SLICING MODEL INTO G-CODE

Author: Archie Shou
Editor: Jim Chen

Date Created: 05/21/22
Date Edited: 05/27/23
Version 1.0

Abstract

In this SOP, the principle of slicing your CAD models for printing is documented.

Contents

List of Figures

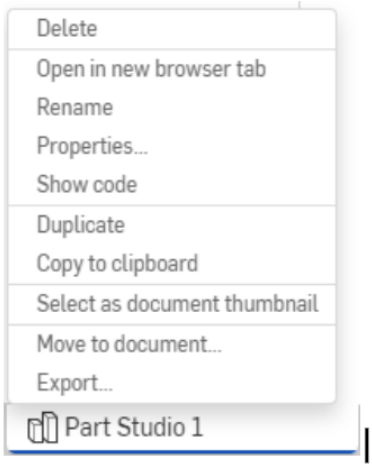
[1] *Footnote 1 etc.*

[2] *Footnote 2 etc.*

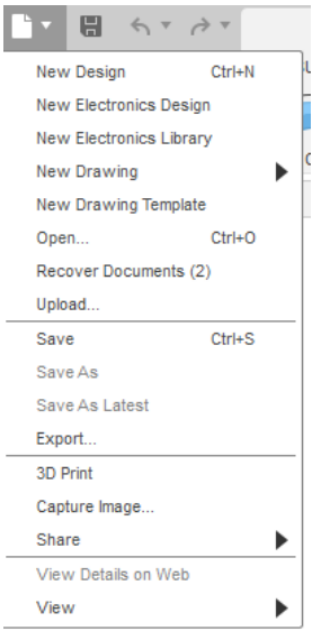
1 FDM Printing

FDM (fused deposition modelling) printers are standard 3D printers at the CAYPT lab. These printers are the most widely us ed and work by extruding plastic through a heated nozzle layer by layer until the part is complete. To print your parts, you need to first install a 3D design software whether that is ONSHAPE or FUSION360 and a slicing software such as CURA. The first step to printing is to convert your design into an STL file. This can be done

on ONSHAPE by right-clicking the bottom of your assembly and clicking export. Ensure that your export format is in STL and name it to something defining so you know what it is.

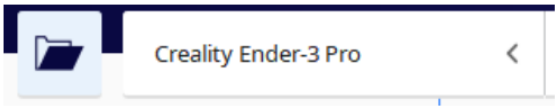


For FUSION 360, first, you need to save your file on the cloud and convert the fusion file into an STL through the export function. If you have CURA installed on the same laptop, you could click the 3D print function and it will ask you to select an app and will directly place your file into CURA.



2 CURA Software

Cura is a slicing software that converts your STL file into Gcode format that allows the 3D printer to follow. To get started with CURA, ensure that on the top left, you have the correct 3D printer selected. To add a printer, you can click the arrow which will prompt you to add a printer or manage your current one. This allows you to change the build size or settings if you upgraded your printer. To import your 3D file, you click the file icon on the top left and select the recently downloaded STL file.



You should see your 3D design pop up in the modelled build plate. By selecting the model, you are given a list of 6 options on the left. These include moving the build, resizing, rotating, flipping the design, or printing it as a support or overlaps. Most of the time, you will not be using the last 3 functions, however, the first 3 are quite useful when printing multiple parts. After you have completed these steps and are content with what you are planning to print, you can click the slice button in the bottom right which will prompt CURA to convert your file into Gcode and will even give you a preview of how the printer will move. You can save this file onto an SD card and begin your print! To access hidden settings and calibration, click the settings icon beside each major settings component.