

Software Guides

Documentation for software used by members in the shop.

- [Fusion 360](#)
 - [Design Tips for Manufacturing](#)
- [Google Docs](#)
- [Inkscape](#)
 - [Design Tips for Manufacturing](#)
 - [Inkscape Software Guide](#)
 - [Notes](#)
- [Janome Digitizer](#)
- [Lightburn](#)
 - [Member Notes](#)
 - [Lightburn Software Guide](#)
- [Prusa Slicer](#)
- [Sure Cuts-A-Lot](#)
- [VCarve Pro](#)
 - [Member Notes](#)
 - [VCarve Guides](#)

Fusion 360

Design Tips for Manufacturing

- Avoid importing SVGs or STLs until late in design, as they are computationally complex to operate on.
- Try using F360 parameters for dimensions you expect may change as you begin prototyping - these allow you to easily re-dimension the model without editing specific features.

Google Docs

Inkscape

Design Tips for Manufacturing

- Use Hairline stroke thickness when designing a file to be cut.
- To ensure the CAM process properly loads the completed design, always convert all objects to paths at the end of your design.
- It's easy to convert objects to paths, but not paths to objects - complete object transformations before converting to paths.
- Use Align and Distribute tools to save time patterning repeating shapes
- Shape builder can shortcut multiple boolean operations when constructing a complex shape from simple geometries.

Inkscape

Inkscape Software Guide

Inkscape is software for creating and editing vector graphics:

<https://inkscape.org/>

Inkscape is a valuable tool that's useful in many parts of the shop.

A common use of Inkscape is to prepare art for import into the software that drives various tools around the shop. Protohaven members use Inkscape to create and edit designs for use with:

- Large Format Laser (LightBurn)
- CNC Embroidery (Artistic Designer)
- Vinyl Cutter (Sure Cuts A-Lot)
- CNC Router (Vcarve)
- CNC Plasma (Sheet Cam)

Inkscape can be used to prepare art for the Large Format Printer.

Inkscape is also a good general purpose tool for creating visuals: drawings, infographics, logos, title blocks, icons.

Download

Inkscape is freely available to download and use for Linux, Windows, and MacOS:

<https://inkscape.org/release/>

Help and Tutorials

Manual

The Inkscape project maintains a comprehensive manual:

<https://inkscape-manuals.readthedocs.io/en/latest/index.html>

The manual is updated regularly, and available for both online (HTML) and offline (PDF, ePub) reading.

Video

A short tutorial to get started with Inkscape:

- Inkscape Tutorial: Complete Starter Guide for New Users (with chapters)

<https://www.youtube.com/watch?v=fzk-suGcqrc>

A comprehensive tutorial series for Inkscape is available from TJ Free:

<https://www.youtube.com/playlist?list=PLqazFFzUAPc5lOQwDoZ4Dw2YSXtO7lWNv>

Some videos from the series that are good places to start:

- Inkscape Lesson 1 - Interface and Basic Drawing

<https://www.youtube.com/watch?v=8f011wdiW7g>

- Inkscape Lesson 10 - Trace Images with Bezier Tool

https://www.youtube.com/watch?v=sagrkdmc_BI

- Inkscape Lesson 11 - Trace Bitmap Tool (Convert Raster to SVG)

<https://www.youtube.com/watch?v=E7HwLTQu2FI>

Notes

- The status line at the bottom of the window gives you information about highlighted objects
- Use the object dimensions area to view or set the dimensions of an object; use the lock to constrain the proportions.
- Use trace bitmap to get a vector outline of a raster image.

Janome Digitizer

Lightburn

Lightburn

Member Notes

Setting Up Lightburn on Your Own Computer

When building projects in Lightburn on your own computer, you do not need to have the profiles installed for the Rabbit laser cutters (the profiles are installed on the workstations connected to the laser cutters - importing your project into Lightburn there should ensure that they will be properly sent to the laser cutters).

It is useful to set the *bed size* in your copy of Lightburn to 900mm x 1200 mm to make sure that you don't create a project too big for the laser cutter.

Lightburn

Lightburn Software Guide

LightBurn is layout, editing, and control software for the large format lasers:

<https://lightburnsoftware.com/>

LightBurn is only available on the desktops dedicated for use with the lasers.

LightBurn is capable of handling all stages of a laser project, from art design through to running the job on the laser.

LightBurn can also import vector and raster art from other sources: you can work on your project in other software and then import it into LightBurn when you're ready to run the job.

Help and Tutorials

LightBurn software has a YouTube page (<https://www.youtube.com/@lightburnsoftware7189/>) with lots of content to help with projects. For those new to laser cutting and etching, these videos are a good place to start:

- Getting Started With LightBurn: Set up & First Project
<https://www.youtube.com/watch?v=v3RDzOrICTM>
- LightBurn UI Walkthrough
<https://www.youtube.com/watch?v=uzFsrUwONbw>
- LightBurn Cut Settings
<https://www.youtube.com/watch?v=nybhYtjEIQU>

Prusa Slicer

Sure Cuts-A-Lot

VCarve Pro

Member Notes

VCarve Makerspace Edition

Vectric VCarve Pro Makerspace edition at Home!

VCarve Makerspace Edition is a special version of VCarve that enables members of a makerspace to use VCarve Pro on their home computers to create projects to use with the Protohaven Avid CNC Router. Members can download the trial version of VCarve Pro and connect it to the Protohaven Makerspace connected Vectric account, allowing unlimited use of the trial version. Projects created on the Makerspace Edition can only be used in the Makerspace that they are connected to, but there are no other limits in terms of features or number of uses of the software. Follow the steps below:

- You must first create a [Vectric account](#). This is required to connect to Protohaven's Vectric Makerspace account.
- Please use the same email address for your Vectric account that you use for your Protohaven membership.
- Connect your Vectric account to the Protohaven Makerspace following these steps
 - Go to this link [Makerspace Invitation](#)
 - Put in your email address for your Vectric account
 - Open your email and look for an email from vectric.com with the subject "Vectric Makerspace Invitation".
 - Open the email and click on the link in the message.
 - Create an account with Vectric (or log into your existing one if you already have signed up).
 - When you get to the page with a section titled "Pending Organization Invitation", click "Accept".
 - Optional * Download and install VCarve Pro software from the link provided on the next page.
 - *note, VCarve only runs on Windows, but it is possible to run on Mac or Linux with a Windows emulator.
- Important Notes about running VCarve on your personal computer
 - You must sign into the software or the files that you save will not be able to be used with the Protohaven VCarve Makerspace edition
 - The trial version is limited. You cannot generate G-code on the trial version. You can create toolpaths, and save the file as a VCarve *.crv file onto a flash drive and bring this to Protohaven. Once you open VCarve on the Protohaven CNC computer, and open your file you will then be able to save your toolpaths to Gcode for import

into Mach3.

- Signing into VCarve on the CNC Computer at Protohaven.
 - When you open VCarve it will ask if you are a "registered member" Meaning you have created a Vectric account and registered that account with the Protohaven Makerspace. Or an "unregistered member" Meaning you do not have an account. Unregistered members will be prompted to register on the spot.
 - When you highlight "registered member" and click Next, there will be a dialogue that asks for your email associated with your Vectric account. Type your email, and click "Send One Time Code"
 - Check your email for the one time code, and put it into the dialogue. Vectric will authenticate your account.
 - A dialogue will confirm your name and the Makerspace Machine you are using.
 - VCarve will open and you will be able to use your files designed on a trial version.

VCarve Allowance Offset Cheat Sheet

From <https://forum.onefinitycnc.com/t/vcarve-offset-allowance-cheat-sheet/14207?u=wmbanderson>, user JoeT :

“ I created a cheat sheet detailing how profile and pocket toolpaths behave when allowance offsets (both positive and negative) are applied (a concept I have to relearn every time I need to use it). Thought I would share in case anyone else struggles like I do, trying to remember whether they need a negative or positive offset. :slightly_smiling_face:
Joe

Cheatsheet: [vcarve-allowance-offset.pdf](#)

VCarve Pro

VCarve Guides

[VCarve Manual](#)