

3D Printing instructions for 3D petroglyph tactiles

Thank you for your interest in printing these free tactile resources!

We are happy to provide these instructions to document our creative process and allow others to re-create our products for educational and outreach purposes.

We invite you to share how you're using these tactiles or to ask us questions by sending an email to punchoutreach@gmail.com. We look forward to hearing from you!

Two methods of 3D printing:

- ❑ Send the STL file(s) to a 3D print shop or makerspace in your area.
- ❑ They will likely charge a fee for filament and services, and potentially delivery.
- ❑ Make sure you specify what size you would like the model to be printed at.
 - ❑ The models are 8 inches wide, but you may have to scale them down if your printer's print bed cannot support that.
- ❑ If they have a website, they will likely have instructions listed there.

- ❑ If you have access to a 3D printer, you can 3D print it yourself with the STL file(s).
- ❑ You will need a program called a slicer, which is dependent on your 3D printer, as well as filament for said printer.
- ❑ Most 3D printers use PLA plastic filament that is 1.75 mm
- ❑ Continue using this instructional packet to learn how to 3D print this way.

Step 1: Understanding how 3D printing works

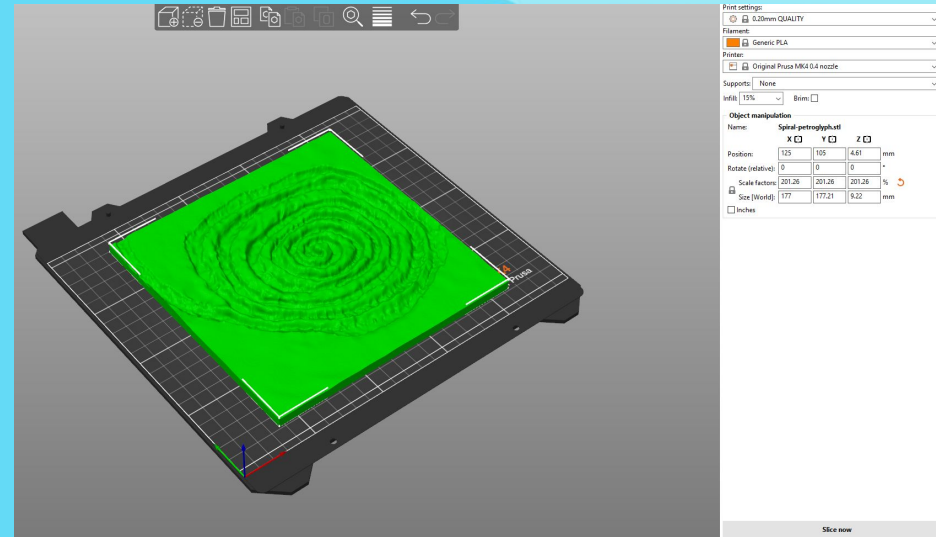
- ❑ 3D prints are objects that are created by extruding thin layers of plastic over and over to build up a 3D object.
- ❑ Much like printing using a normal printer, you must first tell the printer how you want it to be printed via print settings. This is done with a program called a slicer. Once you 'slice' a file, your 3D printer can read and print it correctly.
- ❑ Your 3D printer, if not level or set up right, may mess up! 3D prints can sometimes take a few tries before you end up with the desired product. This is normal.

Step 2: Selecting a slicing program

- ❑ Depending on what 3D printer you are using, you will have to use specific slicing programs to prepare your 3D model file.
- ❑ Some 3D printers have a specific slicer you must use . Others can be sliced using a general slicing program (like PrusaSlicer or Ultimaker Cura) as long as that program recognizes what printer you are using and has its information in its database.
- ❑ When in doubt, you can search the internet for recommendations on what slicing programs are compatible with your 3D printer.

Step 3: Slicing your print



- ❑ Select what printer and filament you're using in your slicing program.
- ❑ Import your STL file.
- ❑ The program will likely visualize what your object will look like on your printer's print bed. Move around and scale the object as needed.
 - ❑ We print the 3D petroglyph tactiles with a width of 8 inches. Your program should scale the model in a uniform matter automatically.





Step 3: Slicing your print (cont).

- ❑ Both petroglyph files do not need supports, as there are no overhanging parts.
- ❑ “Infill” determines what percentage of the inside of the object is solid plastic and how much is air.
 - ❑ 15% - 20% is standard.
- ❑ If you are worried about a large print not sticking to your bed or peeling up at the edges, you can add what is called a ‘brim’.
 - ❑ This will print a thin layer of plastic around your print, and should help with adhesion.



Print settings:

  0.20mm QUALITY ▼

Filament:

  Generic PLA ▼

Printer:

  Original Prusa MK4 0.4 nozzle ▼

Supports: ▼

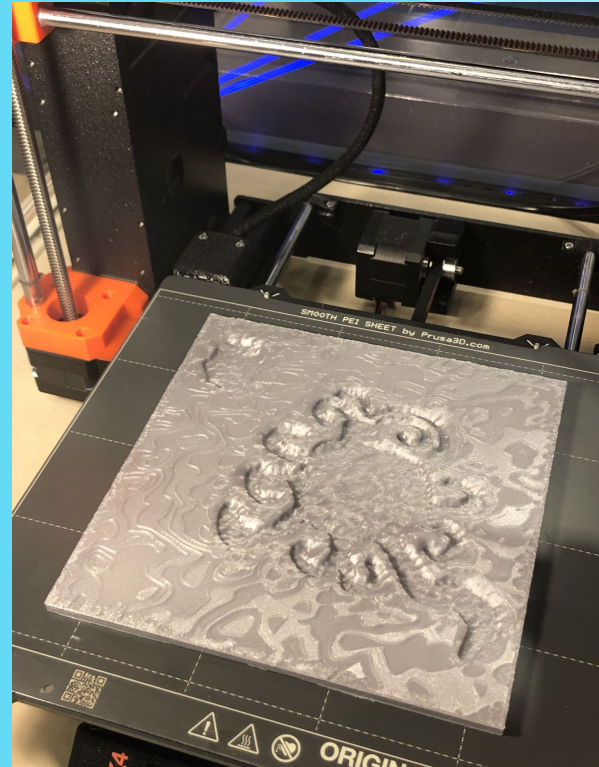
Infill: ▼ Brim:

Step 4: Exporting your print

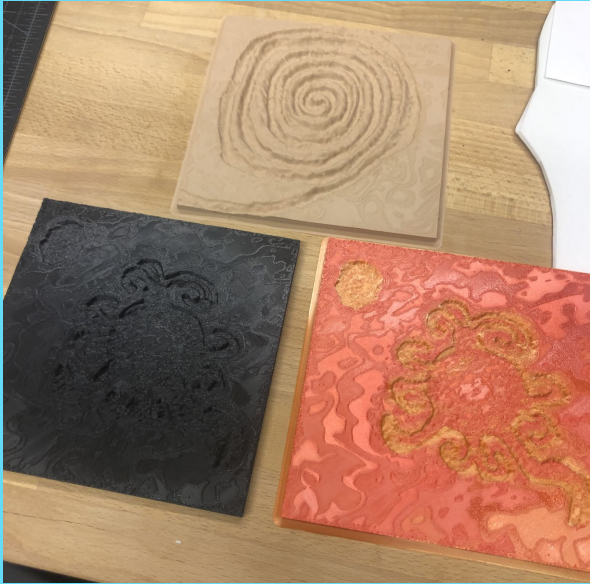
- ❑ Once your settings are to your liking, you can slice your print.
- ❑ Most programs will typically generate a .Gcode file, but depending on your slicer and printer it may accept other file types.
- ❑ You'll want to save this file on some sort of drive that you can plug into your printer.
 - ❑ This could include, but is not limited to, a USB flash drive, a MicroSD card, or SD card.
 - ❑ Some printers can be hooked up to a computer via cable or bluetooth and files can be sent directly, but this is not typically the case.

Step 5: Print!

- ❑ Plug in whatever device you exported your print to, or verify that the file has been sent to your printer.
- ❑ Turn on your printer, make sure that the filament you want to use is threaded into the nozzle, and that the print bed is clean.
- ❑ Find your file in your printer's menu and start printing your file.
- ❑ Most 3D prints are really slow. The petroglyphs can take anywhere around 8-30 hours to print.
- ❑ You'll likely want to watch the printer print out some of the first layer and verify that it looks the correct size and shape.



Sample images using different colors of filament



Tan, Black, and Red / Gold Duo
Tone filament



Cyan and Blue / Green Duo
Tone filament

Credits

- ❑ 3D Petroglyph Tactile STL files created by Caileigh Hudson and Chris Gaines, modified by Will Harris and Kai Hughes
- ❑ Written instructions: Kai Hughes
- ❑ Presentation background: Kai Hughes, inspired by a photo of Chaco Canyon taken by J. Ninneman
- ❑ Presentation images: All photo images taken by Kai Hughes. Digital screenshots are of the PrusaSlicer menu.
- ❑ Special thanks to the PUNCH outreach team and Cherilynn Morrow