

# 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](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, press enter (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!