How we designed an injection-molded case

Created by Mike Doell

https://learn.adafruit.com/how-we-designed-an-injection-molded-case-for-raspberry-pi

Last updated on 2021-11-15 06:13:08 PM EST
Hi there, Mike Doell here! I designed the Adafruit Pi Case and took a few screenshots and photos along the way. This tutorial will go through the process I undertook to design this injection molded case.

Gather all of the relevant cables, sd cards etc that may be used with the board. Put all the things that may be plugged into the board in place.

This gives us an idea of what to look out for as we generate ideas and highlights possible interference between the case and the parts.
Time to start measuring components and building them in CAD. Here is a drawing of the raspberry pi that was built in 3D, to use as an underlay while designing the case.

Sometimes you're lucky and there's a CAD file available to use as a 'negative' - but sometimes we're not so lucky!

Next step is to build and assemble all of cables in 3D. This helps make sure we have plenty of clearance for overmolding, etc. Try to get the chunkiest cables people may end up using.
Design

Once the components are in position, ideas are generated by sketching over printouts or building simple shapes directly in CAD.
After the parts are mocked up in CAD the first parts are printed for a fit check.

At this point we are looking for potential interference and evaluating details like parting lines.

This was an original revision, without the sloped top.

You can use a 3D printer for this, or send it out to a 3D printing service. 3D printers have gotten good enough to provide snap-fit resolution!
Tweak, print, repeat.

Now the fun starts! Refining fit, adding/removing details. Getting the 'feel' of the case.

Making sure you have access to all of the connectors can be challenging - especially with a board like the Pi that has connectors on all sides!

Render

After sorting out the fit and finish we start rendering to figure out the materials and colors. Different materials will affect cost and mold design so it's best to get a good idea of what you want early.
Creating an injection mold

This is the steel we will use to create the inserts for the injection mold. We create an insert for the core and cavity (top and bottom in this case) of each part.

The shapes are roughed into the inserts on the CNC.
Fine details like snap catches are burned into the inserts using a copper electrode on an EDM machine. This gives detail that is too small to be machined with a cutter.

---

**Mold testing**

This is the core side of the tool, it creates the inside of the parts. This is also the side that the ejector pins are on. The pins push the part out of the tool and they also leave the small round dots you see on the inside of a molded part. The hoses are used to move water through the mold base and keep everything at a
constant temperature.

This is the cavity side of the tool, it creates the outside or "show side" of the parts. On opaque parts this is the half that gets polished. *Unless your part is clear then the core and cavity will have the same finish.*

Once we get the tool installed we create our test shots. They are typically clear, it makes it easier to see what's happening during fit tests.
Final fit check

Final fit check with board in place.

We always make things "tool safe" which means that everything is slightly loose.
It is easier to remove material from the tool by machining when you need to tighten it up.
When the parts are too tight or there is an interference you will need to weld the tool (adding material by welding) and remachining or by cutting a pocket in the area that needs to be changed and filling it with material that can be machined again (referred to as "inserting" the tool).
Finished part

Final production parts.

If you look closely at the corner of the clear cover you can see a small round dot. That's from the ejection pin when it pushes the part out of the tool.