Rhino & Grasshopper slicing tutorial

In search of a better way to slice CAD to RAPID and GCODE Vertico organized a Hackathon. This document and the associated content is the result of this hackathon and made together to be shared together. It is free to use but not meant for commercial purposes.

Getting started

Downloading the necessary software

1. Download Rhino 6 link: https://www.rhino3d.com/download
2. Download the slicer file 'Vertico Hackathon Slicer 1.0' – link: https://wiki.bouwkoppel.nl/doku.php?id=software:rhino_grasshopper
3. Open Rhino
4. Go to: File > Open
5. Select your CAD design (make sure it is a .STL file!)
6. STL import options appear (click 'OK')

Setting up the Slicer

Import your file

7. Type in "Grasshopper" in the command line (this opens Grasshopper)
8. Go to: File > Open Document
9. Select Slicer file 'Vertico Hackathon Slicer 1.0' you downloaded in step 2
10. Go to '1.1 Input the base shape here'
11. Right click on 'Mesh' and click 'Set one mesh'
12. Select the CAD design (click on the model you have imported)

Let's Slice!

Generating the G-code

13. Go to '1.2 Global Variables'
14. Select the slicing settings you want
    a. ParallelSlice vs. SpiralSlice: either slice in layers or one continuous layer
    b. Layer Height: slicing height of each layer
    c. Point Count: number of coordinates generated per layer
15. Let the magic happen (wait to load, slicing is happening)

Exporting Coordinates

Using the coordinates that you have generated

16. Go to '10. G-code' – scroll right
17. Right click on the text panel and click 'copy data only'
18. Open your text editor (eg. Notepad++)
19. Paste the text and save the file however you want

Congratulations on making your first slice!