FOSDEM 2021 Presentation
KiCad Project Status

Prepared by Wayne Stambaugh

Project News

- KiCad drives the Open Ventilator Project.
- Version 6 feature freeze on October 1st 2020.
- Version 6 stable release planned for end of Q1 2021.
- Stable version 5.1.9 released on December 28th 2020.
- Funding campaign from December 15th 2020 to January 15th resulted in donations of more than $14K.
- The KiCad Services Corporation matched an additional $10K.
- All project repositories moved to GitLab.

Project News, Continued

- Two new developers promoted to the lead development team.
- Another significant uptick in new developer contributions.
- Moved to Weblate for translations.
- Hired professional graphics designer to create new bitmap theme.
- New bitmap policy to ensure all future images are consistent with the new theme.
- KiCon 2021 planned for September at CERN.

Version 6.0, Core

- New image theme and improved UI coherence.
- Configuration file format changed to JSON.
- Support for multiple version configurations with migration tool.
- Removed legacy canvas support.
- Custom color themes in separate configuration file.
- Use infobar to reduce modal information dialogs.
- Variable expansion in most text objects.

Version 6.0, Project Manager

- Project archive utility.
- CADSTAR project import.
- Project file format changed to JSON.
- Symbol and footprint library editors usable with no project loaded.
- Project tree view files moved to trash.
- Save as support for entire project.

Version 6.0, Schematic Editor

- New file format, library symbols now embedded in schematic.
- Full copy and paste support.
- Custom line, wire, and bus color, width, and style.
- Custom junction color and diameters.
- User defined page numbering.
- Back annotation from board changes.
- Wire drawing improvements.
- Custom color themes.

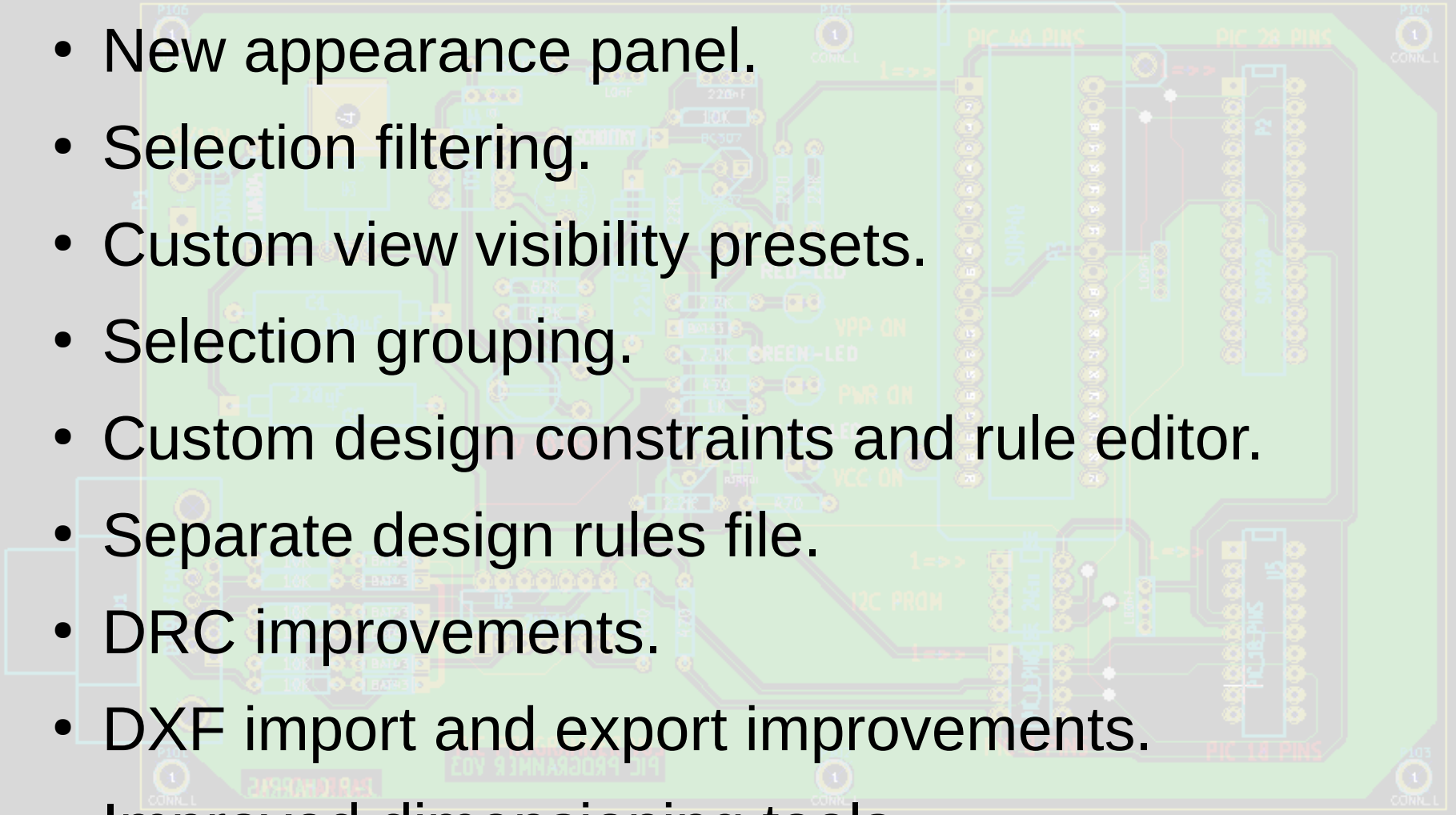
Version 6.0, Schematic Editor Continued...

- Bus definition improvements.
- Bus unfolding tool.
- Improved bus connectivity.
- ERC improvements.
- Object property editing tool.
- Allow hierarchy navigation tool to remain open.
- Inter sheet references.

Version 6.0, Symbol Library Editor

- New symbol library file format.
- Improved inheritance model.
- Alternate pin definitions.
- Legacy library conversion tool.
- Legacy libraries are read only.
- Cut, copy, and paste symbols between libraries.
- Cut, copy, and paste graphical items.
- Support for BOM and board only symbols.

Version 6.0, Board Editor



- New appearance panel.
- Selection filtering.
- Custom view visibility presets.
- Selection grouping.
- Custom design constraints and rule editor.
- Separate design rules file.
- DRC improvements.
- DXF import and export improvements.
- Improved dimensioning tools.

Version 6.0, Board Editor Continued…

- Board layer stack manager.
- Support for user defined layers.
- All layers can now be renamed.
- Support for curved tracks.
- Push and shove router improvements.
- Curved rats option and custom net coloring for ratsnest.
- Support for importing SVG files.
- Thermal relief for custom pads.
- Allow arcs in polygons on non-copper layers.

Version 6.0, Board Editor Continued…

- Support for hatched zones and multi layer zones.
- Support for castellated edge connections.
- Drag footprint with traces connected.
- Add opacity to 3D models.
- Improved default color scheme.
- Back annotation to schematic.
- Geographical annotation.
- Altium and Fabmaster (OrCAD ASCII) board import.

Version 6.0, Footprint Editor

- New appearance manager.
- Object selection filtering.
- Custom layer visibility presets.
- Improved object snapping.
- Allow keep out areas in footprints.
- Allow board edge cuts in footprints.
- Chamfered rectangular pads.
- Custom chamfered/rounded rectangular pads.

Version 7, Core

- Altium library support.
- Database driven atomic libraries.
- Background library loading.
- PNG export for third party tool chains.
- Output automation tool.
- Consolidate overlay widgets into a single system (possibly using ImGUI).
- Keyboard access to all tool actions via an overlay UI widget.
- Implement BGFX graphics abstraction layer.

Version 7, Core Continued...

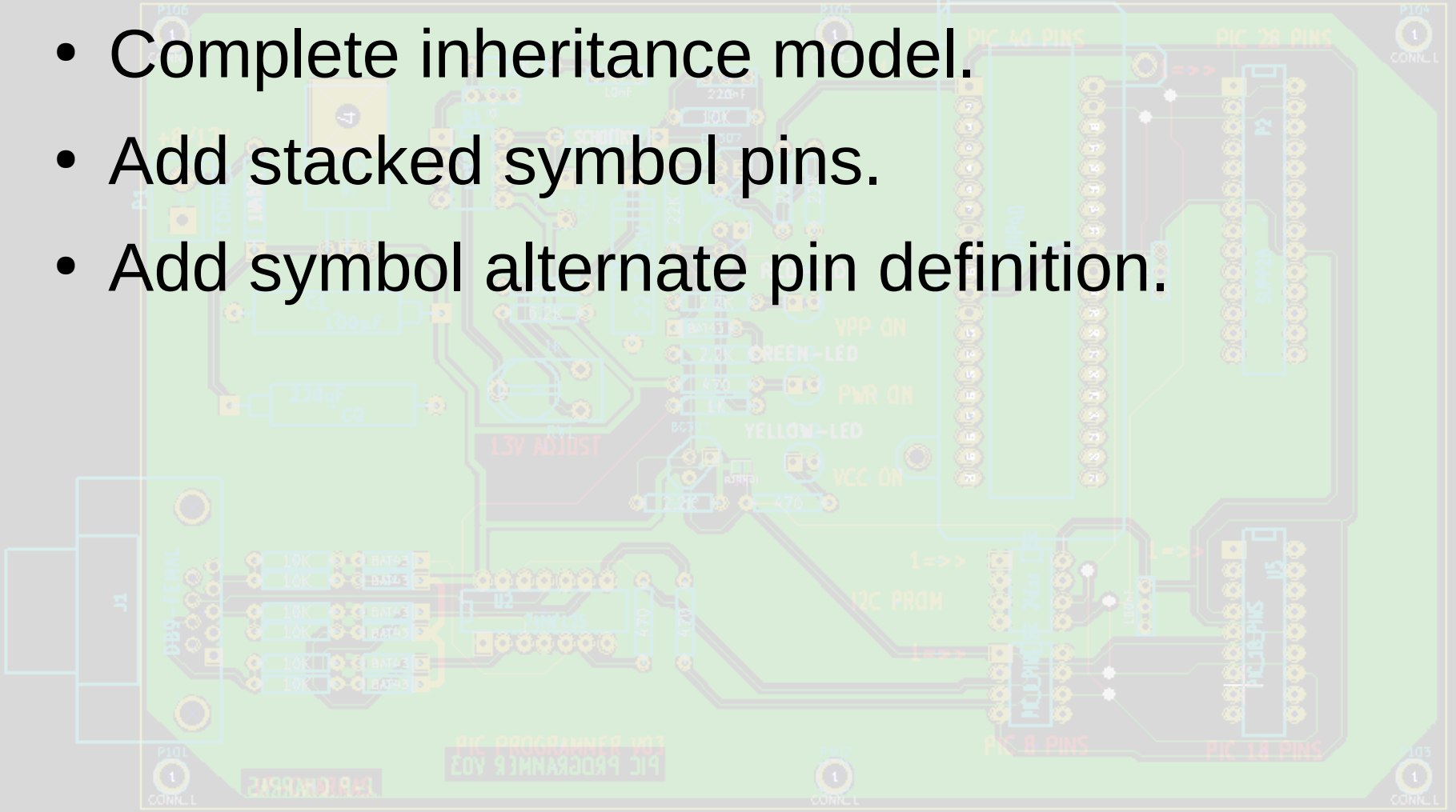
- Custom tool tip widget to allow rich contextual help.
- Embed worksheet in schematic and board files.
- Improve grouping support.
- Allow multiple boards per schematic.
- Multiple document interface.
- Single binary and emulated stand alone mode.
- True net tie implementation.
- Object common properties editor.
- Pin and gate swapping.

Version 7, Eeschema

- Unify symbol browser, symbol viewer, and footprint assignment tool (CvPcb).
- Edit multiple object common properties.
- Edit power symbols in schematic.
- Visual enhancements for graphical and text objects.
- Support non-hierarchical multiple sheet schematics.
- Spice simulator improvements.
- Support hierarchical sheets in Eagle schematic importer.

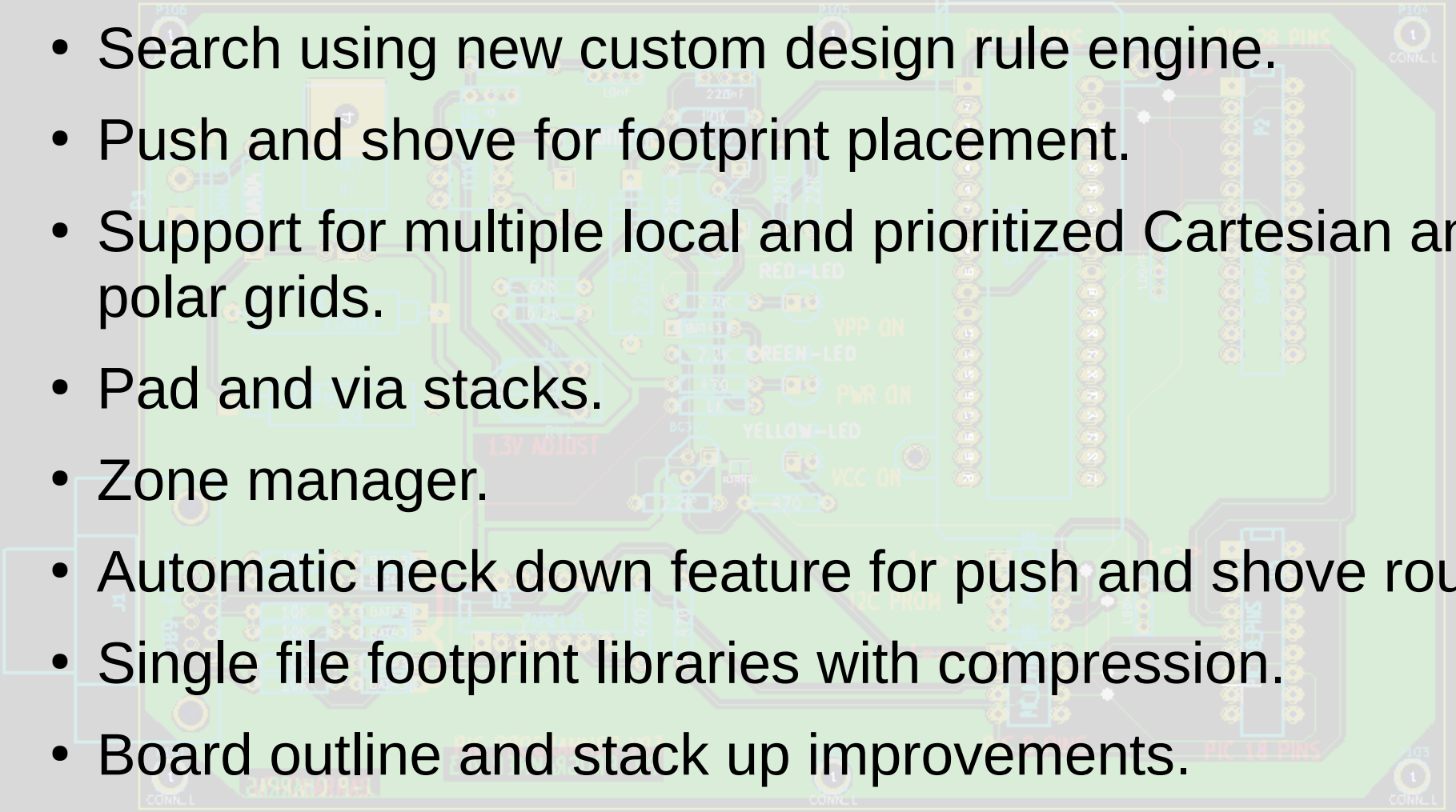
Version 7, Symbol Editor

- Complete inheritance model.
- Add stacked symbol pins.
- Add symbol alternate pin definition.



Version 7, Pcbnew

- Search using new custom design rule engine.
- Push and shove for footprint placement.
- Support for multiple local and prioritized Cartesian and polar grids.
- Pad and via stacks.
- Zone manager.
- Automatic neck down feature for push and shove router.
- Single file footprint libraries with compression.
- Board outline and stack up improvements.



Version 7, Pcbnew Continued...

- Bidirectional 3D viewer cross probing.
- Implement guard rings, zones, and tracks.
- Granular locking feature.
- Multiple net (bus) routing with via placement modes.
- Overhaul 3D model library look up.
- BGA fan out tool.
- IPC 2581 support.

The End

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

Thank you to all of our sponsors and everyone who has generously donated to KiCad.

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

Special thanks to Seth Hillbrand for making this presentation and development room possible.

Hope to see you all at KiCon 2021.