AD, Engraving Machine Control System'.
