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Designing a Simple 3D Printed Rubber Band Car Using FreeCAD

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Located at: <http://www.youmagine.com/designs/designing-a-simple-3d-printed-rubber-band-car-using-freecad>

Short description:

My foray into designing with FreeCAD.

Description:

<https://youtu.be/L9BiA8X-ji0> As a retiree, I design and publish 3D printable mechanisms for entertainment, educational and hobbyist purposes here and on other websites using Fusion 360, as well as provide free consulting to local companies with their use of Fusion 360. Changes to the Fusion 360 "Personal" version I use have made it difficult to support local companies with the Fusion 360 "Commercial" version they use, so we decided to try FreeCAD. My first foray into FreeCAD is a redesign of my Fusion project "Designing a Simple 3D Printed Rubber Band Car Using Autodesk Fusion 360". So if you're interested in designing with FreeCAD, stay tuned as I upgrade some of my previous designs from Fusion to FreeCAD, and create new designs as well! As with any 3D printed mechanism, prior to assembly, I test fit and trim, file, drill, sand, etc. all parts as needed for smooth movement of moving surfaces, and tight fit for non moving surfaces. Depending on your printer, your printer settings and the colors you chose, more or less trimming, filing, drilling and/or sanding may be required. Carefully file all edges that contacted the build plate to make absolutely certain that all build plate "ooze" is removed and that all edges are smooth. I used small jewelers files and plenty of patience to perform this step. As usual, I probably forgot a file or two or who knows what else, so if you have any questions, please do not hesitate to ask as I do make plenty of mistakes. Designed using FreeCAD, sliced using Ultimaker Cura 4.7.0, and 3D printed in PLA on Ultimaker S5s. **Wheels.** <https://youtu.be/xXa-eYr4Zhg> **Axles.** <https://youtu.be/THsnVfGp05A> **Chassis Sides.** <https://youtu.be/7T9PXCJ9AYM> **Final Assembly.** 3D printing and assembly of this model is quite easy. I printed two "Chassis Side.stl", two "Axle.stl" and four "Wheel.stl" at .15mm layer height with 20% infill. Next, I pressed the two "Chassis Side.stl" together and if loose applied small drops of thick cyanoacrylate glue. Then I stretched the O-Rings around the wheels, two O-Rings for each wheel. Finally, I positioned an axle in the chassis assembly then pressed two wheel assemblies onto the ends of the axle. If the wheels are loose on the axles, again apply small drops of thick cyanoacrylate glue, then repeated this process with the remaining axle and wheels. Apply the rubber bands as presented in the original video, then off you go! And that is how I designed a simple 3D printed rubber band powered car using FreeCAD. I hope you enjoyed it!

If you can, please use the online documentation found at <http://www.youmagine.com/designs/designing-a-simple-3d-printed-rubber-band-car-using-freecad> because those may have been updated. Also, there you can interact and provide praise and/or feedback.