Schematic Generation Module™

Create Full Function Schematics from Netlist and BOM

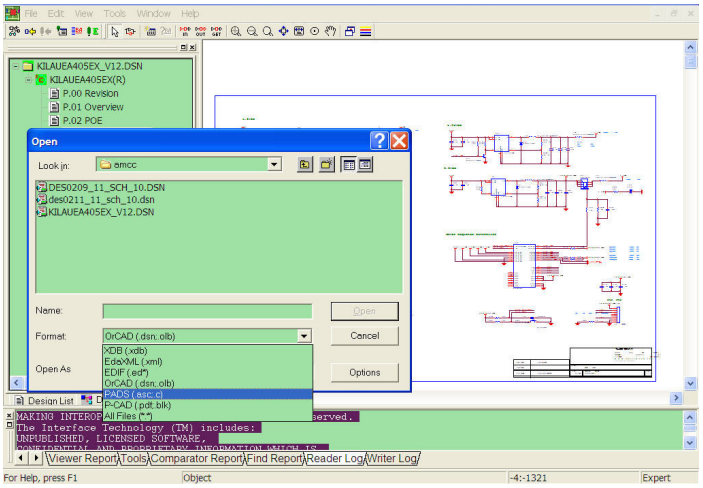
Overview

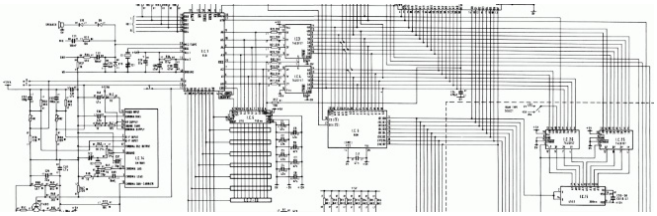
Schematic Generation Module helps electrical engineers, PCB designers and test engineers to visualize, explore and debug netlist and schematic data as well as publish it. Schematic Generation Module supports multiple netlist and schematic formats including OrCAD, PADS, Powerlogic, DxDesigner /Viewdraw, PCAD, etc. and standard netlist formats such as IPC-D-356. Schematic Generation Module generates intelligent schematics from standard PCB netlists and presents them within its own visual environment.

In addition, Schematic Generation Module exports schematic designs to searchable PDF schematics to be navigated using PDF Viewers. Schematic Generation Module produces PDF files that can keep hierarchy, net attributes, and component attributes of the original design, and at the same time, allow net tracing on or between the pages.

Benefits

- View schematics from different vendors in one tool
- Publish an entire schematic design to a single intelligent PDF file that can be opened in free PDF Viewers
- Password protect PDFs
- Output hierarchy, net and Component attributes to PDF
- Allow net tracing in PDF files
- Publish color and B/W PDF schematics
- Generate intelligent PCB-style schematics from PCB netlists
- Generate regular schematics from IPC-D-356 and EDIF netlists
- Save generated schematics to PDFs
- Compare PCB netlists, IPC-D-356 netlists, EDIF netlists, schematic netlists, schematics
- Use scripting for batch-mode PDF production





Schematic Design Viewing

Schematic Generation Module can be used as a vendor independent Viewer, and supports viewing of PADS Powerlogic, DxDesigner / Viewdraw, OrCAD, PCAD PDIF, Altium Designer, EAGLE, EDIF200, EDIF300, EdaXML and other formats.

The Viewer environment is very intuitive and simple to work with. The Design Tree Browser and Search engine make finding any object simple. Using Schematic Generation Module, you can view design hierarchy, occurrence data, or can view design schematic pages layer by layer. You can navigate to any net or component in your schematic with a few mouse clicks. A detailed report can be generated for any object in a netlist or schematic. The Viewer environment supports simple mark-ups that later can be saved in a PDF file.

Creating Schematics from PCB and IPC356 Netlists

Schematic Generation Module supports multiple PCB netlist formats, IPC-D-356 netlist and EDIF netlists as well. A user opens a PCB Netlist in the tool and can use Netlist Browser to navigate through the data with reports available for every object.

Schematic generation is intelligent and configurable. The built-in algorithm recognizes patterns in the PCB netlist and generates a schematic in such a way that it is simple to navigate for a PCB designer or a test engineer. The colors, page size, title block, net/bus recognition, and other parameters, can be selected for each generation run in accordance with user preferences. Symbols can be built using different vendor-styles in order for generated schematics to look familiar to the user of a particular vendor system.

A user can navigate through the generated schematic within the Schematic Generation Module environment or output that schematic to PDF with net tracing capability preserved.

Schematic and Netlist Tools

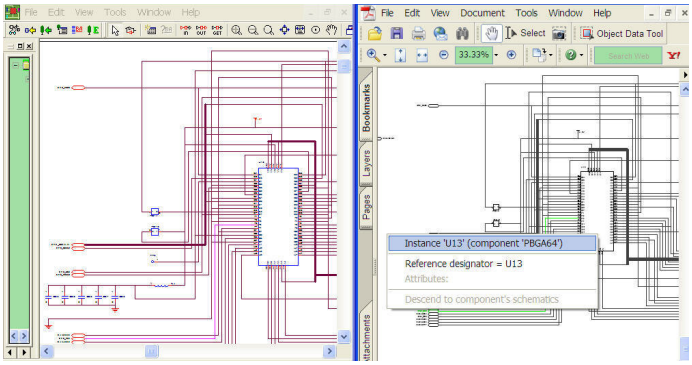
Schematic Generation Module's Netlist Comparator is a useful tool. Users can compare two PCB netlists, two IPC-D-356 netlists or even schematics from one system with PCB netlists from another. This is especially useful when users reside in heterogeneous environments and are working with schematic capture and layout tools from different vendors. The flexible configuration options help users to control the depth of comparison processes. Another utility is a customizable BOM (Bill of Materials).

Creating PDF Schematics

Schematic Generation Module allows you to produce intelligent PDF schematics from multiple supported input formats. Your entire schematic design is saved in a single PDF file that can be password-protected and encrypted to guarantee security of your IP. Components and nets can easily be found in this intelligent PDF, because any text can be searched. Generated PDF files carry up to 10 graphical layers. With the help of the Adobe Reader's "Layers" feature, users can view components, nets, net names, pin numbers, texts, etc., independently.
You can select the level of intelligence you prefer to obtain in the created PDF file. An additional Java-script Toolbar can be included in the PDF file to help trace net segments, or to trace an entire net flow from page to page.
Having component attributes, net attributes, net tracing and net highlighting allow you to browse and understand schematics the same way it would be in a special capture system. The PDFs you produce can be viewed by anyone who has a PDF Viewer. Such functionality helps in sharing design data or in archiving, because once a PDF file is produced it cannot be changed.

Scripting

Schematic Generation Module can be run from a command line. The Enterprise configuration of the tool has licensing for TCL scripting support and batch mode processing. Having scripting available allows users to publish schematics to PDFs from a batch process, and therefore can be used for archiving or documentation purposes.



SCHEMATIC GENERATION MODULE FEATURES

Schematic Readers
- OrCAD
- PADS Powerlogic
- DxDesigner / Viewdraw
- PCAD PDIF
- Altium Designer
- EAGLE
- EDIF200 / EDIF300
- EDAXML

PDF Publishing
- Password protected and encrypted PDFs
- Color and B/W PDFs
- Intelligent PDFs with/without net tracing
- Component attributes, net attributes and layers support
- Java-script Toolbar for net tracing

Schematic Generation
- Pattern recognition and PCB-style look of generated schematics
- Schematic generation from IPC-D-356, PCB and EDIF netlists

PCB and Other Netlists Readers
- IPC-D-356, IPC-D-356A
- Allegro third party Netlist
- Telesis
- PADS PCB
- PCAD PCB
- Expedition KYN
- EDIF netlist and other netlists

Utilities
- Netlist Comparator
- BOM

Scripting
- Command (batch) mode support
- TCL scripting support

Operating Systems Support
Windows 2000, XP, Vista, 7, 8.x, 10

Interfaces to Adobe
Generated PDF files were tested on Adobe Reader 7.x, 8.x, 9.x, 10.x

(All specifications and designs subject to change without notice.)

rev. 19091620

ScanCAD INTERNATIONAL
Simplifying Complex Technology

26437 Conifer Road, Conifer, Colorado 80433 USA
T: +1 303.697.8888 | F: +1 303.697.8580
info@scancad.com | www.scancad.com