
Co-Design

1	Introduction	7
1.1	Overview.....	7
1.2	Modules	7
2	View-based Library Distribution	8
2.1	Check-in and Check-out Lib	8
2.2	Structure of a Library Components.....	8
2.3	Check-in Lib and Check-out Lib.....	9
3	Distribution of Design Data	10
3.1	Overview.....	10
3.2	Design Sharing	10
3.3	ODM	10
4	Web-based concurrent Design Review	14
4.1	Overview.....	14
4.2	Concurrent Design Review	14
5	Glossary of Terms.....	21
6	Conventions.....	22

Version: 1.0	07/10/2006	Co-Design
Language: English	Rau	Product Outline
Contents		P: 3/22

Figures

Figure 1: Data Model in the EDA Library	8
Figure 2: The Data Model of the electronic Part in PLM	8
Figure 3: Check-in Lib and Check-out Lib	9
Figure 4: Block-based Design	10
Figure 5: Design Workflow with ODM	11
Figure 6: Design Engineer: Processes ODM Data	12
Figure 7: Compare Product Structures of Schematics with PCB	13
Figure 8: Review a Board Design	14
Figure 9: Compare Product Structure	15
Figure 10: View Design: Design Structure	16
Figure 11: View Design: Cross Probing	17
Figure 12: View Design: Find and Highlight	17
Figure 13: View Design: File -> Setup -> Edit -> Attributes Dialog	18
Figure 14: View Design: Backannotate Design Changes	18
Figure 15: Example of a Backannotation File	19
Figure 16: View Design: the Redline Dialog	19
Figure 17: View Design: Dimension and Adjustment of Display	20

Tables

Table 1: Functions of Check-in and Check-out Lib	9
Table 2: Example of configured ODM Data	12

1 Introduction

1.1 Overview

Design work may be concurrent between groups in the same company or shared with ODMs. The Co-Design package supports this process with

- view-based distribution of design libraries and design data,
- design module support,
- Web-based concurrent design review,
- structure check of received design data,
- merging design data of the ODM with authoring data of the design engineer.

1.2 Modules

The Co-design package consists of specifically configured modules of Integrate for

- **Check-in, Check-out,**
- Viewing,
- structural comparison.

2 View-based Library Distribution

2.1 Check-in and Check-out Lib

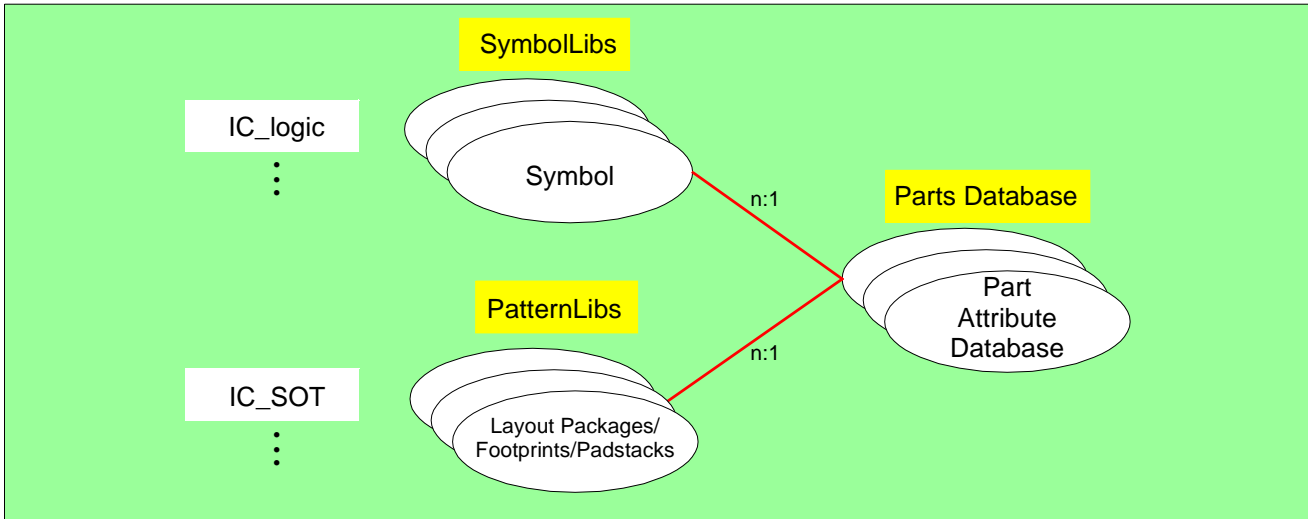


Figure 1: Data Model in the EDA Library

2.2 Structure of a Library Components

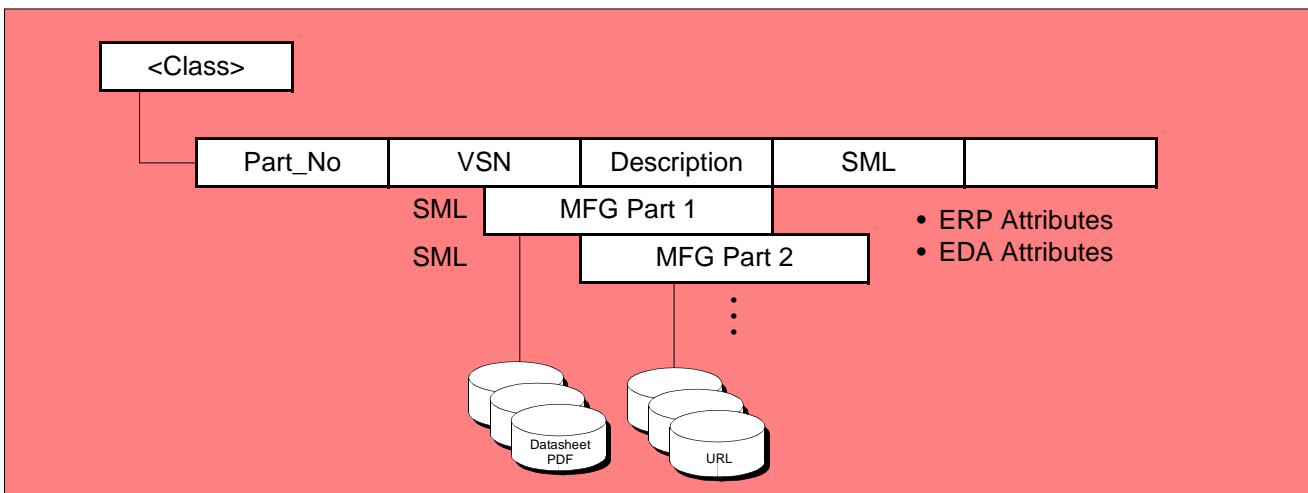


Figure 2: The Data Model of the electronic Part in PLM

2.3 Check-in Lib and Check-out Lib

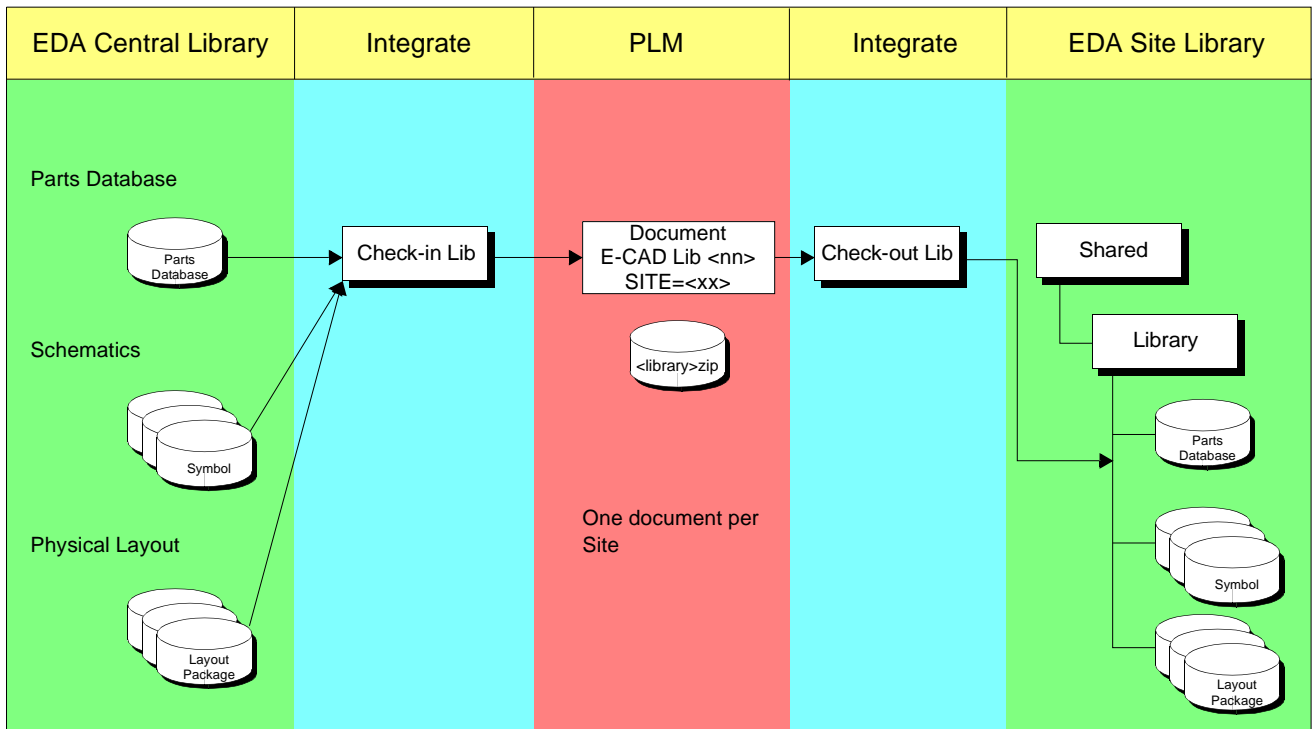


Figure 3: Check-in Lib and Check-out Lib

Module	Functions
Check-in Lib	<ul style="list-style-type: none"> Creates a document in PLM for the Site library if it does not exist. Evaluates the SITE attribute per part and builds a temporary, site-specific library. Packs the library into a zip container and checks it in.
Check-out Lib	<ul style="list-style-type: none"> Checks the library out into the site. Check-out Lib is operated locally so that the library update does not interfere with design work. Creates a backup of the existing library. Updates the library with the changed or new components.

Table 1: Functions of Check-in and Check-out Lib

The document with metadata for the library distribution is site-specific. It is submitted to a workflow. Upon release it is replicated into the sites.

3 Distribution of Design Data

3.1 Overview

Design data are distributed for

- in-house design sharing, or
- forwarding of design data to ODMs.

3.2 Design Sharing

In-house design sharing may be based on design blocks. Blocks are hierarchical blocks in schematics or functional blocks in the physical layout.

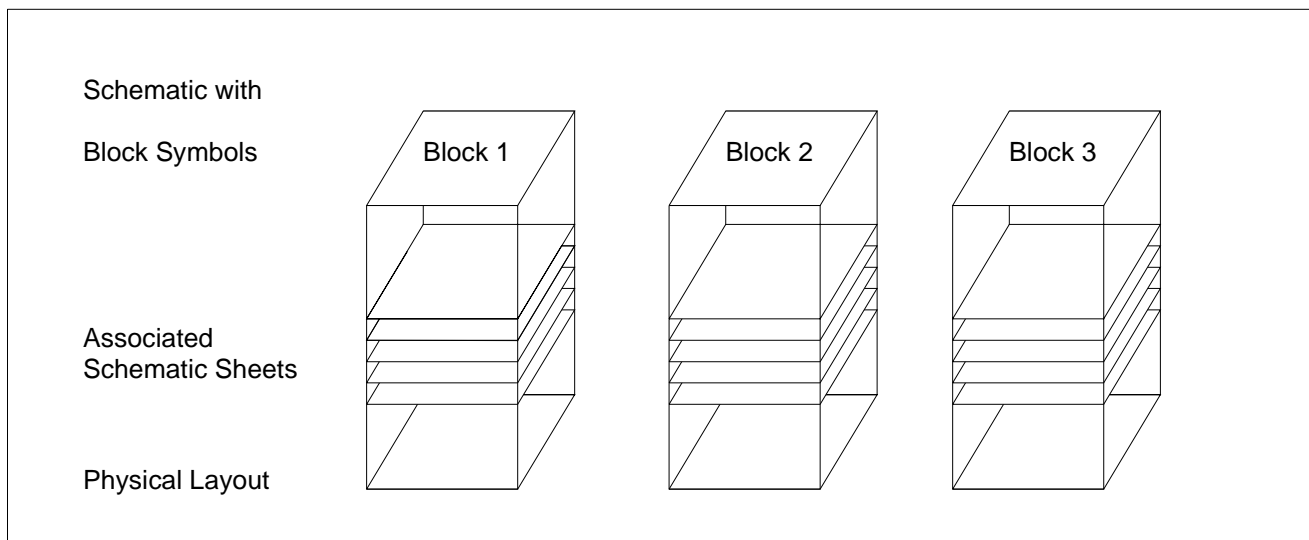


Figure 4: Block-based Design

Design blocks consist of hierarchical schematic blocks and of associated functional blocks in the physical layout with unresolved connectivity references at the block borders.

All blocks must conform to the same design rules. A block approach supports design re-use.

In PLM, in-house design sharing requires establishing of a group. This is configured with **Synchronize Metadata**.

3.3 ODM

Forwarding of design data to ODMs is based on

- an Engineering Release which causes creation and packing of the required data,
- **Check-in** of the data for the ODM.

Required data are configurable and are frequently

- **BOM** document,

- physical netlist,
- reference to the SITE library,
- requirements for the layout of the board.

These are configured for the **Check-in** process. The requirements list is based on a template spreadsheet with attributes to set metadata. The **BOM** module creates a **BOM** spreadsheet for the ODM.

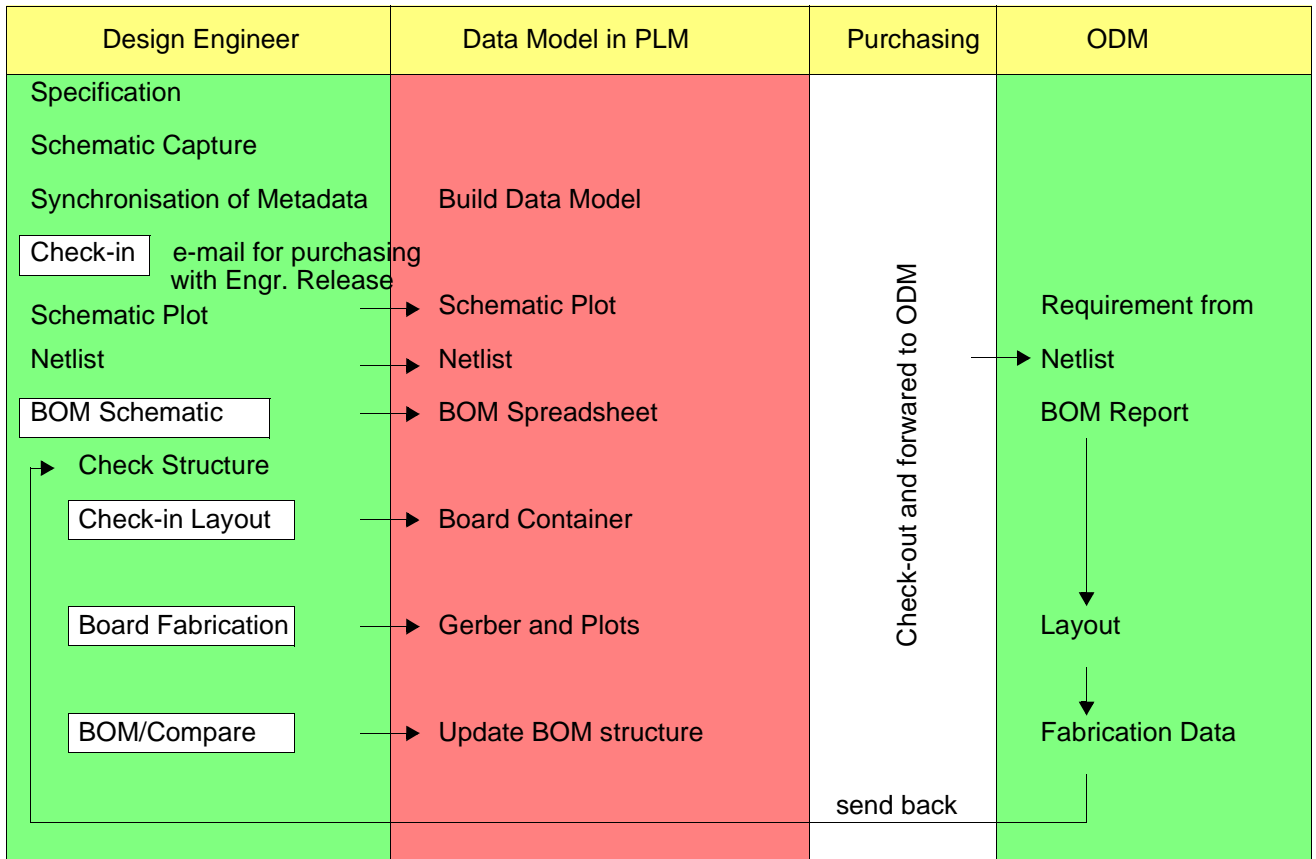


Figure 5: Design Workflow with ODM

The requirements form is a Word document adapted to the pre-requisite metadata by Check-in with Engineering Release.

The ODM then sends back to the design engineer

- A set of fabrication data as per the requirements form.
- The native authoring tool product structure file for comparison check by the design engineer.

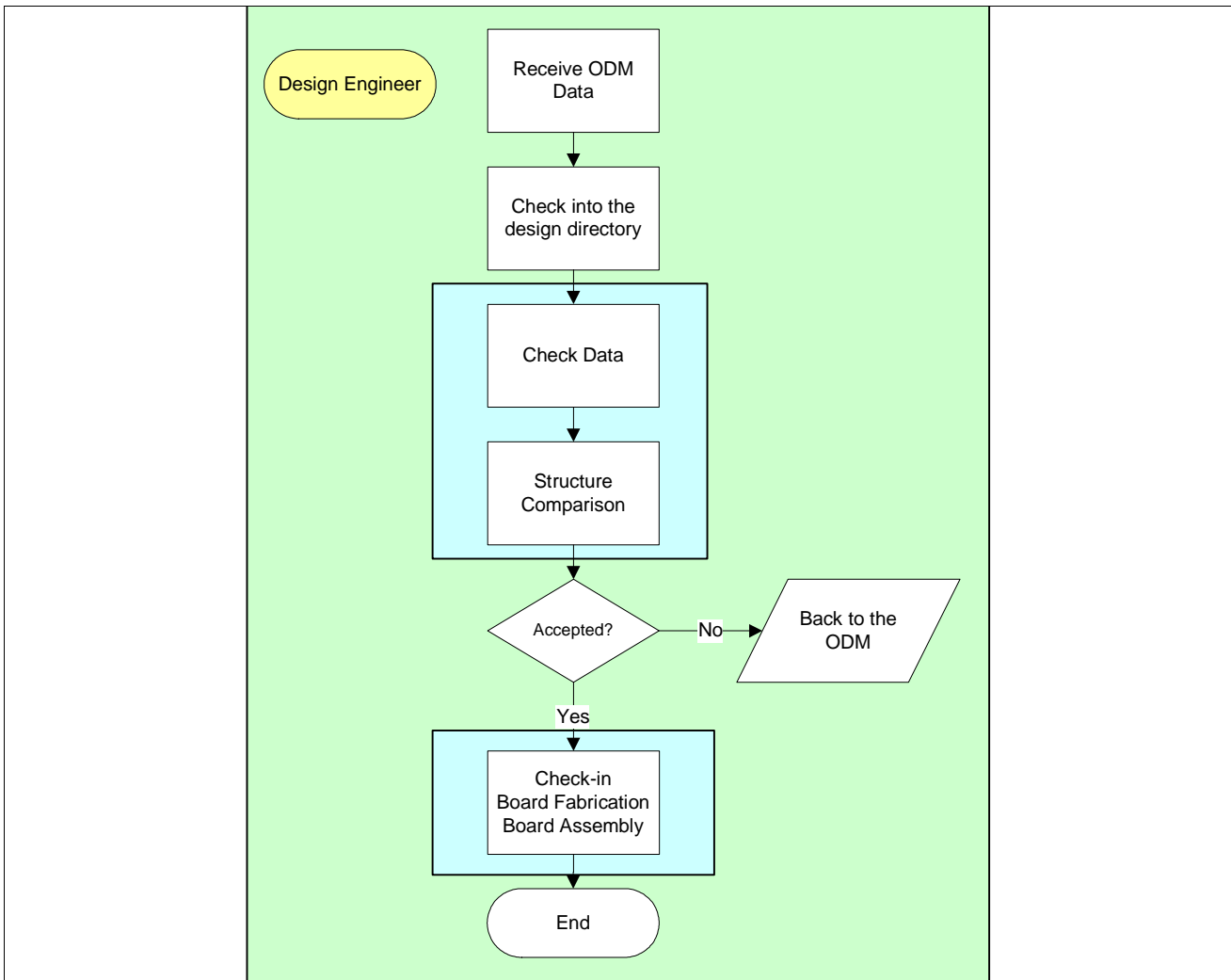


Figure 6: Design Engineer: Processes ODM Data

File Name	Comment
PCB_*_LT.ART	L ayer 1 T op, (Component Side)
PCB_*_L2.ART	L ayer 02 , (Inner Layer)
PCB_*_L3.ART	L ayer 03 , (Inner Layer)
PCB_*_L4.ART	L ayer 04 , (Inner Layer)
PCB_*_L5.ART	L ayer 05 , (Inner Layer)
PCB_*_L6.ART	L ayer 06 , (Inner Layer)
PCB_*_L7.ART	L ayer 07 , (Inner Layer)
PCB_*_LB.ART	L ayer 8 B ottom, (Solder Side, Bottom)
PCB_*_SMT.ART	S older M ask T op (Solder Resist, Top)
PCB_*_SMB.ART	S older M ask B ottom (Solder Resist, Bottom)
PCB_*_SST.ART	S ilk S creen T op (Silkscreen, Top)
PCB_*_SSB.ART	S ilk S creen B ottom (Silkscreen, Bottom)

Table 2: Example of configured ODM Data

PCB_*_SPB.ART	S older P aste B ottom (Solder Resist, Bottom)
PCB_*_GPB.ART	G lue P aste B ottom (Glue mask, Bottom)
PCB_*_SPT.ART	S older P aste T op (Solder Resist, Top)
PCB_*_DRL.<xxx>	D rill File (Drill Drawing)
PCB_*_TPT.<xxx>	T est P oints T op
PCB_*_TPB.<xxx>	T est P oints B ottom
CFP_*_Rev@_Vxx.<xxx>	Component Fitting Plan, PDF-File Assembly Board

Table 2: Example of configured ODM Data (cont'd)

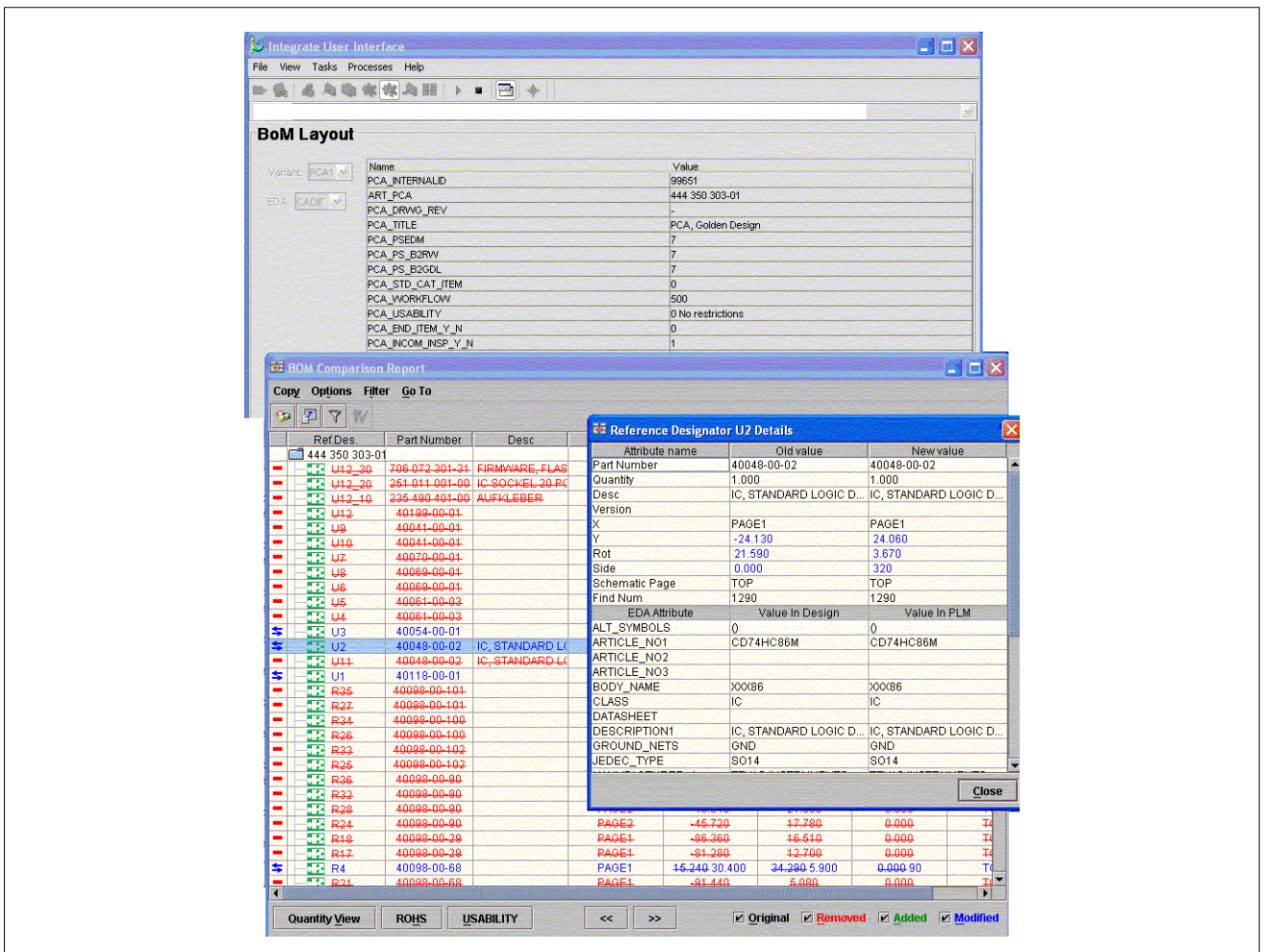


Figure 7: Compare Product Structures of Schematics with PCB

4 Web-based concurrent Design Review

4.1 Overview

Design review is based on structure analysis and comparison and on viewing from the structure or stand alone. Viewing includes redlining, mark-up, dimensioning, and backannotation of **BOM** changes and part number changes into the viewing tool and into the design.

4.2 Concurrent Design Review

The design review is started from the Web in the Review Board.

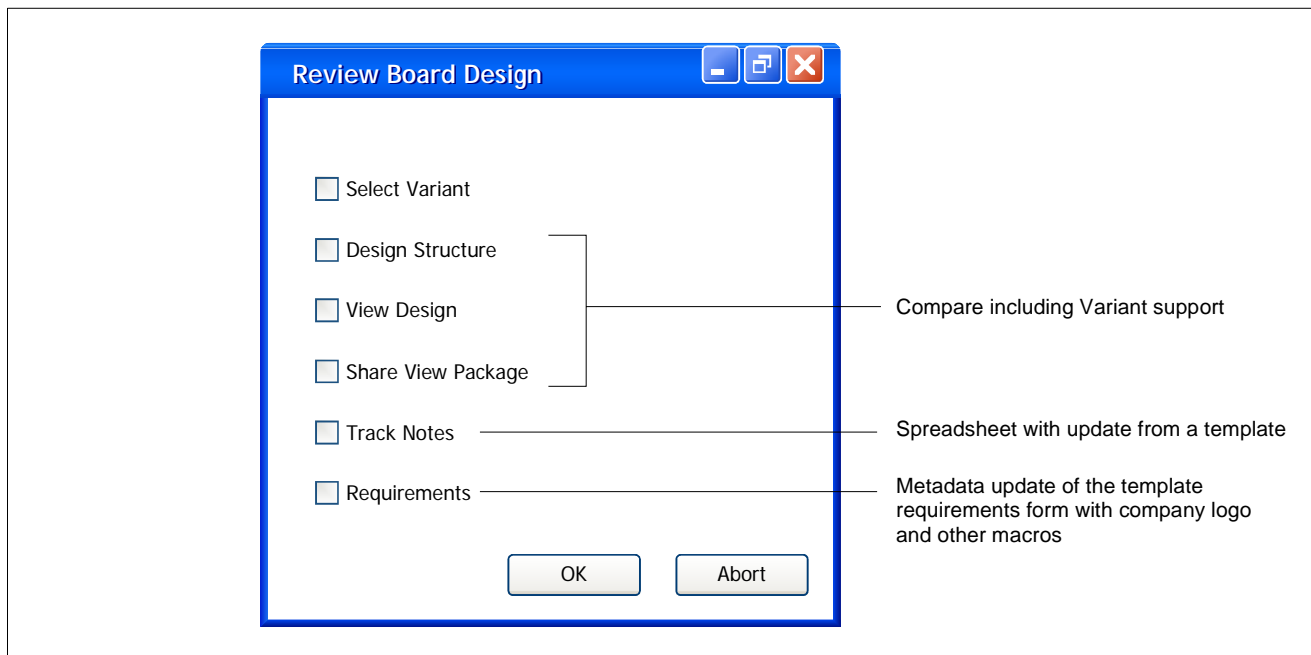


Figure 8: Review a Board Design

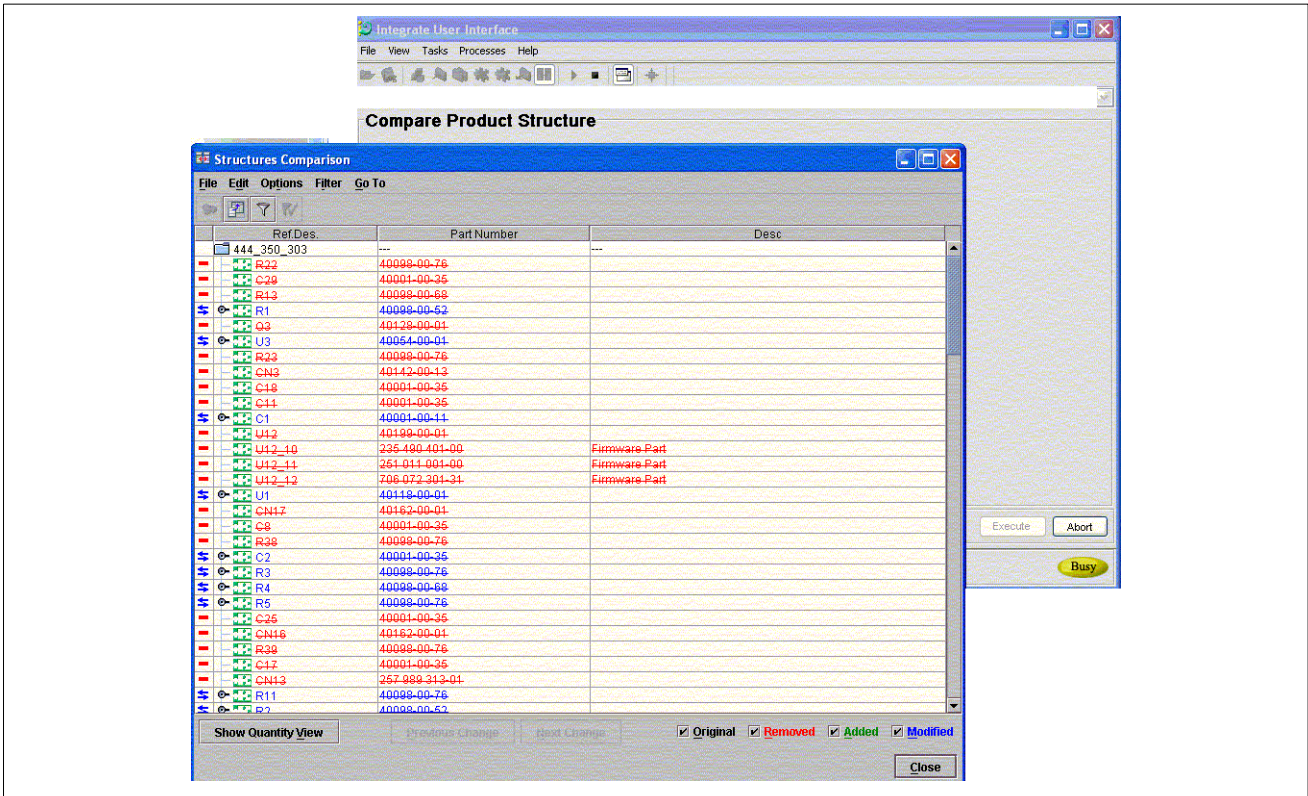


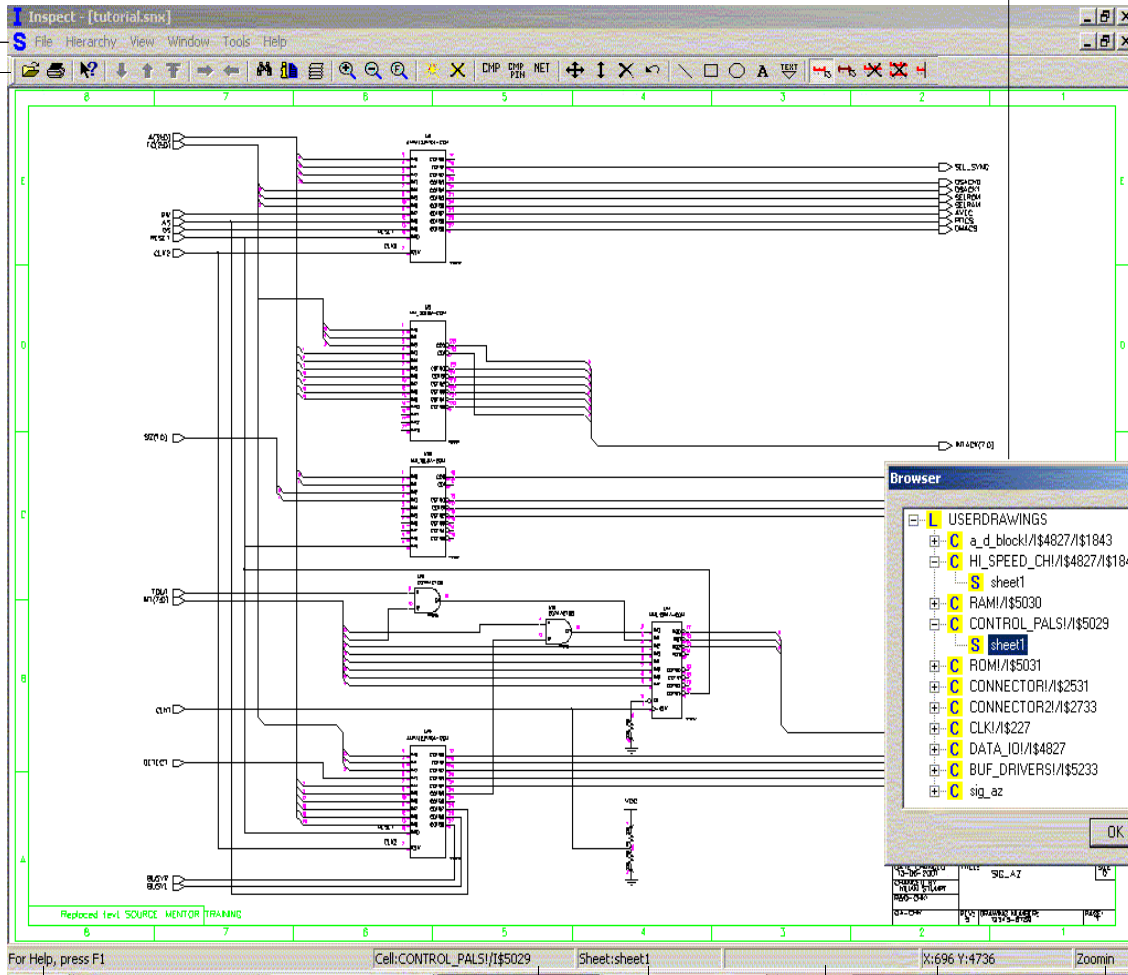
Figure 9: Compare Product Structure

Shortcut Buttons to commands used in interactive operation

Main Toolbar: - all Commands can be executed from here
- the scope of the activated window is shown: S - schematics / B - Board / M - Message file

Header bar: shows the file which has been opened last

View → Browser



There are help texts available on all commands. The help mode selected from the main toolbar displays a "?" cursor. If you click on a command then the help text is displayed.

Inspect numbers the sheets in sequence.

Coordinates of object in extracted file units. Selected mode.

This is the schematic block shown.

Report Field to show the instance name of the object below the cursor.

Figure 10: View Design: Design Structure

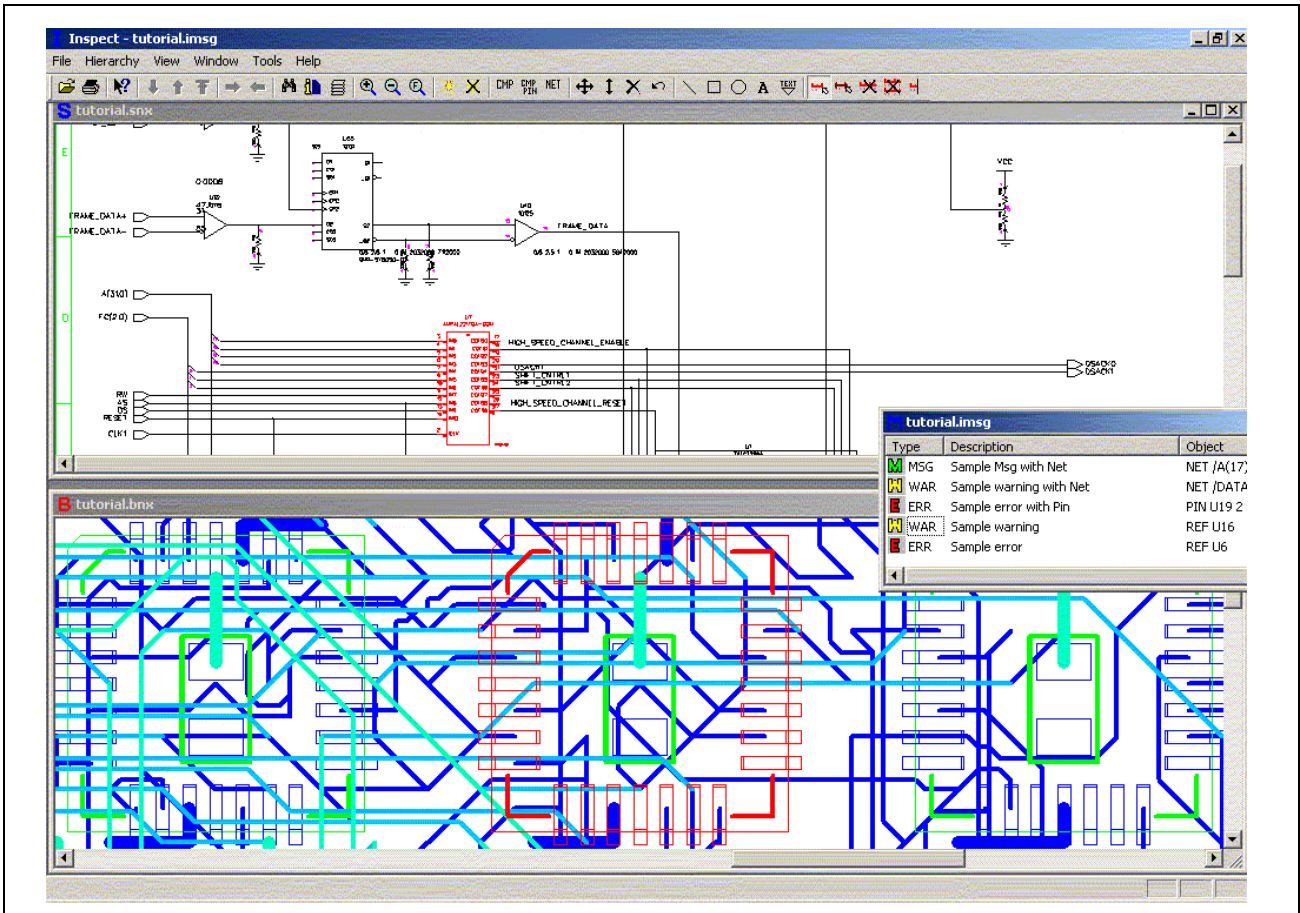


Figure 11: View Design: Cross Probing

The substring selection causes search for all values showing the entered substring. For instance a Find of all REF values starting with D in this design might list the digital components.

Property values in the Component-Pin dialog.

Property Names

Hit the icon in the occurrence list twice to find and highlight the corresponding object in the design with the selected zoom factor.

Select a field for sorting in the occurrence list.

Pairs of value entries are required if you look for a property-value pair.

Here you assign the selected Find to a Shortcut window. Shortcut windows are provided for the selections Component-Pin, Property → Component, and Net. Shortcut windows are opened from the corresponding icons CMP Pin, CMP, and NET. Inspect retains the position and size of shortcut windows between sessions.

Create Excel Sheet with Found Objects. Read into Microsoft Excel with Edit → Paste. The columns to be loaded into the Excell sheet are selected in Setup → Copy.

Global is just a valid selection in schematics. A global Find browses the complete schematic hierarchy, not just the current page.

Type	Field1	Field2	Sheet	Cell
Attr	S800	N1	sheet12	G703
Attr	S800	N1	sheet12	G703
Attr	S800	N1	sheet12	G703
Attr	S800	N1	sheet12	G703

Figure 12: View Design: Find and Highlight

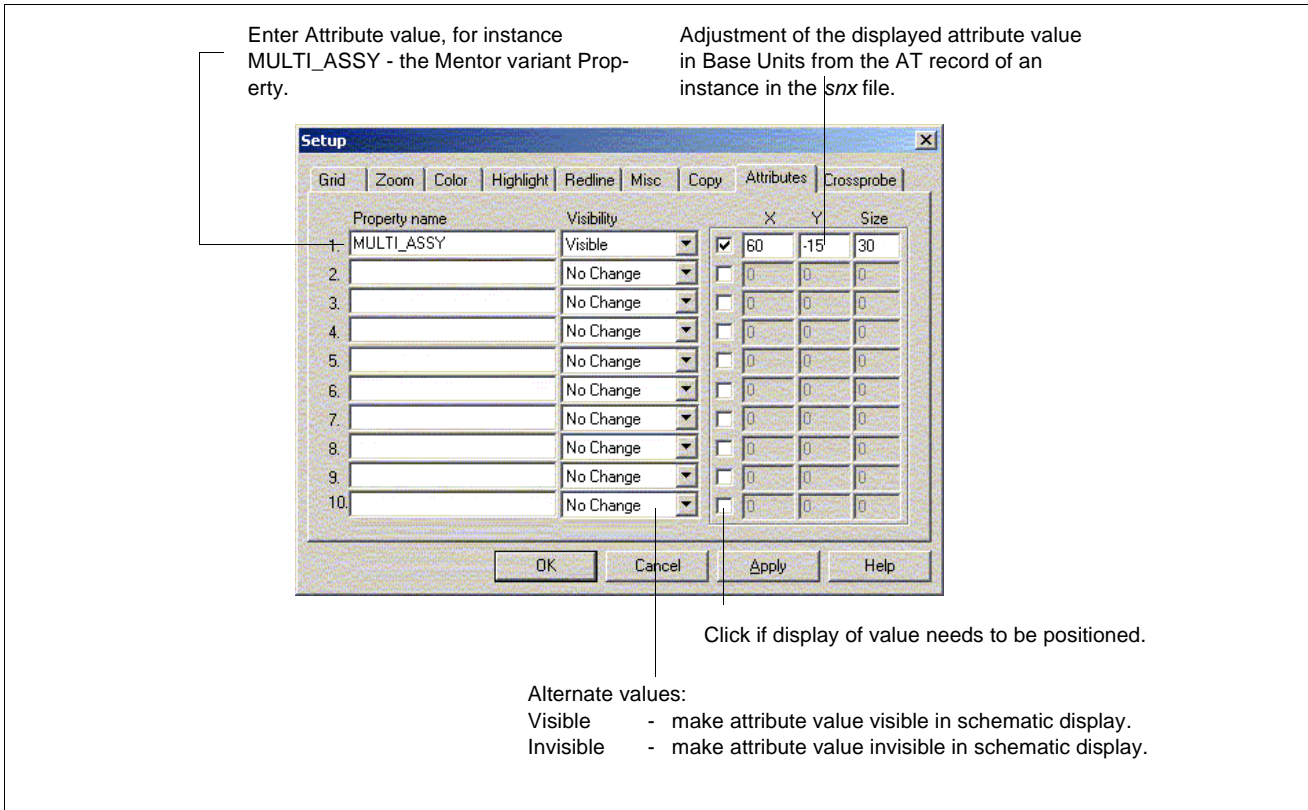


Figure 13: View Design: File -> Setup -> Edit -> Attributes Dialog

Backannotation files for viewing are created with BOM changes and with metadata changes from PLM.

```

<object_name> <change_method> <type> <placeholder> <new value> - Record in <design>.bck

<object_name>:= instance (pcb or schematic), cell or sheet name
<change_method> := {ONE | ALL | SHEET}
ONE - replace specified object only
ALL - replace all objects (name is ignored)
SHEET - replace all objects placed on named sheet. Legal with sheets only.
<type> := {TEXT | ATTR}
TEXT - replace text value. Used with drawing sheets only.
ATTR - replace attribute value
<placeholder> - old text value for texts, attribute name for attribute.
<new value> - new value of attribute or text
  
```

Figure 14: View Design: Backannotate Design Changes

```

# Replace text on component
"sram32kx8" ONE TEXT "[STANDBY]" "replaced STANDBY"

# Replace instance attribute using reference
"D2" ONE ATTR "MODELFILE" "simulation_model_path"

# Replace instance attribute using instance name
"I$462" ONE ATTR "MODELFILE" "sim_model"

# Replace all attributes "company" (ignoring instance names)
"ignored" ALL ATTR "company" "Business Unit"
"ignored" ALL TEXT "1" "one"

# Change page_no value
"sheet1" SHEET ATTR "page_no" "1+"
"sheet2" SHEET ATTR "page_no" "2+"
"sheet3" SHEET ATTR "page_no" "3+"
"sheet4" SHEET ATTR "page_no" "4+"

```

Figure 15: Example of a Backannotation File

Open the file with File → Open Backannotate ...

Note:

The file is opened automatically if *<name>.bck* matches *<name>.snx* in the same directory.

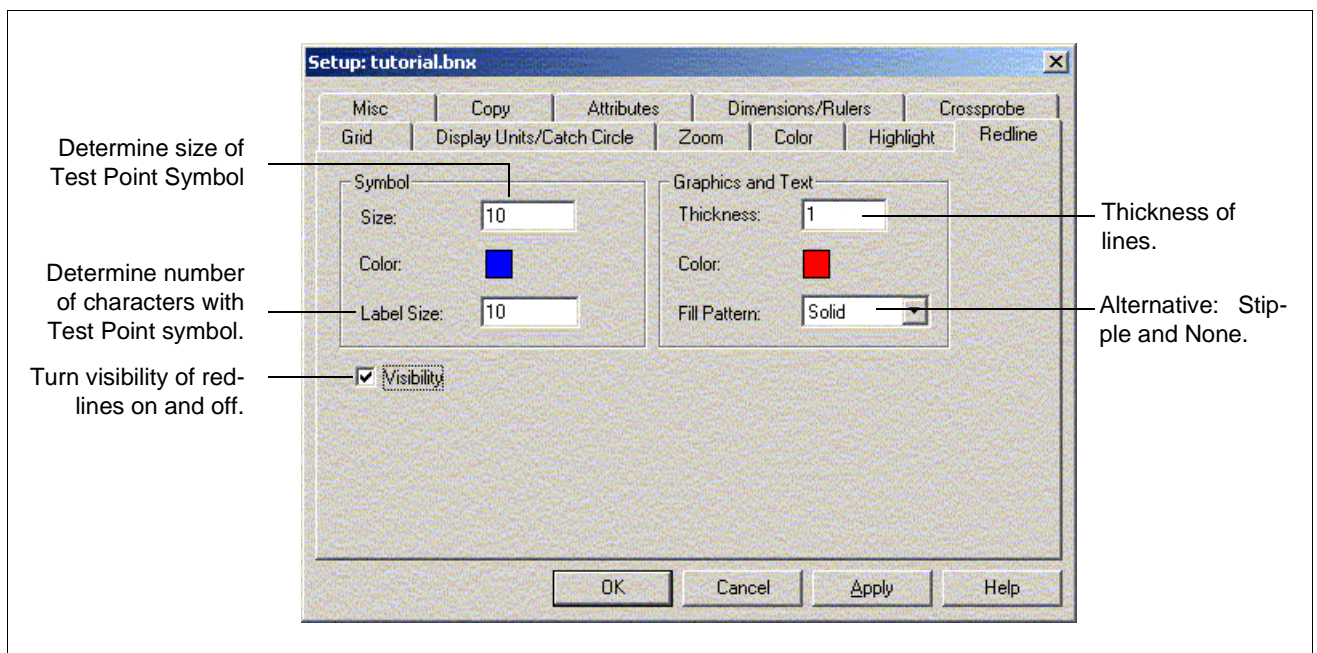


Figure 16: View Design: the Redline Dialog

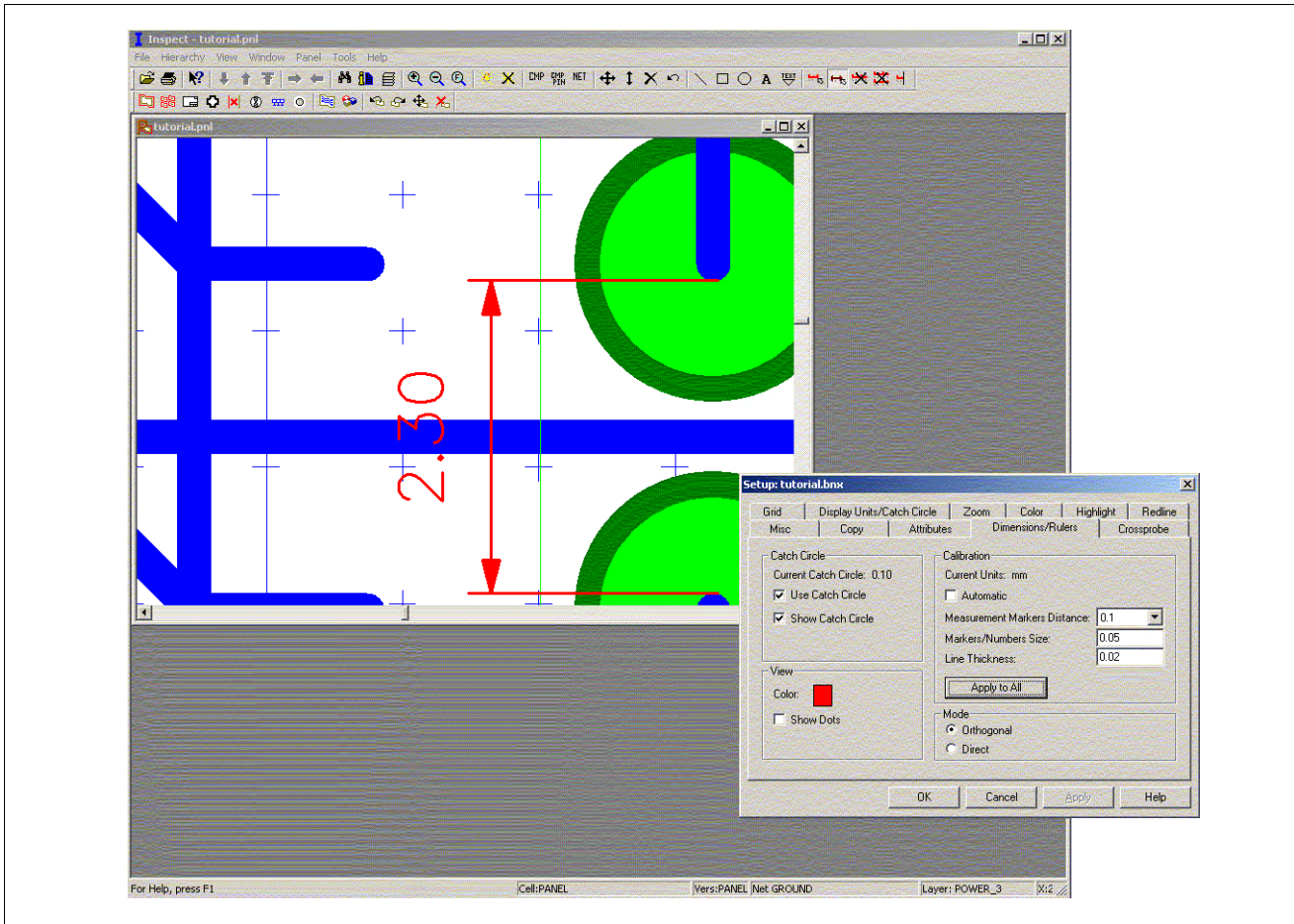


Figure 17: View Design: Dimension and Adjustment of Display

5 Glossary of Terms

ASSY	- Assembly of the Board
API	- Application Program Interface
ATE	- Automatic Test Equipment
BOM	- Bill of Materials
D.O.	- EDA Design Object
ECO	- Engineering Change Order
EDA	- Engineering Design Automation: design environments.
FPL	- Field Programmable Logic
IUI	- Integrate User Interface
Innovation Data	- EDA design data used in ECO and re-use.
PCB	- Printed Circuit Board
PLM	- Product Lifecycle Management
TOC	- Table of contents

6 Conventions

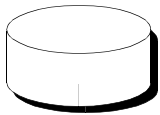
- {A|B} - Alternative A. B default.
- <place_holder> - Replaced by actual value during execution.
- [option] - Optional argument.



- Software process



- Directory



- File

program - Bold: names of executable programs.

/directory - Directory

design.rls - Italics: names of files and paths.

List - Courier: List as screen display.

% - System prompt in Unix.

- System manager prompt in Unix.

* - Wildcard function (any characters).

→ - Select with mouse click.

/1/ - Reference

./<directory> - The directory resides in the location of the actual execution.