

## List of fixed bugs in DipTrace 4.3.0.5 if compared to 4.3.0.4

### General

1. Bug-fix: The sound when hitting the Enter key in the SnapEDA search bar.

### PCB Layout

1. Bug-fix: Sometimes moving a track segment moves adjacent segments incorrectly.
2. Bug-fix: Teardrop for Trace Width Difference may disappear when moving segments.
3. Bug-fix: Panning moved too far in the File/Preview window after zooming out to 0 and back.
4. Bug-fix: Unconnected island in Copper Pour may be not removed.
5. Bug-fix: An incorrect board outline may be created for Tab Routing.
6. Bug-fix: Arcs may be imported incorrectly from KiCad v6 and later versions.
7. Bug-fix: When importing from XML, the Default Width for Layer setting is lost for Shape.
8. Bug-fix: The 3D model by Bottom Comp Outline is displayed without taking into account the board thickness.
9. Bug-fix: 3D Preview freezes if there are polylines with 0 or 1 point.
10. PCB ordering for Aisler works again.

### Schematics

1. Bug-fix: Panning moved too far in the File/Preview window after zooming out to 0 and back.
2. Bug-fix: When Flip Horizontal / Vertical command is applied, the position of some shifted Pin Names is lost.

### Component Editor

1. When Flip Horizontal / Vertical command is applied, the position of Pin Name is lost.

## List of fixed bugs in DipTrace 4.3.0.4 if compared to 4.3.0.3

### PCB Layout

11. Bug-fix: In Windows XP, 7, 8.1, some STEP models are not exported.
12. Bug-fix: An error in pattern name search in the ODB++ structure may cause the export to fail.
13. Bug-fix: Inverted text is not mirrored for print preview / print if "Flip text automatically" option is off.
14. Bug-fix: Changes in Dimension Properties are not updated immediately on the Properties panel.
15. Bug-fix: When the Change Side command is applied, Angle is not immediately updated on the Component Properties panel.

## **Schematics**

3. Selection of very small components with a mouse click has been improved.
4. Bug-fix: Change of pattern is not immediately updated on the Component Properties panel.

## **Pattern Editor**

2. Bug-fix: The display of a hole with a negative diameter in a through pad has been removed.

## **List of fixed bugs in DipTrace 4.3.0.3 if compared to 4.3.0.2**

### **PCB Layout**

16. Bug-fix: Pad and Fiducial are captured by the mouse in an inactive layer.
17. Bug-fix: Ellipse Pad with different Width and Height merges with Copper Pour if Hide Pad Ring in Signal Layer is enabled.
18. Bug-fix: Arcs in the pattern cause a crash when importing PADS ASCII.
19. Bug-fix: In 3D Preview, an obround cutout in the board is displayed as an ellipse.
20. Bug-fix: In Direct3D and OpenGL modes, when Pause Routing is enabled, paused trace disappears until mouse click.

### **Schematics**

5. Bug-fix: Spice Simulator fails to run on Windows XP.

### **Component Editor**

3. Bug-fix: Attaching a pattern to a component with an empty name results in a crash.

## **List of fixed bugs in DipTrace 4.3.0.2 if compared to 4.3.0.1**

### **General**

1. Bug-fix: Components with special characters in the Part Number are not downloaded from SnapEDA.

### **PCB Layout**

21. Bug-fix: Marking settings may work incorrectly with multiple selection of components.
22. Bug-fix: Hundredths are not displayed for the rotation angle value in the Component Properties window.
23. Bug-fix: In Direct3D mode, True Type text may not be displayed in the signal layers.
24. Bug-fix: Clicking the check mark in the Default column affects the presence of the component in the Assembly Variant.

25. Bug-fix: Rotated Ellipse and Obround pads are exported to DSN incorrectly.
26. Bug-fix: Text is shifted when importing from DXF.
27. Bug-fix: Square through-hole pads are imported from PADS with zero hole diameter.
28. Bug-fix: Chamfered and Rounded pads are not imported from PADS.
29. Bug-fix: The "\*" symbol in the name of the component causes the import from PADS to fail.
30. Bug-fix: in KiCAD import, Patterns on the Bottom layer are imported with a "Flipped" attribute.
31. Bug-fix: Hole offset of the pad in the Plane layer is not taken into account in the ODB++ export.
32. Bug-fix: The orientation of the D-Shape pad with an obround hole on the Plane layer is incorrect in the ODB++ export.
33. Bug-fix: STEP models with unicode characters in their name or path are not imported.

### **Schematics**

6. Bug-fix: Marking settings may work incorrectly with multiple selection of components.
7. Bug-fix: In the SPICE Model window, Hint in the search field is displayed in Cyrillic.
8. Bug-fix: Very small components are impossible to select with a mouse click.

### **Pattern Editor**

1. Bug-fix: STEP models with unicode characters in their name or path are not imported.

## **List of fixed bugs in DipTrace 4.3.0.1 if compared to 4.3.0.0**

### **PCB Layout**

34. Bug-fix: Teardrops are not shown if the signal layer color is white.

### **Schematics**

1. Bug-fix: Data changed in the Selected Parts window are not updated in the schematic immediately.
2. Bug-fix: Grouping of objects is lost when switching to another schematic page.

## **List of minor changes and fixed bugs in DipTrace 4.3.0.0 if compared to 4.2.0.1**

### **General**

1. Bug-fix: Empty textboxes in the design area may cause the program to crash.
2. Bug-fix: Programs failed to start if *libgroups.dat* is empty.

## PCB Layout

1. Bug-fix: Network pads with custom color settings were not highlighted.
2. Bug-fix: Pattern was not transferred correctly to the Bottom Layer when the "Copy Placement and Routing" command is used.
3. Bug-fix: Save to Library command may lead to loss of the binding of the size and shape in the pattern.
4. Bug-fix: Components may not be snapped to the grid when placed using the Placement by List feature.
5. Bug-fix: Crash when creating jumper wires in a new project without nets.
6. Bug-fix: The Layers window may not close after opening Layer Stackup Templates if real-time DRC is off..
7. Bug-fix: Pressing the "-" or "+" buttons while creating a connection may result in the program freezing.
8. Bug-fix: The program sometimes crashes when opening a project in OpenGL or DirectX mode.
9. Bug-fix: Pattern preview area was not scaled immediately when changing the width of the library panel for the first time.
10. Bug-fix: Zoom Window did not respect the displayed Libraries Panel.
11. Bug-fix: The program may freeze during a long session in OpenGL mode.
12. Bug-fix: Scrolling the mouse wheel immediately after opening the Layer Stackup window changed the Template.
13. Bug-fix: Incorrect size value may be displayed in the Print Preview.
14. Bug-fix: Changing the All Layers checkbox status in the picture properties did not affect the layer selection lock immediately.
15. Bug-fix: When updating from schematic, custom net color settings may be lost (changed).
16. Bug-fix: When updating from schematic, mounting holes added as components in schematic and then removed, were deleted from PCB. The same with very outdated PCB files.
17. Bug-fix: ViaStyles were not updated immediately after changing the number of layers in the project.
18. Bug-fix: In the Component Properties dialog, the order of parameters in the Parameters and Markings tabs were different.
19. Bug-fix: TrueType marking of the bottom component may not be displayed in WinGDI mode with the Mirror option enabled.
20. Bug-fix: When copied, components were added to the Design Cache together with a numerical index in RefDes.
21. Bug-fix: Neck-down worked incorrectly for Small Pads.
22. Bug-fix: Rectangular terminals (Corner Radius = 0) were not exported to DXF.
23. Bug-fix: Objects imported from DXF may be added to the existing groups.
24. Now round holes are exported to STEP as a circle.

25. Bug-fix: When STEP export to an existing file is canceled, the file size becomes 0 bytes.
26. Bug-fix: When importing Gerber with the option "Pads in Signal Layers" checked for the Bottom Layer, the polygons were mirrored.
27. Bug-fix: Polygons may be imported incorrectly from Gerber.
28. Bug-fix: Gerber export may misplace Origin if a board is panelized with Edge Rails.
29. Bug-fix: Paste Mask Shrink that exceeds the pad size causes incorrect shapes in Gerber export.
30. Bug-fix: When exporting Gerber together with NC Drill into one archive, if Mirror settings of Gerber layers were different, the Mirror settings in NC Drill used the last selected layer (changed to signal layers).
31. Bug-fix: In ODB++ Export, Blind/Buried vias were missing pads in the internal layers.
32. Bug-fix: Layer order for Blind/Buried Vias were exported to IPC-D-356A incorrectly.
33. Bug-fix: Layer order for Blind Vias were incorrect when importing from KiCad.
34. Cutout inside a component is now imported from PCAD ASCII as a Route Keepout.
35. Bug-fix: Arcs were exported to PCAD ASCII incorrectly.

## **Schematics**

1. Bug-fix: Back Annotate did not work after adding a library to the Project Libraries group.
2. Bug-fix: Component alignment may work incorrectly.
3. Bug-fix: The program may crash when hovering over a Design Cache filtered list.
4. Bug-fix: 'Undo' does not cancel the Font Color change for Markings.
5. Bug-fix: Symbol and Pattern preview fields were not scaled immediately when changing the width of the library panel for the first time.
6. Bug-fix: When opening a project, the Grid=OFF status was not restored.
7. Bug-fix: In the Component Properties dialog, the order of parameters in the Parameters and Markings tabs were different.
8. Bug-fix: Network names could be cut off at the edge of the image when printing.
9. Removal of a pin from the net, when the entire net could be deleted, has been improved.
10. Assign SPICE model item has been removed for Net Ports.
11. Bug-fix: Some components from libraries may not be imported when importing a KiCad project.
12. Bug-fix: The program might crash when importing EDIF.
13. Bug-fix: When imported from Eagle format, the schematic was not centered and may not be printed.
14. Bug-fix: Importing an Eagle schematic may result in an error.

## **Component Editor**

4. Bug-fix: Keystrokes on the keyboard may hide the cursor.

## **Pattern Editor**

1. Bug-fix: Keystrokes on the keyboard may hide the cursor.

## **List of minor changes and fixed bugs in DipTrace 4.2.0.1 if compared to 4.2.0.0**

### **PCB Layout**

1. An option of importing DXF in mils has been added.
2. Radial Placement has been duplicated in the Placement menu.
3. Selection of graphics mode for 3D has been added (Tools/ 3D Preview/ Graphics Mode).
4. In the Placement by List mode, a pop-up message appears, if a component is locked.
5. Bug-fix: The border of the group of components containing a shape in the Courtyard layer is formed incorrectly.
6. Bug-fix: Invalid characters in material names cause ODB++ export to fail.
7. Bug-fix: When exporting WRML, the text in the Bottom layer is mirrored.
8. Bug-fix: Scrolling with the mouse wheel changes the model type in the 3D Model window if the model type list is focused.
9. In the 3D board preview, zooming with the mouse wheel has become smoother.
10. In the 3D board preview, the speed of rotation to the projection selected with the buttons has been increased.
11. Bug-fix: In the component model preview, the specified board thickness is not considered in the dimensions of the pads.
12. The display of surfaces of imported STEP and IGES models has been improved.
13. Better color processing in 3DS import.
14. Bug-fix: When working in the interactive BOM and Check Net Connectivity dialogues, hotkeys for switching sides fail.
15. Bug-fix: Inverted text is displayed incorrectly in Windows GDI mode.
16. Bug-fix: The Copper Pour contours shape incorrectly in the Route Keepout layer.

## **List of fixed bugs in DipTrace 4.2.0.0 if compared to 4.1.3.1**

### **General**

1. Minimized modal windows block the program until the window is restored or closed by Esc key.

### **PCB Layout**

1. In DipTrace 64-bit on Windows 10, 3D preview may get stuck when loading models.
2. The Layer Setup window may not close in rare cases, if real-time DRC is on.

3. When exporting Gerber + NC Drill, a wrong Offset may be generated.
4. Space in the project file name results in different file names in Gerber archive when using <project> in filename mask.
5. Copper Pour may be temporarily filled with the wrong line width, if changed from the object properties panel.
6. Default layer color setting for printing is not applied to copper pour.
7. A hole on the board border causes incorrect display in VRML export.
8. VRML models may not be included in STEP export.
9. When pressing the arrow key in the Connectivity Check Result window, selected objects are moved in the design area.
10. When exporting to Eagle, the pads may be rotated 90 degrees.
11. Opening IGES models may cause the program to crash.

### **Schematics**

1. When importing Eagle files, a differential pair netclass may be imported incorrectly.
2. Renaming one Bus Connector of the group may break the connections. The check and option to rename the group at once have been added.

### **Pattern Editor**

1. IPC-7351 pattern generator: in the TO-220 Horizontal family, a 3D model is generated without legs if Pitch\_d value is negative.
2. IPC-7351 pattern generator: in the Header Vertical family, the value in the Pitch\_d field is cleared when editing the Pin Count D value.

## **List of fixed bugs in DipTrace 4.1.3.1 if compared to 4.1.3.0**

### **PCB Layout**

1. If IPC-7351 models are used, STEP export does not work.
2. When entering text into table cells "Shift + U" works as a Hotkey.
3. When importing KiCad 5.1 files, SMD pads on the Bottom layer may be imported incorrectly.

### **Schematics**

1. When entering text into table cells "Shift + U" works as a Hotkey.
2. Back Annotate does not update supplier information.

## **List of fixed bugs in DipTrace 4.1.3.0 if compared to 4.1.2.0**

### **General**

1. Snap EDA: multiple clicks for page switching result in a long "freezing".

## **PCB Layout**

1. The panelization parameters are reset when opening a schematic file in the PCB Layout after starting.
2. D-shape pads are displayed incorrectly in the Mirror mode.
3. The Highlight Net for the Obround and D-shape pads is displayed incorrectly in the Mirror mode.
4. TrueType text is shifted in BOM HTML.
5. DXF with multilayer blocks is imported incorrectly
6. Export of arcs with a very small angle and extra-large radius to Gerber has been changed.
7. Chamfer of the corners of a rectangular board outline may result in incorrect formation of outlines in Gerber when panelizing with Edge Rails.
8. After changing any of the markings parameters for a group of selected objects, the marking for selected Mounting Hole and Fiducial may appear.
9. Dimension objects are not updated automatically after changing the shape dimensions in the shape properties.
10. Ratlines are displayed incorrectly in print preview in the Mirror mode.
11. In print preview, the Flip Text checkbox does not work for all layers if Flip Text Automatically option is off.
12. Courtyard and CompOutline layers are not displayed in print preview in the Negative mode.
13. Assembly Variants settings are reset when updating from schematic.
14. Empty additional fields are duplicated when updating a component from a library.
15. Ratline optimization algorithm has been improved.
16. The algorithm for creating outlines when panelizing with V-Scoring has been improved.
17. Three-point arc on a straight line in the Board Cutout layer causes a failure to create the outlines of the boards on the panel.
18. Size preview is not displayed when changing a shape in the Board Cutout layer.
19. TrueType texts significantly slow down the software at a large scale.

## **Schematics**

1. When importing from Eagle, nets connected to NetPorts may merge.
2. The net line may be redrawn incorrectly after removing a component.
3. Manufacturer and Datasheet fields are reset after Back Annotation.
4. The display of the net name is not updated immediately after changing.
5. The change of the "Number in Bus" parameter is not recorded in Undo and is not saved to file.
6. Empty "Name" fields are duplicated when updating a component from a library.
7. When updating the selected component from a library, other unselected components may be also updated.

## **Pattern Editor**



1. Shape rotation angle in the Pin Terminal does not work when entered manually.
2. The units of measurement are not changed with a mouse click in the View/ Line Width window.
3. When redrawn, a 3D model may be displayed multiple times.

## **List of fixed bugs in DipTrace 4.1.2.0 if compared to 4.1.1.0**

### **PCB Layout**

1. Pick and Place table is not auto-updated when deleting/ creating a board outline with panelization.
2. "Do Not Panelize" is not taken into account in the Pick and Place list.
3. TrueType markings may not be displayed in the Mirror mode.
4. The duplicate component filter may be launched incorrectly.
5. When exporting 3D to VRML, circles are displayed incorrectly.
6. When exporting 3D to STEP, some components may miss pins.
7. In the 3D preview, "Keep All Model Pins" is applied to all patterns with the same name.
8. With a large number of Additional Fields, the component properties window may extend beyond the screen limits.
9. For the outer Plain layer, the pad in Gerber Assembly displays signal layer shape.
10. BOM HTML export does not take into account the selected Assembly Variant
11. When importing PADS files, shape description may cause the import failure.

### **Schematics**

1. When the current page only is printed, the document has no name.
2. The duplicate component filter may be launched incorrectly.
3. With a large number of Additional Fields, the component properties window may extend beyond the screen limits.
4. When importing Eagle, pin information may be placed into the Additional Fields of the components.
5. RefDes from an unconnected hierarchical block may be present in the BOM.

### **Pattern Editor**

1. Some Polygonal Pads from Altium are imported incorrectly.
2. After resizing the Library panel, the design area is not redrawn correctly immediately.
3. Pattern Generator: 3D models of DFN 3 Pins may have no pins.
4. Pattern Generator: 3D model of TO-220 Vertical may have displaced pins.

## **List of changes/fixes in DipTrace 4.1.1.0 if compared to 4.1.0.1**

### **General**

1. SnapEDA engine in all modules
2. Fixed: KiCad import may fail if text contains service characters.
3. Restoring of "Xcursor" mode while reopening the software modules.

4. When changing "Width" or "Height" of a shape, new values are now applied to all the selected shapes.

### **PCB Layout**

1. Fixed: Board Cutout within a component is not included into Gerber export.
2. Fixed: The column of objects on the Objects toolbar is not displayed when scaling the screen.
3. Fixed: Copper Pour on the bottom layer may overlap the display of traces on the inner layers.
4. Fixed: Copper Pour does not apply a far smaller custom Net Clearance parameter to dynamic Vias.
5. Fixed: When panelization of Tab-Routing type is applied, Copper Pour does not take into account asymmetrical Mouse Bites on the board copies.
6. Fixed: Real-time DRC may freeze when editing and optimizing sharp corners of traces.
7. Changed: In projects created with 3.x or earlier, the internal connections of components are considered as optional for routing. Now all pads must be connected in the same way as in 3.x (can be bulk changed manually).

### **Schematics**

1. Fixed: "Flip Symbol -> Vertical" command flips horizontal text by 180 degrees.

### **Component Editor**

1. Fixed: In Library Setup, a new library is displayed separately after saving to a folder connected to the group.

### **Pattern Editor**

1. Fixed: In Library Setup, a new library is displayed separately after saving to a folder connected to the group.
2. In Pattern Generator, the search in the Manufacturer List is now launched upon entering the first character from the keyboard.

## **List of changes/fixes in DipTrace 4.1.0.1 if compared to 4.1.0.0**

1. Raster pictures from older versions are opened correctly in both Schematic and PCB (vectorizing is an option now).
2. 4.1.0.1 files with raster pictures are now opened in 4.0 without crash. Pictures are not visible as they are completely redesigned in 4.1, but other objects are ok.

3. Cross-hair cursor option has been redesigned, it is also saved when you reopen the program.
4. Converting component markings from 3.3 and earlier versions has been polished (lack of some markings was possible).

## List of fixed bugs in DipTrace 4.1 if compared to 4.0

### General

1. In the preview area, the "&" symbol is not displayed in the names of patterns and components.
2. Locked components are not taken into account during alignment.
3. When opening the project and library files, if the file name is entered manually, the default extension is not applied.
4. The programs do not start, if the license registry is damaged. Recover option has been added.
5. Pattern search by the Manufacturer field does not work.
6. Text copying / pasting for non-English symbols does not work properly in Text Edit/Properties.

### PCB

1. When updating from schematic, the Manufacturer field of the components is overwritten/ deleted.
2. If new components are added, positions of text objects, added in Pattern Editor and moved/rotated in PCB Layout, may be changed unexpectedly.
3. Components containing only Mounting Hole are duplicated after updating from schematic by components.
4. Ratline may be kept visible, if pads are connected by several copper pours one by one, and one of connections is made by nested, but not crossed, copper pour with high priority and outer copper pour with low priority (this is possible if outer copper pour area is excluded by other net copper pour with medium priority).
5. When placing components via Placement by List feature, pressing the space bar rotates the component by 180 degrees.
6. Pattern assignment may work incorrectly when importing a third-party software netlist.
7. The behaviour of 45 deg segments when editing tracks has been re-designed.
8. Empty lines are displayed in the Length Matching window when scaling with high resolution.
9. Cutout layer display is overlapped by objects in other layers.
10. All Layer Keepout within a component is not displayed on some layers.
11. Under some conditions, the marking of Static Vias appears.
12. When a component is ungrouped together with static vias it results in via conversion into a pad.

13. DRC does not take into account pattern rotation angle and reports false "Silk - Pad" errors.
14. When a Fiducial is placed on the Bottom layer, the mask settings are generated by default for the Top layer.
15. Design Cache is not updated after Replace Component.
16. Updating/ replacing a component with the one that has a different orientation may cause removal of pins of IPC 3D model.
17. In the context menu on the Layers panel, non-working items are displayed and the layer name can be colored like the layer.
18. When moving objects in Measure mode, their anchor points remain in their original positions.
19. In the Mirror mode, when editing markings, the texts are displayed in a non-mirrored location.
20. Shapes in the Mask layer within a component are displayed on top of other objects on the opposite side.
21. If multiple mounting holes are selected at the same time, entering values in the Properties panel works incorrectly.
22. When a new copper layer is inserted, teardrop layers are not changed together with the trace.
23. When a trace is edited, a hint appears, covering the view.
24. Copper Pour fill around an image in Route Keepout works incorrectly.
25. Copper Pour contours filled and non-filled obounds differently.
26. Copper Pour fill around rotated inverted text ignores its rotation and alignment.
27. WinGDI, vector pictures - fixed: Vector picture polygons are drawn with the line width defined for the respective layer/object in WinGDI mode and print preview (changed to 1px in all cases).
28. In Windows GDI mode, double click does not open Component and Via properties. Incorrect properties dialog is opened for Fiducial.
29. In Windows GDI mode, the visible size of some pads on the opposite side is incorrect at View / Pad Numbers = Show.
30. Polyline made of two points within a component is displayed incorrectly in 3D Preview.
31. When launching 3D Preview of some projects, a lot of "Unable to generate 3D model by entered dimensions" messages appear, but 3D preview for each package works correctly.
32. 3D Models created by Component Outline are sometimes generated with edges of different colors.
33. Components with offset origin on the Bottom Layer are displayed incorrectly in 3D Preview and STEP Export.
34. BOM report includes components of non-placed hierarchical blocks.
35. Large non-active pad hole height parameters for round holes cause filling of pad area with copper pour.
36. In the Gerber export window, the Text and Pictures checkboxes are active for the Board Outline layer.
37. Gerber export preview does not display polygonal pads on the Bottom Layer in mirror mode.
38. When importing Gerber files, some description of arcs gives wrong results.
39. Pictures and tables added to DipTrace ASCII export / import.

40. P-CAD ASCII Export: polygonal pads in non-signal layers are exported incorrectly: ellipses are exported instead of ovals, mirrored text on the Bottom layer is accounted for incorrectly when calculating the size of the design area.
41. Eagle Import: if there is no description of net classes in the .brd file, the import is not done.
42. In DSN export, component rotation angle > 360 degrees is allowed.
43. In DXF export, filled shapes are transferred using a polyline of a specified thickness.
44. DXF import is not done if Turkish regional settings are used.

## Schematics

1. If there is a group of hidden networks, changing the color of an individual network is not possible.
2. Replace Parts returns the markings to the library position.
3. Multipart components created with Create Similar Parts are updated from the library incorrectly (first part is placed instead of all similar).
4. In Windows GDI mode, double clicking does not open the component properties.
5. Possible crash when moving a point of the bus line.
6. Visual T-junctions of bus lines break while editing.
7. Altium Import: obounds and netports with different alignment are imported incorrectly.
8. When moving objects in Measure mode, anchor points remain in their original position.
9. Changing RefDes for the entire multi-part component at once is impossible, if another multipart component with such RefDes already exists.
10. DipTrace ASCII: possible incorrect export of text objects for some patterns.
11. DipTrace ASCII: import / export of images has been added.

## Component Editor

1. The last symbol entered in the Additional Fields may be lost.
2. The mouse pointer does not exit Resize mode when quickly moved from the Libraries panel to the design area.
3. Search fields in the Search Components window are overlapped when scaling at high resolution.

## Pattern Editor

1. When centering the pattern, the offsets of the 3D model are not recalculated.
2. Search fields in the Search Components window are overlapped when scaling at high resolution.
3. Incorrect information is shown in Hint for Fiducial.
4. Keepout value for the mounting hole is allowed to be less than the hole size.
5. 3D Models created by Component Outline are sometimes generated with edges of different colors
6. Pattern Generator does not create a hole for the Mounting Hole family if Type = Copper Ring and Copper inside Hole = Non-Plated.

7. Upon executing the Rotate Group for all objects, under certain conditions, some pins are removed from the model created by the Generator.
8. In the Pattern Generator, the 'b' dimension has been corrected in the picture "Component\_THT\_MountingHole.png".
9. The selected object could be moved in the non-displayed layer.