



FOSDEM 2017 Presentation.

Prepared by Wayne Stambaugh

Project Status

- Stable version 5 pre-release soon.
- Version 6 development road map complete.
- Increased user interest.
- Increase in commercial interest.
- Hackathons.
- **Digi-Key** purchased the kicad.org domain and redirected it to the official KiCad website.



Stable Release 5, New Features

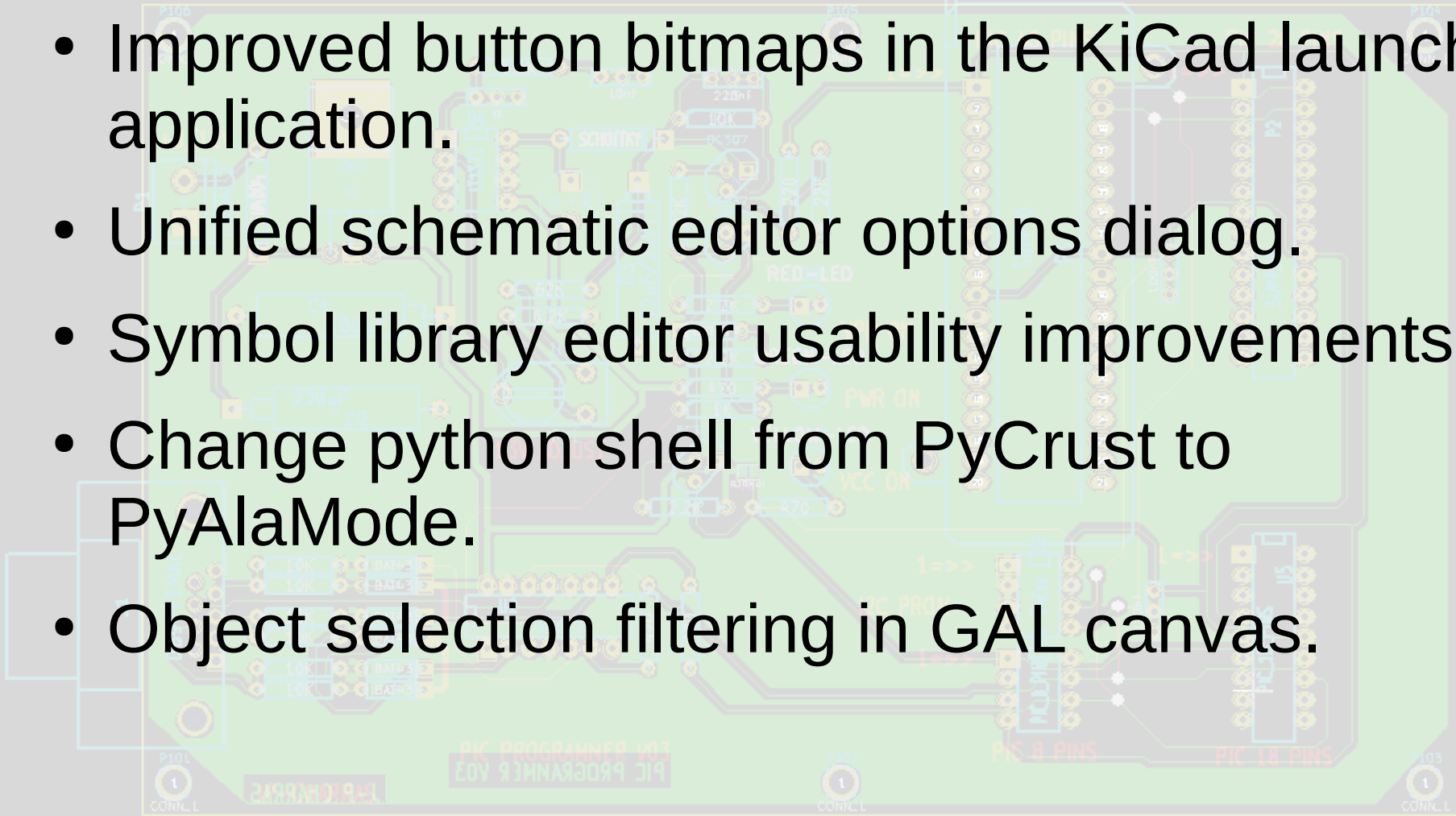
- The KiCad project now uses [git](#) for version control, yeah!!!
- Integrated [SPICE](#) simulation using [ngspice](#) library.
- Schematic editor symbol field alignment tool.
- Net highlighting tool in the schematic editor.
- Regular expression library search capability for symbol library searching.
- Directly update board from schematic editor without intermediate netlist file.

Stable Release 5, New Features Continued...

- All new 3D viewer with **ray tracing**.
- Support for STEP models in boards.
- Direct export to STEP.
- Rounded rectangular pad support.
- Spread footprints on first load of netlist.
- Directly update schematic netlist from board editor without intermediate netlist file.
- GAL canvases have flipped board view.

Stable Release 5, UI Improvements

- Improved button bitmaps in the KiCad launcher application.
- Unified schematic editor options dialog.
- Symbol library editor usability improvements.
- Change python shell from PyCrust to PyAlaMode.
- Object selection filtering in GAL canvas.



Stable Release 5, Code Improvements

- 3D model library plugin manager to accommodate more 3D model file types.
- Schematic I/O plugin manager to provide platform for importing schematics and symbol libraries from third party applications.
- Implement new schematic and symbol library file formats.
- Complete porting of all editing tools in Pcbnew to new tool framework.

Version 6 Road Map

- Port schematic editor canvas to graphics abstraction layer rendering.
- Port schematic editor over to the new tool framework.
- Add GitHub support for accessing symbol libraries.
- Shared object for common schematic code.
- Schematic net highlighting when drawing and editing board traces.
- Allow use of system font in board editor.
- Eagle schematic and symbol library plugin.

Version 6 Road Map, Continued...

- Add Python scripting support for schematic objects.
- Electronic rules check (ERC) improvements.
- Footprint editor library management usability improvements.
- Support for complex pad shapes.
- Pin and gate swapping with automatic forward and backward annotation between schematic and board editors.
- Full clipboard support for schematic and board editors.
- Improve board editor design rule check (DRC) coverage.

Version 6 Road Map, Continued...

- Add support for keep out zones on boards and footprints.
- Add wild card and regular expression search to footprint selection dialogs.
- Hatched zone filling.
- Board file support for externally linked objects.
- Board file support for groups (snippets).
- Board stack up impedance calculator.
- Support for per net color and visibility of ratsnest.

The Distant Future

- Add microwave tool support to board editor.
- Support for importing ODB++ format.
- Tools for part database management.
- Move to plugin based architecture for all major tool implementations such as auto-router, push shove router, etc.
- Unit testing for low level objects.
- Integration with component vendors website for automated ordering.

The End

Thanks to all of the developers who contribute their valuable time and talent to the KiCad project.

Thank you for your interest and continued support of the KiCad project.

Special thanks to Javier Serrano and CERN for making this presentation and development room possible.