Minor changes and bug fixes

DipTrace 5.2.0.4 (compared to 5.2.0.3)

General

1. Added special characters (©, ®, etc.) to the vector font.

PCB Layout

1. Bug-fix: Via Stitching could stop working for the board, regions, or selectively when certain Copper Pour options were enabled.
2. Bug-fix: Incorrect updating of existing vias during Via Stitching.
3. Bug-fix: STEP export failure on a specific model.
4. Bug-fix: Excessive line spacing in text objects when exporting to ODB++ and IPC-2581C.
5. Bug-fix: Distorted TrueType fonts in IPC-2581C export.
6. Bug-fix: Direct3D graphics failure on some drivers after exiting Sleep, Screensaver, or Lock Screen modes.
7. Bug-fix: Copper Pour fill failure when using class-to-class clearance settings in the 64-bit version.

DipTrace 5.2.0.3 (compared to 5.2.0.2)

General

1. Added support for displaying Latin characters with diacritics in the vector font.

PCB Layout

1. Bug-fix: In some cases, the board body appeared transparent when opening the exported STEP file in SolidWorks.

2. Bug-fix: IPC-2581C export failed when an arc was present inside a component.

3. Bug-fix: Screen redraw issues occurred when resizing the drawing area while object highlighting was active.

4. Improved Walk Around and Push Traces functionality.

5. Updated Via Stitching algorithm to properly handle nested Copper Pour regions.

6. Improved overall differential pair routing algorithms.

DipTrace 5.2.0.2 (compared to 5.2.0.1)

General

1. Bug-fix: Occasional false component highlighting in the other application during Cross Selection.

PCB Layout

1. Improved Walk Around and Push Traces functionality.

2. Bug-fix: Various issues in differential pair routing fixed.

3. Bug-fix: Missing vias along board edges during Via Shielding resolved.

4. Bug-fix: Changes in *Related Schematic* were not added to the Undo list and were reverted when undoing a subsequent action.

5. Bug-fix: Dimensions could shift after updating the board from a schematic containing new components.

Schematics

1. Bug-fix: Empty message shown when simulating certain components in SPICE.

2. Bug-fix: *Attached Pattern* dialog always opened the first section of a component instead of the current one from which it was called.

DipTrace 5.2.0.1 (compared to 5.2.0.0)

General

1. Bug-fix: Occasional false component highlighting during Cross Selection.
2. Bug-fix: Minor issues with text and arc boundaries when exporting images.

PCB Layout

1. Bug-fix: Incorrect layer switching when clicking a trace segment between vias of different types.
2. Bug-fix: Incorrect placement of Mouse Bites caused by an arc in the board outline.
3. Bug-fix: Incorrect bounding box for inverted text with increased line spacing.
4. Bug-fix: Vector image redraw issues when mirroring, including reflection across the wrong axis.

DipTrace 5.2.0.0 (compared to 5.1.0.3)

General

1. Improved KiCad 9 import.
2. Bug-fix: Application failed to start at 1024×768 resolution with high scaling.

PCB Layout

1. Bug-fix: Freeze when importing certain ODB++ files created in Altium Designer.
2. Bug-fix: Crash when sequentially importing two KiCad files.

3. Bug-fix: Incorrect net-to-netclass assignment in KiCad import templates.
4. Bug-fix: Unit reading issue when importing Altium files.
5. Bug-fix: Crash when importing certain EasyEDA files through Altium importer.
6. Bug-fix: In specific Altium files, traces from the Bottom layer incorrectly appeared on the Top layer.
7. Bug-fix: In specific Altium files, pads were imported on the wrong layer.
8. Bug-fix: In specific Altium files, components were imported on the wrong side of the board.
9. Bug-fix: In specific Altium files, traces were shifted.
10. Bug-fix: Arcs on copper pours incorrectly imported from Altium files.
11. Bug-fix: Board outline sometimes not recognized in Altium imports.
12. Bug-fix: DipTrace generated mixed layers in DSN files.
13. Bug-fix: Error when importing certain ODB++ files from EasyEDA.
14. Bug-fix: Rotated pad holes not recognized in ODB++ import.
15. Export ODB++: Improvements for `units = mm`.
16. Export BOM (HTML): Improved *Highlight by Mouse* functionality.
17. Bug-fix: Crash in DRC when copper pours extended far beyond board outline.
18. Bug-fix: Component moved unintentionally when selecting an error in Schematic Compare results and pressing arrow keys.
19. Bug-fix: Incorrect length of curved traces in *Layout Information* for specific designs.
20. Bug-fix: Joining two unfinished differential pair stubs during *Push and Show*.
21. Bug-fix: Certain 3D models not exported to STEP due to invalid model type.
22. Bug-fix: On circular boards, some placed components remained in the *Placement by List* dialog.

23. Bug-fix: Incorrect copper pour priority handling during via stitching in some cases.
24. Bug-fix: Extra vias could appear during complex multi-layer differential pair routing.
25. Bug-fix: Differential pair ViaStyle not switching to Blind/Buried under certain conditions.

Schematics

1. Export Netlist: Changed default file extension.
2. Bug-fix: Occasional bus connectivity issues in Altium imports.
3. Bug-fix: Crash when importing specific KiCad schematics.
4. Bug-fix: PDF export omitted spaces in sheet names.

Component Editor

1. Bug-fix: XML export failed after plugin execution.

Pattern Editor

1. Bug-fix: XML export failed after plugin execution.
2. Bug-fix: Pads not imported from certain PADS files.
3. Pattern Generator (THT 5 Vias): Pattern generated with a Y offset.
4. Pattern Generator (PGA): Incorrect placement of W and W1 labels on the preview image.

DipTrace 5.1.0.1 (compared to 5.1.0.0)

General

1. Correct window and message positions for vertical multi-monitor systems.

PCB Layout

1. Improved editing of related trace segments and the option to turn it off.
2. Removing overlapped net names on traces (was possible in solid regions made of traces).
3. Removed optimizing of sharp trace angles for moving and rotation of components with connected traces as a single block.
4. Polishing of push and shove algorithm.
5. Export DSN: parentheses are replaced with underscores for compatibility with some autorouters.
6. Import PADS VX: fixed issues with mounting holes, nets, non-signal layers, via styles.
7. Import Altium ASCII: fixed issues with mirroring and rotation of footprints and obround pad holes.
8. ODB++ Import and Export: fixed pads with incorrect blind holes, component markings, saving import result without “read-only” warning.

Schematics

1. Import PADS VX: fixed issue with pin names

Component Editor

1. Scrolling pin table with mouse-wheel in pin manager.

DipTrace 5.1.0.0 compared to 5.0.0.1

PCB Layout

1. Bug-fix: highlight of dimensions in hidden layer of the opposite side of the board

Schematics

1. Bug-fix: verification of hierarchy reported false “connected differential pair” error in hierarchical schematic with several instances of the block and pairs, connected to them via hierarchical pins.

DipTrace 5.0.0.1 compared to 5.0.0.0

General

1. Restoring window size and position when reopening any of the programs.

PCB Layout

1. Push and Shove Router:
   - significant improvement in pushing vias and "traces with vias";
   - several issues and bugs have been found and fixed.
2. Bug-fix: graphics behavior for round-rect/obround pads in "Highlight Net" feature in PCB Layout (graphics speed optimization was made in 5.0, but some things didn't work as expected).
3. Bug-fix: BOM was not made correctly for the components with the assigned supplier from Octopart. We were forced to remove Octopart support due to their updated policy. Starting from 5.0.0.1 BOM works correctly for all components, and we are looking for a new component supplier API/engine.
4. Bug-fix: significant speed and progress displaying behavior in Via Stitching feature.
5. Bug-fix: traces were not imported from the latest versions of KiCAD file format.
6. Bug-fix: pin signal for spice model was entered and saved correctly only in the field under signal table, but not directly in the table.

Schematics

1. Bug-fix: BOM was not made correctly for the components with the assigned supplier from Octopart. We were forced to remove Octopart support due to their updated policy. Starting from 5.0.0.1 BOM works correctly for all components, and we are looking for a new component supplier API/engine.
2. Bug-fix: non-standard short-circuit settings may cause reporting of nonexistent errors after Schematic start and until ERC window opening.