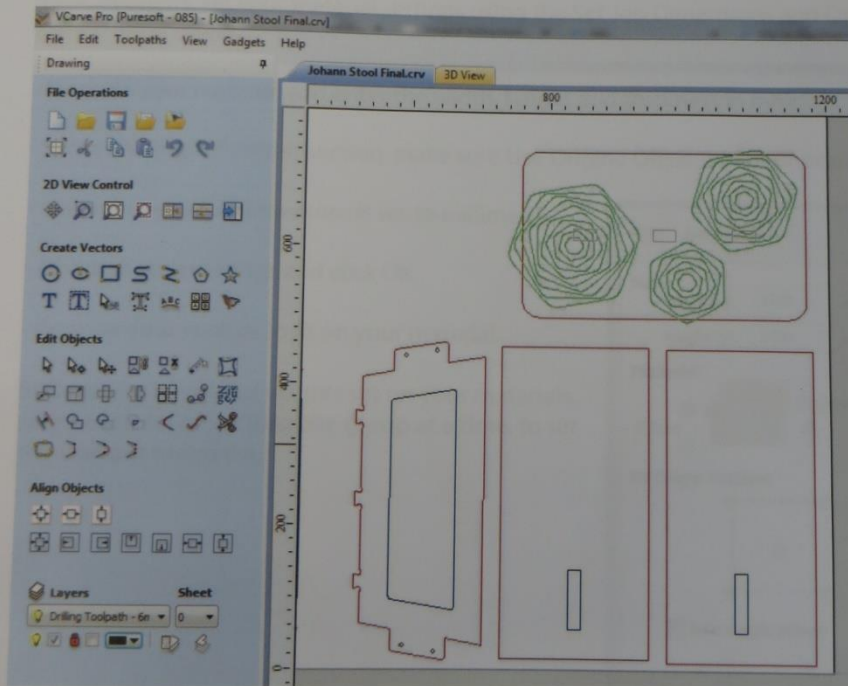


Instructions on 'Introduction to CNC Routing' from MakLab

VCarve - Setting Up Your Toolpaths

Instructions:

- 1) Open VCarve and open the Johann Stool Final.crv file on the desktop.
- 2) To import your engraving vectors, select File > Import > Import Vectors > Select your file. Then position your engraving vectors on the stool top.



Colour Index:

- Red = Outside Cut - 6.3 End Mill
 - Blue = Inside Cut - 6.3 End Mill
 - Purple = 10mm Sunken Pocket - 6.3 End Mill
 - Black = Drilling Toolpath - 6mm Drill Bit
 - Green = VCarve Engraving Toolpath - 90 deg VBit
-

3) Using the Create Fillet tool, add fillets to all your sharp angles on your transverse piece and your sunken pockets. Fillets size are set by radius, so for a 6.3mm End mill, you will be using 3.15 size fillets.

4) For the purpose of this exercise, we have given you a variety of wood offcuts (18mm furniture grade Birch Plywood for the legs and transverse and 18mm white melamine topped plywood for the seat of the stool). For each piece, you will need to take measurements, set your material settings using the Set Job Dimensions and Origin tool on the left hand menu.

- Measure your material and enter the width, height and thickness in mm.
- In the XY Origin Position section, make sure Use Origin Offset is kept unticked.
- Make sure your Unit measure is set to millimetre.
- Check all your settings and click OK.
- Arrange your vectors to fit on your material.

5) Once you have your vectors set on your materials, select each set of vectors, one group at a time, to set the relevant toolpaths.

Job Setup

Size
Width (x): 1200 mm
Height (y): 2400 mm

Material
 Z Zero
Thickness (z): 18 mm

XY Origin Position

Use origin offset
X: 0.0 Y: 0.0

Data Scaling
 Center data in job
 Scale data with job

Units
 inches mm

OK Cancel

Inner Cut Toolpath Settings:

Select Create Profile Toolpath in the Toolpath Menu.

Settings:

Start Depth = 0.0 mm
Cut Depth = 20mm.

In Select Tool, select a 6.3mm End Mill in Metric Tools:

Pass Depth = 4mm
Stepover = 40%
Spindle Speed = 20000 rpm
Feed Rate = 4.0 m/min
Plunge Rate = 2 m/min
Select Apply, then OK.

In Machine Vectors, select Inside/Left and Conventional.

Select the Add Tabs option. Set your tabs to 10mm Length, 5mm Thickness.
Click Edit Tabs and distribute 2 - 4 tabs on your vectors depending on their size.

Name your toolpath with your initials, a good description and the name of the tool used.

Select Calculate, check preview in 3D view, return to normal view.

Outer Cut Toolpath Settings:

Select Create Profile Toolpath in the Toolpath Menu.

Settings:

Start Depth = 0.0 mm
Cut Depth = 20mm.

In Select Tool, select a 6.3mm End Mill in Metric Tools:

Pass Depth = 4mm
Stepover = 40%
Spindle Speed = 20000 rpm
Feed Rate = 4.0 m/min
Plunge Rate = 2 m/min
Select Apply, then OK.

In Machine Vectors, select Outside/Right and Conventional.

Select the Add Tabs option. Set your tabs to 10mm Length, 5mm Thickness.
Click Edit Tabs and distribute 6 tabs on your vectors and distribute them equally.

Name your toolpath with your initials, a good description and the name of the tool used.

Select Calculate, check preview in 3D view, return to normal view.

Sunken Pocket Toolpath Settings:

Select Create Pocket Toolpath in the Toolpath Menu.

Settings:

Start Depth = 0.0 mm
Cut Depth = 10mm.

In Select Tool, select a 6.3mm End Mill in Metric Tools:

Pass Depth = 4mm
Stepover = 40%
Spindle Speed = 20000 rpm
Feed Rate = 4.0 m/min
Plunge Rate = 2 m/min
Select Apply, then OK.

In Clear Pocket, select Offset and Conventional.

Select Ramp Plunge Moves and set distance to 15mm.

Name your toolpath with your initials, a good description and the name of the tool used.

Select Calculate, check preview in 3D view, return to normal view.

Drilling Toolpath Settings:

Select Create Drilling Toolpath in the Toolpath Menu.

Settings:

Start Depth = 0.0 mm

Cut Depth = 20mm.

In Select Tool, select a 6mm Drill in Metric Tools:

Pass Depth = 10mm

Spindle Speed = 12000 rpm

Plunge Rate = 2 m/min

Select Apply, then OK.

Select Peck Drilling and set retract Gap to 4mm.

Name your toolpath with your initials, a good description and the name of the tool used.

Select Calculate, check preview in 3D view, return to normal view.

VCarve/Engraving Toolpath Settings:

Select Create VCarve/Engraving Toolpath in the Tool path Menu.

Settings:

Start Depth = 0.0 mm
Flat Depth = 5mm.

In Select Tool, select a V-bit 90 deg/32mm in Metric Tools:

Pass Depth = 10mm
Final Stepover = 1.3%
Clearance Pass Stepover = 20%
Spindle Speed = 24000 rpm
Feed Rate = 4.0 m/min
Plunge Rate = 2 m/min
Select Apply, then OK.

Name your toolpath with your initials, a good description and the name of the tool used.

Select Calculate, check preview in 3D view, return to normal view.

10) Once all your toolpaths are made, return to the Toolpath menu and save each toolpath to your project folder, by selecting them with the tickbox next to their name. Toolpaths using the same tool bit can be saved together by ticking both boxes at the same time. All toolpaths must be saved as an NC Studio ATC Arcs (mm) (*.nc) file (set as default).

11) Save the project file in the project folder by going to File > Save as > file name.crv (VCarve File format)

NC Studio - Sending your files to cut

Instructions:

- 1) Open NC Studio, then select File > Open and Edit > then select the first toolpath to cut.
- 2) Delete G91 and G28 on line N20, then go to File > Save and Load.
- 3) Once your Gcode is loaded on the screen, you can begin your machine set-up. Go to Operation > Move to Reference Point > Select All Axes (this will move the tool to all the limit switches, helping it calibrate itself in reference to the size of the bed).
- 4) Drill your piece of material into the cutting bed and secure the appropriate tool bit in place.
- 5) Switch to Manual Tab and use the controls to jog manually the drill bit to the centre of your material in order to calibrate the Z axis using the calibrating block. Select Operation > Mobile Calibrator. (Make sure the calibrating block is directly in the path of the tool bit as the tool won't stop unless it touches the calibrating block)
- 6) Move the tool bit using the controls in the Manual Tab to the bottom left corner of the material (make sure the centre of the spindle aligns with the corner). Click on the X and Y boxes on screen to reset them to zero.
- 7) Right click to set Top view in the Trace window and click on Simulate in the top menu to preview the cut. Click on Simulate again to exit the simulation mode.
- 8) Switch on the water bath and the extraction.
- 9) Click on the Start button to start your cut.

Please note: Ear Defenders and Eye Protection must be worn at all times during the operation of the CNC machines. Make sure you always know where to find the emergency stop on the machine you're operating. Loose clothing or jewellery must be removed or tucked away and hair must be tied back when operating the CNC router.
