

PollEx Updates

Period: 2013.08.24~2014.09.30

31 December. 2014



-
- **Window Font** - supporting feature add.
 - **ExtractPDBB**
Add command to read ODB++ folder.
Add parameter which can keep log file.
Add option to read DFMR and DFESR at same time using PollEx command
Ex) PollExPCB.exe /DFMER "PDBB File Path" **"DFMR File Path"** **"DFESR File Path"**
 - **ODB++**
Add to exclude miscellaneous data (MISC) in I/F.
 - **ZUKEN PWS**
Change to read file which is having routing only in WDF.
 - **Mentor-Expedition**
Add readable font, **"std-proportional"**.
 - **Export-Agilent ADS File**
Add to export ADS format (ADFI).
 - **Setting-Board Information**
Add to export file name, number of layers, board information and save into MS/Excel.
 - **Zoom In/Out Display**
Implement mouse pointer location recognizing feature for zoom-in/out.
 - **Properties-Stack up**
Add feature to import/export PCB stack-up.
 - **Tools-Net Color**
Add setting file (*.NCLR) save function after net color setup.
 - **View-Set Capture Window**
Add user-customizing scale for image capture.
Add default capturing mode in environment.
 - **Properties-Local Parts Linkage & Edit**
Add feature to save UPF files into PDBB file path when using button, **"Generate UPF Parts"**.
 - **Setting-Environment/General**
"Don't Copy Same Part into Local": don't make or duplicate same part, UPF during linking operation.
"Automatic Update": when this option is checked, program can detect current version and if necessary update higher version with administrator specified version.
 - **Tools-Picking**

Add pin connected properties on picking tool when users select components pin.

- **Tools-PCB Explorer**

Add feature to hide unselected objects from list for references and nets.

- **Tools-Component Arrangement Plan**

Change to control on/off status for part and reference name in reference list tab.

Add filtering type for user defined pin in CAP environment.

If 1st pin is not same with the package and placed component. user have to mark on actual 1st pin. Can do a marking for that kind of component.

Add features, "**Text Area Fill**" and "**Component Group Fill**".

Add feature to display setting file name on top of window.

Add feature to be displayed setting file name during PDF print.

Add feature to export and show property value list based on property value set in CAP setting.

- **Option-Real PCB Assembly Viewer**

Add to export component shape into CUBOID with component area and height even if there is no 3D package geometry.

New option, "**Show Solder Resistor**": show solder mask on 3D board shape.

- **3D Draw Object**

Add new feature to display in solder resistor "**Real PCB Assembly Viewer**" and "**Component Mounting Emulator**".

- **Restricted PDBB**

New feature to save certain area of board, selected component or component connected nets into another PDBB files. To communicate with outer companies or prevent security issues, new features will help users.



- **DFM-Environment**

"**Reference File Link**": Add new feature to link reference book for each DFM item.

"**Link to All Reference File**": Add new feature to register or delete reference book to DFM item in batches.

- **Board-Routing Slit**

"**Clearance from Overlapped PCB Outline**": Can define the line as 'Routing Bridge' which is overlapped on PCB outline for checking a clearance to certain component.

"**Except Group**": Exclude certain components in checking the clearance between부품 component and routing slit.

- **Component-Component Spacing**

"**Except for SMD pins, not having Metal-Mask**": Exclude SMD pin which is not having metal

mask in checking.

"Except for components, having same properties": Exclude checking between component and component which are having same property values.

"Except for components, having different Property Values": Checking option does not check between components, having different property value.

"Except for overlapped components with selected at reverse side": Does not check for components placed at the other side of BGA.

- **Component-Clinch Spacing**

"Except for components having different Property Value": For components having different property values, exclude them in checking.

"Clinch to Net": Clearance checking between clinch-area and nets.

"Except Comp Group": Option to select nets which will be excluded in checking item, "Clinch to Net".

"Except for Component in same Net": Option to exclude clinch-areas of components, connected in same nets.

- **Pad-Via Spacing**

"Via Spacing to Solder Mask": Add checking clearance between via and solder mask.

"Exception Net": Exclude checking for vias connected to certain nets.

- **Pattern-Connected Pattern Direction**

"Trace": Check only for the case that patterns are connected to pad with line.

"Copper": Check only for the case that patterns are connected to pad with copper.

"Pattern Width Ratio(%) to Pad Size": Check the pattern width ration to pad size.

"Except Net Name Filter": Do not check certain nets. Set net name with filter.

"Right-Angle Pattern to Pad Edge": Option to check whether pad connected patterns are connected with 90 degree.

"Keeping 90 Degree within Given Distance from Pad": Checking option whether pad connected pattern keeps 90 degree within specified distance from pad.

- **Pad-Copper Connected Pad**

"Except Thermal Pad": Excluding thermal pad in checking.

"Filter Type, Pin/Net Name Filter": Checking only for certain nets or pins with naming filter.

- **Board-PCB Outline Spacing**

"Allowable Distance": For clearance checking between PCB outline and components, use checking with given measure base type.

"PCB Outline to Metal Mask Clearance": Checking clearance between PCB outline and metal mask.

- **Pad-Remove Copper from Pad**

"Except Thermal Pad": Exclude thermal pads in checking.

- **Placement - Component Placement Angle**

"Range Angle per Part Name": Can check the placed angle per each component as based on

part name.

Board-Missing Hole

"Component Clearance": Check the clearance between specified components and missing areas which are placed in horizontal/vertical direction.

"Measure Base": Check the clearance from missing-hole or missing-hole objects' center points.

- **FPCB-Coverlay**

"Top/Bottom Edge to Edge": Add checking along X or Y axis in clearance checking from top/bottom coverlay open area edge.

- **Pattern-Keep Out Pattern**

"Exception for Component Connected Pattern": Exclude checking for pattern, connected to certain components.

"Component Group": Specify excluding components groups in checking.

"Exclude Pattern within Distance from Component": Exclude checking for patterns within distance from specified components.

"Select Layer": Add option to specify checking target layers.

"Component Placed Layer": Option for checking patterns only on component placed side.

"Opposite Layer": Option for checking patterns only on component's reverse placed side.

"Inner Layer": Option for checking patterns only on inner layer.

** Can define the number of inner layer from component placed layer.

"Copper": Check only for the case that patterns are connected to pad with copper

"Trace": Pad Check only for the case that patterns are connected to pad with line.

"Except Net Name Filter": Exclude checking for specified nets.

"Exception for Component Connected Pattern": Exclude checking for patterns connected to certain components.

- **Misc-TCP Bonding Mark Option**

"TCP Bonding Mark Count": Add new inequality signs whether TCP bonding mark is much many or much smaller than given value.

"Arrangement for Y Location": Checking whether the value of Y axis of TCP bonding mark is in certain range with tolerance value.

- **Component-Silk to Silk**

"Set Text Line Width as Zero": Option to set text thickness value with 0 in checking.

- **Placement-Mark Placement**

"Except Component Group": Exclude certain component group in checking.

- **Misc-TCP Align Hole**

"Hole Alignment": Checking whether TCP align hole is located at same X or Y axis. Even if, it is deviated in tolerance, it will be passed in checking. (Overlapped slit hole will be exempt in checking.)

"Hole Count": Count the number of TCP align-holes.

- **Board-Data Existence**
 "Except Text": Exclude certain character strings in checking.
- **Placement-Standard Component**
 "Footprint matching to part name": Check whether components' part name and footprint names are matching with CSV document or not.
 "CSV File Path": CSV file path.
 "Component Group Except": Exclude certain component group in checking.
- **Board - V-Cut**
 "Exclude Component in V-Cut Outside Area": Exclude component of V-Cut area outside will be excluded in checking.
- **Drill Size-Under Hole Via**
 "Except Group": Exclude certain component group in checking.
 "Exclude Pad Area": Exclude pad area in checking.
 "Laser Via Definition": Option to set laser-via.
 "Hole Size": Via-hole size. Via-hole size smaller than given value will be treated with laser-via.
- **Board-Routing Slit**
 "Clearance from PCB outline in certain region": In case of routing slits are overlapped with PCB outline, clearance will be checked from PCB outline for overlapped region. ""
 "Except Group": Exclude certain component group in clearance checking with routing slit.
- **FPCB-Tab Sub Mark**
 Add new item to check whether mark for machinery TAB component is well designed or not.
 Change to check clearance between marks and pattern on layer which Tab Sub-marks existing.
- **FPCB-Figure Layer**
 "Check Type": Clearance checking between objects regardless top/bottom.
- **Board-Label Box**
 "Label Box Size Check": Add option to check the size of label box registered with component.
 "Check Object for Polygon Only": Check only for polygon.
- **Board-Min Silk Width**
 "Except Comp": Exclude certain component group in checking.
- **Component-Silk Print Between Two Pins**
 "Select Target Side": Add option to check using component placed layer.
- **Ground Wall**
 "Except for SMD Pins, not having Metal Mask": Exclude SMD pin, not having metal mask in checking.
- **Pad-Via Spacing**
 "Via Spacing to Solder Mask": Checking clearance between vias and solder mask.
 "Except Same Net": Exclude checking for via connected to certain nets.
 "Except for Pad on Via(Only Same Net)": Change to check depending on VIA type.

- "Except for SMD Pins, not having Metal Mask"**: Exclude SMD pin, not having metal mask in checking.
- **Drill Size-Hole Distance**

"Except for Overlapped Hole(Distance 0.0)": In case of clearance checking with PCB outline, exclude hole overlapped with PCB outline.

"Except for SMD Pins, not having Metal Mask": Exclude SMD pin, not having metal mask in checking.
- **Component-Component Spacing2**

Add measure base type option to check overlapping pad and solder resistor.

"Except for SMD Pins, not having Metal Mask": Exclude SMD pin, not having metal mask in checking.

"Except for components having different Property Value": Exclude components having different property value.
- **Misc-TCP Dummy Pad**

"Dummy-Pad/TCP-Pad Ratio": Definition for dummy pad based on TCP-Pad. The dummy pad size between MAX/MIN will be valid.

"TCP Pad to Dummy Pad Clearance": Check clearance between TCP-Pad and Dummy-Pad.
- **Test Point**

"TP on Copper": Add option to check test points on certain on copper connected to certain net.

"PTH as TP": Add option to add plated through hole into test point.
- **Pattern-Pad to Net**

"Fill Distance - Same Net": Add option to check pin and copper-fill for same net.
- **Pad-Thermal Pad**

"Expand Check Area": Add option to find solder mask on bottom layer using thermal pad expansion.
- **Component-U Name Overlap**

"Except Component": Exclude certain components in checking.
- **Board-Fiducial Mark**

"Clearance from Board Corner": Add feature to find fiducial-mark from board corner with delta-X and delta-Y values.
- **Pad-Multi Pattern Connect**

"Number of Connected Pattern": Add to check the number of pattern connected to pads.
- **Component-BGA**

"Clearance Checking": For components which are having matching number of pins, check sub-item.

"Component Spacing on Reverse": Option to check components on the other BGA placed side.
- **Misc-Text Existence**

"Existence of Design File Name": Add feature, "Delimiter" to accept multiple file name.

- **Component-First Pin Mark**

"Overlapping with Objects": Add object overlapping checking for Silk, Via Hole, DIP Pad Hole and Figure Hole.

- **Board-Routing Slit**

"Reverse Side Clearance": In case of components registered in "**Comp Group**" column are placed on top layer, check will be done with the value in "**Clearance**" column. If those components are placed on bottom layer, check will be done with the value in "**Reverse Side Clearance**" column. If "**Reverse Side Clearance**" value is not defined, DFM tool use the value in "**Clearance**" column.

"Recognize as Top Side": Add option to recognize certain components' placed layer as top.

- **Pattern-Connected Pad**

Add pattern width information into result.

- **Component-Center Mark**

Add excluding option for component group or components, having thermal pad.

- **Component-Clinch Spacing**

"Except Same Net": Excluding option for clinch connected to same net in "**Clinch to Net**" item.



- **Input Setting**

Add change file name to be sorted in component group DB editing window.

Add "Sort" button in net class DB and net class setup.

Add feature to setup net-class with AND/OR condition in property column of "Edit Net Class".

- **Excel Export**

Enhance UI in "**User Defined Format Setting**".

Change DFE result MS/Excel report to contain "**Confirm Net Group**".

Add feature to attach "**ECAD Link**" command into MS/Excel.

Add feature to report the number of counting and the number of net groups for each checking item.

- **High Speed-AGND/Power Net Clearance**

"Full Path": For selected power/analog ground net, users can use nets, connected via passive devices.

- **High Speed-Approach Prohibition Via**

"Except Same Net Group": Exclude checking for nets belong to same net group.

- **High Speed-Complete Shield**

Add option, "**Except For Short Pattern**":

Certain net length will be except from target net which is shorter than given length.

High Speed-Copper Cross Over Detect

"Allowable Detour Length": If signal net passes through fill-cut on ground net copper and bypassing fill-cut length is shorter than given value, pin area will be excluded in checking.

- **Differential Pair Nets-Differential Pair Net**

"Nut Shape Anti-Pad": Checking option for differential pair net whether anti-pad in via's inner layer is nut shape or not.

"Except Layer": Excluding option for specified layer.

Add check option between **"Distance"** and **"TP Distance"**.

"Use Edge-to-Edge Segment Distance": Segment distance will be done based on "Edge to Edge".

"Except Serpentine": Exclude serpentine segments in clearance checking between two differential pair nets.

- **Differential Pair Nets-Differential Pair Net2**

"Tolerance(%)": In checking item, "Net Clearance" and "Shield Distance", if there is given tolerance value, adopt smaller value as given tolerance value.

"Pin, Via Escape": Exclude range for given distance value in checking.

"Use Width Value(W)": In clearance checking, "Net Distance" and "Shield Distance", relative values will be used regard to pattern width.

"Show Segment Error": Add option to show results in the unit of segments.

- **Net-Bus**

"Length/Combination Length": Add length checking option for nets having two or more loads.

- **Net-Component to Net Clearance**

Add option to check object locating layer and upper/below 1 or 2 layers.

"Component2": If target components and given net group (component2) components are overlapped, exclude nets connected to component2 will be excluded in checking.

- **Net-Confirm Net Group**

"Data Existence on Layer": Check existence of data on certain layer.

- **Net-GND Isolation**

Add checking option that patterns connected between specified pin are isolated or not.

Add feature whether isolated patterns are connected to main ground net or not.

- **Net-Limit Approach Net**

"Composite Net": Add new option to check for total route path for power line.

Add layer option to check for layers.

"Pin Escape" / "Via Escape": Add excluding option for distance from pin or via.

- **Net-Net Group Shield**

Add layer selection and **"Except Short Pattern"** in **"Vertical Shield Check"** item.

Add option to exclude pin area in checking.

Add option to exclude certain component area in checking.

- **Net-Net Isolation**

Add checking isolation between two components.

- **Net-Net to Net**

Add copy/paste function in inputting distance value.

- **Net-Pattern Sharp Angle**

Add inequality sign option: For selected pattern, check the size of angle is smaller than given value or not.

"**Pin Escape**": Add excluding option for certain area from pin in checking.

- **Net-Serpentine**

New function for differential pair's length matching checks serpentine routing pattern's shape and balancing.

- **Net-Star Connection**

New function checks whether IC pin's pattern are connected independently from capacitor in case of passive components are connected from multiple IC pin.

"Composite Net" : Add new check function for total routing path.

- **Net-Stub**

Change to exclude dummy trace in polygon/copper in "Stub" item.

- "Composite Net" : Add new check function for total routing path.

- **Net-Via Quantity**

Check whether the number of vias of specified net's connected component pad is much many than given numbers. Results' net name is component connected net name.

Add option, "**Connected Comp**" and "**Pad on Via Qty(Min)**".

- **Power-Connected Via**

"Composite Net" : Add new check function for total routing path.

"**Pad On Via**": Add new option check via existence on pad.

- **Power-Net to Net**

Check detour length in case of hole between two patterns.

- **Power-Parallel Jumper**

New checking item to check parallelism in case of jumper used for certain high voltage AC input routing pattern.

- **Power-Power Width**

"**Inner Only**": Check only for inner layer.

"**Exclude Fill Area**": Exclude fill-area in checking.

"Composite Net" : Add new check function for total routing path.

- **Power-Stability Copper**

"**PTH Via Pass**": Option to pass for copper having plated-through-hole via.

- **Power-Supplementary Pattern**

To check segments around certain components, add option to specify starting components

and ending components.

- **Power – Via Option**

Add new checking option to check total via-hole's dimension in case of specified net routing patterns are changed its layers.

- **Component-Approach Pair Comp**

Change checking method so that PolIEx DFE check only close components in clearance checking between components.

Add new feature to check only for comp1 group and comp2 group.

- **Component-Component Direction**

Add new checking for passive components' placement angle.

- **Component-Approach pair Comp/Keep of pair Comp**

Add function to use COC/SILK base in clearance checking between components.

- **Component-Component Shield**

Add checking options, "**Fill – Cut / Inductance Check**" in type-option.

"**Vertical Check**": The above and below layer(s) of placed components are verified.

- **Component-NC Pin**

Pad on Via Exist: Can detect via on NC Pin.

PolIEx DFA

- Modify checking standard in DFA: Users can set decimal point value to define pass/fail.

- **Component-BGA Underfill**

Check whether BGA under-fill spaces are enough reserved or not.

- **Component-Component to Solder**

Add checking the clearance between solder masks of components.

- **Board-PCB Outline Spacing**

Add new checking the clearance between PCB outline and components.

- **Board-Board Outline Spacing**

Add new checking the clearance between panel board's outline and components.

PolIEx Metal Mask Manager

- **Environment**

Add new feature to set environment for "**Metal Mask Manger**".

Add new function to save display option and BOM checker setting option in "**BOM Checker**" environment.

- **Metal Mask DB Setting**

"Get Metal Mask Shape from Gerber": Add new feature to register GERBER's shape into standard metal mask.

- **Metal Mask Type**

Add new several shapes for metal mask shape.

- **Metal Mask BOM Checker**

"Standard Metal Mask DB": Check design's metal mask based on standard metal mask data.

Add feature to show part names in comparing result list.

Modify to save selected option for environment setting in "Metal Mask Manager".

"Line Width on Gerber": Exclude line shape having specified width in checking.

- **Standard of Mask Design**

Add button, **"Standard Mask Edit"** in **"Check DB"** window.

Add feature to automatically create components DB (database) from layout design for components which are not registered in MMDB. Also, automatic registering standard metal mask shape from layout design is possible.

- **Validation Check Report**

Add "Total Fail Count(Component)" item in report so that users can check number of fails for components.