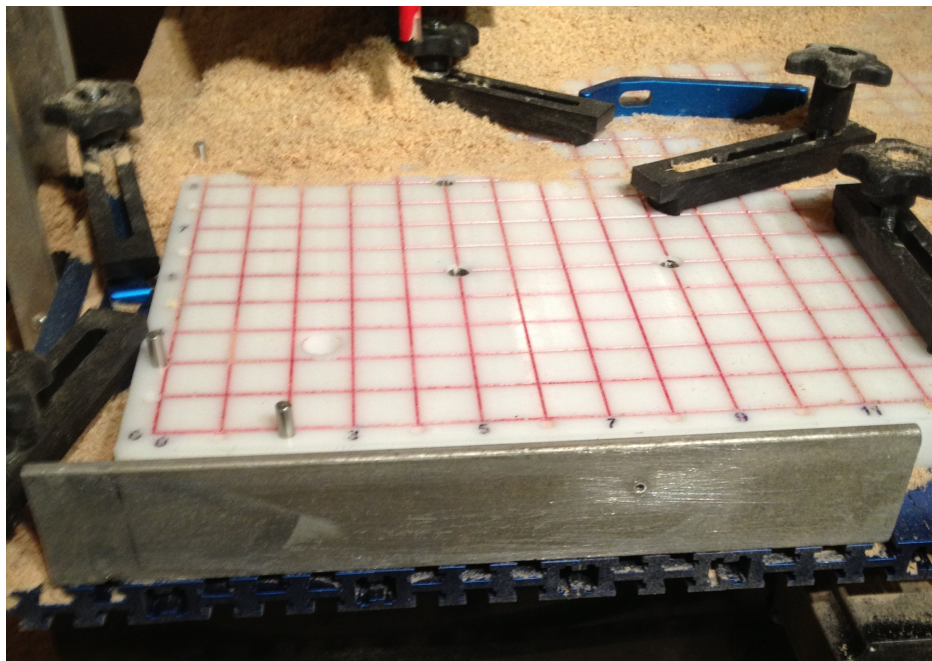


# How to carve Full Through Dovetail joints on your CNC Shark

We can break this down into three main areas: fixturing, geometry and trigonometry, and software (either Aspire or V-Carve).

## Fixturing

You may recall my previous post "Old Shark, new tricks" <http://www.cncsharktalk.com/viewtopic.php?f=4&t=2618> in which we carve the outer edge of a round bowl. Dovetail carving uses the same fixturing, namely a nice piece of Angle aligned with your X axis. This is of course if you want the sides of your box longer than 3 inches.



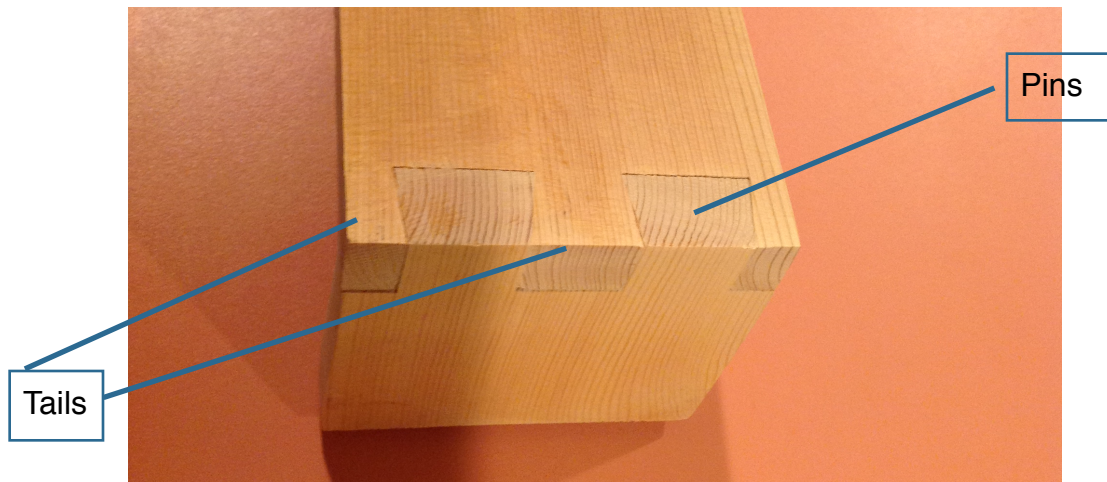
You will also need to make the famous Router mount extension.



This is so you can reach over the edge of your table. Your board is attached vertically to the angle. I clamp a short board to the angle for alignment which is handy when you are doing all sides for a box.

## **Geometry and or Trigonometry**

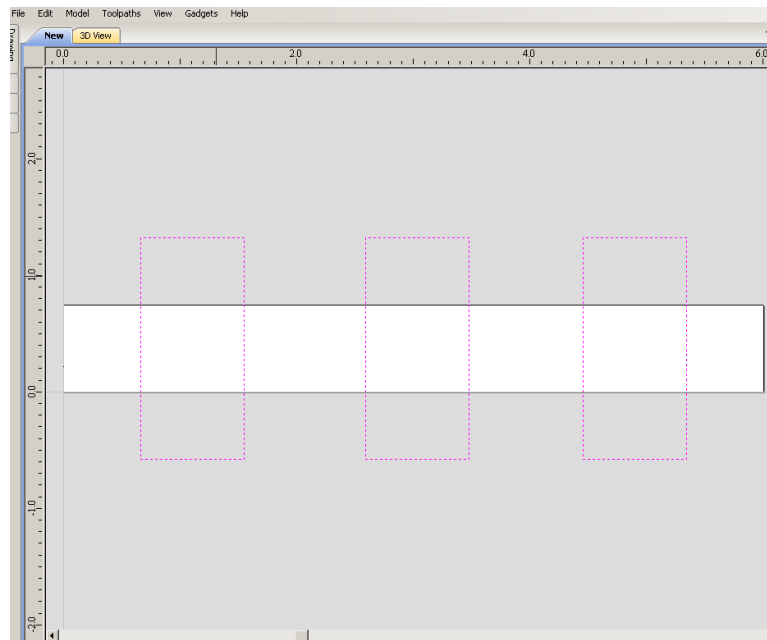
There are two parts to a dovetail joint: The pins and the tails.



Some call the tails at the ends half tails, in the middle is a full tail.

The pins are actually easy to program and cut, more later on that. The tails are more complicated. I broke it down into a two step process using 2 cutters.

I used all Pocket cuts to do this. Start by laying out where you want your tails and define the area between the tails at the smallest point (the board end).

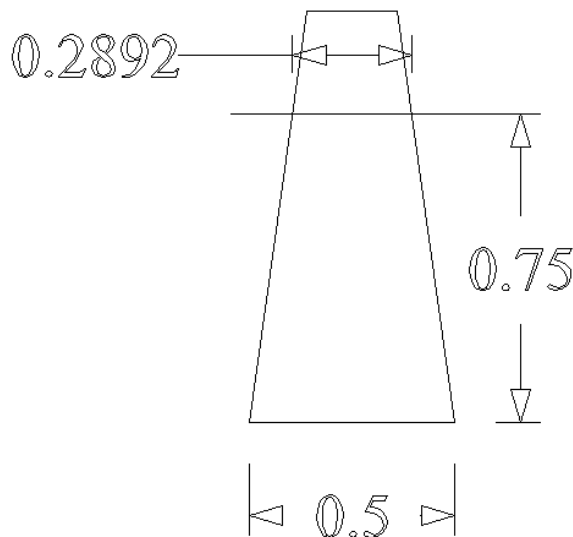


This is a layout for a 6 inch wide board. You'll end up with 2 full tails and 2 half tails. The pocket you will cut out should be more than 1/2 inch wide and longer than the width of the board by more than 1/2 inch on each side. Find out why later.

For the first cut I use a straight end mill, 1/2 inch dia. You can use any diameter that will cut 3/4 deep (width of board).

Now for the angled cut. I use a, wait for it, a dovetail bit. But here's the issue: There's no dovetail bit in the tool database and you can't define one in the form tools. So here is what you need to do:

The dovetail bit I used has a 1/4 inch shank, and is made by Freud, sold in Home Depot. The cutter is 1 inch long and the large diameter at the bottom is 1/2 inch.



It would be too big of a bite to use this at full depth with my Bosch colt without the clearance first. Using trig, you can calculate the diameter of this bit at the 3/4 inch height. It has 8 degree sides. Set up the bit as an end mill at the .2892 diameter. Make the pass depth .75 inch. Keep the stepover small.

## Tool Database

### Tool List

- Imperial Tools
  - End Mills
    - End Mill (0.2892 inches)
    - End Mill (0.25 inch)
    - End Mill (0.5 inch)
  - Ball Nose
  - V-Bits
    - V-Bit (60 deg 0.25")
    - V-Bit (90 deg 0.5")
    - V-Bit (90 deg 1.25")
  - Form Tools
  - Engraving
  - Specialist
  - Drills
- Metric Tools
  - End Mills
  - Ball Nose
  - V-Bits
  - Form Tools
  - Engraving
  - Specialist
  - Drills

New ...

Copy ...

Delete

New Group

Import...

Export...

### Tool Info

Name: End Mill (0.2892 inches)

Tool Type: End Mill

Notes

### Geometry

Diameter (D): 0.2892 inches

### Cutting Parameters

Pass Depth: 0.75 inches

Stepover: 0.0868 inches 30.0 %

### Feeds and Speeds

Spindle Speed: 12000 r.p.m.

Feed Rate: 70.0

Plunge Rate: 30.0

Tool Number: 1

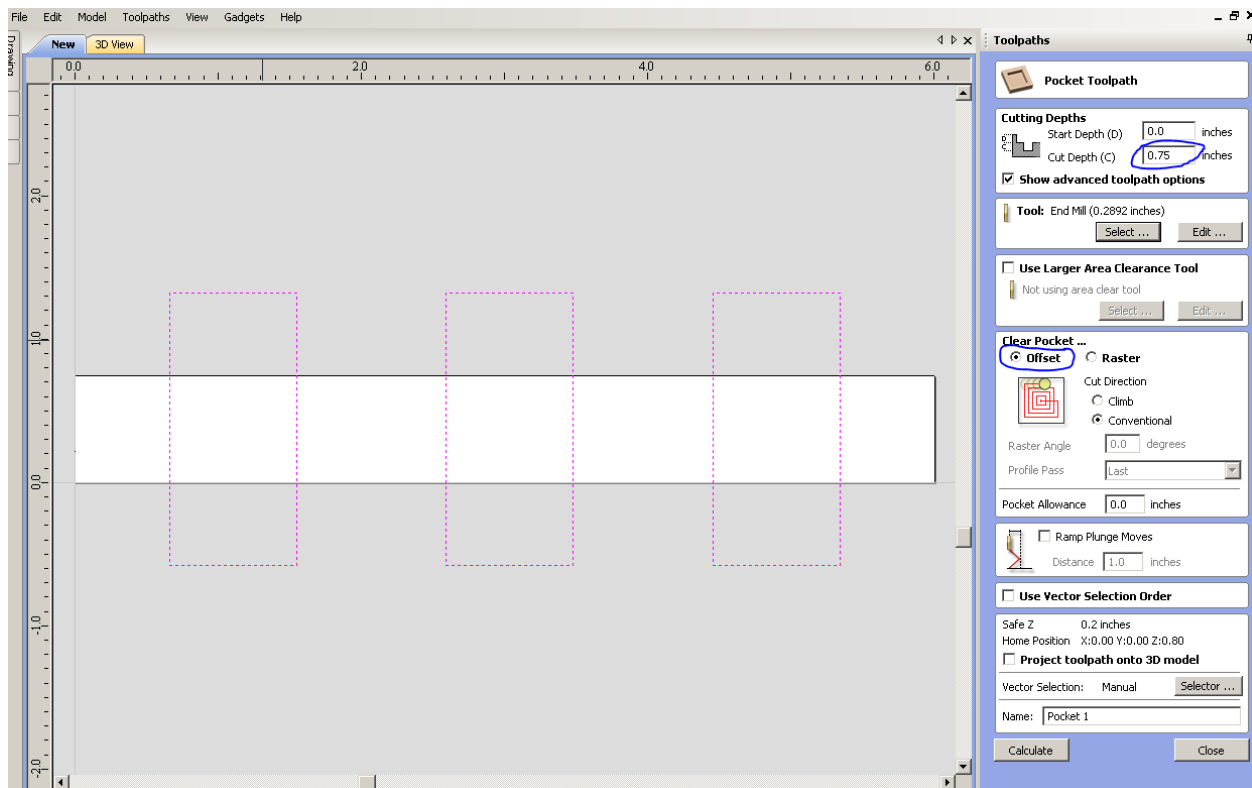


Apply

OK

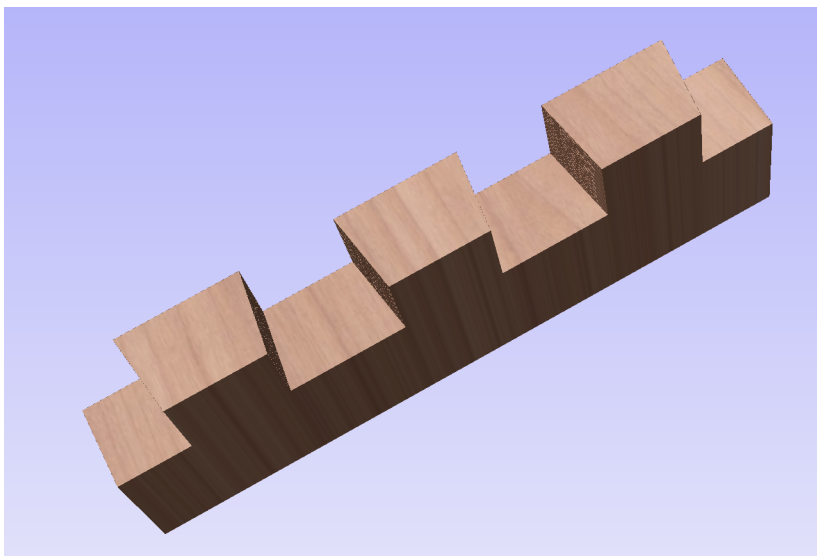
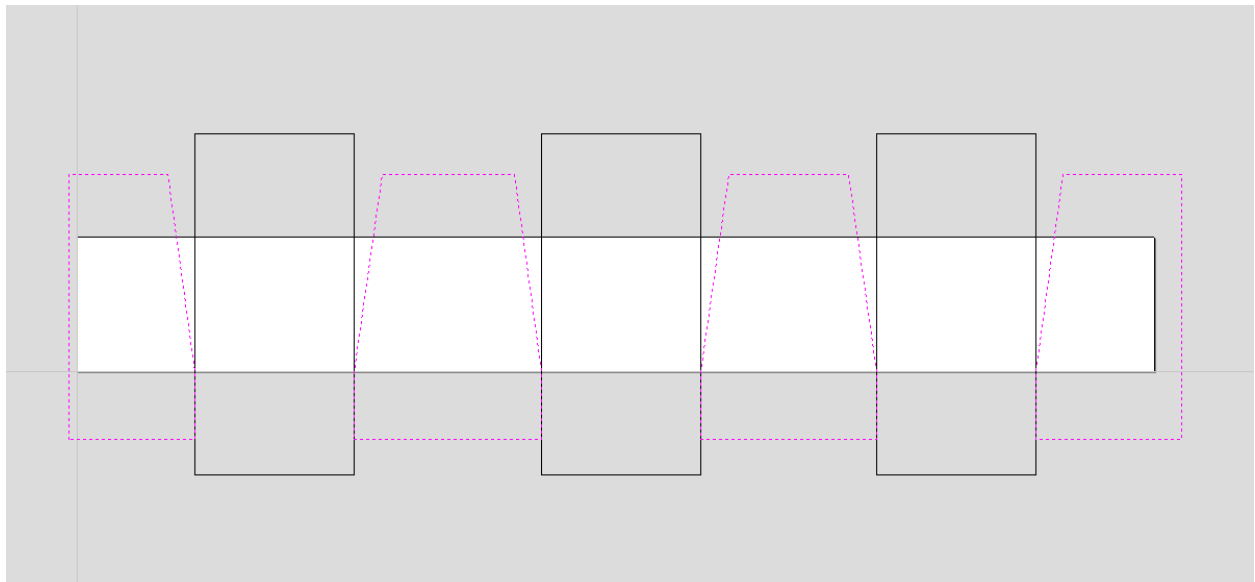
Cancel

## Software



Program your cut with this tool as above. Be sure to use the Offset feature and cut depth at .75. The Offset cut will start in the middle of the pocket and finish on the outside off the board so that when the bit lowers and raises it won't cut something it's not supposed to. (Thus the 1/2+ clearance requirement) This will give you the angled sides of the tails.

Next the pins layout can be created using the rectangles from above as a start point.



Set up your pockets with about a .005 inch offset to make the cut out larger. I used a 1/4 inch end mill for this.

Your two corners will lock together with a few light taps of your mallet.

Happy Carving!