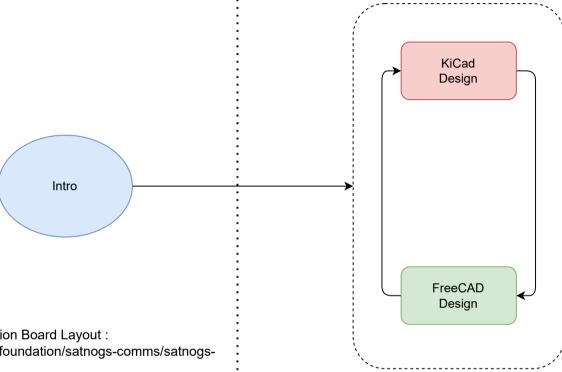# EMI/RFI shield in FreeCAD and KiCad

Notes:

Links

[1] system-design p. 73 section Board Layout :
https://gitlab.com/librespacefoundation/satnogs-comms/satnogs-comms-design-doc/

[2] Section 6 Shielding, Electromagnetic Compatibility Engineering Henry W. Ott

[3] EMI Materials:
https://gr.mouser.com/datasheet/2/987/EMI_CAT_ECE_080615-1902899.pdf

KiCad Design

FreeCAD Design

Intro

Tools:

[1] kicadStepUpMod: https://github.com/easyw/kicadStepUpMod

[2] KiCad Forum : https://forum.kicad.info/t/kicad-stepup-the-sketcher-for-footprint-generation/9582/181

Steps:
1. Set grid origin
2. Set Drill/Place File origin
Tip: Select the middle of lines
3. Add your drawings in User.Drawings layer
4. Export User.Drawings layer in to DXF
5. Import the DXF in to Freecad
6. Convert the drawing in to sketch
7. Merge sketches in to one skecth
8. Save as FCstd file
9. Open kicad_pcb with stepup plugin
10. Save as FCstd file
11. Merge 2 projects
12. Create new body and add the sketch that produced from merge in to this body
13. Rename the body as shield
14. For the exported footprint, get in mind to sketch on the top
15. Create a group with name footprint
16. Use RF PolyLined Sketch to create sketches for footprint
Tip: The sketches must be like open source logo, source Tools [2]
17. Use a random small circle to assign Pad 1 in footprint.
18. Convert to the correct rotation (Mirror) to export 3dshapes