

xCheck Release History

Version 2.3 (Nov 29, 2015)

Enhancements

- Reports zero-length traces as DRC warnings

Bug Fixes

- Fixes crash due to zero-length traces
- Fixes problem saving and restoring empty PCB and schematic file names

Version 2.2 (Nov 7, 2015)

Enhancements

- Added sheet name to more schematic warnings and errors
- Remembers last schematic and PCB filename when program is closed and re-run
- Begins browsing for schematic or PCB filename at location of currently chosen file (if any)

Bug Fixes

- Fixes Xengine permission failure bug
- Fixes problems running on machines set to 120 DPI

Version 2.1 (July 16, 2012)

New features

- Added **stats** report tab
- Added buttons to easily edit PCB in ExpressPCB or schematic in ExpressSCH

Enhancements

- Much faster “minimum overlap” processing between planes
- “Poorly connected” DRC warnings can be disabled by setting **minimum overlap** rule to 0
- Special exceptions made for some built-in library PCB components that assign pin numbers to mounting tabs (e.g. TO-220, TO-220-5). Eliminates verification errors since tab pin does not exist in schematic symbol
- Added “Tie Nets” schematic component and PCB components (4) to enable tying together multiple nets without causing schematic or verification errors
- Added special exceptions for “Tie Nets” components in PCB analysis and in BOM generator
- Reports now refer to components by name if the Part ID is blank (rather than a generic name such as UNNAMED123)
- Improved descriptions in DRC violations involving arcs and circles
- Nuisance DRC warnings suppressed for objects that are not part of a true net
- Suppresses DRC warnings to manually check copper-layer text if there is no possibility of the text intersecting multiple objects – reduces many nuisance warnings
- Progress “fuel gauge” is more linear (doesn’t get stuck at 99% for a long period, for example)
- Improved program icon at various resolutions

Bug Fixes

- Fixed problem where a linked schematic located in the same folder as the PCB would fail to be found by xCheck. Therefore no schematic-related reports would be created if the **Schematic File** box was blank. Would not fail like this if PCB and schematic were in different folders.
- Fixed crash that occurred if doing a Ctrl-C (or Edit->Copy All) when the report that was currently displayed was blank
- Fixed rare Xengine crash that was very dependent on the number and location of components and planes in the PCB
- Fixed crash caused by a “no connect” symbol attached to nothing in schematic
- Fixed schematic netlist errors caused by a comma present in the NetName of a symbol
- Fixed bug where abort button could take a long time to respond during plane collision analysis

Version 2.0 (May 29, 2012)

Merged Xpcb, XGpcb, and Xsch tools, renamed **xCheck** (command line no longer supported)

New features

- Added schematic checking
- Displays formatted reports within the program (no longer relies on external viewing tool)
- Schematic and PCB netlist differences are highlighted for easier comparison
- Reports board statistics (number of holes, pads, components, etc.) during analysis
- Added DRC rule presets to match various ExpressPCB board services
- Added ability to print reports
- Added ability to copy reports to the clipboard
- Includes special “No connect” schematic symbol which xCheck understands

Enhancements

- Overhauled user interface
- Added ability to abort analysis, if desired
- PCB netlist now inherits named nets from schematic whenever possible
- Cleaner netlists: no longer lists single-node nets
- Schematic and PCB netlists are sorted by PartID for easier comparison and readability
- Improved descriptive wording for DRC violations
- Keyboard shortcuts added for “power users”
- Added **Missing Pads** information to verification report

Bug Fixes

- Fixed PCB analysis bug where solo pads (not part of a component) with non-0 pin numbers could be mistakenly assumed to be connected to each other
- Fixed bug where a component’s duplicate pins or mounting pads (unnumbered pins) would be shown as multiples in PCB netlist
- Fixed bug where, in verification report, extraneous pads could be missing from **Extra Component Pads** list if they were also part of an extraneous net

Version 1.3 (May 1, 2012)

- Supports systems with Windows installed on a drive other than C:
- Better error handling if linked schematic not found

- No longer case-sensitive to RefDes (part ID) in schematic and PCB
- Xpcb -n option changed to specify temp netlist pathname (can no longer feed in netlist)
- No longer fails if \Windows\Temp folder doesn't exist

Version 1.21 (April 28, 2012)

- Fixed XGpcb crashing on launch on Win 7 machines
- Fixed tab ordering on Rules tab

Version 1.2 (April 26, 2012)

- Added support for customizing DRC rule thresholds
- Fixed bug related to nets with large number of nodes

Version 1.1 (April 21, 2012)

- Added XGpcb graphical tool
- Package now distributed as a Windows installer

Version 1.0 (April 17, 2012)

- First release version