I don't in any way claim to be an expert at CNC techniques and methods, but I have cut a lot of things out of a lot of sheets of plywood over the years and have developed a system that works well for me. It may not be the most efficient way to work… there are lots of resources that are focused on production and will help you become hyper-efficient. The main benefit is that it limits the things you have to think about, and works pretty well in just about every occasion.

I've toolpathed these Shelter files to cut fairly conservatively, both to make sure that people with CNC tools of varying capabilities can use them and to hopefully make it easier for anyone to have success when cutting them. You are free to make as many changes you would like, but I feel like these are a good starting point

## Bits:

I would rather not have to think about what kind of bit to use, or to have to change bits within a file if I can avoid it. So I try to pick one bit that will do everything I need it to do, and use the largest size of that type of bit that will cut all the features. So ALL these file can be cut with a ¼" bit, and my preference is a 2-flute straight bit. It leaves a reasonable edge and face quality, and can make shallow cuts that would be a problem with an up-cut or combination bit.

There are lots of other options though. An upspiral bit cuts cleanly, but tends to chip the top face veneer and lift small parts, even with vacuum holddown. A downspiral solves the chipping issue on the top face, and will also help hold parts down because of its geometry. Unfortunately that geometry also causes the clips to pack in the kerf, causing friction and heat which is the death of bits.

Combination bits solve some of these problems…they are downspiral at the top and upspiral at the bottom so gives a pretty clean cut. The upspiral section at the tip still causes problems with shallow pockets and you really have to cut at full depth all the time to get much utility out of them.

You can certainly also change bits depending on the feature, such as using an 1/8" bit to drill for the screw holes instead of having to hand drill afterwards. That's fine, but I don't want to have to hang around the tool waiting to change the bit .

## HoldDown and Cutting Strategy:

I have a simple Vacuum holddown system that I can't imagine doing without. If you're cutting features that are depth critical like 3d carvings or parts that have to interlock, then holding the material firmly to the table is critical. If however you cutting shapes out of sheets, holding DOWN is only a part of what it does for you. I feel like its most important job is to hold the parts AND THE REST OF THE SHEET in place. If the material moves a little in the Z-axis you might be alright, but if it moves in X or Y you are in trouble. These are some rules I try to follow:

- Even with a vacuum holddown I try to put clamps at the far corners of the sheets whenever I can. This helps keep the waste part of the sheet from “ooching” around as parts are cut out.

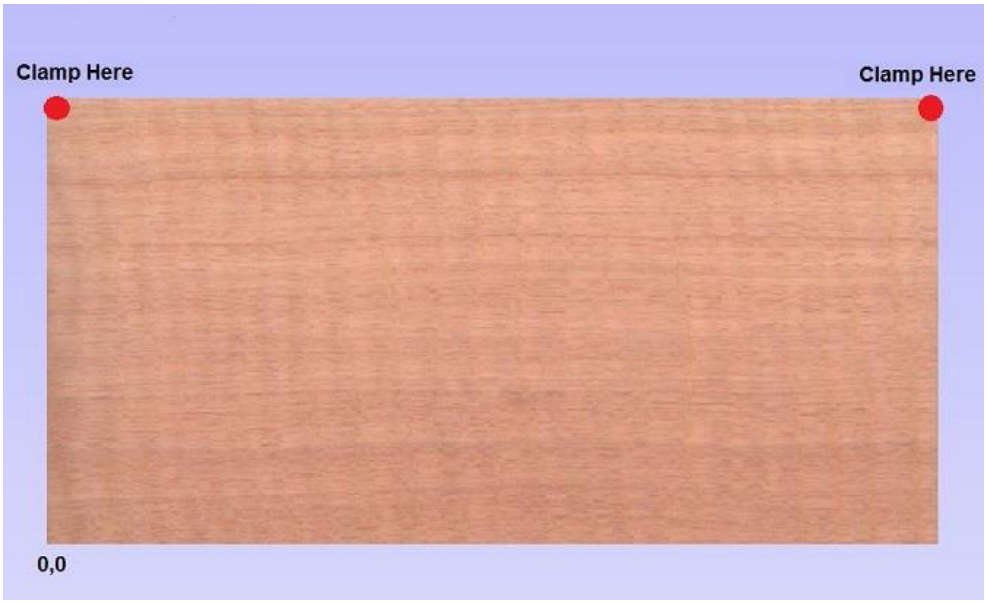


*Figure 1 Clamp at corners if possible*

*Figure 2 Clamped*

- I try to cut small parts first when the holddown is the best. More surface area makes larger parts easier to hold, so do them last.

- Tabs are helpful but can be a pain to remove. I generally only use them with smaller parts, and try to use one tab at the start point (in red circle below, which can be set in NODE EDIT mode).

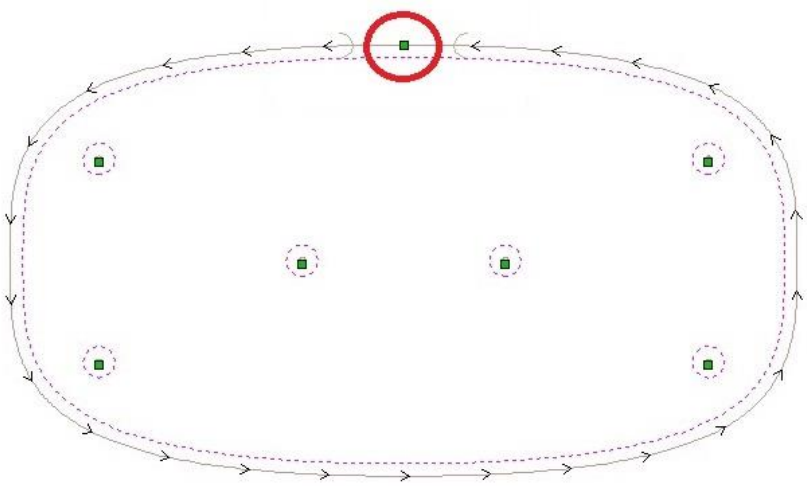

*Figure 3 Single tab at start point*

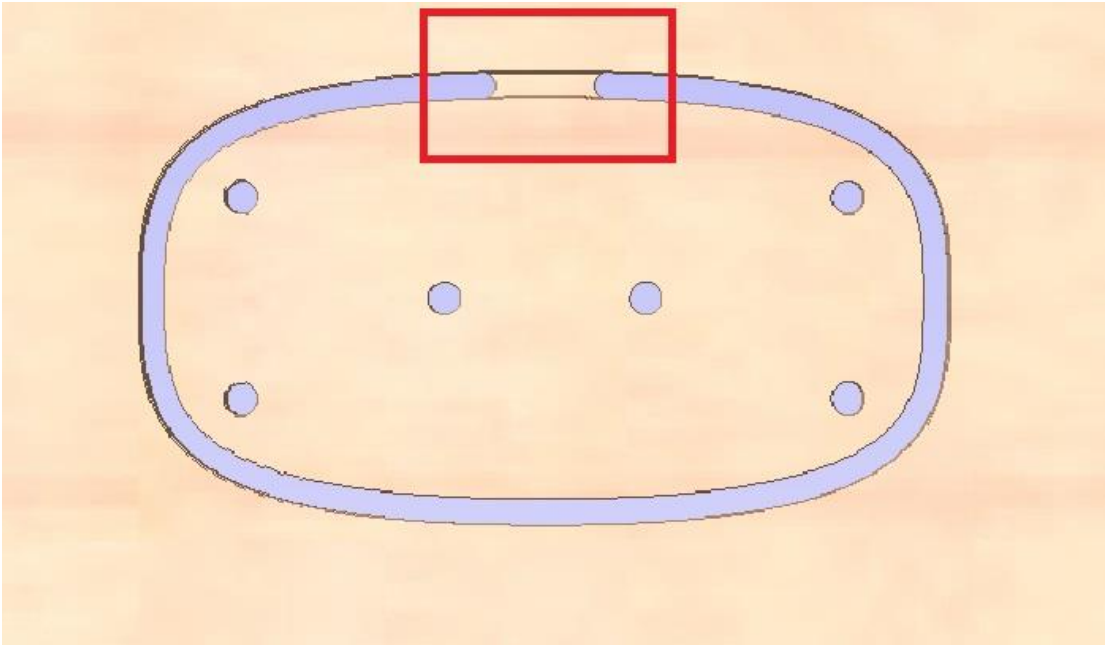

*Figure 4 Preview with tab*

- If you can place the tab so that it's in a section of grain that's parallel to the tab, it will be easier to remove. If you plan ahead, the tabbed area will be hidden and you won't even have to clean it up.

*Figure 5 Place tab in area that won't be seen*

- My exception to the one-tab-per-part rule is in long thin parts, like the door trim pieces. I tab the heck out of them because they will want to chatter, leaving a rough edge and probably breaking a bit

- I generally spend a little extra time picking the order of cut for my parts, and where the start point is for the cut. The goal is to have the parts connected to the sheet for as long as possible before finally cutting them free. Imagine slicing a loaf of bread…you want to hold the loaf at one end and start slicing at the other. As you slice, everything is firmly held until the last minute.

## Ramping:

I ramp into every part that I cut…it makes bits last longer and leaves a better edge. More importantly, since it's cutting a thinner and thinner bit of material as it gets to the end of the final pass, the cutting force is reduced and the part will better stay connected until cutting is finished. For most parts I use a Smooth ramp, varying between 5-8" long depending on how aggressively I'm cutting. With small parts however I use a Spiral ramp, which along with a tab at the start/end point gives me better control of the cut.

## Toolpathing:

The toolpaths are organized based on the way I would cut them. There are two "types" of toolpaths…those that start with "prelim" and those that start with a number.

The ones that start with "prelim" are eventually combined into a Merged Toolpath. Their names tell you what order that are to be cut in the merged file, and what toolpath they are merged into. For instance, toolpaths "***prelim3a-screw location markers"*** and "***prelim3b-cutout wall"*** are merged into toolpath "***3-cutout endwall"*** . Toolpaths that start with a number are meant be cut in that order. You would cut toolpath "***1-****"***, then "***2-****"*** etc.


*Figure 6 Toolpath list*

## Layers:

I've tried to use descriptive layer names to describe the entities contained in those entities. Entities in the "screwhole location marker" layer are meant to cut shallow dimples that indicate where to eventually drill pilot holes for screws. Wedges are on their own layers because of the way they are cut, and occasionally the holes in parts are on their own layers if they are to be cut using a different strategy that other parts, such as when using a spiral plunge.

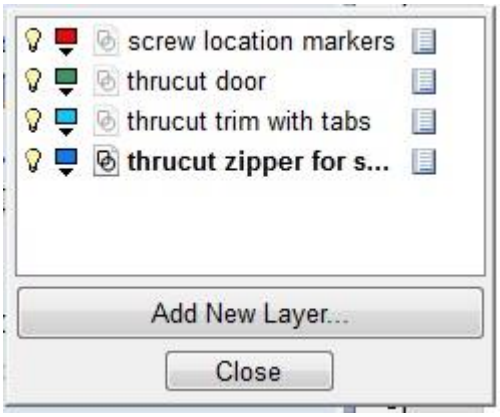


*Figure 7 Layers are your friend*

## Speeds:

I've toolpathed these files to cut at 4"/sec and 11,000 rpm spindle speed, and to cut ½" plywood in 2 passes and ¾" plywood in 3. When I cut these parts myself in my shop I cut ½" in 1 pass and ¾" in 2, so feel free to modify the settings in your tool database to suit your tool and cutting style