Printing hasn't actually reached the point yet where you can just press "print" and get as good an approximation as is physically possible, in the minimum physically possible time.

I have been having problems with a batch of defective filament, the viscosity is too high, so I don't have good recommendations for feed rate.

However I can tell you that you should do an extrusion test, just print a cylinder or rectangle with grid infill or something and determine the maximum reliable extrusion rate of your nozzle, in cubic mm per second. Set the walls something like 70 percent of this, and the infill 100 percent, or something. If the linear speed is too fast, the infill will show through and other defects appear, so the speed needs to be reduced.

Secondly, you have to increase the line width for infill, I use 1.8 times the nozzle diameter, to prevent what I dub shadowing. I haven't heard of this from other authors, however it is a serious issue that greatly reduces the part strength. It's what happens when the nozzle crosses a road that has already been laid down, and the lee side doesn't get enough plastic. If the road being crossed is wide enough, pressure builds up in the nozzle and plastic spurts out and things are fine, though, apparently. The linear speed of the nozzle is not the major factor though it might be a factor at some speeds. You have to make the linear speed of the infill so that the max extrusion rate is not exceeded.

Most people are not acquainted with talking about extrusion rates with printers, but this appears to be ignorance.

Most consumer printers will need to have their perpendicularity compensated for, google for YACS square, use the tangents method, that is the type of data that is commonly used by Goskew, and also the marlin firmware. You will need 3 data points, make sure you save them in a safe place and don't get them mixed up! Print another yacs to make sure, it will save you time in the end.

The diameter of filament has an impact on accuracy and can cause problems with fit. You should print a cylinder and a small square and stuff to check your printer for accuracy before undertaking large prints. I can share the test part I printed, I will make a note to do that.

I always scale parts up by 100.1 in x and y to compensate a bit for shrinkage.

You need to use a bed adhesive for the larger parts and maybe for the smaller ones.

I print in PETG because it is more temperature resistant, PLA is great except it softens at 50 Deg C, so the machine could get destroyed if left in a hot car or in some areas on a hot day in direct sunlight.

All parts should use a brim at least 6 mm wide, I use 8 mm, or they are not likely to stay stuck to the bed. Especially the bigger parts.

Make sure there is nothing on the edges of the build plate for the larger parts. I had to get rid of the paperclips on my printer at the edges of the build plate and glue the glass plate down with shoe goo. It's inconvenient because it's a lot harder to get the parts off, but it's the only way to print the big parts.

Wait till everything has cooled before you try to scrape parts off, that makes it a lot easier as the differential thermal contraction causes stress between the plate and part that works to your advantage.

I have been using glue stick (which is PVA mixed with water), elmers brand, works better than the no name brand, and is easier to clean. However I have had some success with making the first build layer stick using a build plate temperature of 90 degrees, instead, and this warrants more investigation. It certainly works ok for small parts, but I don't know if it will work for the most demanding parts like the base component.

I use a box cutter fully extended from the handle, the sharp edge makes a good scraper. Rock it back and forth in the xy plane to aid disconnection of the part from the plate. Be very careful not to exert excessive force on any part of the printer, or slip and damage anything. Personally I don't like this parts, especially with the glue stick.

The glue stick is kind of messy and hard to get off, and it gets everywhere. I would like to eliminate the use of it using the elevated build temperatures.

An initial layer build temperature of 70 helps bed adhesion with the glue stick, then I drop the build plate to 50 after, that's a hold over from PLA printing and may no longer be helpful with PETG. I use glue sticks from the dollar store, which are PVA mixed with a small amount of water.

Make sure you have enough filament before starting each print. Weigh the roll when it first arrives and write the weight on with a sharpie, ideally, so it's easy to determine how much filament is left. Or you can plan the whole process using CURA, but I have found that bafflingly the material use estimates by cura are not always accurate, they are usually too high actually.

Always check the preview in cura, always. Make sure there is the brim, that support material is where you want it to be, that there is not likely to be any areas that don't have support which need support.

It's not a bad idea to check the gcode with gcode.ws, either, if you use goskew. Sometimes Goskew causes errors that cause some defects, like the infill showing through on some sides. Using the built in marlin perpendicularity correction is probably a better idea than using goskew.

I recommend checking literally all setting for each print when making new cura files, scroll through that whole dialog and double check em all, then save the cura file every time with a

sensible name in a folder for that build. That way you can troubleshoot, and if it goes well you can repeat the stunt by reusing the cura files exactly as they were. It seems like more work, but it will save work overall if you are making more than one unit, and possibly if you are making only one, too.

To remove prints I use a long box cutter blade, there may be better scraping tools but this works pretty well. You have to rock it back and forth and it takes a while, and is kind of stressful, but I have found no other way. There may be better bed adhesives that solve this problem.

You should use a layer height that can form integer millimeters, so 0.25 is ok, 0.333 is good enough, 0.2 is ok, but 0.15 is not, for instance.

You need fresh bed adhesive every time for some reason, I use a paint scraper to scrape off the old stuff. The bed is stuck to the printer so I can't just wash it with water.

Extrusion rate:
For a 0.4 mm nozzle, I am only getting 3.15 cubic mm per second, at 240 degrees, to reliably print without issues. That's for infill, reduce it by about 30% for walls to improve quality. This is lower than it should be according to the community. Some PLA I've tried got more than 4 times this rate.

The 0.8 mm nozzle does about 5.5 cubic mm per second for infill.

Base:
I print the base with a 0.8 mm nozzle, 0.8 mm road width for walls, 1.28 mm wide infill (otherwise you get shadowing defects), 0.25 layer height. You can use a 0.4 mm nozzle, it doesn't save as much time as you'd think to use a 0.8. It seems to save about 30% of the time, for the same infill and wall thickness.
No supports, the support material gets in the holes otherwise and has to be drilled out.
Single wall thickness is fine, cubic infill, I haven't tried others.
There is a compression spline for the bearing to go in, you may wish to pause the print after that part has been printed and make sure the bearing goes in ok. Removeingthe bearing might be impractical, it should be possible to just leave it in place. If it is too big, glue could be used to fix the bearing in place, if it's too small, it could be sanded. So it's not that critical, actually.
It depends on how well calibrated your printer is.
Make sure your bed is not wobbly, that can cause the nozzle to catch the infill near the end of the build, and cause print failures, and even damage the machine.

Regen media holder:
Enable "print thin walls".
This is the hardest component.
Scale everything up by 100.1 on the xy, this applies to all parts but esp here.

There are two models, the grid bottom and the regen media holder walls . Import them into Cura and center them. Set the default print parameters for the media holder walls/axle. Then select the bottom grid and using the menu thing on the left, the fifth icon down, select per model settings, to change the settings for that object. Select grid infill, 4mm line spacing, 0.72 infill line width, 0.45 wall road thickness, slow the linear speed of the infill to match the max extrusion rate of the wall (make it so the extrusion rate is suitable), and say 0 top and bottom layers, so it prints no top or bottom for the cylinder.
Check it in the preview. It should merge the two models into one automatically.
Give the rest at least 2 wall thickness, 25% infill seems to be ok.
You really need the brim and bed adhesive mentioned previously.
The brim should be only on the outside, there is an option that turns this on and off.
Double check everything in the preview, make sure the central column is going to be strong enough, the walls all get printed, the grid is ok and will bond to the rest of it.
Optionally, there is a component in the CAD model called the axle strength support blocker, which is also exported as an stl. You can use this to cause the walls at the top and bottom of the thing to be extra thick, infill higher, etc. however it is optional as the stresses are not really high, except for when it's pushed into the bearing, and/or when the bearing is pushed into its socket. I originally made that for use when I was using a stepper motor and the interface where it met the stepper motor needed extra strength.

Top grid: this is the same deal, the cad model isn't the actual geometry you want, because fusion isn't capable of handling such complex geometry. You have to just slice that object with the right settings, grid infill, make the lines 0.72 mm wide for infill, no top and no bottom, 4 mm line spacing.
Check the preview to be sure it's doing what you want, adjust and try again until it looks good
.

Seal plate:
The seal plate can use the same settings as the regen media holder, it might benefit from two wall thickness. The tolerance of the compression spline matters some but none of the tolerances are critical because we can use grease in the seal to fill in the gap. This allows a large gap to be used in the CAD model, so there is lots of clearance, so if the parts are not quite the right shape everything still works.

Ducts:
These parts are easy to print and might as well be done with the 0.8 mm nozzle. The walls are 1.5 mm thick 0.8 multiplied by two would be 1.6, cura 5.0 can adjust the road widths automatically but I have never been clear if it was doing this or not and haven't checked. It makes nearly no difference.

Motor mount:
Again, also easy to print, can be done with the 0.8 mm nozzle but I prefer the 0.45 for higher quality.

Plugs:
Needs the detail of the 0.45 mm nozzle, otherwise anything is fine. Might as well use 100% infill or thick walls.

Motor cover:
Again 0.8 mm nozzle is fine. Supports are needed for the screw seats.

Pulley:
100% infill, 0.45 mm nozzle, road width of walls has to be 0.45 or smaller to get the detail, other details aren't really important. It might end up slightly too small or too large, in which case scale it, this is the part that needs the most precision, but it's small enough you can try more than once to get it about right. You can also print several in the same print job, scaling each one slightly differently, and pick the best one. It should just fit into the hollow shaft of the motor when you push it in.

Rain covers:
There is only 1 stl, but you need to print 2 identical ones. Again, not a hard component to print, might as well print them with the 0.8 mm nozzle if you can. Notice the screw seats are made in such a way that they don't require support material.

Combining parts onto a single plate:
I haven't tried this much, but it would make a lot of sense, especially if you are making more than one unit, to group parts into batches that minimize the number of times you have to load and unload the printer, especially if you are using the bed adhesive as it has to be scraped off and reapplied every time.