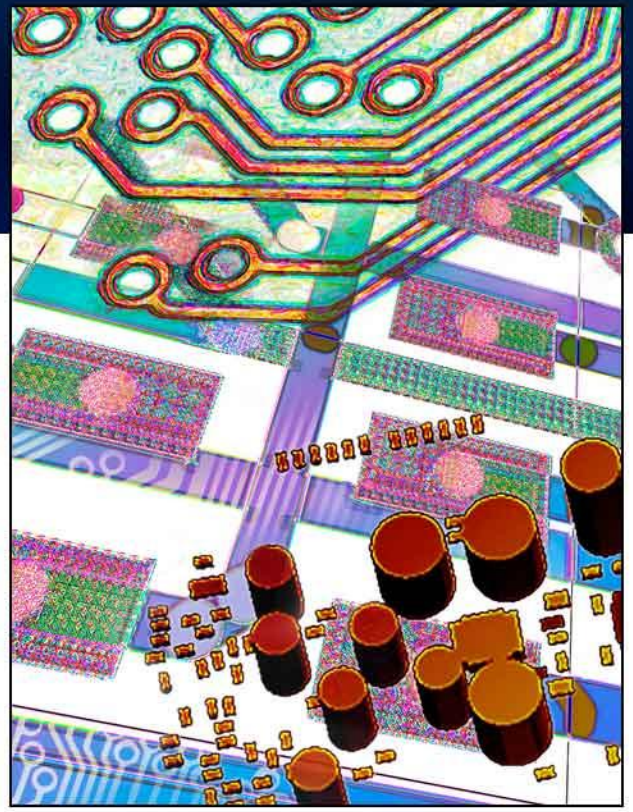


Design Verification

Glenn Son

PCB Team Korea

October 2011

Mentor Graphics®

Agenda

- Gerber 비교
- Gerber와 Design Data간 비교
- Differential View
- Netlist 비교

© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

GERBER COMPARE

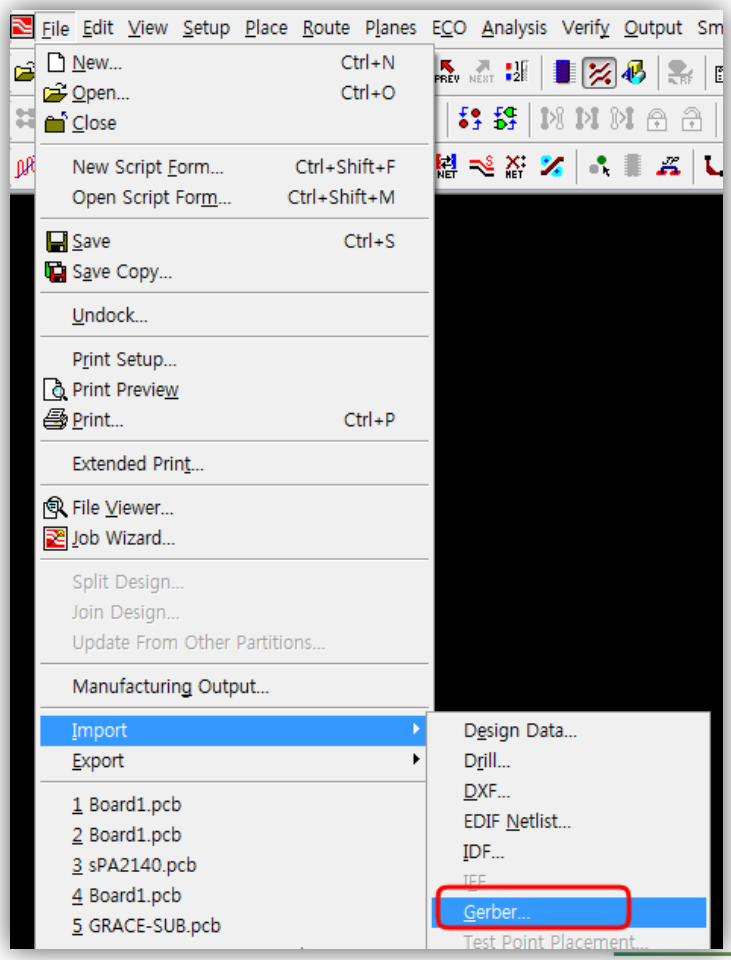
Gerber Compare

- **Expedition**에서 **Gerber**를 그래픽적으로 비교하는 기능입니다.
  - **Board Station Gerber vs. Expedition Gerber** 비교
  - **Expedition Gerber vs. Expedition Gerber** 비교
  - **Expedition Design vs. Expedition Gerber** 비교

© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Gerber Compare

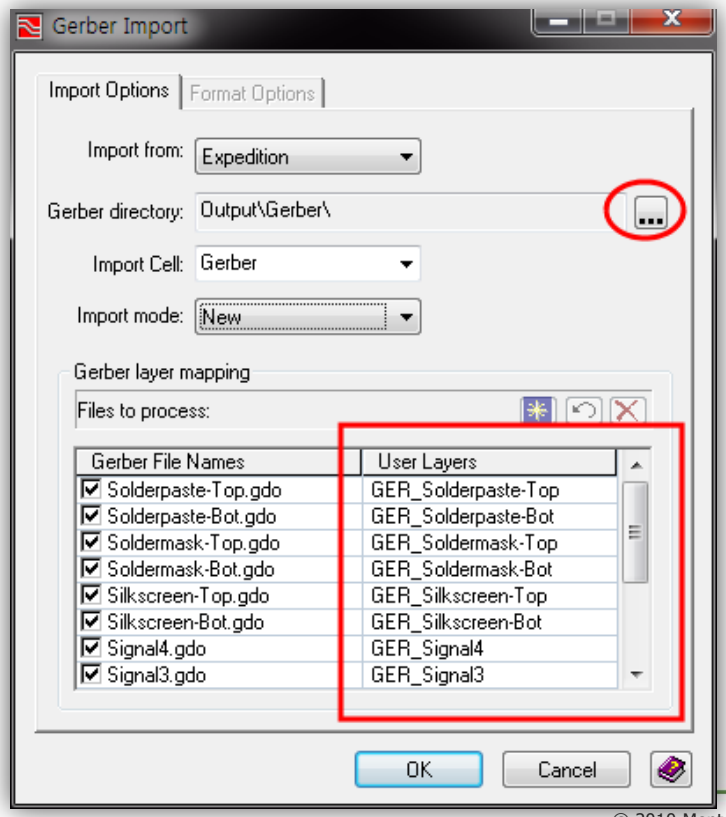
- **Expedition PCB에서 Gerber를 다음과 같이 불러옵니다.**
  - **Fablink XE 또는 Fablink XE Pro License 필요**



© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Gerber Compare

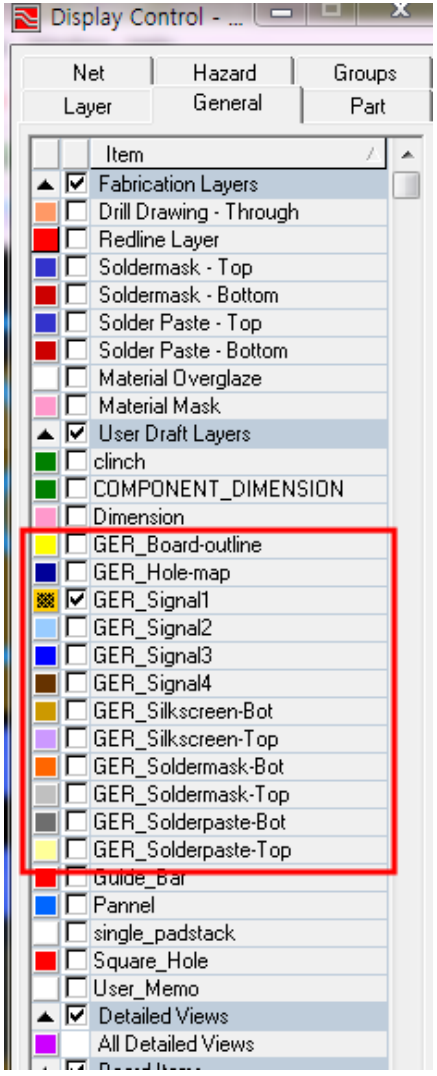
- **Expedition PCB**에서 **Gerber**가 생성되는 폴더를 인식해서 생성된 **Gerber**를 자동으로 불러옵니다. 또는 **Gerber**가 위치한 폴더를 선택할 수도 있습니다. (비교 대상 **Gerber**를 동일한 방법으로 불러옴)



© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Gerber Compare

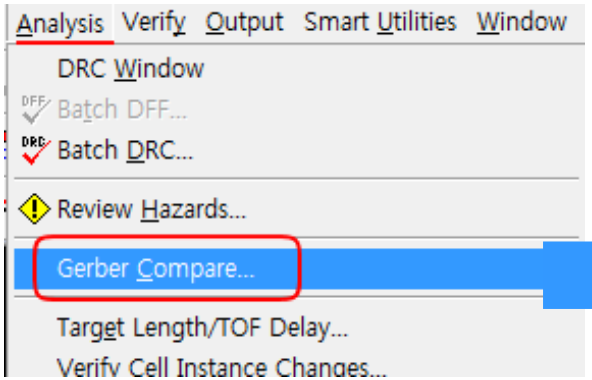
- **User Draft Layers**에서 불러온 **Gerber**를 확인 할 수 있습니다.

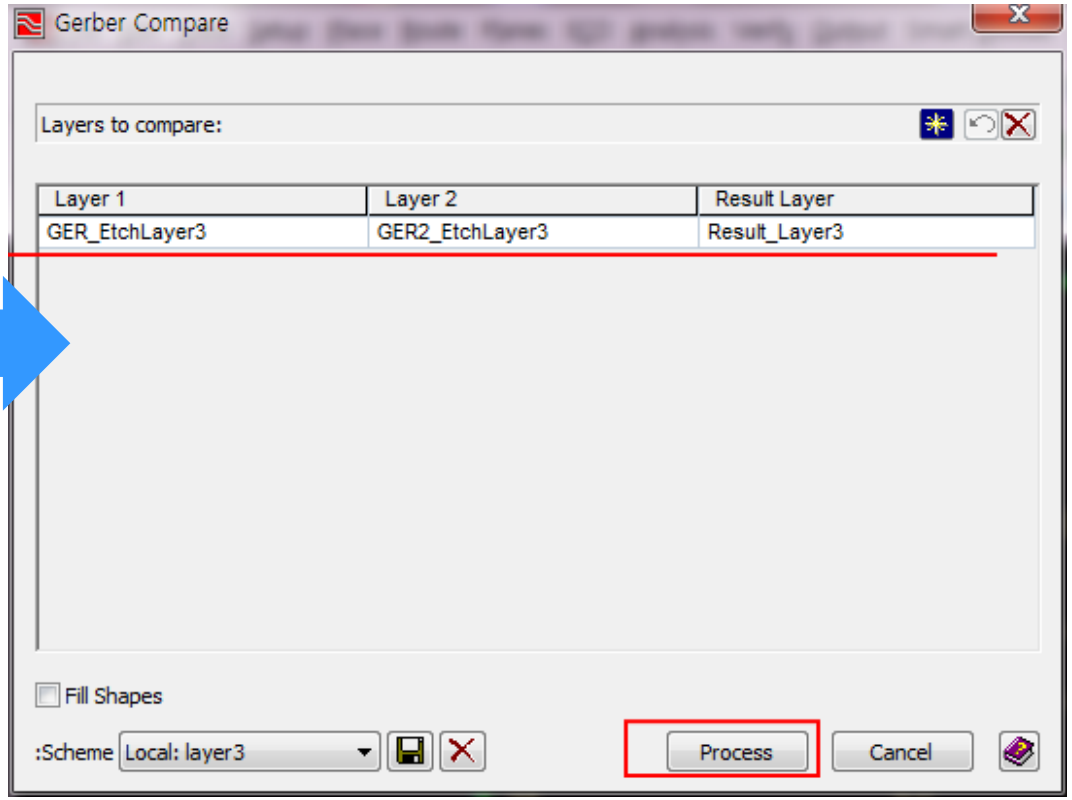


© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Gerber Compare

- Analysis > Gerber Compare 실행



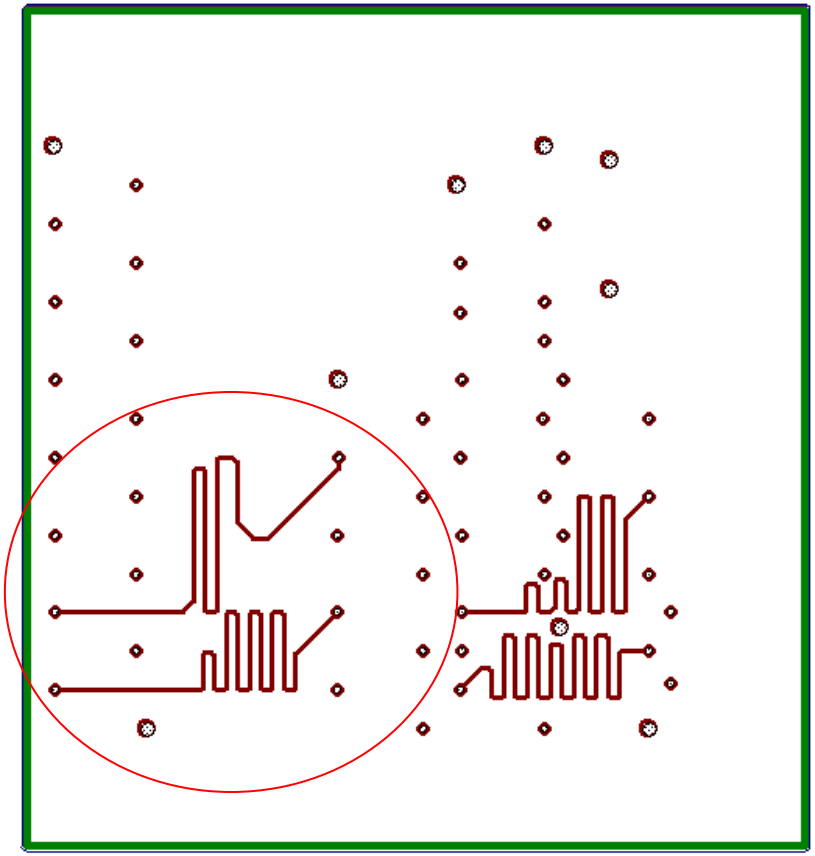


© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

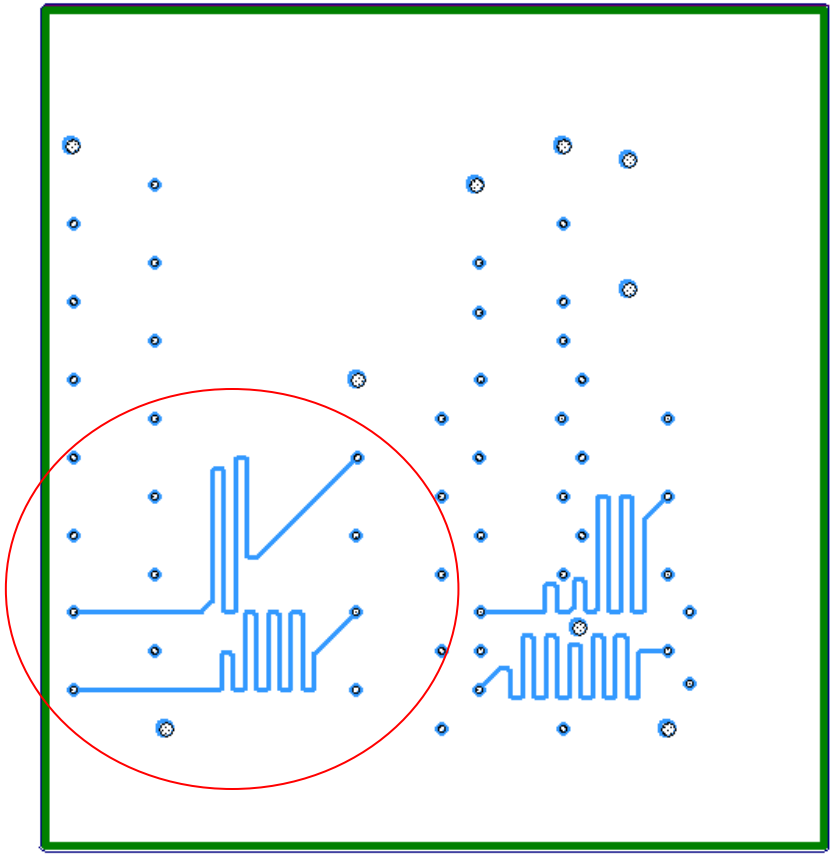
Gerber Compare

- 비교 확인

GER_EtchLayer3



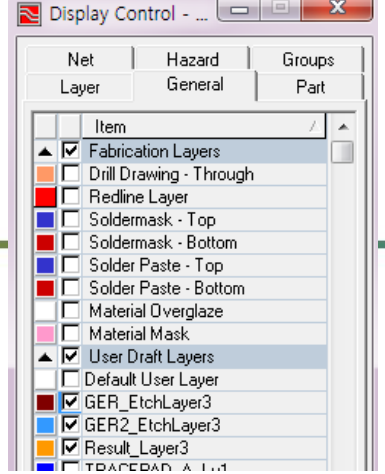
GER2_EtchLayer3



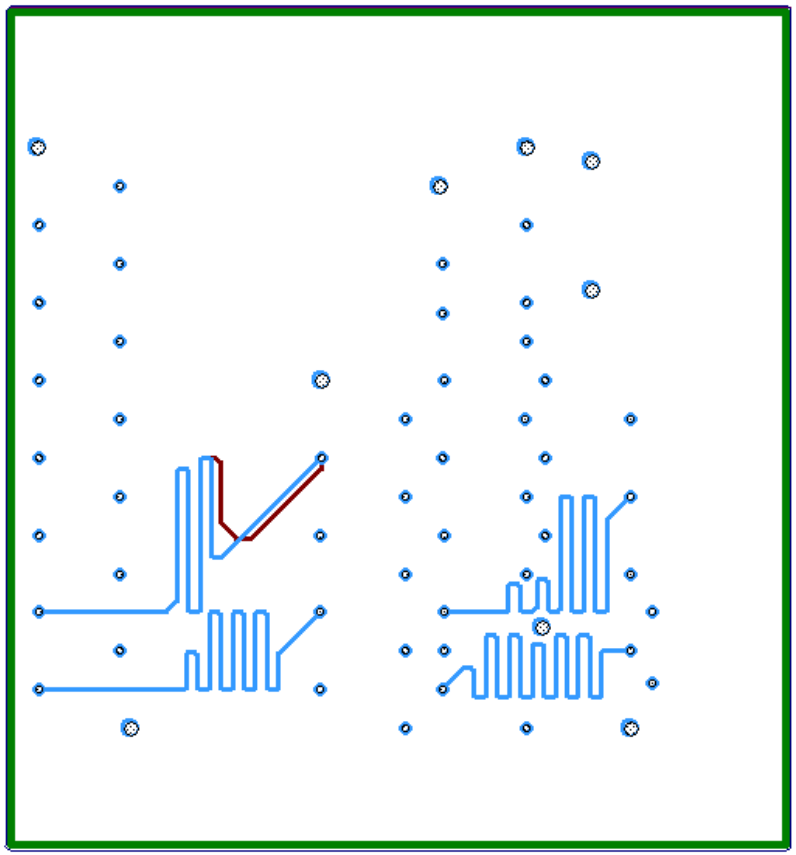
© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Gerber Compare

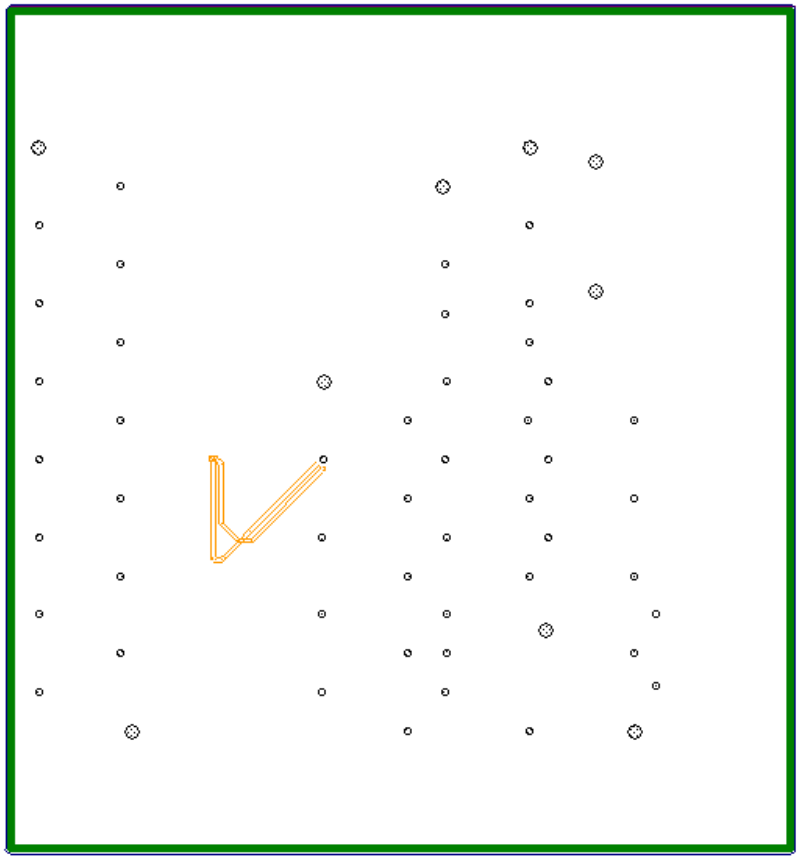
- 비교 확인



GER_EtchLayer3 + GER2_EtchLayer3



Result_Layer3

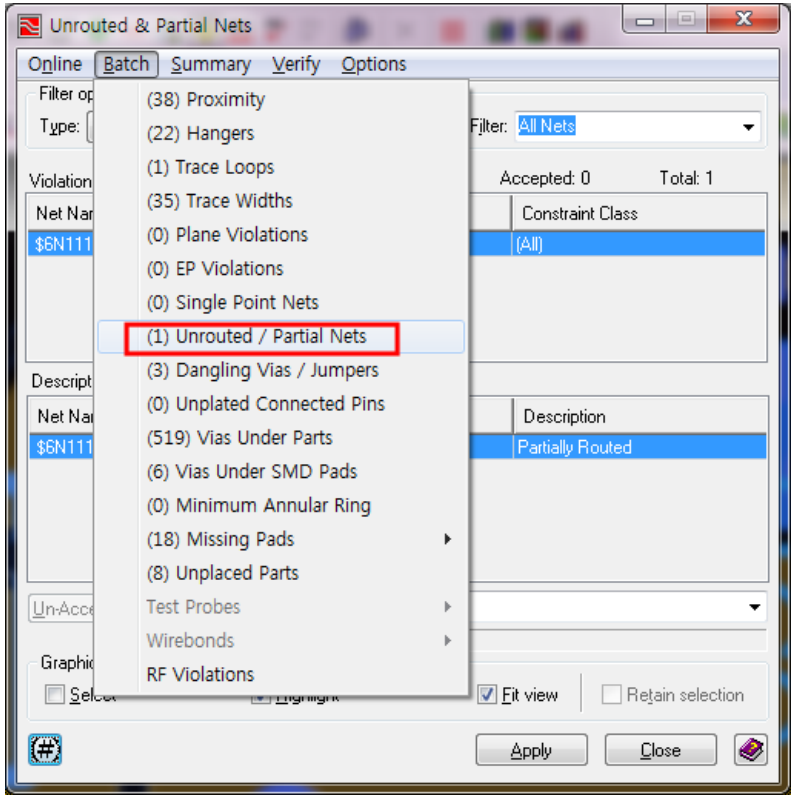


© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

GERBER와 DESIGN DATA간 비교

Gerber 출력 전 준비사항

- **DRC (Batch DRC 포함) Check**를 한 후, **Review Hazards**에서 **Unrouted / Partial Net** 항목 등을 확인합니다.



© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

IPC-D-356 Netlist를 사용한 Gerber와 디자인 데이터간 Open/Short 검증 방법

- **IPC-D-356 Format은 Bare Board Testing용으로 사용되며, Netlist 형식으로 표현됩니다.**
  IPC-D-356 Format을 사용하여 PCB의 Open/Short 검증을 할 수 있으며, 이러한 Open/Short 검증은 PCB 설계 프로세스에 포함하여야 할 단계이기도 합니다.
- **IPC-D-356은 다음과 같이 3가지 Format이 사용되고 있습니다.**
  - **IPC-D-356**
    http://www.downstreamtech.com/Support_CAM/Advisories/IPCD356_Simplified.pdf
  - **IPC-D-356A**
    http://www.solidigm.com/downloads/ipc356a.pdf
  - **IPC-D-356B**
    http://www.ipc.org/TOC/IPC-D-356B.pdf
- **Expedition PCB에서는 IPC-D-356과 IPC-D-356B Format을 출력 할 수 있음.**

* IPC: Institute For Interconnecting And Packaging Electronic Circuits

© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

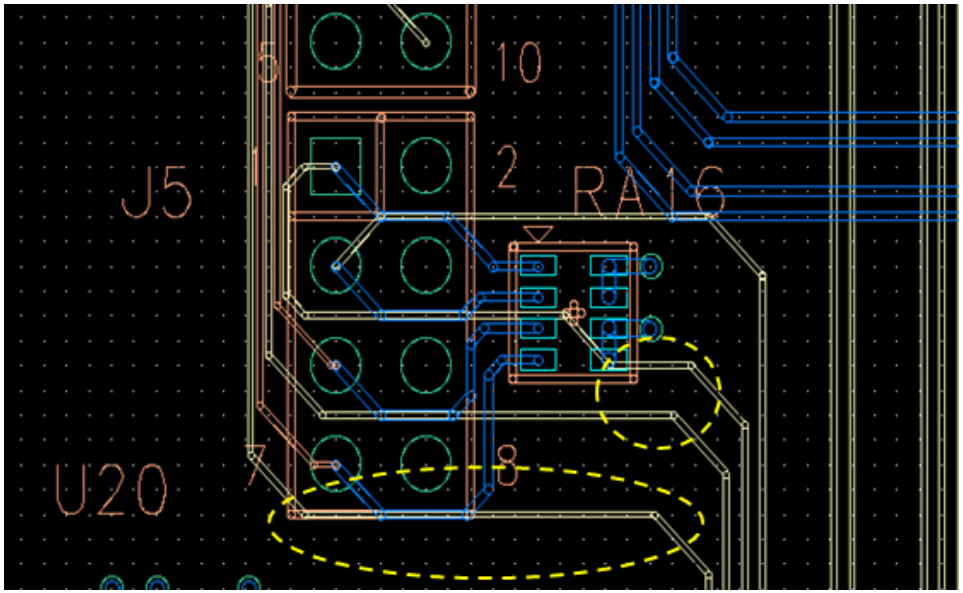
IPC-D-356 Netlist를 사용한 Gerber와 디자인 데이터간 Open/Short 검증 방법

- 범용적인 CAM 툴에서 Open/Short 테스트를 하는 절차는 다음과 같습니다.
  1) Gerber Data를 입력해서 Gerber Netlist 생성
  2) IPC-D-356 Netlist 입력함
  3) Gerber Netlist와 IPC-D-356 Netlist를 비교해서 Open/Short을 검증함.
- IPC-D-356 Netlist는 디자인에서 추출된 파일이며 이 파일을 생산데이터인 Gerber와 비교해서 Open/Short 여부를 확인함으로써 디자인과 Gerber간의 오류 유무를 판별할 수 가 있습니다.
- 또한, 멘토의 DFM 전문 툴인 vSure를 사용하면 손쉽게 Open/Short 검증이 가능합니다.

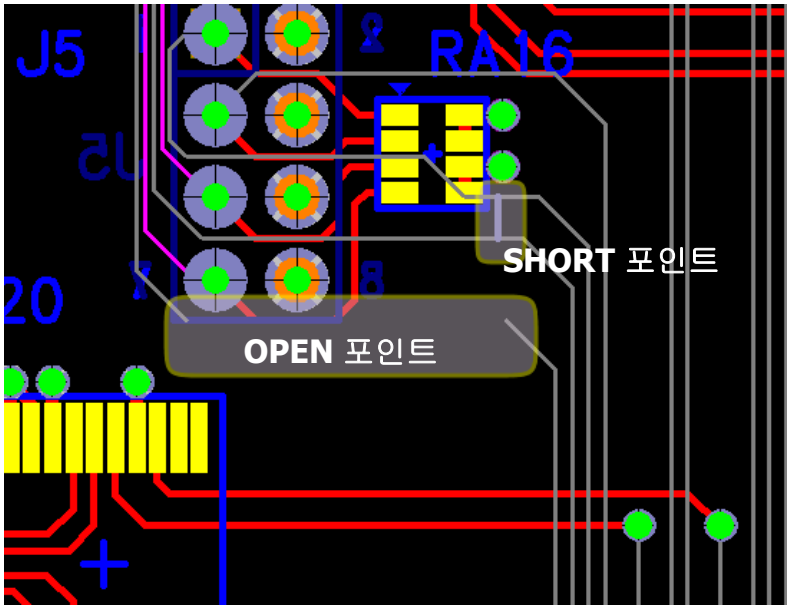
© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

OPEN, SHORT 검증을 위한 데이터 구성

1 원본 디자인 데이터



2 **Gerber** 데이터에서 **OPEN, SHORT** 생성

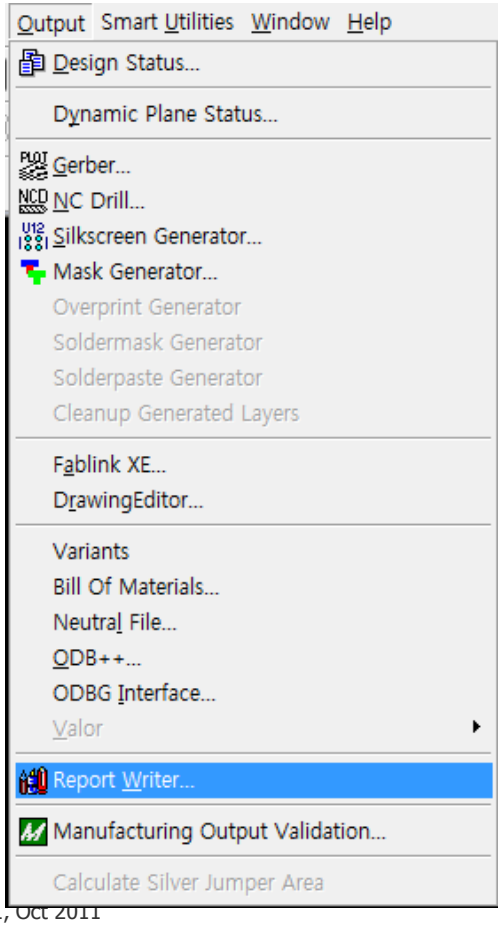


© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

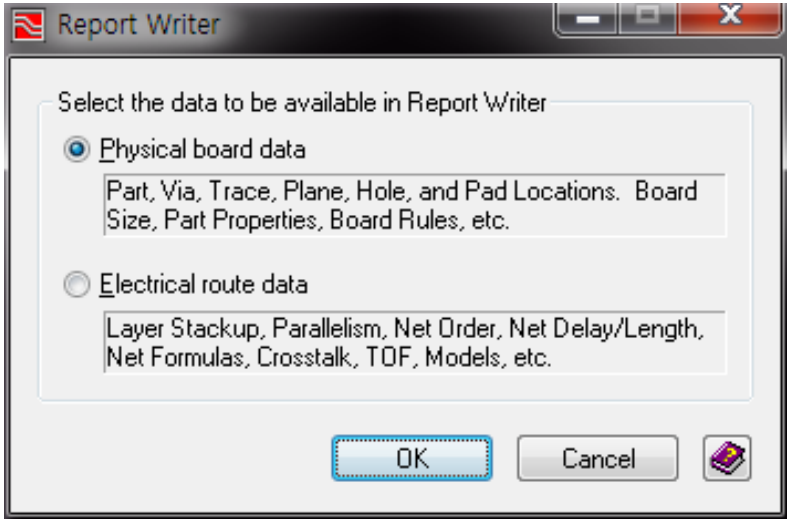
IPC-D-356 Netlist 출력 방법

- **Expedition**에서 **Report Writer**로 **IPC-D-356 data**를 생성하는 방법은 다음과 같습니다.

1 **Output > Report Writer**를 실행



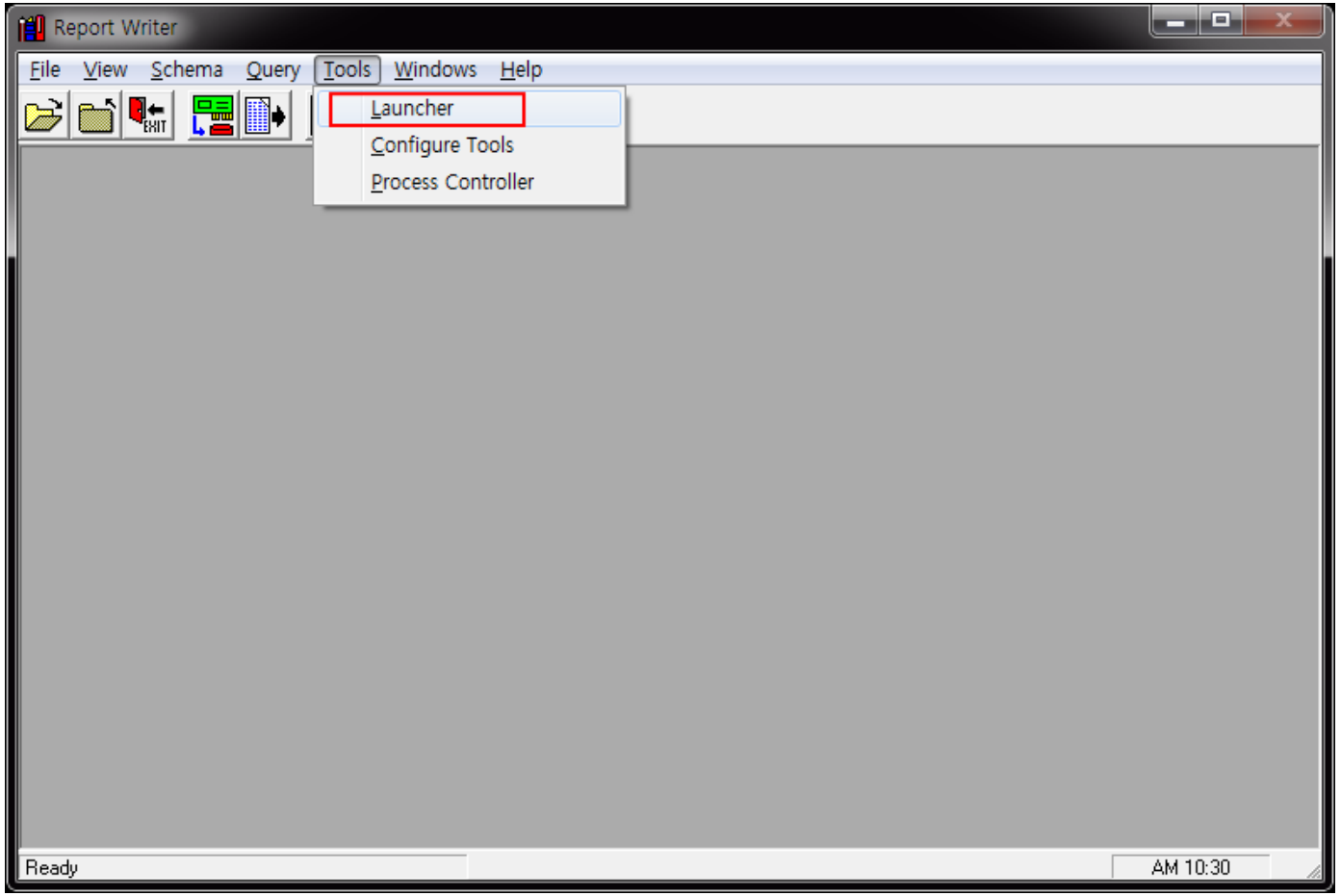
2 **Physical board data**를 선택 후 **OK**



© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

IPC-D-356 Netlist 출력 방법

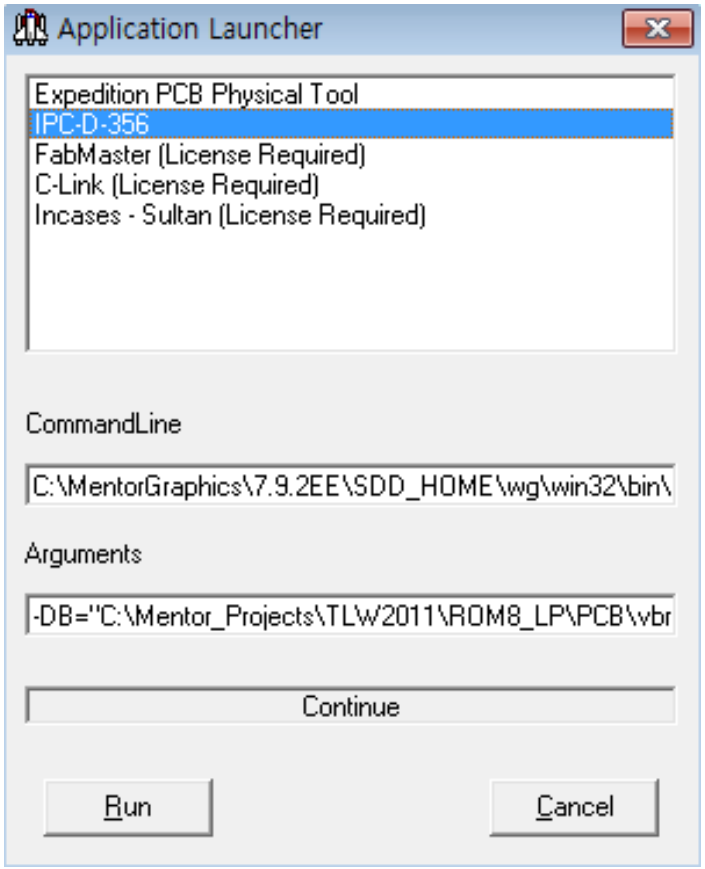
3 **Report Writer**가 실행되면 **Tools > Launcher**를 실행



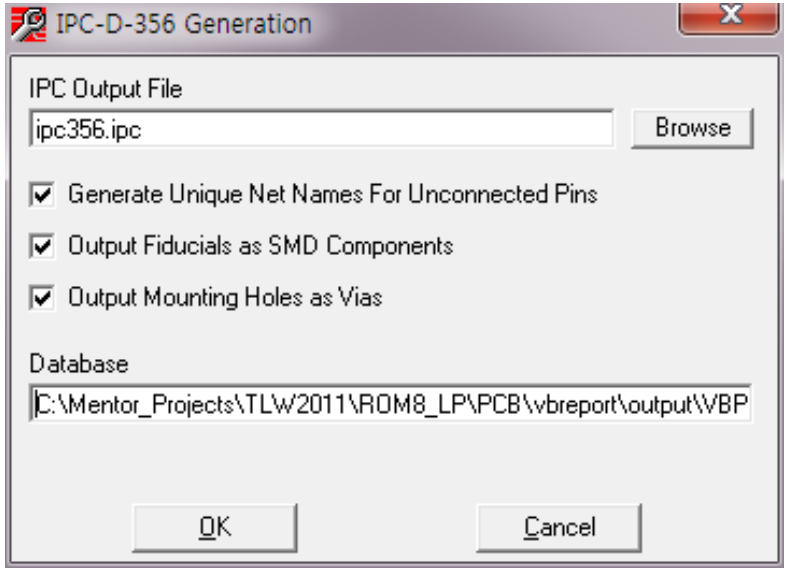
© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

IPC-D-356 Netlist 출력 방법

4 실행된 **Launcher**에서 **IPC-D-356**을 선택 후 **Run**을 클릭



5 **IPC-D-356 Generation**창이 실행되면 **OK**를 클릭

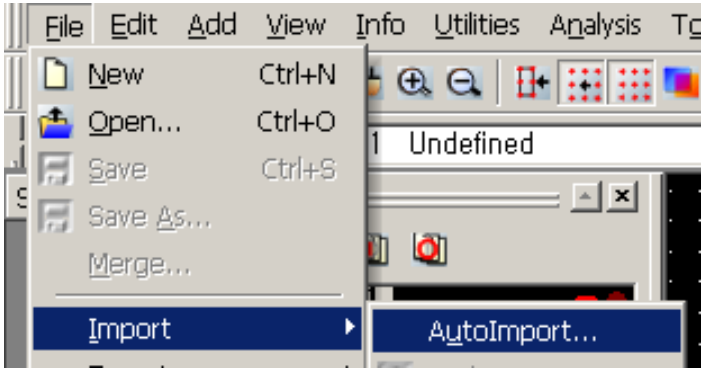


PCB\vbreport\output\ipc356.ipc 파일이 생성

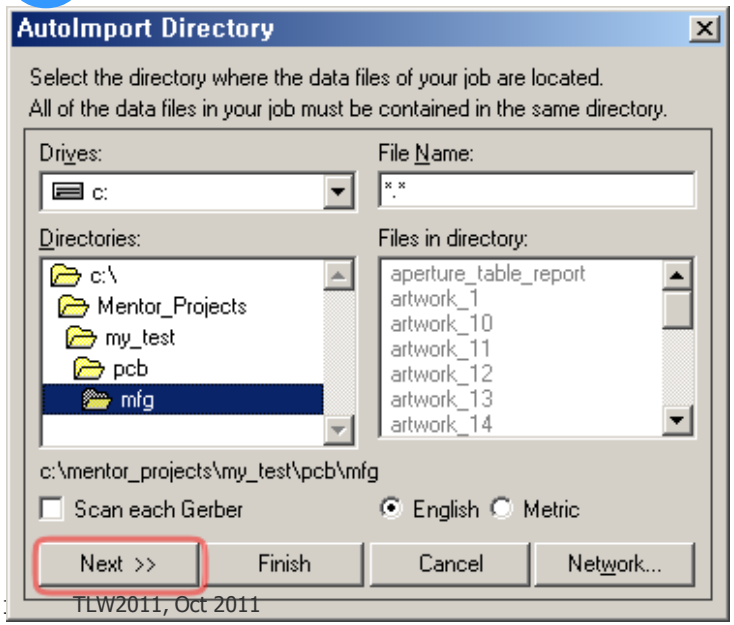
© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

CAM350에서 Gerber & Drill Import

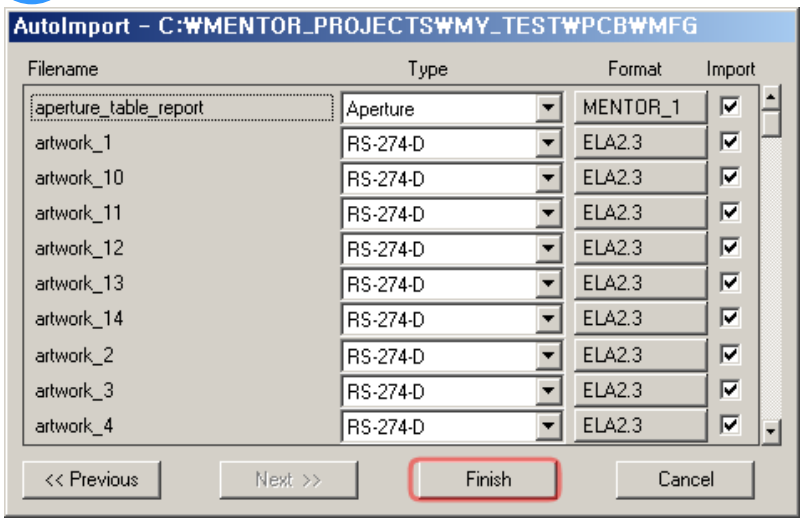
1 **File > Import > AutoImport**

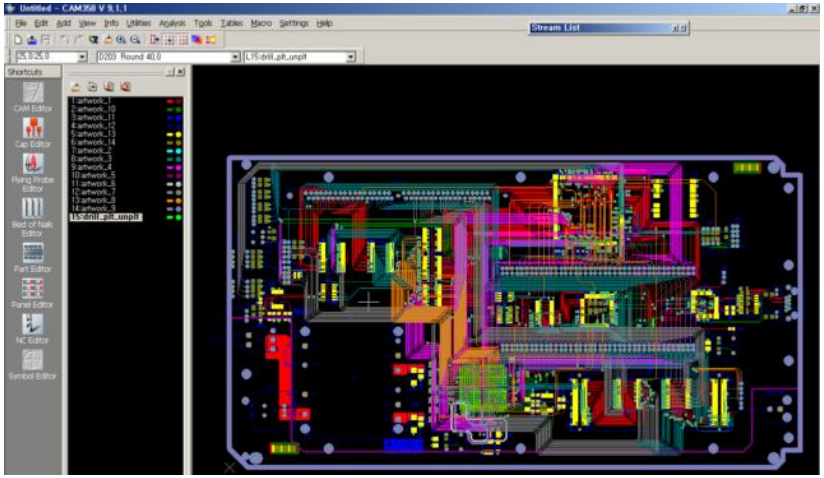


2



3

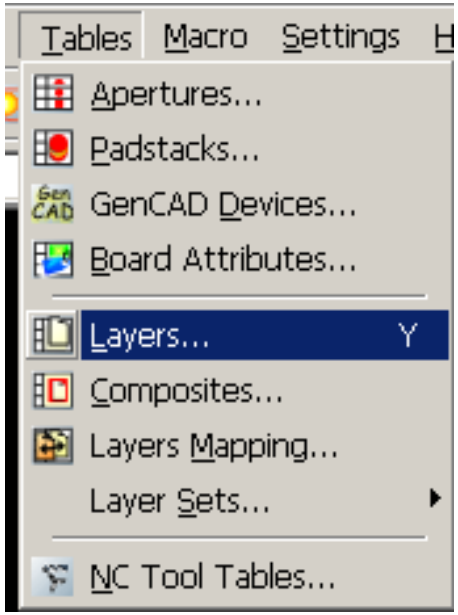




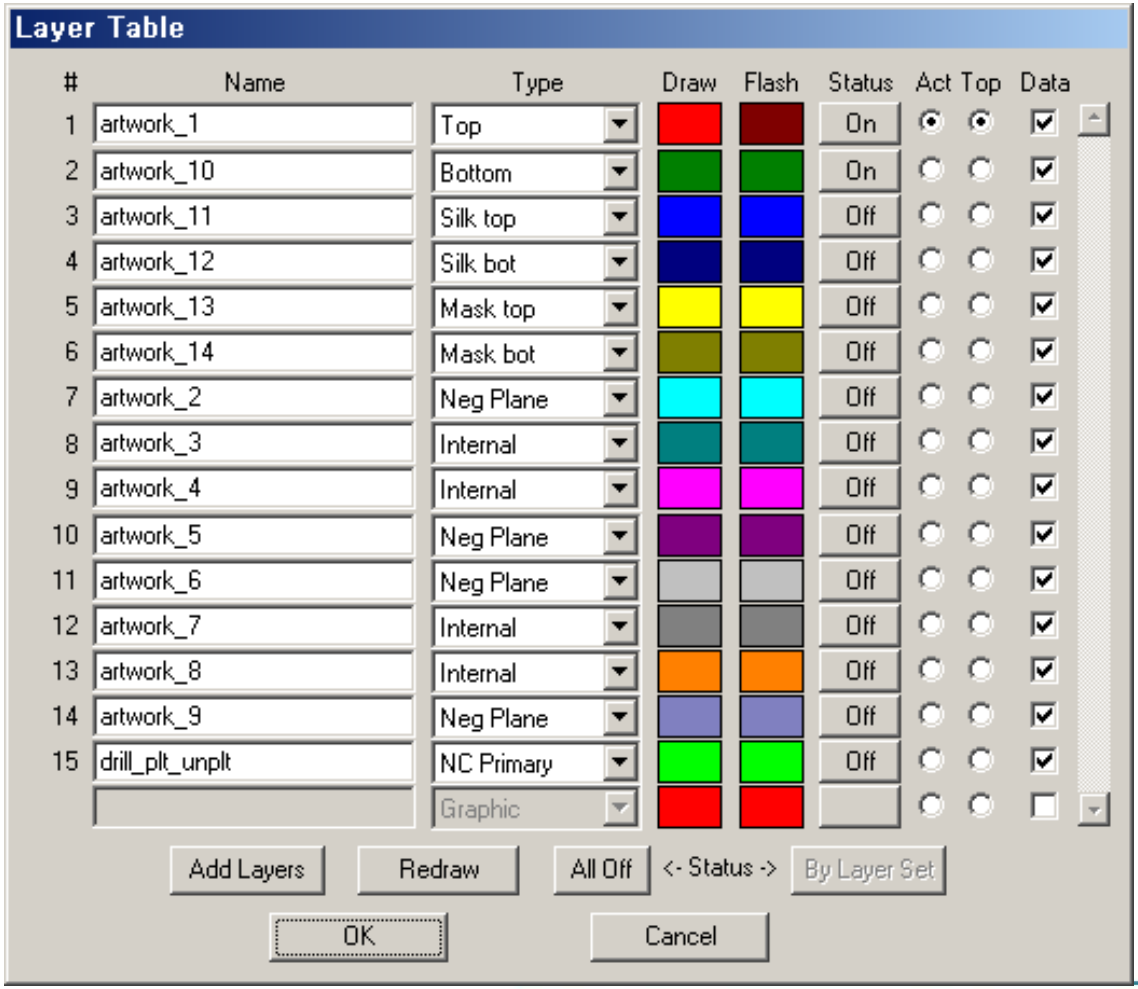
© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

CAM350에서 Layer Type 설정

1 **Tables > Layers**



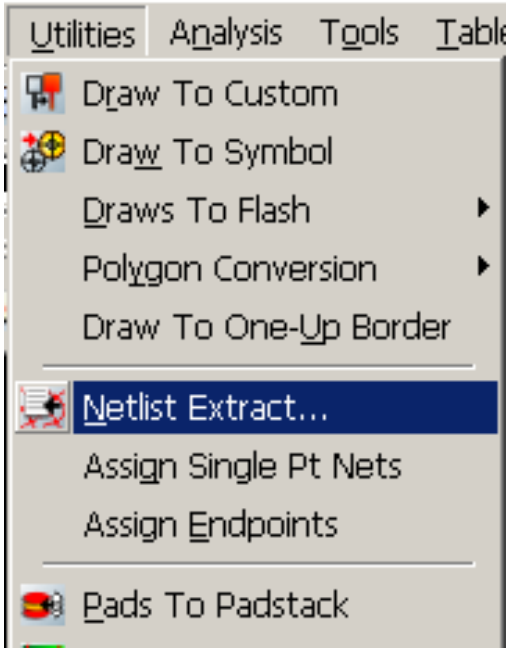
2 **Layer Type**을 설정한다.
Top/Bottom Layer는 반드시 설정해 주어야 한다.



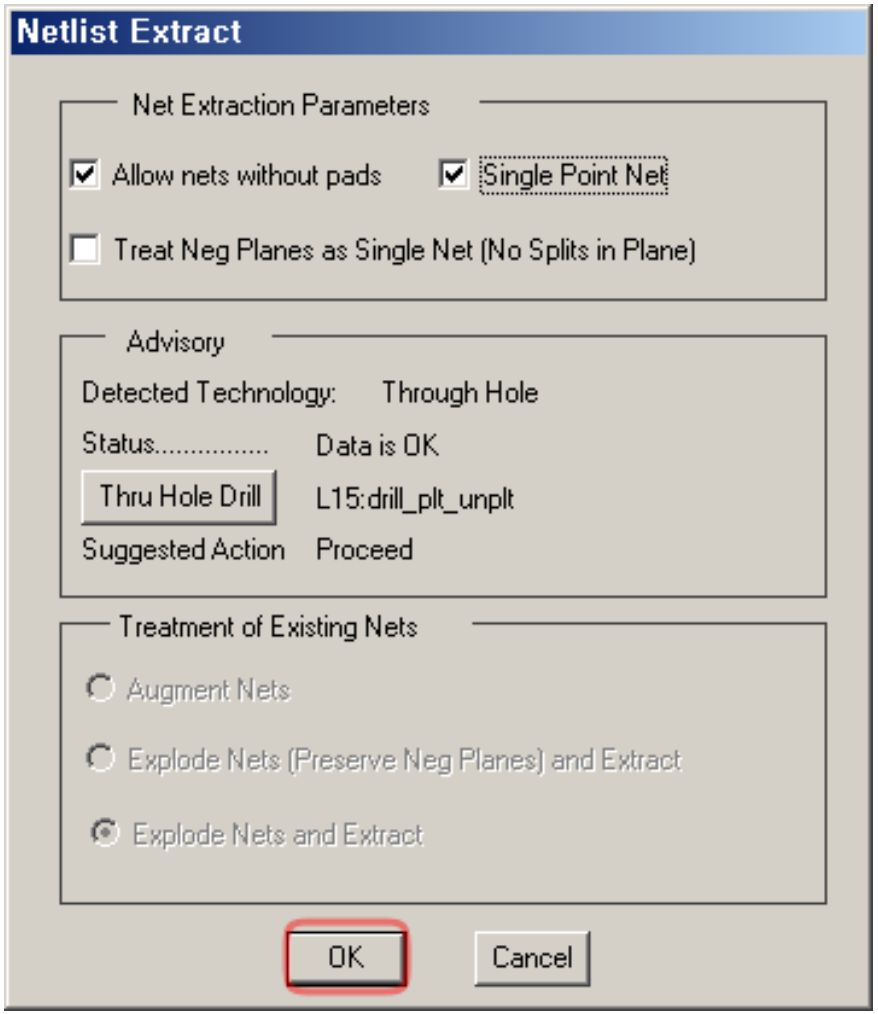
© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

CAM350에서 Gerber Netlist 추출

1 **Utilities > Netlist Extract**



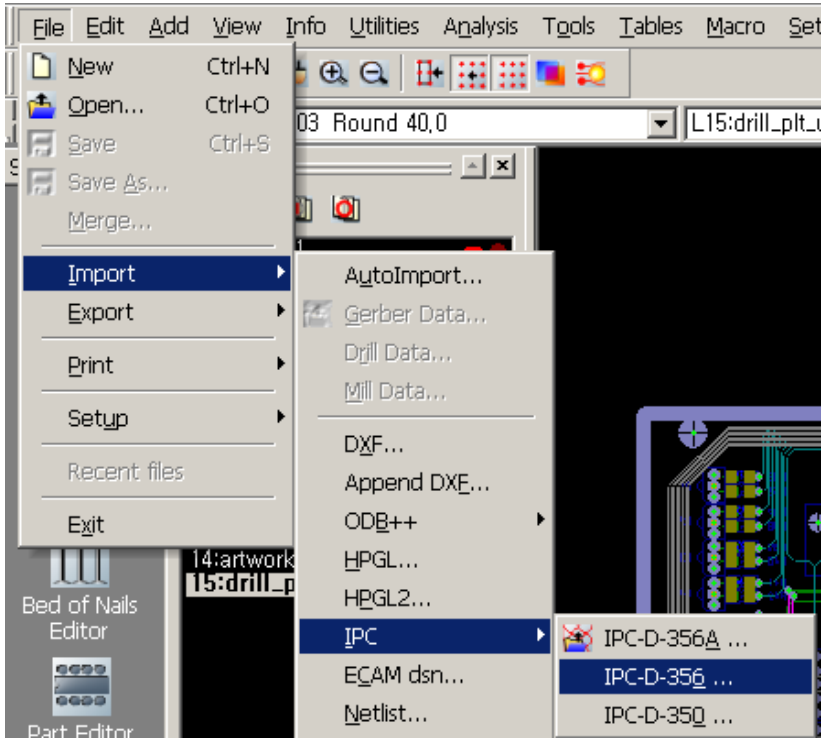
2



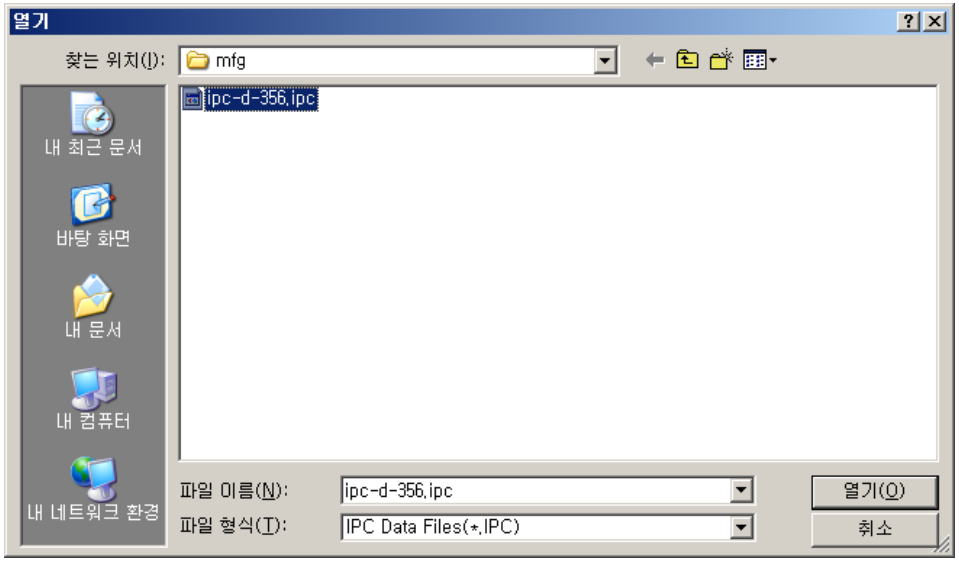
© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

CAM350에서 IPC Netlist Import

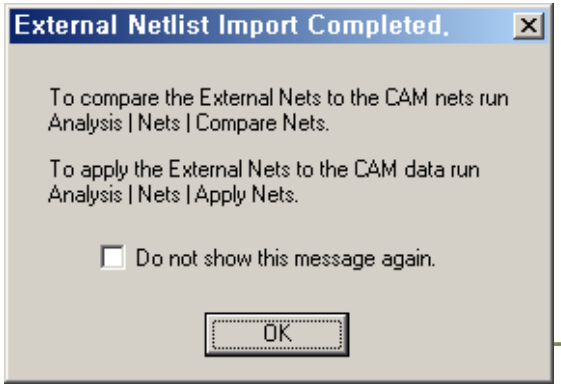
1 **File > Import > IPC > IPC-D-356**



2

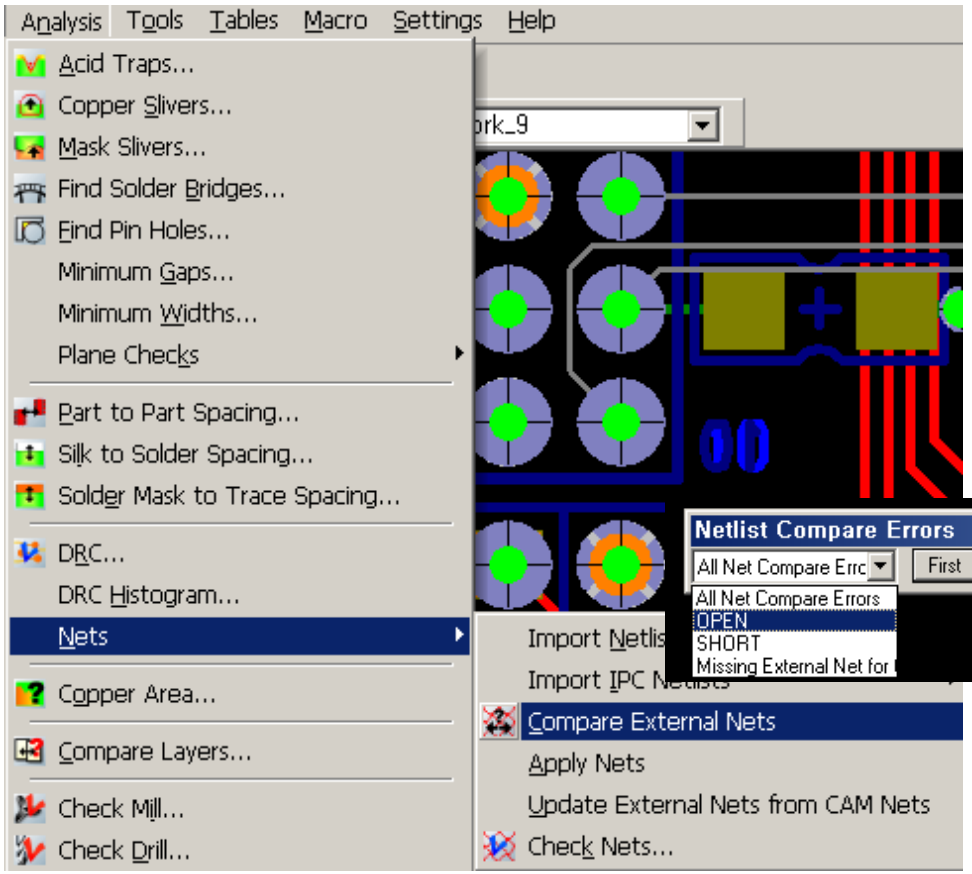


3

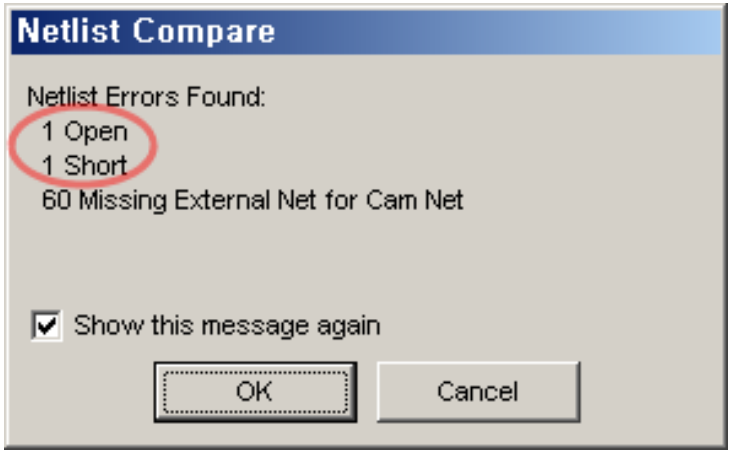


CAM350에서 Gerber Netlist와 IPC Netlist 비교

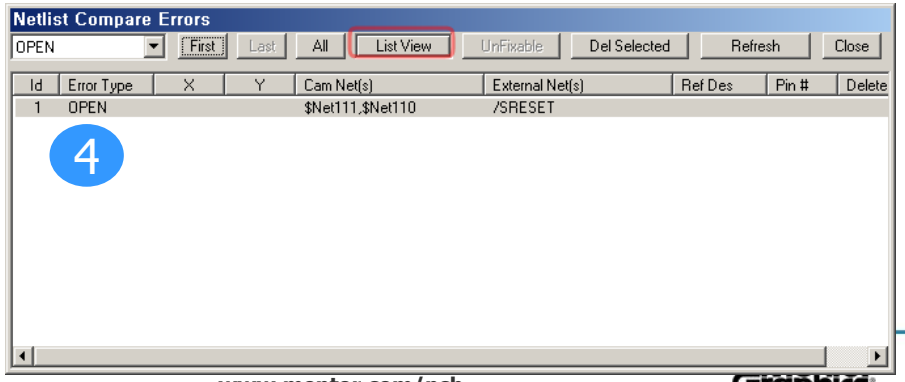
1 **Analysis > Nets > Compare External Nets**



2



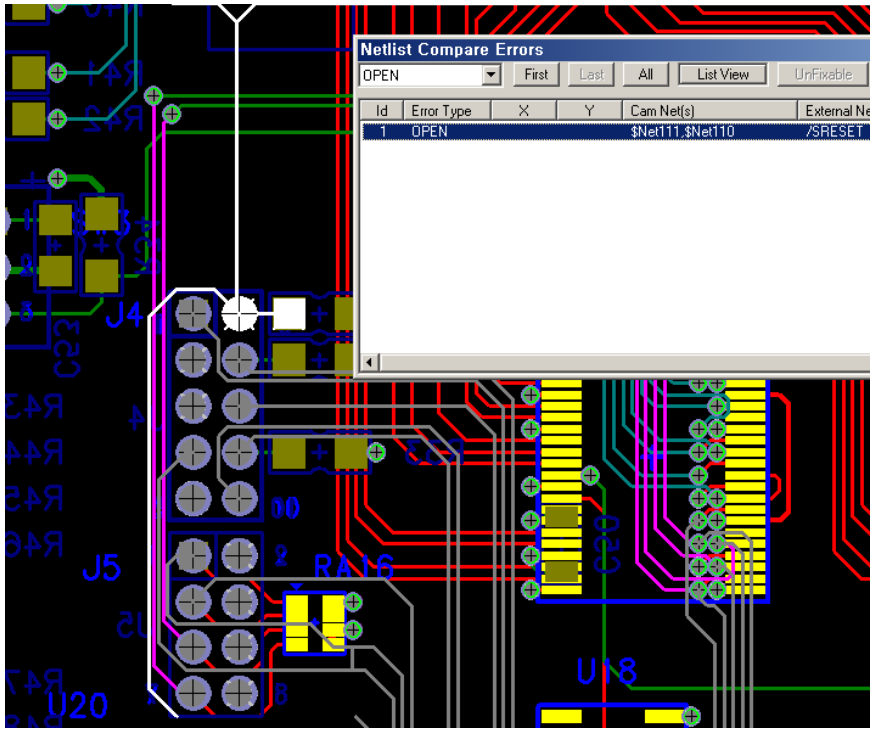
3



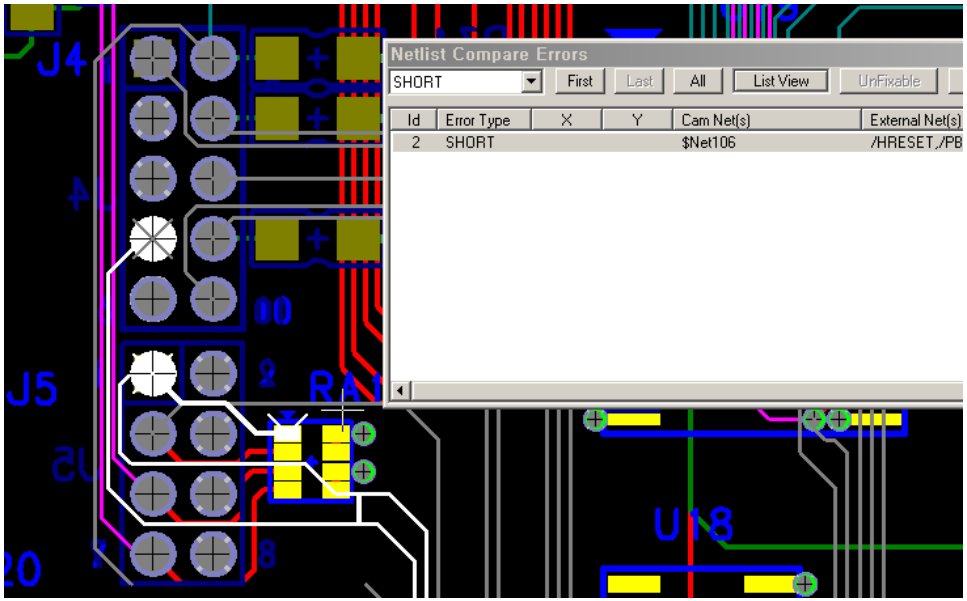
4

CAM350에서 Gerber Netlist와 IPC Netlist 비교

5 Open 상태



6 Short 상태



© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

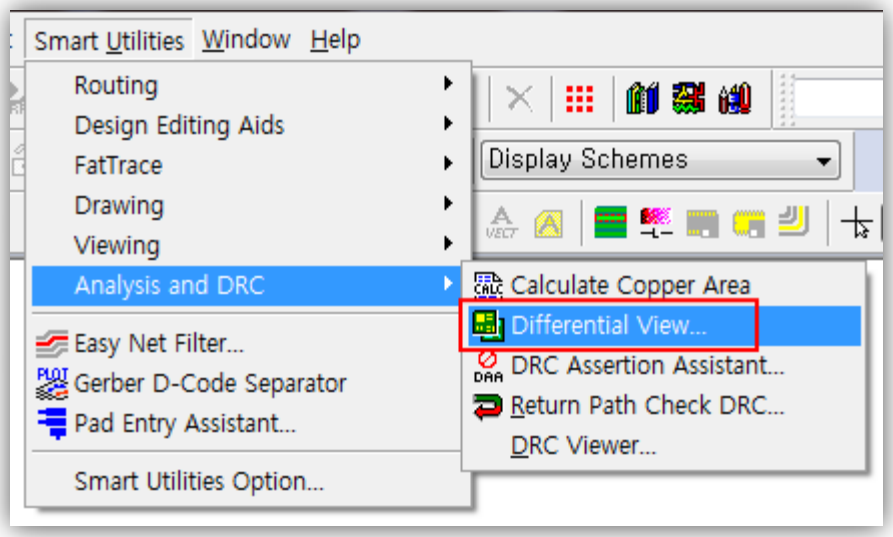
DIFFERENTIAL VIEW

Differential View

- **Expedition PCB**에서는 **PCB**와 **Gerber data**를 비교하는 기능 외에도 **PCB** 데이터간 직접 비교하는 **Differential View** 기능이 있습니다.

1 **ToolBar**의 **Differential View**를 실행하거나

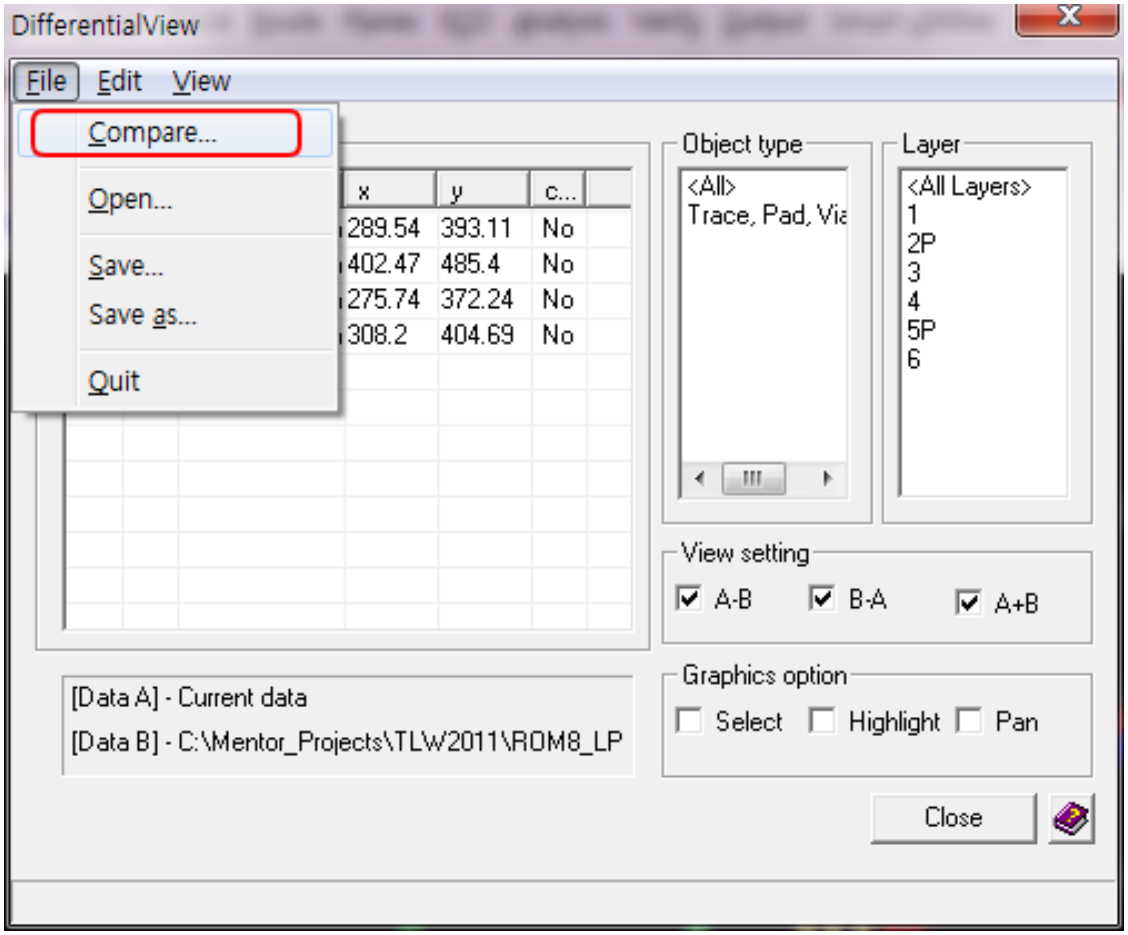
2 **Analysis and DRC > Differential View** 를 실행합니다.



© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Differential View

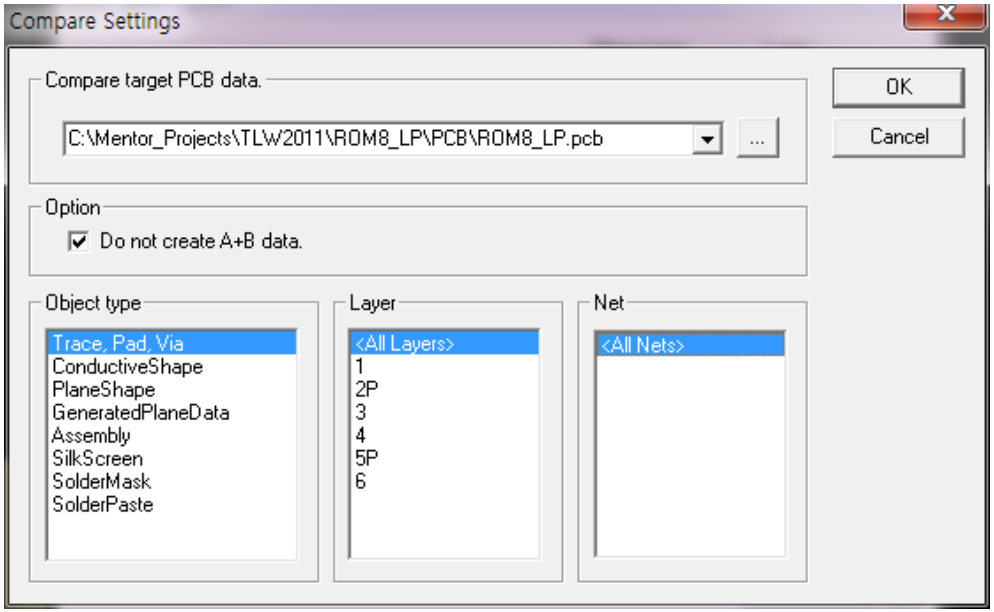
3 **Differential View**창에서 **File > Compare** 를 선택합니다.



© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Differential View

4 비교 **Settings**에서 비교할 대상 **PCB**와 비교 조건을 선택합니다.

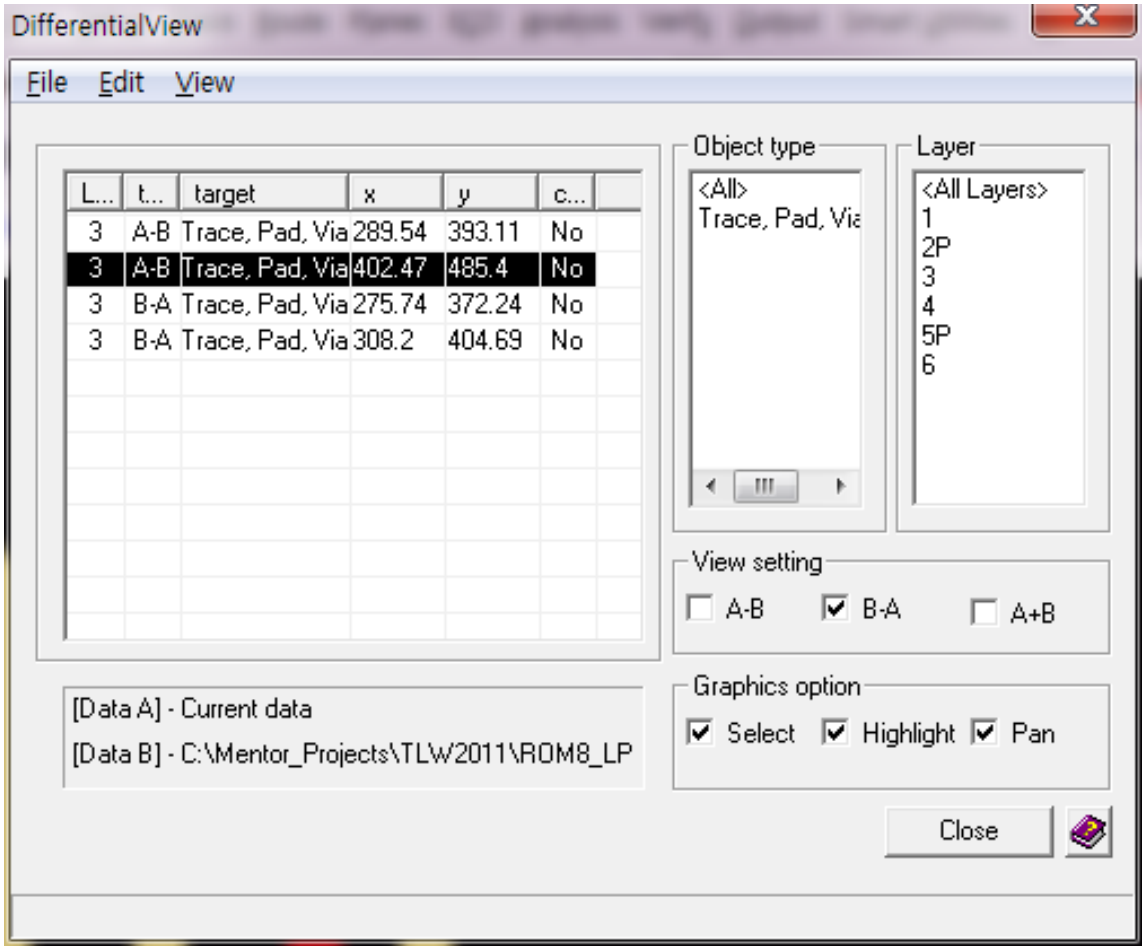


- 현재 **open**된 **PCB**는 **ROM8_LP2**의 **PCB -> A**
- 비교할 **PCB**는 **ROM8_LP**의 **PCB -> B**
- **Option : Do not crate A+B data**
  A와 **B**의 **and(**논리연산**)**한 **data**를 생성합니다. **(**이 부분은 **Check**하여 생성하지 않도록 합니다**.)**
- **Object type :** 비교하고자 하는 **Object**를 선택합니다. 기본적으로 **Trace, Pad, Via**를 선택합니다.
- **Layer:** 모든 **Layer**를 비교시 **<All Layers>**를 선택하고 특정 **Layer**를 비교시에는 원하는 **Layer**를 선택합니다.
- **Net:** 모든 **Net**을 비교시 **<All Nets>**를 선택하고 특정 **Net**을 비교시에는 원하는 **Net**을 선택합니다.

© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Differential View

5 **Compare**를 실행한 결과를 확인합니다.

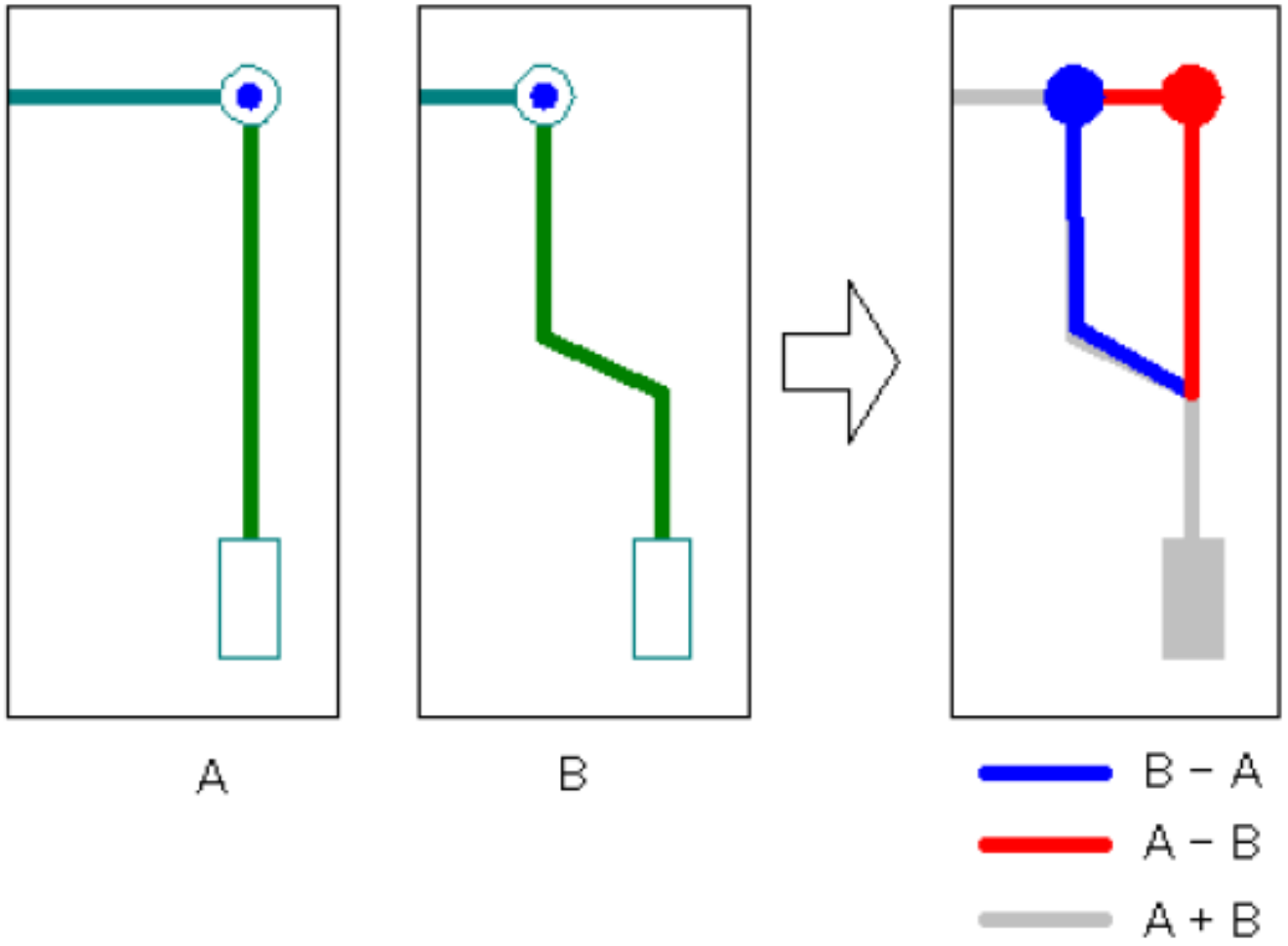


여기서 **View setting**은 **B-A**를 선택하여 수정 전 데이터에서 현재 데이터의 차이점만 확인하도록 합니다.

© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Differential View

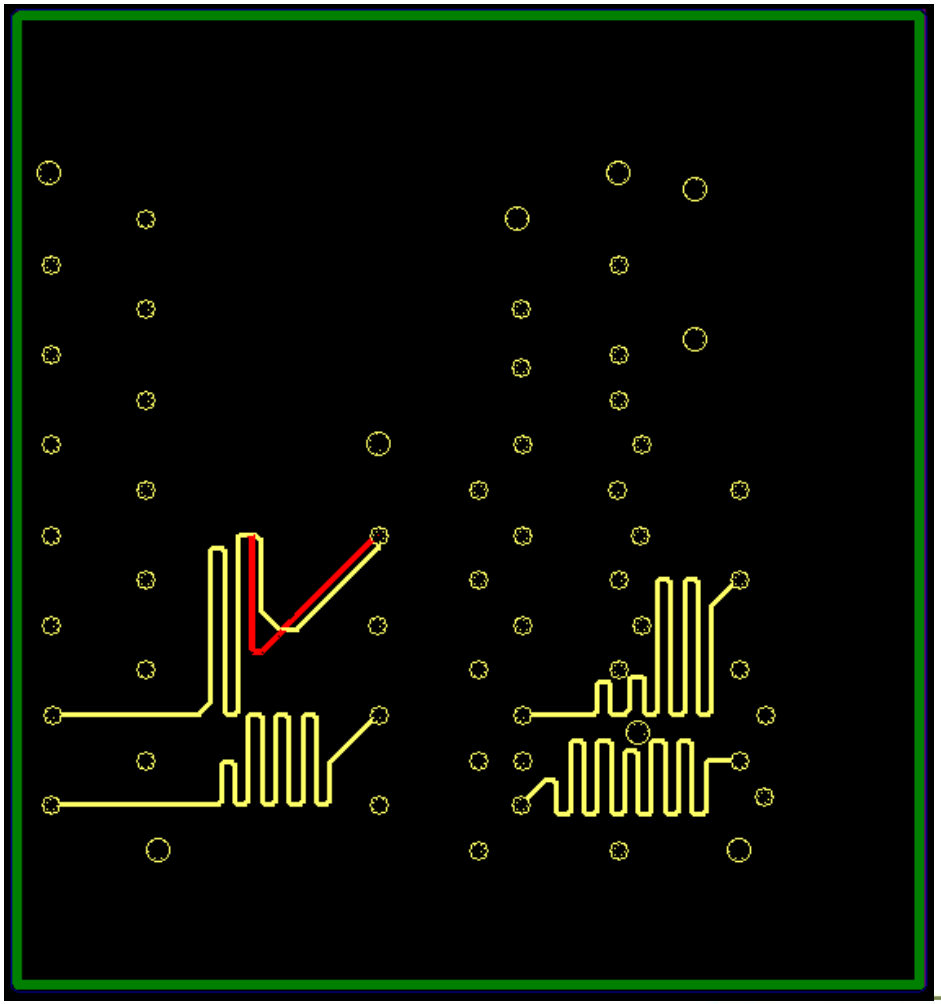
참고.



© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Differential View

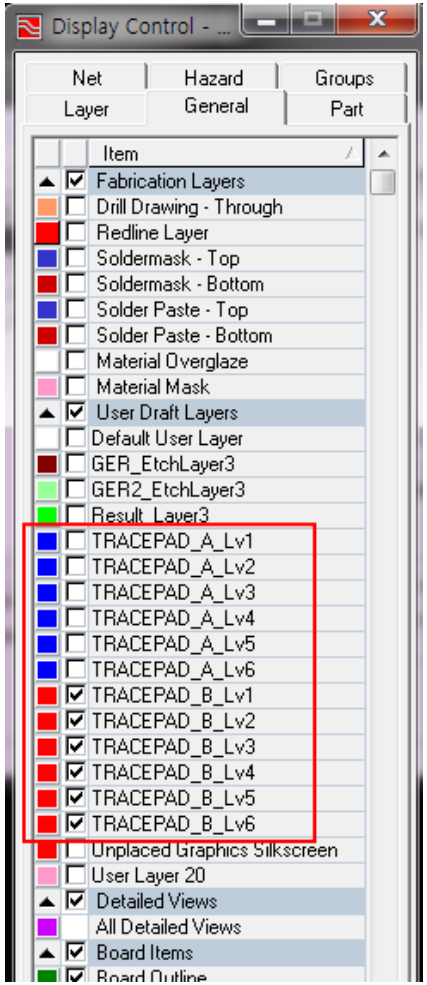
6 비교 예제입니다. 빨간색 부분이 수정 전 데이터의 형상입니다.



© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Differential View

7 **Differential View**를 실행했을 때 **Display Control**에 추가된 **Layer**입니다.

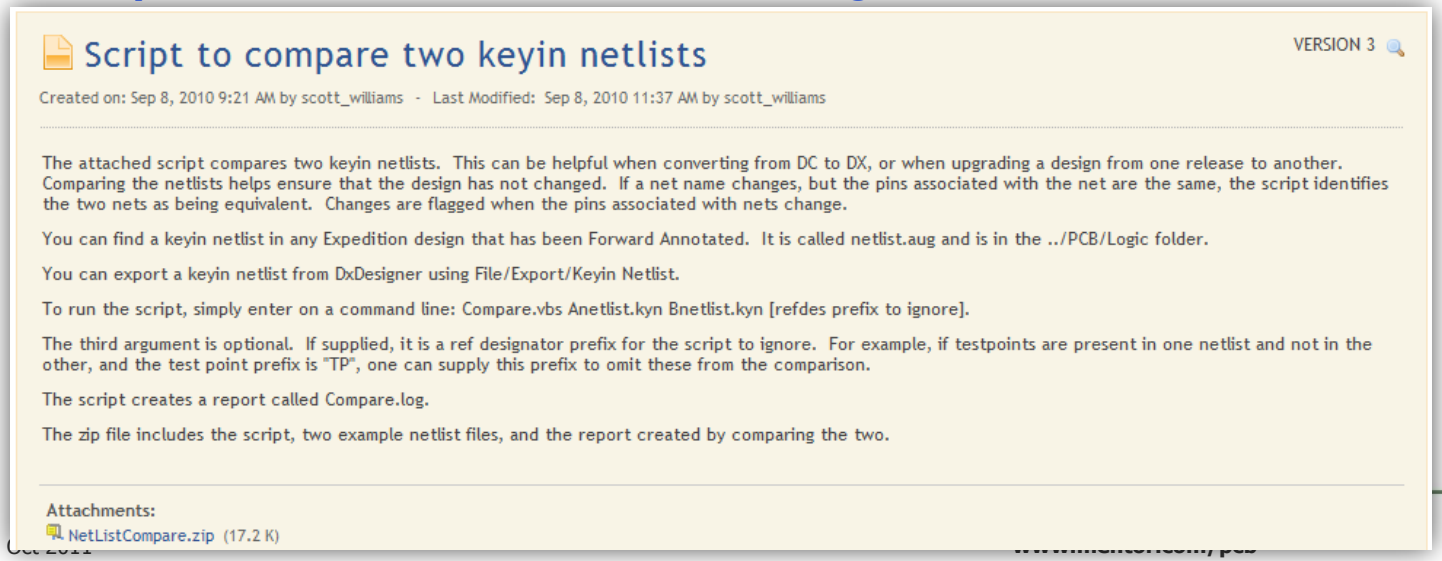


© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

NETLIST 비교

Script to compare two keyin netlists

- **PCB keyin netlist**
  - — Forward Annotation이 완료되면 ../PCB/Logic/netlist.aug 파일 생성
- **Schematic keyin netlist**
  - — DxDesigner using File > Export > Keyin Netlist
- 사용법
  - — Compare.vbs Anetlist.kyn Bnetlist.kyn [refdes prefix to ignore]
- 스크립트 다운로드
  - — http://communities.mentor.com/mgcx/docs/DOC-2207



Script to compare two keyin netlists

VERSION 3

Created on: Sep 8, 2010 9:21 AM by scott_williams - Last Modified: Sep 8, 2010 11:37 AM by scott_williams

The attached script compares two keyin netlists. This can be helpful when converting from DC to DX, or when upgrading a design from one release to another. Comparing the netlists helps ensure that the design has not changed. If a net name changes, but the pins associated with the net are the same, the script identifies the two nets as being equivalent. Changes are flagged when the pins associated with nets change.

You can find a keyin netlist in any Expedition design that has been Forward Annotated. It is called netlist.aug and is in the ../PCB/Logic folder.

You can export a keyin netlist from DxDesigner using File/Export/Keyin Netlist.

To run the script, simply enter on a command line: Compare.vbs Anetlist.kyn Bnetlist.kyn [refdes prefix to ignore].

The third argument is optional. If supplied, it is a ref designator prefix for the script to ignore. For example, if testpoints are present in one netlist and not in the other, and the test point prefix is "TP", one can supply this prefix to omit these from the comparison.

The script creates a report called Compare.log.

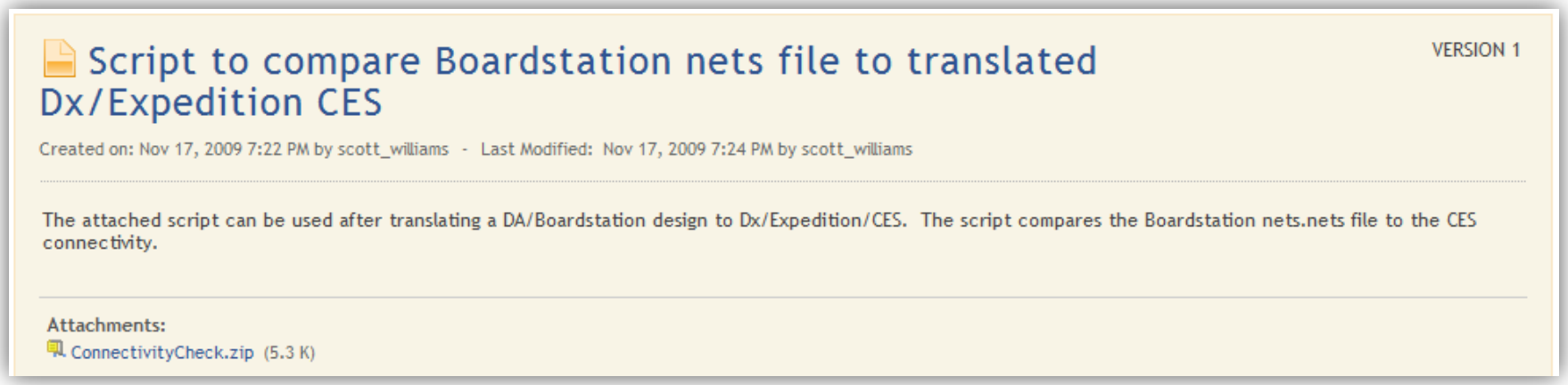
The zip file includes the script, two example netlist files, and the report created by comparing the two.

Attachments:

NetListCompare.zip (17.2 K)

Script to compare Boardstation nets file to translated Dx/Expedition CES

- The script compares the Boardstation nets.nets file to the CES connectivitySchematic keyin netlist
- 스크립트 다운로드
  - http://communities.mentor.com/mgcx/docs/DOC-1820



Script to compare Boardstation nets file to translated Dx/Expedition CES

VERSION 1

Created on: Nov 17, 2009 7:22 PM by scott_williams - Last Modified: Nov 17, 2009 7:24 PM by scott_williams

The attached script can be used after translating a DA/Boardstation design to Dx/Expedition/CES. The script compares the Boardstation nets.nets file to the CES connectivity.

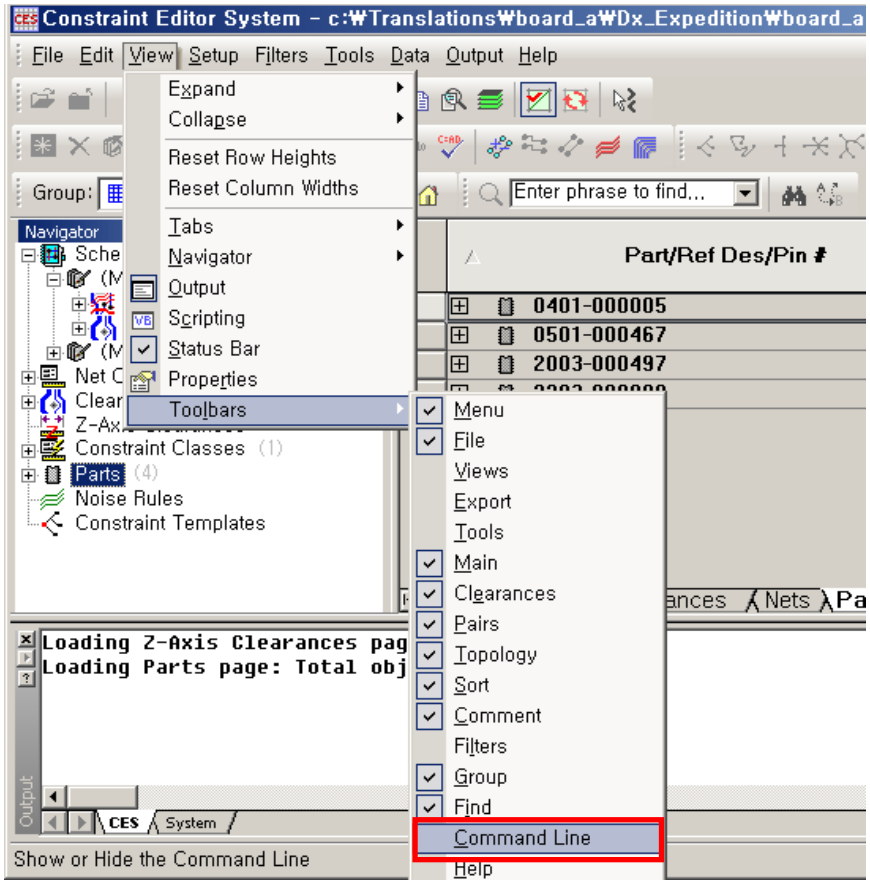
Attachments:

ConnectivityCheck.zip (5.3 K)

© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Script to compare Boardstation nets file to translated Dx/Expedition CES

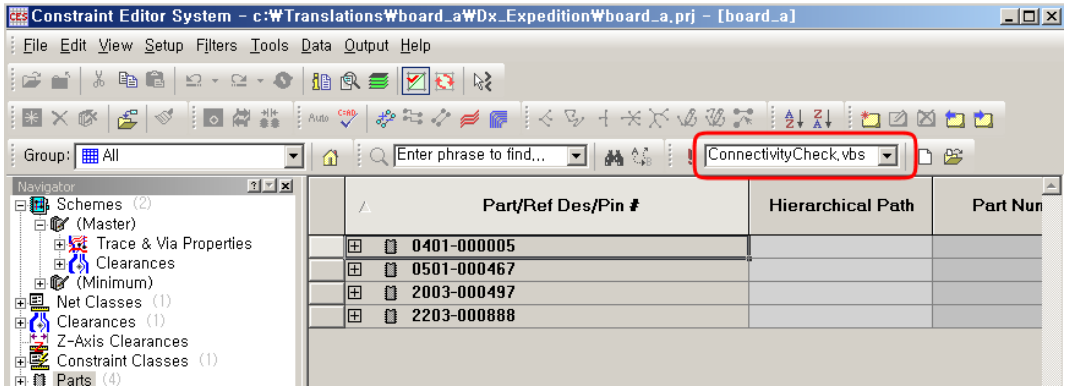
ConnectivityCheck.zip 파일을 압축 해제한 후 ***ConnectivityCheck.vbs*** 파일을 **WDIR** 디렉터리인 **C:\WDIR**에 위치시킨다.
CES를 실행해서, **Command Line** 툴 바를 **Open**한다.



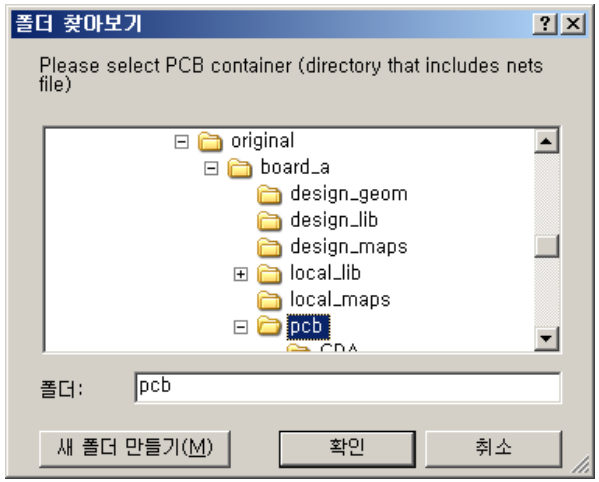
© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Script to compare Boardstation nets file to translated Dx/Expedition CES

Command Line 툴바에서 **ConnectivityCheck.vbs** 를 입력하고 **Enter**한다.



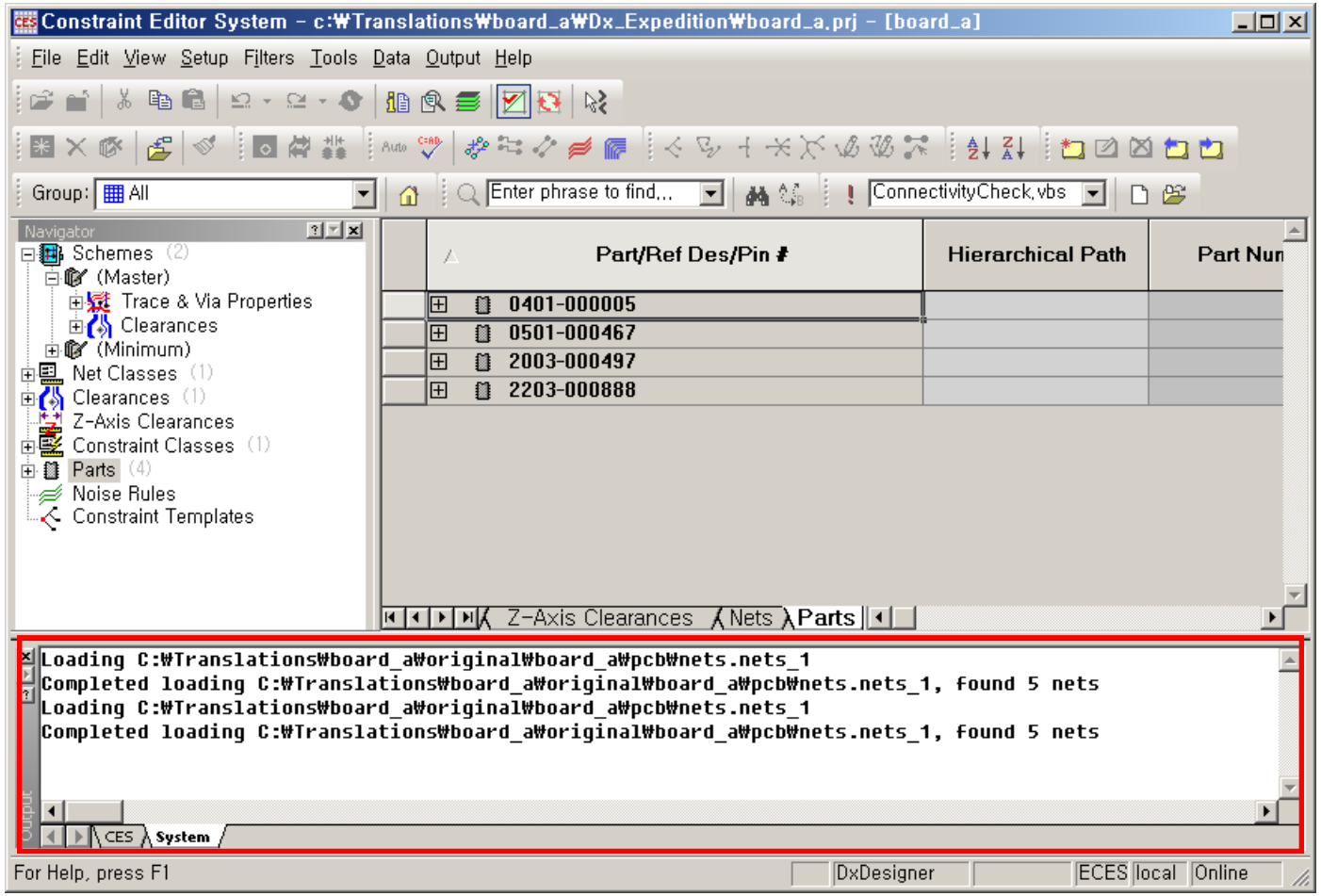
Board Station PCB 폴더를 선택하는 윈도우에서 해당 소스 디자인의 **PCB** 폴더를 선택한다.



© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Script to compare Boardstation nets file to translated Dx/Expedition CES

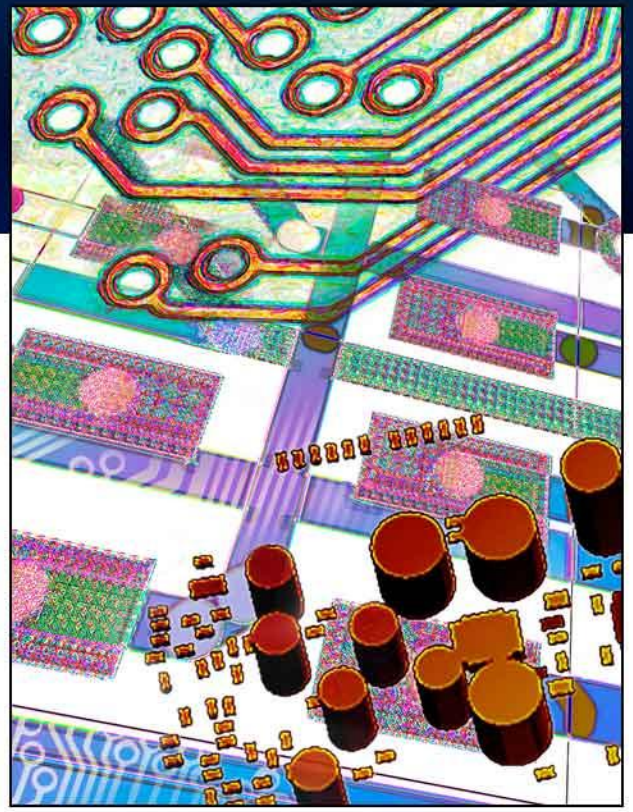
비교 실행 결과가 **Output** 윈도우 **System** 탭에 출력된다.



© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

www.mentor.com

Setting up Oracle Database for DxDatabook



Glenn Son

PCB Team Korea

October 2011

Mentor Graphics®

Agenda

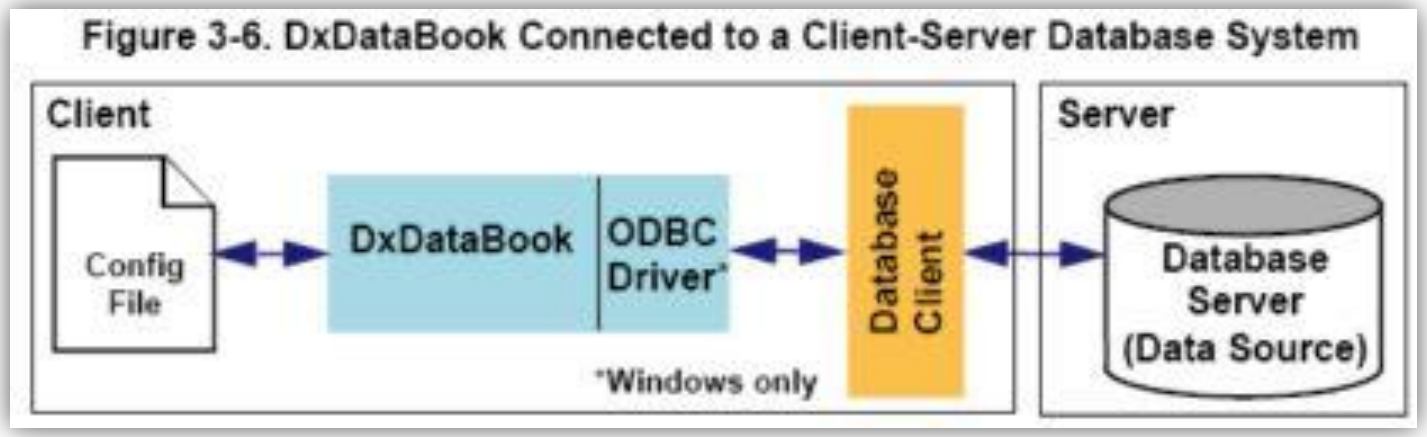
- Oracle 사용환경구성
- Oracle Client 설치 및 설정
- Oracle database 구성 최적화

© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

ORACLE 사용환경구성

Oracle 사용환경구성

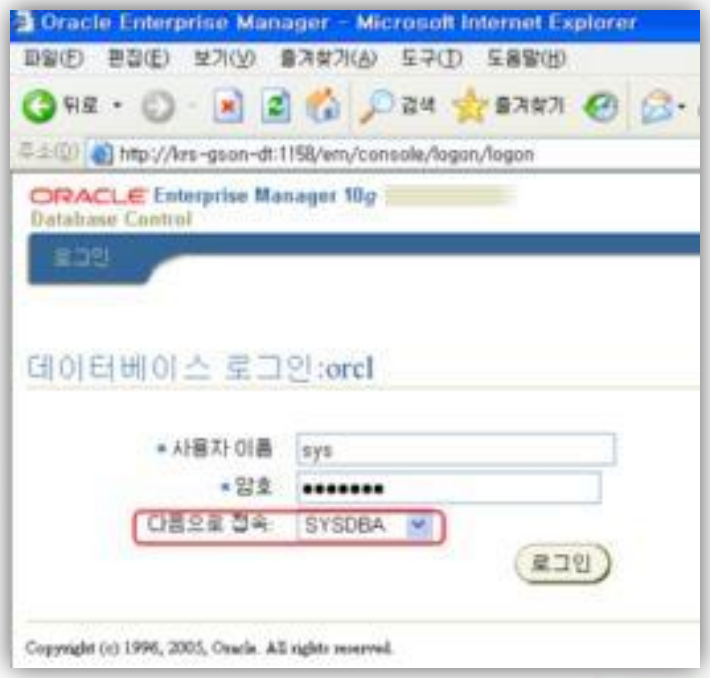
- **DxDatabook**과 **Oracle** 데이터베이스를 **ODBC**로 연계해서 사용환경을 구성하는 방법
- "제어판 > 관리 도구 > 데이터 원본 **(ODBC)**" 등록 불 필요



© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Oracle Server 환경 설정

- **Oracle** 데이터베이스로 사용할 서버에 **Oracle 9i or 10g**를 설치한다.
- **Tablespace**와 **Schema**를 생성하고, 사용자 권한을 설정한다. **Web**을 통해 제공되는 **Oracle Enterprise Manager**를 사용하면 용이하게 구성할 수 있음. **Login**시 **SYSDBA**로 접속하여야 함.



© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Oracle Server 환경 설정

- **Library Partition별로 Table를 생성합니다.**

```
CREATE TABLE "ROMEO"."micro_proc" ( "ID" VARCHAR2(100),
"PART_NUMBER" VARCHAR2(255), "PART_NAME" VARCHAR2(255),
"PART_LABEL" VARCHAR2(255), "DESCRIPTION" VARCHAR2(255),
"STATUS" VARCHAR2(50), "SYMBOL" VARCHAR2(255)) TABLESPACE
"MY_DB";
```


© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Oracle Server 환경 설정

- **PDB**에서 **csv** 파일을 추출하거나, **LMS Catalog**에서 **csv** 파일을 추출한다.
- **MDB**를 사용하고 있으면 **Oracle**에서 **Migration Kit**를 제공하고 있으므로 해당 프로그램을 사용해서 **MDB**를 **Oracle**로 **Migration** 해도 된다.
  - http://www.oracle.com/technology/tech/migration/focusareas/access.html

© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Oracle Server 환경 설정

- 추출된 **csv** 파일을 **sqlldr** 명령을 사용해서 해당 **Table**에 **Import** 한다. **sqlldr**은 **control** 파일이 필요하다.
  - **Control 파일 micro_proc.ctl 예)**

```
load data
infile 'c:\oracle\micro_proc.csv'
replace
into table "micro_proc"
fields terminated by '^'
trailing nullcols
( ID, PART_NUMBER, PART_NAME, PART_LABEL, DESCRIPTION,
STATUS, SYMBOL )
```

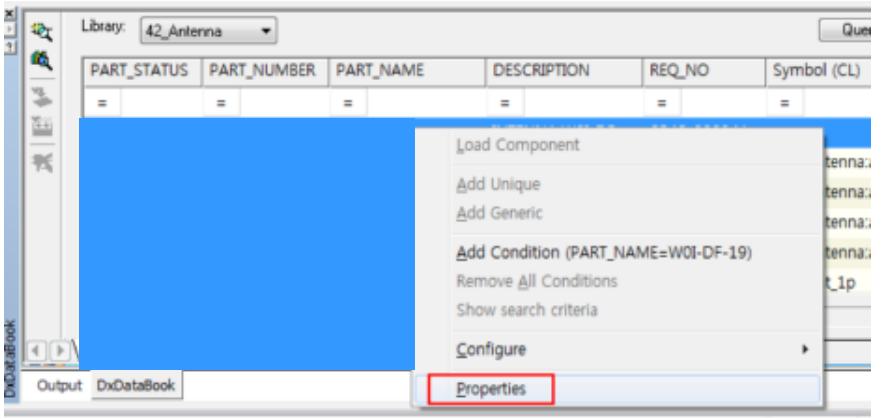
  - **sqlldr 명령 예)**

sqlldr romeo/mentor control=micro_proc.ctl

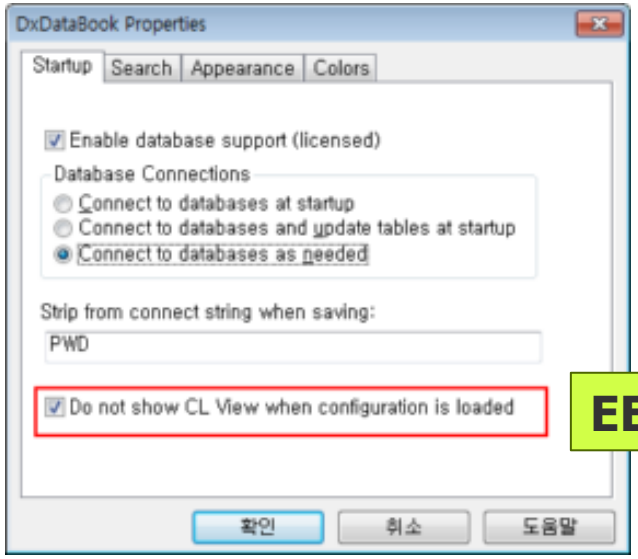
© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Tip: DxDesigner에서 CL View를 보이지 않게 하는 기능

- **DxDatabook에서 Properties 명령을 실행합니다.**



- **"Do not show CL view…." 옵션을 체크해 주면 CL View가 보이지 않게됩니다.**



EE7.9.1 버전에서부터 적용

© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Tip: DxDesigner에서 CL View를 보이지 않게 하는 기능

- 프로젝트 최초 생성시 자동으로 **CL View**를 보이지 않게 적용하는 환경구성

Corp_Config\DxDesigner.xml 파일을 텍스트 에디터로 열어서
<settings> 항목 내에 **DXDB_HIDE_CL_VIEW**의 **value**를 **1**로 정의해 줍니다.

```
<settings>
…
…
<key name="DXDB_HIDE_CL_VIEW" value="1"/>
…
…
</settings>
```

Note: Show (value="0"/) or hide (value="1"/) CL View.

© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

ORACLE CLIENT 설치 및 설정

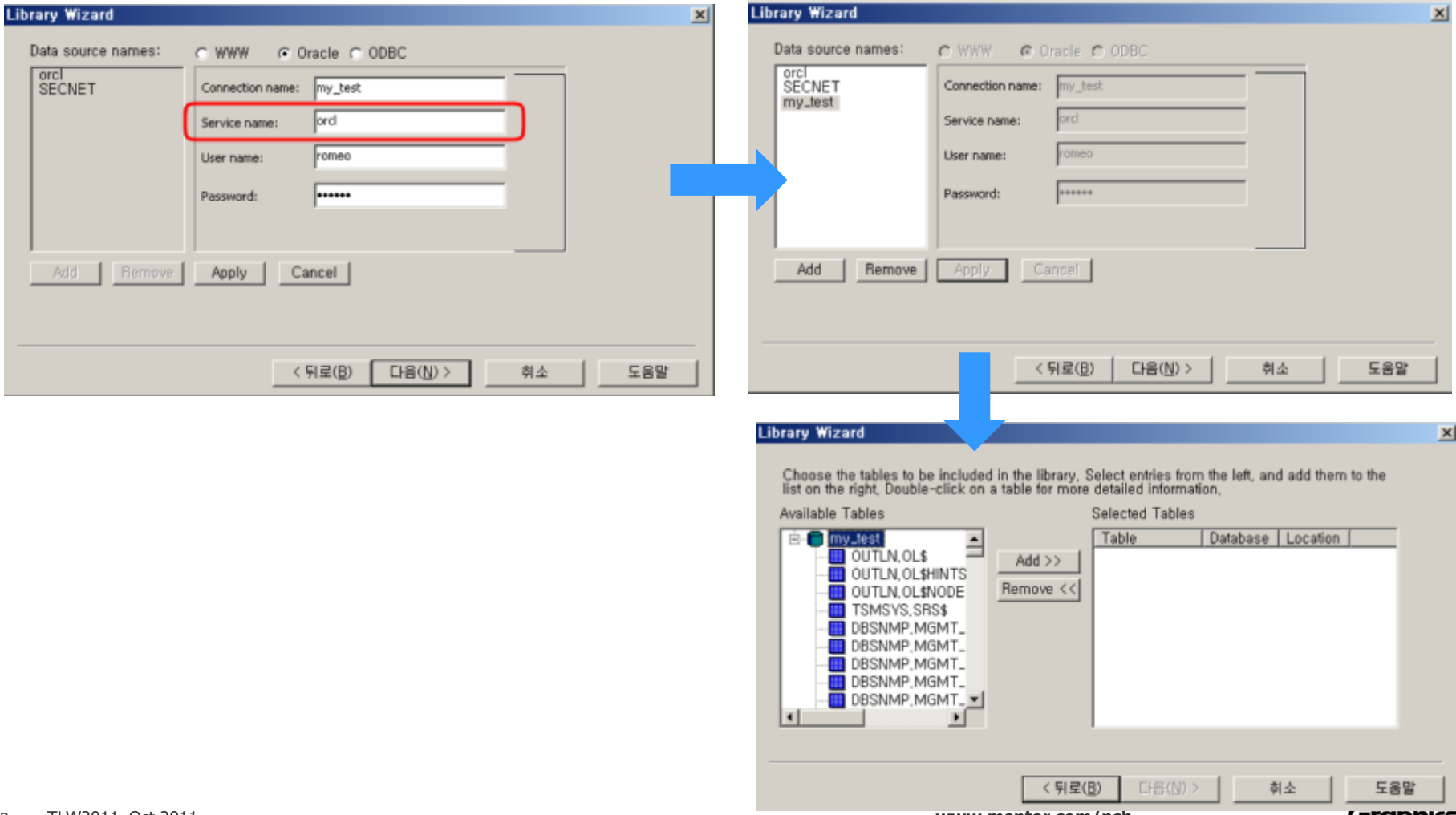
Oracle Client 설치 및 설정

- **Oracle** 서버 버전에 맞는 **Oracle Client** 프로그램을 설치한다.
  - http://www.oracle.com/technology/software/tech/oci/instantclient/htdocs/winsoft.html
- 압축파일을 다운로드 해서 **c:\oracle\instantclient** 와 같은 경로로 압축해제한다. **ODBC**까지 다운로드해서 설치한다.
- **C:\oracle\instantclient** 폴더 내에 서버에서 사용 중인 **tnsnames.ora** 파일을 복사 또는 생성해 주어야 한다.
- 환경변수에 다음과 같이 추가해 준다.
  - **PATH=c:\oracle\instantclient;<기존 값>**
  - **TNS_ADMIN=c:\oracle\instantclient**

© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

Oracle Client 설치 및 설정

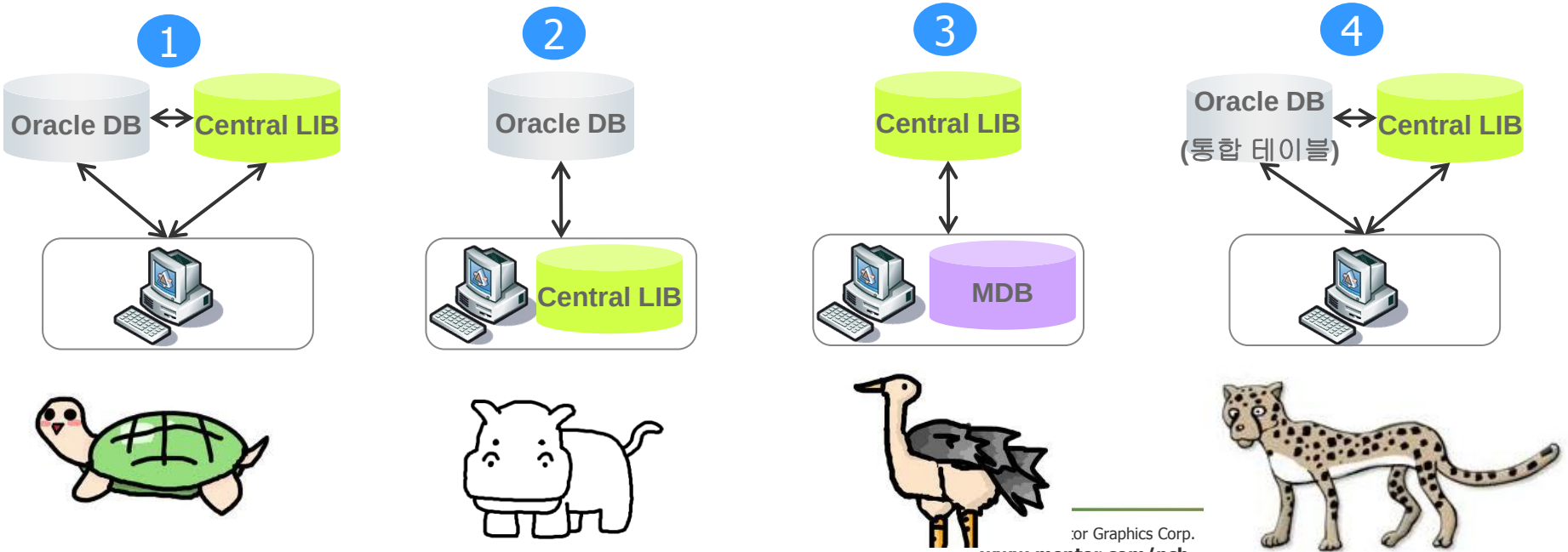
- DxDatabook Configuration 생성



ORACLE DATABASE 구성 최적화

Oracle database 구성 최적화

- 다음과 같은 환경구성을 가지고 **DxDatabook**속도비교
  1. **Oracle,** 네트워크 마운트된 **Central Library**사용
  2. **Oracle,** 로컬 **PC**로 복사된 **Central Library**사용
  3. 로컬에 보관된 **MDB,** 네트워크 마운트된 **Central Library**사용
  4. **Oracle,** 네트워크 마운트된 **Central Library**사용 **(**단**,** 부품 종류별로 구분된 **Oracle DB Table**을 하나의 **Table**로 통합하고**, PDB**는 부품 종류별로 파티션이 구분되어 있어야 하므로 **View Table**을 부품 종류별로 구성**)**



or Graphics Corp.
www.mentor.com/pcb

© 2010 Mentor Graphics Corp.
www.mentor.com/pcb

www.mentor.com