



BluePrint

For Printed Circuit Boards

Importing Data from ECAD Systems.....	1
Mentor PADS Layout 2005 ASCII EXPORT	1
Mentor Expedition 2005 ODB++ EXPORT	3
Cadence Allegro ODB++ EXPORT	5
Trademarks	7

Importing Data from ECAD Systems

This is a guide that is intended to provide you all you need to know to be able to produce the required file(s) from your ECAD System for import into BluePrint. This guide is organized by ECAD System. So simply find your ECAD system in the list below, and follow the instructions provided. Additional ECAD systems will be added as soon as possible (Mentor Boardstation, Zuken, Cadence OrCAD)

For more in-depth information on generating the necessary data file(s) from your ECAD system for import into BluePrint, or for more information on any BluePrint functionality, please see BluePrint's online Help or contact us at sales@downstreamtech.com.

Mentor PADS Layout 2005 ASCII EXPORT

The PADS ASCII data format supports storage of the complete PCB CAD data elements from the Mentor PADS Layout PCB design system. Layout data and netlist data is used for fabrication and assembly documentation for Printed Circuit Board manufacturing. The PADS Layout data has layer based graphical data that is used for lithographic manufacturing of layers of circuit boards.

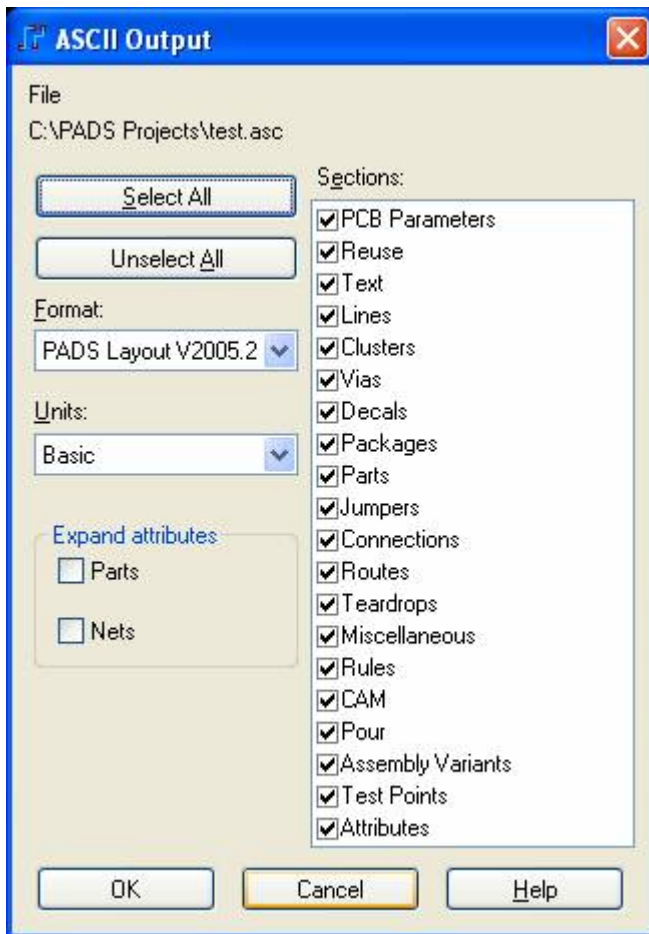
The PADS Layout data also has associated EDA data that assigns nets and components to the fabrication graphics. The PADS Layout data format stores Component Outlines in part decals for top mounted and bottom mounted components. These Component Outlines are stored on various decal layers for top components and bottom components.

BluePrint-PCB requires the full PADS ASCII data output for generating fabrication and assembly drawings. BluePrint-PCB will generate its own intelligent reference designators; however, BluePrint-PCB can optionally display the PCB design reference designator as unintelligent graphics. In addition, BluePrint-PCB can display the Component Outline layers as intelligent component outlines based on the part decal layers for Top and Bottom PCB Views in Assembly drawings.

PADS Layout Command Instructions:

[These instructions are for PADS Layout version 2005, Service Pack 3.]

- Start PADS Layout. Use **File > Open** to load the desired PCB design.
- Select **File > Export ...** to open the ODB++ export dialog.
- Select the file name for the PADS ASCII file to be exported.



The above settings will generate PADS ASCII output sufficient for BluePrint-PCB use. Important settings for BluePrint-PCB include the following:

- ✓ **Sections**
 - All sections should be selected for PADS Layout output. BluePrint-PCB will utilize various fields from multiple sections in the PADS Layout ASCII format
- ✓ And more....

Mentor Expedition 2005 ODB++ EXPORT

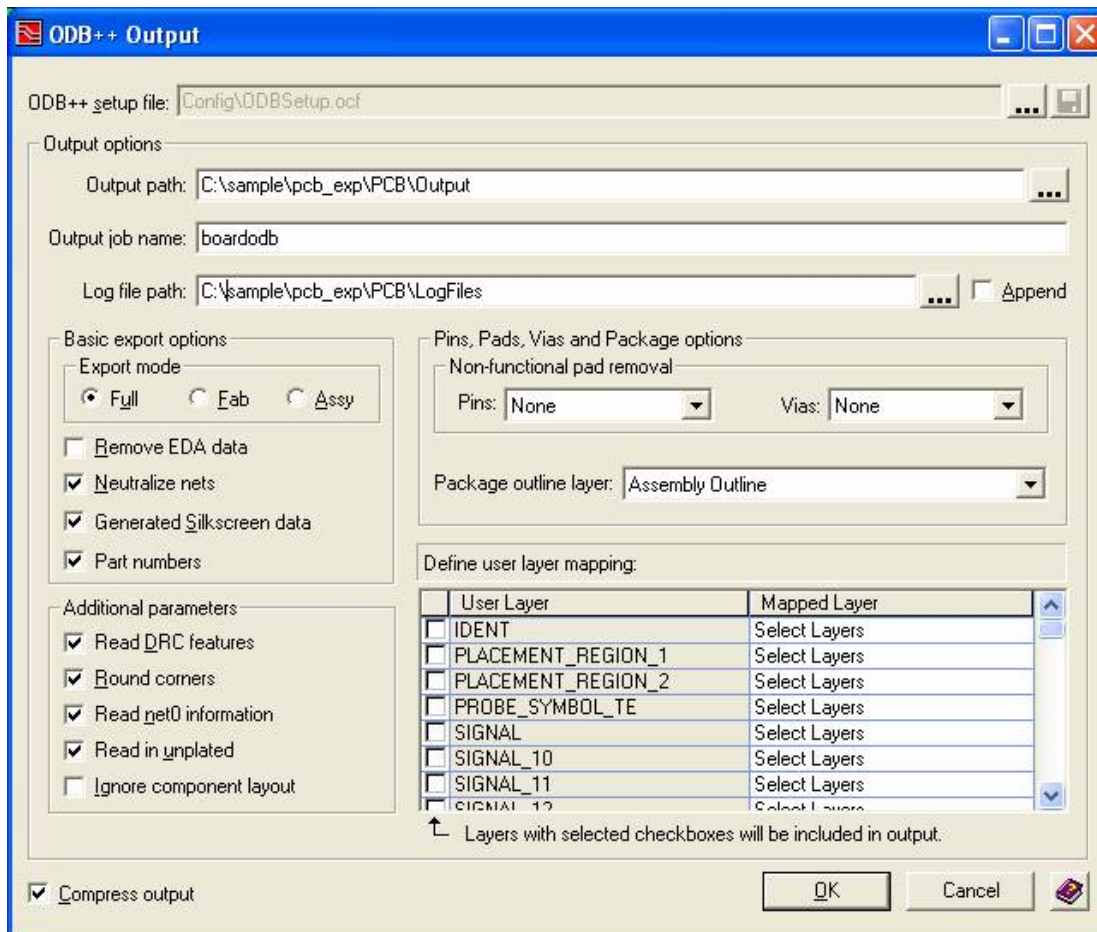
ODB++ database format supports storage of various types of fabrication and assembly data for Printed Circuit Board manufacturing. The ODB++ fabrication data has layer based graphical data that is used for lithographic manufacturing of layers of circuit boards. The ODB++ fabrication data can also have associated EDA data that assigns nets and components to the fabrication graphics. The ODB++ database format also stores Component Outlines for top mounted and bottom mounted components. These Component Outlines are stored on special ODB++ specific layers for top components and bottom components in the ODB++ data format.

Blueprint-PCB requires the ODB++ fabrication data with the associated EDA data. In addition, Blueprint-PCB uses the Component Outline layers to create intelligent component outlines for Top and Bottom PCB Views in Assembly drawings.

Expedition Command Instructions:

[These instructions are for Expedition Version 2005, Service Pack 1.]

- Start Expedition. Use **File > Open** to load the desired PCB design.
- Select **Output > ODB++...** to open the ODB++ export dialog.

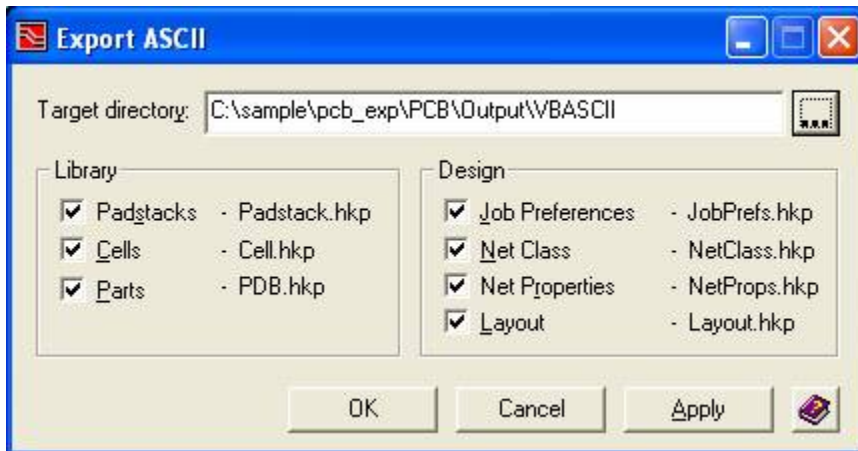


The above settings will generate ODB++ output sufficient for Blueprint-PCB use. Important settings for Blueprint-PCB include the following:

- **Basic Export Settings**
 - o **Export Mode** – “**Full**” – export all fabrication and associated EDA data
 - o **Remove EDA data** – must be unchecked; EDA data is required
 - o **Neutralize nets** – must be unchecked; nets are required
 - **Pins, Pads, Vias, Package Options**
 - o **Package Outline layer** – “**Assembly Outline**” – the assembly outline data is used for Top and Bottom PCB Views for Assembly drawings in Blueprint-PCB
- Select **OK** to export the ODB++ compressed file.

In order to obtain more precise assembly outline graphics, in addition to ancillary layer and drill data, you can optionally export Expedition ASCII data files which can then be imported into Blueprint-PCB.

- In Expedition, select **File > Export > ASCII...** to export the ASCII data files.



- Check all the selections in the above dialog to output all the ASCII files.
- Select **OK** to export the Expedition ASCII files.
- In Blueprint-PCB, use the **File > Import > Expedition ODB++...** command.
- Select the Expedition ODB++ file.
- Select the **Advanced >>** button to optionally select the folder with the Expedition ASCII files.

Cadence Allegro ODB++ EXPORT

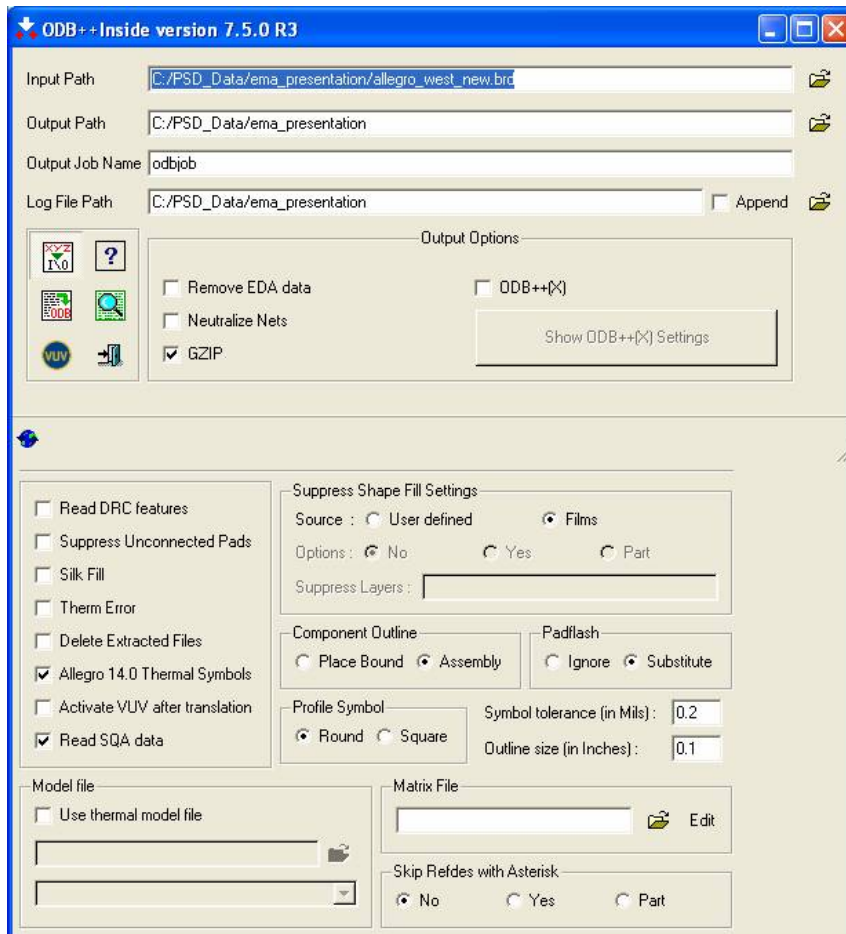
ODB++ database format supports storage of various types of fabrication and assembly data for Printed Circuit Board manufacturing. The ODB++ fabrication data has layer based graphical data that is used for lithographic manufacturing of layers of circuit boards. The ODB++ fabrication data can also have associated EDA data that assigns nets and components to the fabrication graphics. The ODB++ database format also stores Component Outlines for top mounted and bottom mounted components. These Component Outlines are stored on special ODB++ specific layers for top components and bottom components in the ODB++ data format.

BluePrint-PCB requires the ODB++ fabrication data with the associated EDA data. In addition, BluePrint-PCB uses the Component Outline layers to create intelligent component outlines for Top and Bottom PCB Views in Assembly drawings.

Allegro Command Instructions:

[These instructions are for Allegro version 15.7 and ODB++ Inside version 7.5.0 R3.]

- Start Allegro. Use **File > Open** to load the desired PCB design.
- Select **File > Export > ODB++ Inside...** to open the ODB++ export dialog.
- Select the “XYZ I/O” button to show the extended options.



The above settings will generate ODB++ output sufficient for BluePrint-PCB use. Important settings for BluePrint-PCB include the following:

- **Output Options**
 - o **Remove EDA data** – must be unchecked; EDA data is required
 - o **Neutralize nets** – must be unchecked; nets are required
 - o **ODB++(X)** – must be unchecked; ODB++ XML output is not supported by BluePrint-PCB
- **Component Outline**
 - o The **“Assembly”** radio button must be selected; the assembly outline data is used for Top and Bottom PCB Views for Assembly drawings in BluePrint-PCB

Trademarks

BluePrint-PCB is a registered trademark of DownStream Technologies, LLC.

All other registered or unregistered trademarks referenced herein are the property of their respective owners, and no trademark rights to the same are claimed.