

# CAM Software Testing

SLQ Wiki Fabrication Lab 2026/01/14 18:25

# CAM Software Testing

This is an archived page - for the latest version please visit [Vectric VCarve](#)

This page details our experience with CAM software for use on the Multi-cam CNC. New platforms are trialed and documented here. The list of software used (or to be tested) is;

- Enroute
- Vectrics Aspire
- Vectrics Vcarve Pro (makerspace edition)
- Fusion360

For all CAM programs except Enroute we will need to find Post Processors and make a tool library.

## Enroute

This is the supplied and recommended CAM platform. The version supplied was 5.1 - basic. It has extremely limited functionality;

- No drawing functions
- Limited import functions
- Buggy tool and path assignment

We have described most of the tools we use in the Enroute Library.

## Vectrics Vcarve

Recommended by contractors who have worked for SLQ in the past.

## Post Processor

Luckily the in-built AXYZ A2MC (MM) post is close - we compared it to the enroute version created our own modified

version

-refer to [vectrics forum](#) for a discussion.

## Autodesk Fusion360

## Post Processor

According to this forum thread here

this  
post processor should work.

## ILLUSTRATOR & 123DMAKE TO ENROUTE

Seeking to establish a workflow of design in Illustrator or 123D Make (exported as DXF) and layout/ and CAM settings via Enroute. here are my recollections a couple of hours later (figured i should get them down while it is fresh and we can edit/ improve later)

### 123D Make - Enroute

#### 123d Make

1. import an STL model into 123D Make
2. adjust the dimension of the model to the size output you want.
3. set material settings - dimensions of the plate and thickness, size of the tool you will use
4. choose construction method adjust number of surfaces/ shapes you want to cut the orientation of shapes etc
5. export file as DXF

#### EnRoute

1. new file
2. set plate dimensions
3. import DXF
4. Ungroup & delete markings, part numbers, bounding box etc
5. Select each shape to cut and create path offset cut settings? (set tool- feed rates, spindle speed etc; whether its an external or internal cut)
6. tabs on these cut paths (forgot this)
7. add fabrication features like dogbone fillets (drillholes)
8. Prepare output - Set cut strategies, order)
9. Save
10. output to jobs folder
11. Connect to machine via file manager and push outputted job into the machine memory.

#### notes

I wouldn't recommend trying to do too much editing of the shapes in enroute. i'd import files into Illustrator or Corell and make edits to the shapes

## Illustrator - Enroute

this needs testing but i image this wont be too different to exporting a DXF from 123D. one consideration is that you probably want to be careful which shapes/paths you are grouping as you are un likely to be able to separate these for different tool operations.

### Enroute

Ungroup holes before applying paths.

When outputting with multiple strategies - untick 'Maintain Grouping' to ensure the rough passes are all cut first.