

VisualPlace version 4.1

VisualPlace is a program that helps you manage and create component placement files for manual or automated assembly of printed circuit boards. VisualPlace supports the centroid files from various EDA programs. If you are new to VisualPlace, please see the visual tour: from the menu, select 'Help' and then 'Getting started'.

Version 3.0 and higher of VisualPlace use plug-in interface version 3.0. **All plug-ins that were developed for earlier versions of VisualPlace must be adapted to plug-in API version 3.0.** Input plug-ins for centroid files must be adapted so that they collect only components for the top side of a PCB when called for the top side, and only for the bottom side when called for the bottom side. Version 2 plug-ins would load components for both sides in one run. However, the placement *origin* may be different for both sides of a PCB (especially if the project lacks a definition of the PCB profile), and therefore VisualPlace now explicitly calls each plug-in twice, with the origins for top and bottom sides explicitly passed in.

Release notes for version 4.1 build 7662 (current version)

- Support for running under Linux with Wine was improved. The application now shows standard Linux paths and uses the standard GTK+ file open & save dialogs when running under Wine (instead of the virtualized 'C:' and 'Z:' drives).
- Extensions for Gerber files are now also recognized for Eagle, OrCad and Target3001 (in addition to Altium). Note, though, that the current Gerber standard recommends to use X2 file attributes, and the generic file extension '.gbr'.
- When creating a new project, there is now an option to copy the detected production files to the selected project directory. This is especially useful when you set a fixed project directory in the 'Application Settings'.
- There is now an option in the 'Application Settings' for the display resolution, which is useful if the driver or monitor does not support EDID (or reports the wrong value).
- The KiCad plug-in has been updated to support version 10.
- Sorting the component list on package type caused a crash. This issue has been corrected.

Release notes for version 4.0

The release notes for all '4.0' builds have been combined in a single list. Notable features of version 4.0 are a 'butterfly view' showing both top and bottom sides of the PCB at once, netlist support and export to 'BoardView' files.

- Both top and bottom sides of a PCB can now visualized together. A combined top/bottom view is especially useful for checking net connections (on PCBs with components on both sides). The user can select to view the top and bottom side-by-side (horizontally) or vertically stacked.
- VisualPlace now loads netlist data from Gerber X2 attributes, if available. As an alternative, an IPC-D-356 netlist file can also be configured in the project. (If both Gerber X2 netlist attributes and an IPC-D-356 file are available, the Gerber X2 attributes take precedence.)
- A netlist dialog is added (from the menu: Project / Netlist...). This dialog has a table with the names of the nets, and the component pins on each net. Selecting a net puts a marker on each pin in the board view. Double-clicking on a pin in the board view also selects the net. Netlist data must be available in the project for this function. Nets can be renamed in VisualPlace.

- VisualPlace can export 'BoardView' files. These contain essentially netlist data. BoardView files are used by repair facilities. Common BoardView viewers are OpenBoardView and FlexBV.
- The Gerber parser now flags minor flaws and elements and methods that the standard marks as 'bad practices'. VisualPlace catches only 'bad practices' that may hinder analysis or visualization of the Gerber data; many commands that the current standard marks as 'deprecated' are silently handled. There is a new dialog to allow you to review the warnings and notices that the Gerber parser has found in the data files.
- Block selection is now supported in most tables (notably the Bill-of-Materials). This enables you to copy a block of data onto the clipboard (and paste it in a spreadsheet), or to paste data from a spreadsheet into the table.
- Git support is added (in addition to Subversion and RCS).
- 'Step & repeat' has been added to the 'Generic CSV Export' plug-in, to support panels (for CPL output).
- The templates for creating footprints now use TCL for scripting (instead of the mix of INI file and RPN expressions). This makes it easier to create more elaborate templates. The TCL parser is built into VisualPlace, so there is nothing that needs to be installed separately.
- The component list can now be sorted on all columns, including the user fields.
- Better support for high-resolution displays. The markers for component positions and fiducials are now dynamically resized, depending on zoom factor and resolution.
- When aligning the origin, and when moving a component position, you can now use the arrow keys on the keyboard to move the mouse cursor. This may make precise positioning of the cursor easier. The new position can be confirmed with the Enter key (or a mouse click).
- The colours of the solder mask and the silkscreen printing are now read from the Gerber Job file (if present). In addition, these colours (whether read from the Job file or manually set) are now stored in the project (instead of a user setting).
- In the dialog for the board and marker visualization, there is now a button to reset PCB colouring to system defaults.
- Boards with complex contours ("profiles") would only be partially colour-filled. This issue has been corrected.
- Markers for components that were placed at angles that weren't a multiple of 45 degrees, came out warped. This issue has been corrected.
- The Gerber parser could reject files that had negative absolute coordinates. This issue has been corrected.
- The colour set on a custom package could be ignored (depending on the prefix of the component designator). This issue has been corrected.
- When editing a footprint, the preview in the edit dialog was mirrored vertically. (In the list of known footprints, the preview was correct). This issue has been corrected. The same dialog now also allows you to set the terminal style (lug-lead, gull-wing, etc.) to improve the visualization of user packages.
- Walking through the component list while setting checkmarks (or 'tick marks'), function key F7, did not work correctly with the 'auto expand' option. In addition, the setting was not saved (which required you to re-enable the option on every launch of VisualPlace). These issues have been corrected.
- When editing the Bill-of-Materials (or the component placement table), the dialog now immediately shows a notification when the file is read-only. This can occur, for instance, when the file is under RCS control (or Subversion with explicit locking enabled).

Additionally, the dialog immediately checks whether the file is up-to-date relative to the remote repository.

- The 'Application Settings' has a new option to clear cached files and thumbnails.
- The function for creating an archive of a project has been improved on the aspect of removing fabrication files from the file list. References to these fabrication files are also removed from the project file (in the archive), so that the archive will load without warnings. This allows to create archive files for distribution to parties who do not need the full data.
- The F11 function key toggles the view from the design files to the camera. However, hitting this function key by accident, whilst there is no camera attached, would force you to wait a significant timeout, before Microsoft Windows reports that the connection failed. This version adds a 'ping' to test the presence of the camera, with a 1 second timeout (which should be sufficient, because the camera is supposed to be in a LAN, or even directly connected to the workstation).
- Project-local user settings (such as the zoom level) have been spilt off from the project configuration file. The user settings are in a file with the project name, and the extension '.ini', whereas the project configuration is in a file with the extension '.vpp'. The rationale for the split is that in the context of version control, the project file is shared.
- The KiCad support was updated for the 9.0 release. In addition, an improved heuristic is used to detect whether the position file uses mm or inch as the unit (this relates only to the CSV format).
- The dialogs for selecting a footprint lay-out and for editing a footprint have a resizable border, but would in fact not adjust the position and size of their controls. Furthermore, if inches were set as the unit, the footprint editing dialog would save the body size in 'mil', but reload them in 'inch'. These issues have been corrected.
- For Target 3001 support, when a slash appeared in either the value or the package name, either (or both) might be left empty. This issue has been corrected.
- Deleting a thumbnail file from the 'cache' folder caused VisualPlace to associate the wrong project to remaining thumbnails (clearing the cache fixed this). This issue has been corrected.
- In the 'Project Settings' dialog, the 'browse' button for the copper layer (to select the Gerber file) should open a browse dialog, but did nothing. This issue has been corrected.
- The result of the raster-conversion of the Gerber files was no longer stored in cache, regardless of the setting in the 'Application settings'. This issue has been corrected.
- VisualPlace was incompatible with the latest release of the SvnRev utility. VisualPlace was also incompatible with TortoiseMerge (the file compare utility that comes with TortoiseSVN). These issues have been fixed.

Release notes for version 3.3

The release notes for all '3.3' builds have been combined in a single list. Notable features of version 3.3 are improved support for tablets and high-resolution displays, the addition of component totals and costs for assembly in the bill-of-materials editor, and limited dark mode (plus a new opening screen showing the recent projects).

- Daniele Pezzin translated of the user interface in Italian, and donated it to the project.
- Improved support for tablets, notably automatic portrait/landscape flipping. Plus improved support for high-resolution displays and small-screen devices (such as tablets).
- Limited dark-mode is implemented, in addition to user-configurable colours. The dark-mode support is restricted to the main application frame. All dialogs and message boxes still use the standard Windows interface colours.

- There is now an option for automatically sizing columns for the table in the bill-of-materials dialog.
- Two 'user field' functions are added to the bill-of-materials: Total Quantity and Total Price. When a user field is defined with these functions, the fields in it are automatically updated to contain the quantity of components needed for the number of assemblies, and the total costs (per row). For the Total Price function, a user field for the Unit Price of a component must also be defined.

In addition, the bill-of-materials table has a footer row, with the totals of the columns Quantity, Total Quantity and Total Price. These fields also depend of the selected assembly stage, so that these fields give you direct insight of how many components are mounted in a particular stage (per PCB and in total), as well as the total component costs.

- In the Bill-of-Materials dialog, you can now set the number of PCB assemblies to make. When this number is set, the stock is checked against the required number of components, and the field is coloured red if stock is insufficient.
- Files for component positions (CPL files) and for the Bill-of-Materials can now also be in Microsoft's OpenXML format (the '*.xlsx' files), and in the OpenDocument 'spreadsheet' format (used by LibreOffice and OpenOffice).
- The menu items for 'Tick marks' and 'User fields' have moved from the 'Tools' menu to the 'Project' menu.
- The dialog for the 'Placement order & flags' now allows you to explicitly set the placement order of an individual component. More specifically, it allows you mark a component to be mounted *after* another specific component. Note that this function is mostly useful for manual assembly; pick-&-place machines typically impose their own order of assembly (taking feeder pick-up into consideration).
- When visualizing package shapes, a cross is now drawn on 'do-not-mount' components. This helps showing both the component positioning & shapes, and the mount status.
- When archiving a project, you can now add additional files to the archive (archiving a project stores all production files in a ZIP file).
- Improved checking of fiducials; a check was added that only a single fiducial is marked as 'origin'. In previous releases, the *origin* fiducial was called *primary fiducial*. This was renamed in order to clarify that the purpose of this 'primary' fiducial is to serve as the origin for component positions.
- An option was added to the 'Application settings' dialog, for selecting the range of rotations: you can choose between 0...360 degrees and -180...+180 degrees. Positive rotations are still left-turning (counter-clockwise).
- The algorithm used to zoom in and out on the PCB view has been improved, resulting in both improved quality (less blurring) and slightly improved performance.
- When the option to automatically reload the most recent project is turned off (see Tools / Application settings... / Miscellaneous), VisualPlace will now show thumbnail images of the recent projects, and allow you to click on one of these to open it. In addition, VisualPlace now keeps 16 recent projects, instead of 8.
- The Excellon drill file loader now includes support for the 'arc rout' command.
- Plug-ins can now set flags for 'no position' and 'no rotation' on components, if position data in a CPL is incomplete. VisualPlace takes these flags into account.
- Allow for partial match when searching for components in the component list.
- The path to the package definitions file is now configurable. This is useful when you maintain the file with package definitions with an external application.
- The automatic reloading of the most recent project on start-up of VisualPlace, can now be turned off. See Tools / Application settings... / Miscellaneous. The default is still that

VisualPlace reloads the last project on a next launch.

- Various improvements were made to detection of the 'profile' of the PCB (i.e. the board outline). For instance, the Gerber parser now uses the 'AperFunction' attribute on apertures (Gerber X2 format) to check for the 'profile' attribute.
- Serial-mode bar-code readers are now also supported. This is in addition to bar-code scanners in keyboard-wedge mode. Most USB-connected bar-code readers support both serial and keyboard-wedge modes, but applications on occasion only support either protocol. By making VisualPlace supporting both protocols, it may avoid the need to reconfigure the reader when switching between applications.
- Support for the *Revision Control System* (RCS) was added. It offers a solution for version control that is local on a workstation, and that is easy to use and trivial to set up. Apache Subversion is still supported as well.
- Editing the value, package or product number of a component can now also be done by right-clicking on the component (either in the component list, or on the marker).
- When the 'Control' key is held down (for measuring), the mouse cursor now also snaps to the PCB edges. As is the case with snapping to pads, the snapping is off when the Shift key is pressed in combination with the Control key.
- Extra tests and messages were added to differentiate between a CPL file with an unknown format and a CPL file that is empty. Both result in zero placement information being read, but it turns out that it is a common failure to generate a CPL file that is valid, though completely empty.
- There were improvements in the OCR engine (optical character recognition), especially in the area of rejecting markings and small graphics that are not letters or numbers.
- The plug-in for KiCad now also supports BoM files generated by the schematic editor (earlier releases required the BoM to be generated from the PCB editor).
- The plug-in for generic CSV files (for the BoM and CPL files) now also supports 'Unicode' text files with 16-bit characters (in Little Endian format). Both files with and without signature (or 'byte order mark') are supported. Additionally, files with a UTF-8 signature are also detected.
- The TWS Quadra-DVC plug-in now allows you to rotate the exported placement information relative to the PCB design. This is useful for wide PCBs, as the build area of the Quadra is higher than wide. Panel definition for the TWS Quadra was also improved.
- A plug-in for SPRINT was added.

Release notes for version 3.2

The release notes for all '3.2' builds have been combined in a single list.

- VisualPlace switched to a new format for the data on packages and footprints. The new data file is in JSON format. The new format allows for more detailed data on packages. A separate (open-source) tool has been developed to maintain the packages data, see https://www.compuphase.com/electronics/packages_en.htm
- Using the more detailed package data (see previous bullet point), the analysis of component positions and the visualization of the packages has been improved.
- The visualization of the packages (and footprints) is now also based on the new package data format, but in addition, VisualPlace now supports SVG images for package shapes. This allows you to create more accurate shapes for non-standard packages, such as connectors.
- Under menu 'Tools' / 'Application settings', there is a new option to select the preferred naming convention. Options are 'EIA imperial', 'EIA metric', 'IPC-7351' and 'ED-7303'.

- The plug-in for KiCad has been updated to support version 6.
- The ULP for EAGLE now asks you for the output directory, courtesy of Jeroen Lodder.

Release notes for version 3.1

The release notes for all '3.1' builds have been combined in a single list. Notable new features are the addition of a button bar for manual assembly, more accurately establishing the display resolution in DPI and thus a more accurate visualization, and support for label printers that support the ZPL format (notably Zebra printers).

- Juan de la Torre translated of the user interface and help texts (tooltip texts) in Spanish, and donated it to the project.
- Improved heuristics for detecting pads from Gerbers that 'paint' pads instead of 'flashing' them. (The Gerber standard lists painted pads as a bad practice, but many EDA programs do it anyway.)
- The bill-of-materials editor now annotates the assembly stage that a component may be assigned to. In addition, you can now select an assembly stage, after which the values in the 'Quantity' are updated to reflect this.
- When measuring distances, the cursor now displays the distance and the angle, in addition to the ΔX and ΔY values.
- A button bar with a few navigation buttons can be enabled (menu View / Button bar). This bar is intended for using VisualPlace on a tabled (Microsoft Surface Pro) or a panel PC without a keyboard.
- When the Gerber files for a project have X2 pad attributes, the mouse cursor snaps to pad centres in measurement mode. This allows for precise pad-to-pad distance measuring (for example, for pitch or span verification). The mouse cursor switches to measurement mode when keeping the 'Control' key pressed.
- Pad centres are also extracted from Gerber data even if no X2 pad attributes are present in the Gerber files, but no pad information is available in this case.
- Improved fiducial detection.
- Various improvements in OCR (for scanning labels).
- Improved zoom-in / zoom-out, in part because VisualPlace more accurately calculates the monitor DPI using values queried from EDID data.
- Gerber 'job' files are now supported. When creating a new project and a 'job' file is present in the selected directory, VisualPlace gets information on the layers from this file.
- When a component is listed in the CPL, but is absent from the BoM, VisualPlace makes the assumption that the component must not be mounted. When you override this (by removing the 'do not mount' status from the component), VisualPlace would reinstate the 'do not mount' status on the next load. This issue has been fixed.
- Editing fiducials can now be started with a right-click on a fiducial.
- The names of values and packages that VisualPlace recognizes as 'fiducial definitions' can now be modified and extended, see the 'Tools / Fiducial match patterns...' dialog. The patterns are regular expressions.
- Through-hole components can be marked as such (see the Placement List... dialog). When a CPL file has a field for SMD/through-hole type, this is extracted too. In the default sorting, through-hole components appear below SMD components for top and bottom.
- Improved support for barcode scanners: scanning a barcode label jumps to the relevant components if the barcode matches the product number, the value or any of the 'user value' fields in the component list.

- Support for the TWS Quadra DVC pick-&-place machine via a new plug-in. The plug-in for the TWS Quadra Basic pick-&-place machine now gives estimates of the offsets between boards in a panel (under the assumption that the boards are arranged in a regular grid in the panel).
- Support for Zebra Label printers, or other printers supporting the ZPL printer language, was added. Three interfaces are supported: Windows printer (for USB and parallel port), serial (for RS232 and RS485) and network.

Release notes for version 3.0

The release notes for all '3.0' builds have been combined in a single list. Notable new features are Unicode support, support for Gerber X2 and Gerber drill files, and an interface to an inventory system.

- Under the hood, VisualPlace is now converted to a 'Unicode' application. There is no user-visible change for regions that use codepage 1252 (Windows 'Latin 1'), but users in different regions will find VisualPlace more compatible with their installation of Microsoft Windows.
- A new Gerber parser replaces the gerber2dib program (gerber2dib is a derivative of the gerb2tiff program by Adam Seychell). The primary motivation for switching to a new Gerber parser is to support the new Gerber X2 standard. Other features of the new parser are improved detection of the PCB profile (outline) and support for cut-outs.
- Support for Gerber drill files has been added. Note that the drill files must include the 'file function' attribute; otherwise it will not be recognized as a Gerber drill file. Two drill files can now be added to a project (e.g. plated plus non-plated).
- The Excellon drill file parser has been updated with support for slots and step-&-repeat for holes.
- The Bill-of-Materials report now collects components of different assembly stages on separate rows. The name of the assembly stage is indicated as well. Colour marking (conditional rules) now also works on those rows.
- A bill-of-materials can now be updated from an inventory database or a stock management system. In the bill-of-materials editor, see the button 'Import'.
- The number of 'user fields' in the bill-of-materials is increased from 4 to 6.
- A filter was added above the component list (in the main frame). This lets you filter on value, package or any of the user fields.
- The colour for the pads on a PCB can now also be configured. Note that pads are only visualized if you add a solder mask to the project (see project settings). The background colour of the main viewport is now also configurable. The default colour is off-white.
- The optional component overlays are now drawn to represent the component shapes, rather than the footprint shapes.
- You can now edit the title of the reports. The default title is the project name (which is the same as what it was before, but you can now overrule it).
- On all reports, an option is added to save the report in a user-selectable path (in previous releases, the report was opened in a PDF viewer as a temporary file, and you needed to save the report from the PDF viewer).
- The footprint editor now allows you to set the size of a pad, in addition of its position. VisualPlace does not use the size of the pad, but it allows to visualize the custom packages on a PCB.
- VisualPlace comes with a database of common footprint lay-outs, and it can overlay these lay-outs on the PCB image. This enables you to visually check whether the package that is specified in the bill-of-materials fits on the pads of the PCB.

- VisualPlace now supports component rotations in steps of 45° (previous versions supported steps of 90°).
- The Gerber placement data is now supported (also called Gerber X3).
- A plug-in for Pulsonix EDA software was added. The plug-in for KiCad was adjusted to support version 5 of the EDA suite. The plug-in for Target3001! was adjusted too.
- The CPL files created by CircuitStudio are now also supported.
- Support for bill-of-material files created by Ultiboard were added.
- A 'Search and replace' function has added to the bill-of-materials editor and the centroid table editor.
- The 'search' function in the tables can now be restricted to a single selected column. The search function now also keeps a history of recently searched keywords.

Release notes for VisualPlace version 2.1

The release notes for all '2.1' builds have been combined in a single list.

- An assembly stage may now be set as 'cumulative' to another stage. When a cumulative stage is selected as the active stage, all parts in that stage are active, *in addition to* all parts of the other stage.
- When using assembly stages, all components of inactive stages are flagged as 'do not mount' when exporting a centroid file into a machine-specific data format. In case that the active stage is 'cumulative', the components in that stage plus its base stage are included and components in other stages are marked as 'do not mount'.
- The package specifications dialog is now sorted by default on both type and name. Unknown packages are at the top of the list; these are the ones that require your attention.
- The dialog to browse for the defined footprints now uses filtering on keywords that the user types in. In addition, you can now filter on the pin count and/or the pin pitch (by typing them in).
- The footprint lay-out editor is improved. The pin/pad table is automatically adjusted when changing parameter fields (there is no more need to click on 'generate'). There is also a visual preview of the generated lay-out.
- Internet access functions (e.g. for updating a bill-of-materials from an ordering and inventory system) are now based on the WinInet component, which allows to transparently support servers on the HTTPS protocol (as well as HTTP and anonymous FTP).
- For translating package names from EDA-specific libraries to the conventional names, VisualPlace now also attempts generic translations based on parametrized regular expressions (in case a direct match is unavailable).
- In the editor for the bill of materials, there is a new option to copy all fields of a row to other designators in the table. For example, after selecting a row with designator C12, you can ask VisualPlace to copy its value, package, manufacturer product number and all user fields to a list of other designators, such as 'C56 C14 C28'.
- You can now print the bill-of-materials directly from the BoM editor dialog, and you can print the centroid report directly from the centroid table editor.
- In the colour coding rules for the bill-of-materials, you can now add rules for selected stages. For example, rows in the bill-of-materials for components that need to be 'hand-placed', can be given a different background colour, so that they stand out.
- In the 'PCB Layout' and 'Assembly Flipbook' reports, the PCB image can now be an overlay of the silk-screen and the solder mask Gerbers. This makes it easier to locate

components that do not have an outline on the silk-screen.

- When using checkmarks (to mark components that have been handled), the checkmarks are now also displayed on the markers (in addition to on the left of the rows in the component list).
- When marking components as 'do not mount', this status is now also shown in the component list, at the left of each row (in addition to the specific markers that components with this status have).
- When creating a new project and selecting a first Gerber file, VisualPlace searches that directory for more suitable files and suggests to include these too. The dialog for this suggestion has been improved: it now allows you to select or deselect the files in the list of what VisualPlace found.
- VisualPlace now supports milled-out sections in the PCB profile (the Gerber file with the outline of the PCB). In addition, VisualPlace now uses a larger tolerance for its criterion of whether a contour is 'closed'.
- In rare cases, an EDA suite generates the Gerber files in a different rotation than the placement data. In earlier versions, your only recourse was to then rotate the Gerber files with a Gerber editor before loading them into VisualPlace. The current version allows you to specify a Gerber rotation in the project settings.
- VisualPlace now integrates basic support for the Subversion version control system. If production files or user data files (for package specifications or OCR training data) are not up to date or have local (uncommitted) changes, VisualPlace notifies the user. VisualPlace provides a dialog for update and commit operations. For other operations, including the operation of *adding* the files to version control, a general purpose Subversion client is needed. For previewing local modifications, VisualPlace launches an external file comparison utility (such as WinMerge or TortoiseMerge).
- The database with the OCR training data is in a new format. The new file format is 'friendly' to version control systems, as these systems recognize the new file as a text file. This results in the version control tools to merge files on a record-per-record basis (a record holds the definition for a single character lay-out).
- The OCR engine was also updated with heuristics to improve the accuracy of the text recognition. The recognition of footprint shapes was also improved with new heuristics. Some of the new heuristics for footprint scanning require a solder-mask Gerber (or solder-paste mask) to be set in the project.
- The 'open style' markers now have a larger arrow, to more clearly indicate the orientation of the components. The open style markers are especially useful with manual assembly with camera support.
- VisualPlace allows you to manually align centroid files to the Gerber image, for the case that the centroid files uses a different (0,0) origin than the Gerber files. However, VisualPlace assumed that the origin offset between centroid data and Gerber files would be the same for both sides. This made it impossible to correctly align the placement data on the bottom layer in such situations. This issue has been corrected.
- The option to add user fields as columns to the component list was no longer working. This issue has been corrected.
- The Neoden plug-in now includes support for the TM-245P pick-&-place machine.
- Centroid files generated by recent versions of Altium Designer are now supported.
- The file format of Allegro / OrCad is now supported, with a new plug-in.
- Recognition of fiducials was improved for Ultiboard.
- The plug-ins for the file formats of CadStar, GenCAD and Target 3001, failed to operate correctly with manual alignment. This issue has been corrected.

- The parser for Excellon drill files rejected records with only an X or Y coordinate, instead of keeping the missing value from the earlier record. This issue has been corrected.
- When creating a new project that overwrites an existing project, data from the old project might persist in the new project. This issue has been corrected.
- When re-sorting the component list, any checkmarks that had been set were hidden in the component list (though they were still present on the sprites). This issue has been corrected.
- When changing a projects configuration or editing the bill-of-materials, checkmarks that you set on components could get removed. This issue has been corrected.
- When using a camera that does not support PTZ, the PTZ configuration would still be stored in the configuration (but usually be invalid). This issue has been corrected.
- In the 'assembly flipbook' report, the direction of the arrows on the component markers were flipped vertically. This issue has been corrected.
- When exporting a bill of materials with the 'generic' plug-in, the manufacturer product number could not be exported. This issue has been corrected.
- Detection of a board outline would fail if the Gerber file contained overlapping collinear line segments (which you won't spot in a Gerber viewer). This issue was corrected (VisualPlace now detects and merges overlapping collinear line segments).
- Components that are present in the centroid file but absent in the BoM, would be marked as 'do not mount'. However, when editing the centroid data or the BoM, VisualPlace would synchronize the rows of the BoM and the centroid files and the 'do not mount' flag would then be lost. This issue has been corrected.
- Support for a network camera no longer functioned in release 5253, because of a code-generation error in the camera support DLL. This issue has been corrected.

Release notes for VisualPlace version 2.0

The release notes for all '2.0' builds have been combined in a single list.

- When creating the bill-of-materials report, you can choose to give rows that match some criteria a different background colour. In previous releases, the colours and the criteria were predefined and fixed. In the current release, you can define your own criteria.
- Previous releases predefined two of the user fields as 'Supplier' and 'Order nr.' The current release no longer does so. You can, of course, define the two user fields as 'Supplier' and 'Order nr.' explicitly.
- The user fields are now (also) stored per project. So different projects may use different user fields. The most recent set of user fields still becomes the default set for a new project.
- The markers for drill holes, fiducials and registration marks have been changed from red to blue, because red is also the default colour for component markers for the top side of the PCB.
- The browser for the 'standard footprints' (the library of footprints that comes with VisualPlace), now allows you to filter the list on one or more categories or keywords. This makes it easier to select an appropriate footprint from the large list.
- Version 2.0 of VisualPlace no longer worked under Wine (Linux), because recent versions of Microsoft Visual Studio rely on kernel features that Wine does apparently not (yet) implement. This has been fixed by building the parts of VisualPlace that require Microsoft's compiler with an older version.
- VisualPlace did not work under some versions of Microsoft Windows XP, because the installer failed to include a 'runtime' DLL (though many XP systems already had this DLL

because it had been installed via other applications). The current version of VisualPlace no longer depends on this runtime DLL.

- Manual alignment could fail, because VisualPlace did not take the differences in axis systems (between some EDA suites and VisualPlace) into account. VisualPlace now uses an iterative procedure to determine the correct origin after manual alignment.
- The internal Gerber parser has been improved to correctly determine the PCB outline for more complex outline shapes.
- On an import of a bill-of-materials into the one maintained by VisualPlace, the manufacturer product number was not copied from the imported bill-of-materials. This field is now being copied.
- The OCR engine has been slightly improved. More importantly, the original release of version 2.0 (which introduced OCR capability) did not include the standard glyph database, so OCR had to be trained from the start. The database was missing due to an oversight in the installer. This version includes the glyph database.
- Assembly stages may be defined, for an assembly process that has multiple stages. For example, when using two pick-&-place machines in sequence, it allows you to assign some components to the first machine and some to the second machine. Assembly stages can also be useful when the project is partially hand-assembled, because the component list is sorted on the 'active' stage.
- From the bill-of-materials, you can now print labels for the components on a label printer. Currently, only DYMO LabelWriter printers are supported. Two example label lay-outs are added for the DYMO LabelWriter plug-in. Since the label printing is implemented as a plug-in, it is easy to add support for other label printers.
- There is a new field in the bill-of-materials editor: 'product number'. This field is intended for a reference number of label that uniquely identifies the component. In previous versions, it was common to implement this feature using a 'user column'. The bar-code reader now matches to the product number before looking at any 'user fields'.
- The package browser combines the 'system' packages (from the package database of VisualPlace) and the user-defined packages are now in the same list. For user-defined footprints, the description and the pitch (for generated footprints) are saved as well. The package editor can now also be launched from the browser.
- VisualPlace now includes an OCR engine, to scan designator labels from the silk-screen. This eases the generation of placement data from scratch (when only the silk-screen images of a PCB are available).
- The overlay markers for items that are marked as 'do not mount' have changed. The new overlay markers show the orientation of the 'do not mount' part (if the orientation and the package are known for that part).
- When searching for text in a table (such as the bill of materials), you can now give options for finding partial words and for wrapping back from the end of the table to the top.
- Scanning the Gerber files for verification and optimization of the centroid files has improved again, for a few complex cases.
- Scanning the Gerber file for the outline of the PCB now also works for those PCB design programs that draw fiducials in all Gerber files (for example, UltiBoard). Outline scanning is also more robust for PCB design programs that use a minimalistic Gerber syntax.
- The Gerber parser (gerber2dib) now handles the (optional) 'file attributes'. These attributes are specified in the Gerber format specification 'J1' and later. Using these attributes, the project dialog now detects additional production files and suggests to add them to the project.
- Editing of all tables has been improved. When a cell is being edited, horizontal scrolling is

enabled in case the text exceeds the cell width. Text can also be marked, to copy part of a cell to the clipboard (previously, only a full cell could be copied to the clipboard). Key combination Ctrl+D copies a cell from the previous row, and then moves down.

- In the bill of material, you can mark components with a colour. When creating a 'PCB layout' report, the respective components are marked in the same colour.
- On loading a project (and after creating a new project), VisualPlace checks for a few common problems and shows a warning in a tip. There can be multiple warnings, and those are collected and shown at the same time.
- The Help menu has been simplified. There is now only a single option for the 'user manual' (instead of the introductory guide and the reference).
- For interface to many pick-&-place machine formats, VisualPlace can interface with the PCBSynergy program via a plug-in. The PCBSynergy program has to be installed separately.
- Support for bitmap images as the basis for the PCB graphic has been improved. When using images in 'Window Bitmap', JPEG or PNG formats, VisualPlace checks the DPI setting in the image and allows you to adjust the setting if it is found to be invalid.
- There is a new plug-in for Europlacer support. This plug-in supports both input and output. It allows to create 'program files' that are suitable for the inneo, Xpii and FLEXYS series of machines. VisualPlace can also extract centroid data (and fiducials) from programs created by the Europlacer software.
- The export of the centroid files in 'generic CSV' format (a format supported by most pick & place machines) now supports templates. The output generation settings can be stored under a template name, so when generating CSV files for different machines or different assembly facilities, the appropriate settings for each can quickly be restored. The file with the templates is in the 'shared data' directory, so it can be shared by multiple workstations (in a network).
The same plug-in now also supports the option to export components on both sides of the PCB in a single output file, and to select the assembly stage to export (the default is that all stages are exported).
- Support for Target 3001 and KiCad has been made up-to-date to the recent releases of these EDA suites.
- A new colour bar in the component list shows at a glance whether the component is at the top or at the bottom side of the PCB. The colour of the bar is the same as that for the markers (which is configurable by the user).
- Coordinates may now be displayed relative to the primary fiducial. You can set this in the Application Settings. If no primary fiducial is defined, the coordinates will still be relative to the lower left corner of the PCB.
- When there appeared space characters in the values of a 'module positions' file of KiCad, these were not always parsed correctly. This has been corrected.
- A static analysis tool turned up various syntax issues and a few resource leaks. These issues have been corrected.
- Adding columns to a bill-of-materials file would fail if columns were skipped. For example, when the existing bill-of-materials has 5 columns and you would define column 7 (but leave column 6 undefined/empty), data would not be written in column 7. This issue has been corrected.
- If one or more 'user fields' were defined as an empty string, the bill-of-materials could no longer be edited. This issue has been corrected.
- When converting from a bill-of-materials in a VisualPlace format before VisualPlace version 1.9 to the format of VisualPlace 1.9, unknown user fields and (translated) package names that mismatched between the bill-of-materials and the centroid files, could lead to

the old user fields being removed (or orphaned). This issue has been corrected.

- Conversion of bill-of-material files from an EDA suite to VisualPlace format could result in the list of designators to be present twice for each row. This is caused by the original designator list having a different column header than what VisualPlace expects for its intrinsic format. The bill-of-materials editor would then preserve the existing (unrecognised) column and add its own. This issue has been corrected by fully rewriting the bill-of-materials file when specific columns in the original bill-of-materials file are unrecognized.

Release notes for VisualPlace version 1.9

The release notes for all '1.9' builds have been combined in a single list.

- The PCB image can now be rotated in increments of 90 degrees. Image rotation does currently not work in 'camera-assisted placement mode'. Functionality like aligning the centroid files to the silk-screen image and adding/moving component positions, are *not* available if the PCB image is rotated.
- In the component list, components with an unknown package/footprint are on a red background and their orientations are not shown in the markers. This is because VisualPlace needs to have information on the footprint to determine the 'zero orientation' of the component.
- The 'Package specifications' dialog allows you to create a lay-out for the package/footprint, or to select one from a library of common footprint lay-outs. VisualPlace comes with a library of footprints.
- The 'Verify components centroids' dialog now also uses the solder mask (or solder paste mask) for accurate verification of the component centroid. The silk-screen image serves as an indicative reference to find the approximate centre, after which VisualPlace uses the specification of the package to map the pins of the component to the pads on the PCB. This *does* require that for each component, a package specification is selected or created (VisualPlace comes with a set of common footprints).
- To improve project loading speed, the rasterized Gerber files (bitmap images) are now optionally stored in a cache directory. This function is enabled by default. You can disable it via the 'Application settings' dialog.
- The bill-of-materials editor has been modified to save the edited data in the file, while preserving any extra data that may exist on a row. This functionality allows to import the bill of materials in other applications (or spreadsheet programs) that add their own columns, while still being able to edit the bill of materials in VisualPlace.
- Additionally, the bill-of-materials editor allows to import other CSV files, to complement the data in the BOM.
- In Linux, Okular is now supported in addition to Evince to view the documentation. You can choose the PDF reader in the 'Application settings' dialog (in the 'Tools' menu).
- The barcode reader interface is now only initialized if the barcode scanner is detected, and it is uninitialized when the barcode scanner is removed. This improves the operation of VisualPlace with some (hot-plugging) barcode scanners.
- CadStar (by Zuken) is now supported via a new plug-in.
- There is a new output plug-in to create CSV centroid files. Various variants can be selected for the output (field separator, field order, unit of measurement). More importantly, the desired zero-orientation of components of several classes can be specified. This is convenient when creating position data files for Europlacer machines, for example.
- The centroid export plug-in for the NeoDen TM-220A has been expanded to also support the model TM-240A. This plug-in comes with full source code.

- Bill of Materials files can now also be generated by plug-ins (so you can export the bill of materials in a format of your choice, provided that a suitable plug-in is available). One plug-in, for CSV output with configurable columns and options, is provided.
- There is an 'undo' option for operations on centroid markers (move, rotate, auto-centre).
- Fiducials are now also printed on the PCB lay-out reports.
- When converting the project to the intrinsic format of VisualPlace, VisualPlace now keeps a section to quickly restore the project to the EDA-specific format (without needing to adjust the project settings). The 'intrinsic' format of the centroid files is now mainly intended for internal use; to generate (corrected) CSV files, use the new CSV centroid export plug-in.
- In all dialogs that had a table or grid, the 'Escape', 'Enter' and 'Tab' keys were not functional to close or navigate through the dialog (because the table control used these keys for editing and navigating through the table). It has now been changed so that an 'Escape' moves the focus away from the table, after which the keys are available for navigation through the dialog. When currently editing a cell, the first press on 'Escape' closes the edit line and another press is needed to move the focus away.
- The excellon drill file parser has better autodetection of the format (precision and leading-zero versus trailing-zero suppression).
- The calibration target for camera calibration is now a PDF file instead of a bitmap image, for increased resolution. There are in fact two PDF files for the same calibration target: one pre-set for A4 size paper and one pre-set for letter size paper. Note you often need to fine-tune the scale of the printing, so that the grid of the calibration target is exactly 160mm wide and high. This is a limitation of the accuracy of common (laser) printers (or rather: of their drivers).
- When zooming up or down, VisualPlace frequently reset the scroll position back to the upper right corner of the PCB. This issue has been corrected.
- When creating a new project, the resolution of the Gerber scan conversion had no default value, and therefore the scan conversion dropped back to low resolution. This issue has been corrected.
- Auto-flip mode deselected all components in the component list when flipping the board. For manual flip, clearing the current selection is required, but for auto-flip, the selection should be kept. This issue has been corrected.
- Some IP cameras were not supported, due to their response being in a slightly different format as what VisualPlace expected. This issue has been corrected.
- Some (deprecated) Gerber file formats caused VisualPlace to crash. This issue has been corrected.
- Pressing the 'Shift' key *after* pressing and holding the 'Control' key made the cross-hair cursor disappear. This issue has been corrected.
- Manually added fiducials would only be positioned correctly when the 'alignment origin' was zero (meaning that the origin of the centroid file matches the origin of the Gerber files). As misaligned centroid files are common, this issue was a nuisance. It has been corrected.

Release notes for VisualPlace version 1.8

The release notes for all '1.8' builds have been combined in a single list.

Version 1.8 brings changes to the user interface and adds a function/dialog to verify the centroid position of all components.

- You can now add general purpose notes to a project. The presence of these notes is marked with  icons; these icons may be moved to any position on the PCB. Hovering

with the mouse cursor above the icon will show the message. For each note, you can choose whether to also include it on the reports for the PCB lay-out and/or the bill of materials.

- A new dialog allows you to compare the component positions in the centroid file to the positions determined from the silk-screen. When using this function, VisualPlace scans all footprints from the silk-screen and compares these positions with those from the centroid file. The tolerance for the comparison is configurable (the silk-screen scanning resolution is limited to 1 mil, however). This function also lets you quickly scan for possible inaccuracies in the centroid file.
- The dialog for 'Package placement corrections' has been merged with the dialog for 'Package specifications'. The information of the two dialogs is interrelated. So it is more practical (for the point of view of the user) to have a single dialog for entering the specifications and corrections for a package.
- A barcode scanner is supported for selecting components. The barcode scanner must have a USB interface and be configured for keyboard emulation. The scanner is active in the main screen and in the dialog for the Bill of Materials.
- Footprint scanning has improved, with more shapes being recognized and less 'false positives'.
- The menu lay-out has changed. There is a new main menu item, 'Tools'. A few dialogs that were previously accessed through the 'Edit' menu, are now found under 'Tools'.
- For the 'PCB lay-out' and the 'assembly flip-book' reports, the mask Gerber is no longer included in the printed image of the PCB. This provides for a less cluttered image on the reports.
- In the 'PCB lay-out' report, the labels for 2-pin SMD components are printed vertically, so that the label fits better inside the component outline.
- Alignment of Excellon drill files to the Gerber plot files has improved; it is no longer necessary to set the origin of the files to the lower-left corner of the PCB. There is also better support for oblong holes.
- The camera calibration controls for the scale, the barrel distortion and the keystone distortion now move with the minimum step size when clicking to the left or to the right of the track-bar marker. This allows for fine adjustment with a touch screen.
- The 'swipe' algorithm for scrolling the PCB has improved. Slow swipes are now more accurate.
- VisualPlace now also reads fiducials from the centroid files and puts centre marks on them. In addition, there is a new dialog to mark an existing feature on the PCB as a fiducial. Export plug-ins can use the fiducials to recalculate the coordinates.
- The resolution for the Gerber scans can now be chosen, per project. The default resolution is 600 dpi. Higher resolutions increase the accuracy of the displayed data, lower resolutions reduce the memory requirements of VisualPlace (and speed up many operations).
- The plug-in for KiCad now also supports the format of the current stable release (2013-07-07 BZR 4022).
- The plug-in for ULTIboard now also supports the latest releases of the product.
- In the Bill of Materials table, you can now also add new components (in previous releases, you could only edit existing components, or delete components).
- When changing the footprint in the Bill of Materials, the centroid files are now updated as well, *provided* that the centroid files are in VisualPlace's intrinsic format. (If they are not, you must export the centroid files first.)
- In the main view, the mouse-wheel now zooms in and out. The mouse-wheel works in both

design-mode and camera-mode (provided that a PTZ camera is installed).

- Flags for 2-pin components are now also (optionally) saved in the project file. This is especially useful when working with the intrinsic format of VisualPlace, because the 2-pin flags are not saved in the centroid files (and there are no format-specific corrections for the intrinsic format). For EDA-specific formats, saving the flags in the project file is optional, for the intrinsic format, the flags are always stored in the project.
- A special case was added to the colouring of the PCBs, to avoid a horizontal 'stripe' on some shapes of PCBs.
- The treeview buttons for expanding and collapsing groups looked weird on recent versions of Wine. This bug has been corrected.
- When opening an archived project, VisualPlace could prompt to reload production files, claiming (incorrectly) that these had been changed since they were archived. This bug has been corrected.
- Some dialogs required values with a fractional part to use a decimal period, even if the user interface was set to a language where a decimal comma is used. This bug has been corrected.
- Converting Gerber files could fail, especially if a mask was included. The failure was due to a memory allocation error combined with path length limitations. This bug has been fixed.
- When pasting text with embedded TAB characters into a cell in a table, the text would not be displayed correctly. This bug has been fixed by replacing TAB characters with spaces.

Release notes for VisualPlace version 1.7

The release notes for all '1.7' builds have been combined in a single list.

- *Any 'action' plug-ins that you have developed, must be adapted.* In the plug-in interface of version 1.7, the function `vp_Configure()` has an extra parameter. The change in the plug-in interface was motivated to better support a new type of plug-ins ('output' plug-ins, to generate files for a specific pick & place machine).
- You can now subdivide a PCB into zones and sort the component list with the zone code as a criterion. Zones are useful for manual assembly, in two situations: 1) when using a camera for assembly and the board is too big to fit in the camera view, and 2) as a compromise between viewing *all* components of a particular value/footprint at once and viewing *just one* component at a time.
- The Project Settings dialog has been redesigned to show only the options that are relevant for a particular configuration. This means that when you toggle the project from single-sided to double-sided, the dialog expands to show more fields (and collapses again when switching back to single-sided).
- VisualPlace no longer substitutes Gerber files with bitmapped files (TIFF files) when you create (or modify) a project. The advantage is that if you edit a project, it will list the original files on which the project was based. For some operations, such as automatically finding the origin of the design files, the original Gerbers are required. Note that loading a VisualPlace project takes a few seconds longer, because scan-converting a Gerber file takes more time than loading a bitmap file.
- You can add a solder mask Gerber file, or a solder paste Gerber file to a project. Adding this file (or two, for PCBs with components on both sides) may clarify positions of components in case a component does not have a good outline on the silk-screen.
- VisualPlace can also load a drill file. The purpose of the drill file is to help align the PCB below the camera (for camera-assisted manual assembly) —and especially in the cases that you need zones because the PCB is too large to fit in the camera view.
- You can now also create plug-ins for exporting the centroid files in a specific format. See

the documentation for the details on creating plug-ins for VisualPlace. Source code for an example 'export' plug-in (NeoDen TM-220A pick-and-place machine) is included.

- You can now scroll the board by 'swiping' it, like is commonly done on touch panels.
- If you are using VisualPlace to create centroid files, the new option to automatically centre the marker in the footprint, makes this process more efficient. For new components, you will only have to click somewhere inside the footprint shape and the marker gets centred automatically. Note that for complex or open footprint shapes, automatic centring may fail—in these cases, you must still position the marker by hand.
- The plug-in for branching to the Octopart component database now has a configurable URL and query prefix string. The plug-in can therefore be more generally used with other databases—provided that these databases offer a search interface with parameters on the URL (so-called 'GET requests').
- Several message boxes have been replaced with a pop-up message window in the lower-right corner of the main window. This pop-up window is less intrusive than a message box that requires you to click 'OK' before continuing.
- The scanning and conversion of Gerber files has improved. This improvement leads to better visualization of the PCB and, more importantly, a better ability to automatically align the centroid files to the PCB image.
- When using a PTZ camera, you can now zoom out by clicking and holding the left mouse button (for roughly half a second). As before, a click inside the camera image zooms in on that spot. You can now also zoom back out with a click-&-hold (previously, you needed to use a menu command or press the '-' key).
- When running under Linux, configuring of Wine to use the native version of the GDIPLUS DLL is no longer required (even for Wine 1.3 and earlier). For context-sensitive help, Evince is still required, though.
- Components on the bottom side of the PCB sometimes had the orientation inverted. This bug has been corrected.

Release notes for VisualPlace version 1.6

The release notes for all '1.6' builds have been combined in a single list.

- VisualPlace supports network cameras for a real-time preview of the PCB. VisualPlace overlays the markers (for locating the components) over the video image. Cameras with pan and zoom are supported (so-called PTZ cameras), to enlarge the part of the PCB where the component must be placed. When doing manual placement, VisualPlace therefore functions both as a magnifier and as a component locator. (At the moment, only network cameras are supported. A PTZ camera is required for some of the functionality.)

The set-up for network cameras includes sliders to adjust for barrel/pincushion distortion and keystone distortion. The camera can be calibrated using the provided grid image and a special 'calibration' demo project.

- Support for Wine has been enhanced. VisualPlace can now generate the reports in PDF format under Linux and display these. On-line help is now also available in Linux, provided that 'Evince' is installed (VisualPlace requires a feature in the PDF reader that currently only Evince has). With recent versions of Wine (1.4), no reconfiguration of Wine is necessary—with older versions of Wine, it is still needed to install the 'native' GDIPLUS DLL.
- In all tables, when text does not fit in the cell of a table, a tooltip is now displayed (with the full cell text) when the mouse cursor hovers over the item. (There are also other minor improvements in the grid editing.)
- A new plug-in connects VisualPlace to the Octopart database. When you enable this plug-in (see Edit / Application settings) and you double-click on a row in the BOM dialog (see

Edit / Bill of Materials), a browser will open with the selected part in the Octopart database. The information that the plug-in passes to Octopart is configurable. This plug-in comes with full source code, to also serve as an example how to write plug-ins for VisualPlace.

- A plug-in for the DIY Component Carousel (published in Circuit Cellar, November 2012 issue) is included. This plug-in also comes with full source code. Note that you need to install the software from the Component Carousel to make this plug-in functional.
- Support for KiCad has changed to support the upcoming new format for centroid files.
- VisualPlace comes with an ULP for EAGLE. This is just for convenience (you may also use other ULPs, as long as they provide the data that VisualPlace needs).
- VisualPlace now registers two sound events. These are useful for manual PCB assembly, because they give an audible cue that a new part is needed. Two short WAV files for the events are in the 'media' directory below where VisualPlace is installed. Note that VisualPlace registers the events, but does not assign the sound files to these events by default. If you wish to use the sound, use the 'Sounds & Audio devices' applet in the Windows Control Panel.
- Sometimes, VisualPlace would create its INI file in a location where Windows blocked writing. VisualPlace now has improved checks for this situation, and puts the INI file in the 'application data' directory for the current user when it detects that it is installed in a read-only path. If VisualPlace is installed in a location that gives read and write access, the INI file is still in the same path as the VisualPlace application itself.
- When Gerber files were changed outside VisualPlace, this was detected, but the changed files were not re-converted and re-loaded. This bug has been fixed.
- On boards with components on both sides, VisualPlace could incorrectly give the message that (one of) the centroid data files was changed. This bug has been fixed.
- When a small PCB was centred inside the main view, the coordinates of the cross-hair cursor (shown when you press and hold the Control key) were incorrect (because the margins added to the top and left were not taken into account). This bug has been fixed.
- The 'opacity' setting for drawings in the generated reports did not work well. This bug has been fixed.
- Some EDA-specific plug-ins could insert random text into unused user fields. This bug has been fixed.

Release notes for VisualPlace version 1.5

The release notes for all '1.5' builds have been combined in a single list.

- Adding positions for components that lack placement information, or adjusting the centroid position of components has become much more convenient and efficient. Noteworthy are the heuristics that VisualPlace provides to automatically find the centre of a shape —see the presentation (on YouTube) for an introduction.
- For accurate positioning, there is a new, larger cross-hair cursor. This cursor is used in alignment (PCB origin), component positioning and measurements.
- VisualPlace now checks for changes in the production files that are made by other programs, and prompts the user to reload (and possibly reconvert) the fabrication files.
- All files that are part of a project can now be archived from within VisualPlace. The archive is a standard 'ZIP' file. VisualPlace stores in the archive all files that are referenced in the project (images, bill of materials, centroid files and the project data file itself). You can also open and archived project directly from VisualPlace (that is, without need to extract the ZIP file first), but modifications that you make to an archived project will not be stored back into the ZIP file.

- For the help system, there is now a better integration with PDF readers, allowing for context-sensitive help where appropriate. The Adobe Reader and Sumatra PDF reader are supported (other PDF readers are supported as well, but these may not support context sensitive help).
- VisualPlace can automatically toggle from the top side to the bottom side of the PCB, depending on the side that the selected components are on. The active side is written in the caption of the VisualPlace application.
- More zoom levels, giving finer zoom steps, and extending the zoom-out level down to 10% of the original size. The zoom levels have also been renamed. The zoom levels are now related to the display resolution. Therefore, at a zoom level of 100%, the image of the PCB will now be at the real size of the PCB. Previously, zoom levels were relative to the resolution of the PCB images, which is typically 600 dpi (the typical resolution of a display is 96 dpi). Zooming in and out is now also quicker, and therefore more responsive.
- When you let VisualPlace create the PCB artwork from (silk-screen) Gerber files and also provide a Gerber file for the PCB outline ('PCB edges' or 'contour routing'), VisualPlace now displays the PCBs in colour, instead of black-&-white. The colours for the PCB image (background and 'text print' colours) can be selected by the user. The defaults are green for the PCB background and near-white for the silk-screen print, but you can now choose to have brown or red PCBs. The colouring can also be switched off. Colouring of the PCB images is only functional for images that are extracted from Gerber files.
- When changing the sort order in the component list, any new columns that are specific to the new options are adjusted to a non-zero width, to ensure their visibility.
- In the BOM report, if the same component is on both sides of the PCB, these components are now combined on a single row (previously, the BOM was per side of the PCB).
- All reports have been modified to handle components marked as 'do not mount' in a more explicit manner. The 'assembly flipbook' excludes these components and the other reports print a notice for them.
- When changing a Gerber file (and applying it to the VisualPlace project) after having exported the centroid file to the VisualPlace intrinsic format, the alignment of the centroid file was not preserved. This would require the user to re-align the centroid file to the new Gerber artwork (after this changed Gerber file is set in the project). This flaw has now been fixed.
- In the dialog for specifying package/footprint names and translations, the attribute '2-pin' was not saved correctly.
- When the text in a grid did not fit inside the cell, a triangular 'arrow' indicates truncated text.
- When deleting components from the BOM, the centroid file was not always written back correctly. (When components are removed from the BOM, these components are now also deleted from the centroid files.)
- In the BOM report, the line to separate the rows was not drawn at the bottom of the row for rows that spanned multiple lines.
- In grid editing, column resizing and column sorting interfered with each other; trying to resize a column would frequently result in sorting on that column too.
- When pressing an unrecognized key combination with the 'Ctrl' key, the mouse cursor would stay a cross-hair, even after releasing the Ctrl key.
- Editing cells in a grid has improved. When a string in a cell is very long, the text did not always scroll horizontally.
- Some plug-ins for EDA suite support could fill any undefined 'user fields' with random data.
- Bug fix: the images in the reports did not show correctly when the 'fit to page' scaling

option was selected.

- Bug fix: some special characters did not appear in the PDF reports, due to an incorrect "font encoding".

Earlier versions

No history was kept for the release notes for versions 1.0 to 1.4. Roughly speaking, version 1.0 supported only KiCad and required the silk-screen images as bitmaps. The plug-in system to also support other EDA suites came with version 1.1, along with Gerber support via gerb2tiff. Reports and printing support were added in version 1.2. This was also the first version to run well under Wine (for Linux & OSX).

I have no recollection of what changed in version 1.3; it was probably primarily a bug-fix release. Version 1.4 followed quickly and supplanted the printing support with PDF reports. It also made a few user-interface enhancements.