LED light bar for Lemons race car numbers

DISCLAIMER: This solution HAS NOT BEEN TESTED UNDER RACE CONDITIONS. There is a non-zero chance that it will fail in some way when you really don't want it to. That said I am reasonably sure it will last 24+ hours over the course of the weekend. Have backup parts ready and a backup lighting plan if these prove to be less durable than expected.

Faced with the task of lighting up our car numbers, I chose to build rather than buy my way out of the task. Armed with a new 3D printer and free parametric CAD software, I came up with this (picture from a pre-installation test):

Note that step 0 should be to have highly visible numbers to begin with. We switched to black-on-white numbers for the 2022 HPR 24 hour race anticipating that our numbers would need to me more visible. I am considering using retroreflective white vinyl also to remove any possibility of the numbers not being visible. I have also added a lower light bar to our setup – this may not be necessary especially if used with retroreflective vinyl.

Lighting is provided by an LED strip from Amazon that has a thin strip of VHB tape to affix to the light bar:
https://www.amazon.com/dp/B00RT2WLQS?ref=ppx_yo2ov_dt_b_product_details&th=1

They are fastened to the car using M5 rivnuts and fasteners, 16-20mm should be long enough. Post-installation I don't see the need for the 4 fasteners in the design, I would say its safe to skip the two middle fasteners. I used Loctite on the fasteners because racecar.

I also used VHB tape as an enhancement to the fastener solution:
https://www.amazon.com/dp/B007Y7EWRO?psc=1&ref=ppx_yo2ov_dt_b_product_details
Put a strip of this on the surface that mounts to the car, it does double duty as a gasket and attachment point.

I spent an enormous amount of time tweaking parameters for this specific application. It is likely your car is different and will require something as unique. To that end, in addition to STL files for my printer and car, I have included a .scad file that can be used with OpenSCAD. There you can tweak parameters in the Customizer window, all of which I will cover here.

First things first: To print my canned STL files on your 3D printer, make sure it has a vertical build area of at least 160mm. Since they print vertically, make sure your slicer adds a raft at the bottom for stability. The two pieces are designed to be joined together in what should be an obvious fashion, I use super glue to stick them together.

The STL files create a 310mm long light bar that the referenced LED strip fits into, tiliting the light at a 15 degree angle toward the car door. This is what I settled on for my car, which has a Lemons-legal 12" high 3-digit number. Your car is probably different, so keep reading for how to customize your own solution.

I was introduced to OpenSCAD by the TDISore crew chief, Chris, when working on a previous attempt to light numbers using trailer license plate lights (which went about as well as it sounds.) OpenSCAD is a parametric computer aided design program, meaning that rather than drawing something in a tradition CAD program, everything is coded using a domiain specific language. Here is a screenshot of the .scad file in action:
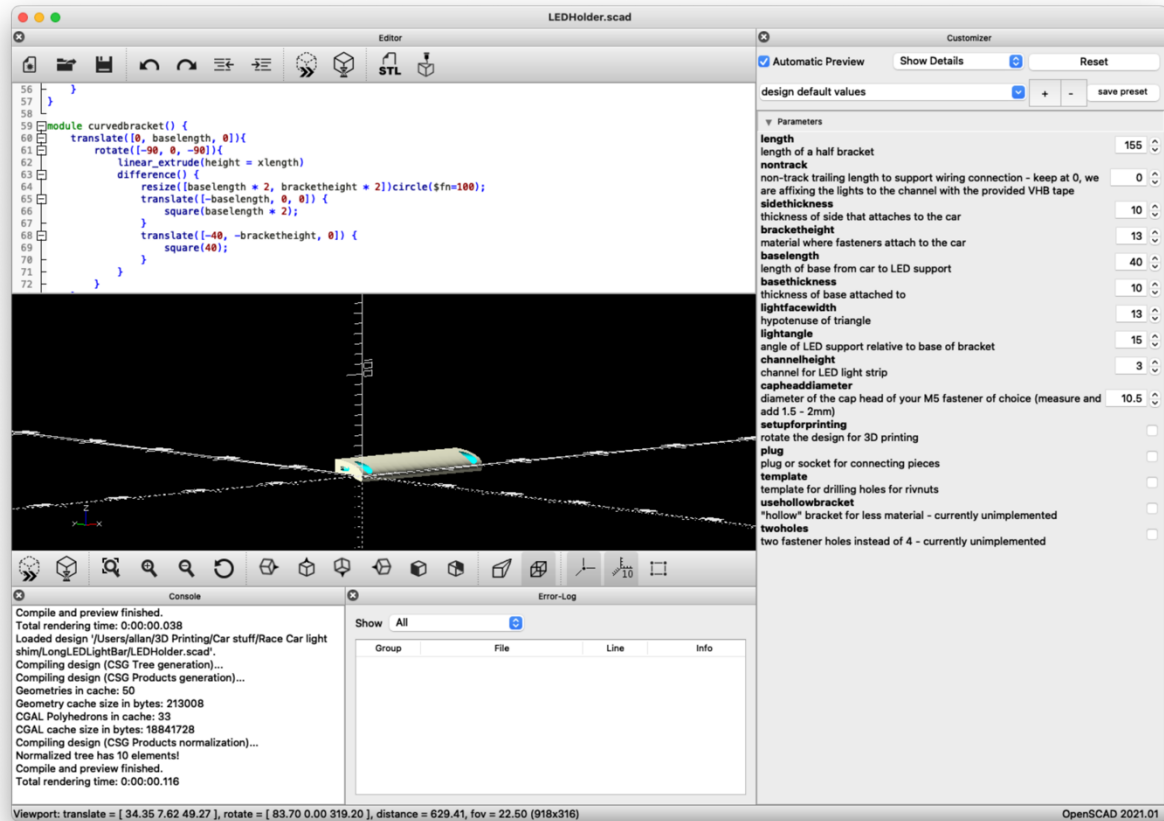
The top left corner contains the Editor window. Technically this is where you can edit the code that describes youyor model, but OpenSCAD supports the use of external editors, and if you want to get the most out of OpenSCAD I would recommend doing so. Visual Studio Code with an extension is what I use:
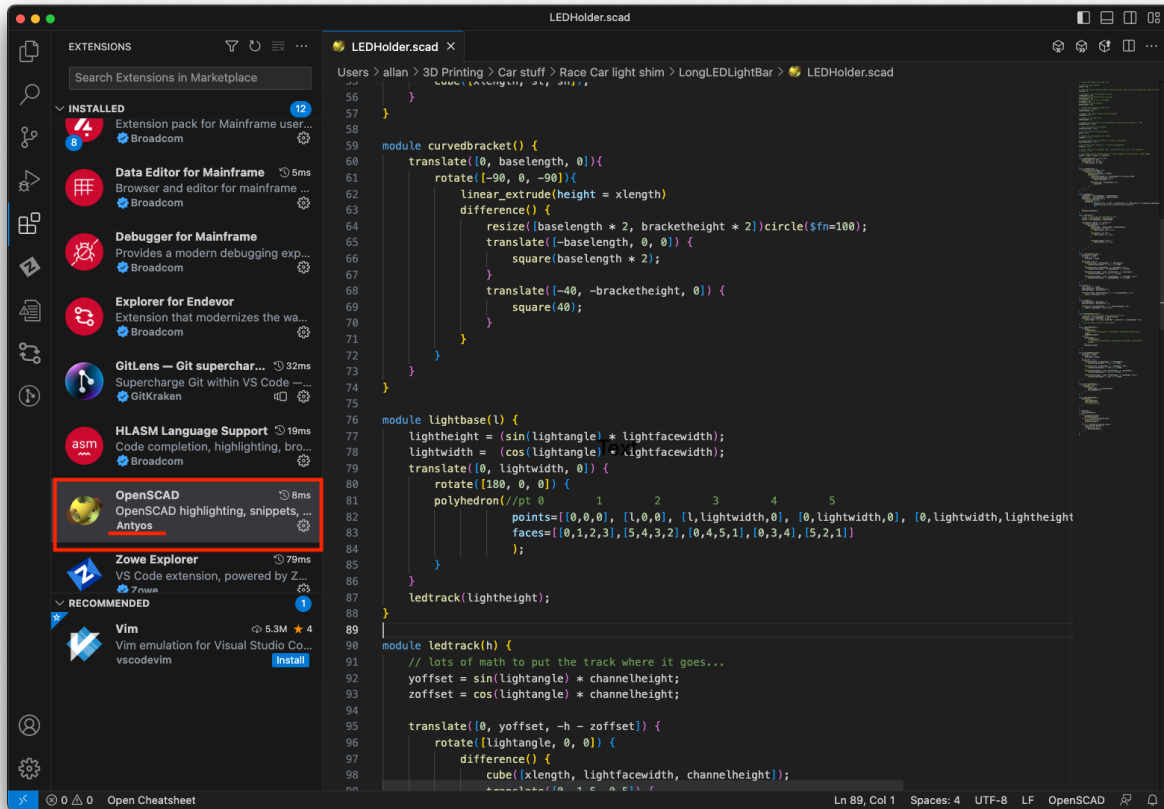
If you are not planning to build something from scratch and just need to tweak some parameters, you need the Customizer window:
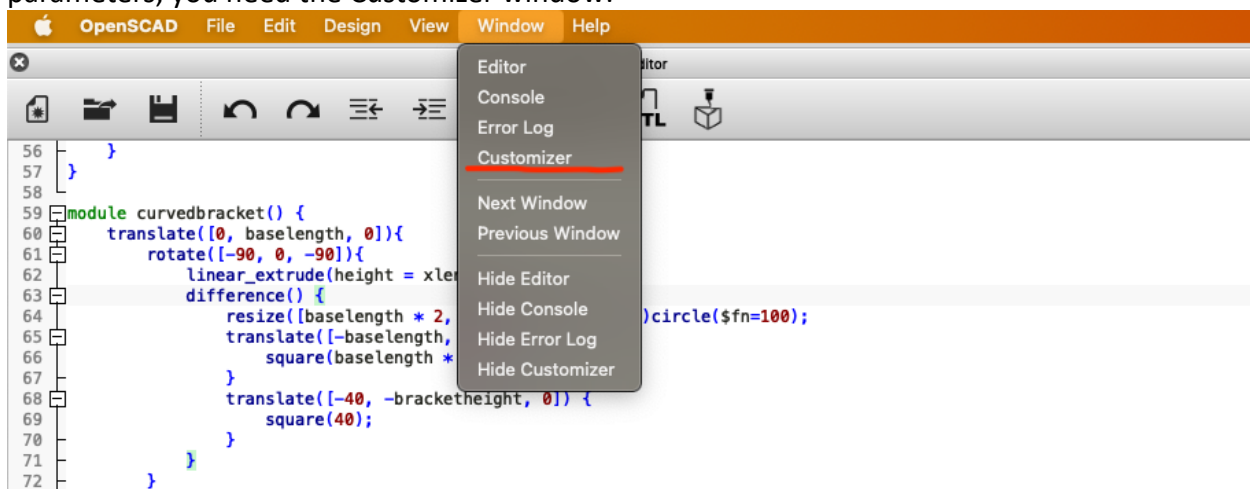
Here is a closeup of the Customizer window for the light bar:
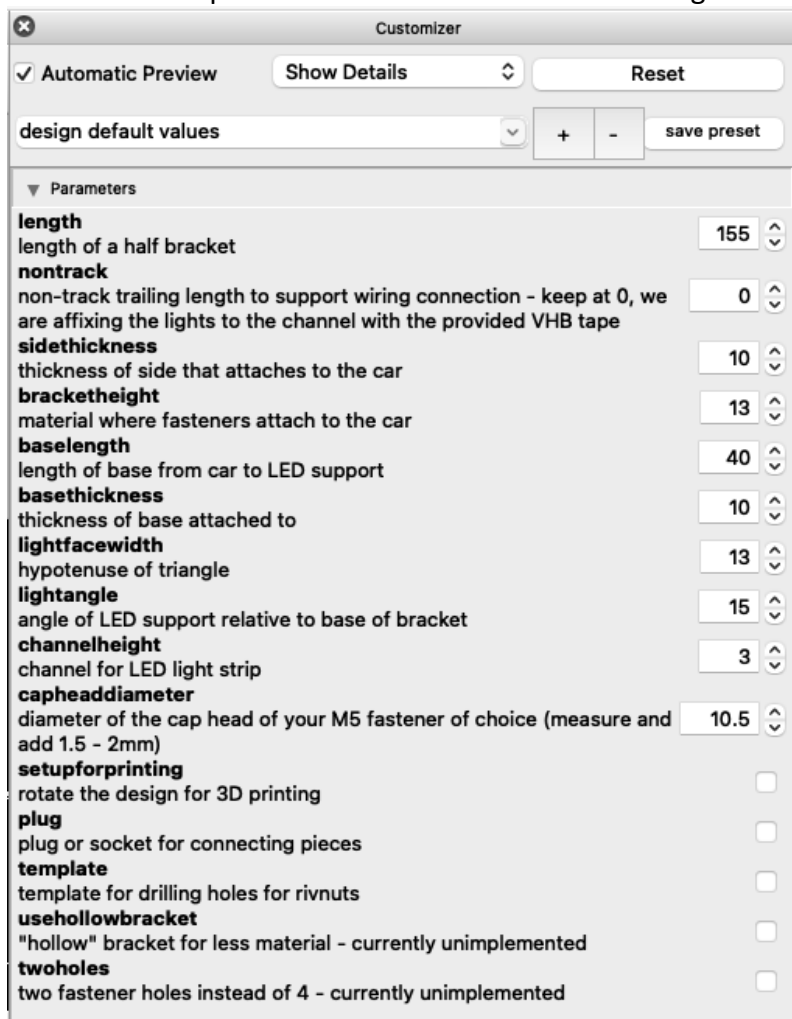
There are some that should not be changed: lightfacewidth, channelheight, and nontrack come to mind. These may be removed in a future release.

Assuming you read this far, experimenting with baselength, and lightangle should get light pointed at your numbers in a suitable way. setupforprinting orients the part in a vertical orientation suitable for printing, and plug toggles between the two halves of the bracket. template will show a drilling template for rivnuts or bolts.

To export to an STL file, first render your parameter changes and then export to STL. The two buttons to do so are below.

Note that there are some unimplemented features that I am working on, and by that I mean occasionally revisiting.

Hope that explains everything – the STLs and .scad file will be on Github for download at https://github.com/lingoslinger/LEDLightBar

I will answer questions as I am able to. Thanks for your interest!