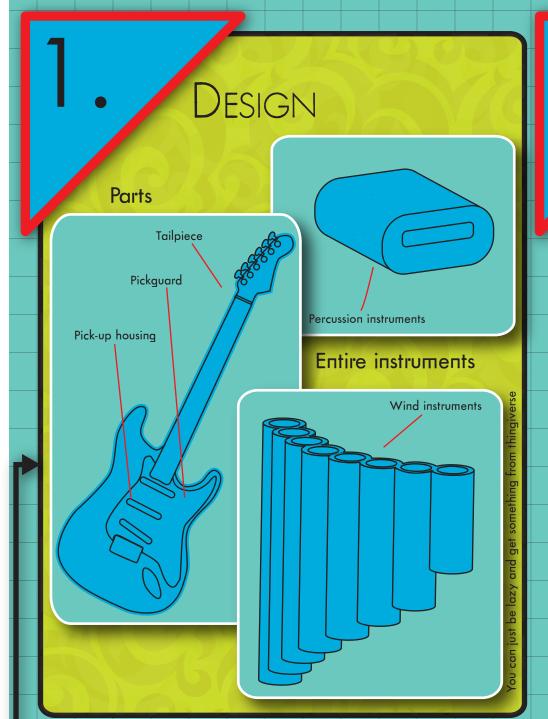
PRINTING MUSICAL INSTRUMENTS WITH A REPRAP



3D MODELLING

Software

Sketchup Solidworks or OpenSCAD > Free

Guidelines

Check and fix > Software: NetFabb Studio Basic (Free) > Check for the correct dimensions and

rotation relative to the printing bed

> (Automatic) Repair > Export .stl (repaired)

> Easy to use

printer.

When making

thickness a

layer-height

thin-walled object, make your wall

multiplier of your

> Accurate

Get to know your printer

first, so you know what

Be careful of overhang

level of detail works.

- > Limitations in shape > Good for technical parts → Difficult to learn Rhino or Blender
 - > Freedom in shape
 - > Allows organic shapes > Expansive plugins

Be careful to model within the size limitations of your 3d

Materials

⇒Stonger Higher temperatures

пеедед **PLA** >More flexible >Natural

Different colors of filament will have different printing properties

Export to .stl Lower quality equals faster

SLICING

Software for slicing include Slic3r, Skeinforge and Cura. Each of these have their own speciality, but developments are so fast, that no recommendation can be done. Just try several and use whichever feels like it produces the best results for you. This guide however, is written based on Slicgr.

Things to think about in printing settings:

Layer height »While a smaller layer height will increase the detail in your object, it also increases the printing time. Number of shells

»A single horizontal shell will give the fastest printing speed, but might leave you with a weaker object. »Multiple vertical shells are always recommended, as the bottom and top layer are usually where problems arise. >Infill density

»A density of 1 means that the product is solid, but the printing time will be a lot lower if you decrease it; start with 0.3 and see whethe to go higher or lower from there. >Infill pattern

»Several options are available, honeycomb or rectlinear will be both strong and fast to slice and to print. Skirt

»This will draw a line around your object, allowing the extruder to flow through and ensure correct extrusion on the

»If your object has problems with sticking to the bed, it can be useful to set a brim, don't over use it though, because you will need to cut it off after printing.

»Faster is usually a good thing, try the limits of you printer with a few test objects. A lot of speed can be gained by having the infill print speed faster than the perimeters.

Note: Save your settings!

1. Open Slicer

2. Open .stl file

3. Check printer settings

4. Check filament settings

5. Check print settings

6. Export the .gcode!

If properly calibrated, the printer settings should remain the same every time you print. But if you want your printed object -notcentered on the bed or if you interchange your extruder, this is where the new nozzle settings should be specified.

> Not only is this where you'll have to make adjustments if you change the filament, it is also where you specify the printing and bed temperature. A useful feature for finetuning your prints is the extrusion multiplier. This setting allows to correct it if your printer is extruding just too much or little material. If your pinter has a cooling fan, enable it! It will inprove the quality of your prints.

Temperatures

»Extruder: 230°C »Bed: 110°C >PLA »Extruder: 185°C »Bed: 65°C Start from these values and adjust from how your objects print.

PRINTING

Things to check:

- Is the filament feeding correctly into the extruder? Is the printing bed secured?
- Is the bed clean?
- Is the bed level? Is your room ventilated?
- Are there no loose parts on your printer? Do all the axes roll smooth?
- Does the extruder home correctly?

For the first few layers, stay close to you printer and check that: The printed material is sticking to the bed.

That the filament is feeding correctly. That the objects seems to be getting the right

The the printer is doing what you expect it to be

If this is all going fine, you can only wait! After the pinter finished, let the bed cool before you remove the object; if you remove your object too soon, it may warp.

1. Open Pronterface

2. Connect the printer

3. Press monitor printer

4. Open .gcode file

5. Check the gcode

6. Check printer

7. Print it!

Choose the right port and press connect.

This will allow you to see the temperatures of the extruder and

> Pronterface will display the size of the printed object as well as the estimated printing time on the right, check if this complies with what you'd imagine. By double-clicking on the graphic representation on the left, you can see what the printer is intending to do. Shift-scroll to go through the layers. If some things seem brong, got back to slicing.

Common problems+solutions

- > Too much/too little extrusion, blobs or holes at the
- » Check if the extruder is operating correctly » Adjust the extrusion multiplier
- » Adjust the printing temperature > Extruder (hobbed bolt) jamming
- » Remove and reinsert filament >> Clean hobbed bolt with a small brush
- > The printer makes weird unnecessary movements » Try slicing again with other settings
- > The objects doens't stick to the bed » Clean the base an/or stick (kapton) tape on it
- » Increase either the bed or extrusion temperature

» Use a brim in the slicing settings

FINISHING

> The finished product likely has some brims on the bottom which can be removed using a knife or file. » Doing this with a machine like a dremel is unadvisable, as the material melts quickly.

> The surface can be smoothed with sanding paper or by using acetone on ABS or a heater on PLA.

Printed object

Printing bed

Roll of filament

Yes --- Is the instrument as you expected it to be?--- No

TFRATING

So, did it sound like you want to? Can you improve it? Whatever your answer is, go back to the start and print more things!

ROUBLESHOOTING

>Not the right dimensions? >Warping? >Inconsistent layers?

Just try again and learn from your mistakes.

If you keep having trouble, you can get help at the awesome communities at forums.reprap.org orhttp://www.reddit.com/r/reprap