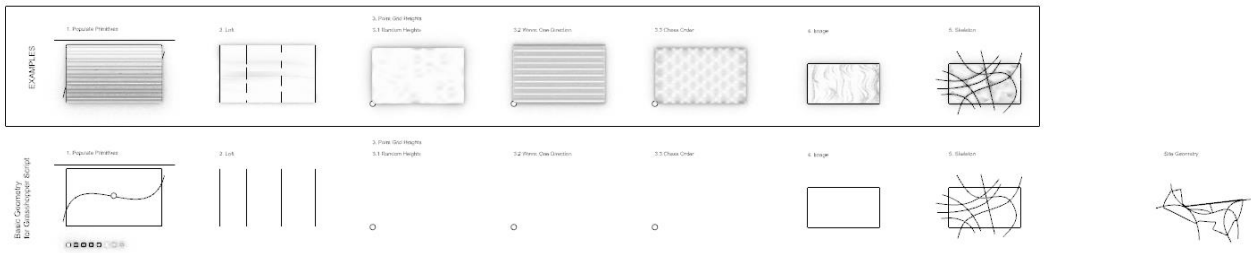


File Structure:

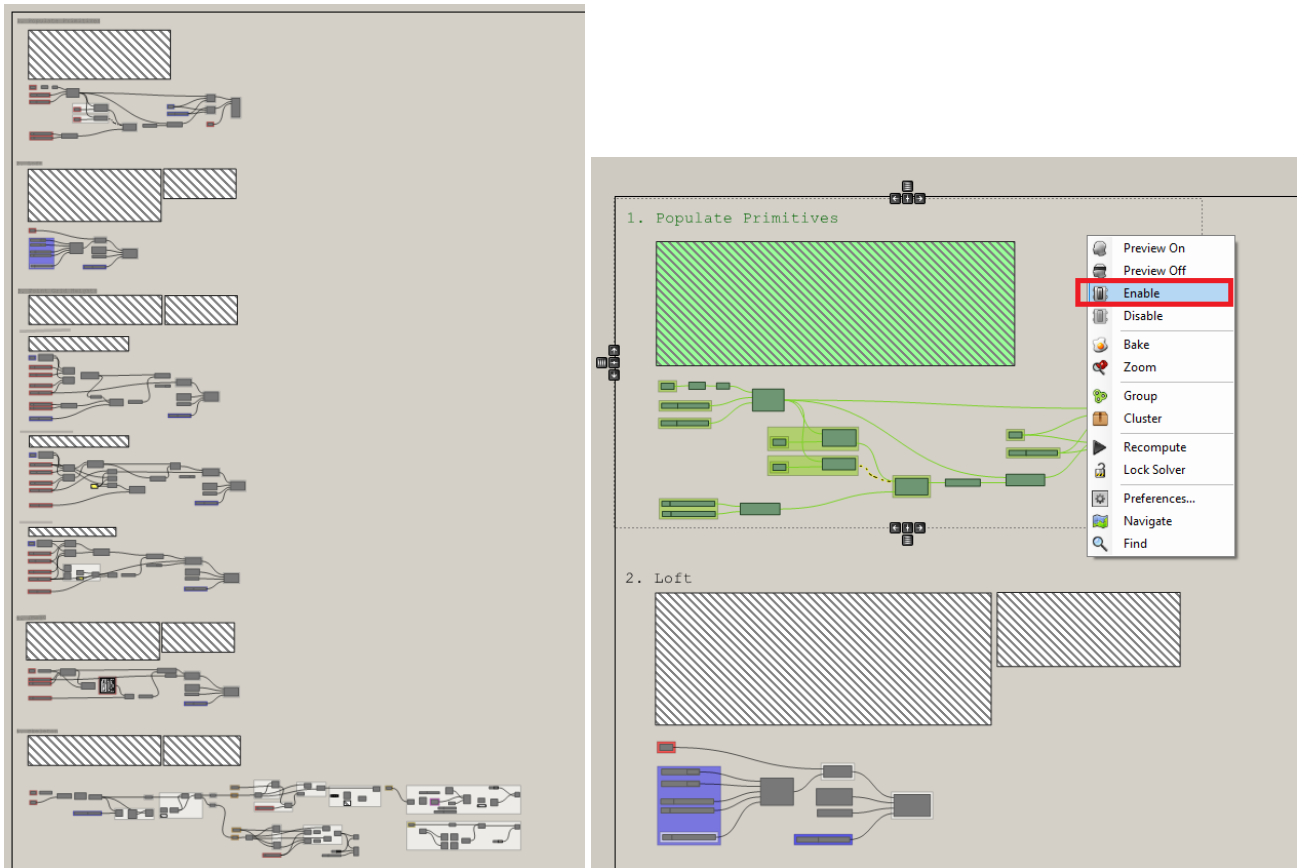
In Rhino File:



In upper row examples of the created geometry are located. In the lower row basic geometry needed to create a surface in grasshopper are presented. All the elements are connected with Grasshopper scripts. Fell free to change this geometry or assign a new one.

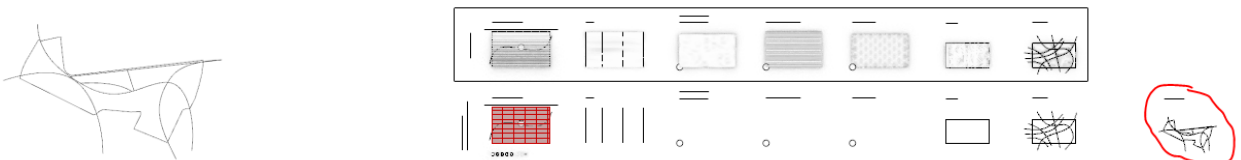
For each example there is an according Grasshopper script (look at the name).

Initially all the elements in Grasshopper file are disabled. To work with the specific method select all the nodes of it and with RMC (on any free space) "Enable" it. Then you will see created objects in Rhino.



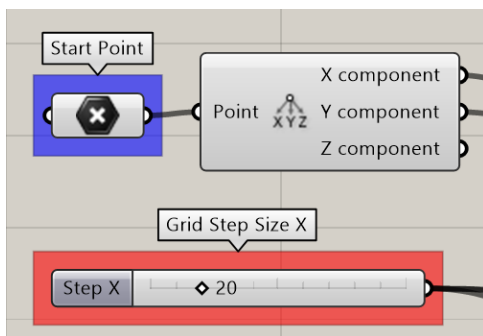
Go through all the methods and experiment with them. If you are advanced in Grasshopper feel free to add something new. You can also try to apply any of the methods to the geometry of our site (on the right side in the Rhino file)

Site Geometry

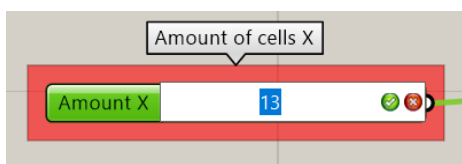


Some Basics:

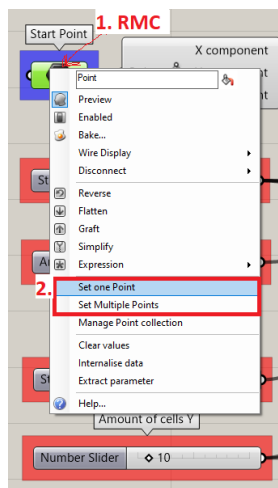
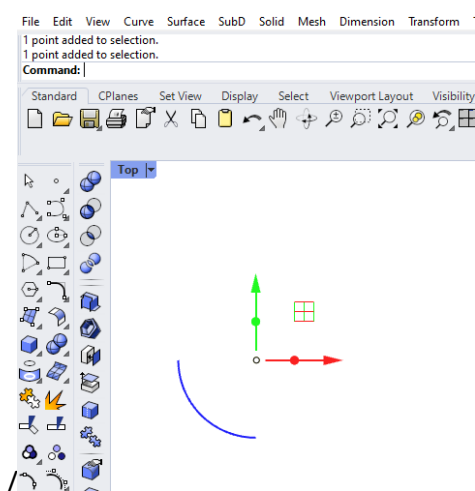
Red and Blue elements are input data or parameters that could be changed to create different results.



Working with the sliders better double click on it and write desired number to avoid unnecessary recalculation of the script, that are sometimes very time-consuming.



To assign another point(s) or curve(s) select it in Rhino and with right mouse click (RMC) on the element select “Set one Point”/ “Set Multiple Points”



To bake created geometry select the node that contains it. With the RMC select “Bake”, then chose the layer in Rhino, where you want to save it.

