How to use "Solidoodle 3" 3D printer :

Installing the software (Repetier-Host Solidoodle)
- Follow this page : http://support.solidoodle.com/hc/en-us/articles/201546664-How-To-Install-Repetier-Host-For-Windows

Preparations
- Put the 3D printer and the roll of abs sludge plastic in the proper place.
- Connect the power cable.
- Give a thin layer of a mixed liquid (aseton + abs sludge plastic), so the plastic will stick on that layer. We can break it easily in the end.

Calibration :
- Generally, see this page : http://wiki.solidoodle.com/solidoodle-1
- Setting the Z offset : http://wiki.solidoodle.com/first-layers-setting-the-z-offset
  - This is important as we add a layer of glass on the original print-bed
  - To get the best result, the extruder must be "almost touch" the print-bed
- We find that the extruder and the print-bed did not go to home position. After some trials, we realise that we did not have the default G-code : http://wiki.solidoodle.com/setting-and-editing-your-gcode

How to print :
- This page gives you complete instruction : http://support.solidoodle.com/hc/en-us/articles/201486604-How-to-Print
- Open the software Repetier-Host Solidoodle
- Connect to the printer by clicking "Connect"
  - If it is not worked, check the COM number in the "Print Setting", you must change it to COM number of the printer (see in "Control Panel - Device Manager")
  - Sometimes we have to take out the USB cable, and plug it in again
- Go to tab "Manual Control"
  - Click the button "Heat Extruder" (temperature : 215^0 C) and "Heat Printbed" (temperature : 105^0 C). This must be done first, as it needs long time to heat the extruder and the printbed.
    - Be aware, when you "Kill the Job", or close the application, the heaters will be turned off automatically.
    - Although the button is on, sometime the heater is not turned on. It can be checked via watching their temperature on tab "Temperature Curve". Then you should turn the button off and on again.
  - Turn on the "Fan" as well.
- Go to tab "Object Placement"
  - Add object (*.stl file)
  - Arrange object position
- Go to tab "Slicer"
  - We can change the configuration of slicer on "Print Settings". For example, we use

customised configuration here ("3m .5f su 1").

- Faster do not mean better. Sometimes it may not give time to the plastic to stick enough. It is the trade off.
  - Click "Slice with Slicer". Then you will get the G-code.
- Click the "Run Job" button to start printing
  - You may use "Kill Job" to finish it completely, or "Pause Job" to pause it momentarily.

Created by : Handy Wicaksono, Rudino Salleh (CSE - UNSW)