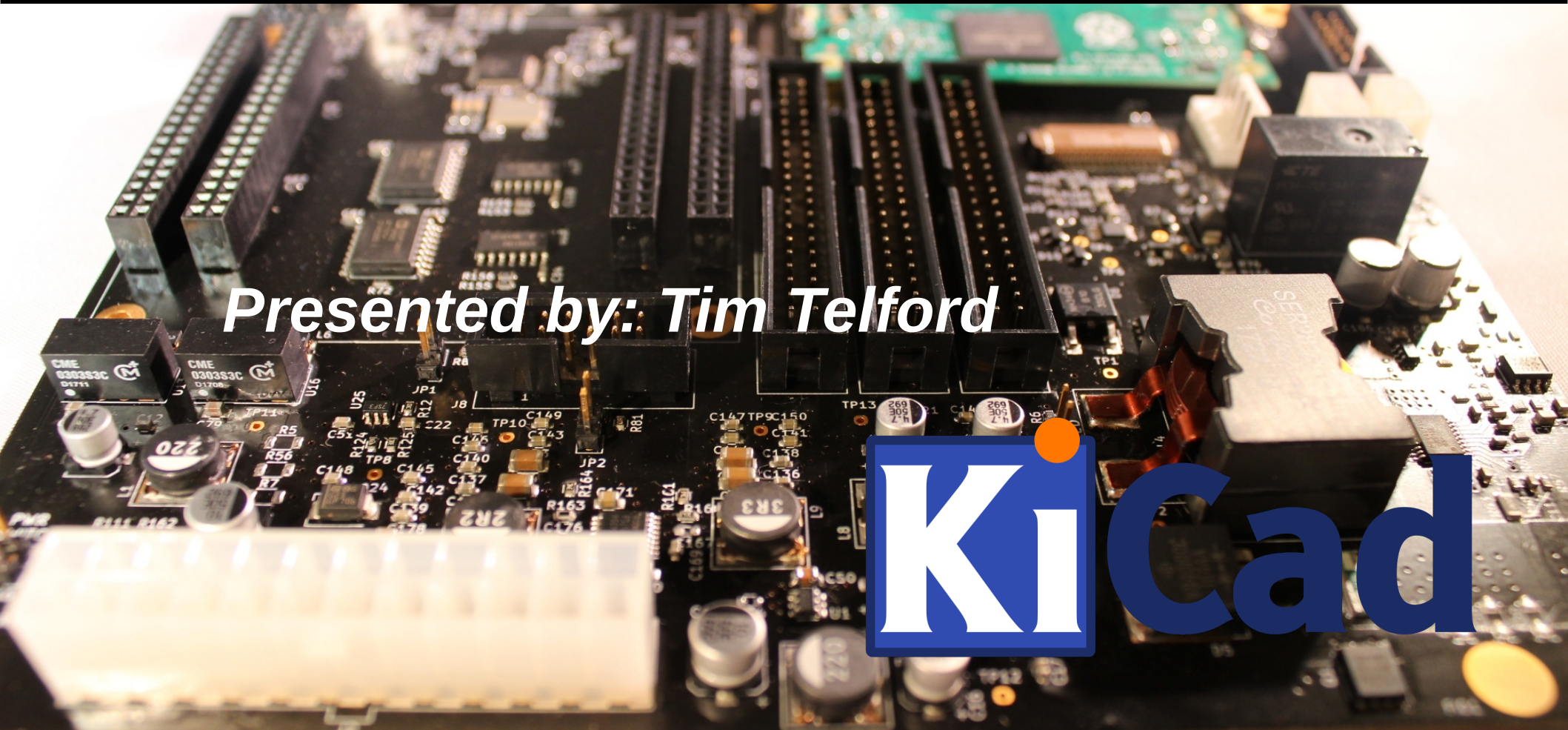
KiCAD EDA Training – Part 1

DEVTANK



Introduction

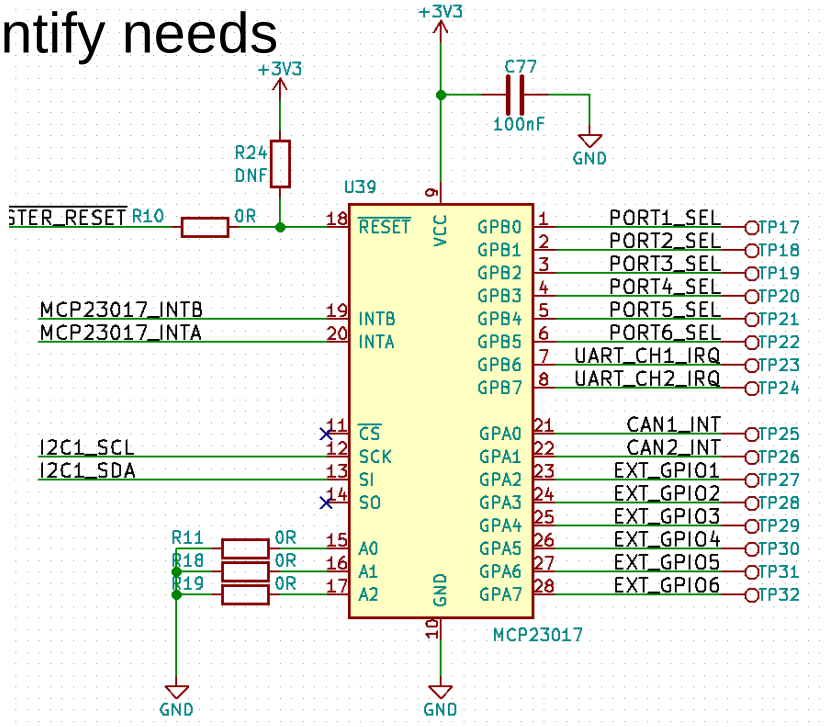
- Congratulations on picking this fantastic tool!
- House-keeping
- Useful References:
  - http://kicad-pcb.org/ (KiCad home page)
  - https://en.wikibooks.org/wiki/Kicad/file_formats (useful reference for file formats)
  - https://docs.kicad.org/ (online manuals/how to guides)
  - https://github.com/devtank-ltd (Devtank training material/examples)
- Please ask lots of questions and try the worked examples as we go through the day.

Timings (Day 1)

- 8.00 Introductions
- 8.30 - 10.00: Schematic capture & worked examples (Eeschema)
- 10.15 – 10.30: Break
- 10.30 – 12.00: Schematic capture & worked examples
- 12.00 – 13.00: Lunch
- 13.00 – 14.30: PCB Layout
- 14.30 – 14.45: Break
- 14.45 – 15.45: PCB Layout
- 15.45 – 16.00: Washup/Lessons learned

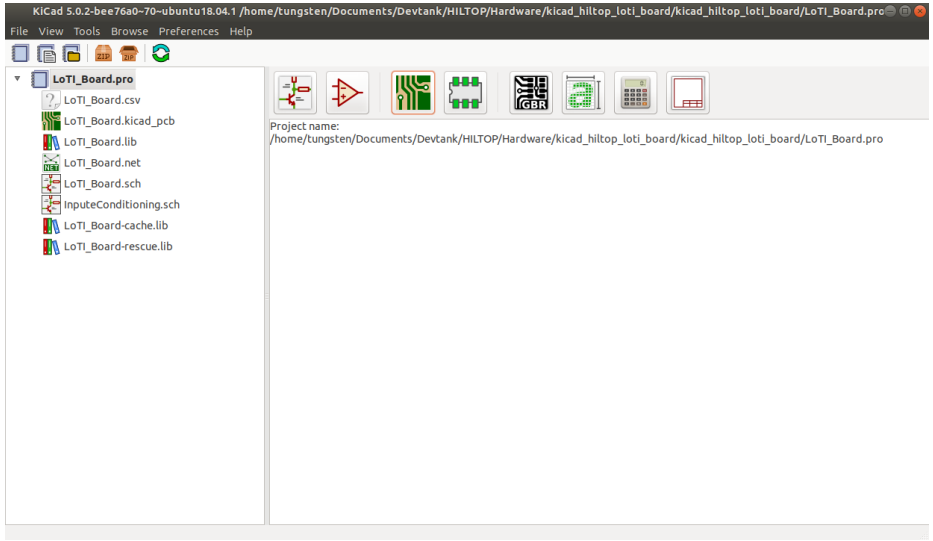
Day 1 Agenda (1st half)

- 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



KiCad – Tool Overview

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



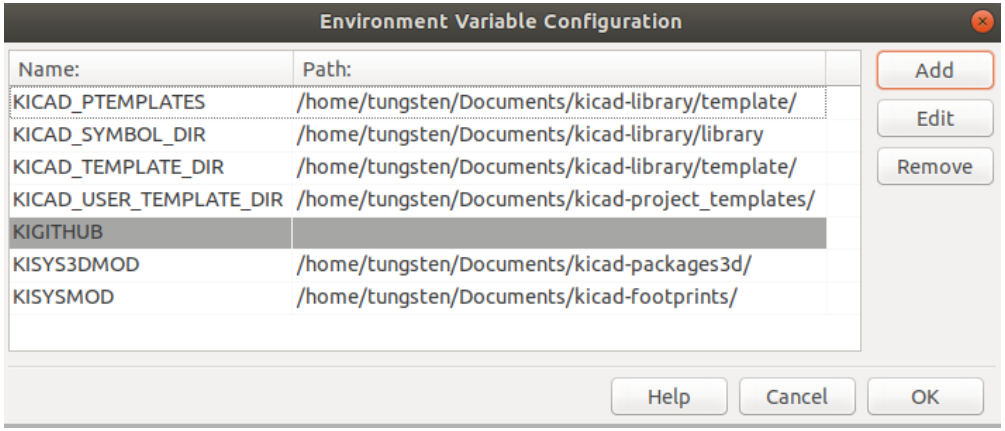
Example Projects

- 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_board
  - https://github.com/devtank-ltd/kicad_hiltop_loti_board

    https://github.com/devtank-ltd/kicad_training

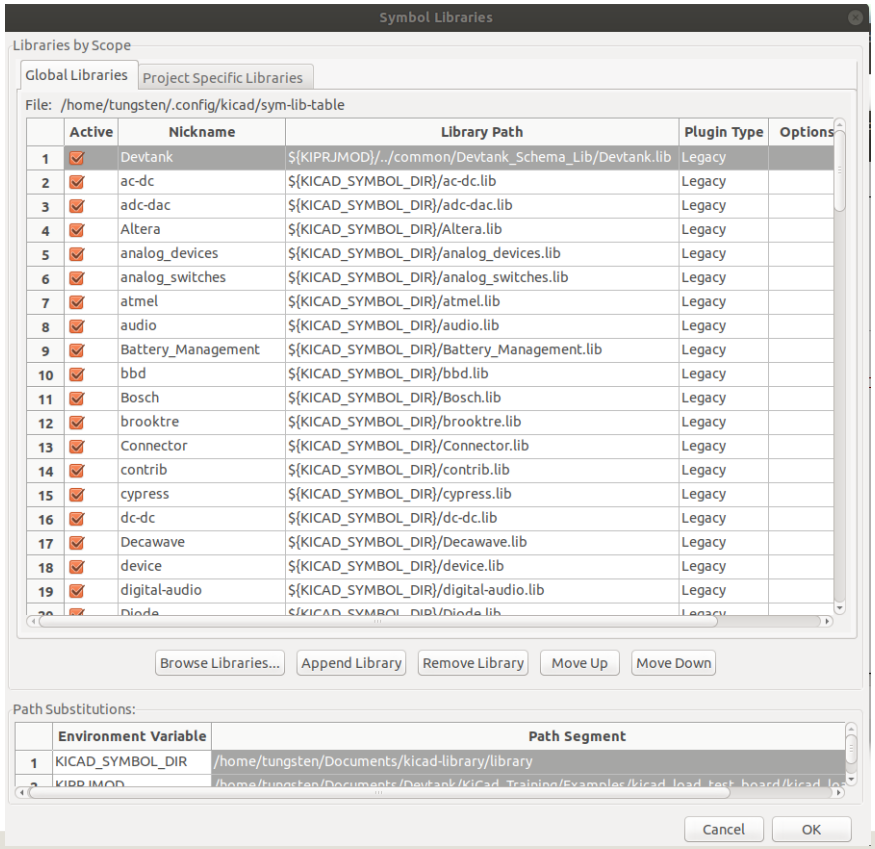
Eeschema Setup

- Configure paths
- Local and remote paths
- Hotkeys: Help > List hotkeys (or Ctrl+F1)
- Page Settings and Drawing templates



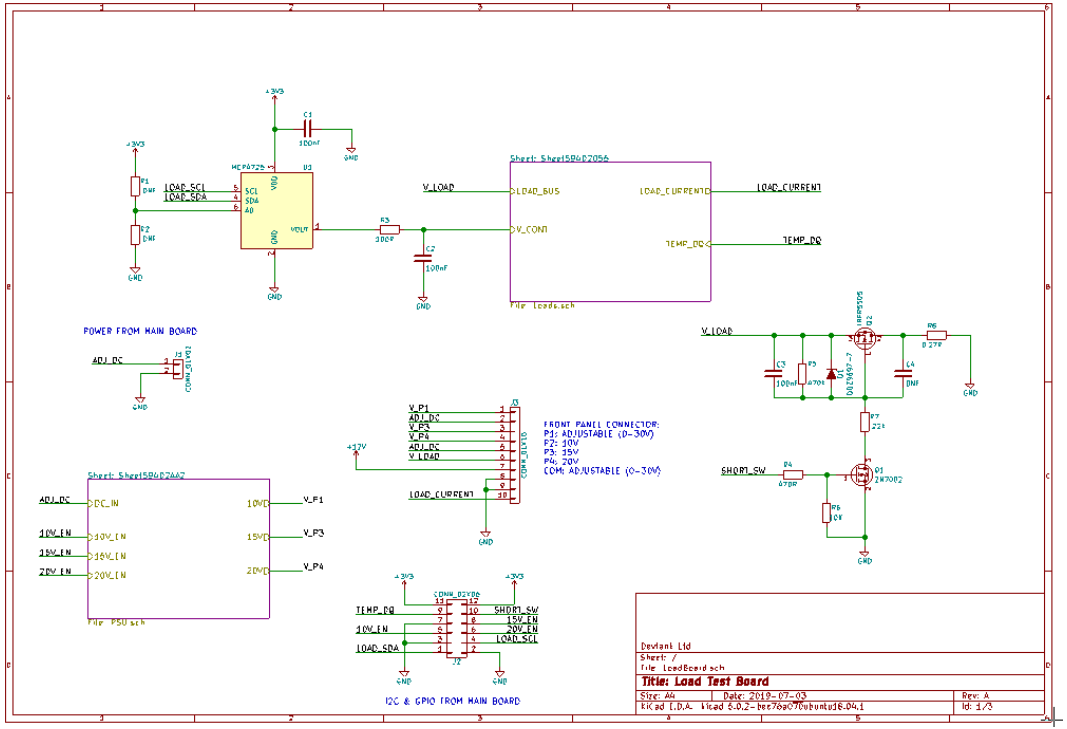
Eeschema Setup (2)

- Manage Symbol Libraries



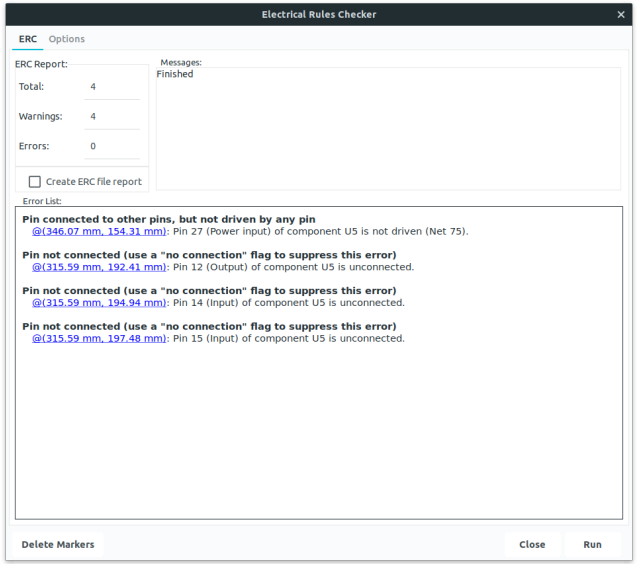
Eeshema Tool Features

- 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)



DRC Checks

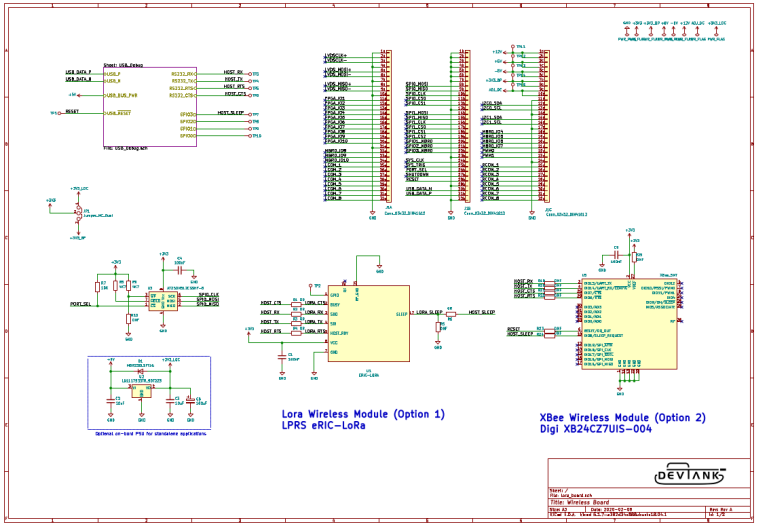
- How does it work?
- The tool is only as good as it‘s masters inputs!
- Tips and Tricks
- How to clear those tricky errors!



Starting from Scratch

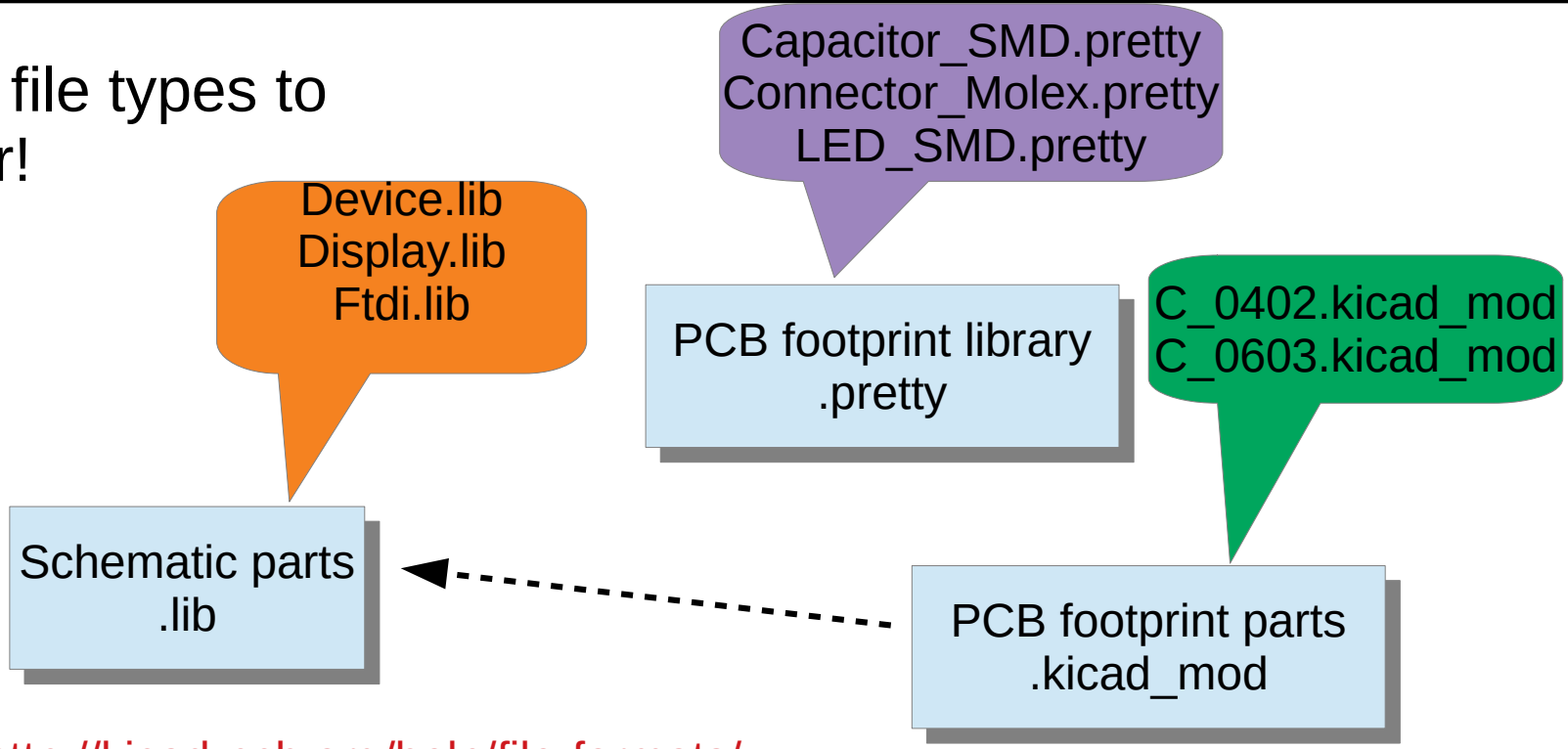
- Create a new project
- Add your own schematic parts
- Connect up the parts using wires
- Add a hiearchical block
- Add connectors
- Design rule checks
- Example (2)
- References:
  https://github.com/devtank-ltd/kicad_training/blob/master/PDF_examples/Lora_board.pdf

  https://github.com/devtank-ltd/kicad_training/blob/master/PDF_examples/eRic_LoRa_Datasheet_V1.44.pdf



Library Management

- Important file types to remember!



Useful reference: http://kicad-pcb.org/help/file-formats/

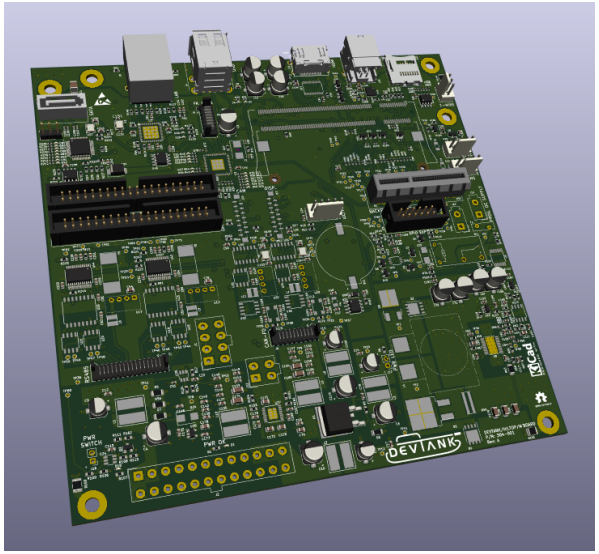
New Part Creation

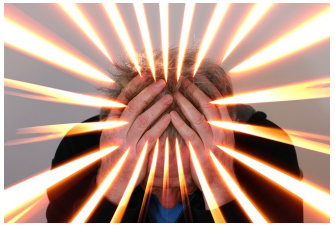
- Symbol Library Editor Tool 
  - Create the shape and connections/pins
  - Add properties and datasheet references
  - Aliases/Footprint filters
  - Duplication of parts from other libraries – don‘t reinvent the wheel!
- External Tool Wizards

  http://kicad.rohrbacher.net/quicklib.php (recommended tool)
- Worked Example (3)

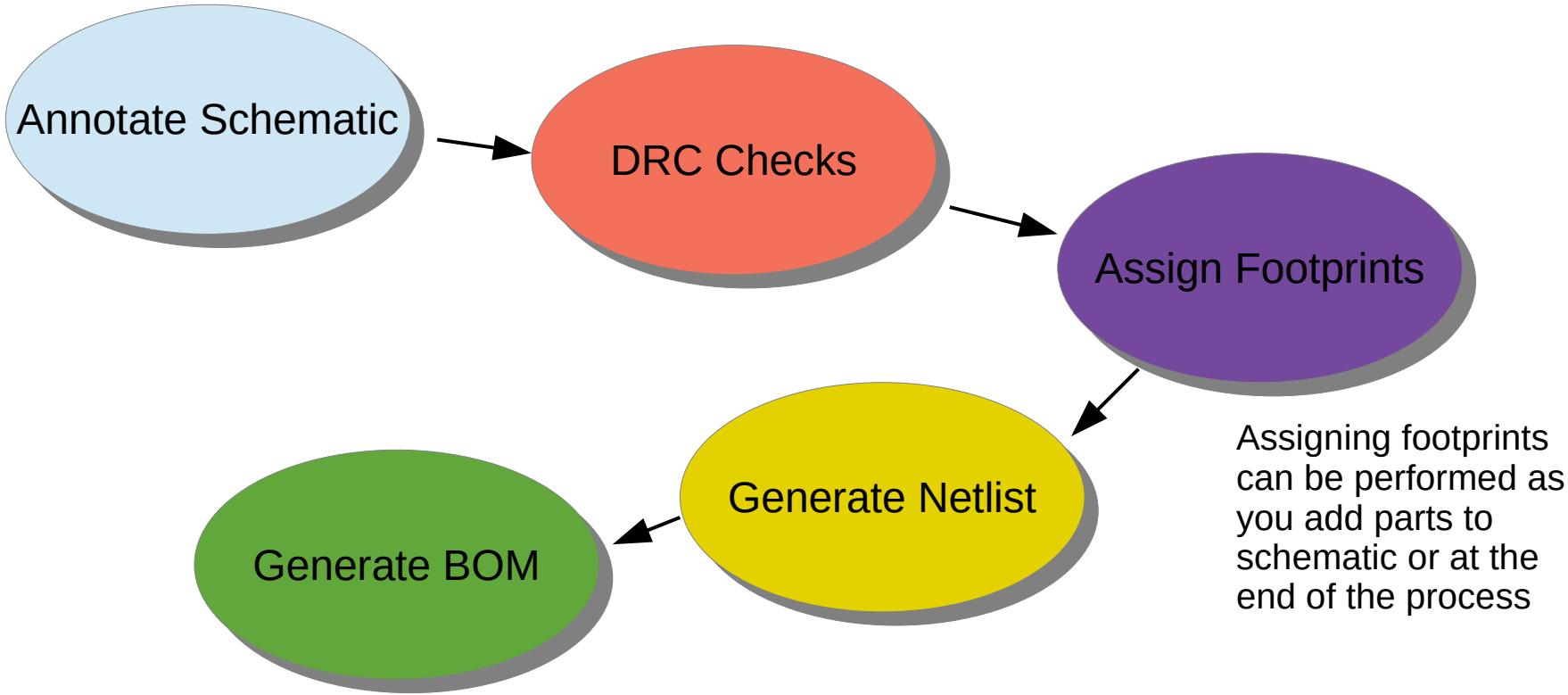
Day 1 Agenda (2nd half)

- 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.





Getting ready for PCB layout

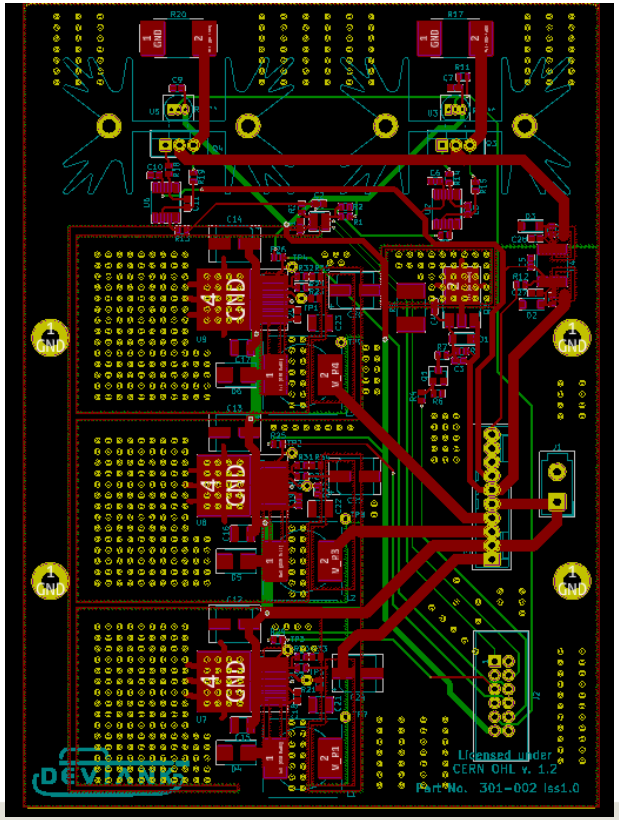


PcbNew Setup

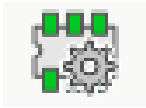
- Ensure footprint libraries are setup
  - >Manage Footprint Libraries option
- Choose your board layer stackup (note – this can be changed later)
- Define design rule settings
  - Net classes (Clearance, track width, via dia etc…
  - Custom via and track widths
- Using a common dummy project will expedite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

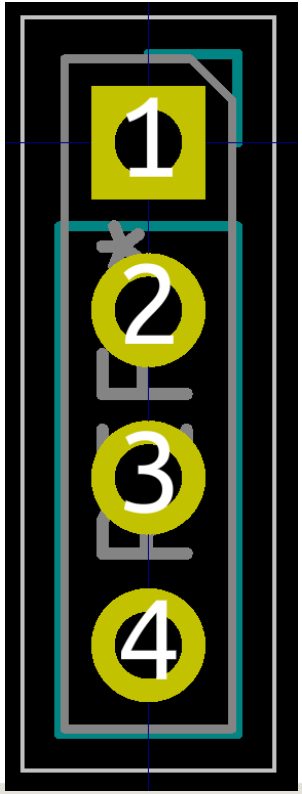
PCBNew Tool Features

- 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 (4)



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 (6)



Start a PCB from Scratch

- Setup layers
- Assign PCB footprints
- Import netlist
- Assign signal attributes
- Design board outline
- Place parts
- Track & Gap
- Design Rule Checks (DRC)
- Worked Example (5) – IoT Board Layout

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

Questions

Day 2 Agenda

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