

# THINGS TO CONSIDER

## Stock Measurement Vs. Nominal Measurement

**Nominal Measurement** is what hardware/lumber yards use as the average across the stock. This makes up for tolerances through the manufacturing process.

**Stock Measurement** is the true measurement of the material you will be using. With laminate material this can change from brand to brand. This is very important when you want to machine 3D files.

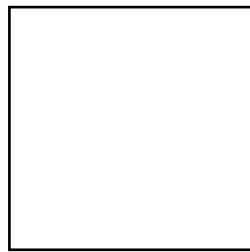
**Ex.** a 2"x4" piece of lumber is a nominal measurement. Its stock measurement is generally closer to 1.5" x 3.5"

## Internal Radii

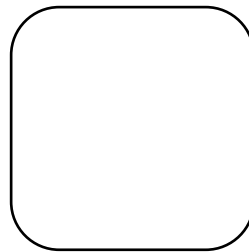
This is what an **INSIDE** corner will look like when a tool goes into the area to cut.

**Ex.** 1/2" cutting tool will leave a 1/4" radius on a corner.

Original Design

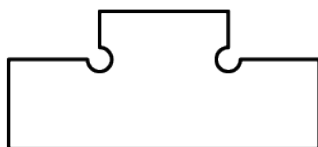
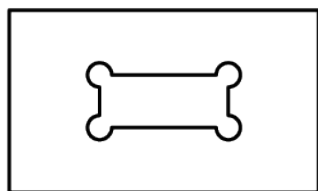


Internal Radii

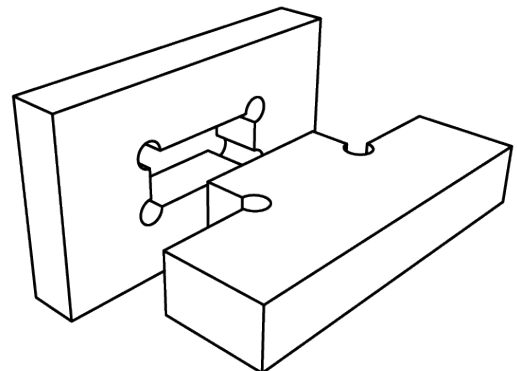


## Dog Bones

Dog bones is a technique to create relief for internal corners for slot fitting parts. They are created by drawing 2D vectors. They can be created by using the Aspire software (refer to vector packet) or drawn in your preferred software.



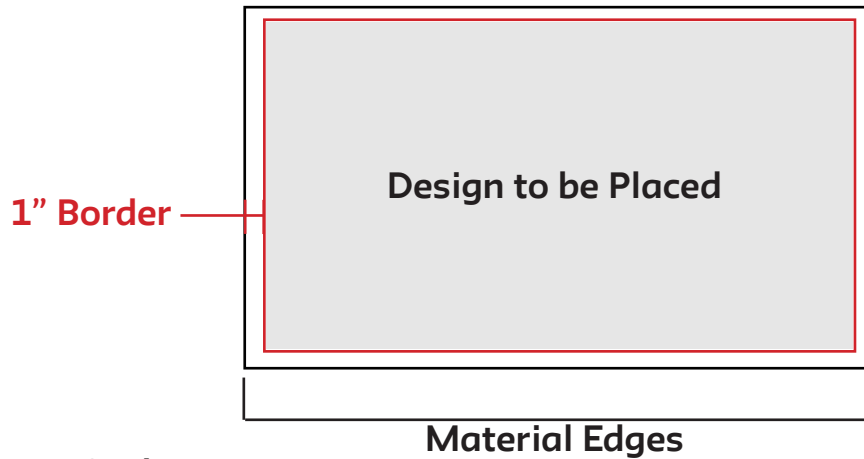
2D Vectors



Final Cutout

## Placement of Design

In order to secure material to the machine there **MUST** be the use of fasteners. Those might include screws, nails, tape, jigs or several of these methods combined. With that said, there has to be a border along the edge of any material that does **NOT** include any design components. This border shall be no less than **1"** set in from material edge.



**\*Not Drawn to Scale\***

## Order of Operations

When creating your cut file in Aspire you have to consider order of operations. The order of toolpaths created is very important.

**The rule of thumb: start with the smallest or least intrusive cuts first then work outwards or most intrusive cuts last.**

Small profile cuts or pockets can be cut first. 3D surfacing toolpaths can run next (if applies to your design).

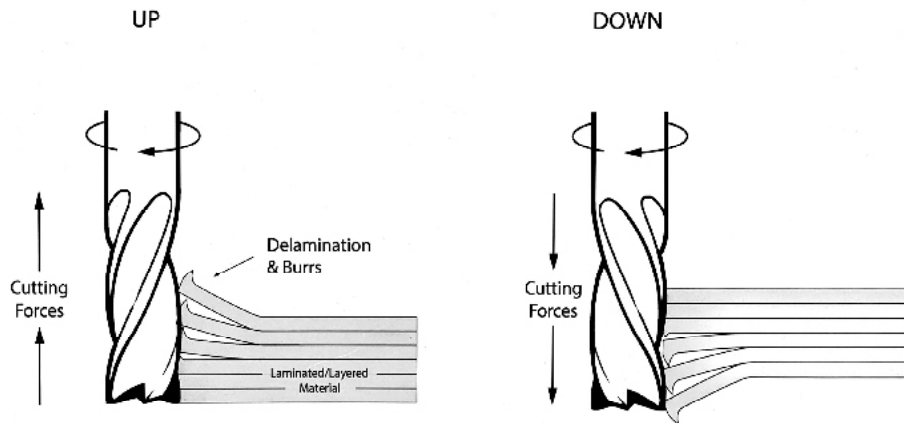
Large profiles should always run last (we want the bulk of material connected for as long as possible before cut into).

## Machining Plywood

When machining plywood a **Down Spiral** tool must first be used to protect the surface quality.

This only needs to be machined to break the first layer of veneer “0.008” is a good depth to start with it. This does slow down the process.

If surface quality is not important you can just use an **Up Spiral** tool for the whole cut, but it will chip the plywood.



## Climb Vs. Conventional

**Conventional** takes a less aggressive chip from the material but leaves a fuzzier edge. We recommend using this for the bulk of the cutting. This preserves the tooling and is easier on the machine.

**Climb** takes a more aggressive chip from the material but leaves a cleaner edge & can lead to tool breakage.

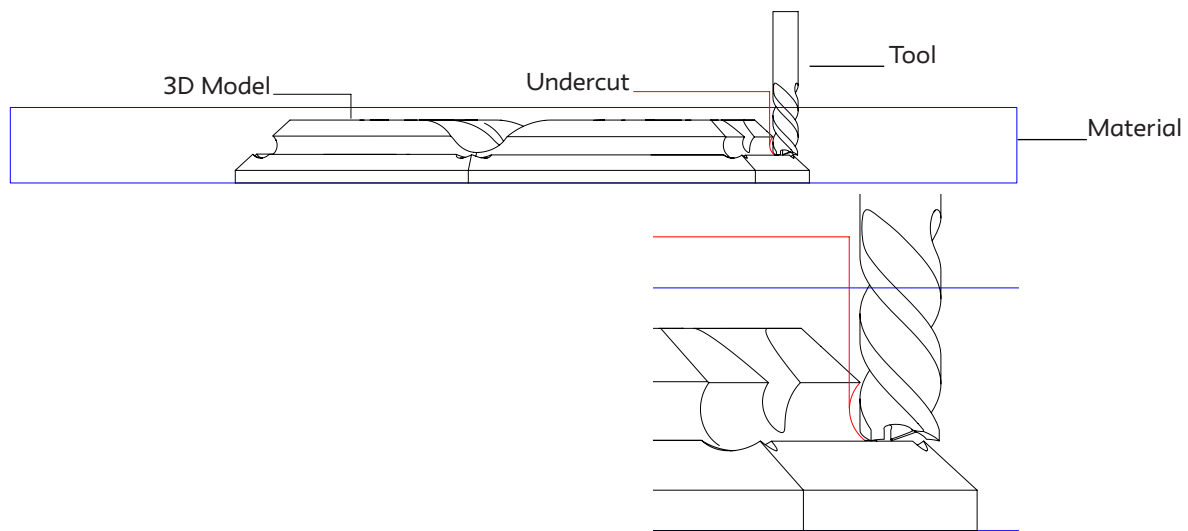
If you want to use climb we recommend using Conventional first to remove 90% of the material and create a finishing pass that is climb.

## 3D Machining

The **3D model** must be the **exact size or smaller** than your stock material. If you are creating a digital model that is one inch thick, the stock material should than be at least one inch or slightly thicker.

Remember the stock material's thickness gets measured to the thousandths "**0.001**"

**Undercuts** are any angle that goes underneath the top surface of your model. Our machines **cannot** cut undercuts and will leave a 90 degree cut instead. Keep this in mind when designing.



## 2 Sided Machining - "Flip Files"

Flip files are done whenever a model has geometry on both sides requiring the stock to be flipped, reregistered, and machined on the 2nd side. Often models can be sliced in half, machined, and then glued back together.

If you feel your model needs to be flipped, you will need to fill out a form for Technician approval. You will need to send it at least 1 week prior to you needing your file cut.

The Technician will get back to you to set up a time for a further consultation if approved.

The form can be found here: <http://bit.ly/flipfile>