![Adafruit logo]

Ember 3D Printer

Created by Ruiz Brothers

https://learn.adafruit.com/ember-3d-printer

Last updated on 2021-11-15 06:25:10 PM EST

Table of Contents

Overview

Today we're taking a look at the Ember 3D Printer from Autodesk. We've been printing our projects for several weeks and wanted to share some workflows and techniques.

This is a DLP, SLA 3D printer, so its completely different than FDM 3D printing. Here's a basic look at how it works.
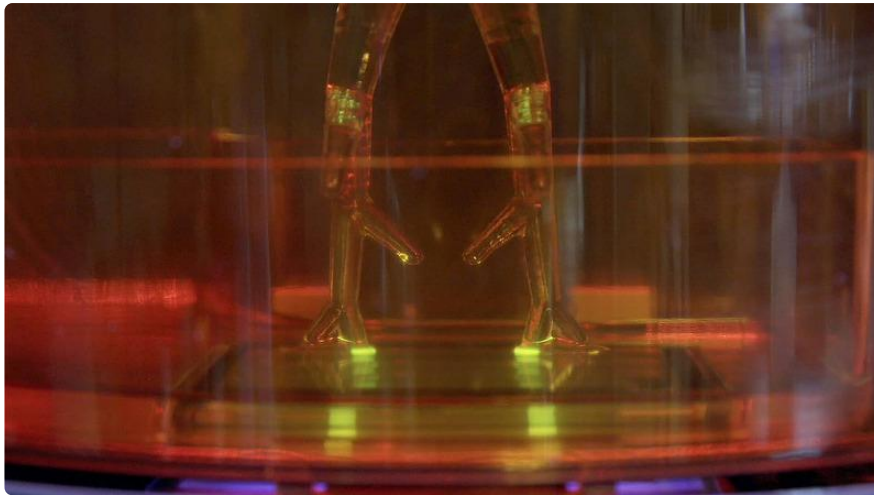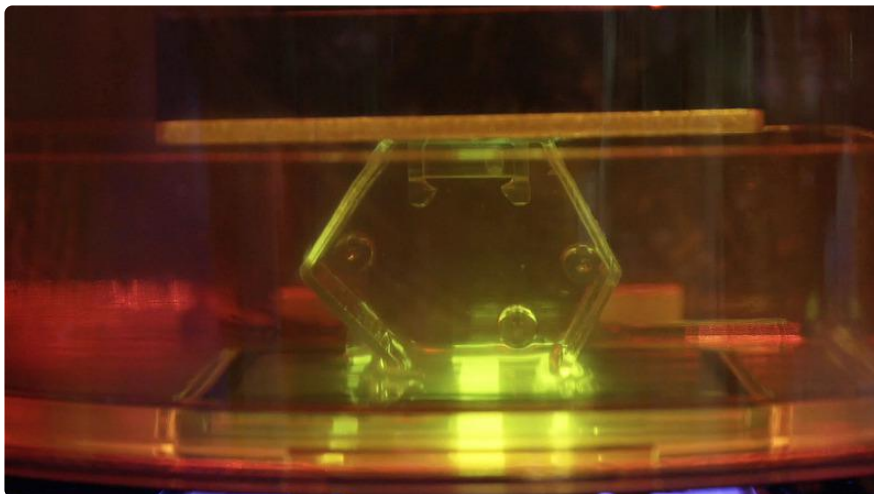
How DLP SLA Works

This printer uses a DLP projector to cure photosensitive resin using UV light.

The projector shoots UV light as a full slice for each layer. This allows it to print in resolutions can be as fine as 10 microns!

A metal build plate moves up along the z-axis as the print emerges from the resin tank.
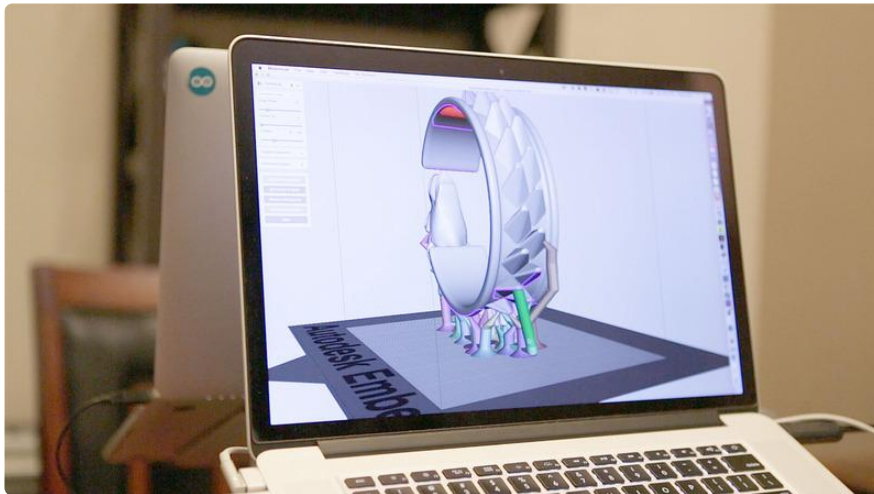

The ember printer uses a cure and peel technique. This exposes and separates for each layer to create a high definition part.

When UV light is exposed to the resin, it is hardened and solidified. The resin tank separates by moving away from the build plate and returns for the next layer.
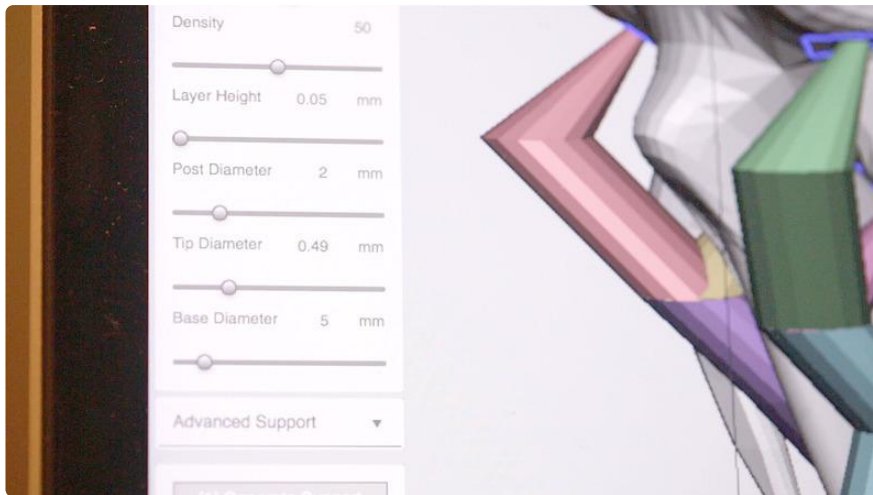
Print setup

Orientation

Setting the proper orientation for an object can be a little tricky - you'll want to test different angles to get prints to stick to the build plate. Also keep in mind parts are printed upside down - because the build plate is oriented that way when printing.

Tip diameter

You can optimize support structures to print with a tighter hold by upping the tip diameter. Under the support generator options you can update the defaults from .15mm to .5mm, this will increase the width so it holds objects more firmly - making print fails less likely. Can can also add your own custom supports to accommodate for any overhangs.
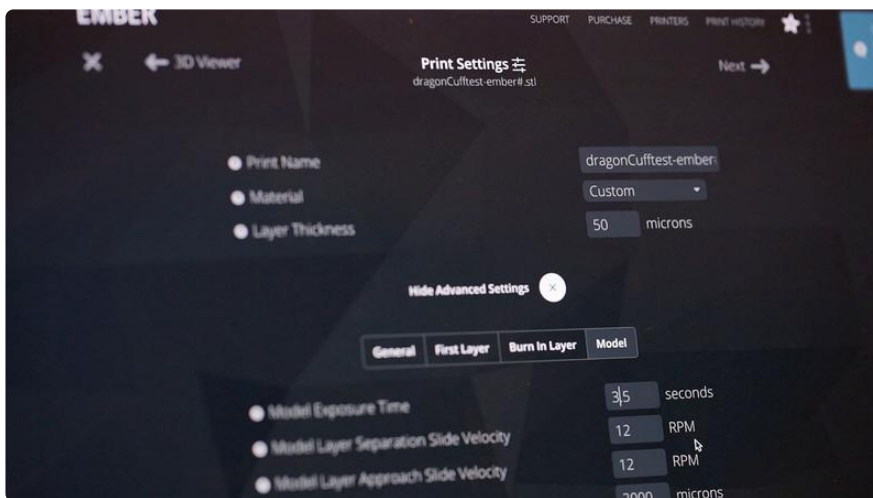
Resolution

Next up we'll need to slice the models for printing - this is done through the ember printer website. Here you can preview models to make sure everything fits correctly. To speed up prototyping we can print at a lower resolution of 50 microns and then adjust the model exposure time to 3.5 seconds.

The slice preview lets you visualize how the part will be printed and is actually just a stack of PNGs. You can also monitor your print job from any mobile device.

Fixing the Hall Sensor

If your printer can't align the tray, then you may need to adjust the hall sensor. This is easily done by opening the front cover and adjusting the screw nut on the sensor so that it protrudes 2mm above the tray floor. Watch the video above for a walk through.

Processing

Workspace

To operate the printer, you need to handle the resin with safety in mind so you must wear nitrile gloves, safety glasses and lab coat or apron - you know, safety first.

Each print needs to be cleaned and processed, so it can be a little messy. This requires a cleaning station with handy tools that will be specialized for handling resin parts.

So your tools, containers and anything you touch can get resin on it, so you need to be cautious.
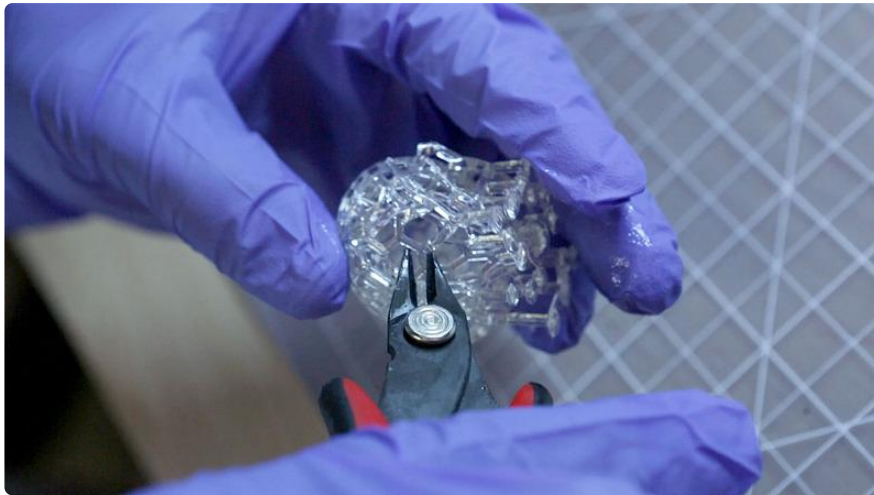
Support remove

To remove support material, we recommend using flush diagonal cutters. Start by snipping off the tips from the supports and then removing branches as large groups. Try to cut the tips away from the part to prevent any surface damage.

Wear a face shield to protect yourself from any flying bits when breaking apart smaller pieces. Take your time when cutting complex structures. If you have support structures inside your model, you can use tweezers to hold them in place while snipping them into smaller bits.

Without Supports

Enclosure parts smaller than 30mm can actually print without any supports! STL models are prepared using Autodesk's Mesh Mixer. Here you can set up support structures to hold models to the print head.

Curing

After parts are printed, we need to fully cure the surface by submerging it in a plastic jar full of alcohol. We need to let it soak for about 5 mins before taking it out and patting dry with a paper towel. Rough edges and support bits can be sanded away but this leaves a chalky look - this can be fixed with a bit of mineral oil - thanks nature!
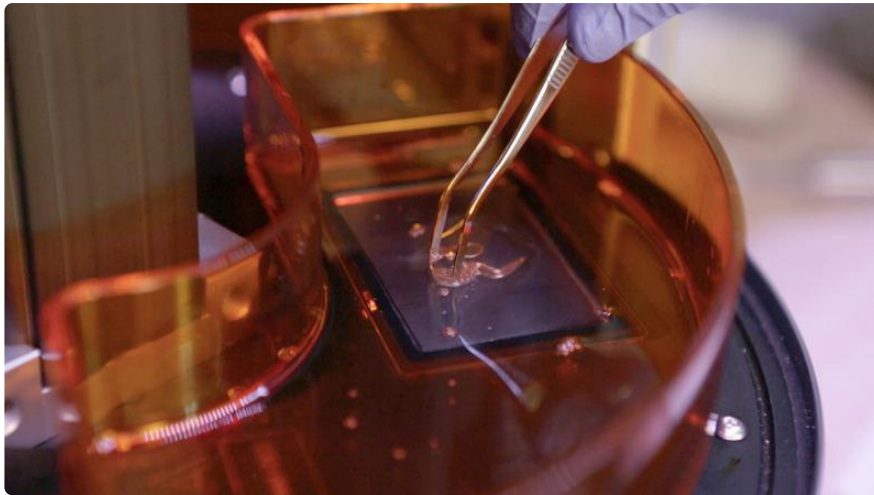
Maintenance

Keeping the resin tank clean is important. As prints fail, material builds up and floats around the tray. These can be difficult to spot, so we can use a hair comb to filter out any pieces left behind. Use tweezers to fetch out any bigger pieces.
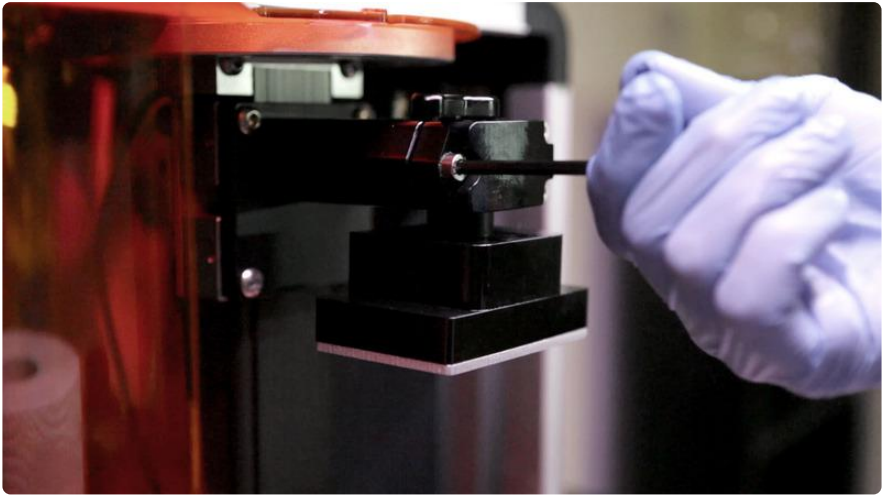
[Download Comb STL]

https://adafru.it/eET

Carefully swipe around the edges of the resin tray window and remove any build up.

If theres a lot of bits and pieces, use a fine mesh strainer to quickly filter out smaller debris into the second tray. The build head is easy to clean after each print, just wipe off any remains.

Leveling

We'll need to relevel the build head before starting a new job - mainly because it misaligns after removal. So we can loosen a the build plate and move it up, and then we'll tighten up the screw.
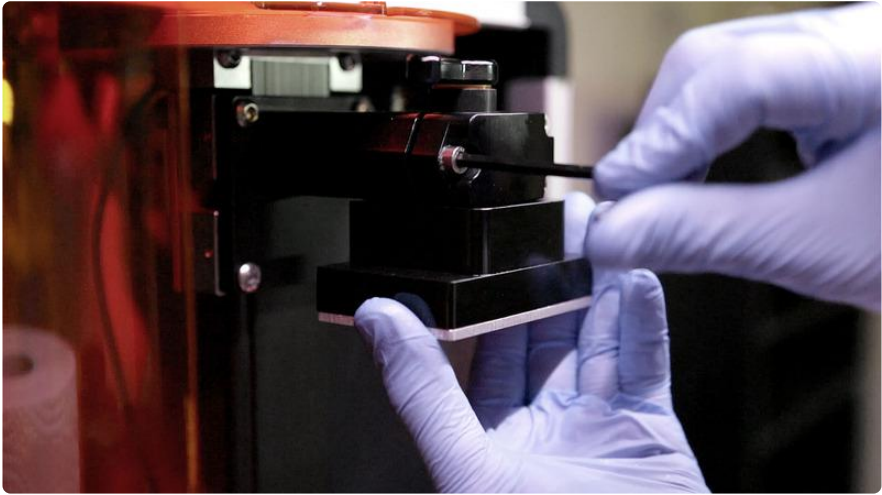
Then we'll initiate the leveling procedure - It'll start homing the build head down until it reaches the resin tank window.

From here we can loosen the build head - let it drop down and push on all four corners to lay it flat. Once its all straight and level, we can tighten it back up and complete the process.