GerbTool V14.X Release History

Version 14.2

The following issues have been addressed in Version 14.2:

366   Add $$TIME and $$DATE variables to macro language.

759   Drill DRC errors don't show on the highlight layer.

893   Pad to Pad DRC with customs measuring correctly, but reporting error when none exists.

901   Stacked pad removal not taking polarity into account.

944   DRC Values display to thousandths of a mil, but only allow entering to tenths.

1043   Convert Drawn Pads dialog box needs to store position in registry.

1251   GerbTool text should stay "text" when outputting to other formats that have "text" entities.

1340   Annular ring check problems on a particular dataset with stacked pads.

1393   Problem with the text in a particular dataset after Advanced Panelization.

1633   Add support for ODB++ XML format.

1758   Multiple rotations were used on the stepped images in a Panel. After exporting to Gerber and re-importing, only the original non-rotated images came in.

1919   Problem with polarity when exploding customs.

1960   Gerber import of zero length draws should be converted to flashes.

1963   DXF import text problem.

1969   Suggestion to make 360 degree arc mode the default.

1976   Stencil Generation Enhancements to replace Home Plate function.

1979 $$DSNNAME truncates filenames.

1980   A particular netlist extract is wrong; has a pad connected that should not be.

1983   Edit Tool from menu or dialog box doesn't display selected tool.

1984   Request for warnings on/after import.

1987   Customs coming in with incorrect orientation from Gerber.

1988   Drills missing from Export in S&M format.

1990   Error running the DRC Information Check "Legend on PTH".

1992   When arc almost forms a complete circle, data shifts out of place.

1995   Fit to page with Batch Mode Printing does not fit each page -- only the first.

1996   Gerber Import handling arcs incorrectly.

1997   Measure problem to polygon resulted in serious errors.

1998   Isolated Pads removal, removes non-isolated pads.

2000   ODB++ import incorrectly rotates some pads on SM and SP layers.

2001   Polygon export problem.

2002   Raster to Vector conversion problem.

2003   Import Wizard has problem interacting with path dropdown list.

2004   Minimum Trace Information check reporting incorrect result on polygons.

2006   Polygons being incorrectly identified as stubs in DRC check.

2007   Cam350 v8 files not importing correctly.

2008   Convert composite to layer problem on polylines drawn with rectangular apertures.

2009   Auto stencil conversion not working on rounded rectangles.

2010   Testpoint problem determining end points.

2011   DRC sees negative pads as errors.

2012   Intermittent crash on Find Duplicates when selecting all layers.

2013   HomePlate melf and oblong not working well on pads with a certain orientation.

2017   Aperture bug with rounded corners.

2021   Convert Drawn Pads to custom gives problem with flipped arcs on a particular design.

2024   Netlist macro command may not use the correct option settings.

2027   Stencil Auto Conversion dialog box change requests: Add Close button, layer selection list, ability to save shape conversion list, and additional fields to show original pad sizes.

2028   Request to add an Edit button to the Stencil Shape Setup dialog box, and move the Setup Shapes menu command to top of Tools|Stencils menu.

2029   Undo label should be updated to reflect the generic nature of the stencil command.

2030   Stencil conversion shifting and shrinking resultant pads.

2031   Offspring export precision too small.
2032   During Offspring export polarity is lost on polygon voids within negative plane layers, and negative plane layers are not getting set with a polarity type of "negative." During Offspring import, a polarity type of "negative" and layer type of "inner" should set the GerbTool layer type to Negative Plane, and layer colors of a file exported to Offspring from GerbTool should be maintained on re-importing the file to GerbTool.
2033   Offspring import crashes on a particular database.
2037   Improper parameters for auto panel cause memory allocation failure.
2039   Isolated pad removal not removing all isolated pads.
2040   Export DPF includes unused zero-sized apertures.
2041   Converting draws to raster polygon hangs.
2042   Convert composite to layer drops clear flashes from a particular dataset.
2046   DXF export poly/outline issues.
2048   Offspring export/import problems with a particular dataset: Incorrect drill sizes were exported from mixed NC layers. Mill data was not exported as continuous paths. Layer tags for paste layers were not being exported. Layer colors of the exported offspring file did not match on re-importing.
2049   Offspring import polygon void problem with this dataset.
2050   Acid trap detection is not working properly.
2051   Paste mask optimize not removing drawn pads.
2052   PADS import missing pads and refdes text with this dataset.
2053   COMBINEMILLPATHS macro command spelled wrong in Help system.
2054   Analysis Information check results do not print from context menu.
2055   Stacked pad removal does not remove pads that are not at the exact same location.
2056   COMBINEMILLPATHS Layer parameter problem.
2060   COMBINEMILLPATHS macro command parameters not correct in documentation.
2061   GerbTool shouldn't allow custom aperture names that start with a number.
2062   Netlist wrong on a particular dataset.
2065   DRC border check not catching items that actually touch border.
2066   Offspring import leaves funny filename in MRU file list.
2067   Selection filter changes during a macro are not reflected properly in selection filter tool bar upon macro exit.
2068   Composite to layer problem will only complete 1 conversion per GerbTool session.
2069   Macro command problem dealing with group mode.
2070   DXF hatch import problem.
2071   Request for macro access to layer comments.
2072   Demo version is being used for production NC work.
2075   Raster to Vector command fails on a particular dataset.
2080   Bottom test points being ignored on a through hole board.
2083   Polygon Engine exceptions not currently handled by EmitGTD code.
2084   Analysis Drill/Pad check doesn't report on pads that are smaller than drill.
2089   DRC Trace to Trace not working properly when no netlist is present.
2090   GETDCODEINFO and PUTDCODEINFO macros don't have access to rotation.
2095   Crash related to group selections by UserData.
2096   Request to change group selection by net or UserData, to allow filtering out other data, such as drill hits.
2097   GerbTool is not recognizing the %IPNEG parameter and thus isn't displaying 274-X data correctly.
2099   Netlist generation appears to be dropping voids on a particular dataset.
2108   Copper Sliver fixed with a dark poly when it should have been fixed with clear data.

May 9, 2005

Version 14.1

The following issues have been addressed in Version 14.1:

195   Aperture converter rounding error.
255   Add hole chart problems when rotation is specified.
281   Request to be able to remove or rename a command/macro in the My Commands tab of the Navigator via right-click menu.
301   Can't undo virtual panelization.
333   Orthogonal line snap not available during Add Vertex.
334   Ortho line snap not behaving properly during Add Slot.

383	Aperture size appears wrong when imported aperture data are not centered on 0,0.
403	Request for automatic method of removing Virtual Panel S&R codes.
409	CREATELAYER macro command fails to return layer number to $$STATUS
495	Teardrops dialog box is not adapting to units being microns.
796	ODB++ Import symbol translation problems.
810	Netlist Generation may be wrong if negative data present.
929	Solder mask generate doesn't prompt to create a netlist.
975	Edit|Dcode|Polarity leaves cursor fragments.
1017	Display of raster filled polygon arcs not consistent with regular arcs.
1062	Some Pad to Trace errors indicate they are fixable, but fixing them does nothing.
1091	Composite Setup dialog box needs updating.
1125	Drawing with anything other than Round, Square, or Rectangle displays as round.
1203	Need UI method to notify users when warnings/errors have been logged.
1265	Macro commands for IPC-D-356/A import and export need to be updated.
1358	Netlist Compare produces invalid errors.
1359	DRC errors incorrect when imported aperture data are not centered on 0,0.
1403	When moving vertex, orthogonal switch is disabled.
1413	Analysis|Copper Area dialog text should reflect usage of "NC layers" instead of "Drill".
1446	Rotate command does not update Navigator after adding apertures.
1561	Netlist Generation fails on a particular data set.
1565	Some polygons from a particular imported design contain only a single point.
1607	Macro Variable $$CURRENTDDRILLTOOL not working.
1617	DRC Auto Fix problem on a particular data set.
1629	Request to support Gerbers from Japan with decimals in them.
1654	DRILLTOGERBER command obsolete, need NCTOGERBER command.
1657	EXPORTDXF should be updated to reflect current product functions.
1660	GERBERTODRILL should be replaced with GERBERTONC and updated appropriately.
1704	Exploding transformed drill patterns results in non-NC data
1724	DRC Auto Fix problem on pad-to-pad errors.
1737	TBar width fix is not adjusting the passive size correctly.
1756	ODB++ import problems on specific design.
1759	Panel Rotate command doesn't store last value entered.
1789	DRC fixes should not use negative (clear) data if at all possible.
1800	Exported net locations are incorrect on some custom pads.
1804	Netlist Compare error when working from PADS Import Netlist Only
1821	Solder mask optimization can crash when user attempts to abort.
1848	PowerPCB 5.0 ASCII file not reading in correctly.
1849	Changing legends assigned in the NC Table is not reflected in the drill hole chart.
1855	When exporting ODB++ after performing Advanced Panelization, re-importing the data fails.
1861	Problems adding tabs if amount of space available is between minimum required and the amount required to insure it will fit if the path is compensated.
1865	DXF Dimension Text not importing.
1866	A particular file loads incorrectly when dropped into GerbTool.
1868	Report files disappearing upon close of GerbTool.
1869	NC Export of advanced panelization designs has tool export problem.
1870	NC export with virtual panel not working correctly.
1871	Export order not coming out correctly with pilot holes, even after optimization.
1874	Import Wizard detects a particular file as leading zeroes suppressed, but file is trailing zeroes suppressed.
1877	DRC Trace to Trace check failing to detect error on parallel lines at 45 degrees.
1879	IPC-D-356A Flying probe export hangs on a particular design.
1880	Copy command problem when windowing data and selecting multiple destination layers.
1881	Polygons arcs are segmented for DPF export.
1883	Spurious drill hits being created by Excellon export/import.
1884	Sieb & Meyer Mill Import - non-compensated path following compensated path gets compensated.
1888	Postscript Export incorrectly scaling certain flashes.
1890	Crash using Add|Drill in SR2 after adding a new tool.
1900	Gerber Import is incorrectly reading polygons that consist of a single 360 arc.
1901	Clockwise arcs from version 9 & 10 design files not loading properly when opened in V14.

1902   Sieb & Meyer import problem. The command state is not being terminated for drilled slots.
1904   GerbTool crashes when trying to import a particular set of Gerbers.
1905   Macro Command STOP crashes when used in a submacro.
1906   Assert/Memory leak in Fire 9000 initialization.
1908   Reset gauge causes asserts and partially aborts command.
1909   Pads rotated incorrectly on import of specific ODB++ design.
1911   Dropped void during netlist generation.
1912   Mill paths appear corrupted after changing the database origin, then saving and reloading.
1913   Simple Panelize dialog box is no longer working properly for virtual panels.
1916   Exporting Gerber file in incremental coordinate mode causes crash during import of same file.
1917   Testpoints not exported correctly via IPC-D356A export.
1918   Unexpected warning message appears during Gerber export.
1920   NC output problems when exporting in Modal mode.
1922   Problem during Import Mill for a design that had a stepped and repeated mill path, and the tool assigned to the path was not defined.
1923   Gerber Export crashing after ODB++ files loaded.
1924   Drill file import problem.
1925   Feed Rate not output in Excellon formats.
1926   Crash during Postscript Output.
1927   File name parameter of macro TEXT command not working.
1929   DXF file scale should not be changing during Import.
1930   Crash if DXF file is open in another application and user tries to access the import setting from the Import Wizard.
1932   Importing a particular Gerber file causes a stack overflow.
1934   ScaleLayers macro not executing on the correct layers.
1935   ApReport macro not executing on the correct layers.
1938   Crash in the Simple Panelization dialog box.
1943   Unknown Netlist Compare errors on Blind and Buried design.
1944   GerbTool allows the application to be shut down during file export, causing incomplete files.
1946   Invalid layer compare differences reported with a particular NC layer of a virtual panel.
1947   Change request for up-path display to not automatically coming on after optimization.
1948   Auto-centered customs not displaying correctly during open of an older design.
1949   Netlist Compare reports invalid errors on a particular design.
1950   Improper "Unable to determine Format" message on what appears to be an OK Drill file.
1951   Crash importing a particular Gerber file.
1952   Setup Layers crash after cutting a layer from a particular dataset.
1953 $$LTNC is not getting its value set.
1954   MERGEGERBER gives syntax error on attached macro which uses a variable for the filename.
1955   Polarity behaving like a separate command even when used as a parameter.
1956   Crash on ODB++ import after warnings of duplicate custom not accessible.
1957   ODB++ problem with features not matching Gerbers and Valor VUV.
1961   Gerber LN command interferes with Merge Poly option during import.
1962   IMPORTNC macro command incorrectly documented as MERGENC.
1964   When mills are mirrored, compensation ends up in wrong direction.
1965   File Merge problems with NC Tools.
1968   Problem with View All after import.
1970   Importing IPC-D-356 file fails to map layers properly in all cases.
1971   Attempts to add polygon from UI fails.
1972   Add Polygon dialog box shouldn't allow the selection of apertures that are not valid for draws.
1973   Older version GTD files causing a crash during conversion.
1974   Net Compare producing a problem net open result.
1977   Macro POLARITY command not working properly.
1982   Polygon Engine abort during netlist generation on a particular dataset.

<div align="right">February 7, 2005</div>

Version 14.0 Service Release 2

The following issues have been addressed in service release 2:

1850   Netlist compare problems with a particular dataset.
1863   NC import dialog box problem with tool table name when accessed from the Import Wizard.

1873   External net names with single quotes being lost.
1875   Polygon corruption during Gerber Import with a particular dataset.
1876, 1878   Small arc corruption problems during Gerber Import with a particular dataset.

Version 14.0 Service Release 1

The following issues have been addressed in service release 1:

212   No grid display in Aperture Editor.
701   Netlist Extract not consistent for polygons touching at corners.
853   PostScript accuracy problem.
1117   Gerber Export: Unneeded polygon slicing to expose disappearing voids.
1321   Polygon Engine Error: Disappearing circular void.
1328   Export DXF should maintain Layer Names with numbers in them.
1346   IPC-D-356A Export Issue (Blind and Buried).
1379   Netlist regeneration during Analysis fixing needs "Ignore this message" option.
1380   Netlist problem - Polygon Engine dropping a clearance.
1425   Need ability to compact/renumber NC Tool Tables.
1496   Fix error option should not be available on the Generate Paste Mask dialog box.
1538   Double mill path visibility change request.
1571   Macro POUR command not working with Hatch option.
1573   Gerber format detection gets m.n wrong on a particular dataset.
1662   GETTOOLINFO macro command must be updated for v14 NC changes.
1688   Custom NC Tab placement/size issue.
1720   Edit|Clip not working correctly on Polylines and Raster Polygons.
1729   Display anomaly when adding polygons interactively.
1732   Tool list not updating in Navigator after certain commands.
1748   Create Mill From problem with plunge.
1802   Customer requests Layer Compare Tolerance value to be saved in registry.
1807   Double mill path UI change requests.
1822   Solder Mask & Paste Mask commands use non-standard methods of status bar prompting and activity indication.
1823   Flatten Panel isn't available for users with Advanced Panelization feature.
1824   Import 356A not always mapping bottom layer correctly.
1827   Paste Mask corner option not working.
1828   Combine Mill Path problem.
1830   ACR $rot keyword not working properly.
1834   Import detection says Gerber is trailing-suppressed when it is actually leading-suppressed on a particular dataset.
1835   Unable to export Sieb & Meyer format drills via macro commands.
1838   File Merge command is not mapping tool tables appropriately, map dialog is not called.
1840   IMPORTFORMAT macro command isn't working properly.
1841   Unused tools are being exported.
1842   Need a way to remove tools from tool list via macro commands.
1844   GETTOOL and PUTTOOL should be updated appropriately for v14 capabilities.
1846   Crash trying to import specific DPF file.
1851   Crash during save after running netlist generation on a particular design.
1853   HPGL export crash with Fill mode.
1854   Ambiguous custom apertures names resulting from a particular dataset.
1858   Problems reading a particular Gerber file that contains aperture macros defined multiple times.
1862   Macro drag & drop select polarity keyword not handled properly.
1867   Analysis of silkscreen causes crash.

Version 14.0 Initial Release

This document will contain a list of issues that are addressed in future service releases.