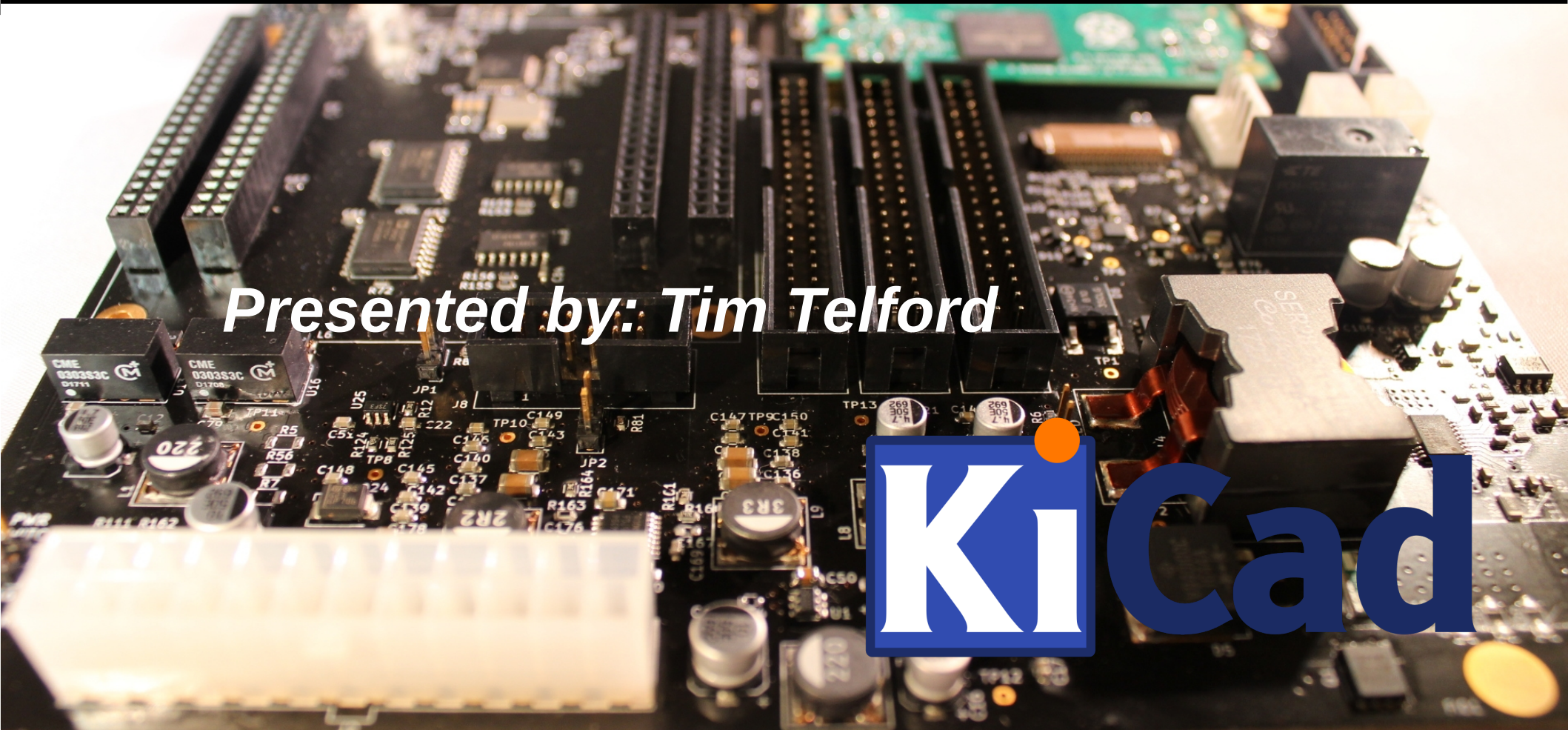
KiCAD EDA Training – Part 2

DEVTANK



Day 2 Agenda

- Differential track and routing (PCBNew)
- Impedance matching
- Centralised company libraries
- Using Git (github) for library management and backups
- BOM generation tools
- FreeCad Integration and 3D model-footprint alignment

Timings (Day 2)

- 8.00 – 10.00: Advanced Schematic tool features
- 10.00 – 10.15: Break
- 10.15 –12.00: Avanced PCB tool features
- 12.00 – 13.00: Lunch
- 13.00 – 14.30: Advanced library management tips and tricks using Git
- 14.45 – 15.45: FreeCAD Integration and 3D model footprint libraries.
- 15.45 – 16.00: Washup/Lessons learned

Recap from Day 1

- Any specific questions/topics to cover?
- Real life problems/issues?

BOM Generation

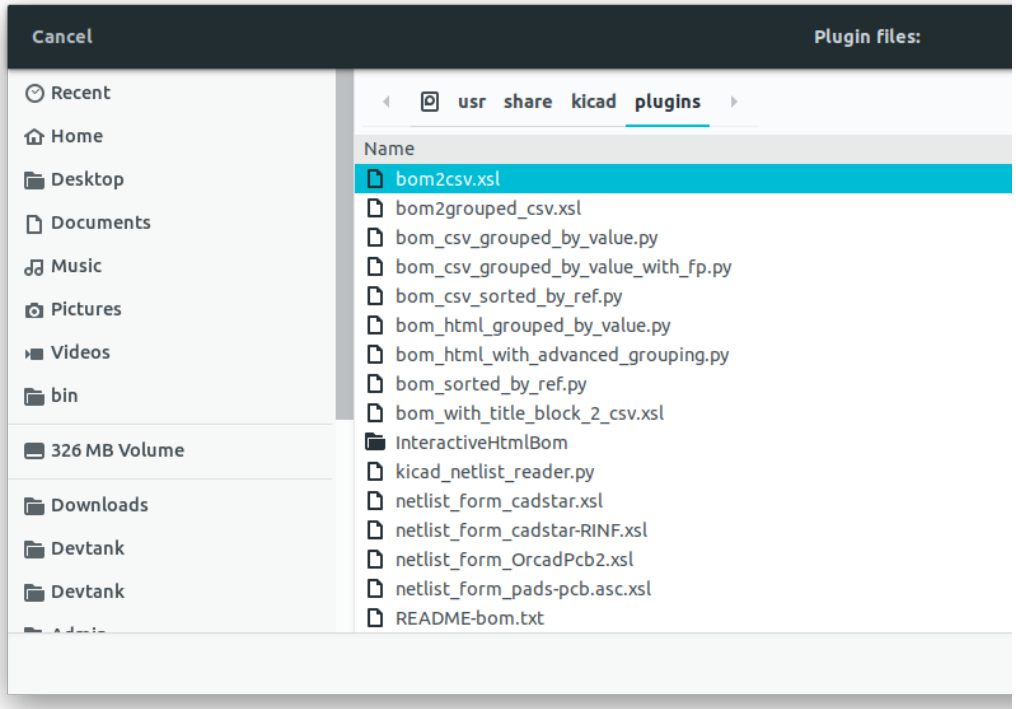
- BOM export feature is fully customisable by the user.
- KiCAD has a plugin install feature for BOM tools
- At Devtank we use KiCad_BOM_Wizard developed by HashDefineElectronics
  - https://github.com/HashDefineElectronics/KiCad_BOM_Wizard
  -

Adding BOM Export tools

- Add your choice of BOM plugin script
- Linux: usr/share/kicad/plugins

  Windows:

  c:\Program Files\KiCad\bin\Scripting\plugins\
- **Example (1)**



Importing pictures into your design

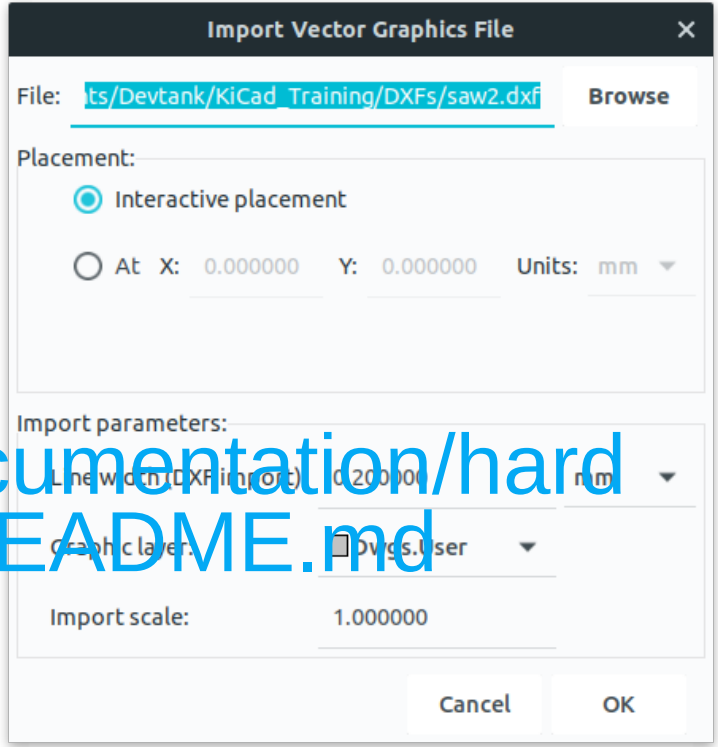
- Bitmap to Component Convertor
- Example (2)

Drawing Templates

- Page Settings (this tool can be a little quirky!)
- Using the drawing template editor
- Help: https://docs.kicad.org/5.1/en/pl_editor/pl_editor.pdf
- Example (3)

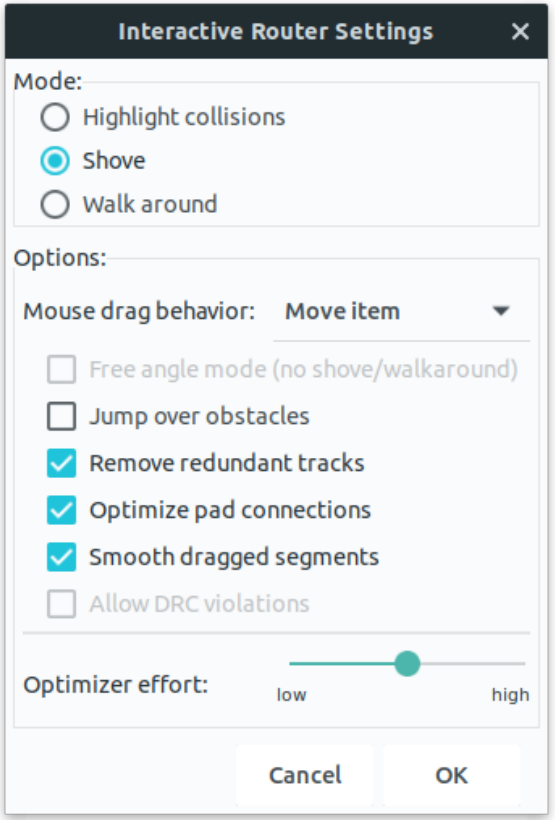
Importing Dxf files

- Using mechanical drawings
- Layer management
- Dimensioning
- https://www.raspberrypi.org/documentation/hardware/raspberrypi/mechanical/README.md
- Example (4)



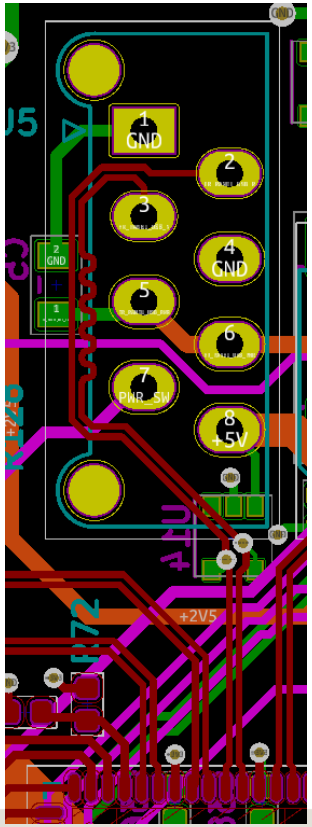
Intelligent Router Optimisation

- Router options:
  - Push & Shove,
  - Highlight Collisions
  - Walk Around
- Example (5)



Advanced PCB Features

- Differential tracking and routing - worked example:
- Please clone the following repository:
  - https://github.com/devtank-ltd/kicad_hiltop_motherboard
- Key features:
  - Impedance length matching
  - Tune skew/phase
- Example (6)

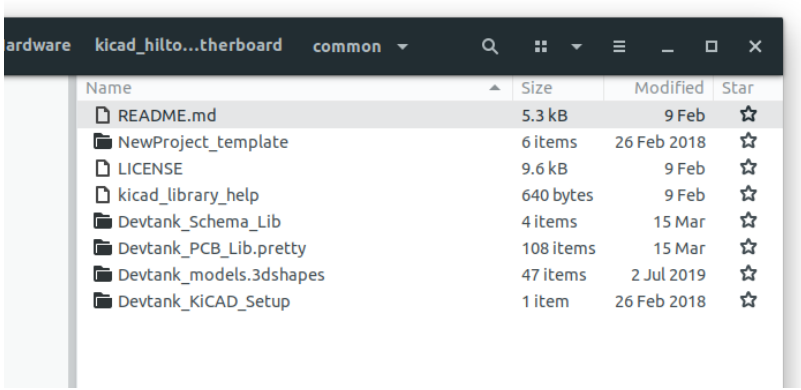


Project Management & Collaboration

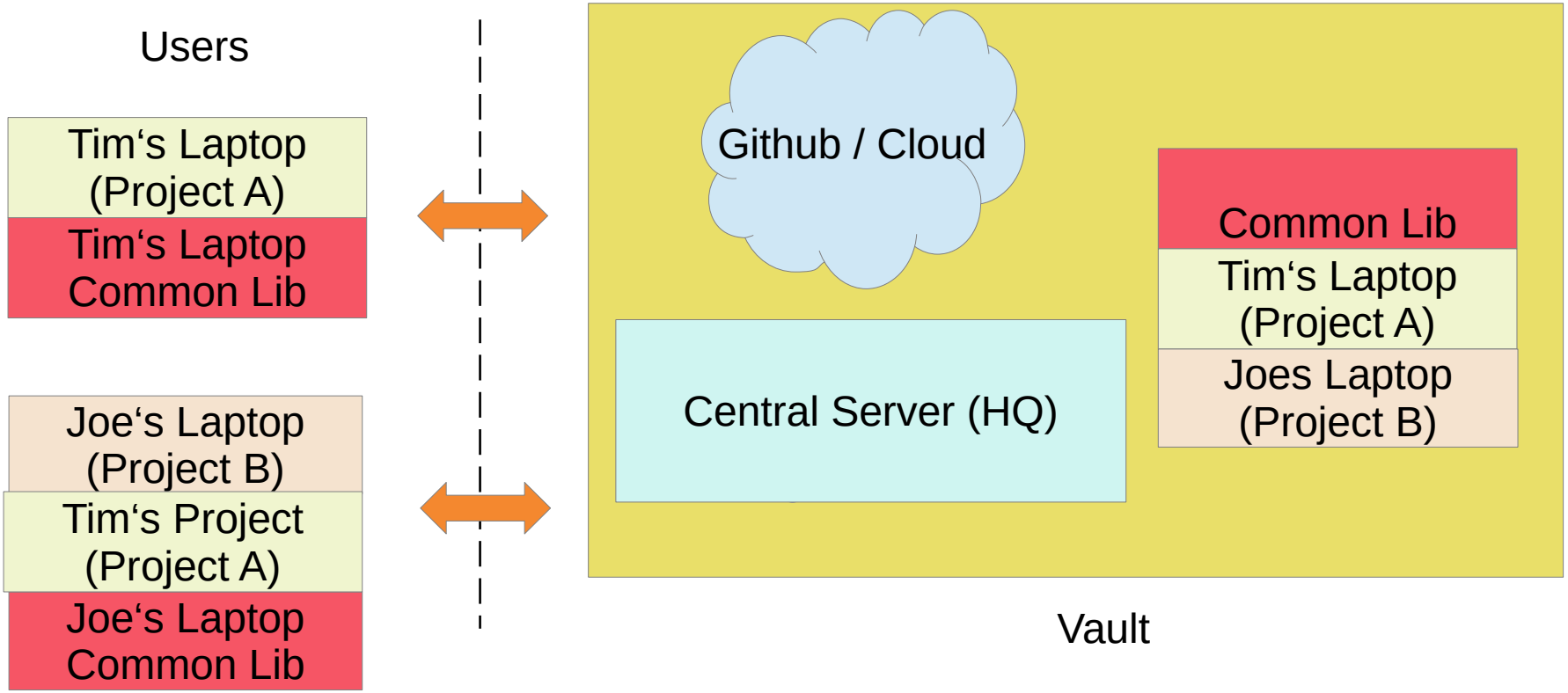
- Knowledge base:
- Peer Design Reviews
- Release processes / Change Control
- Central server

Common templates

- Establishing a project baseline
- Default design rule settings
- Library management – keeping up to date with changes



Git Work Flow



Git Tools

- Git is cross platform (Windows, Linux, MacOS)
- Most popular version control tool software in the world
- Windows options:
- Git for Windows -https://git-scm.com/download/win

  Tortoise Git - https://tortoisegit.org/download/
- Linux (bash)

  Sudo apt-get install git
- MacOS

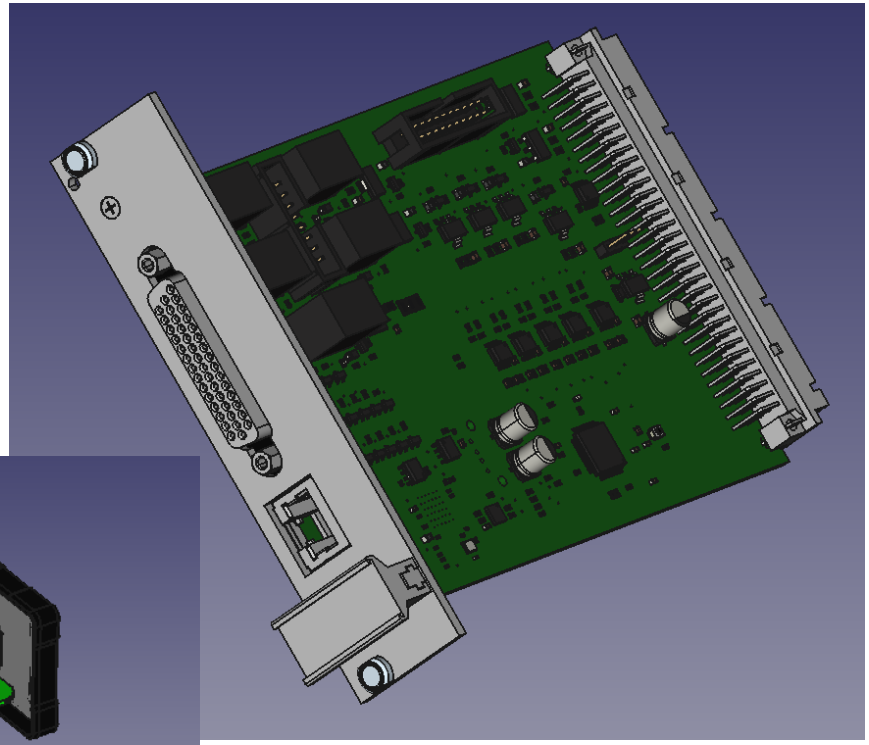
  https://git-scm.com/download/mac

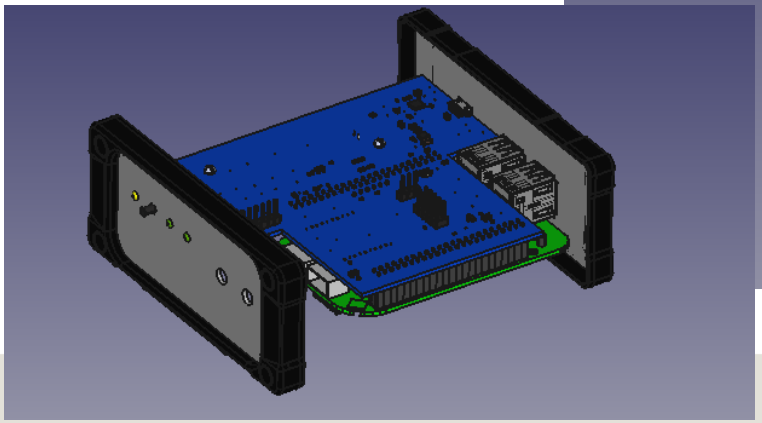
Advanced Library Management (Github)

- Managing repositories and best practice
- Using kicad with git: Recommended video. https://www.youtube.com/watch?v=oXzJFrLo77Y
- Cloning public libraries:
  https://github.com/devtank-ltd/kicad_devtank_common
  https://github.com/Digi-Key/digikey-kicad-library
- Useful git commands:
  - git clone --recursive <<insert URL>>
  - git commit -a -m “relevant log comment“
  - git log
  - git status

FreeCAD Integration

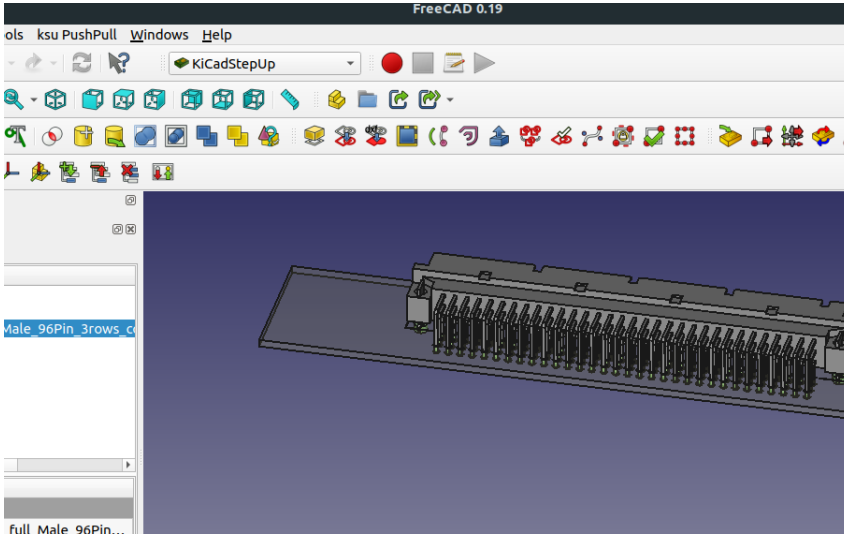
- Adding additional 3D Models to your footprint designs.
- Check PCB alignment with mechanical models
- Using the KiCAD Stepup Tool Plugin





Stepup Tool for FreeCAD

- Precisely align footprints with models
- Add material information – vrml and step model generation
- Design new footprints
- https://wiki.freecadweb.org/KicadStepUp_Workbench



Questions