A dedicated toolbox for Mesh + Readout Zone in KiCad

Aude GRABAS & Valmir DO NASCIMENTO JÚNIOR

Aude.grabas@cea.fr
Valmir.donascimento@cea.fr
Part 1 – Introduction to KiCad
KiCad, an open source EDA Software
Designing MM detectors at Saclay using KiCad
The plugins make KiCad very powerful

Part 2 – MM dedicated plugins
Plugin demo
Plugin development
Automated netlist generation
Next steps

Summary
KiCad, a free and open source EDA software

- created in 1992 by Jean-Pierre Charras from Grenoble University
- multi platform: GNU/Linux, Windows and Apple OS X
- supported by CERN & Society Foundation
  [https://cernandsocietyfoundation.cern/projects/kicad-development](https://cernandsocietyfoundation.cern/projects/kicad-development)
- under continuous development – v6 foreseen for end of 2021

**KiCad front panel**
TPOT MM prototype and its HV filter board for sPhenix
256 strips – 512mm x 256mm readout zone
2 inner layers with μvias, grid GND
HV contacts through FSI connectors – connection to FE through mec8 edge connectors
TPOT MM prototype and its HV filter board for sPhenix
256 strips – 512mm x 255mm readout zone
2 inner layers with µvias, grid GND
HV contacts through FSI connectors – connection to FE through mec8 edge connectors
TPOT MM prototype: insolation mask, kapton, silkscreen, mesh… all masks

TPOT RPHI ZigZag silkscreen mask

TPOT RPHI silkscreen mask

Stephan, Maxence V, Arnaud B, Aude G
But also for other projects: picosecond, omnis, babylaxo…
« The objective of [its] development is to **enhance its functionality**, making it an **efficient tool** for PCB design, which people can use to share their design information without compromising productivity.

This will ensure that there are **no artificial barriers** to the sharing of information, so that design and development knowledge can flow more freely »

Quote from: https://cernandsocietyfoundation.cern/projects/kicad-development
• What does «enhance its functionality» mean?

→ Users can **develop third-party tools, the so called PLUGINS**, that can interact with KiCad to add custom functionalities.
THE PLUGINS MAKE KiCAD VERY POWERFUL (3/3)

• How can it be an **efficient tool** for detector layout design?

→ Exploit the potential of the « **footprint generator wizard** » to automatically draw
  (1) Mesh mask (pilar) – (2) insolation mask (wall) – (3) readout zone (pixels or strips)
• How can it be an **efficient tool** for detector layout design?

Exploit the potential of the « **no artificial barriers** » to automatically generate a KiCad **netlist** that is a key element for an **efficient** and **error – free** routing process.
Part 1 in a nutshell

Saclay Bulk workshop is now using KiCad to draw masks required for all bulk process steps

Thomas Bey is using KiCad to route MM readout PCB + other MM related objects

We have decided to develop dedicated KiCad plugins for

- Mesh mask (pilars) generation
- Insolation mask generation
- Readout zone footprint
- Automated netlist generation

And now, time for demo!
• KiCad uses a two language model:

- **C++**
  - Performance
  - Portability
  - Compiled

- **Python**
  - Rapid prototyping
  - Easy to understand
  - Interpreted

[Diagram showing the relationship between C++, SWIG, and Python, leading to Core application and User Plugins with Pcbnew API.]
Creating an insolation mask

KiCad Home page

We choose a wizard
• Meshes with different patterns
• Readout zones with strips (1D and 2D)
• Problem:
  • Meshes with a large number of pads are challenging to build (memory management)
  • Rebuild the mesh preview consumes a long time
  • KiCad footprint files become too large

![Graph showing the footprint creation time vs. number of pads.](image)

- Editor doesn’t respond
- Editor keeps working

<table>
<thead>
<tr>
<th>Type</th>
<th>Taille</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fichier KICAD_MOD</td>
<td>619 438 Ko</td>
</tr>
</tbody>
</table>
• The objective: Use a CSV file that describes the connections between the detector components as an input to KiCad.

• How can we do it?

CSV file → Python script → Netlist output file (readable by KiCad)
AUTOMATED NETLIST GENERATION (DEMO)
**Footprint wizards:**
- Include a feature to generate segmented meshes
  - The pattern is repeated every X and Y

- Optimization: export mesh footprint directly as a Gerber File
  - Saves time when creating meshes with millions of pads

**Netlist Generator Script:**
- Add file verifications to ensure the plugin good functioning
- Detector symbol generation
Remember MPGD meeting from November:

- Interfacer les scripts existants de génération de schématiques, empreintes et symbole avec le plugin DetGen Kicad → ongoing

- Concevoir un script pour :
  - La génération des masques d’insolation → done
  - La génération des plans de masse → done

- Implémenter une arborescence projet (gestion librairie, versionning…) → to be done

- Implémenter un script qui prend en compte les pixels exotiques 😊 → done
- Implémenter un script qui gère le multiplexage au niveau de la schématique → done (but to be tested ;)

- ….. Et plus selon besoin utilisateur

A lot has been achieved thanks to Valmir !
Thank you for your attention

aude.grabas@cea.fr
valmir.donascimento@cea.fr
• Extensive use of the Pcbnew API

Python scripts
  • Insolation, mesh and detector wizards

Pcbnew API

C++ functions

• Use of existing plugins
WHEN ELECTRONICS DESIGNERS MEET DETECTOR DESIGNERS

Freely inspired from https://xkcd.com/730/ and from Theodore Zorbas Poster
Definition: a **symbolic** representation of the **physical connection** between different electrical **components** of a system

Physical connections == **green line** with a name (**net name**)  
Electrical component == **standardized drawing** or ‘box’

Example in the electronic world: RC filter for HV supply
MPGD Definition: « a symbolic representation of the physical connection between the readout zone and the FE board connectors and the HV elements

Physical connections == green line with a name (net name)
Component == ‘box’ (no standard component for detector readout zone)

2 Examples in the detector world:
• 16 pixels / strip readout zone connected to 1 FE connector
• 16 pixels / strip multiplexed signals connected to 1 FE connector
**Wikipedia Definition:** A footprint or land pattern is the arrangement of pads or through holes used to physically attach and electrically connect a component to a PCB.
**MPGD Definition**: A detector footprint is the arrangement of *pixels* or *strips* of a read out zone to physically attach and electrically connect it to a detector PCB.
MPGD Definition: (1) A detector footprint is the arrangement of **pixels** or **strips** of a read out zone to physically attach and electrically connect it to a detector PCB

(2) A **mask footprint** is a pattern designed for the bulk process machine: insolation mask for the mesh plots, silkscreen mask for resistive paste, insolation protection wall,…
Definition: A netlist is a list of components in a circuit and a list of the nodes that connect the components to each other.