Sue2 User Manual (Version 1.0)

Michael H. Perrott
http://www-mtl.mit.edu/~perrott
January 26, 2004

Table of Contents

Introduction

Sue2 is an updated version of the Schematic User Environment (SUE) package (based on version 2.4.1) written by Lee Tavrow for integrated circuit design. The updates applied to create Sue2 are quite substantial, and include a new netlisting and canvas memory allocation framework, migration to the most recent TCL/TK package for Windows, and a general update of the visual interface. Sue2 is open source software – see the COPYING file within the Sue2 directory for details on copying/licensing issues.

The current version of Sue2 supports only Spice netlisting. Future versions will include support of Verilog and IRSIM. Also, Spice netlisting does not yet include support for buses – that capability will also be added in the future.

Setup (Windows)

The Sue2 install package is provided as a self extracting executable file called setup_sue2.exe. To install, simply run setup_sue2.exe in Windows (i.e., double-click on setup_sue2.exe in Windows Explorer). You may install the main Sue2 directory wherever you like. To run the program, click on the desktop icon for Sue2 that was created during the install process.

If you would like to use CppSim with Sue2, you should **instead** install the CppSim package for Windows found at the CAD tools link at http://www-mtl.mit.edu/~perrott. This self-extracting package includes Sue2, CppSim, and the MinGW and MSYS packages which provide g++, make, sh, and other routines. Note that Cygwin is no longer required, but Matlab 6.5 (or higher) for Windows is needed in order to run CppSim.

Known Bugs

1) The undo function is broken.

Basics

Start Sue2 by clicking on its desktop icon that was created during installation. You'll see four windows pop up, which include the main canvas, one schematic listbox, and two icon listboxes. At the top of the main canvas are menu options File, Window, Edit, Sim, and the Help Message Window which initially states "Welcome to Sue2 (version 1.0) – see the COPYING file for details on copyright/licensing issues".

The various menu options are straightforward to understand, and their bindkeys are listed next to their respective command. Bindkeys, as well as other configurable parameters, can be changed in the .suerc file contained in the Sue2 directory that was created during installation.

Three key things are worthwhile to keep in mind as you use Sue2:

1) Pay attention to the Help Message Window during command operations – it provides information on bindkeys activated while a given command is in effect.
2) To break out of any given command mode, hit the **Escape** key.

3) Schematic and icon libraries shown within the listboxes can be altered by clicking on the listbox menu button located below the listbox Windows header banner and above its item selections. Note that schematic and icon libraries are always placed within the Sue2/SueLib directory and are named within the sue.lib file located in the Sue2 directory in order to be active within the listbox menus. The first entry within the sue.lib file corresponds to the default schematic library placed in the schematic listbox, and the next two entries correspond to the default icon libraries placed in the icon listboxes.

New Schematics, icons, and libraries

Creation of new schematic or icons is straightforward using the File menu commands in the main canvas window.

To create a new schematic/icon pair, it is best to first create the schematic and then press the **K** key to create its associated icon view. To be recognized within the schematic or icon listboxes, cells must be placed within a directory whose name is in the Sue2/sue.lib file and whose location is in the Sue2/SueLib directory.

To create a new library, you must create a new directory corresponding to the library name within the Sue2/SueLib directory using Windows commands (i.e., no Sue2 commands support this), and add the library name to the Sue2/sue.lib file.

Existing Schematics

A. Opening

As an example of viewing a schematic, click on sd_synth inside of the schematic listbox. The main canvas will display the sd_synth schematic. If one then clicks on sd_synth_fast within the schematic listbox, the main canvas removes the sd_synth schematic (asking for it to be saved if it has been altered), and then loads the sd_synth_fast schematic. Now click on the schematic listbox menu button to change the schematic library to spice. Clicking on inverter then loads an inverter schematic. Change the schematic library back to CppExamples, and click on sd_synth_fast to return to its schematic.

B. Window Operations

As an example of performing window operations within the main canvas window, hit the **z** key to zoom into the sd_synth_fast schematic, the **Z** key to zoom out of it, and the **f** key to fit the view to the window size. Zoom into a particular part of the schematic by pressing the right mouse button and then dragging the mouse over the region to be zoomed into. Panning is accomplished by either hitting the **arrow** keys, or by holding the **Ctrl** key and then dragging the mouse while the left mouse button is held down.

As an example of descending down the schematic hierarchy, select instance xi12 (xorpfd) using the left mouse button, and then hit the **e** key. Select instance xi0 (dff2), and then hit the **e** key again. Hit the **k** key to switch to the icon view of the cell, and then hit the **k** key again to switch back to its

schematic view. Hit the **Ctrl-e** key combination twice to return to the top level of the sd_synth_fast schematic hierarchy.

C. Editing

As an example of performing basic edits, move an instance or wire by pressing and holding the left mouse button over it and then dragging the mouse. If you try the same operation while holding the **Shift** key, then movements will be restricted to be in either the horizontal or vertical direction. Select a number of instances and wires by pushing the left mouse button and then dragging the mouse over the items to be selected. Additional items can be added to the current selection by holding down the **Shift** key while pushing the left mouse button over the items to be added. The entire set of selected items can be moved by then pressing the left mouse button over one of the selected items and then dragging the mouse. Selected items can be rotated by hitting the **r** key, can be flipped in the horizontal direction by hitting the **x** key, can be flipped in the vertical direction by hitting the **y** key, can be copied using the **c** key, and can be deleted by hitting either the **backspace** or **delete** key.

D. Adding Instances

As an example of adding instances to a schematic, select the delay item from the icon1 listbox using the left mouse button, and then place it into the main canvas window using the left mouse button. Now select the add2 item from the icon1 listbox, but allow multiple instances of it to be placed by holding down the **Shift** key as you continue to press the left mouse button in the main canvas window.

E. Modifying Instance Parameters

Double-click on an instance to view and edit its parameters. To change the parameter definitions for the module corresponding to an instance, you need to modify/add appropriate text within its icon view (as described later in this document).

F. Placing Wires

To place wires, press the **w** key within the main canvas window and then press the left mouse button over an instance pin, wire, or other part of the canvas. Click the left mouse button to create wire corners or to end the wire at an instance pin or another wire. Double-click on the left mouse button to force the end of a wire. Note that wires are different than lines – you use wires to connect instances inside schematics, and lines for creating labels within schematics and objects (such as lines, triangles, or rectangles) within icons.

G. Placing Pins, Global Supplies, and Naming Nets

The placement of pins or naming of nets is performed by selecting the appropriate icons within the devices library, which is displayed by default in the icon2 listbox. **Input**, **output**, and **inout** pins are appropriately named as such in the devices library. Global supplies (i.e., power and ground) are instantiated using the **global** icon. Naming of nets is accomplished using either the **name_net** or **name_net_s** icons. Once the pin and name_net symbol is placed within the schematic, double-click on it with the left mouse button in order to specify its name.

Existing Icons

A. Opening

Existing icons can be opened by using the Load command within the File Menu of the main canvas window. The schematic view of the cell will be loaded if it exists – in such case, display the corresponding icon view by pressing the **k** key. Alternatively, an icon within an existing schematic can be opened by descending into the cell using the **e** key and then pressing the **k** key (if necessary) to switch to the icon view of the cell.

B. Editing

As an example, load the leadlagfilter icon by using the Load command within the File menu and pressing the **k** key. Move objects in this icon as you would in schematics by pressing the left mouse button over them and then dragging the mouse. Likewise, note that objects can be rotated, flipped, copied, and deleted in the same manner as done in schematics.

Text

Create text by pushing the **t** key followed by the left mouse button. Modify text by double-clicking on it with the left mouse button and then performing edits as you would in other Windows applications (with the exception that **Tab** is used to create lines, and **Return** to end edit mode for the text object). Only three sizes are available for the text – the size of a given text segment may be varied while in text edit mode by holding the **Shift** key and then pressing either the left, middle, or right mouse button. Also, the text can be changed to either left, right, or center justified by holding the **Control** key and then pressing either the left, middle, or right mouse button.

Lines

Create a line by pushing the **l** (as in line) key followed by the left mouse button, and then double-clicking on the left mouse button at a different point on the canvas. Multi-segment lines are created by single rather than double-clicking on the left mouse button at each desired breakpoint of the line, with a double-click of the left mouse button being used to end the line. Press the **Shift** key to limit the drawing of line segments to the vertical or horizontal direction. Once a line is created, its various breakpoints can be modified (including the end points) by first double-clicking on the line, and then pressing and holding the left mouse button over the given breakpoint followed by dragging of the mouse.

Rectangles, Triangles, and other Polygons

Rectangles, triangles, and other polygons are created with multi-segment lines, so that the above description for lines applies to these objects.

Circles, Ellipses, and Arcs

Create an arc by pressing the **a** key followed by pressing (but not holding) the left mouse button, moving the mouse until the appropriate size and shape for the arc is achieved, and then pressing the left mouse button. Modify an existing arc by first double-clicking on it with the left mouse button

and then moving its breakpoints as you would for lines. Circles and ellipses are created by modifying an arc such that it nearly closes on itself (Sue2 does not allow the arc to be completely enclosed, but you can get close enough that the circle or ellipse looks reasonable).

C. Parameters and Instance Names

Parameters for a given icon are created and displayed according to text statements placed within the icon view. The instance name for a general cell is simply a parameter called **name** which has a default value of **x**. Instance names for primitive elements such as MOS transistors, resistors, inductors, and capacitors are also parameters called **name**, but have default values of **M**, **R**, **L**, and **C**, respectively (see the icon views of elements within the spice library to see examples of this). Note that Sue2 will automatically append numbers after the default values of the instance names once the given cells or elements are placed within a schematic and saved.

As an example, look at the icon view of the cell leadlagfilter located in the CppSimModule library. At the top of the green rectangle corresponding to the symbol block outline is the statement **$name**. The **$** symbol indicates that the text to follow is a variable whose value should be displayed, so that **$name** directs Sue2 to display the instance name when the leadlagfilter icon is placed in a schematic. Below the green symbol block outline are statements such as **fp=$fp**, which direct Sue2 to display the values of the various parameters when the instance of the icon is placed in a schematic. Much below the green symbol block outline are statements such as **–type user –name fp –default 1.0**, which declare parameters (**fp**, in this case) along with their default values.

To add a parameter **example_param** to an existing icon, simply add the text statement **–type user –name example_param –default 1.0** to its icon view. To display the value of this parameter when the icon is placed within a schematic, simply add the text statement **example_param=$example_param** to its icon view.

When creating an icon from an existing schematic view (by pressing **K**), text lines will be automatically generated to declare the instance name (**name**) and an example parameter (**example_param**), along with statements that will display these parameter values. Simply modify the **example_param** statements to reflect a desired parameter name, and add additional statements to create extra parameters.

D. Netlisting Directives

Netlisting directives for a given icon are contained in its icon view. Most icons created in Sue2 do not require such directives, but they are required when primitives (such as capacitors, resistors) are created.

As an example, look at the icon view of the **capacitor** symbol contained in the spice library. First, as mentioned above, notice that the icon declaration contains a default value of **C** rather than **x** (i.e., **-type user –name name –default C**). The default value for instance names corresponds to the first letter of their instance name when netlisting. For instance, capacitors have a default value of **C**, inductors have a default value of **L**, MOSFET transistors have a default value of **M,** and instances that are composed of other instances and primitives have a default value of **x** (which is the default case). Second, notice the text statement within the **capacitor** icon that reads **–type fixed –name spice –text {$name $t1 $t2 c=$capacitance}**. This statement declares that the SPICE netlist statement

corresponding to this primitive is composed of its instance name (which will start with C) followed by its terminal names and the value of the capacitance parameter. Future version of Sue2 will allow statements for Verilog and IRSIM.

Netlisting and Printing

Netlisting and printing to eps files are straightforward using Sue2 menu commands. To create SPICE netlists, select **Create Spice Netlist** under the **Sim** menu item in the main canvas window. Alternatively, you can create SPICE netlists in batch mode by directly running the executable sue_spice_netlister.exe contained in the Sue2/bin directory (this method is used when running CppSim). To create an eps file corresponding to a given schematic, simply select **Create eps file** under the **File** menu item in the main canvas window.