A crash Course in KiCad





Introduction



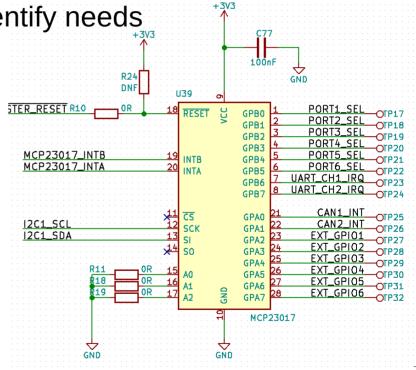
- Congratulations on picking this fantastic tool!
- House-keeping
- Useful References:
 - http://kicad-pcb.org/
 - http://kicad-pcb.org/help/file-formats/
 - https://github.com/devtank-ltd
- Please ask lots of questions and try the worked examples as we go through the day.

Agenda (1st half)



Introductions to the tool suite and identify needs

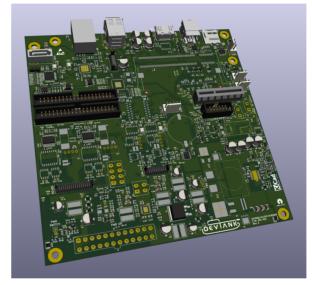
- Schematic Capture (Eeschema)
 - Initial setup (getting started)
 - Basic tool features
 - Worked examples
 - Hints and Tips
 - Library Management
 - New part creation
 - BOM Generation



Agenda (2nd half)



- PCB Layout (PcbNew)
 - Initial setup
 - Basic tool features
 - Netlist import
 - Part placement
 - Tracking and routing
 - Footprint creation
 - 3D Viewer
 - Manufacturing data pack
- Wash up review of learning outcomes.

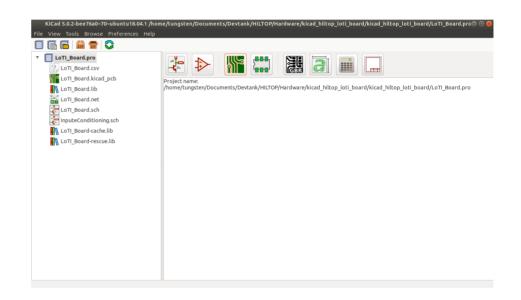




KiCad – Tool Overview



- Schematic Tool
- Symbol Library Editor
- PCB Tool
- Footprint Library Editor
- Gerber Viewer
- Bitmap Editor
- Calculator
- Template Editor



Example Projects

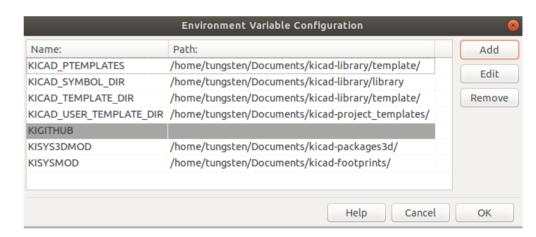


- Please go to Github and clone the following projects:
 - https://github.com/devtank-ltd/kicad_devtank_common
 - https://github.com/devtank-ltd/kicad_load_test_b oard
 - https://github.com/devtank-ltd/kicad_hiltop_loti_b oard

Eeschema Setup



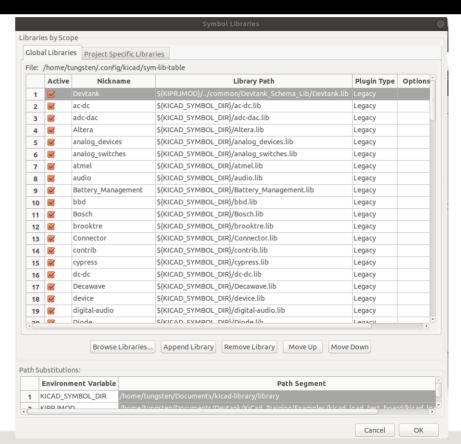
- Configure paths
- Local and remote paths
- Hotkeys
- Page Settings and Drawing templates



Eeschema Setup (2)



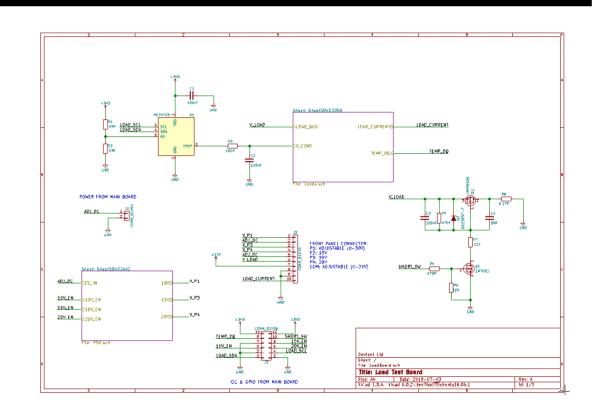
 Manage Symbol Libraries



Eeshema Tool Features

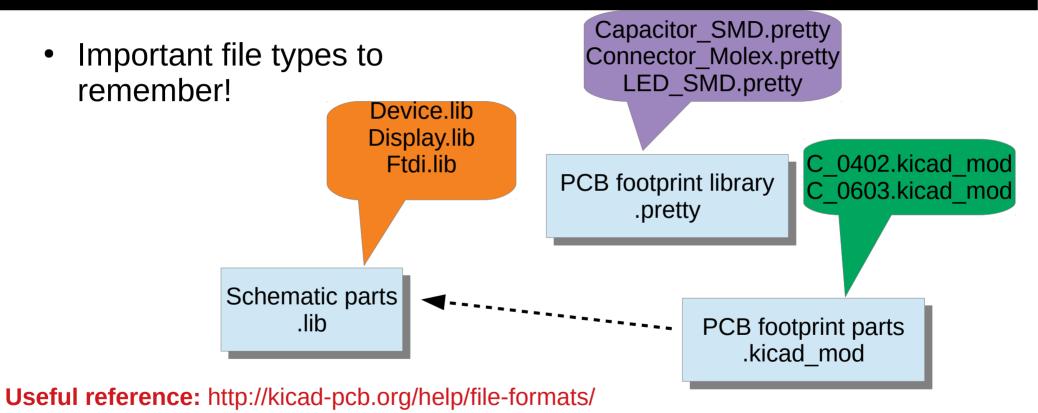


- Drawing Templates
- Place components
- Place power nets
- Wire up circuits
- Annotations
- Heiarchical sheets and labels
- Highlighting nets, hot keys
- Part copy, grid management
- Worked Example (1)



Library Management





New Part Creation



Symbol Library Editor Tool

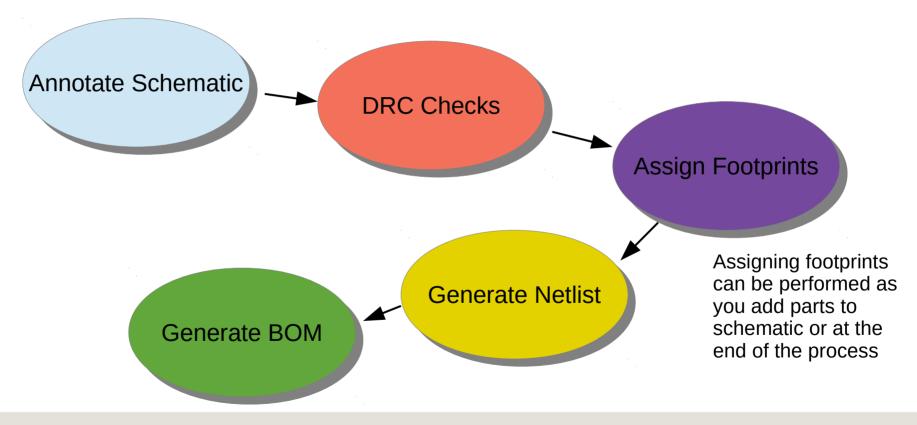


- Create the shape and connections/pins
- Add properties and datasheet references
- Aliases/Footprint filters
- Duplication of parts from other libraries don't reinvent the wheel!
- External Tool Wizards

http://kicad.rohrbacher.net/quicklib.php (recommended tool)

• Worked Example (2)





PcbNew Setup

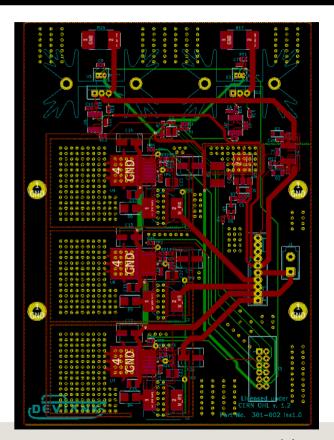


- Ensure footprint libraries are setup
 - >Manage Footprint Libraries option
- Choose your board layer stackup (note this can be changed later)
- Define design rule settings
 - Net classes (Clearance, track width, via dia etc...
 - Custom via and track widths
- Using a common dummy project will expedite this process!

PCBNew Tool Features



- Import Netlist
- Defining your board outline and understanding the layers (toggle visibility) – dimensioning and measuring tools
- Place footprints (positioning, locks, adding parts directly)
- Working on Grids (and avoiding alignment issues)
- Cross selection/highlighting Eeschema to PCBNew
- Track and Routing (Router options)
- Hotkeys
- Copper Pour and Keepout Zones
- Worked Example (3)



New Footprint Creation



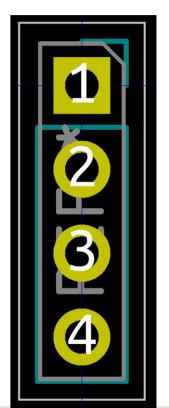
Footprint Editor Tool



- Create the outline shape and connections/pads
 - Normal layers defined are: F.CrtYd, F.SilkS & F.Fab*
- Add properties and datasheet references



- Duplication of parts from other libraries
- 3D Part references (more on this later)
- Worked Example (4)



Finishing touches



- DRC Checks
 - Unconnected nets
 - Clearance rule checks
 - Parts overlap etc.
- Manufacturability checks
- 3D Viewer
- Generate Manufacturing data
 - Gerbers
 - Pick and Place files
 - Step, svg model files etc.

Washup – Review Day 1



- Confirm the days objectives have been met.
- Are there any subjects we would like to re-visit in more detail.
- Q&A

Advanced Training Topics

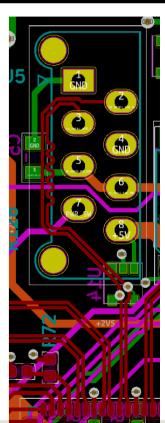


- Differential track and routing (PCBNew)
- Impedance matching
- Centralised company libraries
- Using Git (github) for library management and backups
- BOM generation tools
- FreeCad Integration and 3D model-footprint alignment

Advanced Tool Features



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

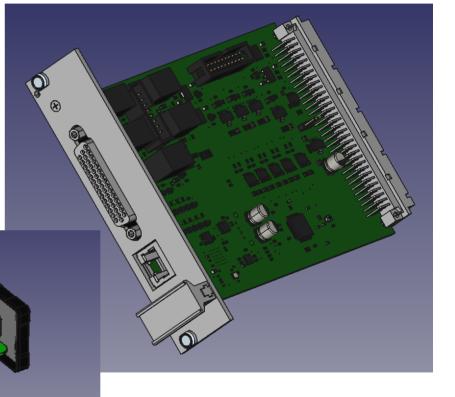




FreeCAD Integration



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



Advanced Library Management (Github)



- Managing repositories and best practice
- Useful git commands:
 - git clone --recursive <<insert URL>>
 - git commit -a -m "relevant log comment"
 - git log
 - git status

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_BO M Wizard

Questions



