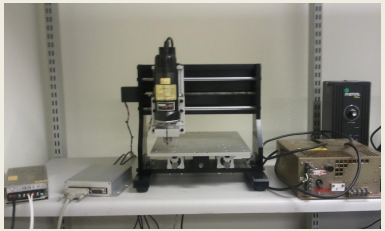
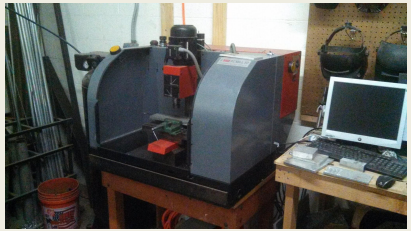
CNC Router Introduction

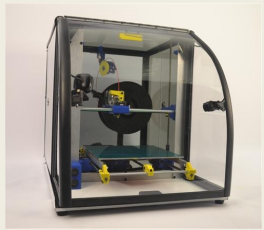
Using the ShopBot Desktop MAX CNC router at PCC

What is CNC

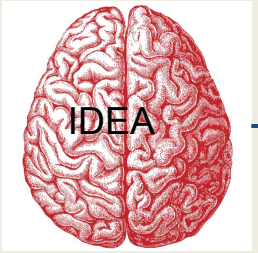
Stands for **C**omputer **N**umerical **C**ontrol

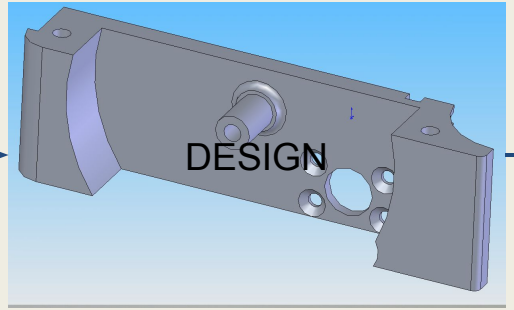


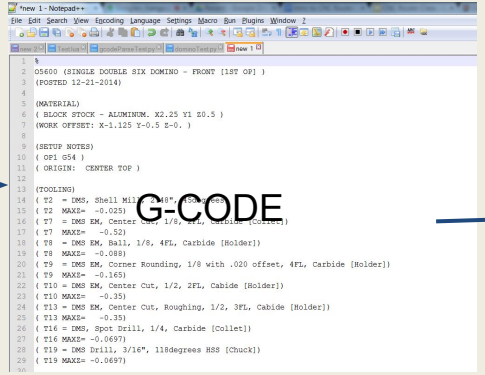


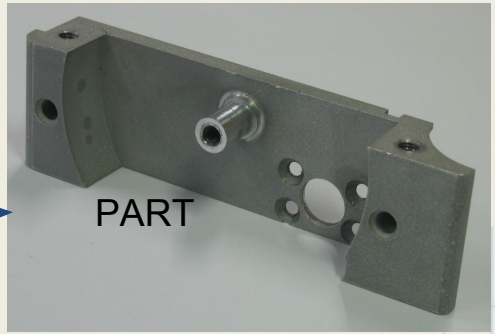


CAD and CAM


IDEA


DESIGN


G-CODE
%
O5600 (SINGLE DOUBLE SIX DOMINO - FRONT [1ST OP])
(POSTED 12-21-2014)
(MATERIAL)
(BLOCK STOCK - ALUMINUM. X2.25 Y1 Z0.5)
(WORK OFFSET: X-1.125 Y-0.5 Z-0.)
(SETUP NOTES)
(OP1 G54)
(ORIGIN: CENTER TOP)
(TOOLING)
(T2 = DMS, Shell Mill, 1", 2FL, Carbide [Holder])
(T2 MAXE= -0.025)
(T7 = DMS EM, Center Cut, 1/2, 4FL, Carbide [Collet])
(T7 MAXE= -0.52)
(T8 = DMS EM, Ball, 1/8, 4FL, Carbide [Holder])
(T8 MAXE= -0.089)
(T9 = DMS EM, Corner Rounding, 1/8 with .020 offset, 4FL, Carbide [Holder])
(T9 MAXE= -0.165)
(T10 = DMS EM, Center Cut, 1/2, 2FL, Cabide [Holder])
(T10 MAXE= -0.35)
(T13 = DMS EM, Center Cut, Roughing, 1/2, 3FL, Cabide [Holder])
(T13 MAXE= -0.35)
(T16 = DMS, Spot Drill, 1/4, Carbide [Collet])
(T16 MAXE= -0.0697)
(T19 = DMS Drill, 3/16", 118degrees HSS [Chuck])
(T19 MAXE= -0.0697)


PART

G-code

G-code is the most commonly used language for NC

- A completely text based programming language
- Called “G” code because of all the instructions starting with the letter G
- Programs consist of “blocks” of instructions (lines)
- Blocks consist of “words” which are a letter and number
  - G00 - rapid movement
  - G01 - cutting movement with a feedrate
  - X, Y, Z - movement in those axis (for example X0.0 Y1.0 Z2.0)
- Multiple flavors of G-code
  - FANUC
  - YASNAC
  - ShopBot uses OpenSBP but will also read standard G-code:
  - http://www.opensbp.com/
  - http://www.shopbottools.com/ShopBotDocs/files/ComRef.pdf)

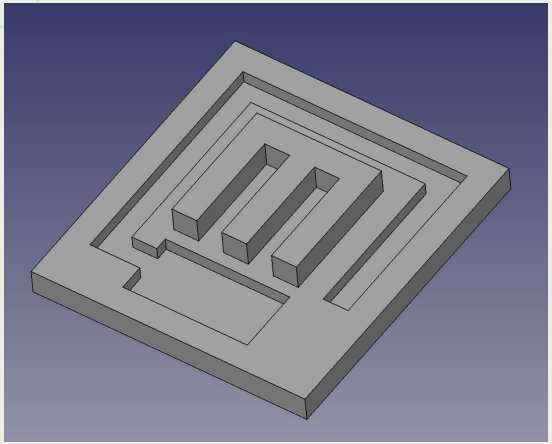
Limitations

Keep the limitations of a 3 Axis machine in mind when designing pieces to create (“design for manufacturing”)

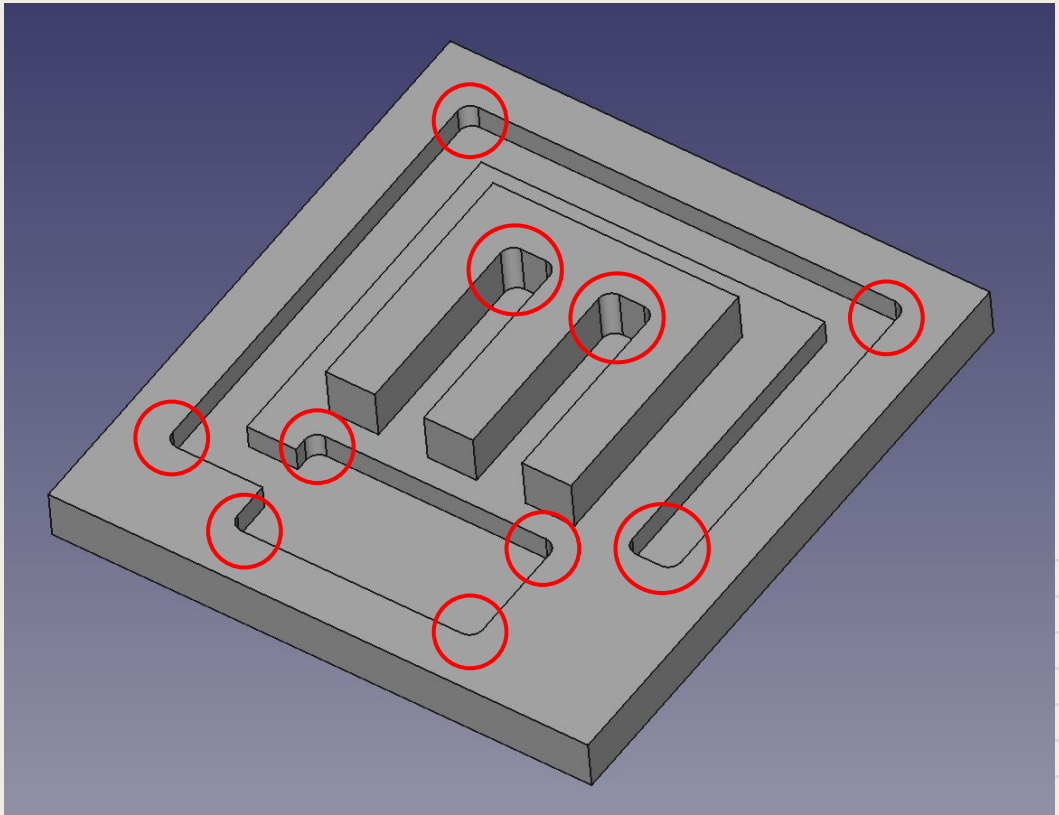
- No “undercutting” for the most part
  - Some bits allow a little undercutting, such a dovetails for example
  - If your piece is double sided, you will need a plan for “registering” it when it’s flipped over
- Interior corners will be rounded by the radius of the bit
  - V-bit engraving can mitigate some of this for artwork, making it look like a carving
  - Smaller bits makes smaller inner “fillets” obviously

Rounded Corners Limitation

What you want...



...what you get!



Bit Types

End Mill
- For 90 degree edges
- Flat Bottom with sharp edges

Ball Nose
- For rounded edges

Bull Nose
- Combination of a Ball Nose and End Mill
- Flat bottom with round edges

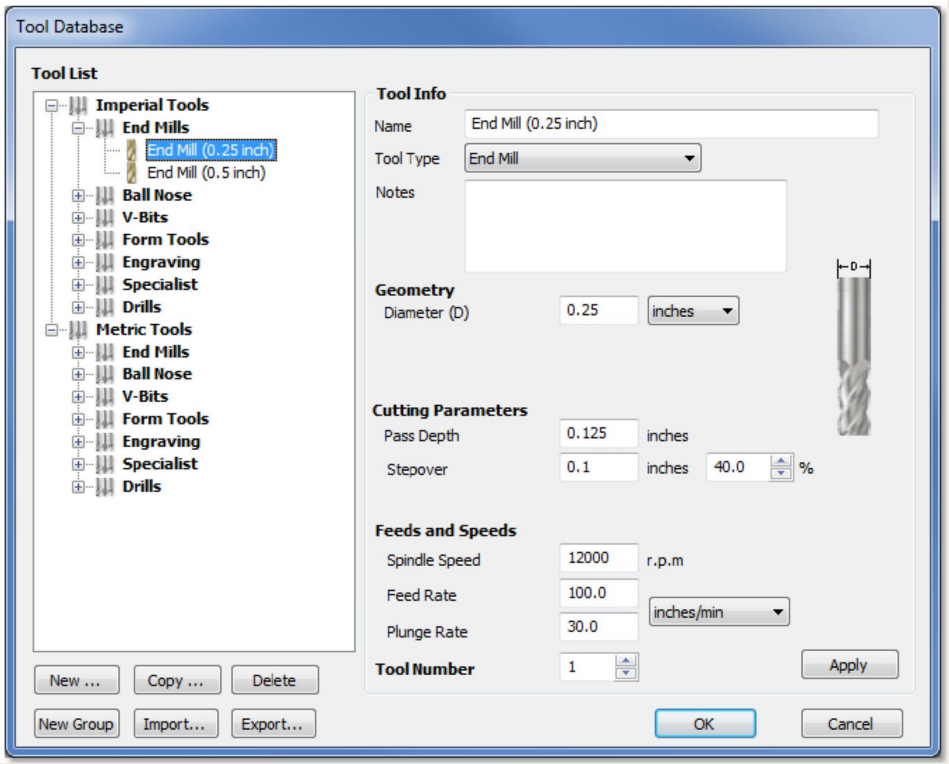
V Bit
- Commonly used for lettering in sign making, fluting, etc,

Engrave
- For small precision work.

End Mills

- 1 or 2 flute bits are mostly used
- Straight
  - cheap, easily sharpened, use only on wood and plastic
- Upcut (conventional endmill)
  - pulls chips up and out of the way (for conventional direction)
- Downcut
  - leaves a cleaner cut
  - exerts less pull up on parts
- Compression
  - combo of upcut and downcut
  - for plywood, composites, and laminates

Tool Database Setup



Process

1. Design CAD Model - Inkscape, SketchUp, AutoCad, Illustrator, VCarve Pro, Fusion360, Rhino, etc.
2. Import to CAM software (VCarve)
3. Create tool paths
4. Export G Code
5. Run G code on machine

Our Setup

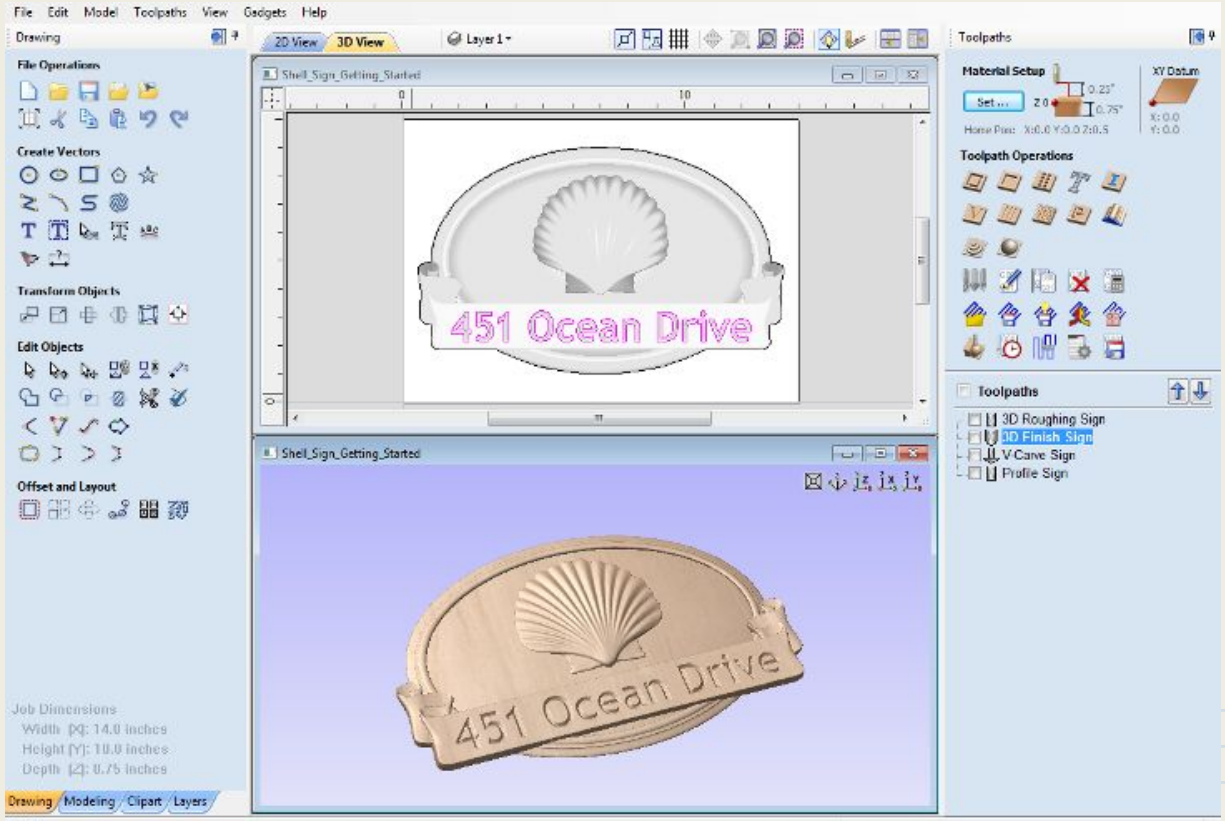
Vectric VCarve Pro 9.0

- Commonly used CAM software for CNC routers
- simple UI, intuitive
- downloadable demo that only has g-code output disabled
- full software available in the Multimedia Lab and Fab Lab computers
- many tutorial videos online

ShopBot Desktop MAX

- 24” x 36” (x,y) cutting area 3.5” (z) clearance under gantry 5.5” z travel
- Resolution: 0.00025” (.00635mm)
- manual hold down
- simple to use
- resume feature on tool break
- able to rotate a job in Vcarve to fit board

VCarve Pro Screen



Importing a file into VCarve

Supported Vector formats

- DXF, DWG, EPS, PDF, SKP and AI

Supported Bitmap formats

- BMP, JPG, GIF, TIFF, PNG

VCarve Pro can import a single 3D model to do relief designs (STL, OBJ, etc.)

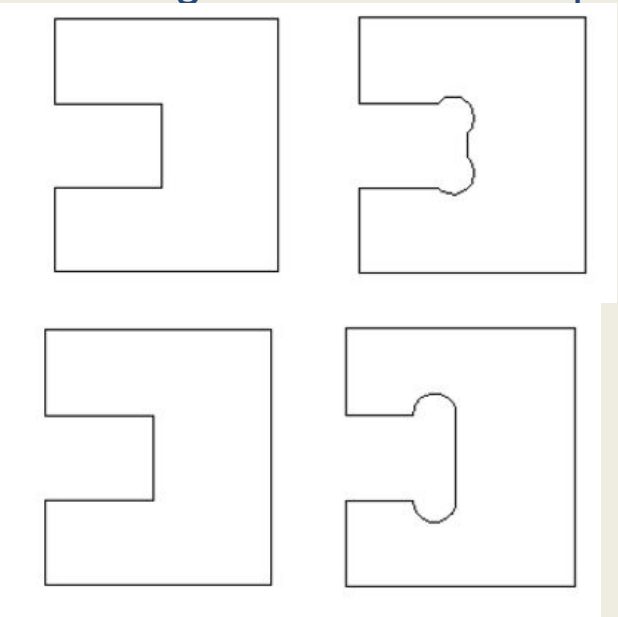
Tool Path Operations

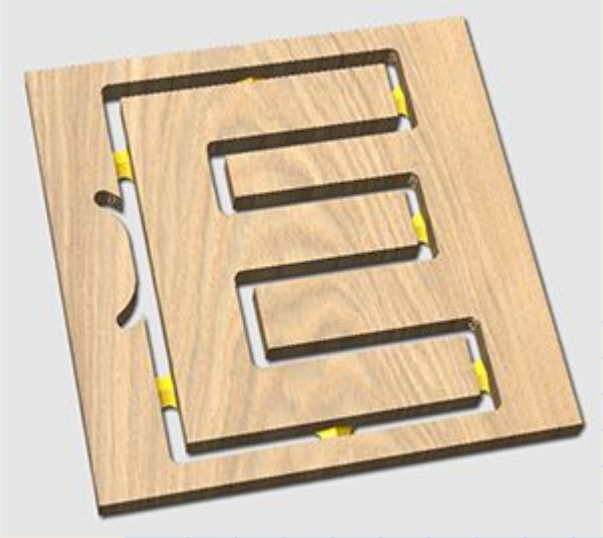
- Profile*
- Pocket*
- V Carve*
- Drilling
- Quick Engrave
- Flute
- Texturing
- Prism Carving
- Inlay Toolpath
- Hold-down Tabs*

*Necessary for test

Profile

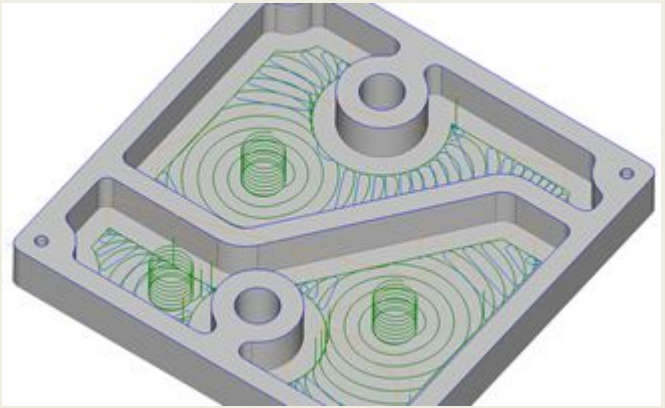
Used for cutting slots, dados, grooves,
Tabs and onion skinning
Dog Bone vs T-Bone pathing





Pocket

- Create a “valley” or recessed surface

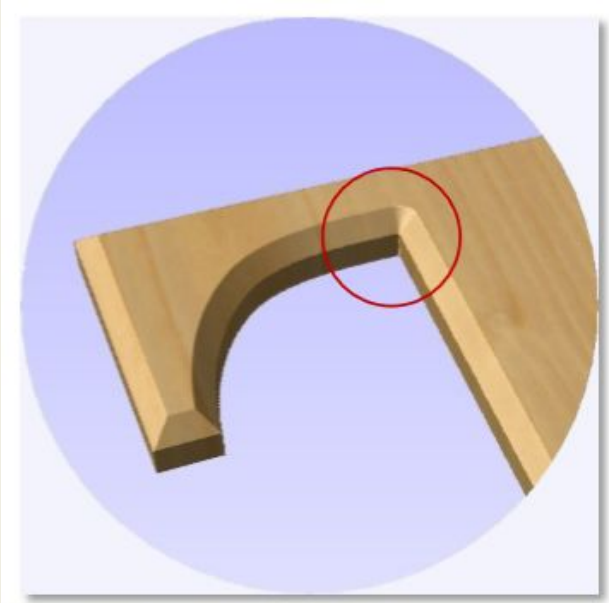


V Carving

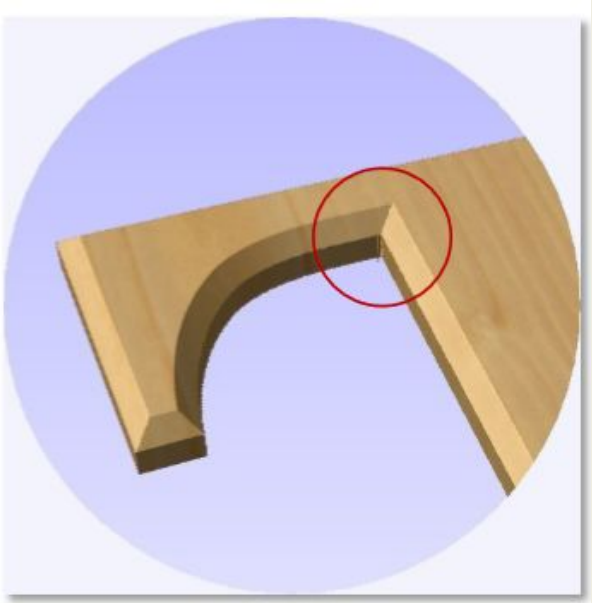
- Creates carved in features (looks great on images and lettering)
- Uses V bit
- Able to produce sharp(ish) inside corners

VCarving - Sharp Corners

Getting sharp internal carving



Sharp Internal Corners **not selected**



Sharp internal corners selected

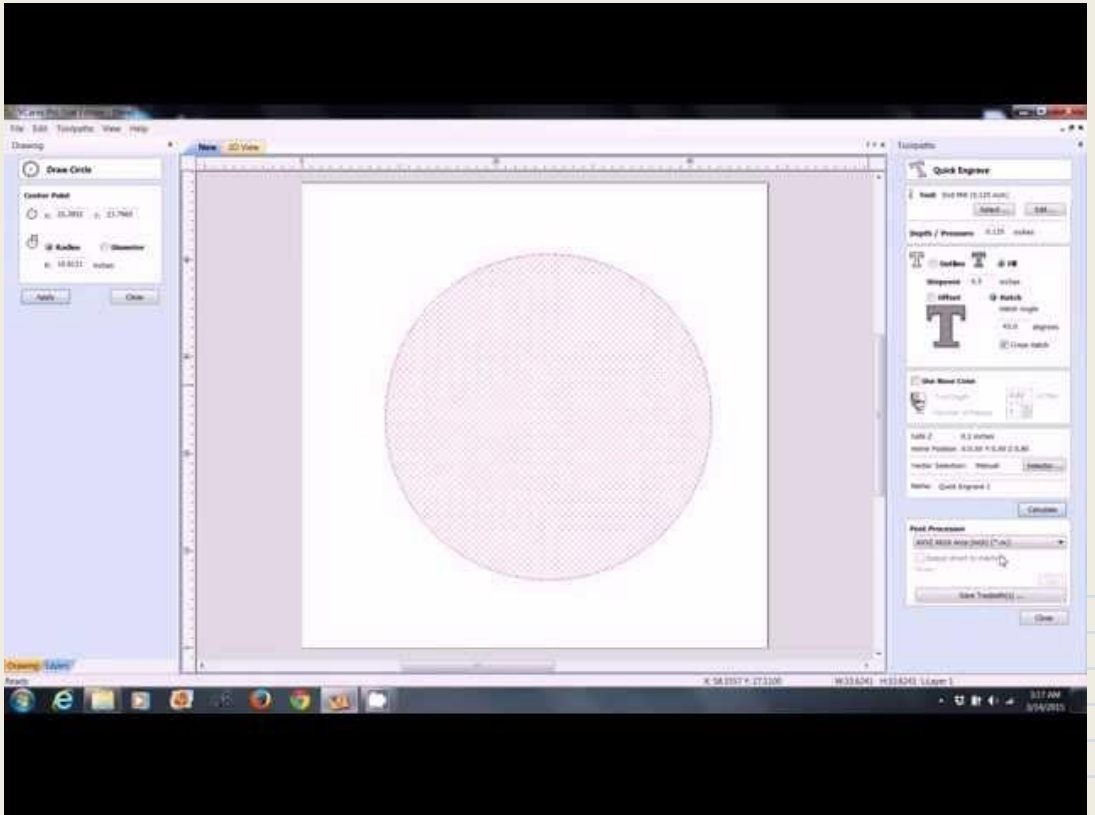
Drilling Operation

- Straight Down operation
- Not for End Mills
- Pecking
  - Used to clear chips that get built up
- Drill bit must match collet size

*Might be easier and faster to just create holes with a smaller end mill and spiral down.

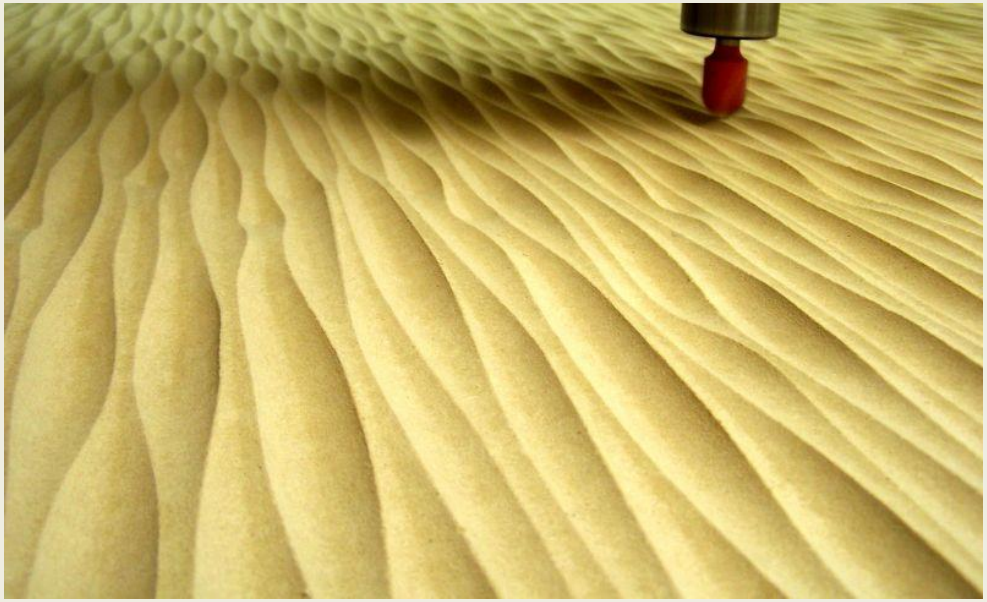
Quick Engrave

- For doing really quick operations and exporting directly from tool menus
- Supports hatching and simple outline (on the path)



Texturing

Allows you to fill an area with random texturing



Prism Carving

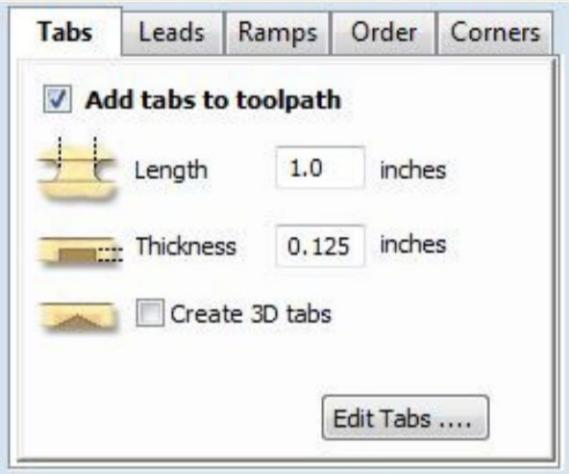
- Like the inverse of v-carve
- Produces features that are raised with the peak mid feature
- Great for creating stand alone letters

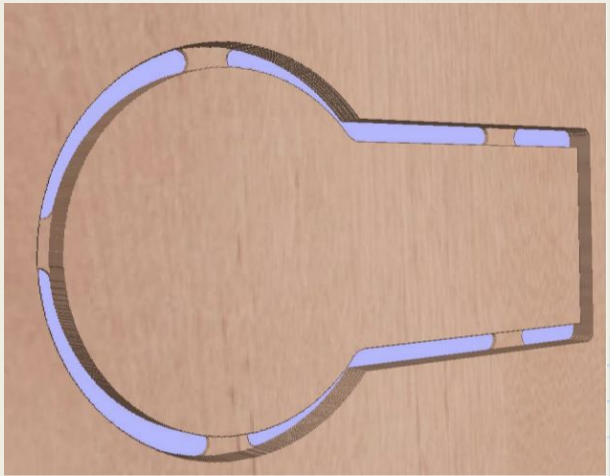
Inlay Tool

- Creates an outer and inner Operation
- Usually cut inlay and pocket out of 2 different materials
- Use v bit to get sharp inside corners in pocket and inlay
- Inlays are cut upside down

Hold-down Tabs

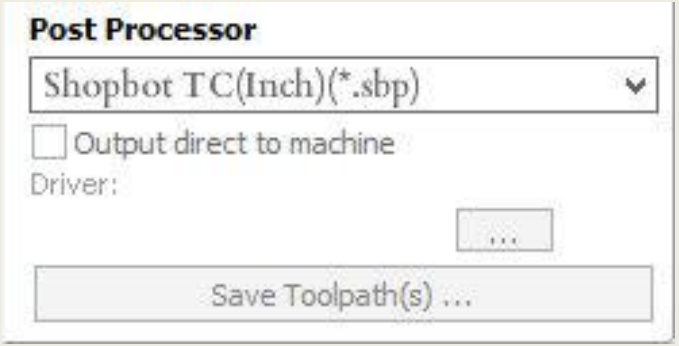
Tabs hold the material in place once the part has been cut out so that the material doesn't break free during the cut process. This is VERY IMPORTANT and will be covered in more depth when we discuss how to hold down material. Failure to do so can result in serious injury, because the part will go flying off the table if not secured, causing injury and damaging the machine.





Exporting Toolpath Files

- Each Tool should be its own Export (.sbp file)
  - Keep in mind tool order
    - Consider adding operation number at the front of your file name
    - Consider adding tool description in file name
    - **Make sure correct post processor is selected Shopbot TC (inch or mm)(*sbp)**



DEMO

Demo V Carve

Safety Information

- If ANY problems, don't hesitate - **E-STOP or Spacebar**
- Avoid Gantry : It can easily knock you over
- Wear Hearing, Eye Protection, and possibly breathing protection
- Keep hands & face away from tooling while running
- Bits can be hot while removing
- Keep loose clothing and hair away
- Careful with Bits and Mills in General (they are EXTREMELY sharp)
  - EXHIBIT A: A makerspace members scar from reaching into a Haas mill and brushing his arm against a RESTING tool in the changer

DISFIGURED FOR LIFE!
(don't be like that guy)

Connect Power

Start Up Process

- Make sure E Stop is reset before turning on.
- Make sure dust collector is connected and path to home position is clear
- Make sure the USB cable is connected to the shopbot laptop and the shopbot itself.
- Make sure the Z homing tool plate is disconnectd and the path to home position is clear.
- Make sure machine is plugged in, then flip the red switch and make sure the key is in the horizontal position (turn clockwise)

Tool Movement options

The “Easy” Control Panel

This panel provides essential machine information and controls like input indicators, position information and homing buttons for all of the axes.

The Keypad

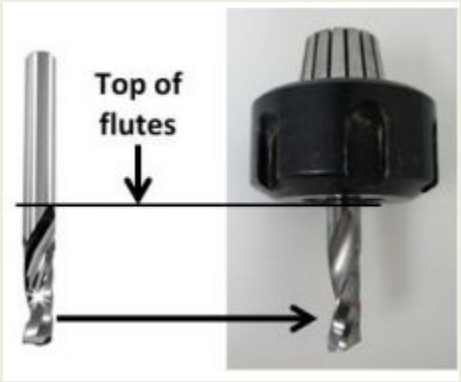
This window will allow users to manually move the X-, Y- and Z-axes of the machine. Click on the blue arrows to move the router and gantry.

The Keyboard option

Keyboard option:
You can also move the X- and Y-axes with the cursor buttons on the computer keyboard. Use the “Page Up” and “Page Down” buttons on the keyboard to move the Z-axis up and down.

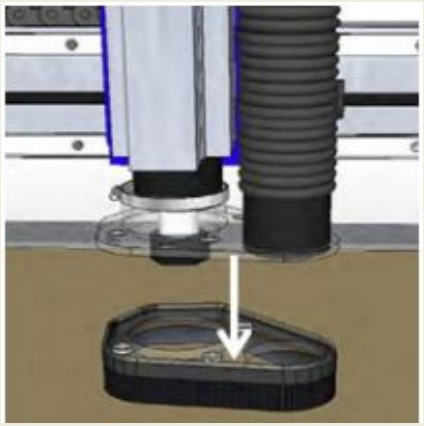
Install Bit

- Locate the bit that you set up in your Vcarve software
- Press the spindle collet into the spindle nut and listen for the “click”
- Slide your bit into the collet. Ideally, the shank of the bit should full up at least 75% of the collet to provide sufficient gripping surface. Make sure that the collet only grips the shank and not any of the flutes of the bit.



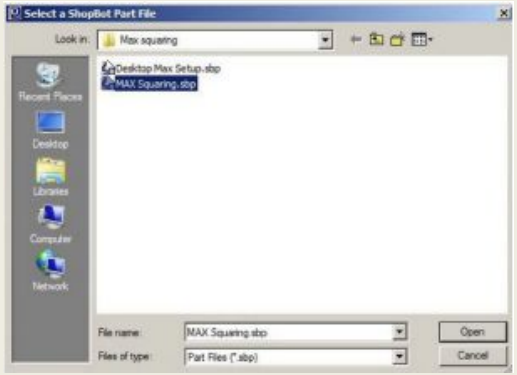
Install Bit

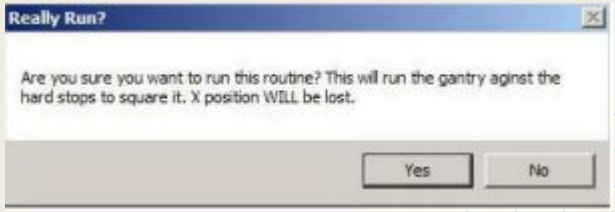
- Remove the magnetic dust foot by pushing down.
- Hand-thread the collet nut onto spindle. It should go on easily. Do NOT force it. Remove and try again if it is not going on easily. Once the collet is finger-tight, reposition the bit if it has slipped.
- Use the collet wrenches to fully tighten the nut. Over-tightening can damage it. Under-tightening will allow the bit to slip during operation.
- A firm one-hand squeeze is usually sufficient.



Squaring the gantry

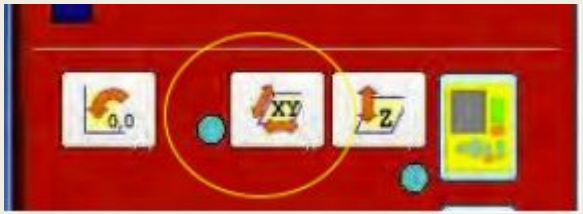
- *You only need to square gantry if it has been moved around or if the gantry was crashed since last job*
- Locate and open “MAX Squaring.sbp or “DESKTOP Squaring.sbp”
- Click START to run the squaring routine
- When prompted, click “Yes” to run the squaring routine
- This will make a loud scary noise as it runs the gantry head into the axis corners! (its ok its supposed to do that)





Zero the X & Y axes

- To home the X & Y axes, click the middle XY button. The machine will move through an automatic zeroing routine.
- After running the routine, the X and Y locations in the control panel should now read .000 in
- The router should now be in the home position, which is the lower right corner of the deck



Holding Down Material/Sheets

If you have room, you can drive screws into the excess material itself, OUTSIDE of the cut path of the bit. Do a perimeter outline to verify where the cut path is.