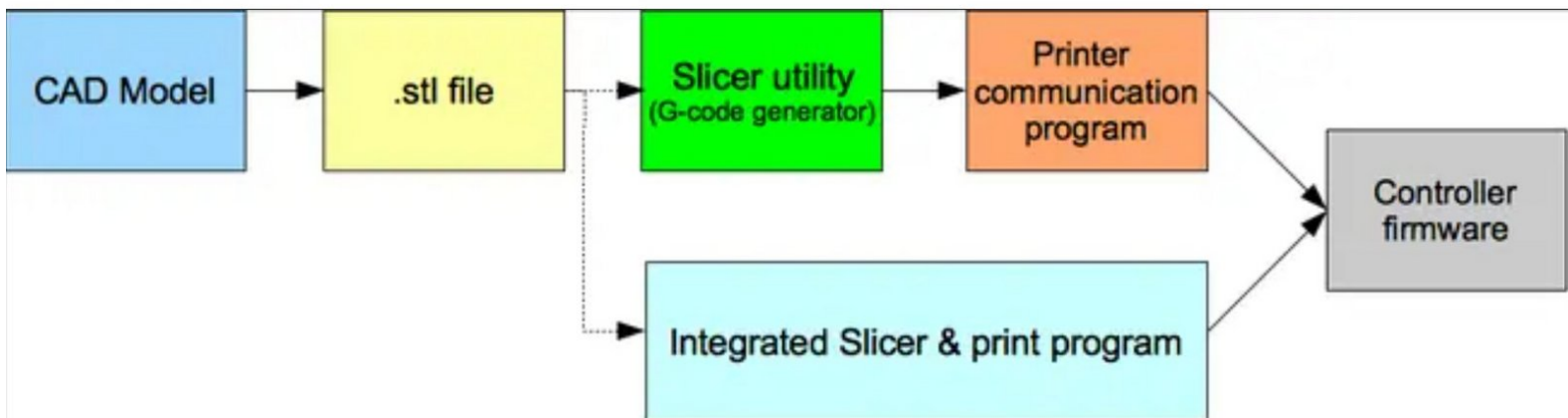# Introduction to 3D Printing

This is the first of a three-part sequence to using Kenny, the official 3D printer of R@M. This guide gives an overview of how to 3D print a model.

Written By: Dillon James Capalongo

This document was generated on 2022-11-02 08:47:25 PM (MST).

© 2022   dozuki.umd.edu/   Page 1 of 9
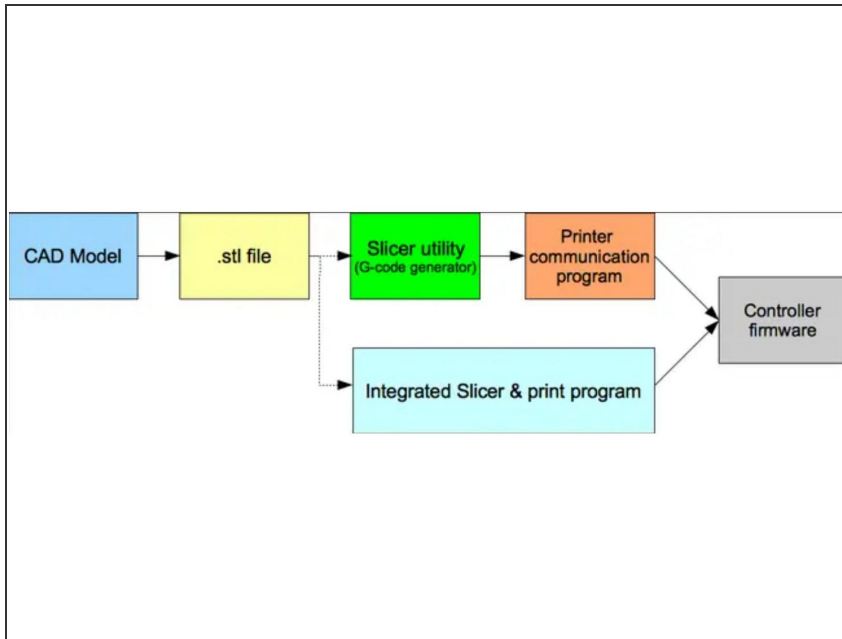
Introduction

This is the first of a three-part sequence to using Kenny, the official 3D printer of R@M. This guide gives an overview of how to 3D print a model.

🔧 TOOLS:

- Desktop / Laptop Computer (1)
- CAD Model (1)

This document was generated on 2022-11-02 08:47:25 PM (MST).

© 2022   dozuki.umd.edu/   Page 2 of 9
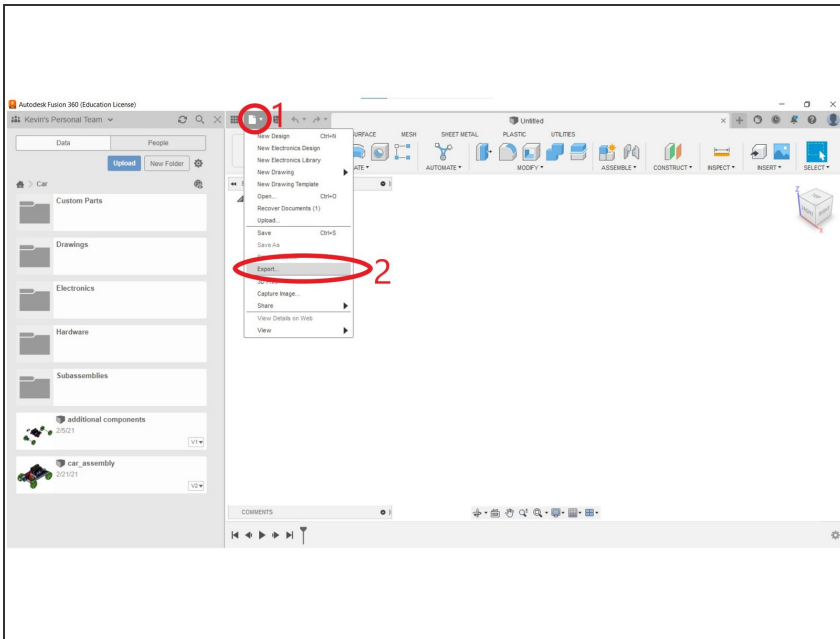
Step 1 — Overview

- 3D printing is a process that involves a CAD model, a slicing software, and a 3D printer.

Step 2 — Create a CAD Model

- The first step is to create a model of the object you would like to 3D print with a CAD software such as Autodesk Fusion 360, Autodesk Inventor, or SolidWorks.

- Refer to the Mini-Sub guide series for more information and practice with modeling.

This document was generated on 2022-11-02 08:47:25 PM (MST).

© 2022   dozuki.umd.edu/   Page 3 of 9
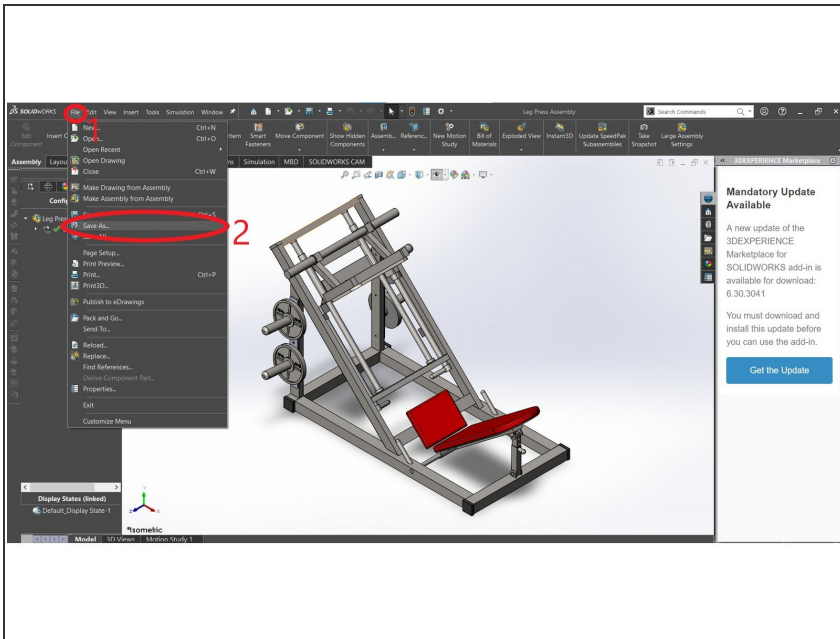
Step 3 — Create an STL File with Fusion 360

- After you create your model, you have to export it as a .stl file, which stands for stereolithography file.

- With your model open, locate and click on the "File "menu at the top of your screen.

- Click on "Export"

- A new window should appear. Give the STL file a name. Then, change the file type to .stl . Finally, give the STL file a location you want it to be exported to.

- Click on "Export"

This document was generated on 2022-11-02 08:47:25 PM (MST).

Step 4 — Create an STL File with Inventor
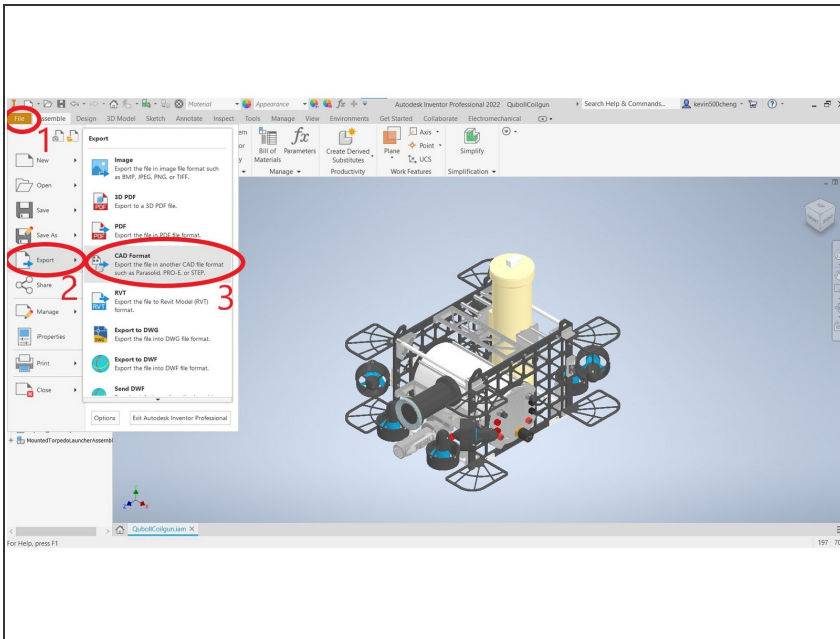
- After you create your model, you have to export it as a .stl file, which stands for stereolithography file.

- With your model open, locate and click on the orange "File "menu at the top of your screen.

- Click "Export"

- Click "CAD Format"

- Select the folder you would like the STL file to be saved in.

- Name the STL File.

- Change the file type to "STL Files (*.stl)"

- Click "Save"

Step 5 — Create an STL File with SolidWorks

- After you create your model, you have to export it as a .stl file, which stands for stereolithography file.

- Locate and click on the "File" button at the top of the screen.

- Click on "Save As"

- Select the folder you want the STL file to be saved in.

- Name the STL File

- Change the file type to "STL (*.stl)"

- Click "Save"

This document was generated on 2022-11-02 08:47:25 PM (MST).

Step 6 — Use Slicing Software

- After you have created an STL file from your model, open the slicer you plan to use for your 3D printer.

- Upload the STL file into the slicing software.

- After adjusting the settings, slice the file. The slicer should then create a GCODE (*.gcode) file.

- Please refer to the "Using PrusaSlicer" guide on more details for 3D printing with Kenny.

This document was generated on 2022-11-02 08:47:25 PM (MST).

Step 7 — Starting the 3D Print

- After getting the GCODE file to the 3D printer:
  - via USB Drive or SD Card:
    - Select the file you wish to print.
    - Start the print
  - via a printer communication software (Pronterface):
    - Connect your computer to the 3D printer
    - Select the file you wish to print and begin the print.
  - Please see the guide on "3D Printing with Kenny" for more information.

This document was generated on 2022-11-02 08:47:25 PM (MST).

Step 8 — Printing

- For the first few layers, watch the print as to make sure your print does not fail.

- After the print has finished, remove the print from the printer.

- Please see the guide on "3D Printing with Kenny" for more information.

This document was generated on 2022-11-02 08:47:25 PM (MST).