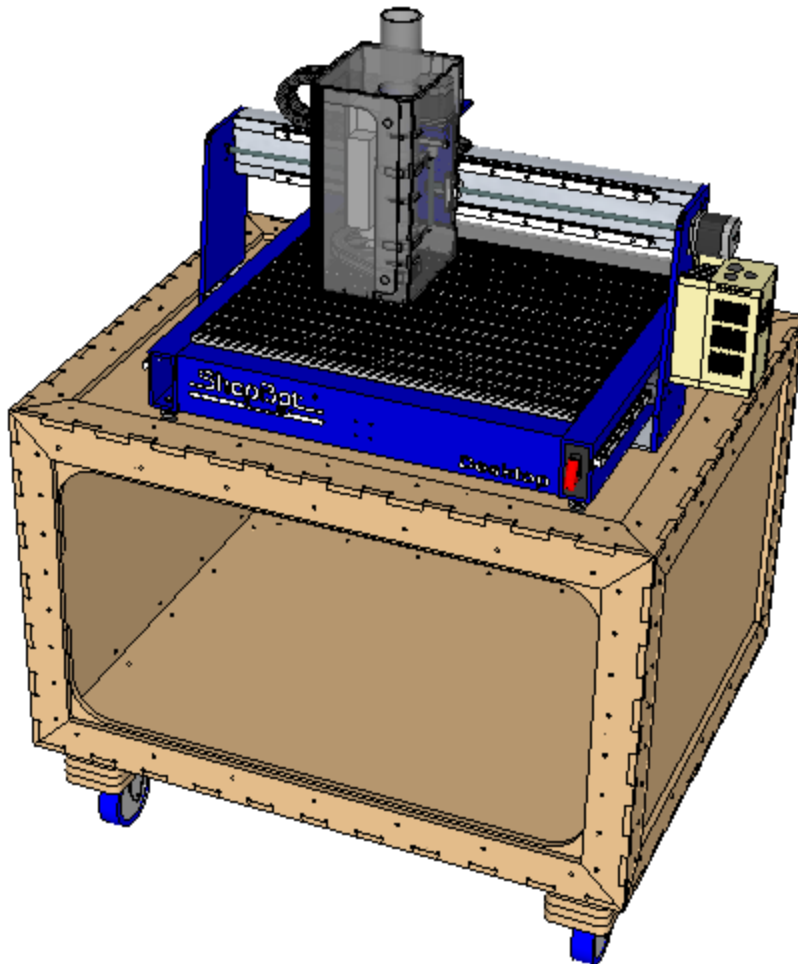


Make a Maker Crate



Working with the VCarve files:

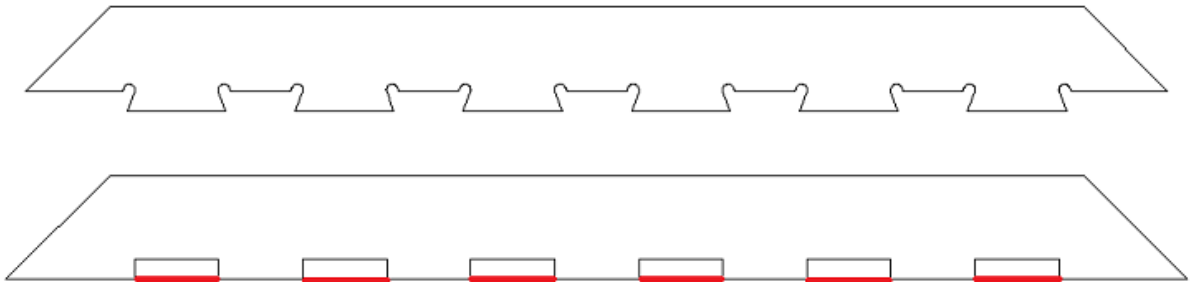
These files cut 1 of the rolling Maker Crate tables. They all use a 1/4" bit, except for the ShopBot logos which use a 60 degree v- bit or 1/8" straight bit to engrave. I suppose it's possible to cut the crates without vacuum holddown, but I wouldn't bet on it! Please let me know if you have success

Materials and hardware for one Maker Crate:

- 3 @ nominal 1/2" ply. I like Radiata Pine plywood because the outside veneers are nice and thick, so the dovetails are sized at 0.47" thick for that material
- 1 @ 3/4" Advantech for skids (enough for 2 crates)
- 132 @ 1/4-20 nuts Stafast part # 142074T

- 16 @ $\frac{3}{8}$ -16 nuts Stafast part # 381674T
- 132 @ $\frac{1}{4}$ -20 x 0.93" decorative bolts Stafast part # BB142025UHD
- 14 @ $\frac{1}{4}$ -20 x 4 $\frac{21}{64}$ " decorative bolts Stafast part # BB1420110UHD
- 4 @ Large plate 5" casters with brakes Castercity part # K3BP5x1-1/4-S3
- 16 @ $\frac{3}{8}$ -16 x 1.75" hex bolts and flat washers
- 4mm hex wrench or bit for each, or $\frac{5}{32}$ "

Sheets 1 and 2 each have parts that have flat-cut dovetails cut into them. The location of the dovetails are marked in the VCarve file with line segments across the opening of the dovetail pairs, shown in red below.



These line segments are all on their own layer, and to create the intermediate file a toolpath of those line segments is generated using the Profile option “on” the line, setting the cut depth at something shallow so that you’re positive that it won’t try to cut in multiple passes

2D Profile Toolpath

Cutting Depths

Start Depth (D) inches

Cut Depth (C) inches

Tool: End Mill (0.25 inch)

Passes: 1

Machine Vectors...

☐ Outside / Right

☐ Inside / Left

☒ On

Direction ☐ Climb ☒ Conventional

Allowance offset inches

☒ Use vector start points (don't optimize)

Tabs

☐ **Add tabs to toolpath**

inches

inches

☐ Create 3D tabs

The steps for creating the dovetail file:

1. Select all the line segments on the dovetail layer
2. Create a Profile toolpath of just those entities, with a very shallow cut depth, no ramps or tabs, and the "ON" option set.
3. Save that toolpath. They are included in the ShopBot files folder, but DO NOT cut them...they are just examples

I tend to cut all the dovetails first while I have good vac holddown, but you can create separate files from selected lines in Partworks if you want to do parts individually. You can also create you own parts and dovetails...the only thing to remember is that the dovetails always cut to the "left" of the direction of the line segment, so be aware of the start and end points of your line segments.

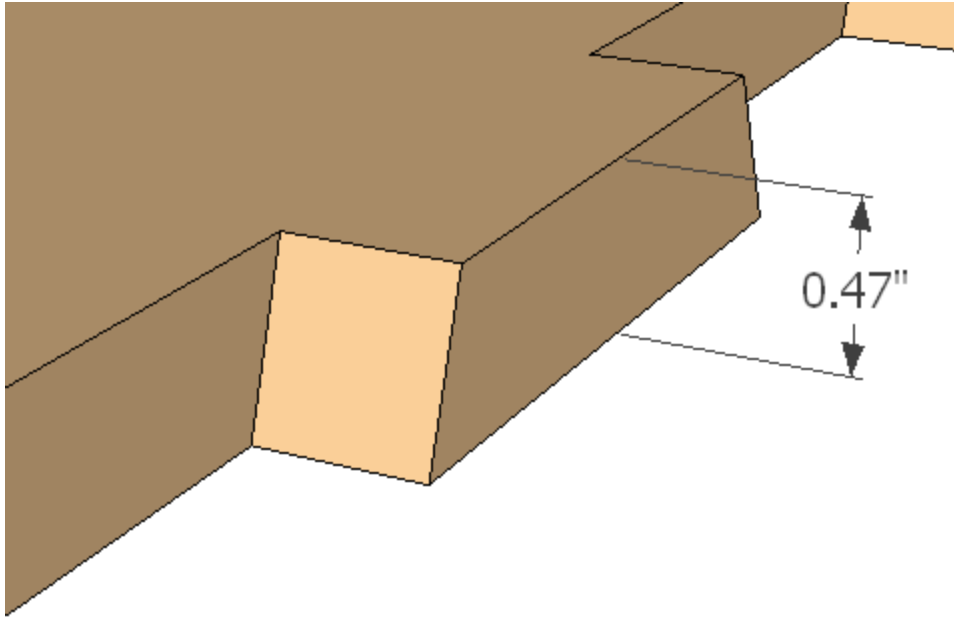
The cutout is done as any other. I cut full depth with a 1/4" bit so have the start points, cut order, and ramps setup for that, but as always you can modify as needed.

Running the EdgeTailer to create flat-cut dovetails:

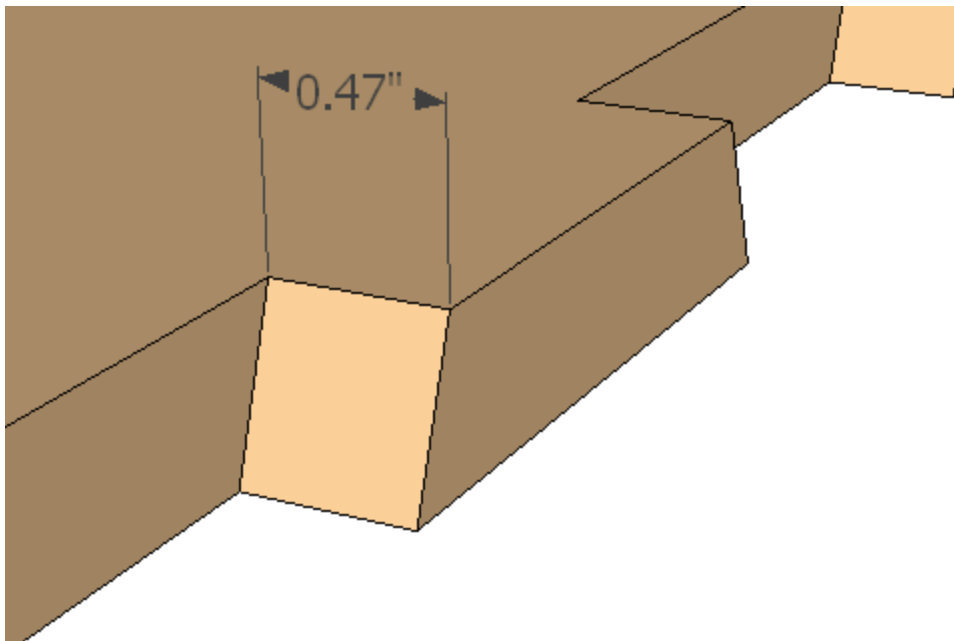
- Open the EdgeTailer program.
- Select the file you want to convert. It's the file that you created in PartWorks full of line segments defining the location of the dovetails
- A settings window will open that SHOULD have appropriate values by default. I've tried to explain them below, but if you get a chance to look at the labels and tooltips and see if you can figure it out before reading the settings info below it would help me refine them.
- Ignore the radio button for custom configuring dovetails for now...that lets you customize the dovetails on a face-by-face basic, read from the "angles.txt" file. All the dovetails in the crate are 20 degrees so you can use the default angle.
- Click the Convert button to select an output file name and write the 3d file. That's the file you'll be cutting

EdgeTailer Setting:

Accurate material thickness... the thickness of the material you're cutting. The more accurate that value is the better your dovetails will turn out

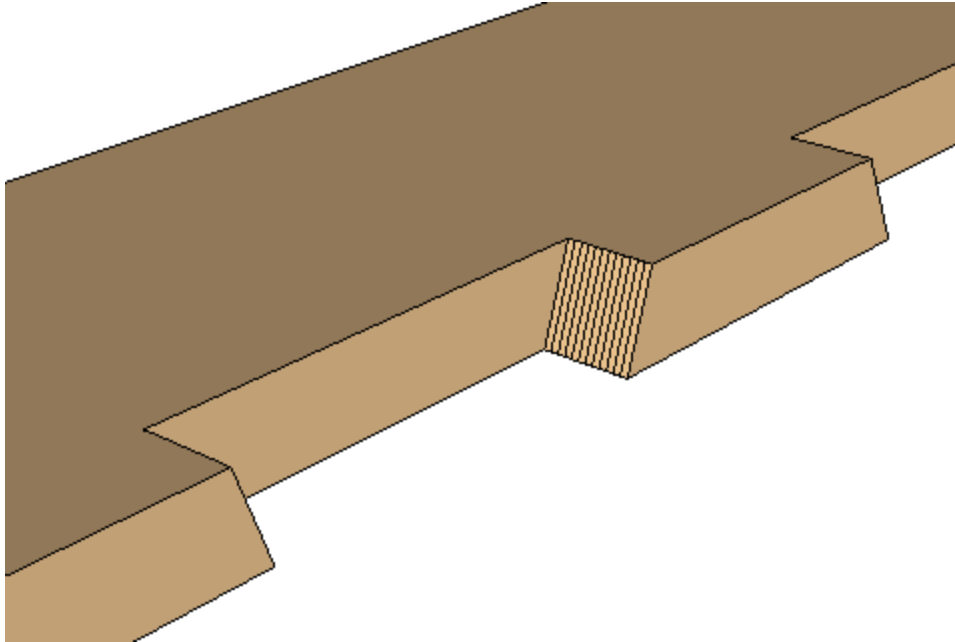


Depth... This will be the same as material thickness if you are joining pieces of the same thickness, but you can also join materials of different thicknesses...this is the thickness of the piece that you're joining to your dovetailed piece.



Bit Diameter... just what it says

Resolution...sets how many steps when cutting the dovetailed faces. This is expressed in a percentage of the bit diameter so that you don't have to rethink this value as material size changes.

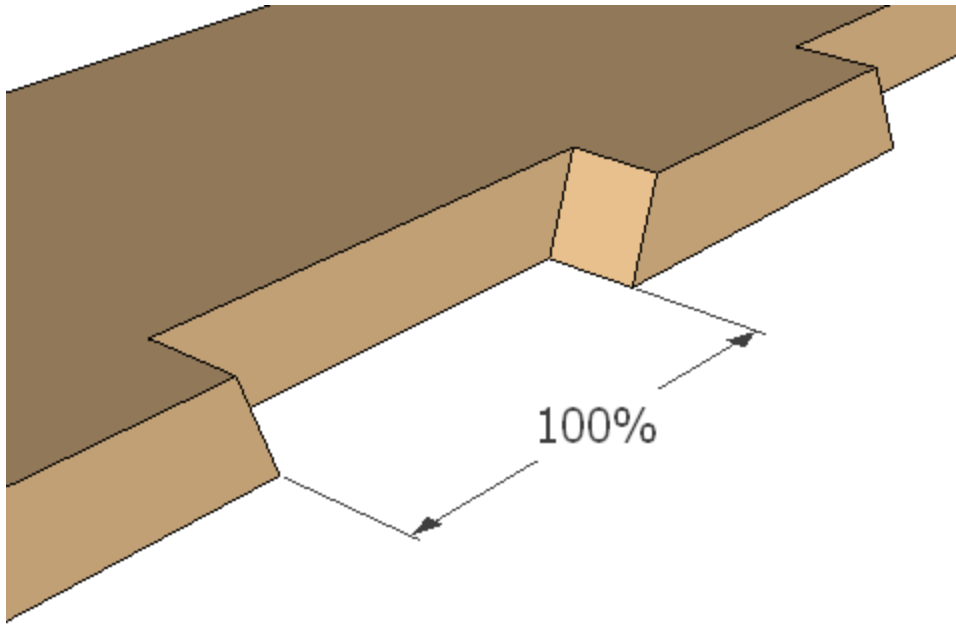


Dovetail Angle...There are two ways to specify the face angles of the dovetails. If all the faces are the same angle just click that radio button and enter the face angle in degrees...all the dovetails will be cut at that angle.

If however you want to have different angles on faces, check that option and it will read the faces for each pair from a text file named “angles.txt”, which is just a list of paired face angles, separated by commas. An example angles.txt file is included in the zip file. NOTE: If you use an angles.txt file it MUST be in the same folder as the Edgetailer program

X and Y speed, Z speed...Once again pretty self explanatory. These values can be a little higher than regular cutting speed because you aren't taking much of a cut.

Tolerance...How tight the fit will be, specified as a percentage of the drawn size. A value smaller than 100 will give a tighter fit, a value larger will give a looser fit. I tend to use 97-98% which gives the fit I like



I know this is very rough...comments and suggestions are always welcome,

Bill

P.s. There's also a routine for cutting dovetails in the bottom of slots called the SlotTailer for connections like the one below...I'll pass that on later!

