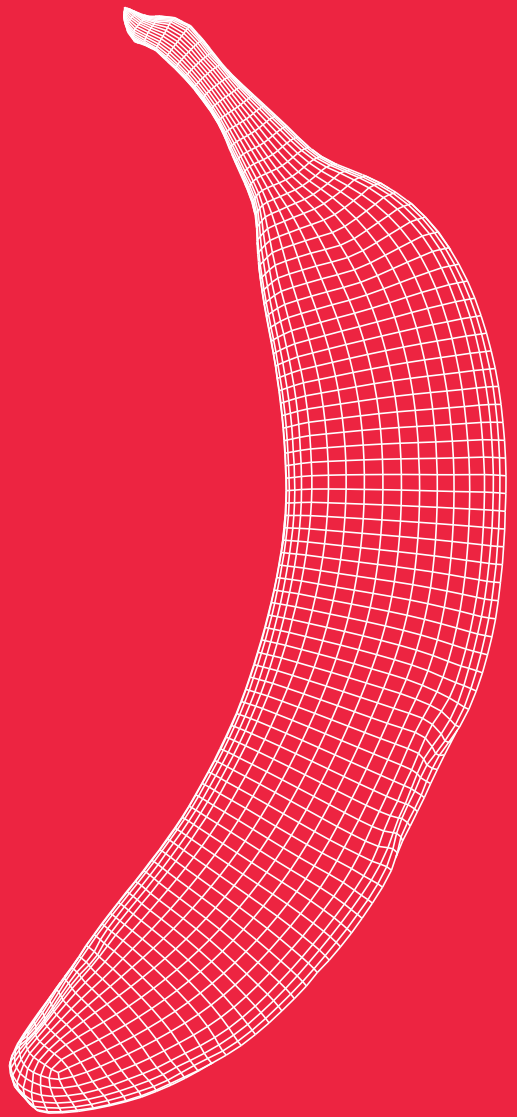
3D Printing Basics

Table of Contents

Document prepared by Emily Carr University.
Shops + Studios, Digital Fabrication Lab.
Content, illustrations and 3D imagery, Logan Mohr.
Design, illustrations and editing Michelle Peters.

Creative Commons License 2024 (CC BY-NC-SA 4.0).



Introduction

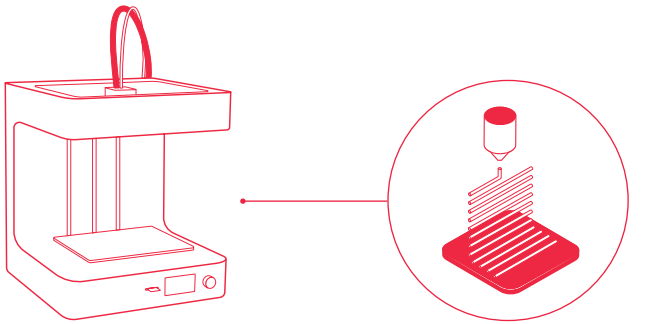
In combination with 3D modeling software, 3D printing is a powerful prototyping tool that can be used to create unique forms and quickly explore ideas through a range of materials. 3D printing is an additive process, in which material is deposited layer by layer to create a three dimensional form.

All 3D printed objects begin as 3D models which can be created with a variety of 3D modeling programs. Knowing how to create a 3D model is the first step in creating a 3D printed object.

Printers

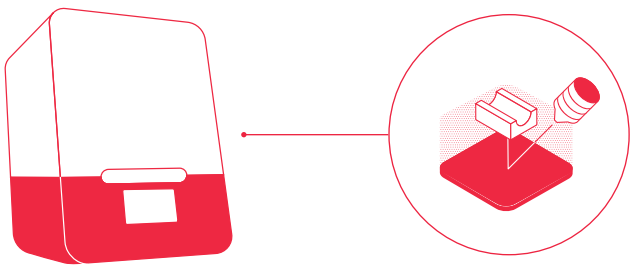
FDM Printers

Fused deposition modeling (FDM) is the most common 3D printing process: thermoplastic filament (such as ABS, PLA or PETG) is heated to its melting point and extruded in layers to build a 3D object. FDM printing is low-cost and great for prototyping.



SLA Printers

Stereolithography (SLA) is another common 3D printing process: curable photopolymer resin is hardened layer by layer with UV light to build a 3D object. SLA printers excel at creating small details and very accurate parts.



Designing a Model

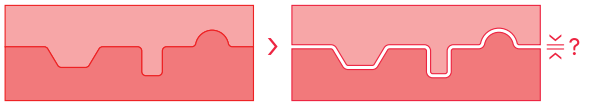
fit clearances
layer height
infill density
wall thickness
draft angles
support material

TO CONSIDER WHEN DESIGING AND PRINTING

Fit Clearances

Clearances are important when creating 3D printed parts that will join together or work as an assembly. A clearance defines how much physical space there is between two parts or surfaces in a 3D model. If possible, it is always a good idea to test print and check the clearances of a design before the final draft is printed.

FIT CLEARANCE ON PARTS THAT WILL JOIN TOGETHER



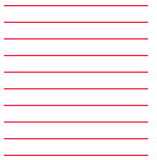
Typical Clearances + Accuracy of 3D Printers

	STANDARD RESOLUTION FDM PRINTING	HIGH RESOLUTION SLA PRINTING
PRESS FIT	0.1 – 0.15 mm	0.05 – 0.10 mm
SLIDING FIT	0.2 – 0.3 mm	0.15 – 0.2 mm
WALLS	0.8 mm thickness +	0.5 mm thickness +
HOLES + SMALL FEATURES	1 mm +	0.2 mm +
DIMENSIONAL TOLERANCE	XY +/- 0.12 mm	XYZ +/- 0.05 mm

Layer Height

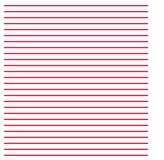
Layer height refers to the thickness of the layers of material that make up a 3D printed object, and is sometimes referred to as the resolution of the print. Typically, there are three to four resolutions to choose from when preparing a file.

FDM PRINTING



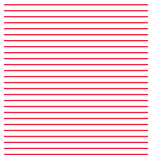
LOW RESOLUTION
0.3 mm layer height

MEDIUM RESOLUTION
0.2 mm layer height

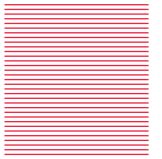


HIGH RESOLUTION
0.1 mm layer height

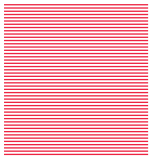
SLA PRINTING



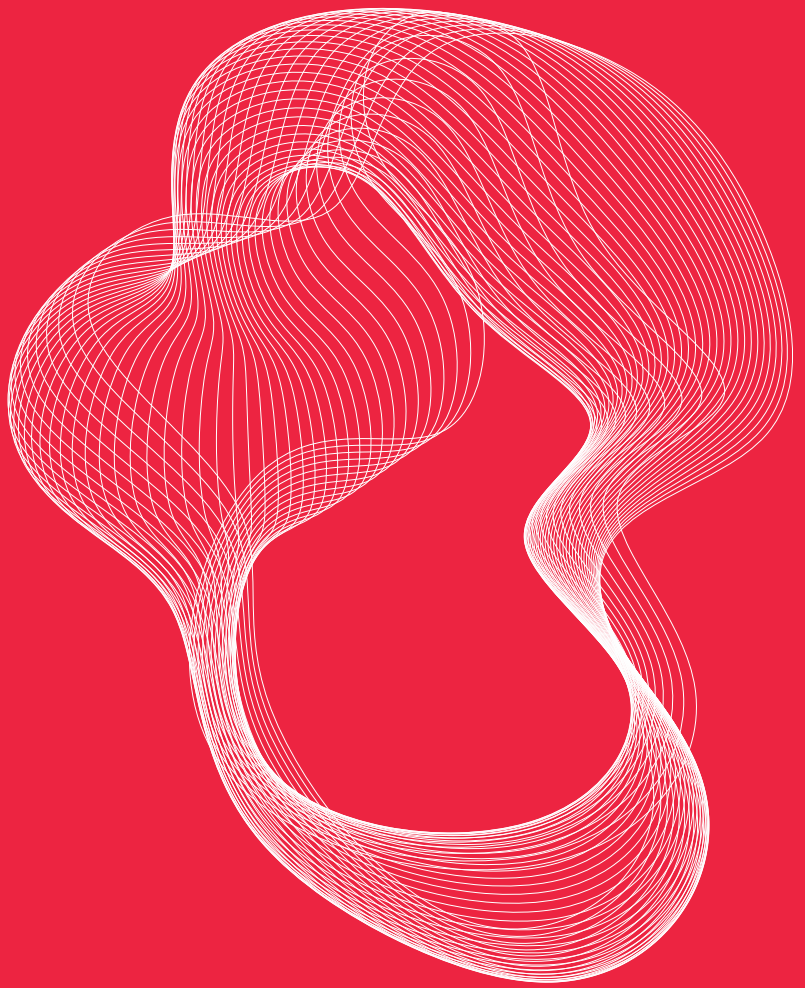
LOW RESOLUTION
0.1 mm layer height



MEDIUM RESOLUTION
0.05 mm layer height



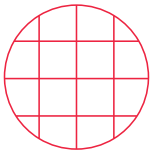
HIGH RESOLUTION
0.025 mm layer height



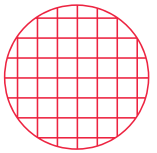
Infill Density

FDM PRINTING

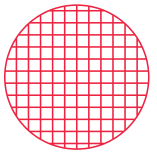
The interior of FDM printed objects is rarely solid material. Printing an infill pattern of plastic on the inside of the object reduces the amount of material used when printing while still creating a strong part. Infill density is specified by a percentage of the object's interior volume that is filled with plastic. A variety of 3D patterns can be used depending on the function of the printed part.



LOW 10%



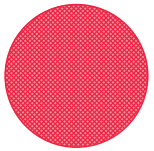
MEDIUM 20%



HIGH 35%+

SLA PRINTING

The interior of SLA printed objects always have an infill density of 100%.

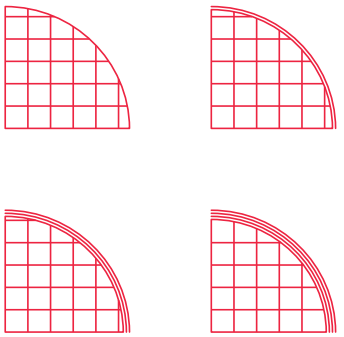


100% SOLID INFILL

Wall Thickness

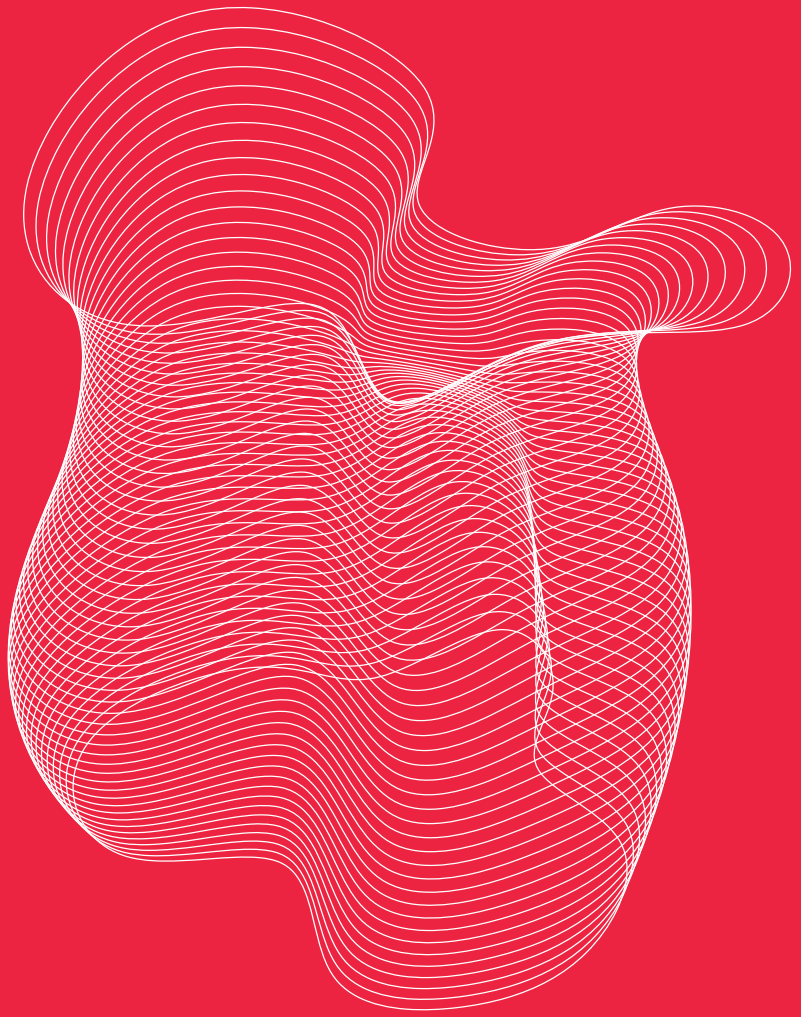
FDM PRINTING

The exterior surface of a 3D printed object is created with a wall or 'shell' of material. This wall can be made thick to strengthen a part, or thin to use less material and print more quickly. When more walls are added, they are created on the interior of the part, and do not change the external dimensions.



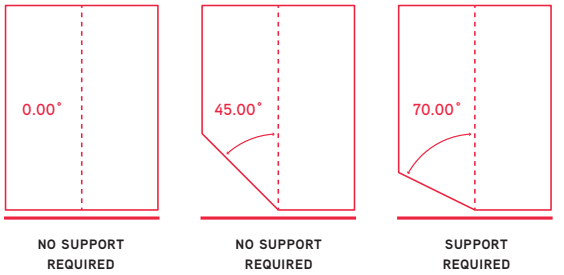
SLA PRINTING

SLA printing uses a standard wall thickness setting that does not need to be changed.



Draft Angles

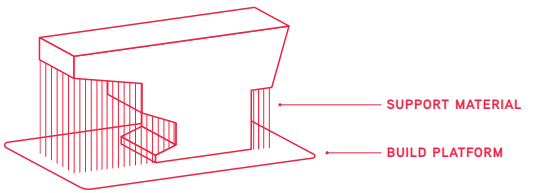
Draft angles in 3D printing refer to any features of a 3D model that do not print perpendicular to the printer's build surface. If areas of a model have severe draft angles, additional support material will need to be printed beneath. Generally, angles below 65° are able to be printed without support material.



Support Material

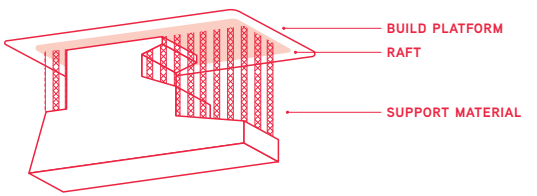
FDM PRINTING

Support material is printed beneath areas of an object that are beyond the allowable draft angle. It acts as a temporary support for layers of the print as they are built. If support material is not used where it is needed, the next layers of material may not have enough material beneath them to connect to. Support material does not completely bond to the body of the print, and can be broken or washed away depending on what type of printer is used.



SLA PRINTING

Support material is also generated in SLA printing, but in a different way. A raft is built on the build platform, with scaffolding extending to the printed object and attached at touchpoints.



File Preparation

exporting
slicing

STEPS BEFORE PRINTING

Exporting a 3D Model

Closed Volume

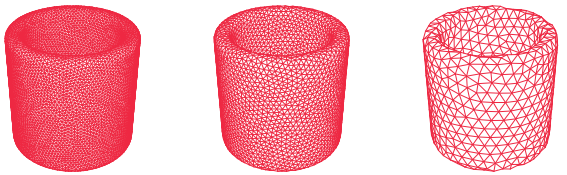
In order to convert a 3D model into a file that a printer can understand, it must be a solid, closed volume — this is also referred to as a model being 'watertight'. Unjoined surfaces, edges or unintentional gaps in a 3D model can create problems during export and while slicing.

.STL or .OBJ

3D models should be exported as an .STL or .OBJ file type. These file types represent the form of the 3D model as a triangulated mesh, which can be read by a slicer program that creates code for the 3D printer.

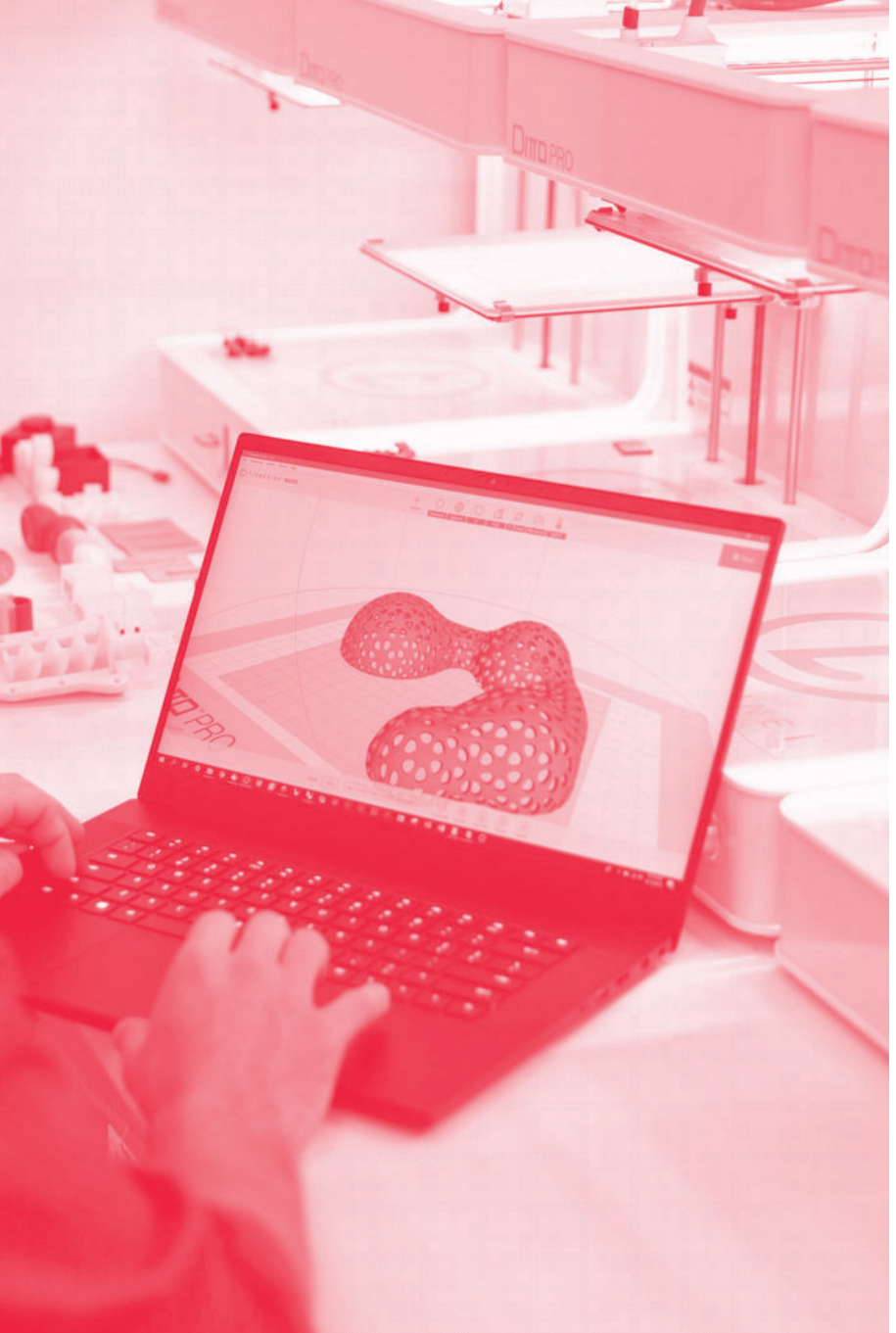
Resolution

It is important to check the resolution of these types of files when exporting. If the resolution is not high enough, it can affect the surface and accuracy of a 3D printed object.



HIGH RESOLUTION | MEDIUM RESOLUTION | LOW RESOLUTION

RECOMMENDED RESOLUTION .STL OR .OBJ FILES	0.01 mm / 0.004 in



Slicing

Once the 3D model is exported, it is time to open it in a slicer program. This is the last step before sending the file to the 3D printer. Importing the .STL or .OBJ will place it directly in the center of the printer's build area. Review the slicer settings and make any necessary changes.

After a 3D model is sliced it is exported as a .gcode file. G-code is a code format that is used to control many digital fabrication tools.

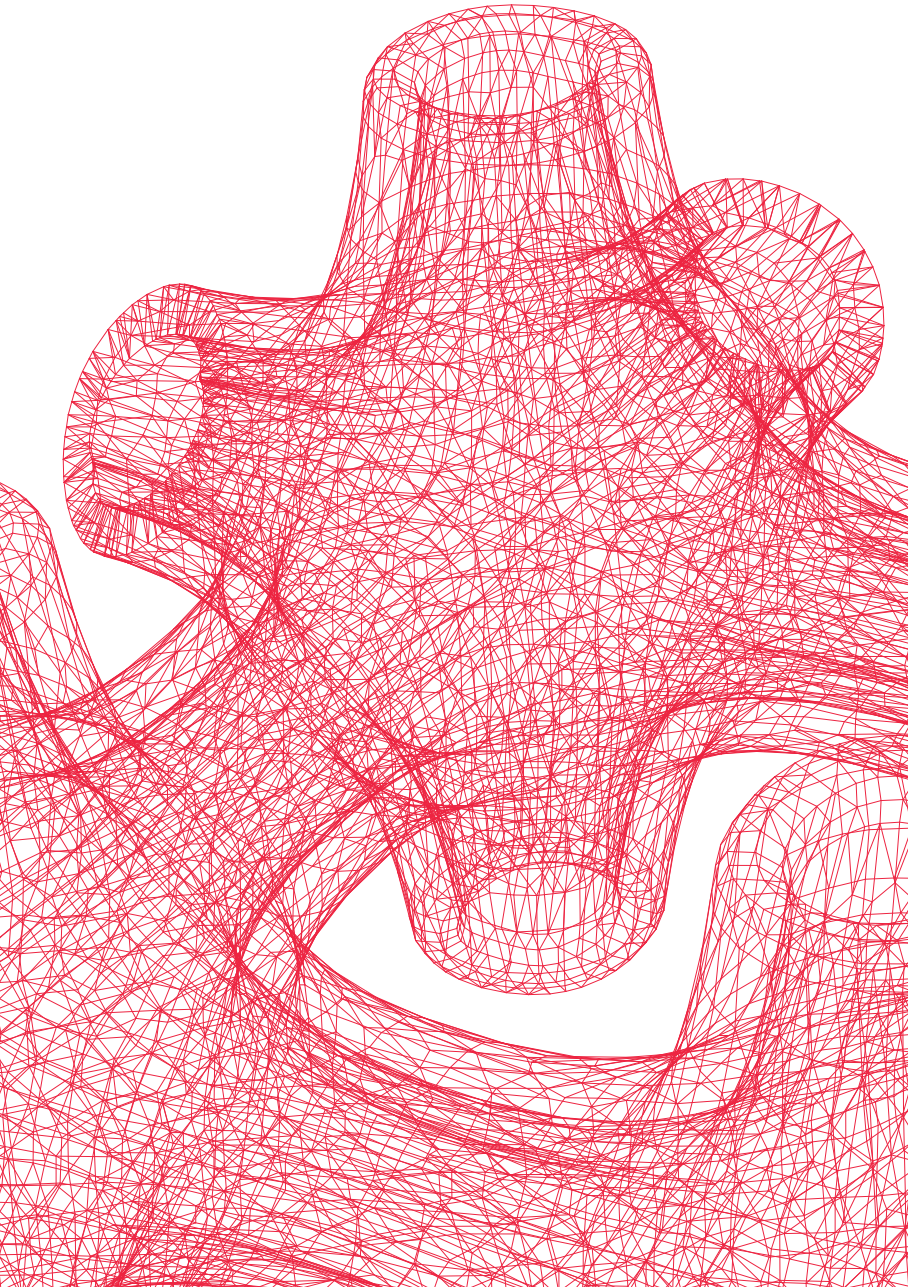
General slicer settings are subject to change with changes in 3D printer technology and material availability. For most up-to-date recommended settings, visit the Digital Fabrication Lab Moodle.

SCAN TO VISIT
DIGFAB MOODLE

What is .gcode?

This is a small portion of a .gcode file—a long list of X, Y and Z coordinates that define where the 3D printer will move in order to recreate the 3D geometry of a part. It is not necessary to edit these files after they are created.

G1 F1800 X62.04 Y81.641 E21.48884
G0 F3600 X61.761 Y81.907
G1 F1800 X61.496 Y81.641 E21.50047
G0 F3600 X61.217 Y81.907
G1 F1800 X60.951 Y81.641 E21.51213
G0 F3600 X60.672 Y81.907
G1 F1800 X60.407 Y81.641 E21.52377
G0 F3600 X60.128 Y81.907
G1 F1800 X59.862 Y81.641 E21.53542
G0 F3600 X59.583 Y81.907
G1 F1800 X59.318 Y81.641 E21.54706
G0 F3600 X59.039 Y81.907
G1 F1800 X58.774 Y81.641 E21.55869
G0 F3600 X58.494 Y81.907
G1 F1800 X58.229 Y81.641 E21.57033
G0 F3600 X57.95 Y81.907
G1 F1800 X57.685 Y81.641 E21.58196
G0 F3600 X57.405 Y81.907
G1 F1800 X57.14 Y81.641 E21.5936
G0 F3600 X56.861 Y81.907
G1 F1800 X56.596 Y81.641 E21.60524
G0 F3600 X56.316 Y81.907
G1 F1800 X56.051 Y81.641 E21.61687
G0 F3600 X55.772 Y81.907
G1 F1800 X55.507 Y81.641 E21.62851
G0 F3600 X55.227 Y81.907



EMILY CARR
SHOPS + STUDIOS