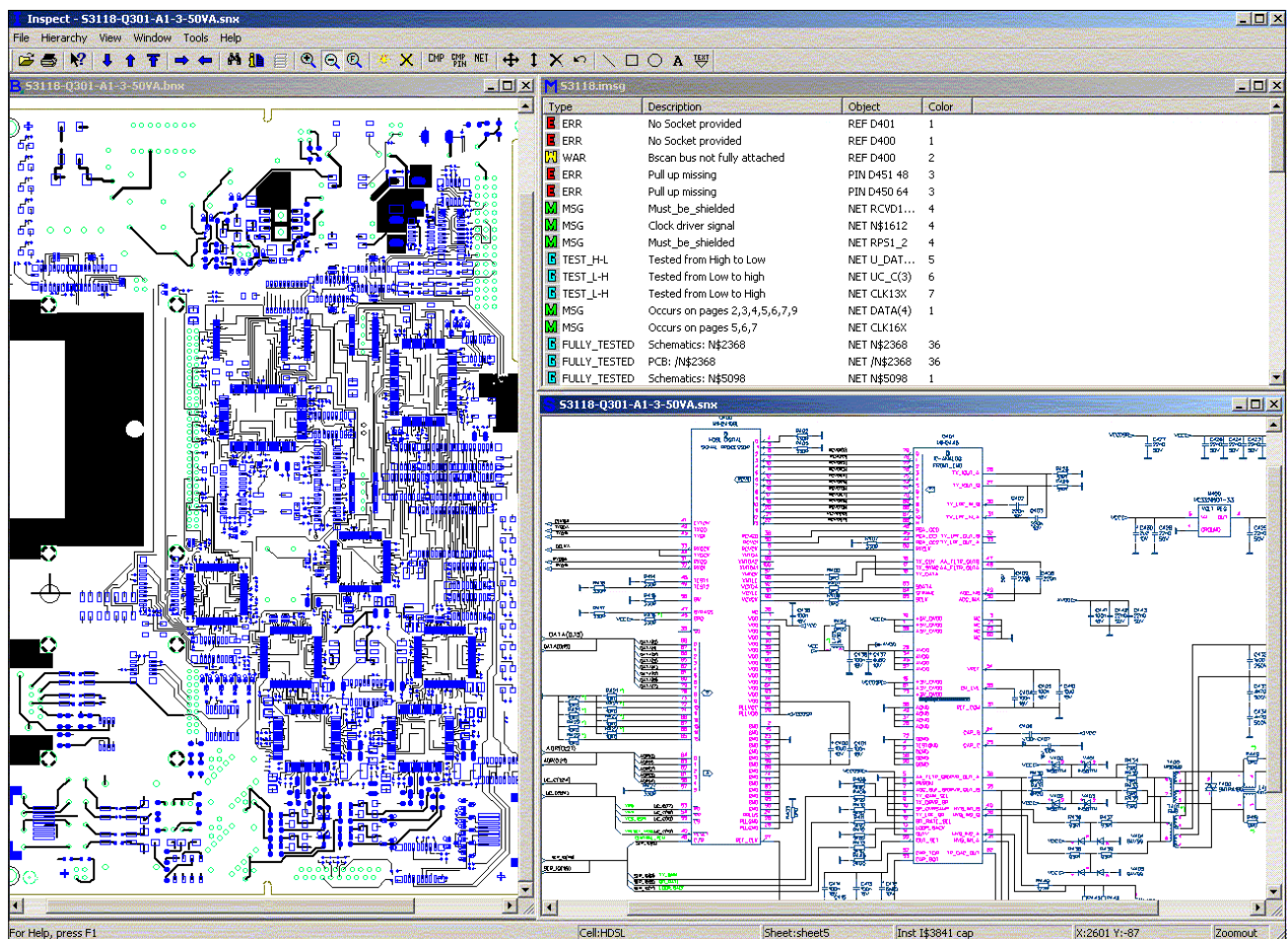


Inspect

Inspect is an analysis tool to view and analyze complete schematic and PCB databases of any Engineering Design Automation System. All EDA objects like unnamed nets, power pins, and all component attributes can be found and highlighted. Inspect cross highlights nets, pins and references between two designs which may be schematic, schematic and PCB or two PCB designs.



Features

- Inspect complete schematic and PCB databases including library elements.
- Find and highlight any property, instance, net or pin in schematics and PCB layout.
- Browse through hierarchy and multi-page designs.
- Set up of multi-highlight and layer colors.
- Find and highlight with double-click on items of message file with results from manufacturing. The message file is plain ASCII Text which may come from a test machine.
- Plot schematics and pcb layers.
- Display schematic and pcb design hierarchy.
- Inspect reads compact intermediate ASCII files with design data in the SNX and BNX formats.

Inspect

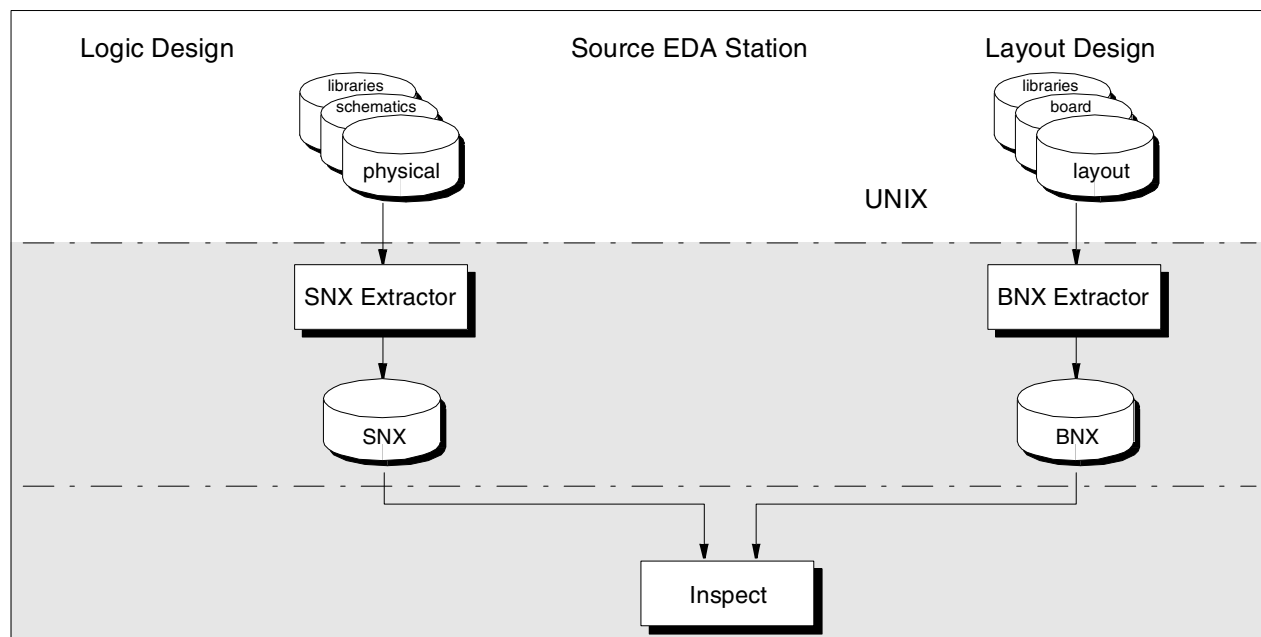
Application Areas

- ✓ **Test Engineering** Put error messages into file from which Inspect highlights defective components, faulty nets, shorted or open pins. Cross probe Board layout and schematics.
- ✓ **Service** Execute Inspect from Internet page and browse circuit schematics and pcb layout.
- ✓ **Manufacturing and Quality Control** View board layout and circuit schematics on the PC with Windows; Exchange data marked-up in Word or Power Point via Outlook with external vendors.

Benefits

- Search and highlight any object of the EDA design such as property, net, pin, or component in schematics and PCB data.
- Cross-highlight nets, components, and pins between schematics and PCB.
- Find and highlight nets, components, and component pins from ASCII files generated by production test machines.
- Redline text, circles, and arbitrary geometries with user selectable hue and color.
- Forward search results into Excel sheets.
- Execute Inspect with double-click on the design file in the Eigner PLM system.

System Architecture



- Extract Schematic data with the SNX extractor into a compact neutral ASCII file format which includes all design data.
- Extract PCB Layout data with the BNX extractor into a compact neutral ASCII file format with all data.

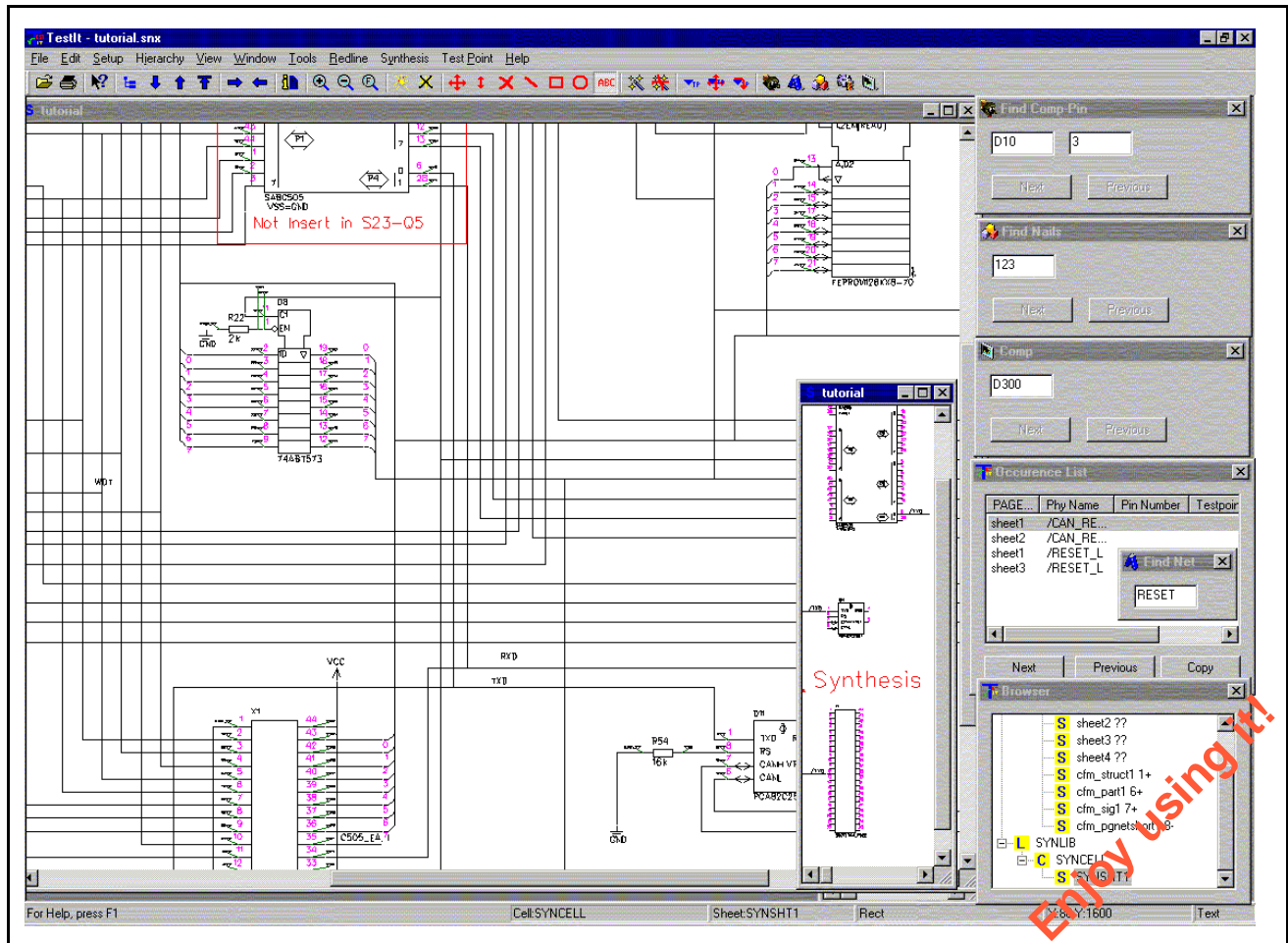
Supported Systems

Schematic and PCB extractors are available for Mentor Graphics, Zuken System Designer and PCB, Innoveda, Cadence Concept_hdl, Allegro, Zuken Cadstar, Orcad and many other EDA Systems.

Test_it!

Test_it! places all schematic data of an Engineering Design Automation system at your finger tips for quick test analysis, and repair.

All EDA Schematic objects like nets (named or unnamed) component references and pins are found and highlighted. Powerful functions aid in tracking errors and in communicating specifics analysis results.



Features

- Analyse complete schematic database.
- Find and highlight any component, property, pin or net.
- Browse through hierarchy and multi-page designs.
- Click on pin to create synthesis sheet with all components connected to this pin.
- Redline any object and add text.
- Find and highlight from ASCII list which comes from a test machine.
- Browse design for any object and export resulting table in Excell sheet.
- Backannotate test needles from ASCII file into schematics.
- Edit and place test needles in schematics.
- Optionally execute Test_it! from within Fabmaster® with cross-probing.

Test_it!

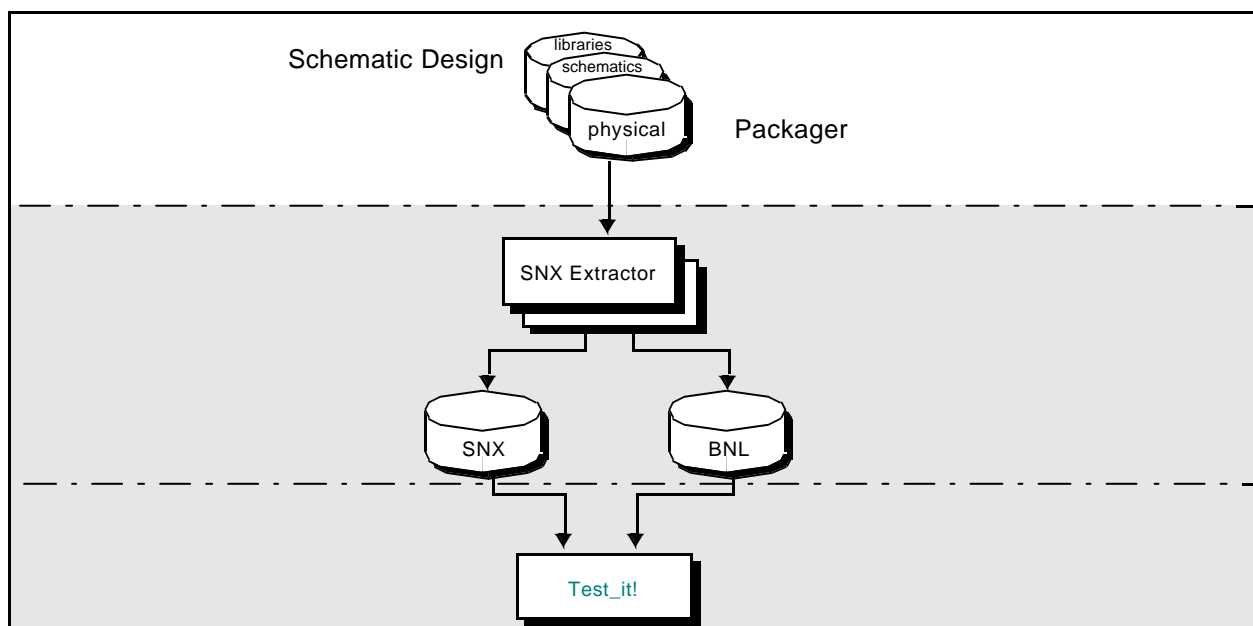
Benefits

- Search and highlight any object of the EDA design such as property, net, pin, or component in schematics and PCB data.
- Cross-highlight nets, components, and pins between schematics.
- Find and highlight nets, components, and component pins from ASCII files generated by production machines.
- Redline text, circles, and arbitrary geometries with user selectable hue and color.
- Forward search results into Excel sheets.
- Execute [Test_it!](#) with double-click on the design file in the Eigner PLM system.

Applications

- **Test and Repair** Put error messages into file to highlight defective components, faulty nets, shorted or open pins. Analyse net pairs. Find and highlight component-pins. Find and edit test needles. Follow nets through the design.
- **Service** Execute Test_it from Internet page and analyze in circuit schematics.
- **Manufacturing and Quality Control** Redline schematic parts. Export schematic display into Microsoft Windows[®]. Export item lists into Excell. Analyze design from occurrence list.

System Architecture



Extract Schematic and Packager data with the EDA extractor into a compact neutral ASCII file formats which includes all data.

Supported Systems

Schematic and PCB extractors are available for Mentor Graphics, Visula SCM, Cadence Concept_HDL, Veri-best Capture, Innoveda Viewdraw, Pads, Orcad, Cadstar, Zuken System Designer, and others.