CabinetSense Wiki

CabinetSense Wiki

Toolpath Generation for Vectric Software

Introduction

CabinetSense now offers an easier way of creating Vectric toolpaths. Instead of manually creating toolpaths, assigning DXF layers and hard wiring the depth into the toolpath, you can have Vectric pick up the depth from the DXF and the toolpath strategy from the layer name.

We have written 3 Vectric gadgets to automate this process.

Important: This strategy can only be used with V10.0 (or later) of Vectric products. The gadgets will check the version number of the Vectric product that you are using and terminate if it is an older version.

The Gadgets

We have written 3 Vectric Gadgets to help you with toolpath generation saving you hours of effort and reducing (and potentially eliminating) the errors that can be made.

- **CabinetSense Tool Setup**: This gadget will generate base toolpaths for any given tool that you select from the tool database. For example, a ¼" end-mill might need toolpaths for cutting out your parts, pocketing and profiling (*inside* and *on* vector).

- You store these toolpaths in a toolpath template and name it specifically for the tool. You would create base toolpath template files for each tool that you use with your CNC. You end up mimicking how your machine works… every tool in your machine does a specific task(s). The toolpath templates that you create for each tool does the same thing.

- The next gadget will help you process an actual job.

- **CabinetSense Toolpath Checker**: This gadget will review the CabinetSense Layers in your job and ask you to select any missing tool template files. You can also create toolpaths for unique processes on the fly (EG. Generate a toolpath using a specific tool designed for the OVVO fastener).

- When the checker has completed its review, it will have generated all necessary toolpaths to handle all the layers in your job. This ensures that you have a complete set of toolpaths for generating your G-Code.

- Better yet, as your material thicknesses change, or construction depths change, these generated toolpaths will automatically pick up these changes eliminating the re-cuts caused by neglecting to change the manually assigned tool depths.

- **CabinetSense Apply Toolpaths:** This is a modified version of the *Vectric Apply Template to All Sheets* gadget. The difference being that this gadget can take the toolpaths that have been generated in the job and use them to apply to all sheets in the job. This saves you from having to save all the generated toolpaths into a job toolpath template file, reload the job and then use the Vectric version to process the templates.

CNC Options

The CNC Layers tab in the machining database has attributes that give you control over how your toolpaths are generated.

Add Hole Diameter when Diameter <=: This states the largest drill bit that you use with your CNC. Anything larger will require a pocketing type strategy to clear out the hole. CabinetSense will select the strategy

automatically based on this value.

Pocket Clearing Tool Widths: The tool widths that you use to clear out pockets. This is a comma delimited string containing the sizes of bits. You may want to enter nominal sizes rather than actual. For example, enter .25 rather than the actual sharpened size of the bit (eg. .24992). As long as your actual bit is not larger and only a tiny fraction smaller, everything will continue to work properly. CabinetSense will select the tool diameter for all inside profile and pocket strategies based on which tool diameter is the largest that can fit inside the pocket.

3D DXF Depth Strategy: this group defines how your tool path names will be formatted and when (and if) an inside profile strategy would be used instead of a pocket.

> ***Cam Package****:*
>
> *choose between "none" and Vectric.*
>
> > *Vectric layer names are generated as follows:*
> >
> > *[strategy]..[layer name|tool diameter|.[DT|#depth]*
> >
> > *where:*
> >
> > > *strategy is: Drilling, Pocket, In.Profile, Out.Profile, Line.Profile*
> > >
> > > *tool diameter is: d(12mm)*
> > >
> > > *depth is: #1, #2, #3 (generic) or #12mm, #13mm (using depths as values)*
> >
> > *examples:*
> >
> > > *Drilling..d(5mm).DT*
> > >
> > > *Drilling..d(0.25).#11mm*
> > >
> > > *Pocket..d(0.25).#2*
> > >
> > > *Line.Profile..OVVO 0930.#0.5*

Question: Why do we have different toolpaths for different depths?

Answer: Vectric only allows one depth to be associated with any toolpath. If you need to drill 10mm, 11mm and 12mm deep holes using a 5mm drill bit, Vectric requires 3 different toolpaths. Failing to use different toolpaths will result in this error from Vectric

> ***Pocket Strategy****: There are times where an inside profile can be used to clear out a pocket. Inside Profiles are generally more efficient than pockets, but quality of cut may suffer. Choose been efficiency and quality.*

Tool overlap %: if you have chosen to use an inside profile to do pocketing (where appropriate), enter the minimum % tool overlap required before the inside profile is used.

example: if you have a 10mm wide pocket and are using a 5mm end mill, the tool can theoretically be used to clear out the pocket using an inside profile. More likely, there will be a bit of material left in the middle that was missed. Specifying a 20% overlap would mean that 20% of the tool will be running over the same area that the opposite side did.

Drill Depths: choose between the maximum (non-drill through) depth for each different sized drill bit or all of the depths. The main reason why you might choose to use the maximum depth is that Vectric could then process all of them with the same toolpath resulting in a more efficient traversing of your sheet good.

example: if you have 3 different drilling depths (ie 10mm, 11mm, 12mm) for your 5mm drill bit, you can choose to drill the actual depths, or the maximum depth (12mm) for all of them.
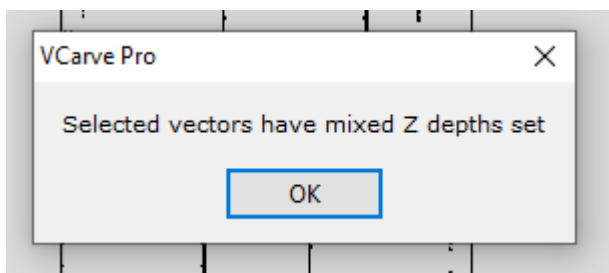
Depth Label: choose between a generic depth label and using the actual depth as the label. The reason to use one over the other is readability over toolpath template size. Using the generic label, will only result in more toolpaths being generated when your job has more different depths than previous, while using the actual depths will result in a toolpath being generated (and saved) for every different depth you ever use.
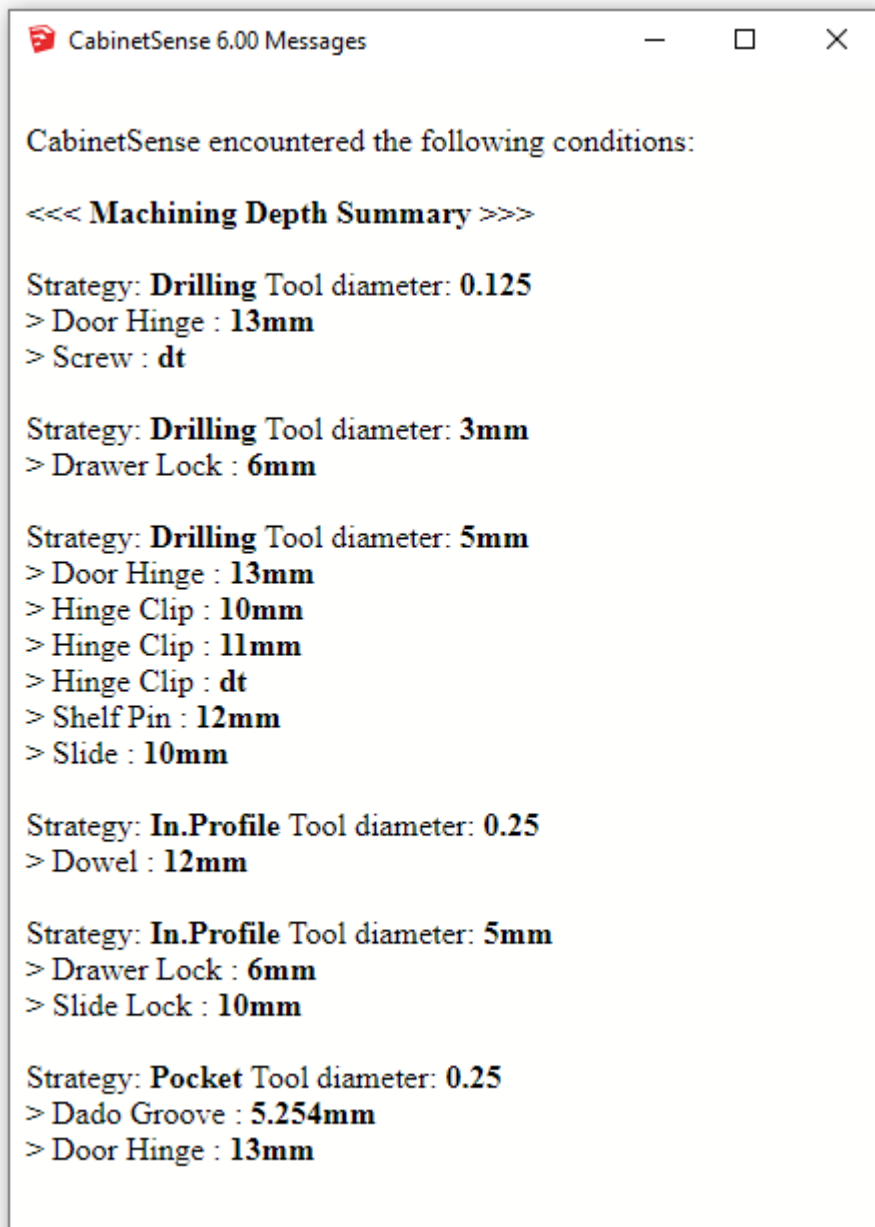
example: if you have 3 different depths (10mm, 11.2mm, 12mm) to be drilled using your 5mm drill bit, the formats for the depths would be #1, #2, #3 with the generic label and #10mm, #11.2mm, #12mm when using the actual depth. Remember that the actual depth is picked up through the DXF and the label has nothing to do with it.

Layer Report: CabinetSense can create a report showing you how your machining operations are being translated into the Vectric specific layer naming convention. This can help you review and revise your machining information before sending your job to the CNC.
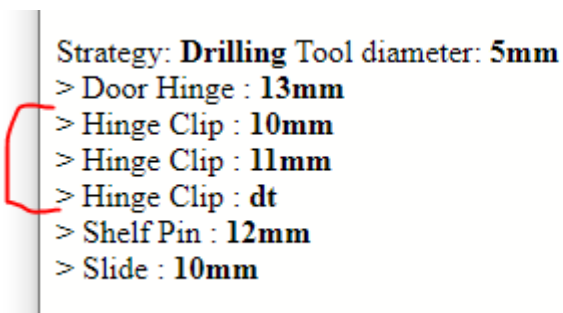
Layer Report Example

The report lists the type of machining operation and depth of cut required for each Vectric strategy and Tool diameter used. There is an on-screen report as well as an HTML version written to your export folder.

CabinetSense 6.00 Messages   — □ ✕

CabinetSense encountered the following conditions:

<<< Machining Depth Summary >>>

Strategy: **Drilling** Tool diameter: **0.125**
> Door Hinge : **13mm**
> Screw : **dt**

Strategy: **Drilling** Tool diameter: **3mm**
> Drawer Lock : **6mm**

Strategy: **Drilling** Tool diameter: **5mm**
> Door Hinge : **13mm**
> Hinge Clip : **10mm**
> Hinge Clip : **11mm**
> Hinge Clip : **dt**
> Shelf Pin : **12mm**
> Slide : **10mm**

Strategy: **In.Profile** Tool diameter: **0.25**
> Dowel : **12mm**

Strategy: **In.Profile** Tool diameter: **5mm**
> Drawer Lock : **6mm**
> Slide Lock : **10mm**

Strategy: **Pocket** Tool diameter: **0.25**
> Dado Groove : **5.254mm**
> Door Hinge : **13mm**

Explanation of Items listed in the above report:

- There are three hinge clip holes being drilled at different depths. This might be a definition problem in the machining database and could need cleaning up.

Strategy: **Drilling** Tool diameter: **5mm**
> Door Hinge : **13mm**
> Hinge Clip : **10mm**
> Hinge Clip : **11mm**
> Hinge Clip : **dt**
> Shelf Pin : **12mm**
> Slide : **10mm**

- You will be using an inside profile strategy with a ¼" tool bit to machine your dowels. Dowels are defined as 8mm in diameter and drilled to a depth of 12mm.
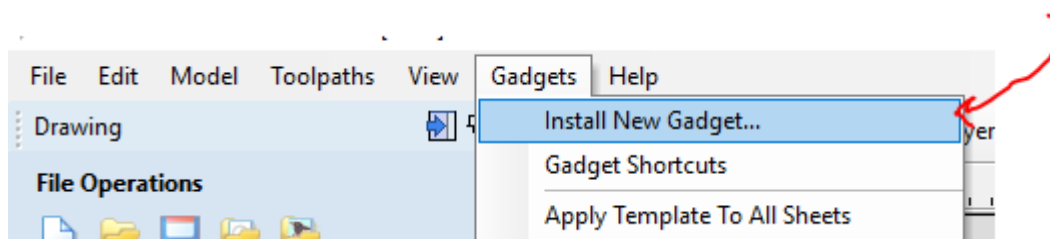

- You will be using a pocket strategy using a ¼" tool bit to machine your Door Hinge (most likely for an 8mm dowel) as well as a Dado Groove.
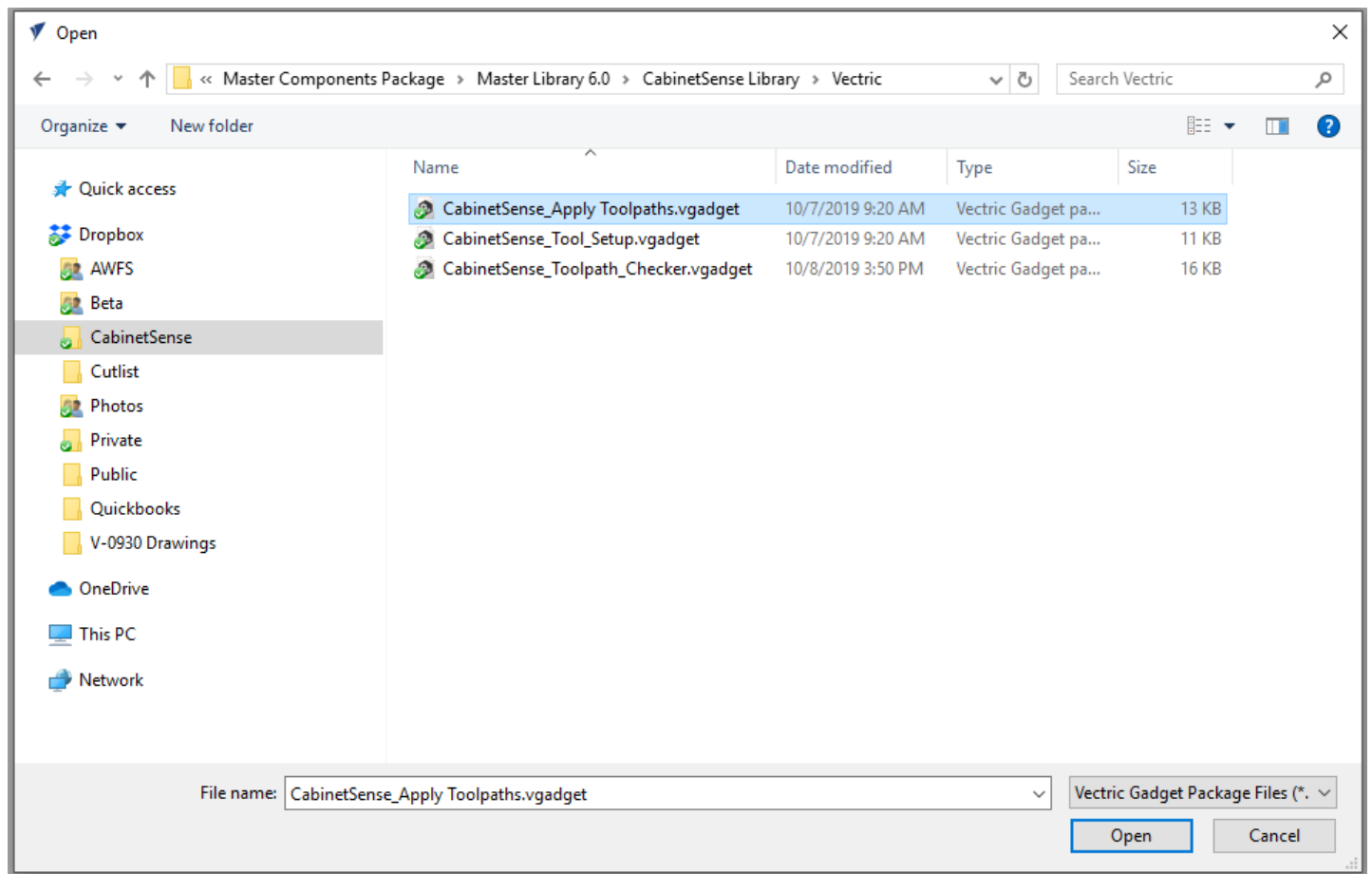

Installing the Gadgets

Open your Vectric software and choose the Gadgets>Install New Gadget Menu option.
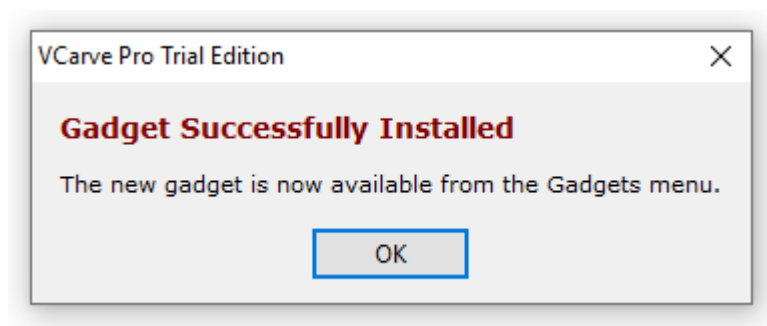
Navigate to your CabinetSense Library and open the Vectric folder.

NOTE: you may have to download the latest library from the CabinetSense website.

Select one of the gadgets and click Open.

Install all 3 gadgets.

Using the Tool Setup Gadget

Use this gadget to create toolpath strategies for each tool that you use with your CNC.

Tool_Setup   G CabinetSense

NOTE: This gadget was expressly written for the CabinetSense 3D DXF depth strategy and may not work effectively in other situations.

Create a tool strategy template for each of the common tools used with your CNC. Combine multiple tool templates together to form a coherent strategy for processing your sheet goods.

Vectors File:	C:\Users\paul\Dropbox\CabinetSense\Master Components Package\Master Library 6.0 beta\CabinetSense Library\Vectric\Toolpath Vectors Imperial.crv	**Select Vectors...**
Tool:	Amana Downcut (0.25)	**Tool...**

Strategies used with this tool:
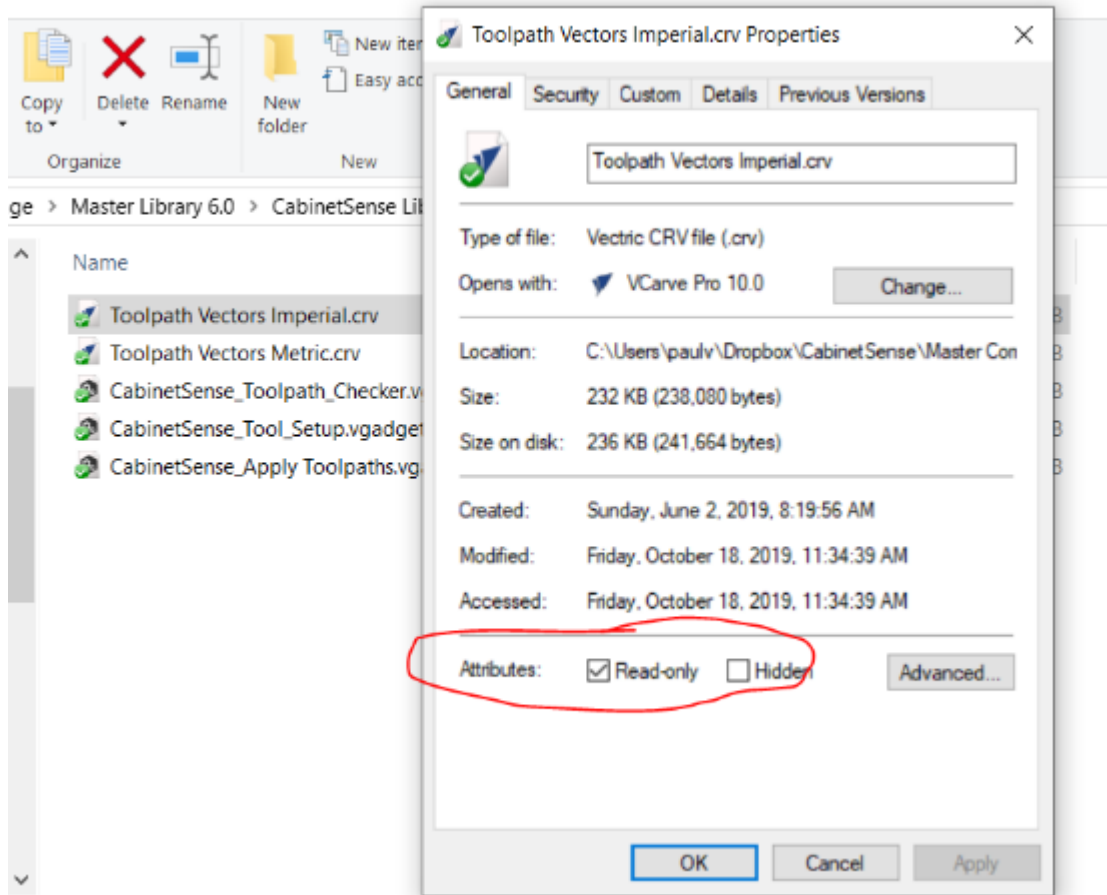- ☐ Drill
- ☑ Inside Profile, Line Profile and Pocket
- ☑ Fast Cut
- ☑ Slow Cut

☐ add Toolpath Note   Amana

OK   **Cancel**   Version 1.03.0

Select Vectors…: There are two vector files included in your CabinetSense Library. One is for clients using imperial measure while the other is for metric. Your selected file will be recalled whenever you run the gadget.

Note: These Vector files should have their file property set to Read-Only as it is only to be used as a basis for creating new toolpath templates and should never be over-written.

**Tool...**: Select a tool from your tool database.

Strategies used for this tool: There are four straegies to choose from. Select the one(s) that are appropriate for the the tool selected.

> *Drill*: If the tool is a drill bit, this will be the only strategy that you need to select.

- *Inside Profile, Line Profile, Pockets*: If you have selected an end-mill type of tool, you most likely want to generate toolpaths to handle these types of strategies.

- *Fast Cut:* This is to handle the cutting out of your parts. This name infers that you can cutout the piece at normal speeds. Only select this strategy for the tool that you want to use to cut out your parts.

- *Slow Cut*: This is to handle the cutting out of your small parts. You might use an onion skin and/or slow down your cutting speed.

Add Toolpath Note: You may have tools from different manufactures. You can use the toolpath note to add an identifier to your toolpaths so that you can differentiate tools. For example, differentiate an Amana ¼" end-mill from a Royce ¼" end-mill.

Generating the toolpaths: After making your selections, press the OK button to generate the toolpaths.
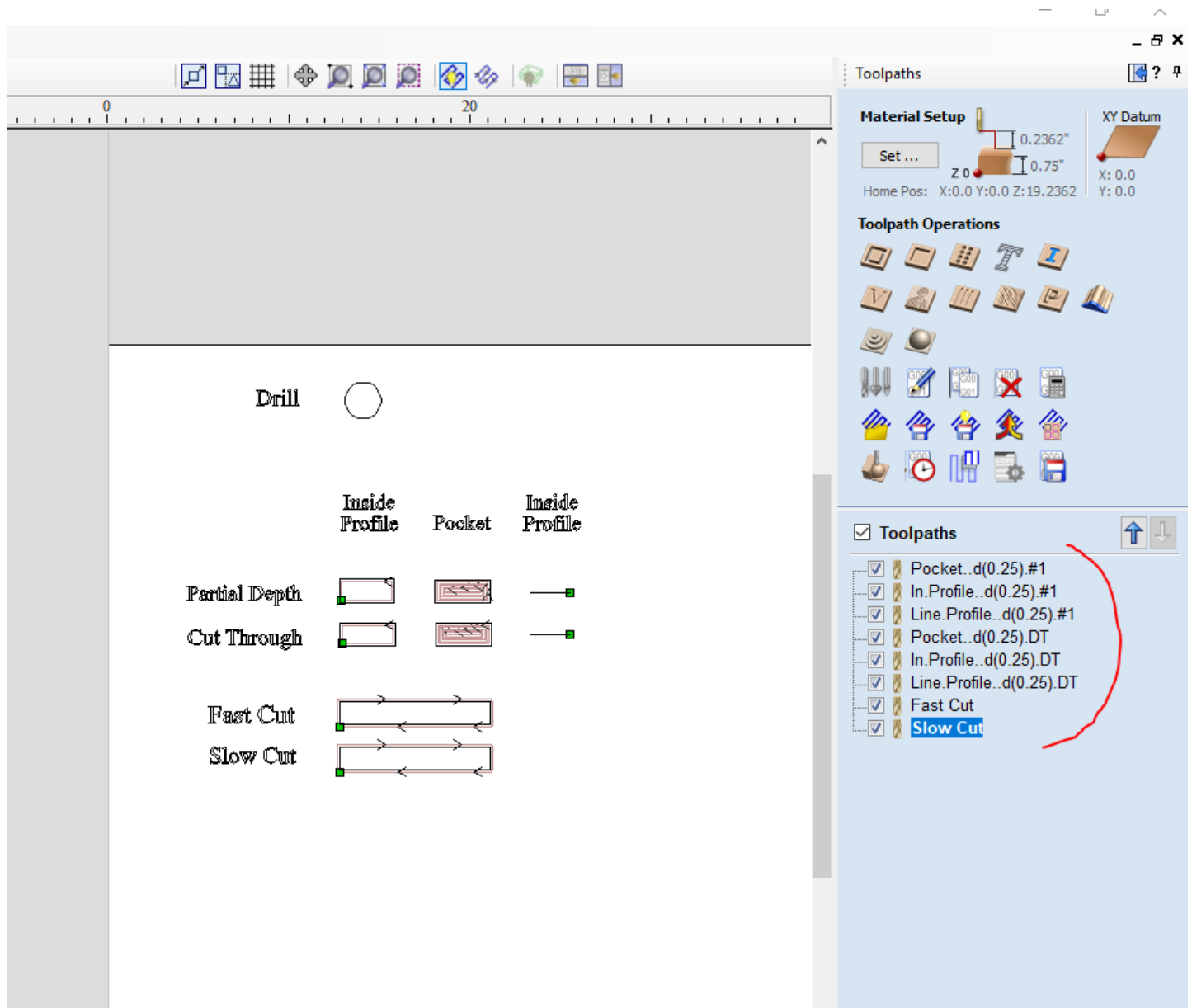
The generated toolpaths are very basic. You would need to edit each of them to add any additional processing criteria (such as ramps, offsets, dwells...).

Once you have edited the toolpaths, save them to a template file. These will be used in the next CabinetSense gadget.
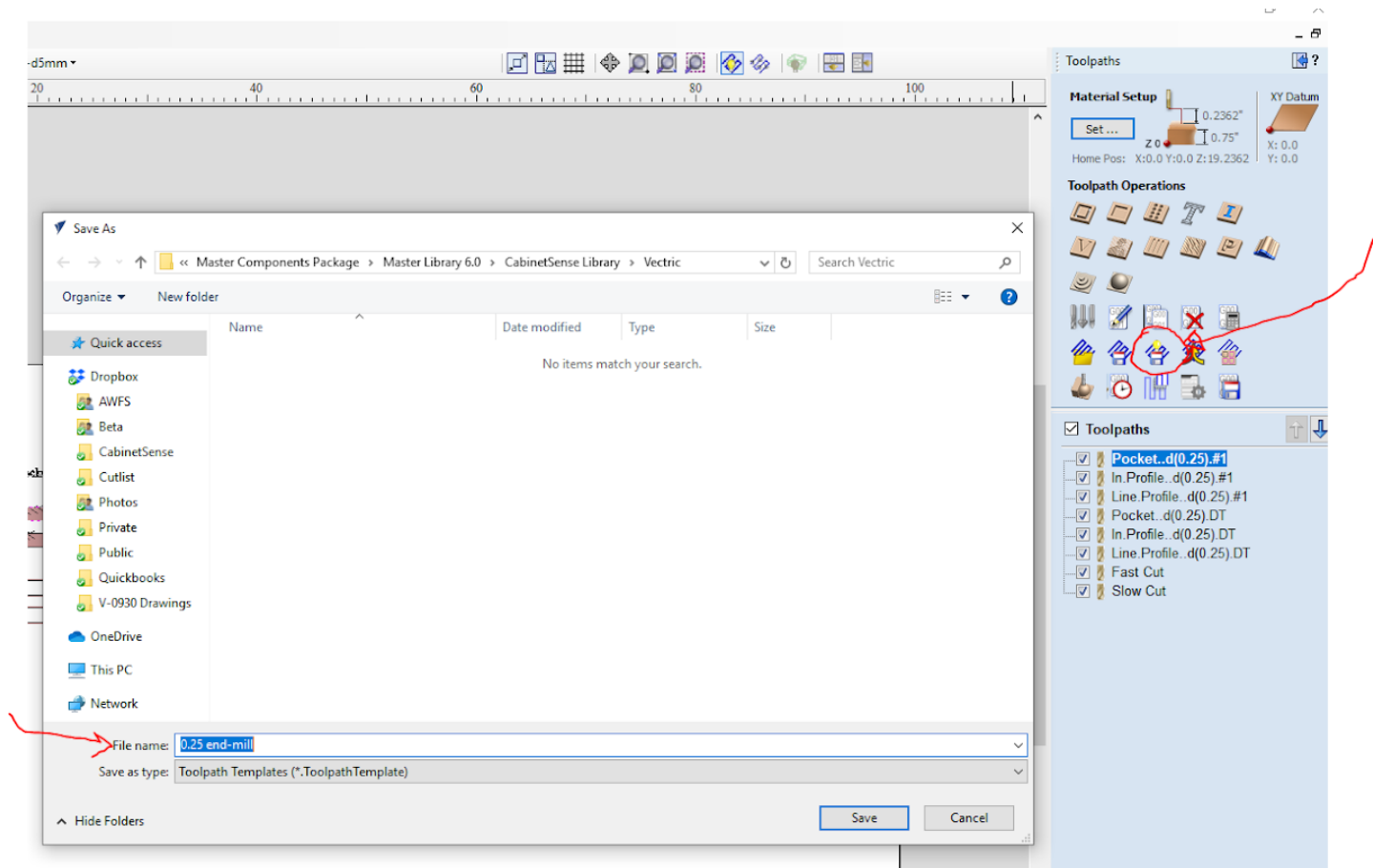
Preparing your toolpaths for your first job

When you have finished declaring all your tools, you now have a folder of several toolpath templates, one for each tool that you had setup. You can use these toolpath files in the next gadget (CabinetSense Toolpath Checker) to create your toolpaths needed for an actual job. This might require you to select several of the tool files (depending on which strategies are required for the job).

You might consider creating a CNC toolpath file that is an amalgamation of these individual tool files thus mimicking your CNC setup. That way, the next gadget would only require this one file to check and generate the actual toolpath definitions for your job.
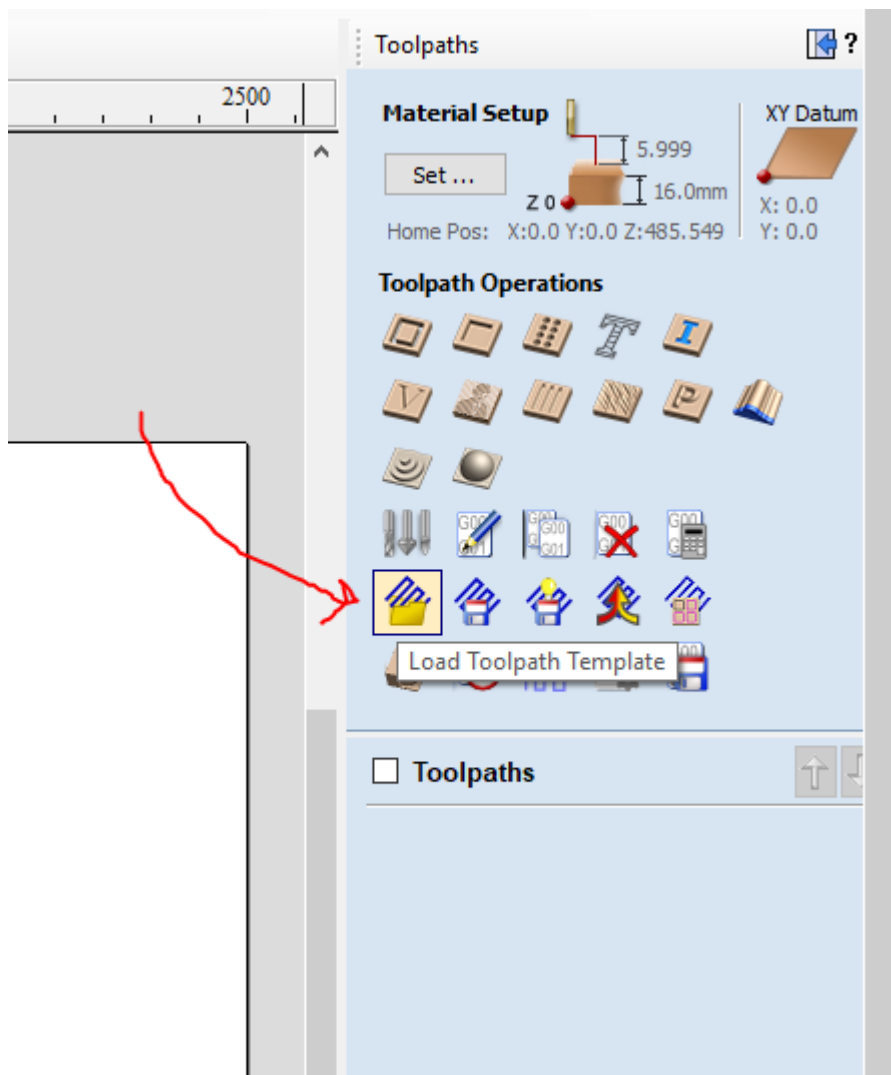
To create a composite toolpath file, start with a new Vectric job and then open each of the toolpath templates (created in the previous step) that you want to use.
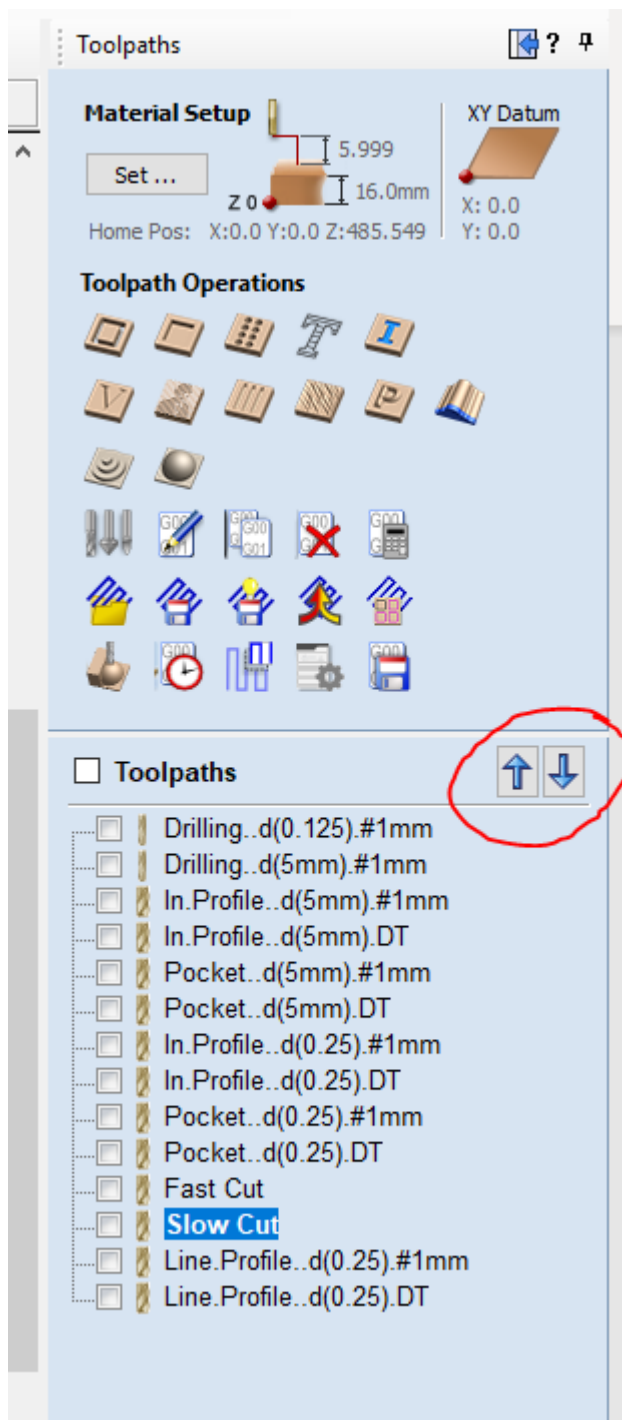
After loading all of the tool templates, re-sequence the toolpaths in the order that you want them processed in your job.
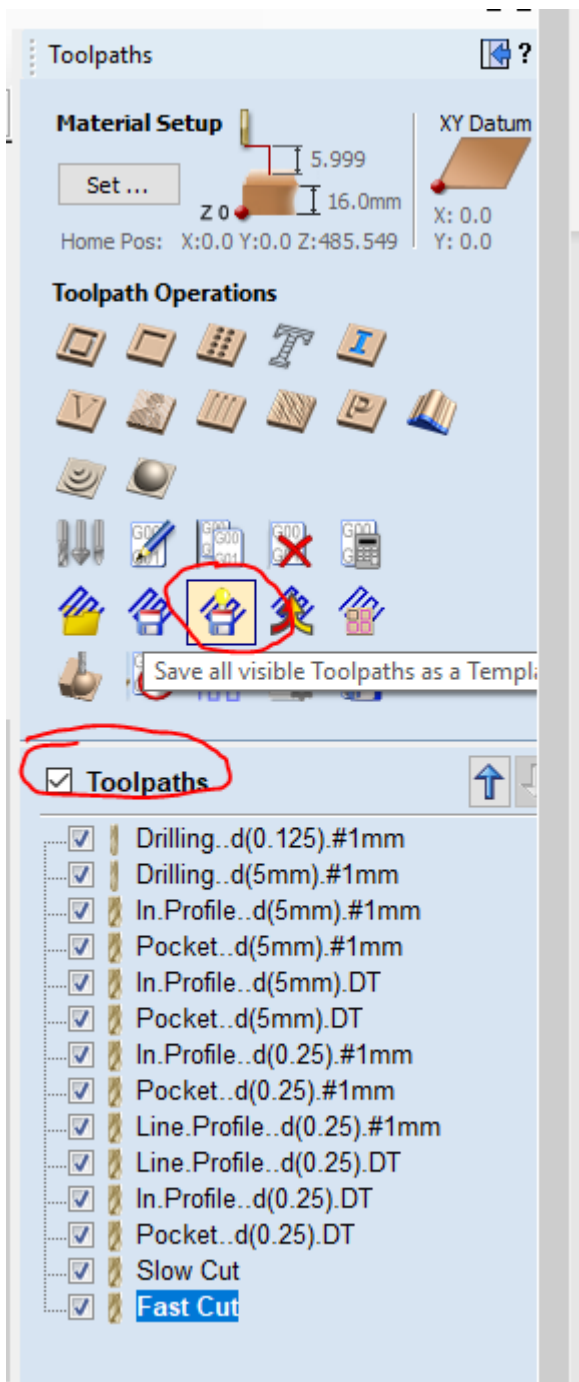
Finally, save the templates into a new toolpath template file.

Using the Toolpath Checker Gadget

Open the DXF files generated by CabinetSense by either using the Vectric Import function or DXF Batch Processor Gadget (available from the Vectric website).
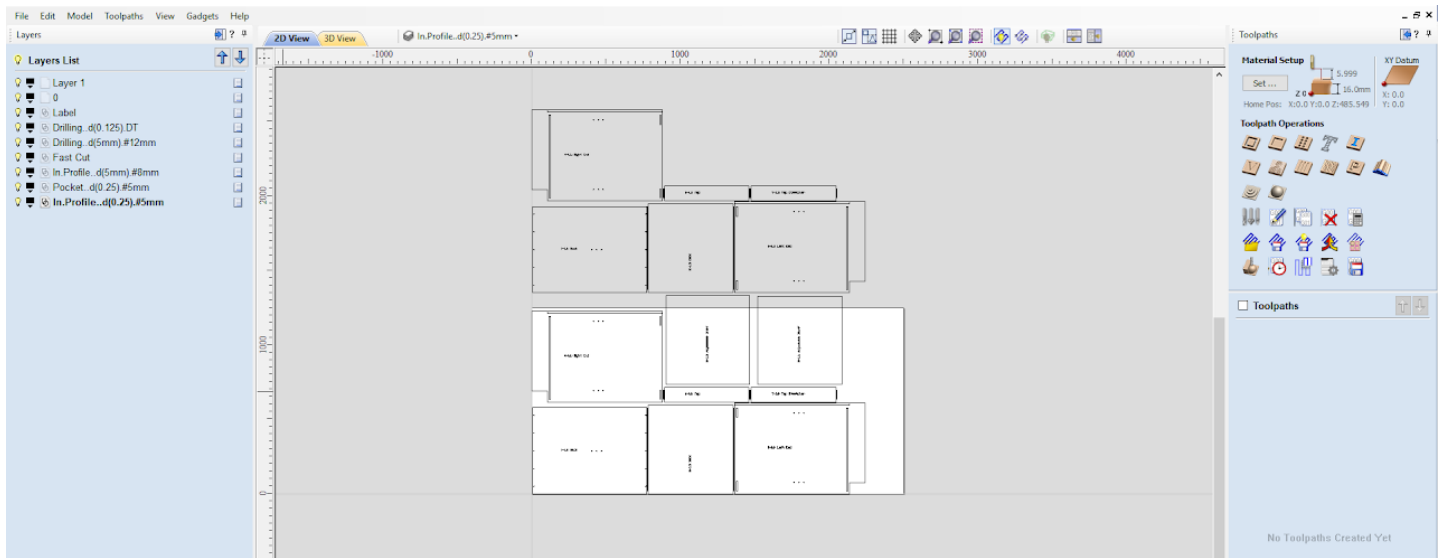
What this gadget will do: CabinetSense will use the toolpaths created with the Tool Setup gadget, match them to your job layers and create the required toolpaths for them. It will insert toolpaths immediately after the one it matched ensuring that your toolpath order is maintained.

Without any toolpaths defined, run the CabinetSense gadget.
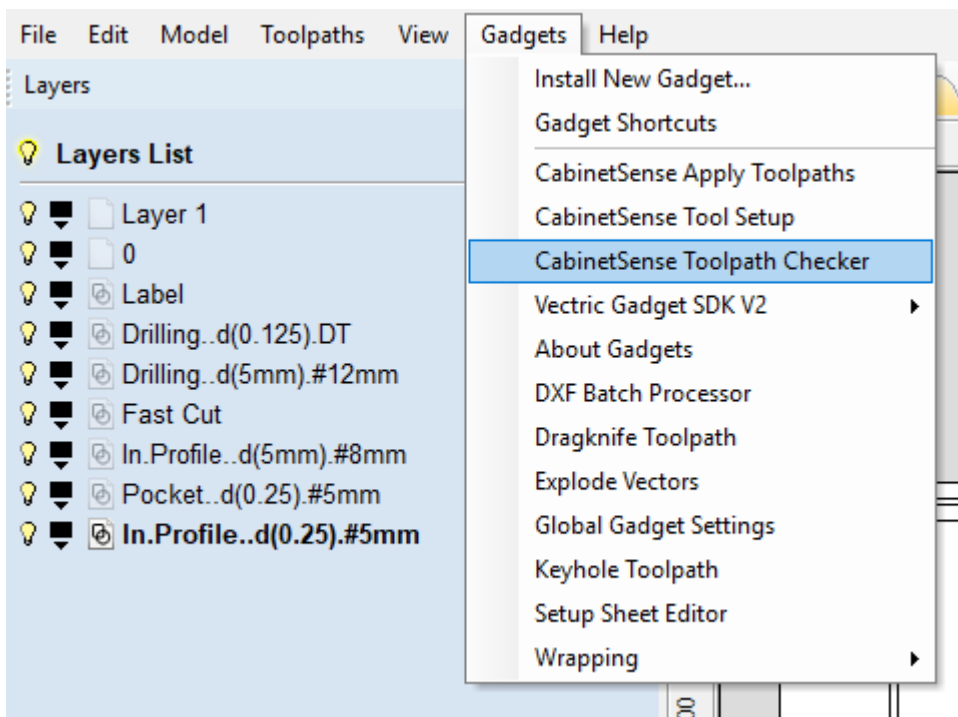
If you are using the CabinetSense Optimization method of Front-Back Layers, you may see this dialog asking you to select the sheet face to process.

The gadget will check each layer in your job against the toolpaths (where there are none at this time). For each layer that has no associated toolpath, you will be asked to select the tool to use.

You can choose a tool from your tool database OR you can select a toolpath template that you previously defined that can satisfy the request.
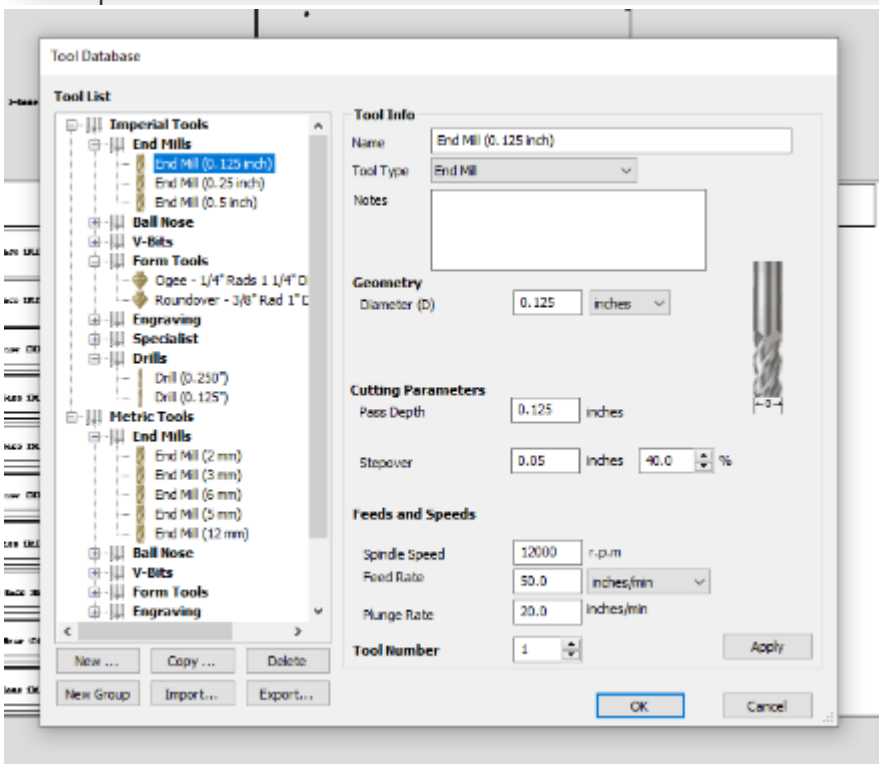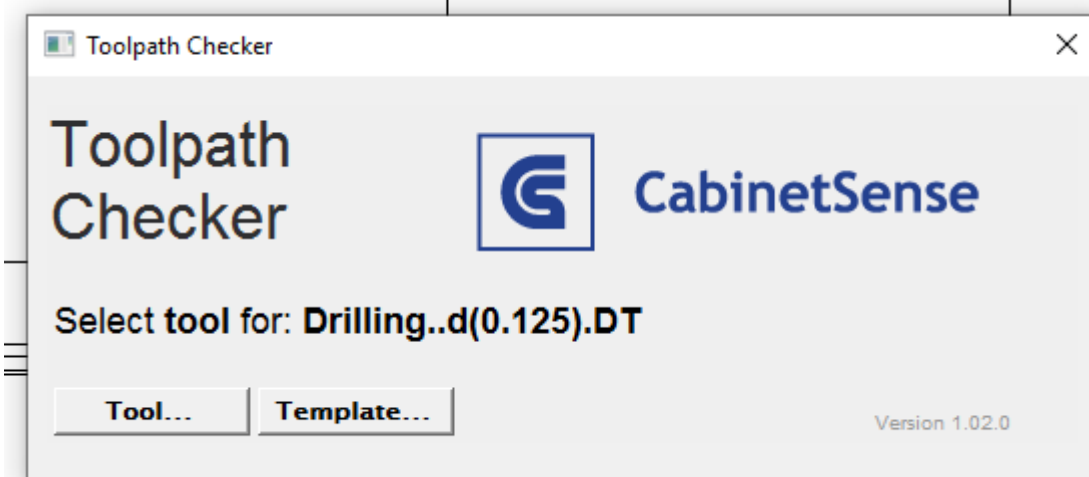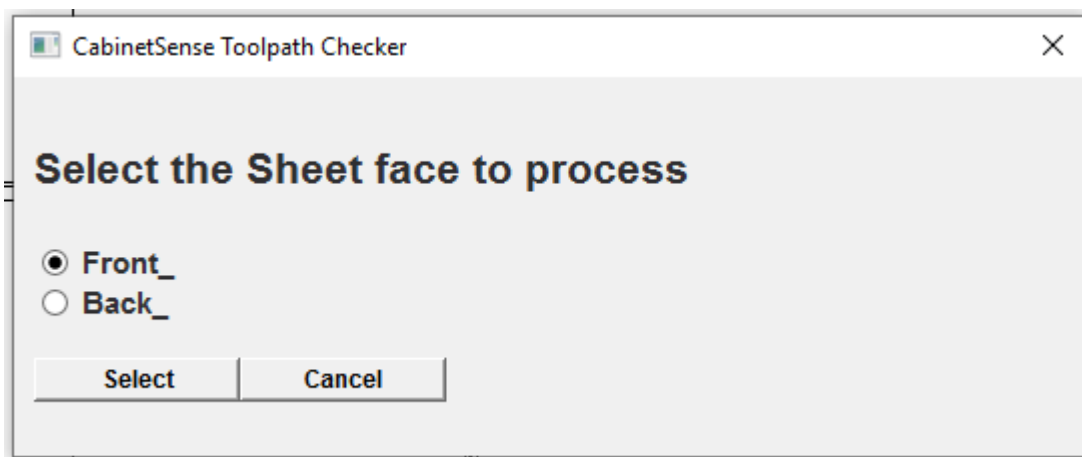
Click the **Tool...** button and you will be presented with your tool database.

In the image above, you are being asked for a drilling tool with a 1/8" diameter (0.125). Select the tool from your database that can handle this request.

Click the **Template...** button to open the toolpath selection dialog. Naviage to where you have stored your tool templates.

Select the one that is appropriate for the request being made.
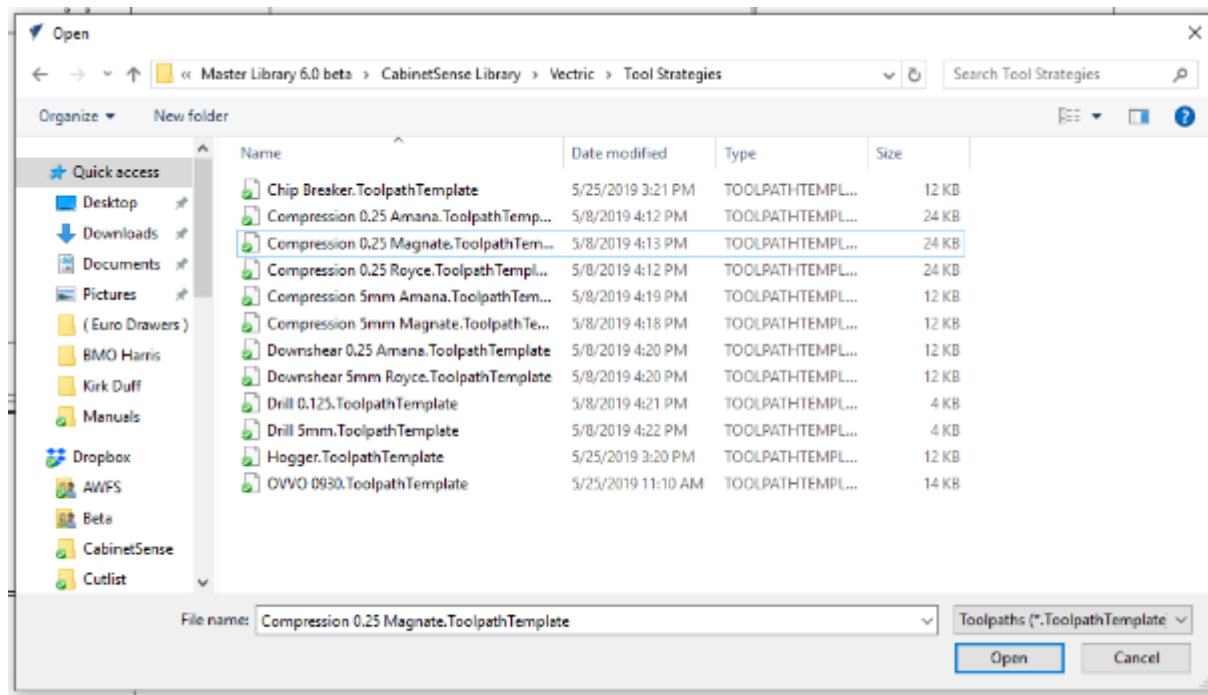
If you forget what tool you are being asked to select, the information is also displayed in the bottom left hand corner of the Vectric screen.
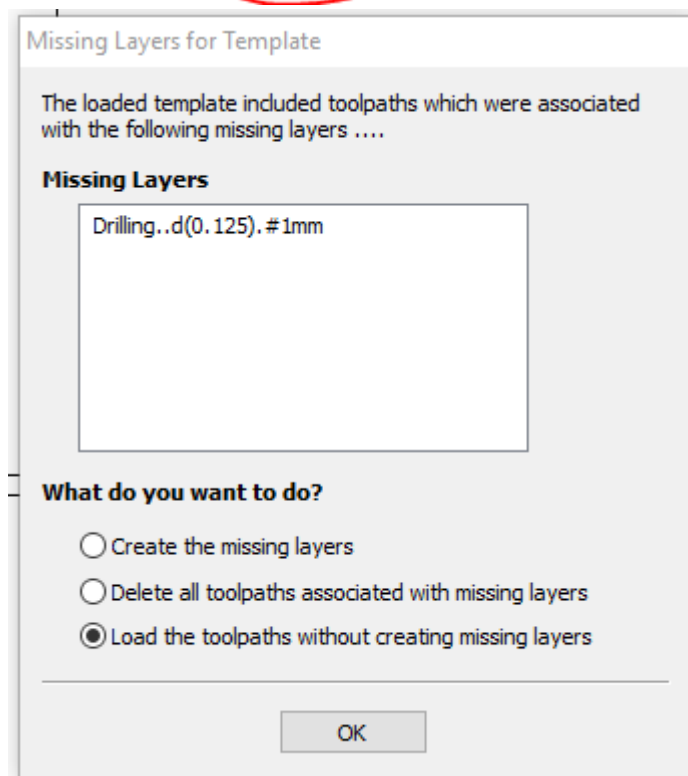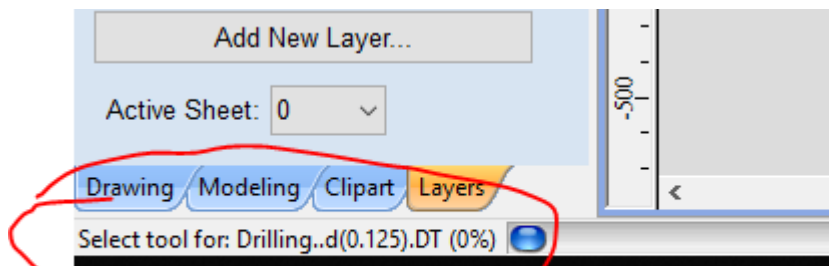
If you had selected a toolpath template, you may receive the following message. The layer defined in the toolpath does not exist in your job's list of layers.

CabinetSense will use this toolpath layer to generate a proper layer for your job.

You can set the option to **Load the toolpaths without creating missing layers**.

After selecting the tool, the gadget will create the toolpath specific to your job.

Go through each request for tooling until the gadget confirms that all toolpaths have been created.
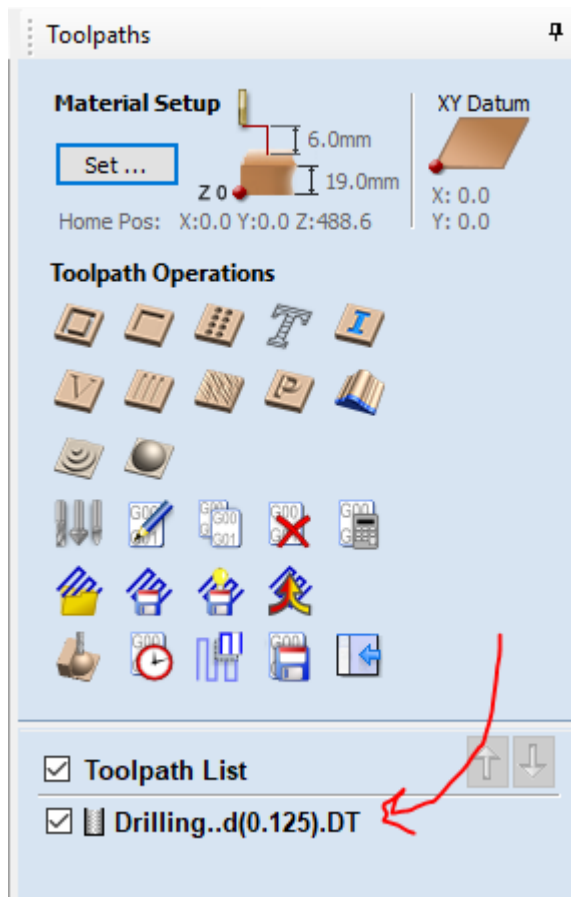
When the gadget finishes, you will receive one of the following messages.
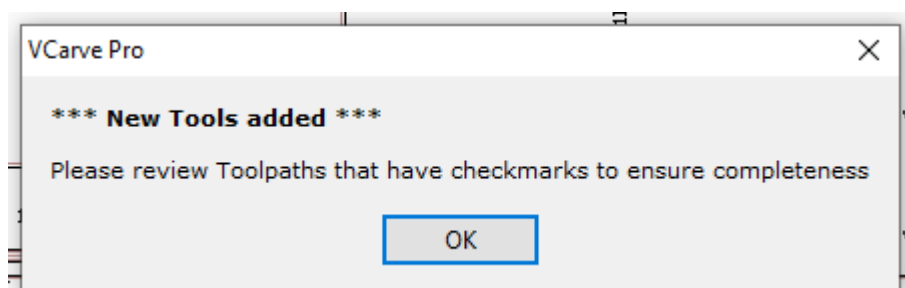
When this message is given, CabinetSense has added some default toolpaths for missing tools. These are basic templates and are missing any tweaks that you might want to do (such as ramps, dwells…). Edit the toolpath(s) and add the any optimizations that you wish to make for the toolpath. All new toolpaths will have a checkmark beside the toolpath.

CabinetSense has added a specified number of new toolpaths to your job. In this case, they are duplicates of toolpaths already in your toolpaths… the only difference being the depth to machine. Your existing toolpaths should have already been optimized, so there is nothing for you to review for these new ones.
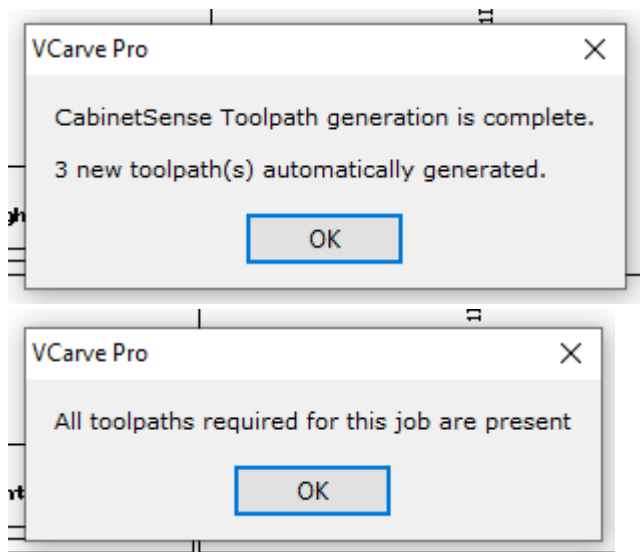
No new toolpaths were generated and your toolpaths are complete for the job being processed.

If needed, reorder your toolpath and/or edit any generated toolpaths as required.

Using the CabinetSense Apply Toolpaths Gadget

CabinetSense Apply Toolpaths

This gadget will apply the selected toolpath template to every sheet in the current job and optionally save the created toolpaths for each sheet in the specified folder using the selected post processor.

The toolpaths for each sheet will be prefixed by "Sn-" where 'n' is the number of the sheet for the toolpaths.
e.g If the Template has a toolpath called "Cut Out" the associated toolpaths for each sheet will be ...
   "S1-Cut Out"
   "S2-Cut Out"
   etc.

If the "Save toolpaths after calculating" option is checked, after applying the selected template to all the sheets and calculating the toolpaths, the toolpaths will be automatically saved as well.
If the selected post-processor supports toolchanging, or all the toolpaths in the template use the same tool, the toolpaths for each sheet will be saved as Sheet1, Sheet2, Sheet3 etc using the suffix of the chosen post processor.

Template File: **Select Template ...**

☐ **Save toolpaths after calculating**

Post Processor Name: | Mach2/3 ATC Arcs (mm) (*.txt)   ▼ |

Output Folder: | C:\Users\paulv\Desktop\DXF | **Output Folder...**

OK   **Cancel**   Version 1.2

Sheet 1

Toolpaths

Material Setup   XY Datum
Set ...   5.999
            Z 0   16.0mm   X: 0.0
Home Pos: X:0.0 Y:0.0 Z:485.549   Y: 0.0

Toolpath Operations

■ Toolpaths

- ☐ Drilling..d(0.125).#1mm
- ☑ Drilling..d(0.125).DT
- ☐ Drilling..d(5mm).#1mm
- ☑ Drilling..d(5mm).#12mm
- ☐ In.Profile..d(5mm).#1mm
- ☑ In.Profile..d(5mm).#8mm
- ☐ Pocket..d(5mm).#1mm
- ☐ In.Profile..d(5mm).DT
- ☐ Pocket..d(5mm).DT
- ☐ In.Profile..d(0.25).#1mm
- ☑ In.Profile..d(0.25).#5mm
- ☐ Pocket..d(0.25).#1mm
- ☑ Pocket..d(0.25).#5mm
- ☐ Line.Profile..d(0.25).#1mm
- ☐ Line.Profile..d(0.25).DT
- ☐ In.Profile..d(0.25).DT
- ☐ Pocket..d(0.25).DT
- ☐ Slow Cut
- ☐ Fast Cut

When you have run the Toolpath Checker gadget, you may be with toolpaths that resemble something like this.

Notice that there are several toolpaths that are not checked. These toolpaths may or may not be used in the job being processed.
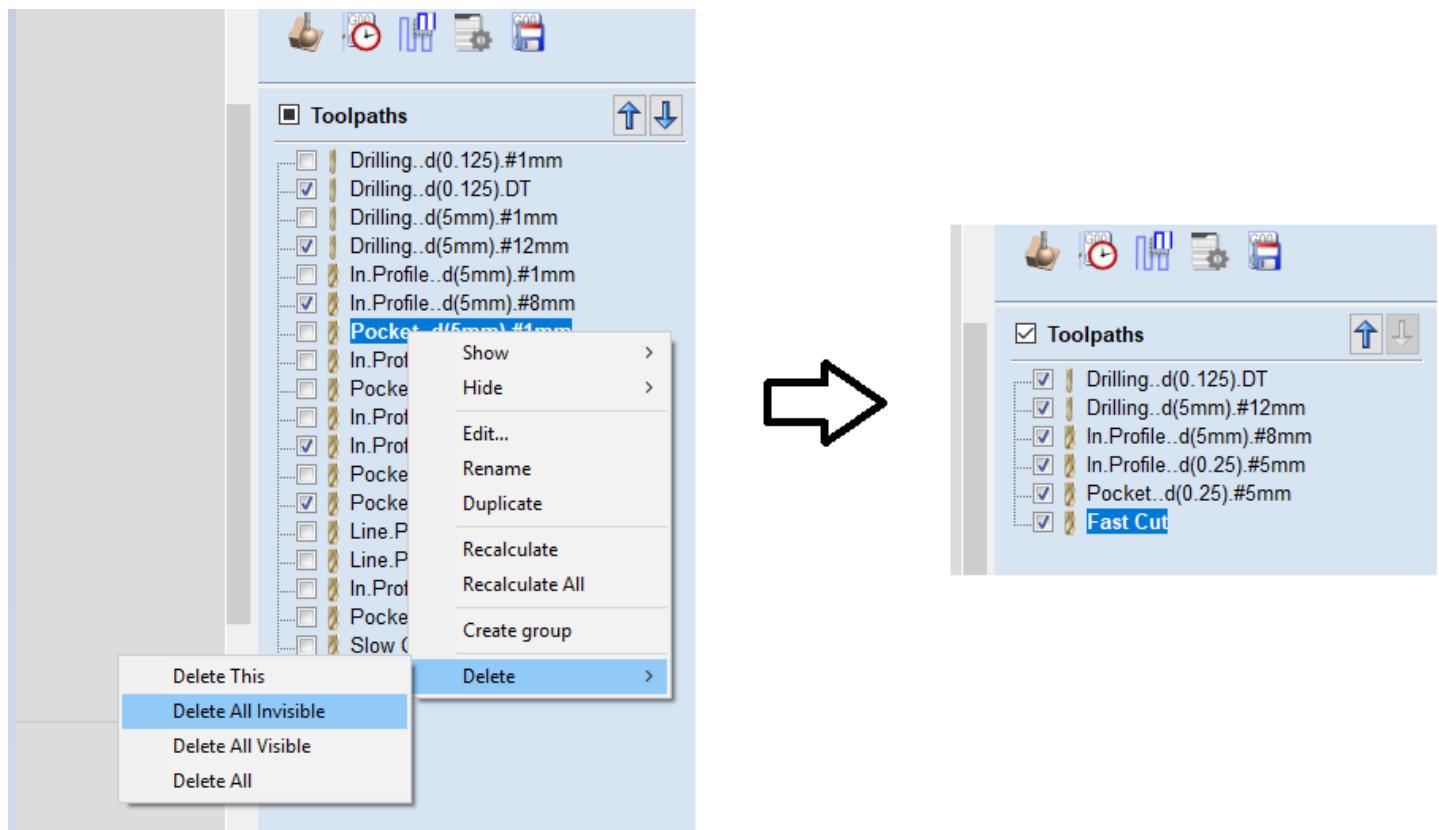
If you have a large number of parts in your job, you can speed up processing time by doing a bit of cleanup.

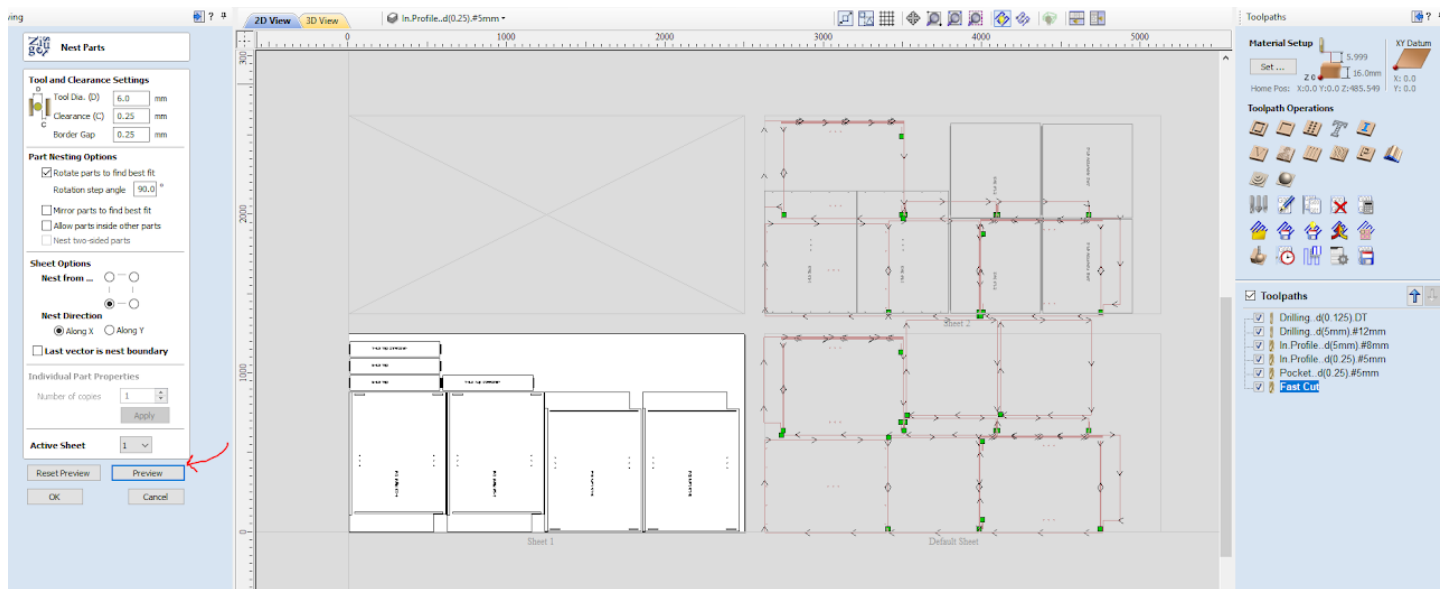Press the recalculate button so that all of the used toolpaths become checked.

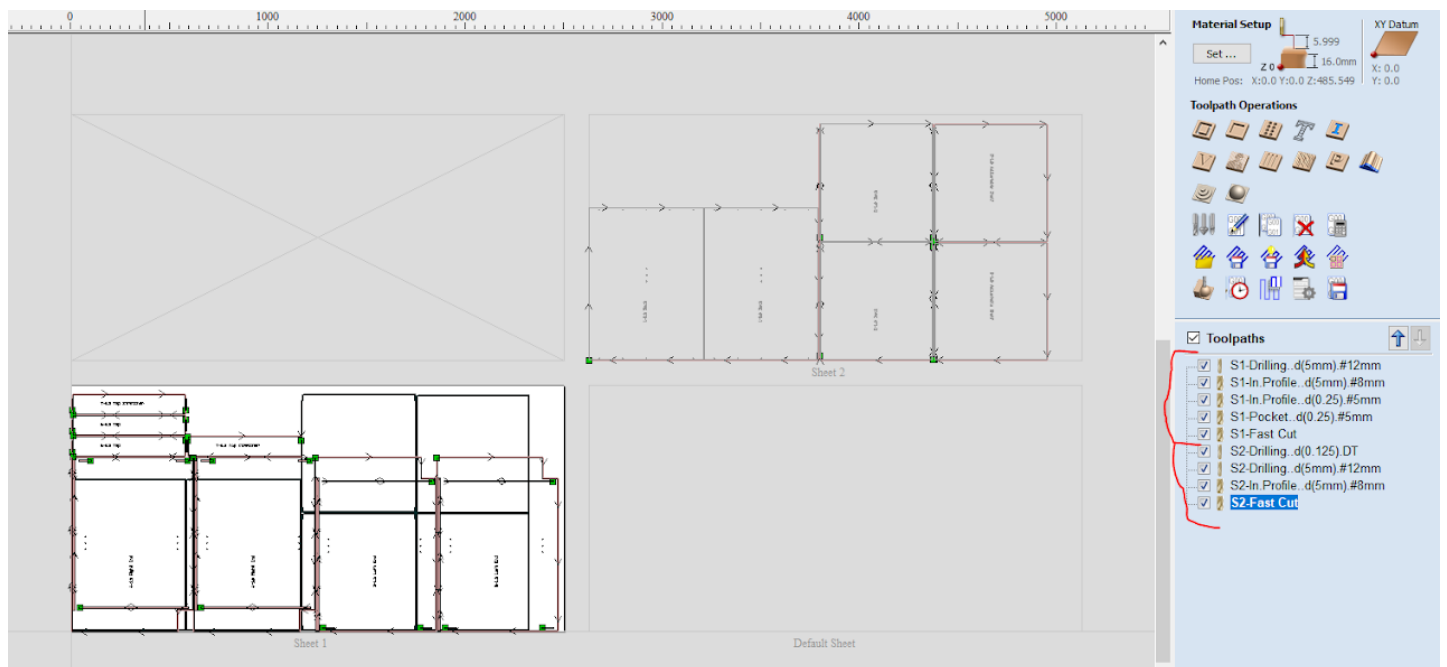Now you can delete all the unused toolpaths.

The smaller set of toolpaths will speed up the Apply toolpaths gadget.

Nest your parts...

and now run the CabinetSense Apply Toolpaths Gadget. This gadget will use the toolpaths listed in your job rather than opening a template file. Other than that, this gadget is identical to the Vectric version.

Generate your GCode and you're ready to process on your CNC.