FreeCAD instructions to create Mesh

1. PartDesign Workbench
2. Create Body, Create Scetch
3. Chose XY Plane
4. Create rectangle (2 dots symbol)
5. Coincidence Constraint s.th. bottom left corner at origin
6. Freely modify ground shape
7. Fix vertical and horizontal distances. Build up your 2D mesh form by using appropriate constraints.

8. Finish sketch by closing sector
9. Pad selected sketch (symbol right)
10. Chose length

11. Go to Model section
12. Go to Mesh Design Workbench
13. Select created Pad
14. Go to Meshes > create mesh from shape
15. Chose standard (and other parameters)
16. Export the meshed pad as .stl