

# *Introduction to Electronic Circuit Design*

Richard R. Spencer and Mohammed S. Ghausi

## Instructions for installing the files on this CD assuming that the introECD installer did not work.

This README file is specifically for the CD that accompanies the text, the original README file that accompanies the MicroSim installation disk is included at the end of these notes. *Please read and follow these instructions carefully or you will not be able to make use of the example files included on this CD.*

Be sure that you have administrator privileges on the machine you are installing this software on.

**This CDROM contains the following (you must install these items in the order listed):**

- 1** The *installer* for the MicroSim evaluation version of *DesignLab* release 8. You should install this program using the default directory suggested by the installer (on your C drive). If you install it anywhere else, the library links in the enclosed PSPICE files will be incorrect and you will have to change them in each individual schematic file before the files will run. To install this program double-click on SETUP.EXE and follow the directions given. **Notes: 1)** Be very patient when the installer updates your registry information (near the end of the install) - it may take a couple of minutes and **if you restart during this time you can corrupt your registry**, which may require you to reinstall your operating system. **2)** Do NOT install the Adobe Acrobat Reader program included with this program - it is an old version and may cause you problems. If you need a copy of Acrobat, you can download one for free from <http://www.adobe.com/products/acrobat/>.
- 2** A folder called *IntroECD*. This folder contains all of the PSPICE simulation files, Mathematica notebooks, Matlab scripts, and Excel spreadsheets for the examples and exercises in the book. **You should put this folder at the top level on your C drive (i.e., do not put it in another folder).** The PSPICE files are further organized into folders for each chapter. There are pdf files that describe the simulation files available. **If you do not put this folder at the top level on your C drive, the library links in PSPICE will be incorrect and you will have to change them in each individual schematic file before the files will run.**

**NOTE:** After moving this folder over to your hard drive, you must change the "read only" attribute of all of the folders and files. To do so, right-click on the IntroECD folder and select "properties". Go to the "general" tab and uncheck the "Read Only" attribute and click "Apply." A dialog box should open asking if you

want to make this change in just this folder, or this folder and all subfolders and files as well; select all subfolders and files and click "OK." If this dialog box does not appear, you may have to go into the subfolders one at a time, select all files (Ctrl-A) and change the attribute (you can do this for all of the files in each folder at one time, so it won't take too long). Depending on your operating system and settings, the IntroECD folder and the other folders within it may not say they are Read Only. In this case, you may need to go into each folder one at a time, select all of the files and remove the Read Only attribute.

The IntroECD folder also contains all of the *companion sections* referred to in the text along with Appendixes C and D. These are provided as pdf files.

- 3 A folder called *Userlib* containing the custom model libraries produced for this text. After you have installed DesignLab, you should place the contents of this folder in the folder "C:\MSimEv\_8\UserLib", which was put on your system by the DesignLab installer script.

In addition to the material on this CD, you will need the MATLAB program if you want to run the MATLAB scripts included here, a copy of the Mathematica program or the Mathreader program if you want to look at the mathematica files, and Excel or a compatible spreadsheet program if you want to run the spreadsheets. A student version of MATLAB can probably be purchased from your university bookstore and the Mathreader program can be downloaded for free from [www.wolfram.com/products/mathreader/](http://www.wolfram.com/products/mathreader/).

Also, **prior to using PSPICE, you need to configure the MicroSim Schematics editor** to know about the model libraries used in the book. To do this, you first need to start the program. If you have installed it correctly, you should be able to go to the Start menu, go to "Programs" then to "DesignLab Eval 8" and then select "Schematics." Once you have started the program, go to the "Options" menu and select "Editor Configuration..." The "Editor Configuration" dialog box will open and you will see a list of libraries and some buttons. Click on the "Library Settings..." button. The "Library Settings" dialog box is now displayed and has a list of device libraries shown. The student version of the program limits you to ten libraries, so you must delete one of the existing libraries to make room for the book libraries. In the library list, select "\*BREAKOUT [.slb]" and click on "Delete". Next, click on "Browse...". The "Browse" box should now open and you should be in the directory "UserLib". You should see two files; "Book.slb" and "Book2.slb" (if you don't, be sure you performed step 3 in the installation procedure above). Open "Book.slb" and you will be returned to the Library Settings dialog box with "C:\MSimEv\_8\UserLib\Book" showing up in the "Library Name:" box. Uncheck the box labeled "Package" if it is checked. Click on the "Add\*" button. Now click on "Browse..." again, and open "Book2.slb". Uncheck the "Package" box and click on the "Add\*" button again. You can now click on the "OK" button in the Library Settings dialog box and then click on "OK" in the "Editor Configuration" dialog box. The schematic editor should now be set up to function properly with the schematic files included with the text. **NOTE:** You will see an error message from PSPICE Schematics

telling you that it can't find the files Book.plb and Book2.plb. You may safely ignore this error message, you don't need these files.

Should you ever want to have the Breakout library of schematic symbols available again, simply delete go back through this process but delete one of the other libraries and add the Breakout library. The Breakout library contains basic symbols that you can then customize. There are also other symbol libraries provided by Microsim that you can use. They are all in the folder C:\MSimEv\_8\lib.

**To run the MicroSim installation program, double-click on SETUP.EXE and follow the directions given.**

**The original MicroSim README file follows:**



This Read Me file contains last-minute information for all MicroSim programs. The information in the manuals and Help that are provided with your programs also contain essential information to help you begin using your MicroSim programs.

This Read Me file is divided into the following sections:

- Installation
- What's New with Your MicroSim Applications?
- General Information
- Tech Tips
- Documentation Errata

## **Installation**

### **Installing Release 8**

◆ There is a limit to the length of file names (for example: C:\MicroSim\Evaluation\DLab\Release8 ) when installing MicroSim programs. If you install with a file name that exceeds the maximum amount of characters allowed, towards the end of the installation, you will receive an error message or the installation will be canceled. When installing MicroSim DesignLab, there is a limit of 110 characters or less for the destination directory. When installing other MicroSim programs, the character limit is 220 or less.

### **Installing Online Manuals**

◆ When you received your MicroSim programs, you also received online manuals with the Adobe Acrobat Reader. For information about how to use the Adobe Acrobat Reader, refer to the [Adobe Acrobat Reader Online Guide](#) provided with your MicroSim programs.

**To start the reader**

- 1 In Windows 95, point to the MicroSim Program group.
- 2 Point to Accessories.
- 3 Click Online Documentation to launch the Reader.
- 4 Click the appropriate manual.

#### To view the Adobe Acrobat Reader Online Guide

- 1 In Windows Explorer, move to the [\<Acrobat Reader directory>\Help](#) directory.
- 2 Double-click [Acrobat.pdf](#).

#### Searching the online manuals

These manuals are supplied with an across-book index and search mechanism. Use the toolbar buttons shown below to start a search and view the results, respectively:



For information about how to use the search feature, refer to the Adobe Acrobat Search Online Guide contained in [\<Acrobat Reader directory>\Help](#). (In Windows Explorer, double-click [search.pdf](#).)

**Note: MicroSim recommends that you install the online manuals once, in a single location. You do not need to install the online documentation for both the registered version and the evaluation version.**

#### Installing PCBoards

◆ During installation of PCBoards, you are prompted for the [sp.pas](#) file. You must have the floppy in the drive when installing this file. On some Windows 95 machines, if you press Browse without the floppy in the drive, the installation will crash. Because Uninstall will not work after this type of crash, re-install the software over the first installation attempt.

### What's New with Your MicroSim Programs?

Refer to the manuals and Help provided with your programs for more information about what's new for Release 8.

#### Schematics

◆ If you are upgrading from a previous version (7.1 or earlier), review the following functionality changes to Schematics.

##### **Undo information is kept after a simulation is finished**

**Earlier Versions:** Undo information was unavailable from the create Netlist or Simulate commands.

**Release 8.0:** You can now undo after using these commands. (Your schematics are still automatically saved to be consistent with the simulation results).

##### **Connectivity Via Label netlisting option default setting**

**Earlier Versions:** In version 6.3, there was only a global default. All designs were controlled by the current setting. In version 7.1, Schematics stored this setting with the topmost design. Any subschematics used the setting of the topmost design.

**Release 8.0:** Instead of being controlled by the topmost design's setting, the Connectivity Via Label option (and the associated Append Page Number to wire labels option) within each subschematic is used during netlisting. Subschematics created by someone else can be used without having to know the original setting.

If you used Connectivity Via Label in hierarchical designs under version 7.1, do one of the following:

- Update each subschematic to have this option set (from the Options menu, select Restricted Operations).

Or

- Add the following option to the [SCHEMATICS] section of [MSIM.INI](#):  
[NITopSchConnViaWireLabel=ON](#)  
This option will cause netlisting to revert to using the setting from the topmost schematic.

**Note: This affects ALL schematics.**

### Zoned Borders

When you open Schematics, the default border that displays is Zoned, but you can disable the zoned borders and enable the single-line border (Outline) from previous versions of Schematics by doing the following:

- 1 From the Options menu, select Editor Configuration, then click Page Settings.
- 2 Choose Outline for each page size that you want to display a single-line border.

### Default font is Arial

**Earlier Versions:** The default font for Schematics was Modern.

**Release 8.0:** The default font is Arial. Also, the Display Preferences and Text Properties dialog boxes support only TrueType fonts. If you use Modern or another non-TrueType font, you may see text alignment differences in existing schematics due to differences in how Windows scales the fonts.

You can go back to using Modern as your default font by adding the following item to the [SCHEMATICS] section of [MSIM.INI](#):

[BASEFONTNAME=MODERN](#)

This will change the default font from Arial to Modern.

### Changing fonts for individual layers

**Earlier Versions:** The default font was changed by using Editor Configuration on the Options menu.

**Release 8.0:** You can change fonts for individual layers by using Display Preferences on the Options menu.

### Enabling and disabling toolbars and status line

**Earlier Versions:** Display Options on the Options menu enabled and disabled the display of toolbars and the status line.

**Release 8.0:** Enable Toolbars and Status Bar commands on the View menu.

### Multiple toolbars

There are now multiple toolbars you can individually enable and disable in the Toolbars dialog box. All simulation-related buttons are in their own toolbar.

### Options/Display Preferences vs. Options/Set Display Level

**Earlier Versions:** Set Display Level on the Options menu globally enabled and disabled the display of schematic items such as markers and title blocks.

**Release 8.0:** You can use Display Preferences on the Options menu to configure colors, fonts, and line styles, as well as globally enable and disable the display of schematic items (layers).

### **Editing Graphics and Text Properties**

In addition to changing the default settings of layers in Display Preferences, you can change display properties for individual layers in the current design by doing the following:

- 1 Click to select the item.
- 2 From the Edit menu, select Graphics Properties, or select Text Properties.

You can also double-click the object to display the appropriate dialog box.

### **PRINTERLINEWIDTH option now obsolete**

**Earlier Versions:** This item could be added to the MSIM.INI file to make wires and buses thicker when printing.

**Release 8.0:** Display Preferences on the Options menu provides width control for both display and printing.

### **Block/Graphics resizing**

**Earlier Versions:** Blocks in the schematic editor and graphics in the symbol editor were resized by holding down the <SHIFT> key and dragging with the right mouse button.

**Release 8.0:** You can resize blocks and graphics by using handles. Select a block or graphic to display the handles and drag a handle to resize it to the appropriate size.

### **Automatic pin creation on hierarchical blocks**

If you associate an existing schematic or XNF file with a block, pins corresponding to the interface ports with the subschematic or XNF file are automatically created and placed on the block.

### **Hierarchical schematics with multi-signal interface ports**

**Earlier Versions:** Pin names on blocks and symbols had to match exactly the labels used on underlying interface ports, and vice versa.

**Release 8.0:** Netlisting of hierarchical symbols and blocks has been extended to allow bus pins to connect to individual signal ports.

The following scenarios can happen:

- A block with 1 pin A[0-2] and a subschematic can have 3 interface ports A0, A1, A2.
- A block with 3 pins B0, B1, B2 and a subschematic can have 1 interface port B[0-2].

### **Selecting items**

In Release 8.0, selecting items includes the following functionality:

- To select all segments of a wire or bus, hold down the <ALT> key as you select a segment.
- To select an extended area, hold down the <SHIFT> key at the same time as selecting an area.
- You cannot select a part and one of its attributes at the same time.
- When you select wire and bus attributes, in addition to the segment that owns it becoming colored, a dotted line is drawn from the attribute back to the segment that owns it.

### **Part Browser - wildcard searching**

**Earlier Versions:** To search for parts using wildcards in part names, previously you needed to enter the pattern (e.g., 74\*) and press <ENTER> to see the list of names.

**Release 8.0:** You do not need to press <ENTER>; as you type in the name, matching items are displayed in the list automatically.

### **Edit/Convert Block & File/Symbolize**

These menu commands have been changed to include the necessary path information, to ensure that when the symbol created is used, the associated subschematic can be found.

### **Searching for subschematics and local symbol libraries**

**Earlier Versions:** In versions 6.3 and earlier, subschematics and local symbol libraries were previously searched for in the following order until found:

- the directory of the containing schematic
- along the library path

**Release 8.0:** Searching for subschematics was changed to the following:

- if referenced by a hierarchical block, the directory of the schematic containing the block
- if referenced by a hierarchical symbol, the directory of the symbol library containing the symbol

This change was made so that subschematics that were associated with hierarchical symbols could be put in the same directory as the symbol library and having subschematics with the same name in other directories would not be inadvertently used.

### **Updating schematics from PLSyn**

The defaults for the AUTOPLACEX and AUTOPLACEY .INI items that control placement when updating the schematic from PLSyn have been changed from 100 and 150 to 200 and 200, respectively. This may cause pages created by adding PLDs from PLSyn to have a slightly different appearance.

### **NC symbol in Schematics**

**Earlier Versions:** The NC symbol was removed from [port.slb](#) because it could create disconnects within subcircuits that produced inaccurate simulation results.

**Release 8.0:** If you have schematics using this symbol, replace it with the new NoConnect symbol (attaching it directly to the pin of the part) to indicate that the pin is intentionally left unconnected. The old NC symbol is in [obsolete.slb](#) (the library that contains symbols that is not being shipped with this release). If you still want to use NC, you can configure [obsolete.slb](#) by using Editor Configuration on the Options menu.

The [obsolete.slb](#) is located in the `\MSim_8\lib` directory.

## **General Information**

### **Printing Online Manuals**

◆  
When printing the online manuals, you can print the whole manual, the current page, or a range of pages. If you are having difficulty printing the whole manual, there is a chance that your printer doesn't have enough memory to sustain the size of the .pdf file. In this case, use the page range printing option.

## **Schematics**

◆

### **Warnings are created for parts not included in the simulation**

When creating a simulation netlist, Schematics now generates warnings for parts that are not included in the simulation. Parts must have a **TEMPLATE** attribute with a value to be included in the simulation. To suppress these warnings for parts that are layout specific, such as connectors, you can add an attribute **MODEL=\*IGNORE\*** to the symbol. We have done this for the symbols for connectors in connect.slb.



The OrCAD Translator does not translate signal names with a backslash at the end of the signal name correctly, e.g., **OUT\** is not correctly translated.



### **Troubleshooting**

If you get an "Unable to start application" error when starting a simulation or viewing simulation results, but you see PSpice or Probe start, try the following:

In the [SCHEMATICS] section of the MSIM.INI file, add the following line:

**WaitForApp = 30**

This is the number of nanoseconds that Schematics will wait if it is not able to detect that the simulator or another MicroSim program has started. The default is 20 nanoseconds. Typically, no wait is needed, but on some systems it is helpful.



### **Unexpected word wrapping when editing textboxes**

At certain zoom levels, depending on the font, text may unexpectedly wrap when you click to edit the text. This occurs when the enclosing textbox's width is extremely close to the end of the longest line. For single-line text, this wrapping will cause all but the last word to disappear.

To correct this, make the enclosing textbox wider by dragging one of its handles.



### **Repainting problem with dialogs overlapping inserted bitmaps (Insert Picture)**

Windows 95: Inserting color bitmaps on a schematic can cause repaint problems for dialog boxes (such as File Open and Print) and other open windows on your desktop. This problem typically occurs with 16 color bitmaps with a palette.

If you do not have the latest device drivers for your video card, try converting your bitmap to a 256 color or true-color bitmap using a graphics editor (such as Microsoft (R) Paint) to correct the problem.

## **PCBoards**



### **Incorrect file names in post-processing log files**

In the post-processing log files, the file names <filename>.dlg and <filename>.glg listed in the photoplot and drill files are incorrect. Ignore the extra ".dlg" or ".glg" in the full file name.

The files are correct, but the log file report names them incorrectly.

For example:

c:\Projects\<filename>.glg.g01 (incorrect)  
c:\Projects\<filename>.g01 (correct)



### **Tools/CCT: Edit Do File**

If the layout file name is not in DOS style 8.3 format (that is, if the name has spaces in it), the menu command will not start the text editor and open the .do file. The .do file will still be sent to SPECCTRA and will be processed as usual. To edit it, start a text editor and look for a name with (normally) the first six letters of your layout file name, followed by ~1.do.



## PSpice A/D & Basics+

◆ In the libraries provided with your applications, there are a number of parts that use delay line models that are not supported by the simulation. The use of these models may produce erroneous simulation results. The names of these parts and their libraries are as follows:

- 74121, 74122, 74123 in 7400.lib
- CD4098B, CD4538B in CD4000.lib
- 74LS122, 74LS123 in 74LS.lib
- 54L121, 54L122, 54L123 in 74L.lib
- 555D (Mixed A/D model for the CMOS version of the 555 Timer) in [dig\\_misc.lib](#).

In addition, the Subckt DELAY (a generic delay line model in [dig\\_misc.lib](#)) is also not supported by the Basics+ package.

## Libraries

◆ The following are all-new libraries created for this release:

maxim	phil_bjt
phil_rf	siemens
phil_fet	zetex

The following libraries have new and/or updated models:

anlg_dev	burr_brn	diode
ediode	elantec	harris
igbt	jpwrjbt	motor_rf
nat_semi	opamps	polyfet

## Tech Tips

- ◆
- Q "I get a blue screen in MicroSim Schematics when I run it, and I can't place any parts."  
A The evaluation software should be run from the MicroSim Program group on the Start menu, rather than from the Run command. The evaluation programs use a differently named msim.ini configuration file, which needs to be specified as part of the Shortcut. This command line specification of the msim\_evl.ini file is built into the icons in the MicroSim Program group on the Start menu. An example shortcut target line (for MicroSim Schematics) is:  
[C:\MsimEval\\_8\PSCHED.EXE -i msim\\_evl.ini](#)
- Q "Can I run Release 8 under Windows 3.1/3.11?"  
A No. Release 8 will not run under Windows 3.1/3.11. Release 8 will run under Windows 95 or Windows NT.
- Q "Why is the installation taking so long?"  
A Some screen saver programs (when loaded and running in the background) can slow down the installation program by as much as a factor of 3. To improve the performance of the installation, disable your screen saver for the duration of the installation.
- Q "Should I let the installation program update my Registry?"

A If you update the Registry, you can start a MicroSim program automatically by double-clicking a file of a registered type. For example, you can double-click an .sch file and start Release 8 MicroSim Schematics by opening that file. Similarly, you can double-click on an .lib file to start Release 8 MicroSim Parts.

Q "Why is Acrobat Reader saying it can't find this file?"

A If you performed a Typical installation, the online manuals were not installed on your local hard drive. When you access an online manual, Acrobat Reader looks for the file on the MicroSim CD-ROM, and when it's not in your CD-ROM drive, you get a message that it can't find the file. To install the online manuals on your computer, select Custom installation.

Q "Can I get extra copies of the manuals?"

A You will find online copies of our manuals in a number of places: on your MicroSim CD-ROM, on the evaluation version CD-ROM, and most recently, it is accessible on our Web site (available for downloading).

Q "How do I specify trace colors in Probe using Markers in Schematics?"

A Do one of the following:

On the Simulation Toolbar, click the Select Marker Color button, then select a color, and place a marker.

Select a marker, then click the Select Marker Color button, and select its color.

Using different colors for each marker makes it easy to identify traces in Probe. If you do not specify trace colors in this way, Probe will automatically assign unique colors to each trace as it is drawn until all of the available colors are used once. After each color has been used once, Probe uses them again, but adds symbols to each trace (which are correlated to the trace variables in the legend of the plot window) to keep the traces unique.

If you mix methods (that is, specify some colors explicitly and let Probe select others), you may reach a point where two traces have the same color (and use of symbols). To correct this, from the Tools menu, select Options, and in the Use Symbols frame, select Always.

## Documentation Errata Schematics User's Guide

◆ p. 3-41: In the section "Changing the Get Recent Part List Size," the second paragraph should read as follows:

To change the length of the list, use a text editor to edit the MAXMRPLISTSIZE item in the [SCHEMATICS MRP LIST] section of the [msim.ini](#) file.

**MAXMRPLISTSIZE=<length of list>**

◆ In the schematic editor, the Snap-to-Grid option is no longer available from the Display Options dialog box under the Options menu. Enabling Stay-On-Grid (which is enabled by default) will force items to stay on grid.

◆ In the symbol editor and schematic editor, there is now a Restore All Defaults button on the Display Preferences dialog box under the Options menu. This button restores the originally installed values for all layers.

Electronic circuit design comprises the analysis and synthesis of electronic circuits. To design any electrical circuit, either analog or digital, electrical engineers need to be able to predict the voltages and currents at all places within the circuit. Linear circuits, that is, circuits wherein the outputs are linearly dependent on the inputs, can be analyzed by hand using complex analysis. Simple nonlinear circuits can also be analyzed in this way. Specialized software has been created to analyze Electronics Index. Circuit design.

Introduction. The electrons always flow in a full circuit needing to get back to where they started (ie the battery), although as we will see to different terminals of that battery. This also sounds kind of complicated, but from the point of view of designing electronic circuits (rather than designing the components themselves) you don't need to go any deeper, but I will be referring to electrons again in future. For any electronic circuit to work there must be a complete circuit. This means that there must be a connection made out of conducting material that goes in a circle from one terminal of the battery through the equipment and then back to the other terminal of the battery.

Introduction to Electronic Circuit Design - 2 volume set [Richard Spencer, Mohammed Ghausi] AbeBooks.com: Introduction to Electronic Circuit useful for many engineers even those who may never actually design a circuit because it is likely that they will fabricate, A basic understanding of circuit design is useful for many engineers -- even those who may never actually design a circuit -- because it is likely that they will Introduction to Electronic Circuit Design:United States Edition,Richard Spencer,9780201361834,Electrical Engineering, Circuits and Electronics.