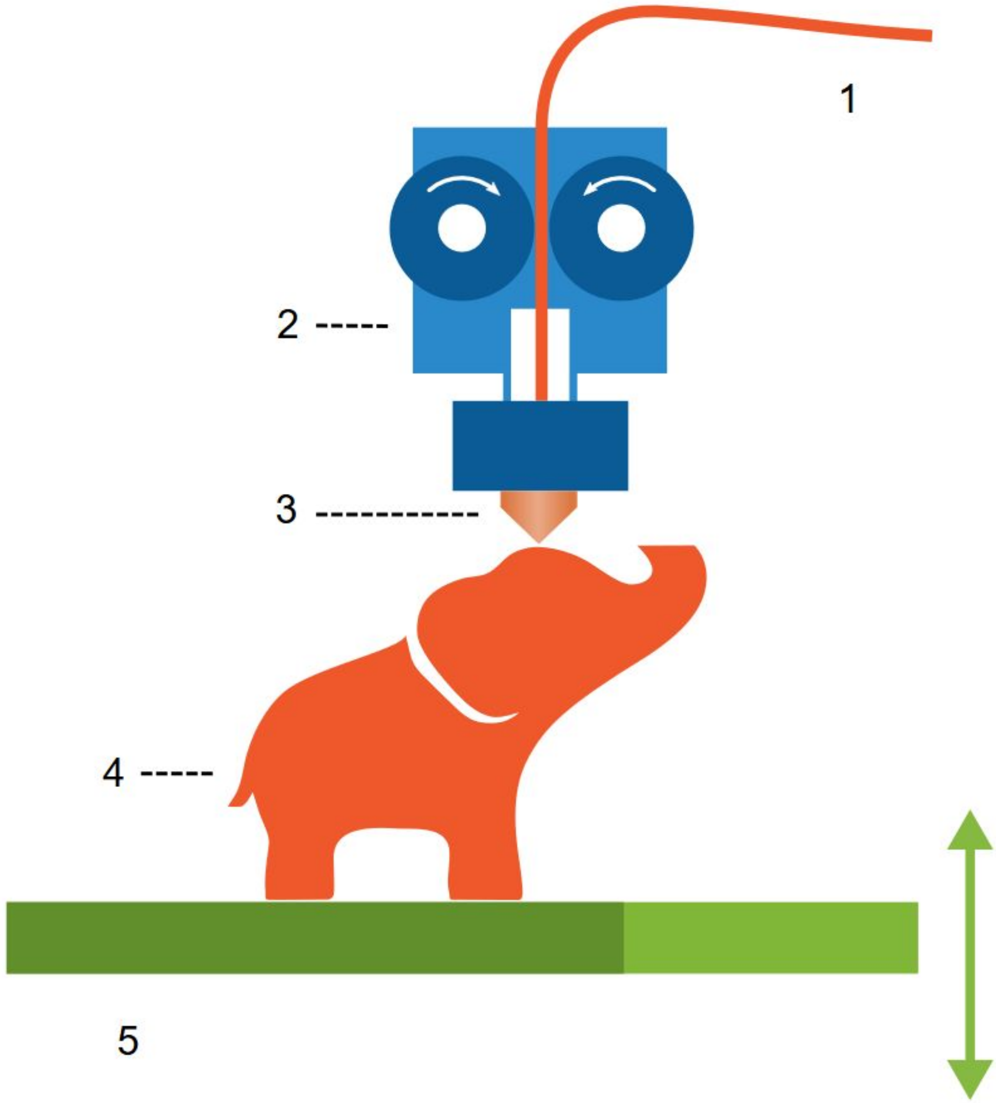
3D Printing and Modeling Orientation Packet



What is 3D Printing?
3D printing is an additive manufacturing technique, as opposed to traditional subtractive machining. Instead of starting with a block of material and cutting it down into a desired shape (as you would with a mill, lathe, or chisel), a 3D printer creates a solid object out of a fluid, plastic, or powder by building the desired geometry up layer by layer, solidifying as it goes.

Common 3D print types include:
- ***Stereolithography (SLA)*** - Actually the first type of 3D printing ever invented, SLA printers work by focusing a laser in a vat of liquid resin. Where the laser hits, the liquid resin hardens and solidifies. When finished, the completed model is removed from the vat and cleaned.
- ***Selective Laser Sintering (SLS)*** - SLS works similar to SLA in that a bed of loose powder is subjected to a laser, fusing the powder where it hits. After each run of the laser, a new layer of powder is applied and the process repeats. When complete, the solid 3D print is pulled out of the bed of loose powder and cleaned off.
- ***Fused Deposition Modeling (FDM)*** - Though slow, it is the most popular form of 3D printing among hobbyists and educators, because of its low cost and ease of use. FDM printers melt and extrude thermoplastics (meltable plastics), tracing out an object layer by layer with its nozzle (much like you would draw a picture with a pen, except with a third dimension), gradually increasing it's Z-height (vertical position) until each layer is complete. The final object includes no additional material (except for supports) and requires little cleaning.

(type some of these terms into youtube, to see them in action!)

https://penandplastic.com/3d-printer-types/ (Summary of 3D Print Types)

What Applications are there for 3D Printing?
- ***Prototyping*** - 3D printing enables you to rapidly create sample geometries, check measurements and fits before investing in the creation of a final object.
- ***Medical models*** - 3D printing is capable of creating incredibly accurate models of organs/etc. That are used for training and surgery practice.
- ***Custom components*** - A necessarily unique item like a prosthetic arm can be created to fit a user exactly.
- ***Small run production*** - Traditional manufacturing techniques like injection molding are very expensive and are generally not financially viable unless you are making 10,000+ items.
- ***Replacement parts*** - One-off prints
- ***Geometrically complex objects*** - 3D printing can create objects that traditional subtractive methods cannot (ex. Hollow objects, interlocking parts in a single piece, etc)

How are 3D Printable Objects Created?
3D Printable objects are called 3D models. They are composed of a collection of coordinate data that define the shape of the object, kind of like colored pixels define an image, but with an additional dimension. They can be derived from 3D photo or depth scans of a physical object or generated entirely in the digital world using a 3D modeling program. There are many kinds of 3D model formats, but a standard output format is either STL or OBJ.

Once an object exists digitally, it still needs to be converted into a format that a 3D printer can interpret. That's where a **Slicer** comes in. A **slicer** is a program that builds out a layer-by-layer **toolpath -** essentially a route for the printer **toolhead** to follow in order to lay down the print material in the correct location - kind of like a topographical map, but in more detail. The **toolpath** is specific to the 3D printer being used, while the 3d model is not. Most **toolpaths** are stored in a file format called **GCode,** which lists all the X,Y,Z coordinates the tool must go to. GCode is used by all sorts of machines, not just 3D Printers.

3D Printing Tools used in this Program:

1. 3D Modeling Software: **Fusion 360**
2. 3D Slicing Program: **Cura**
3. 3D Printers: **Creality FDM printers** (CR-10 & Ender 3)

Resources & Training Process:

1. **Fusion 360:** Cloud Based 3D Modeling Software.
    a. Create an Autodesk account associated with your pcc.edu email address. Write down your password in your design notebook. (Autodesk Student User Account)
    b. Log into Fusion 360 on your computer using your email address.
    c. Pick *one of the two* Tutorial Collection and start to work through the tutorial videos:
    d. Go to Autodesk Intro Course: https://academy.autodesk.com/software/fusion-360 **Watch the Following to get started:**
        i. **Beginner Videos 1,2,3,7,8,10,12,13,14,15**
        ii. **Intermediate Videos 1,2,3,4,5,6,8,9,10,11,12,13,14,15,50**
    e. Go to Lynda.com, Log in using your PCC login information. Navigate to “Fusion-360-Essential-Training” Course. **Watch the Following Videos to get started:**
        i. **1. Introduction *Videos 1,2,3***
        ii. **3. Understanding the Fusion 360 Environments *Videos 1, 2,3,4,5***
        iii. **4. Targeted Video Collection *Videos 1,3,4***
        iv. **10. Model: Create *Videos 1,2,3,4,6,7,8,9,10,11***

    f. Try and follow along with the videos, play around with creating various geometries, incorporating as many features as possible.
    g. Export your geometry as an STL *(See Autodesk Intermediate Video 50)*

2. **Cura:** Free 3D Model Slicing Program
    a. Follow Instructor tutorial to import and prepare you file for printing.
    b. The following terms are important **variables** for setting up a print. Ask your instructor for help as needed when setting up your file.
        i. *Filament Type:* Print Material, usually PLA.
        ii. *Filament Diameter:* Usually 1.75mm
        iii. *Print Quality* (Layer Height): Usually between 0.1 -0.3mm (lower value = greater print resolution, but also a slower print time.
        iv. *Wall thickness:* Thickness of model surface wall. Usually between 0.6-1.2mm. (Thicker = stronger but slower, more material).
        v. *Print temperature:* Must match print material, Usually around 210C for Nozzle, 70C for Bed when printing in PLA
        vi. *Infill:* Percent of model that is not-hollow. Usually ~20% for non-structural prints. (Lower value = less material, faster print time, but weaker)
        vii. *Supports:* Additional support structures are generated to hold up model **overhangs**. You can define a threshold angle, the smaller the angle, the more supports there will be, but they must be manually removed from print.
        viii. *Bed Adhesion:* For models with a small bed contact area, adding additional material around the model helps to keep print from shifting, disconnecting during print.

3. CR-10 / Ender 3 3D Printer Quick Reference Guide

Basic Maintenance

1. Keep the Print Bed Clean (Glass is a great build surface, but must be clean to work)
    a. Remove any built-up print material by hand or with scraper tool, taking care not to scratch the glass.
    b. Regularly remove Glass Plate from bed and clean with water or denatured alcohol. Dry thoroughly with microfiber cloth.
2. Check to make sure Nozzle is clean
    a. Clear away any excess material from nozzle tip (use tweezers if hot).
3. Make Sure Bed is Leveled
    a. 1 sheet of copier paper should just fit between nozzle and bed at all points, with some friction.
    b. Menu settings to move Nozzle along bed: *Prepare > Move Axis > 10mm > X/Y*
    c. Adjust height of bed by tightening/loosening the four thumb screws under bed.

Print Prep

1. Filament Preparation:
    a. Take note of the type (PLA/ABS), brand, and color of filament you will be printing with (different colors can have different physical properties) - make sure it matches the material profile selected in your slicing program. If no profile exists, select the most similar material or create a new material profile if ideal print settings are known.
    b. Confirm you have enough material to complete your print - the slicer should tell you the weight/length of filament required for the print (remember to add the length between the nozzle and the feeder to your total required filament length, ~20cm).
    c. If changing filament type, set the nozzle temp to 100C *(Control > Temp)* and wait for it to reach that temperature, then hold open the filament feeder and pull the old filament out. With the feeder still open, push the new filament through to the nozzle until it is met with resistance, release the feeder.
2. Pre-Heating:
    a. Automatic: If your machine is has pre-set warmup conditions, then simply select *(Prepare>PLA Print/ABS Print*) depending on material used.
    b. Manual: To warm up the printer manually, select *(Control > Temp > Bed > ##)* and *(Control > Temp > Nozzle > ###)*. Recommended warm-up conditions for PLA are Nozzle @ 185C and Bed @ 60C.
    c. The Main Screen will reflect the current and target temperatures of the Nozzle and Bed at all times.

Printing

1. Insert SD Card with gcode file (generated by slicer) into 3D printer.
2. Select *(Print from SD Card> "YourFile.gcode")*.

(The printer will reset target temperatures to those defined by material selected in slicer program. Your file will now begin to print once those conditions are reached.)

3. Confirm that target temperatures on main screen match material profile/desired conditions.
4. If the target temperatures are incorrect, they can be manually adjusted by selecting *(Tune > Temp. > Nozzle/Bed >###).*
5. Watch print for first few layers to ensure proper adhesion to the build plate. This is where most prints fail.

Warning: If you pause your print, the nozzle will remain heated. Never leave the nozzle heated while the printer is idle (temperatures below 100C are fine for between prints), nozzles left on can cause the filament inside to burn and harden, creating extrusion issues.

Last Edited 2019-04-07

3D Printing Rules & Safety

For questions and concerns, contact
cascadefablab@pcc.edu

Hazards:

- Extruder and motors are HOT during operation
- Extruder and motors may be HOT at any time
- PINCH POINTS while machine is moving
- Removal tools are SHARP

Dos:

- Do Clean Up
- Do inform a mentor of machine errors or damage
- Do check that print nozzle heater is off when printer is stopped
- Do inform a mentor of missing tools or supplies
- Do ask a mentor if you have any questions or concerns
- Do exercise caution when using cleaning tools

Don’ts:

- Don’t use the printers unless you have been trained
- Don’t attempt to modify or fix a printer without mentor approval
- Don’t let 3D printing become lazy engineering!

Last Edited 2019-04-07

Free 3D Model Repositories

3D Orchard
https://3dorchard.com/
Small collection of free models and design tutorials. Categories: Hobby, tutorials

African Fossils
http://africanfossils.org/
Collection of about 200 fossils including Hominids, animals, and tools. Categories: Science, history, education

Dremel 3D Idea Builder
https://3dprinter.dremel.com/3d-printing-model/anatomical-heart
Small set of models with lesson plans. Categories: Education, science

GRABCAD
https://grabcad.com/
Primarily models related to engineering. Categories: Science, engineering, art

Libre 3D
http://libre3d.com/index.php
Small set of free models. Primarily hobby. Categories: Hobby

MyMiniFactory
https://www.myminifactory.com/
Mostly free models, some paid. Large collection of tested, printable models. Extensive collection of scans of ancient art and famous landmarks. Categories: Accessibility, hobby, education, ancient art

NASA
https://nasa3d.arc.nasa.gov/
Models of asteroids, NASA space crafts, rovers, equipment, details of lunar surface Categories: Science, engineering, education

NIH 3D Print Exchange
https://3dprint.nih.gov/
High detail medical models, molecular models, and a variety of lab equipment. Some models will require editing and/or specialized 3D printing equipment to print properly. Categories: Science, engineering, education

Pinshape
https://pinshape.com/
Large collection of free and paid models.Some adult content. Categories: Hobby, education, art

Thingiverse
https://www.thingiverse.com/
The largest, free repository. Wide variety of materials. Categories: Education, hobby, art.

3D Print Training Worksheet:

Fill in this sheet and turn in when complete. This will fulfill your training.

1. **Are there any terms in the documentation that you are unfamiliar with? Please list at least 3 and ask the instructor to define them. Record their responses here.**

2. **Answer the following questions:**

- What is a Slicer? ______________________________

-What type of 3D print filament is most often used in our lab? ______________________________

-What is the recommended Nozzle/Bed Temperature for this material? ______________________________

-What are the max dimensions of the CR-10? The Ender 3? ______________________________

-What 3D model formats are most common? ______________________________

Complete these tasks:

- ❏ Build a printable 3D Model using Fusion 360
  (Required tools: Sketch, Extrude, Cut, Fillet, Mirror)
    - ❏ Export as 3D Model
- ❏ Level a Printer Bed
- ❏ Prepare and Slice a 3D Model using Cura
- ❏ Start a Print on a 3D Printer

Last Edited 2019-04-07