

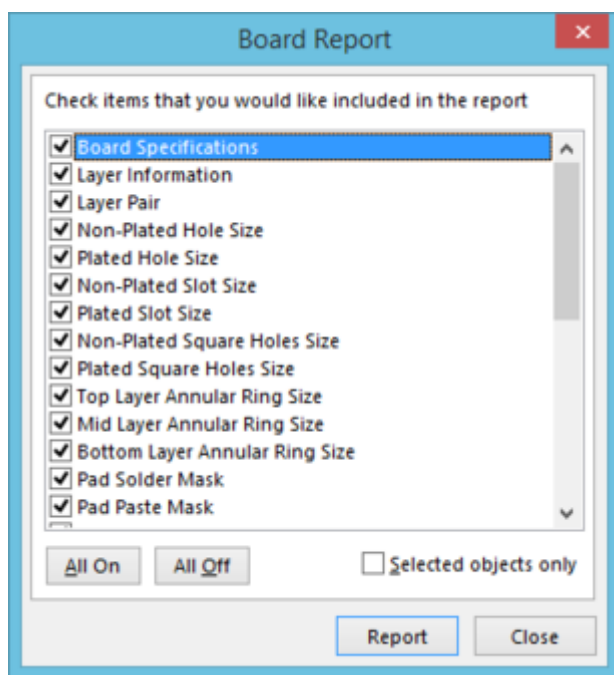
Board Report

Modified by Phil Loughhead on Nov 2, 2014

Other Related Resources

[PCB Information \(Dialog\)](#)

Parent page: [PCB Dialogs](#)




The *Board Report* dialog.

Summary

This dialog allows the designer to specify the content to be included when generating a detailed report for the board.

Access

The dialog is accessed from the PCB Editor, by clicking the **Report** button in the *PCB Information* dialog. The latter is accessed by clicking **Outputs | Reports |  Board Information**, from the main menu.

Options/Controls

- **Items to include** - this main region of the dialog lists all of the items that can be included in the report. To include information on a particular attribute of the board, simply ensure its associated check box is ticked. Supported items for inclusion are:

- **Board Specifications** - general information about the board size, and the number of components on the board.
- **Layer Information** - how many primitives (arcs, pads, vias, tracks, texts, fills, regions, component bodies) are on each used layer of the board, along with total usage for each primitive type.
- **Layer Pair** - the defined drill layer pairs, along with a breakdown of the number of vias starting and stopping between those pairs.
- **Non-Plated Hole Size** - the number of pads and vias for each hole size of this type.
- **Plated Hole Size** - the number of pads and vias for each hole size of this type.
- **Non-Plated Slot Size** - the number of pads for each slot size of this type.
- **Plated Slot Size** - the number of pads for each slot size of this type.
- **Non-Plated Square Holes Size** - the number of pads for each hole size of this type.
- **Plated Square Holes Size** - the number of pads for each hole size of this type.
- **Top Layer Annular Ring Size** - the number of objects (pads and vias) for each annular ring size on the top layer.
- **Mid Layer Annular Ring Size** - the number of objects (pads and vias) for each annular ring size on a mid layer.
- **Bottom Layer Annular Ring Size** - the number of objects (pads and vias) for each annular ring size on the bottom layer.
- **Pad Solder Mask** - the number of pads for each specified, and unique, solder mask expansion value.
- **Pad Paste Mask** - the number of pads for each specified, and unique, paste mask expansion value.
- **Pad Pwr/Gnd Expansion** - the number of pads associated with unique **Clearance** values specified in defined power plane clearance rules.
- **Pad Relief Conductor Width** - the number of pads associated with unique **Conductor Width** values specified in defined power plane connect style rules, whose **Connect Style** is set to Relief Connect.
- **Pad Relief Air Gap** - the number of pads associated with unique **Air-Gap** values specified in defined power plane connect style rules, whose **Connect Style** is set to Relief Connect.
- **Pad Relief Entries** - the number of pads associated with unique **Conductors** values specified in defined power plane connect style rules, whose **Connect Style** is set to Relief Connect.
- **Via Solder Mask** - the number of vias for each specified, and unique, solder mask expansion value.
- **Via Pwr/Gnd Expansion** - the number of pads associated with unique **Clearance** values specified in defined power plane clearance rules.
- **Track Width** - the number of objects for each unique track width used in the design.
- **Arc Line Width** - the number of objects for each unique arc line width used in the design.
- **Arc Radius** - the number of objects for each unique arc radius used in the design.
- **Arc Degrees** - the number of objects for each unique arc angle used in the design.
- **Text Height** - the number of objects for each unique text height used in the design.
- **Text Width** - the number of objects for each unique text width used in the design.
- **Polygon Clearance**
- **Net Track Width**
- **Net Via Size**
- **Routing Information** - information on routing completion (as a percentage), along with a breakdown of the total number of connections, how many have been routed, and how many remain.
- **All On** - click this button to quickly include all items in the report (checking their respective

check boxes).

- **All Off** - click this button to quickly exclude all items from the report (unchecking their respective check boxes).
- **Selected objects only** - enable this option to have the report generate information for each of the include items, but only with respect to design objects currently selected in the design workspace.
- **Report** - with all items enabled for inclusion into the report as required, click this button to generate the report. The report itself is generated in HTML format (Board Information - <PCBDocumentName>.html).

The report is generated into the directory specified by the **Output Path** field, on the **Options** tab of the *Options for PCB Project* dialog. In the *Projects* panel, it will appear as an entry in the project's Generated\Documents sub-folder.

Source URL: [http://documentation.circuitmaker.com/display/CMAK/PCB_Dlg-BoardReport\(\(Board+Report\)\)_CM](http://documentation.circuitmaker.com/display/CMAK/PCB_Dlg-BoardReport((Board+Report))_CM)