DBX Power Table
User's Guide

This document contains information about the Power Table DBX utility. This document is formatted so that it is suitable for both viewing on your screen and printing hardcopies. To print this document, choose the File Print command while the document is loaded in Microsoft Word or WordPad.

Contents

1.0 Introduction to Power Table

A Power Table is a collection of lines and text organized to help the designer easily determine the nets that are associated with power pins in a given schematic. A power pin is defined as component pin with its Electrical Type set to Power. This includes hidden pins (pins that are part of a component but not displayed on any symbol of that component). For more information on setting a pin's Electrical Type or hidden pins, see P-CAD 2004 Library Manager Users Guide and Reference.

1.1 Overview

The Power Table program reads component, attribute and net information from an open Schematic design. This data can then be selectively used to place a Power Table in that design. The user can decide which data will be contained in the table and set various output options. Power Table is written using DBX, P-CAD's Application Programming Interface. Capabilities included in this application are for use with P-CAD Schematic V13 or later.

1.2 Running Power Table

Before running Power Table, run the Schematic program and open the design file for which the table is to be generated. Start the Power Table program by running PowerTab.exe from the Start, Run command line.

2.0 Gathering Design Info

Upon startup you will be presented with the Power Table dialog box. Notice the Run button is grayed out at this time. Power Table has not collected design information yet, so Run is not a valid option.

2.1 Hidden Pins Only

Before pressing the "Gather Design Info" button, decide if your Power Table will contain only hidden power pins. If this option is unchecked all power pin information will be included in the table. If you change the state of this option you should click the "Gather Design Info" button to refresh the design information.

2.2 Gather Design Info.

When pressed, the "Gather Design Info." button will establish a connection with the schematic design and extract text styles, sheet names, component refdes, component attributes and net names. These items will then show up in the various list boxes for you to use in your Power Table.

The "Gathering Design Info. Please wait." message will display after pressing this button. Power Table is busy looking through the Schematic design file for table data. This message will go away when the process is finished. The larger the design, the longer this process will take.

3.0 Specifying Table Data

After Power Table is done "Gathering Design Info", the next step is to select the components, attributes and nets to be included in the Power Table.

3.1 Selecting Components for the Table

To select which components will be in your power table, click on the Components tab.

There are two list boxes: **Components not Selected** is a list of components with power pins that will not appear in the power table. **Components Selected** is a list of components with power pins that will appear in the power table.

Use the **Add**, **Remove**, **Add All** and **Remove All** buttons to move components from one list to the other. Add the reference designators to the list in the order that you wish them to appear in the table.

3.2 Selecting Attributes for the Table

To select which attributes will be in your power table, click on the Attributes tab.

There are two list boxes: **Attributes not Selected** is a list of attributes that will not appear in the power table. **Attributes Selected** is a list of attributes that will appear in the power table.

Use the **Add**, **Remove**, **Add All** and **Remove All** buttons to move attributes from one list to the other. Add the attributes to the list in the order that you wish them to appear in the table.

3.3 Selecting Nets for the Table

To select which nets will be in your power table, click on the Nets tab. There are two list boxes: **Nets not Selected** is a list of nets with power pins that will not appear in the power table. **Nets Selected** is a list of nets with power pins that will appear in the power table.

Use the **Add**, **Remove**, **Add All** and **Remove All** buttons to move nets from one list to the other. Add the net names to the list in the order that you wish them to appear in the table.

4.0 Specifying Table Placement Options

To select table placement options, click on the Options tab. You will be presented with the dialog box below:

Options are as follows:

- **Column Spacing,** specify the table's column spacing in mils.
- **Hidden Pins Only,** used in conjunction with the "Gather Design Info." button to filter out non-hidden power pin from the Power Table.
- **Row Spacing,** specify the table's row spacing in mils.
- **Sheet Name,** pick the name of the sheet that the table will be placed on.
- **Text Style,** pick the text style name that will be used in the table.
- **Title,** type in the Power Table Title, this text will be centered at the top of the table.
- **UL X Origin,** specify the upper left x axis table corner in mils.
- **UL Y Origin,** specify the upper left y axis table corner in mils.

5.0 Generating the Power Table

Once you have selected Components, Attributes, Nets and Options for the table press the Run button. A Power Table will be added to the open Schematic design.

If an error occurs, the program will halt and an error message will be displayed. Some errors will result in the placement of an incomplete Power Table in your schematic design. Delete the incomplete Power Table from the design, correct the problem and try again.

Note: If changes are made to the Schematic design after Power Table generation, you will need to delete the table from the design and run Power Table again to ensure accuracy.