

Tutorial

SUE Design Manager

Micro Magic, Inc.

Version 5.3



DataPath Compiler, SUE Design Manager, MAX Layout Editor, MAX-LS Layout System, MAX-3D 3D Layout Editor, MAX-View Layout Viewer, and MegaCell Compiler are trademarks of Micro Magic, Inc. All other trademarks, service marks, registered trademarks, or registered service marks may be the property of their respective owners. All specifications are subject to change without notice.

SUE Design Manager Tutorial
Copyright ©1996-2008, Micro Magic, Inc.
All rights reserved. Printed in USA.

The information in this document has been carefully verified and is believed to be accurate. Micro Magic, Inc. assumes no responsibilities for any inaccuracies that may appear in this document. In no event will Micro Magic, Inc. be liable for direct, indirect, special, exemplary, incidental or consequential damages resulting from any defect or omission in this document, even if advised of the possibility of such damages.

Micro Magic, Inc. reserves the right to change, modify, transfer or otherwise revise this publication without notice.

Table of Contents

Part 1

| | |
|---|---|
| Getting Started with SUE Design Manager | 1 |
| Introduction | 1 |
| SUE Features | 1 |
| Getting Started | 1 |
| Starting Up SUE | 1 |
| SUE Screen Layout | 3 |
| Library List Boxes | 3 |
| SUE Pull Down Menus and Hotkeys | 4 |

Part 2

| | |
|----------------------------|----|
| Schematics And Icons | 7 |
| Introduction | 7 |
| Drawing A Schematic | 7 |
| Creating a New File | 8 |
| Beginning the Schematic | 9 |
| Selecting Icons | 11 |
| Editing Icons | 14 |
| Generators in SUE | 16 |
| Wiring and Connectivity | 17 |

Part 3

| | |
|-------------------------------------|----|
| Simulating Circuits With SPICE | 23 |
| Simulating Circuits | 23 |
| Running SPICE | 23 |
| Displaying SPICE Waveforms From SUE | 24 |
| Net Names When Netlisting | 26 |

Part 4

| | |
|---|----|
| Higher Level Schematics And Verilog Simulation | 29 |
| Higher Level Schematics | 29 |
| Verilog | 29 |
| Making An ICON For Our Schematic | 31 |
| Placing Our ICON Into The Schematic | 35 |
| Creating Behavioral Verilog Models | 39 |
| A More Complex Example | 43 |
| Displaying Design Hierarchy, and Controlling The Simulation | 47 |
| Mixed-Mode Verilog Simulation | 49 |
| Buses | 51 |

Part 5

| | |
|-------------------------------|----|
| Cross-probing with MAX | 55 |
| Cross-Probing With MAX | 55 |

Table of Contents

Part 1

Getting Started with SUE Design Manager

Introduction

Welcome to SUE Design Manager, from Micro Magic, Inc. SUE is more than just a schematic capture program, it is a complete graphical user environment.

SUE Features

With SUE, you will be able to:

- Draw, view, and edit schematics, icons, random graphics and text.
- Use the built-in netlist to Verilog, NGSPICE, Berkeley SPICE.
- Cross-probe directly with Verilog, NGSPICE, and MAX (the Custom Layout Editor from Micro Magic).
- Interactively run Verilog, with logic values displayed directly on the schematic.
- Automatically attach documentation, Verilog models, etc. to schematics.
- Enjoy automatic version control.

This tutorial will carefully walk you through these features and many more.

Getting Started

Before we get started make sure someone has already installed SUE at your site.



To run some parts of the tutorial you will need to have programs like SPICE and Verilog.

We include two free simulators with the release: **ngspice** for SPICE simulation and **cver** for Verilog simulation. Before you use SUE for design work, you should point to your simulators using the **setup_sue** script. Refer to the *SUE User Manual* for details.

Starting Up SUE

To run the tutorials, you need to make a personal copy of the SUE tutorial directory.

Step 1

- To install the SUE tutorial, type:

```
mmi_tutorial
```

at the UNIX prompt. Make sure that the appropriate tutorial is selected and then click on **Install Tutorial**. The default installation directory is “`mmi_private/tutorial`” in your home directory. You can also select a different directory for installation. Don't worry if the directory doesn't exist, the script will make it if it can. You can also use this to reinstall a clean copy of the tutorial.

Step 2

- To start the tutorial, you must first “`cd`” (change directory) to the directory where the tutorial was installed. If you used the default directory, type:

```
cd ~/mmi_private/tutorial/sue
```

Otherwise, replace the above directory with the directory you selected. This directory contains files that you will need to run the tutorial.

Step 3

- Then to start SUE type:

```
sue
```

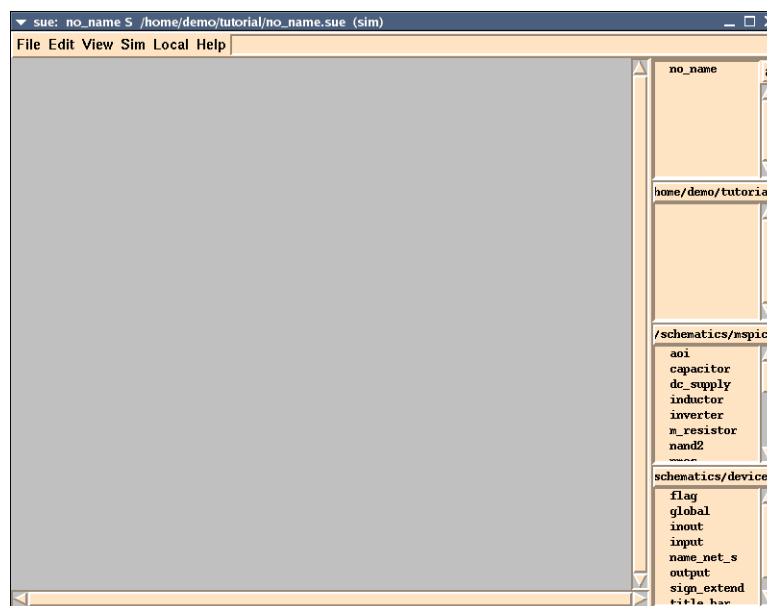
A window should come up that looks something like Figure 1-a. All the colors can be changed to your own personal preferences in the `sue.rc` (or `.suerc`) file. Figure 1-b shows SUE with a black background.

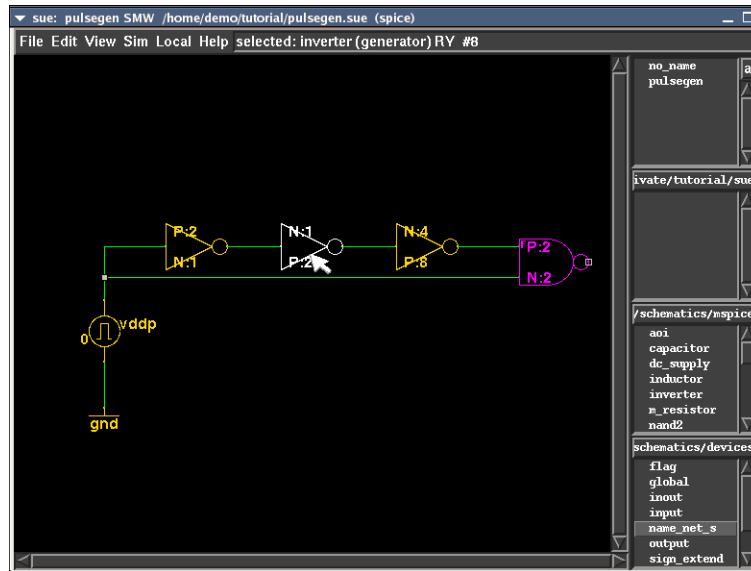
- To bring up SUE with a black background, add the following line to your `sue.rc` file:

```
color_scheme black_background
```

It is strongly suggested that you use the default background and colors for this tutorial. Numerous times during this tutorial we mention the color of an icon or wire and some of those are different when using a black background.

Figure 1: SUE Window in a) Default Colors; b) Modified Colors





SUE Screen Layout

On the very top of the window the title bar should say:

SUE: no_name S <path_to_cell> (spice)

“no_name” means that you have not specified the file (schematic) you wish to edit. The “S” means that you are editing a *schematic*, as opposed to an icon. The “(spice)” means that you are currently in **SPICE simulation mode**.

Below the title bar you see the menu bar which contains the following menu items: **File, Edit, View, Sim, Local** and **Help**. These are pull down menus working much like any window-based application using click and drag.

Directly to the right of the **Help** menu is the **SUE Message Area**. It currently says “Welcome to Micro Magic SUE (MMI_SUE5.3.0).” Note that the 5.3.0 (or whatever the number is) refers to the version of SUE you are running. If you want to see more information about the version, go to **About SUE** in the **Help** menu.

Library List Boxes

Down the right side of the window are several small **Library List Boxes**.

The top one is the **Schematic List** box. This lists all current schematics that have been loaded into SUE. Currently only “no_name” should be listed.

All of the rest of the **List Boxes** are for icons. (In some tools, *Icons* are called *Symbols* — they are the same thing). Each **Icon List Box** displays the loaded icons in a given library, which is also a UNIX directory.

The UNIX directory of the Library in each Icon List Box is shown at the top of the List Box, clipped from the left. You can change what Library goes in which List Box by holding down **Button-1** on this directory name and selecting a different library from the list that pops up. Libraries are automatically added to this list when you load any element from them. You can also load all of the icons in the Library by selecting **Autoload directory**, and add or subtract List Boxes with this menu.

The top Icon List box typically contains the library you are currently working in. When you create new icons, they will show up there. The other List boxes contain other libraries. In this example, the mspice and devices libraries are shown. The **devices Library** is necessary since it contains the all-important I/O icons: *input*, *output*, *inout*, and *global*. It also contains other handy general-purpose icons. The **mspice Library** is useful for low level circuit design since it contains parameterized transistors, simple gates, and the like.

SUE Pull Down Menus and Hotkeys

SUE pull down menus work just as in most other window-based applications. Each command is listed with its hotkey shortcut next to it (if it has one). In addition, when you drag the cursor over an entry, a description of the command is displayed in the **SUE Message Area** (to the right of **Help**).

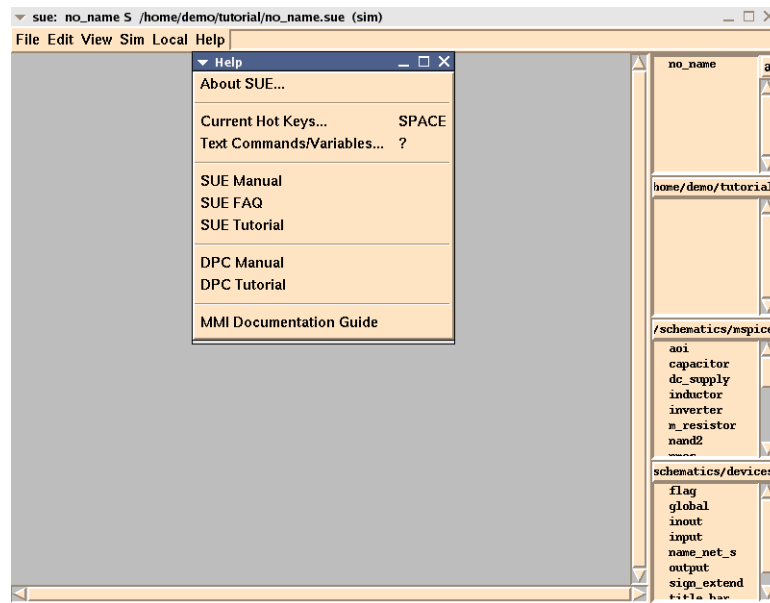
- Step 1**
- Try this out. Go to the **Edit** menu and move the cursor from command to command. Can you find the help description? What is the hotkey to add a wire?

SUE menus can also be torn off — that is, turned into top level windows that don't roll back up when you're done with them. This can be useful if you need to go to them frequently and they don't have a hotkey (or you can't remember it).

- Step 2**
- To tear off a menu, simply click on the “-----” (dashed line) at the top of the exposed menu. Try tearing off the **Help** menu now.

You should now see something like Figure -2. You can manipulate this window like any other Xwindow (window manager dependent). For example, you can move it by clicking and holding down the **left mouse button** (**Button-1**) over the top title bar and dragging it around. Or you can close the window by holding down the **right mouse button** (**Button-3**) over the top title bar and selecting **Close** from the menu that pops up.

Figure 2: Help Menu



Tip

You can also just click on the menu and the pull down will stay down until you click on the desired item.

Scroll bars are shown across the bottom and right side of SUE (and are found in the List boxes). These work like the scroll bars in most other applications. Additionally, the mouse scroll wheel can be used to scroll through List boxes.

- Step 1**
- To view the complete SUE on-line manual, select **SUE Manual** from the **Help** menu. In addition to providing detailed information on all of SUE's commands, modes, and features, the SUE manual explains the philosophy behind SUE. After finishing this tutorial you should read through the manual. In fact, it would be very useful to read the sections on SUE Philosophy and Selection at this time.
- Step 2**
- To learn about all the Micro Magic, Inc. software in this distribution, select **MMI Documentation Guide** from the **Help** menu.
- Step 3**
- Now hit the **Space** key (or **Current Hot Keys** in the **Help** menu). You will now see all of the hot keys in alphabetical order, plus a description of what they do. This window is context specific, so when you are in a mode like **Add Wire** and hit the **Space** key, you will see just those hotkeys which are active in that mode. Hit **Space** again to close the hotkey window.
- Step 4**
- Now close the **Help** menu.

Part 2

Schematics And Icons

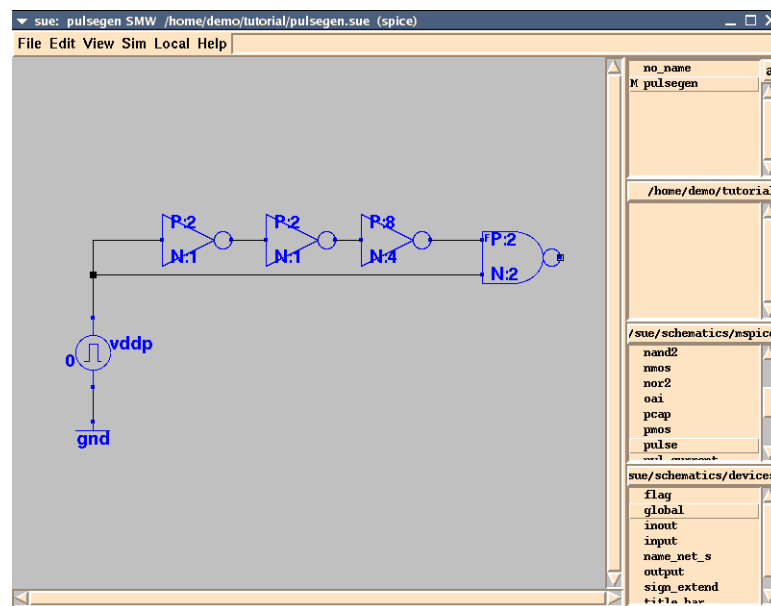
Introduction

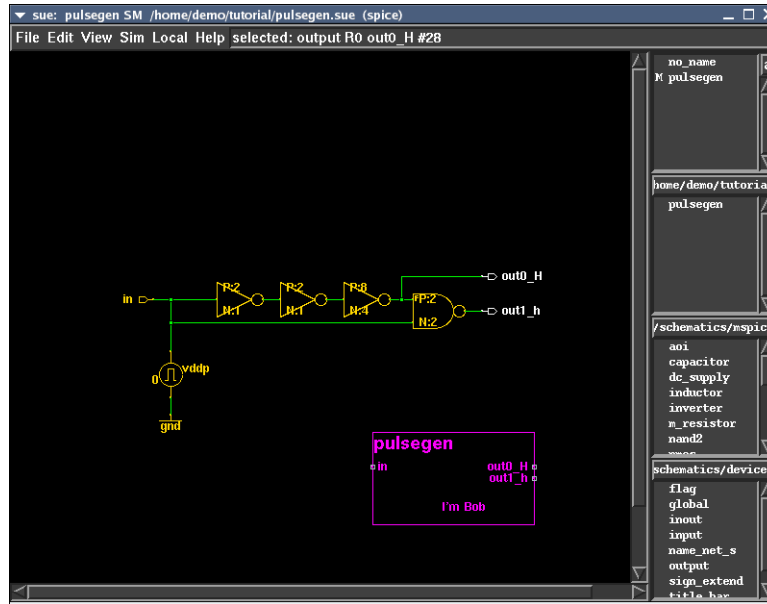
Drawing A Schematic

OK, we are now ready to draw a schematic using SUE.

We are going to draw a very simple (and not very useful) circuit that will generate a pulse. The circuit we are going to draw is shown in Figure 3. (If you were to use the black-background color set, this schematic would look like Figure 3-b.)

Figure 3: Simple SUE Schematic





Creating a New File

CREATE NEW
SCHEMATIC

First we will create a new schematic with a given name.

Step 1

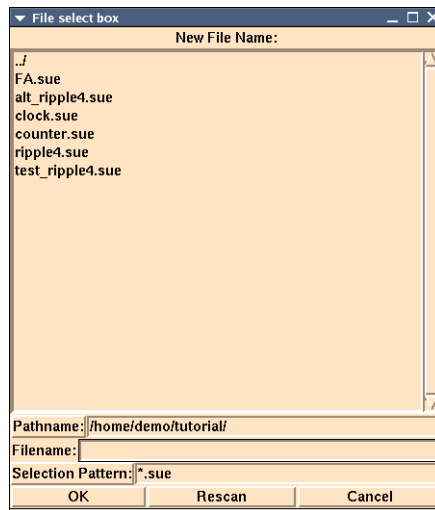
- Pull down the File menu and select New Schematic. This will bring up the File Select box, as shown in Figure 4. Note that you could have also used the hotkey **Ctrl-n** to bring up this menu as well. That is, while holding down the **Control** key, type “n”.



Tip

Using the hotkeys for the most common commands will make you much more productive when using SUE.

Figure 4: The File Select Box



The File Select box lets you select a directory along with the schematic name. This is the directory where the schematic will be saved.

- Step 2**
- Type the name of the schematic “**pulsegen**” (or any other name you like) near the bottom where it says “**Filename:**”. (If the cursor isn't blinking to the right of “**Filename:**”, click the mouse button there first.) Now either hit **Return**, or click the **OK** button.

You should now see “**pulsegen**” in the top Schematic List box and also in the title bar at the top of the window. The title bar tells you that you are now editing **pulsegen**.

Now to draw our schematic.

Beginning the Schematic

To draw a schematic we are going to do 3 things:

- Add icons to the schematic,
- Modify properties of those icons, and
- Draw wires.

ADDING ICONS

First we add icons from the Icon List boxes:

- Step 1**
- Find the “inverter” icon in the mspice Library Icon List box. You might have to scroll to find it (using either the scroll bars at the side, or the scroll wheel on your mouse). Next, click **Button-1** over “inverter” and then move the cursor over to the schematic window where you should see the inverter icon appear. Drop the inverter where you want it by clicking **Button-1** again.

Notice that an “**M**” has appeared to the left of the “**pulsegen**” name in the Schematic List box, at the top right of the SUE window, and also in the title bar of the window. The “**M**” tells you that the cell is *modified*.

SUE doesn't let you exit if you have any modified schematics or icons without verification. Let's try it to see what happens.

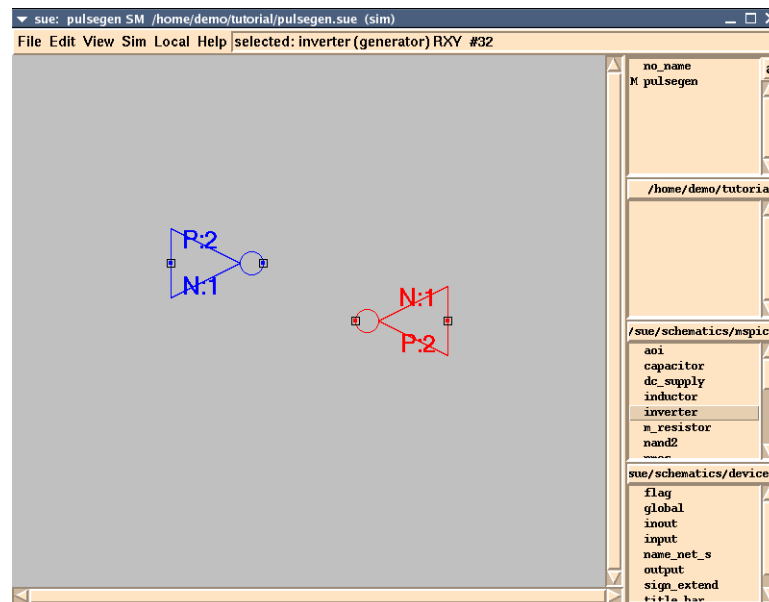
- Step 2** ■ Select Exit from the File menu or hit **Ctrl-d**. The Cells Modified dialog box will come up and tell you that you have modified cells.
- Step 3** ■ Click on Cancel since we don't want to exit yet. If you had clicked Exit and Lose Changes, SUE would have quit WITHOUT saving your modifications.

MANIPULATING AN ICON

Let's add another icon, but this time we'll try something fancier.

- Step 4** ■ Click **Button-1** over the “inverter” in the Schematic List box again and move the cursor to the schematic.
- Step 5** ■ Hit the **Space** bar on the keyboard. This brings up a list of all of the things you can do before dropping the inverter. An abbreviated list is also displayed in the SUE Message Area. Hit the **Space** bar again to make the window go away.
- Step 6** ■ Hit “x” to flip the inverter in the X direction.
- Step 7** ■ Hit “z” to zoom in around the inverter. Hit **Shift-z** to zoom out around the inverter.
- Step 8** ■ Click on **Button-1** to drop the inverter. Your schematic should now look something like Figure 5.

Figure 5: Two Inverters



Selecting Icons

In SUE, most editing operations apply to the **selected object**. Objects, whether *icons, wires, text, arcs, or lines*, change color to red if they are **selected** (the colors referred to here are the default colors specified for this tutorial in the `sue.rc` file).

You probably noticed that if you put the cursor over an icon, its color changes to white. This is known as the **active object**. *When you move the cursor over a selected object, its color changes to yellow.*

- Step 1** ■ Move your cursor over the left inverter. It should change to white. This inverter is ready to be **selected**.
- Step 2** ■ Move your cursor slowly past the center of the icon. Notice that the icon will turn blue again momentarily. What happened? The icon is defined as what is blue and thus the inside is not part of the icon. This may seem weird at first.
- Step 3** ■ Click **Button-1** when the inverter is white. It should change to yellow which means it is both **selected and active**.
- Step 4** ■ Move your cursor away from the inverter. It should change to red which means it is **selected but not active**.

ROTATING AN ICON

SUE lets you do lots of interesting things to the selected object. For example, let's take the inverter you just dropped and rotate it

- Step 5** ■ Select the rotated (flipped in the X direction) inverter.
- Step 6** ■ Hit the **r** hotkey twice to rotate it back to the original orientation.



If two objects are close together, move the cursor around a little bit to get the desired object to become active. Also, by making an object active you can quickly tell what is included in that object since only that object will change color.

Selected objects are also described in the **SUE Message Area**.

- Step 7** ■ Select the inverter you just rotated, and you should see something like:

```
selected: inverter (generator) RY #23
```

This is telling you that you have selected an inverter which is a generator (more about that later) and which is flipped (R=Rotated) in the Y direction and has the unique identifier of "23" in this schematic. The identifier number will probably be different in your schematic. The first inverter should have an orientation of **Ꝁ0** (no flipping or rotating).

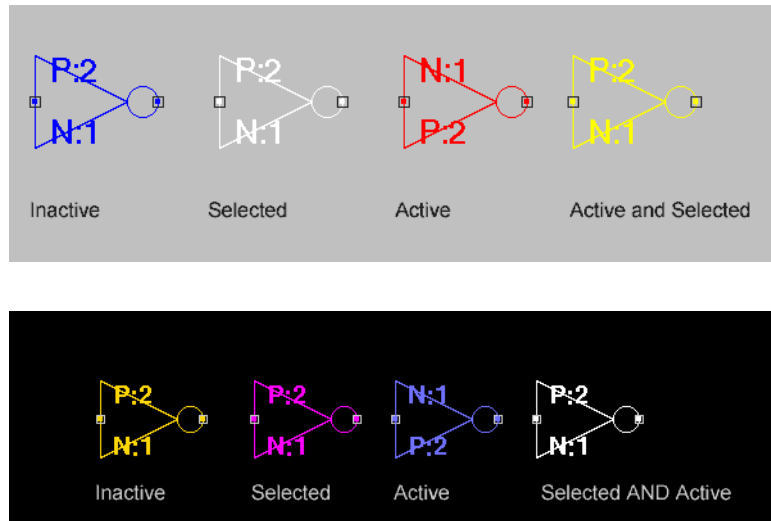
To recap, objects can be in one of 4 states:

- **blue** — default (inactive, unselected)
- **white** — active

- **red** — selected
- **yellow** — active, selected

If you were to use the black-background color scheme, the four states would have the colors shown in Figure 6-b.

Figure 6: Object in Different States: Default (Inactive); Active; Selected; Active AND Selected



The two main ways to **select objects with the cursor** are by **clicking Button-1** on an active object as described above, or by **dragging a select box** around a region. By default, the previous selection will be cleared when making a new selection. To prevent this, simply hold down the **Shift** key when selecting, and the new selection will add to the old selection. Also, clicking **Button-1** over a blank part of the schematic (an area with nothing active), clears the selection.

ADDING AND SELECTING ICONS

Now you try.

- Step 1** ■ First, add a few more icons to the schematic.
- Step 2** ■ Click and hold **Button-1** down and drag a white selection box around a few objects and then release **Button-1**. All of the objects completely inside of the selection box should turn red and be selected.
- Step 3** ■ Drag a box around another portion of the schematic. The icons that were only in the first box will deselect and the new ones will select.
- Step 4** ■ Hold down the **Shift** key and drag another box. The previously selected icons should stay selected.
- Step 5** ■ With the **Shift** key still down, click **Button-1** over an active, selected (yellow) icon. Only that icon should deselect.

MANIPULATING SELECTED OBJECT

Now let's do something to the selected object.

- Step 6** ■ Select one inverter. Remember, you can either click **Button-1** when it's active to select it or drag a box around it with **Button-1**.
- Step 7** ■ Rotate the inverter by hitting the **r** hotkey. You could also use the menu command **Rotate** in the **Edit** menu. The inverter should now be facing down.
- Step 8** ■ Rotate the inverter three more times by hitting **r** three more times (this is why the hotkeys are good to learn). The inverter should return to its original orientation. To rotate in the other direction, counterclockwise, use **R** (**Shift-r**).



You can also type **u** for “undo” to undo your last action. SUE gives you 100 levels of undo!

- Step 9** ■ Hit the **u** hotkey to undo the last rotation.
- Step 10** ■ Hit the **u** hotkey 3 more times. Is that what you expected?

You can also Move, Flip, Duplicate, Delete, and Modify the inverter. All of these features and more are found in the **Edit** menu. **Edit** menu features work on the selected object or objects. Once again, you can either use the menu or the hotkeys.

DUPLICATING SELECTED OBJECT

Let's try duplicating an icon:

- Step 11** ■ Select an inverter and type **d**. This will make a *duplicate* copy of the inverter and offset it slightly so you can see it. You are still in duplication mode which you can tell because the cursor is a “pointing hand”.
- Now click and hold down **any mouse button**. This allows you to drag the duplicate to where you want it to go.
- If you hold down the **Shift** key while duplicating, the duplicated inverter locks to the X or Y direction, based on the original.
- Step 12** ■ Finally, release the mouse button to drop the icon in its new resting place.



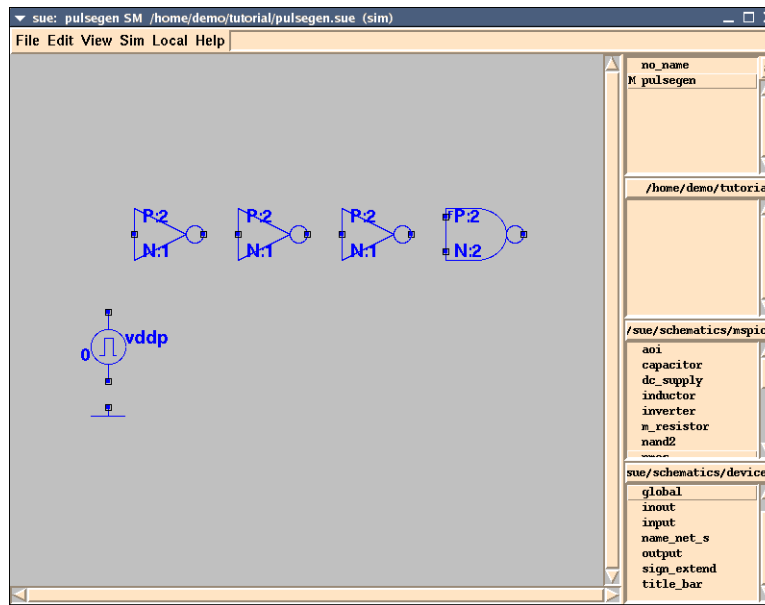
Once you are in any mode, such as duplication mode, the **SUE Message Area** tells you what to do next. You can also hit the Space bar to get a full list of available hotkeys and more information.

CONTINUE BUILDING “PULSEGEN”

It's time to get back to making our pulse generator.

- Step 1** ■ Remove all the icons except for three inverters. Move them if they aren't already lined up in a row. You can move the inverters by first selecting it and then move it by holding down **Button-3**.
- Step 2** ■ Add a “nand2” gate (also in the mspice Library) to the right of the inverters.
- Step 3** ■ We now need two more icons, the “pulse” from the mspice Library, and the “global” from the devices Library (at the bottom). Place them to the left of everything else as shown in Figure 7.

Figure 7: Icons Placed in Window



Editing Icons

Every icon has **properties** (also known as *parameters*). At the very least, an icon usually has a **name** property so you can specify its name for netlisting. The icons that come from the **mspice Library** are for transistor-level design and have parameters for transistor widths and lengths.

EDIT ICON PROPERTIES

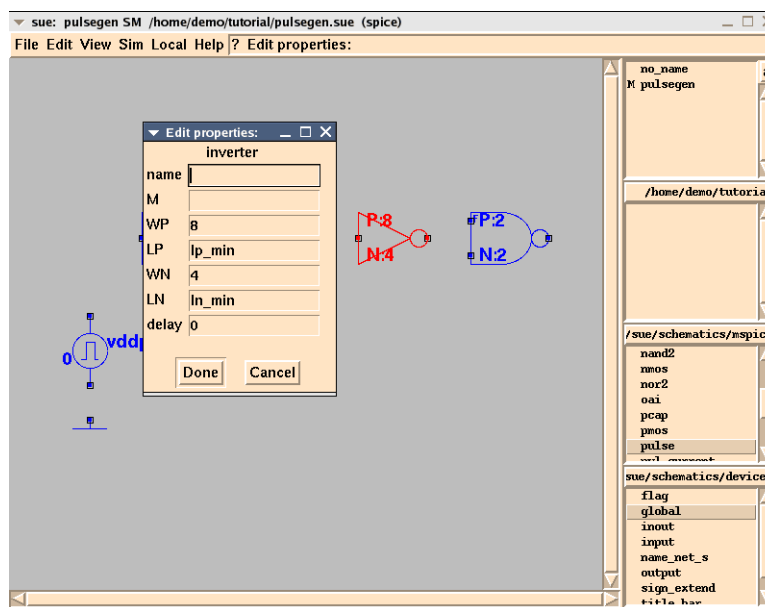
Let's edit the transistor sizes of the third inverter to make them larger.

- Step 1** ■ Make the right-most inverter **active** (white). It should have a P/N ratio of 2/1.
- Step 2** ■ Double-click with the **left mouse button** (**Button-1**) on the active inverter. This brings up an **Edit Properties** dialog box as shown in Figure 8 and selects the inverter.



If the cursor is a "?" or the **SUE Message Area** displays "? <message>" then SUE is waiting for you to close a popup window. You cannot continue with SUE until you close that window, usually by clicking on either the **Done** or **Cancel** button. If you don't see the popup, look at all of your iconified windows or see if the popup got moved behind some other window.

Figure 8: Edit Properties Pop-up for Inverter



- Step 3** ■ Change the value of **WP (Width of P)** from **2** to **8** and the value of **WN (Width of N)** from **1** to **4**. This will quadruple the size of this inverter.

- Step 4** ■ Click the **Done** button. The inverter should now display a P/N ratio of 8/4.

ASSIGN "GROUND" TO GLOBAL

Now let's assign the correct name to the **global** icon. The **global** icon turns its attached wire into a global signal like a power supply. For our **pulsegen** circuit we want the **global** to be 'ground' to provide a reference for our pulse generator.

- Step 5** ■ Make the **global** icon *active* and then double-click on it. This brings up a popup.
- Step 6** ■ Type "gnd " into the **Edit Properties** popup for the **name** property.
- Step 7** ■ Hit **Done**. The **global** icon should now have the label **gnd** on it.



vdd and gnd will be automatically translated by SUE to the correct levels (1 or 0) for Verilog.

EDIT "PULSE" ICON

The **pulse** icon will generate a string of pulses for SPICE simulation. You can change the duration, amplitude, delay, period, etc. of the pulses by editing the pulse icon properties.

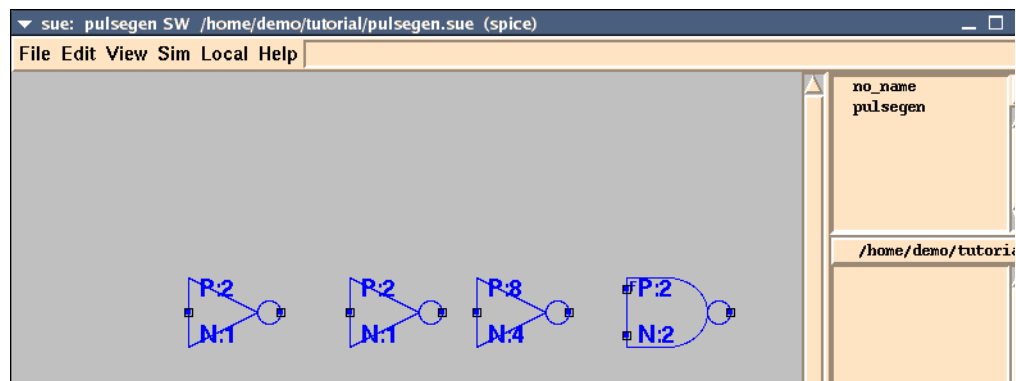
- Step 8** ■ Make the “pulse” icon active and then double-click on it. This brings up a popup.
- Step 9** ■ Hit Done when you are finished reviewing the popup.

Since you have done some work you should now save your circuit.

- Step 10** ■ Do this by selecting Save from the File menu, or type the hotkey **Ctrl-s**.

Notice that the M next to **pulseggen** disappeared from the schematic list box. The title bar of the window now has **SW** next to **pulseggen**, as you can see in Figure 9. This means that this is a Schematic, and it has been Written.

Figure 9: Save Work on Schematic: Title Bar Showing “SW”



Generators in SUE

Many of the sample gates provided with SUE are generators. A SUE generator takes arguments that can structurally change its icon and/or schematic (unlike parameterized icons) and creates a new icon/schematic. Hence, for every generator, a multitude of cells can be generated, depending on the combination of its arguments.

EDITING “NAND2” GENERATORS

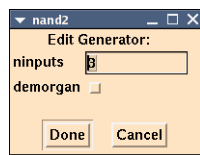
For example, the “nand2” icon is actually a generator. You can edit the number of inputs and/or specify that you want to use a demorgan equivalent icon. The schematics under the icon will also change based on the number of inputs.

- Step 1** ■ Select the “nand2” gate by clicking on it with the **left mouse button**.
- Step 2** ■ Push into the schematic (hotkey: **e** or from the **View** menu, select **Push Into**.)
- Step 3** ■ Look at the schematic. Notice that it has 2 pfets and 2 nfets.
- Step 4** ■ Look at the icon view by using the **c** hotkey, or selecting **Display Other View** from the **View** menu.
- Step 5** ■ Notice the Verilog property line which should say:

“-type fixed -name verilog -text {assign \#\$delay out=!(\$In0&&\$In1);\;}”

This is the Verilog model for a *2 input nand gate*.
- Step 6** ■ Pop back up to the pulsegen schematic (hotkey: **Ctrl-e** or from the **View** menu, select **Pop Out Of**.)
- Step 7** ■ To change the generator properties, hold down the **Shift** key and double-click on the nand2 gate with the **left mouse button**. The pop-up for the nand2 in will appear, as shown in Figure 10.

Figure 10: Edit Generator Form for nand2



- Step 8** ■ Change “ninputs” (number of inputs) to 3 and click on **Done**. Notice that the icon updates to have 3 inputs.
- Step 9** ■ Push into the icon again (hotkey: **e**) and notice that the underlying schematic has changed. There are now a total of 6 transistors.
- Step 10** ■ Look at the icon view (hotkey: **c**). Notice that the Verilog model is now for a *3 input nand gate*.
- Step 11** ■ Pop back up to the to the **Pulse_generator** schematic (hotkey: **Ctrl-e**) and undo the change (hotkey: **u**). The nand gate should be back to only 2 inputs.

Wiring and Connectivity

We are now ready to wire up the circuit. First, look closely at the ports of each of the icons. Do you see a hollow square surrounding each port? If not, zoom in until you do. The hollow square, called an *‘open’*, signifies that the port is **unconnected**.

Ports and wire ends show connectivity with one of three symbols:

- **hollow square** (*‘open’*) signifies unconnected
- **<nothing>** signifies one connection

- **solid square** ('*solder dot*' or just '*dot*') signifies two or more connections.

Remember to pay attention to this when you wire up the circuit. If you think that you have connected a wire to a port but you missed, the '*open*' won't go away.

WIRING UP SCHEMATIC

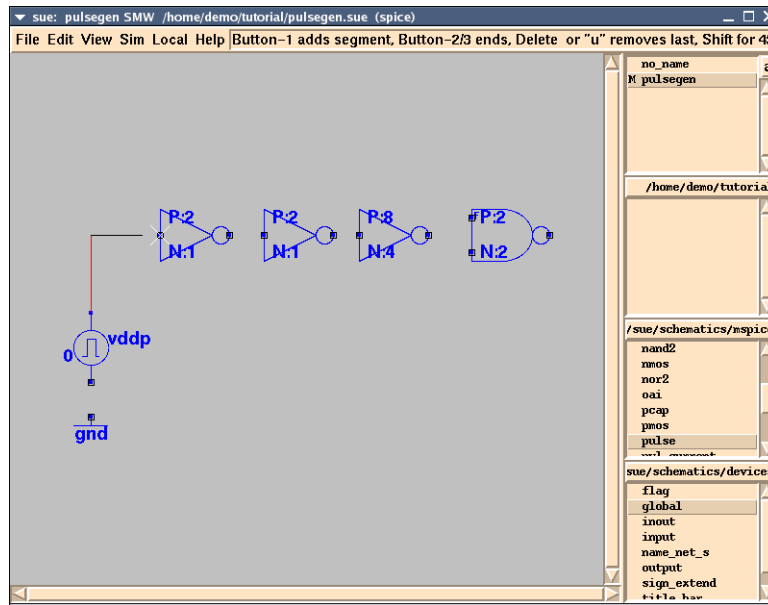
Now let's add some wires.

- Step 1** ■ Select **Add Wire** from the **Edit** menu or type the **w** hotkey. This puts you into "**wiring mode**".
- Step 2** ■ Look at the **SUE Message Area**. It will tell you what the buttons do, or you can hit the **Space** bar to see what all of the buttons and hotkeys do.
- Step 3** ■ Move the cursor to the port on the top of the **pulse icon**. Click the **left mouse button (Button-1)** on top of the **open**. This starts a new wire at that point.
- Step 4** ■ Move the cursor up (vertically) away from the port on top of the **pulse icon**. A dark green wire should follow your cursor. When you get up to about the level of the inverter input port click the **left mouse button (Button-1)** again and move the cursor to the right. The mouse click creates an anchor point which allows you to put a bend into the wire. See Figure 11.



Note that all of the colors mentioned in this tutorial are based on the default color map. If you are using a black background, your colors will be different in some cases.

Figure 11: Drawing A Wire



Step 5

- When you get the mouse over the left (input) port of the inverter hit the **right mouse button (Button-3)** to end the wire. Did the opens go away? If not, you missed. You can either start over with undo or add another wire to finish the connection.

Try zooming in for a better view of the wiring connections.

- Place the mouse pointer over the connection, and push forward on the scroll wheel to zoom into the schematic. Zoom out again by pulling back on the wheel. If you do not have a scroll wheel, you can use the “**Zoom In on Cursor**” hotkey **j** and the “**Zoom Out**” hotkey **Shift-z**.



Tip

SUE tries to help you wire by placing a white “X” at the nearest open port to the cursor, as shown in Figure 11. If you like that location then simply double-click with the **left mouse button (Button-1)** and SUE will wire it up for you.

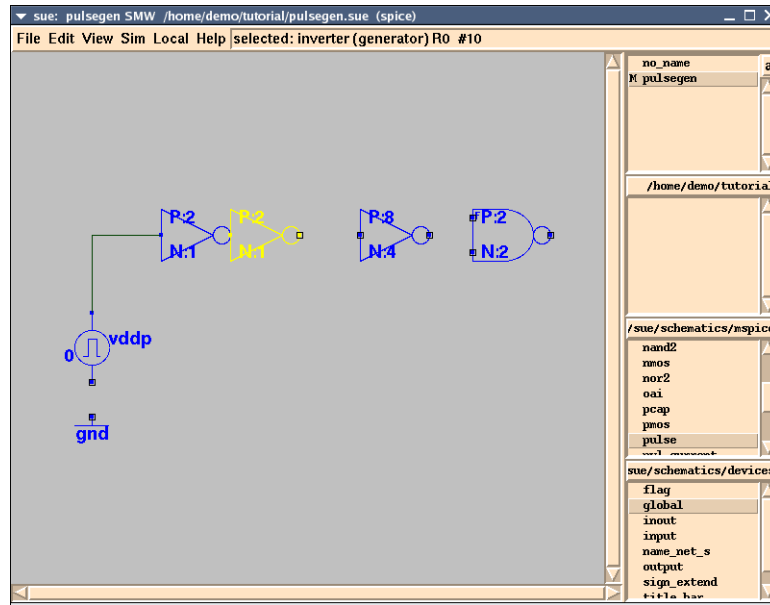
WIRING BY ABUTMENT

When you move icons that are connected to wires, the wires stay connected. You can use this feature to add wires by abutting ports from two different icons you want to connect.

- Step 6** ■ Move the middle inverter left so that its input port overlaps with the output port of the inverter on its left, as shown in Figure 12.

To **move** the inverter, position the cursor so that the inverter is **active** (white) and then press and hold **Button-2** or **-3**. You are now in **move mode** and the inverter will follow the cursor until you release the button. Once you release the button, the opens between the two inverters should go away. If not, repeat until they do.

Figure 12: Wiring By Abutment



- Step 7** ■ Now, hold down **Button-2** or **-3** again and move the middle inverter to the right. These two inverters are now wired up. You can use this method of wiring instances as you are placing or duplicating the instances.

CONNECTING TWO WIRES

If you draw a wire that crosses another wire, the two won't connect. In order for them to connect there must be an overlap on at least one of the endpoints of the wire or at a port. To get two wires to connect, just start or stop the new wire over the old wire.

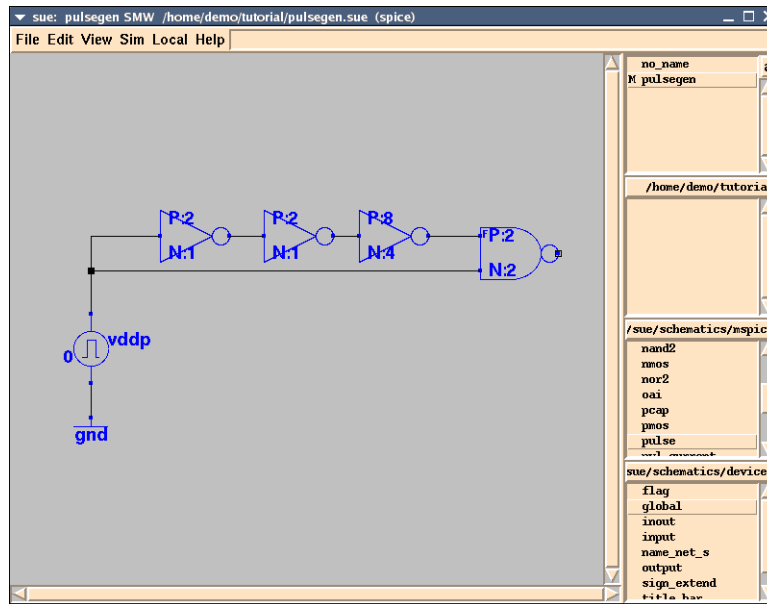
- Step 8** ■ Begin a new wire starting from the center of an existing wire and draw it perpendicular to that wire. Did SUE add a dot to the "T" intersection? If not, try again until a "T" intersection is formed with a dot indicating the connection.
- Step 9** ■ Undo (hotkey: **u**) this wire segment.



Tip When you are drawing wires or moving objects, the connection information (dots, opens, etc.) is not recomputed until you are done. For example, when you move an inverter to make a connection, the dot doesn't go away until you release the mouse button.

- Step 10** ■ Now wire up the other pieces so that your schematic looks like Figure 13. Don't worry about the ports yet; we will add them later in the tutorial.

Figure 13: Sample Schematic



Tip

If at any time you are in a mode and something goes wrong, you can type **Ctrl-c** to abort that mode. For example, if your wire isn't going the way you want it to, typing **Ctrl-c** will abort that wire.

- Step 11** ■ Before going on to the next section, save your schematic (hotkey: **Ctrl-s**).

Part 3

Simulating Circuits With SPICE

Simulating Circuits

The main purpose of creating schematics is to netlist, simulate and verify them. In SUE this is a snap! In the next few sections you will get a chance to simulate the same circuit in different simulators: SPICE and Verilog.

Running SPICE

If you don't have a SPICE simulator installed, Micro Magic has provided a publicly available simulator with this tutorial called NGSPICE. The program has been set up so that the user can simply install the build and by default can run NGSPICE using the `mmi25.mod` SPICE models.

Step 1

- First, make sure the following two things are set up:
 1. You have a circuit loaded into SUE that you want to simulate. The “pulsegen” circuit you just entered will work fine. If you just started SUE again, load the circuit using the **Open** command in the **File** menu. Your schematic should look like Figure 14.
 2. Insure that you are in **SPICE mode**. On the title bar of SUE you should see ‘(spice)’ which means you are in **SPICE mode**. Otherwise, go to **Change Simulation Mode** in the **Sim** menu and click on **spice** and then **Done**.

Step 2

- OK, pay close attention. To run SPICE you have to do the following set of complex tasks:

Type **h**

That's it!

This single command (SPICE it in the **Sim** menu) does the following:

- First, it creates an SPICE netlist of the current schematic which is the same as the hotkey **Shift-n** or **SPICE Netlist** in the **Sim** menu.
- Second, it runs a script that starts SPICE either on your local machine or on a remote server depending on the configuration (assuming the script is set up).
- When the SPICE job is finished, it starts the Micro Magic NST Waveform Viewer (included with SUE) and loads the SPICE data file (the `.tr0` file) into it. This can also be done by **Ctrl-i** or **Init Probe** in the **Sim** menu.



If you have two monitors on your machine, you can have SUE start NST on the other monitor by typing the line "**setenv PROBE_DISPLAY other**" at the UNIX prompt before starting SUE.

Displaying SPICE Waveforms From SUE

Now you will start to see some of the real power of SUE.

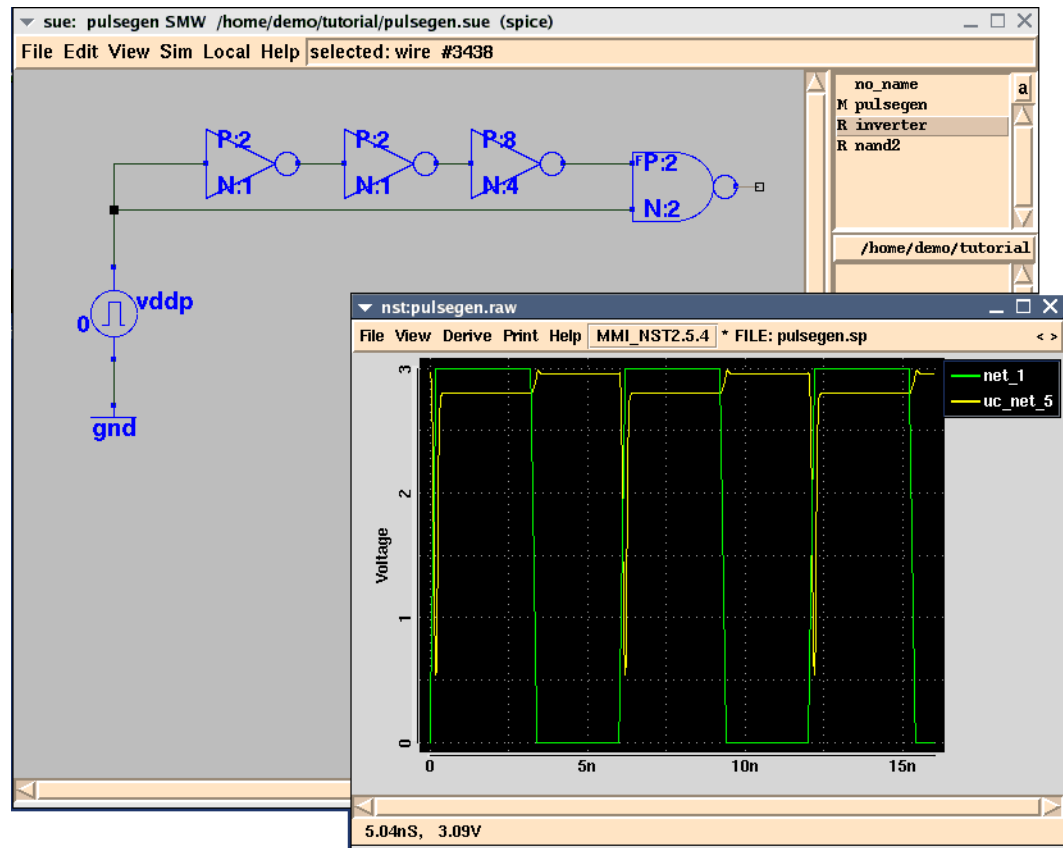
PLOT NETS;
DISPLAY
WAVEFORMS

To see how your circuit simulated, you would simply select one of the wires in your schematic and then type **p** or Plot Net in the **Sim** menu. The waveform for that wire should show up in the NST window.

- Step 1** ■ Select the wire at the top of the pulse icon and hit **p**.
- Step 2** ■ Select the wire (the *open square*) at the output of the **nand2** gate and hit **p**. You might have to carefully select the 'open' on the output of the **nand2** gate if you didn't add a wire there.

Do you see the pulse output? Remember the pulse is low going because we have a **nand2** instead of an **and2**. Your waveforms should look like Figure 14-b.

Figure 14: Displaying SPICE Waveforms: a) “pulsegen” Schematic; b) NST With Two Waveforms



Note that if you changed any of the parameters for the **pulsegen** circuit, your waveforms will look different.

There are lots of plotting options in the **Sim** menu such as **Plot Old Net**, **Unplot Net**, **Plot Net & Remember**, etc. **Plot Old Net** will plot the same net from the last SPICE run. So you can tweak a device size or change a gate, rerun SPICE with **h**, and compare the old and the new waveforms!



Tip

NST can do a lot more than just plot waveforms. You can add panels, print, compute power, and do arithmetic derivations on waveforms! (See the NST manual for details).

PLOT NET HIERARCHY

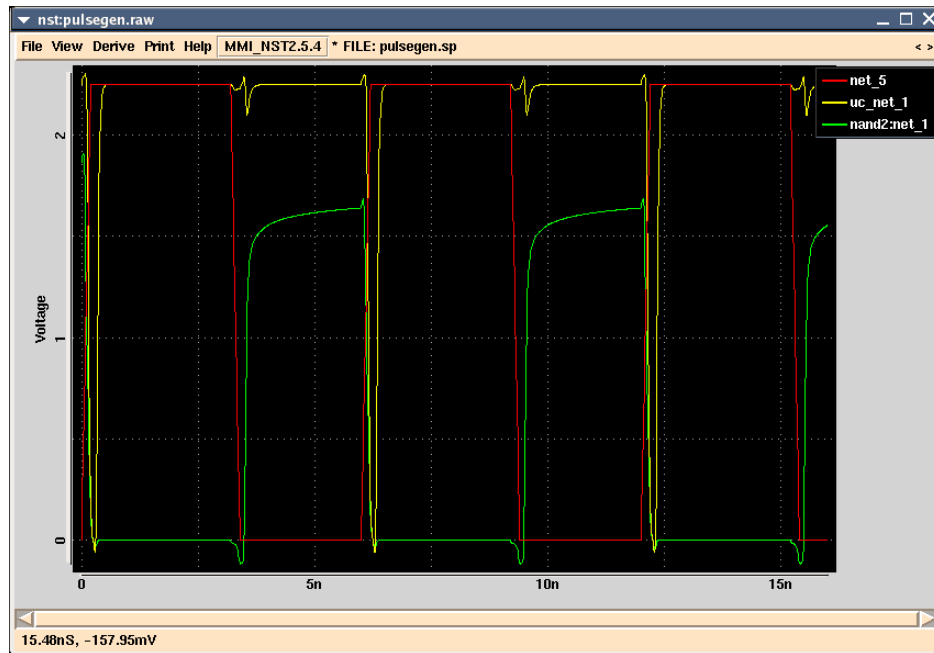
You can plot net waveforms at any level of the hierarchy just as easily as at the top. SUE does all of the messy translations for you.

Step 3

- In this circuit there are only two levels of hierarchy: the one we are looking at, and the transistor level circuits of the inverters and **nand2**. Push into the **nand2** gate (select it and hit **e** or **Push Into** from the **View** menu).

- Step 4**
- Now select the net between the 2 series nfets and hit **p** to plot its waveform. You should now see something similar to Figure 15. (The colors of your plotted nets may be different.)

Figure 15: Three Waveforms Plotted



- Step 5**
- Pop back out to the pulsegen circuit with **Ctrl-e** or **Pop Out Of** in the View menu.



Tip

You can always go directly to a schematic without pushing or popping by clicking on its schematic name in the **Schematic List** box. However, SUE will assume that the schematic you went to is at the top of the hierarchy so you should use push and pop whenever you can.

Net Names When Netlisting

Did you notice that you haven't named a single net, yet you just finished simulating a schematic with a SPICE simulator? In SUE, you only need to name ports and globals. SUE will create unique names for all other unnamed nets.

There are three ways to name a net in SUE:

- By connecting the net to an I/O port with either the “input”, “output”, or “inout” icons.

- By connecting the net to a global signal name with the “global” icon.
- By adding a “name_net_s” icon to a net and labeling it with a name. (Do this by double-clicking [Button-1](#) on the icon, and then entering a net name.)

Otherwise, SUE makes up a name for the net of the form: “net_#”. SUE will also make up unique names for devices and icons if you don't name them.



If you have trouble double-clicking on the small “name_net_s” icons to name them, you might use the **Edit Selected** command in the **Edit** menu, or first name them and then move them onto the desired net.

If you want to see what SUE has named a wire, just select it and look at the **SUE Message Area**. Note that you have to netlist ([Shift-n hotkey](#)) first to see the net name.

Part 4

Higher Level Schematics And Verilog Simulation

Higher Level Schematics

Verilog

We have already simulated our little pulse generator in SPICE. Now we want to run a Verilog simulation. Unfortunately, you can't simply set nodes in Verilog, you need to have a test file to drive them. To accomplish this we will first make an *icon* for our cell, and then wire our icon up to a clock generator that has been provided for you.

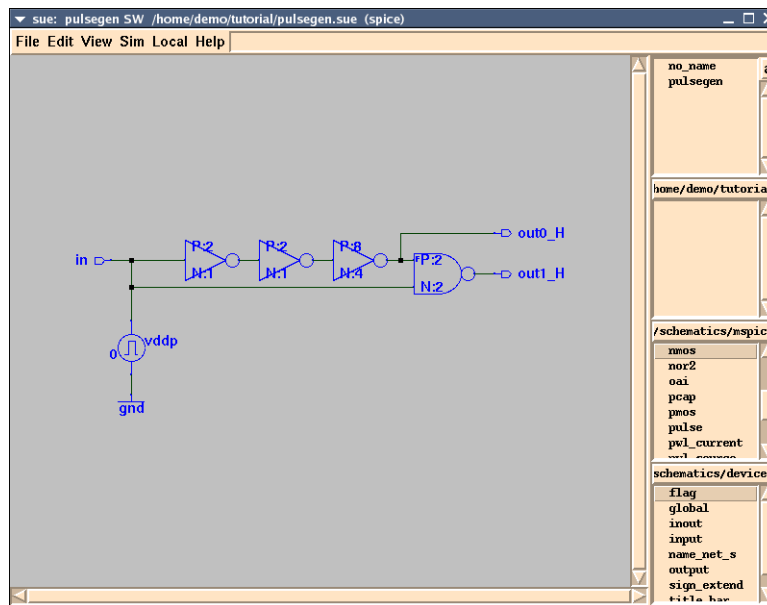
Adding Ports

ADD I/O PORTS

Before we make our icon we need to add some I/O ports to our circuit. We are going to place and name them as shown in Figure 16.

- Step 1**
 - To attach the I/O ports, start by getting an input from the devices Library and attach it to the wire above the pulse. Remember, click with the left mouse button (Button-1) to *select* or get an icon from an Icon List box, and use the right mouse button (Button-3) to *move* an instantiated icon.
- Step 2**
 - Next, attach an output icon to the output of the nand2. Since we would like to look at more than one output, duplicate the output port (*select*, then *d*) and wire it up to the output of the third inverter as shown in Figure 16.

Figure 16: Adding Ports to a Schematic



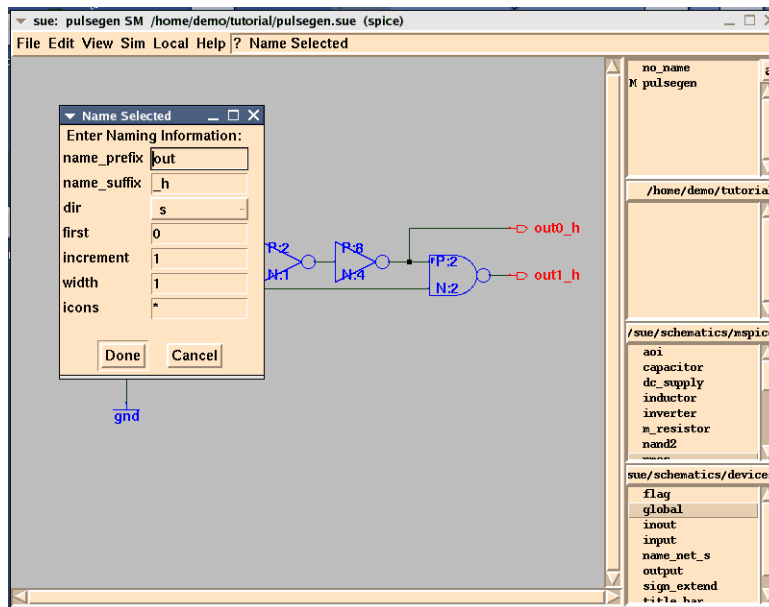
- Step 3** ■ Now name the I/O port. Double-click on the input port to call up the Edit Properties dialog box.
- Step 4** ■ Type “in” into the name box, then click Done or hit Return.

For the outputs we will use a little trick that allows us to name multiple signals. It's not much for only two I/O's, but when you have 32, it helps a lot.

Naming Multiple Signals

- Step 1** ■ Select both of the **output** icons. You can do this by clicking the **left mouse button** on one of the outputs and **Shift-clicking** (hold down the **Shift** button while clicking with the **left mouse button**) on the other output. Or you can simply hold down the **left mouse button (Button-1)** and drag the selection box around both icons. Don't get any other icons (like the **nand2**) or you will name those also.
- Step 2** ■ Go to **Name Objects** in the **Edit** menu. The **Name Selected** dialog box (see Figure 17) will appear. Type “out” in the **name_prefix** box. Type “_H” for **name_suffix**. Change the direction from north (**n**) to south, “s”. This controls the direction of the bus numbering, so if set to **s**, bit 0 is on top and will increment down or south. When you are finished click **Done** or hit **Return**.

Figure 17: Name Selected Popup



Step 3 ■ Save your circuit (hotkey: **Ctrl-s**).

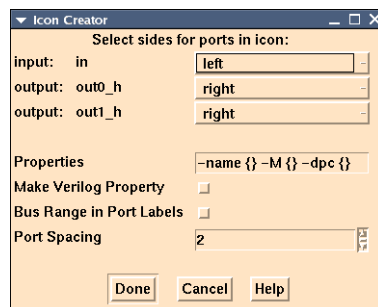
Making An ICON For Our Schematic

CREATING AN ICON

We are now ready to create an icon for the `pulsegen` schematic. Schematics only need icons when you want to instantiate them into other schematics — which is what we are going to do.

Step 1 ■ To make an icon, select **Make Other View** from the **View** menu (or type **Shift-c**). This creates an icon for `pulsegen` and puts you into that icon view. The **Icon Creator Menu** will open, as shown in Figure 18, allowing you to tailor the icon to your needs. For now, we'll be using the default values, so click on **Done**.

Figure 18: Icon Creator Menu



You should now be looking at the **icon view** of **pulsegen** as shown in Figure 19 (below). You can tell it's an icon since the title bar of the SUE window now has as 'I' instead of an 'S' in it. Also, since **pulsegen** now has an icon, it appears in the Icon List box for this directory.

Only the following elements can be placed in icons, which are essentially just pictures:

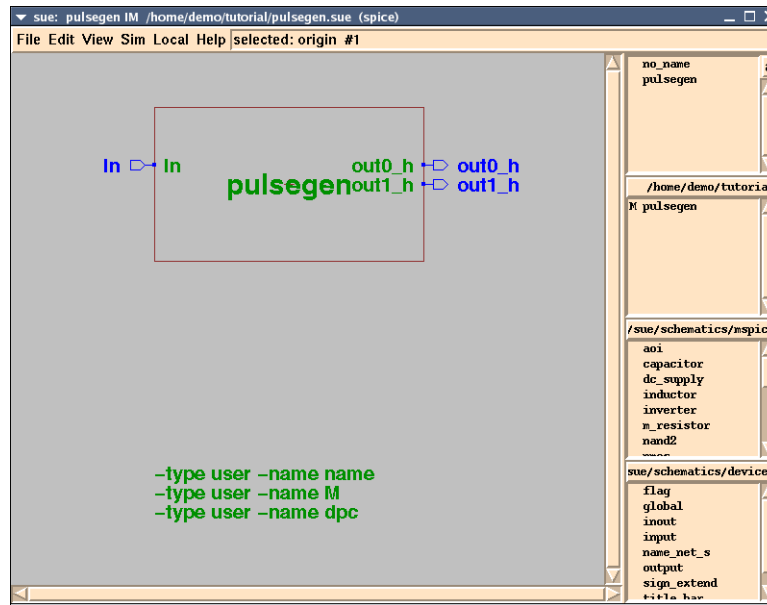
- **I/O's:** *inputs*, *outputs*, and *inouts*. At the locations of the origins of these icons, small solid dots will be displayed when the icon is placed into a schematic. No other icons can be placed into an icon (except a **title_bar** which is for documentation only).
- **The origin of the icon**, which is notated with a "+". All placement, rotation, flipping, etc. of the instantiated icon is relative to this origin. Schematics don't have origins, only icons do. The only thing you can do to the origin is move it around.
- **Property Definitions**, which are text lines that begin with a dash ("-"). Don't add text to an icon starting with a dash unless it is a **Property Definition**. The **Name Property** for the icon, for example, is defined by the line "**-type user -name name**" and will then appear in the **Edit Properties** dialog box if you double-click on an instantiation of this icon (discussed more fully below).
- **Picture elements**, like *lines*, *arcs*, and *text*. Don't add wires to icons.

When SUE makes an icon, it automatically draws a rectangle to represent the icon, and places the schematic name in the center. SUE also places all of the I/O's from the schematic into the icon, with inputs in alphabetical order on the left, and outputs on the right, and adds the **Property Definitions** (the default values **name**, **M**, and **dpc** in this case) entered in the **Icon Creator Menu**:

As mentioned earlier, the I/O icons get replaced with small solid squares. To help identify these ports, SUE automatically duplicates the port label, putting a copy inside the icon rectangle.

Our new icon for **pulsegen** is shown in Figure 19

Figure 19: Icon for Pulse Generator



Editing the ICON

Once your icon has been created, you can further customize it.

You can resize your icon rectangle by double-clicking on it with the **left mouse button** (Button-1). Small squares (called *edit markers*) will appear in the corners of your rectangle, which you can then drag to where you want the vertex to be. If you hold down the **Shift** key, the entire side will move.

You can add new text, or change text within the icon. You can also enter new properties for the icon.

Adding Text

EDITING ICON: ADD TEXT

Let's add some text to our icon. First, we will move the name "pulsegen", and put it in a more convenient and readable location for when we look at schematics with this icon instantiated.

Step 2

- Click on the text "pulsegen" and notice that it becomes active as an object. Using the **right mouse button** (Button-3) drag this text out of the center, toward the upper left of the icon rectangle.
- To add new text, use the **t** hotkey or select **Add Text** from the **Edit** menu. Move the cursor to where you want the text to start and double-click the **left mouse button** (Button-1). Notice that directions are provided in the SUE Message Area to the right of the **Help** menu.

You can use standard editing commands like the arrow keys, backspace, delete, etc. Emacs users will find that simple Emacs commands also work.

- To copy text, hit the hotkey **Shift-t** or choose **Duplicate Text** from the **Edit** menu.



To change the font size of text you must be actively editing the text. Make sure to select it by double-clicking (**Button-1**) on the text. **Button-2** then brings up a menu of text sizes.

You can also change the text anchor using **Button-3**.

In order to personalize our icon, let's add a new property called "**my_name**":

- Step 3**
- Select the text "**-type user -name name**" and duplicate it (**d**). This is just to save some typing. Move the duplicated text underneath the other text.

- Step 4**
- Double-click on the copy and change it to read:

```
-type user -name my_name -default t Bob
```

You just defined a new property called "**my_name**" and set its default value to "**Bob**". The "**my_name**" property will now show up in addition to the other properties when you double-click on the instantiated icon.

DISPLAYING A PROPERTY

Once you've defined a property, you want to use it. The simplest way to use a property is to display it. The way you display a property is by preceding it with a "\$" in a text line.

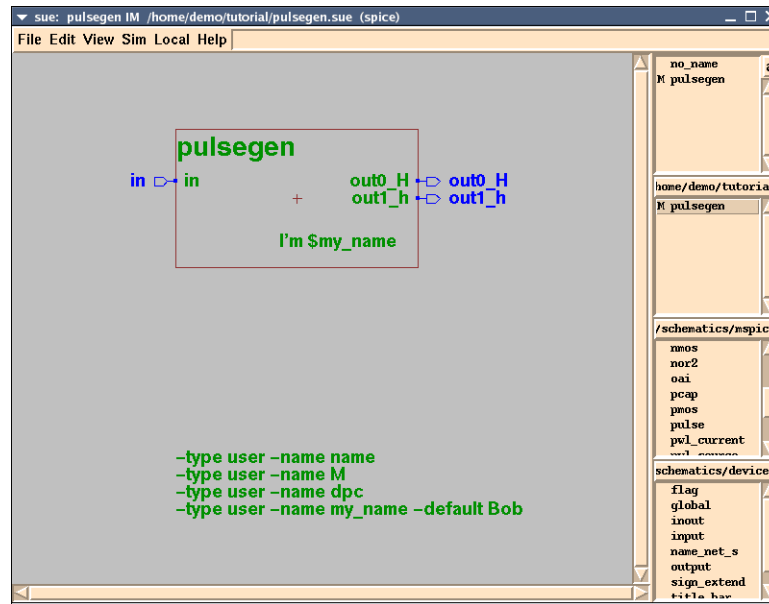
- Step 5**
- Hit **t** to add text, and click **Button-1** inside the icon rectangle. Now type the line:

```
I 'm $my_name
```

and hit Return.

Your icon should now look something like Figure 20.

Figure 20: Icon with Text Added



- Step 6** ■ Now type **c** or select **Swap Views** in the **View** menu to get back to the schematic.

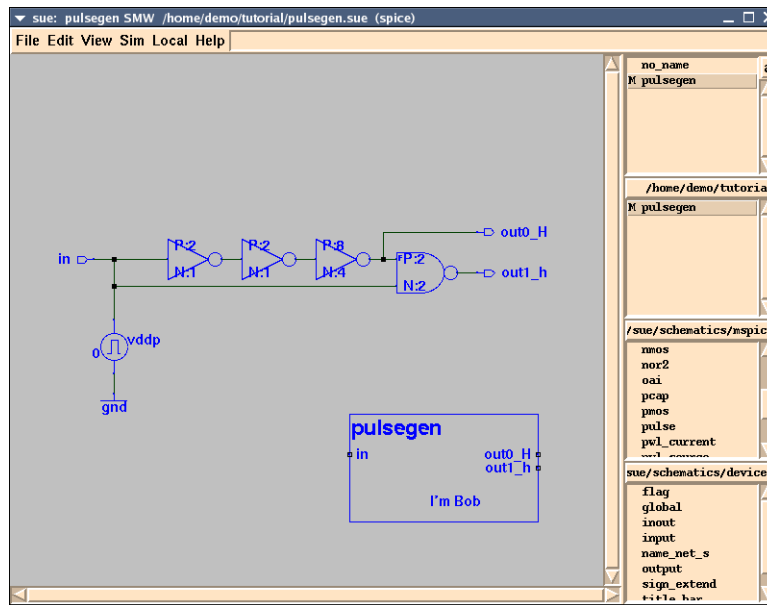
Placing Our ICON Into The Schematic

PLACE ICON IN SCHEMATIC

We are now going to place a copy of the “pulsegen” icon into the “pulsegen” schematic. This is handy for documentation since we can see both the schematic and the icon that goes with it at the same time. Don't worry, SUE is smart enough to know it's the icon for this schematic and to avoid recursive calls when netlisting.

- Step 1** ■ Select “pulsegen” from the Icon List box (the **second** list box from the top).
- Step 2** ■ Place the icon toward the bottom of your schematic, as shown in Figure 21.

Figure 21: Schematic with Icon



Modifying The ICON

Just in case your name isn't Bob (yes, I know there are lots of you) let's change the value of 'my_name' in this instantiated pulsegen icon.

- Step 1**
- Double-click on the pulsegen icon and change "Bob" to your name in the Edit Properties dialog box.

The icon should now say "I'm <your name>". Notice that SUE automatically added the property "my_name" and the default "Bob" to the Edit Properties dialog box.

ADDING TITLE BAR TO SCHEMATIC

Finally, let's add a title bar to our schematic. The title bar is another documentation feature. It automatically includes the schematic name, file name, date modified, and owner to the schematic. You can easily customize it to include your company name, too.

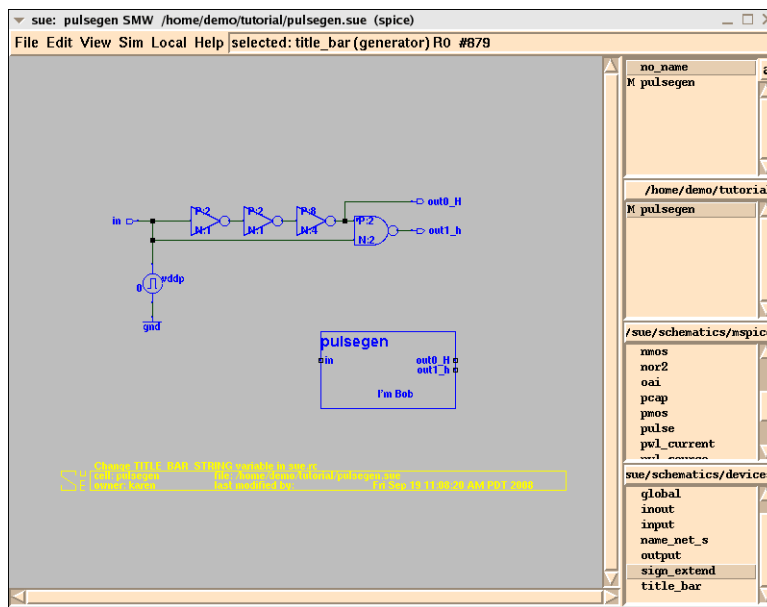
- Step 2**
- Select the "title_bar" icon from the devices Library and place it at the bottom of your schematic.

Your schematic should now look like Figure 22. (The actual text in the title bar will be different than the illustration. Change the text in the title bar by editing the appropriate field in the sue.rc file.)



The **sue.rc** (or **.suerc**) file allows you to set various features within SUE, such as the Title Bar text, colors, hotkey definitions, filename parameters, etc. See the section on “User Customization” in Chapter 1 of the *SUE User Manual* for information and instructions.

Figure 22: Schematic With Title Bar



The “**title_bar**” icon is a generator. If you **Shift-double-click** on it and change it to an “**sccs_title_bar**”, then SUE will put your cell under SCCS control, also, when you save it.

- Step 3** ■ Save your schematic with **Ctrl-s**.

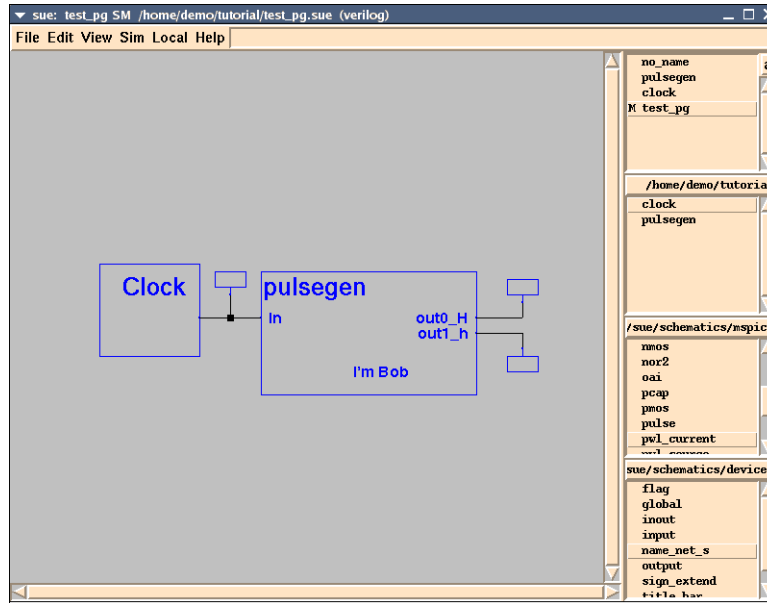
Using Pulsegen

We are now going to build a test circuit to run our **pulsegen**.

- Step 1** ■ Open the file “**clock.sue**” (**Ctrl-I** or Open in the File menu). This is the clock generator we already made for you.

- Step 2** ■ Make a new schematic with **Ctrl-n** or **New Schematic** in the **File** menu. Call the schematic “**test_pg**”. This new schematic is going to look like Figure 23.

Figure 23: Test_pg Schematic



PLACE ICONS TO BUILD CIRCUIT

Now build the circuit.

- Step 3** ■ Drop a “pulsegen” icon
- Step 4** ■ Drop a “clock” icon
- Step 5** ■ Drop three “flag” icons from the devices Library. Attach them to the outputs of the “clock” and the “pulsegen” icons (refer to Figure 23 if needed). We will describe these later.
- Step 6** ■ Wire it up.

SIMULATE IN VERILOG

You are now ready to simulate your circuit in Verilog.

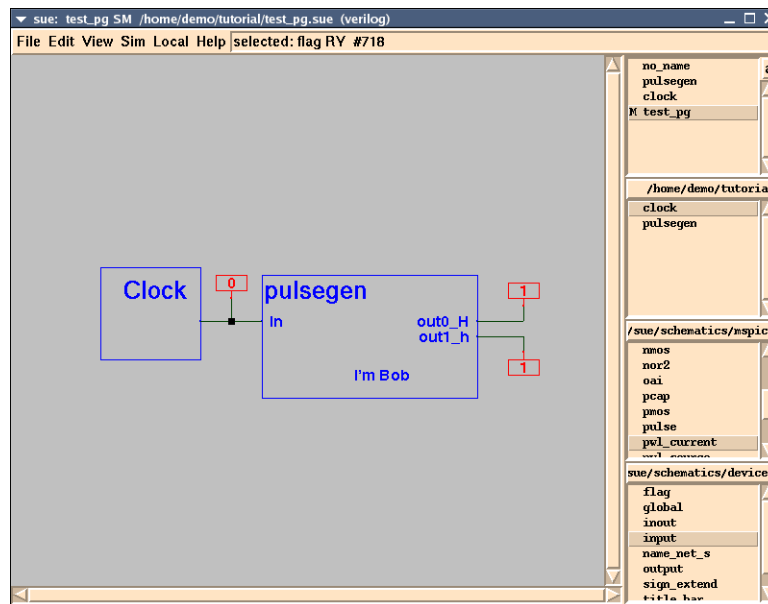
- Step 7** ■ Change simulation mode to Verilog (Change Simulation Mode in the Sim menu, click on Verilog and hit Done or Return).
- Step 8** ■ Hit **Shift-n** or **VERILOG Netlist** from the Sim menu. Notice that the Sim menu changed again since we changed the simulation mode to Verilog.

You just built a Verilog netlist using our “pulsegen” and a behavioral model of “clock” (which has been provided).

Now to run the simulator:

- Step 9** ■ Hit **Ctrl-i** or **Init Probe** in the **Sim** menu. This starts up an interactive Verilog simulation.
- Step 10** ■ Type **s** or **Step Verilog** in the **Sim** menu to step the time forward. This sends several cryptic lines to your Verilog simulator, which causes it to advance the time step 10 units. A “0” should appear in the flag nearest “clock”, and a couple of “1”s on the output flags, as in Figure 24.

Figure 24: Step Verilog Shows Changes in Flags



You probably figured out what the flags are by now. They report the state of the nets they are attached to and are updated after every time step.

- Step 11** ■ Step time a few more times.

The clock is designed to transition every other Verilog time step. Notice that the “out0_H” flag changes every cycle and is the inverse of the clock. This makes sense since it is just the clock followed by three inverters. But what happened to our pulse on “out1_H”?

In SPICE all transitions take a finite time, unlike Verilog where you have to specifically add delays to transitions. What's happening is that the inverters are taking no time (or 0ns), so in Verilog our “pulse” lasts for 0ns! To fix this we will change the inverter model to add a delay.

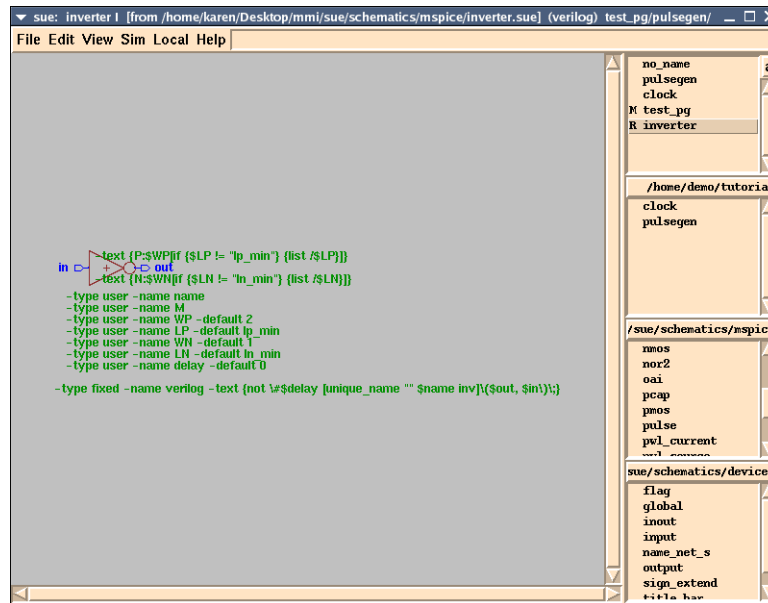
Creating Behavioral Verilog Models

We are now going to edit the schematic so that the inverters have a non-zero delay.

- Step 1** ■ Select and push into “pulsegen” (hotkey: **e**).
- Step 2** ■ Select and push into an inverter (hotkey: **e**).

- Step 3** ■ Now type **c** or **Swap Views** in the **View** menu to switch to the icon view of “inverter”.

Figure 25: Icon View of inverter



- Step 4** ■ Look at the Verilog property for the inverter:

```
-type fixed -name verilog -text {not\#$delay[unique_name]" "$name
inv]\($out,$in)\; }
```

The text above tells the netlist how to netlist this cell when in Verilog Mode. Notice the “**#\$delay**” in the verilog string above. This adds the value of the delay property to the inverter. Because it is a user property, it shows up in the **Edit Properties** popup for the icon. Also, the actual net names are substituted for “**\$out**” and “**\$in**” during netlisting.

- Step 5** ■ Pop back up to pulsegen (hotkey: **Ctrl-e**).
- Step 6** ■ Double click with **Button-1** on each of the inverters and change the delay from 0 to 4.
- Step 7** ■ Save pulsegen with **Ctrl-s**.

SIMULATE
“TEST_PG” CIRCUIT

Now let’s get back to our pulse generator test circuit, “test_pg”, and simulate it again to see if we get a pulse.

- Step 8** ■ Pop back up to test_pg (hotkey: **Ctrl-e**).
- Step 9** ■ Re-netlist by hitting the **Shift-n** hotkey or **VERILOG Netlist** in the **Sim** menu. Then do an **Init Probe (Ctrl-i)** to restart the simulator.

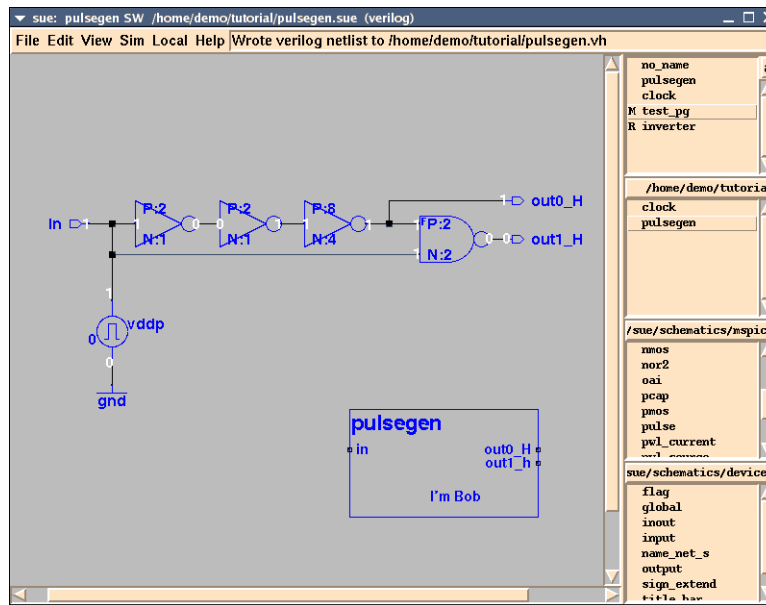
The previous simulation will be stopped and a new one will be started.

- Step 10** ■ Step time (s) a few times and see what happens.

You should see that “out0_H” follows “in”, but with a one-cycle delay. Also, now “out1_H” generates our pulse! (negative going).

- Step 11** ■ Select the pulsegen icon and push into it (e).
- Hit the **Ctrl-o** hotkey or Display Term Values in the Sim menu. Do you see the white 0's and 1's? Look at Figure 26.

Figure 26: Display Term Values



In addition to the flags, you can have SUE display all of the terminal values in a given schematic with Display Term Values. This is great for debugging!

Creating A More Complex Verilog Model

For the inverter in the above example, the Verilog model could be described in a single line. If you need to write a more complex Verilog model, you can attach a Verilog text file to a schematic. Just because SUE looks like a schematic editor doesn't mean that everything you do has to be with schematics.

VERILOG MODEL ENTIRE CIRCUIT

Instead of modeling just the inverter, this time we'll model the entire **pulsegen** circuit from the above example.

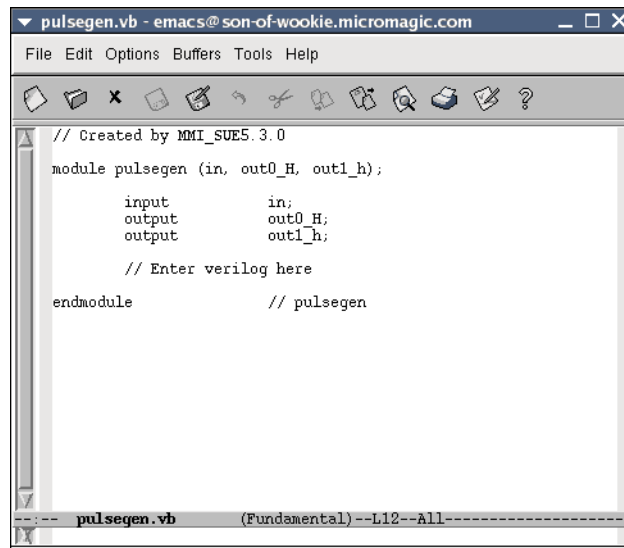
- Step 1** ■ Return to the **pulsegen** schematic if you aren't already there. Make sure none of the icons are selected.
- Step 2** ■ Hit the **Shift-e** hotkey or Edit Verilog in the Sim menu. A dialog box will come up telling you that it can't find a Verilog file for “pulsegen”. Simply hit the Done button and SUE will make one for you, and then bring up the file in your favorite text editor (EMACS, VI, etc.).



SUE chooses the editor to use based on your *EDITOR* UNIX environment variable. For example, to set to *VI*, type “**setenv EDITOR vi**” in UNIX, and restart SUE. The default editor is *Emacs*.

Your text editor file will look like Figure 27.

Figure 27: Template for Verilog Model



Notice that SUE automatically creates a template for the Verilog model, adding the port information to the file. You only need to add the model for “pulsegen” after the line:

```
// Enter verilog here
```

Instead of creating the pulsegen Verilog model, please wait for the next section, which has larger and more interesting example and includes Verilog models at many levels.

Step 3

- Save the file and close the text editor window.
 - If your editor is Emacs (the default), type **Ctrl-x**, **Ctrl-s** to save, followed by **Ctrl-x**, **Ctrl-c** to exit.
 - In VI type **:wq** to save the file and quit.

SUE keeps track of your Verilog file and ‘attaches’ it to your schematic. You can view or edit it any time by simply selecting **Edit Verilog** from the **Sim** menu, as you did before. Since the file exists now, SUE will bring it up for you automatically. You can do this either from the schematic, or by selecting the icon in a higher level schematic.

Notice that SUE has named your behavioral Verilog file “**pulsegen.vb**” (note the “b”). If you want to write and attach your own behavioral Verilog files, they must have the “.vb” extension. Otherwise, SUE won't find them.

You may have also noticed that when SUE creates a Verilog netlist, it gives us the extension of “.vh”. Why doesn't SUE just use the extension “.v” like everyone else? Simple, so SUE doesn't step on your existing “.v” Verilog files. The only time SUE uses the “.v” extension is to import port information from a user-written Verilog RTL file using Load Verilog I/O's in the Sim menu.

Before going on to a more interesting example, let's recap what you have done so far.

You have:

- Drawn a schematic and built its associated icon.
- Simulated the schematic in SPICE and Verilog.
- Learned lots of SUE features.

And you have done all of this from inside of SUE, without once having to run any other tools externally.

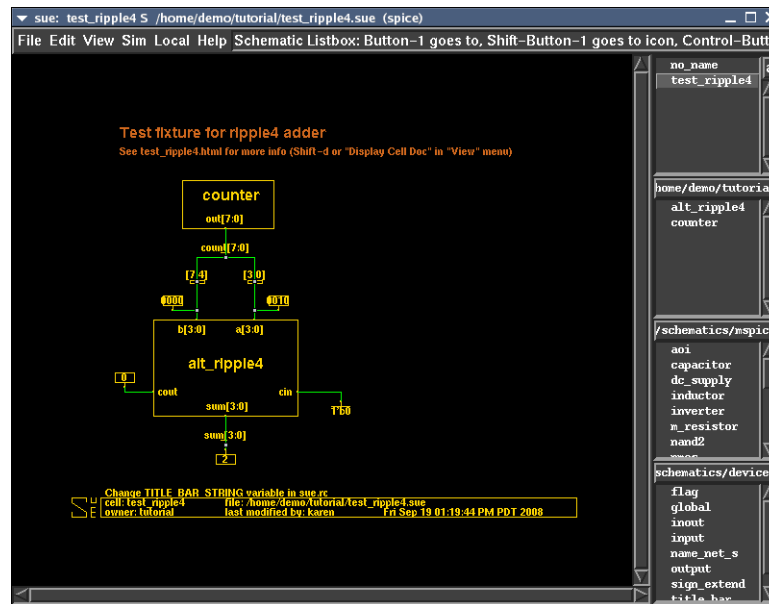
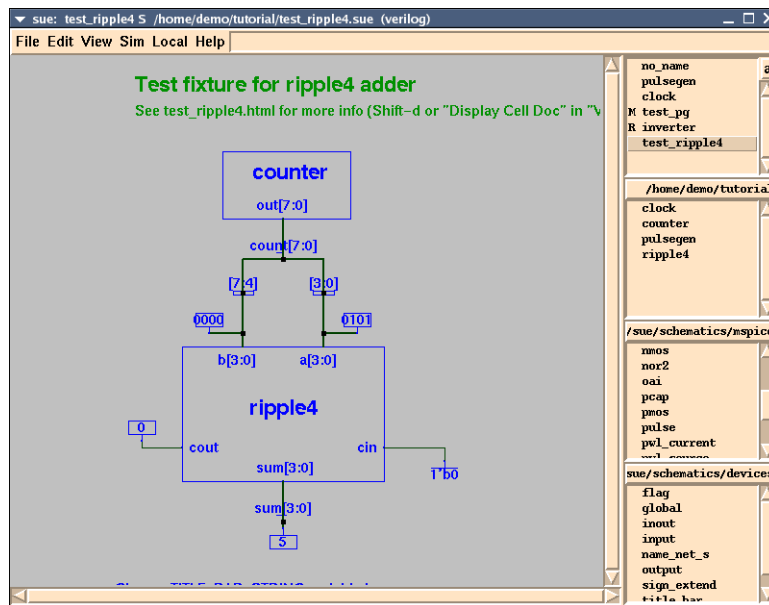
Not bad, huh?

A More Complex Example

Now let's look at something a little more complex and which has a few levels of hierarchy. The circuit is a simple 4-bit ripple-carry adder.

- Step 1**
- Open the file “**test_ripple4.sue**” located in the tutorial directory. You should see something similar to Figure 28. (If using the black-background color scheme, the file would look like Figure 28-b.)

Figure 28: Demo “4-bit ripple-carry adder” Schematic



This example has text documentation along with it as described by the text in the schematic. Let's look at it.

Caution



If you are reading this tutorial in a browser and bring up the documentation, it will replace this tutorial in the browser window. After reading the cell documentation, just hit the “Back” arrow in your browser to return to this tutorial.

Step 2

- Hit **Shift-d** or **Display Cell Doc** in the **View** menu to view the attached documentation for this cell.

You can also attach plain text files, in place of HTML files. SUE looks for the “.html” or “.doc” extensions to files with the same names as the schematic you are editing. It brings up “.html” files in your browser, and “.doc” files in your text editor.

Step 3

- Now look over the **test_ripple4** example. Push into all of the cells. This example has documentation, buses, and lots of other interesting things not covered in your simple **pulsegen** example.



SUE is smart enough to remember where you just were. Let's say you just pushed down five levels and you want to pop back up four levels. To accomplish this just repeat **Ctrl-e** four times. To push back down type **e** four times. This is because selected objects stay selected even when you switch cells and return.

PUSH INTO/ POP OUT OF NETS

Step 4

- Select the net which connects to the “cout” pin on the “ripple4” icon.

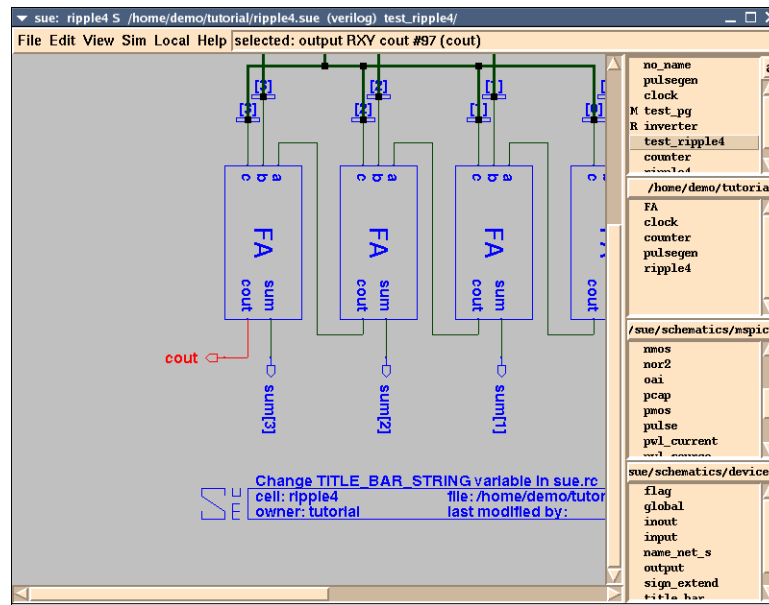
Step 5

- Type hotkey **e** to push into the icon.

Step 6

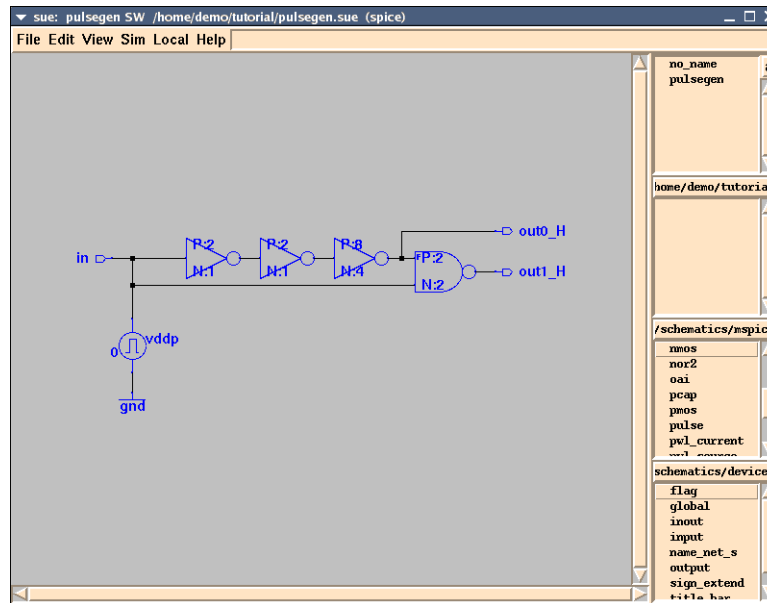
- You should see something like Figure 29. SUE pushed down into the instance the net is connected to, highlights that net in the **ripple4** schematic and zooms in on it.

Figure 29: Push Into Selected Net



- Step 7**
 - Type **e** again, and SUE now pushes you into the “FA” schematic and highlights the `cout` net.
- Step 8**
 - Type **Ctrl-e** to go back to the `ripple4` schematic, and then type **v** to view the entire schematic.
- Step 9**
 - Select the `a[3:0]` net, then type **Alt-e** (or select **Pop Connected** from the **View** menu). You should see something like Figure 30. SUE popped up one level of hierarchy, and highlighted the net which connects to `a[3:0]` and the instances connected to this net.

Figure 30: Pop Up In Hierarchy Of A [3:0] Net



Displaying Design Hierarchy, and Controlling The Simulation

One of the more powerful features in SUE is that it can change what you simulate on the fly. The Display Design Hierarchy feature allows you to not only display the hierarchy, but to control what level of the design gets simulated.

During netlisting, SUE uses a very simple algorithm to decide what to do with subcells. If the subcell icon has a Verilog property and you are in Verilog simulation mode, then it uses that property to netlist the cell, ignoring any schematics or hierarchy below it. Otherwise it netlists the corresponding schematic.

Let's look at the ripple4 icon to see if there is a Verilog property.

- Step 1**
 - Select the “ripple4” icon and push into it with the hotkey **e**. You are now looking at the ripple4 schematic. We need to look at its icon.
- Step 2**
 - Hit **c** or Swap Views in the View menu. Now you should be viewing the ripple4 icon.

You should see a line that looks like:

```
-type auto -name verilog -text ...
```

This is the Verilog property. It tells SUE exactly how to netlist this icon in Verilog. If this seems complicated, realize that it is simply a function of the name of the icon and the port names and can be generated with the Create Verilog Property command.

- Step 3**
 - Hit Create Verilog Property in the Sim menu and move the text it creates below the existing Verilog property. You should see that they are the identical.

- Step 4** ■ Once you have verified that they are the same, remove the second copy (hotkey:q).

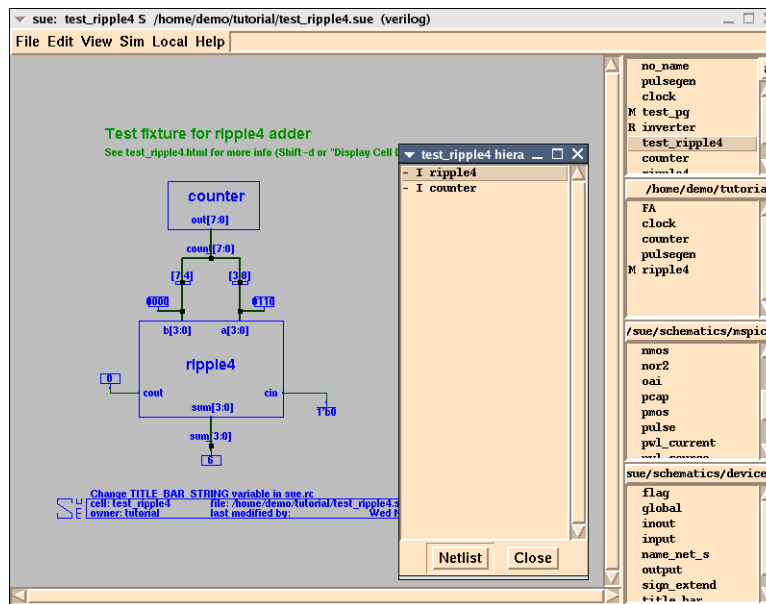
If you wanted to simulate the ripple4 schematic, you could remove this property. Instead of removing the property, we can tell SUE to ignore it with the Display Design Hierarchy command.

Using Display Design Hierarchy To Netlist

Now we are going to use Display Design Hierarchy to control the netlisting.

- Step 1** ■ Return to “test_ripple4” by hitting Ctrl-e or Pop Out Of in the View menu.
- Step 2** ■ If you aren't already in Verilog mode, change simulation mode to Verilog (Change Simulation Mode in the Sim menu, click on Verilog and then Done or Return). The title bar will show the current simulation mode.
- Step 3** ■ Netlist the test_ripple4 schematic (Shift-n).
- Step 4** ■ Now select Display Design Hierarchy in the Sim menu. It will netlist your cell and pop up the Design Hierarchy menu on top of your schematic, as shown in Figure 31.

Figure 31: Design Hierarchy Dialog Box



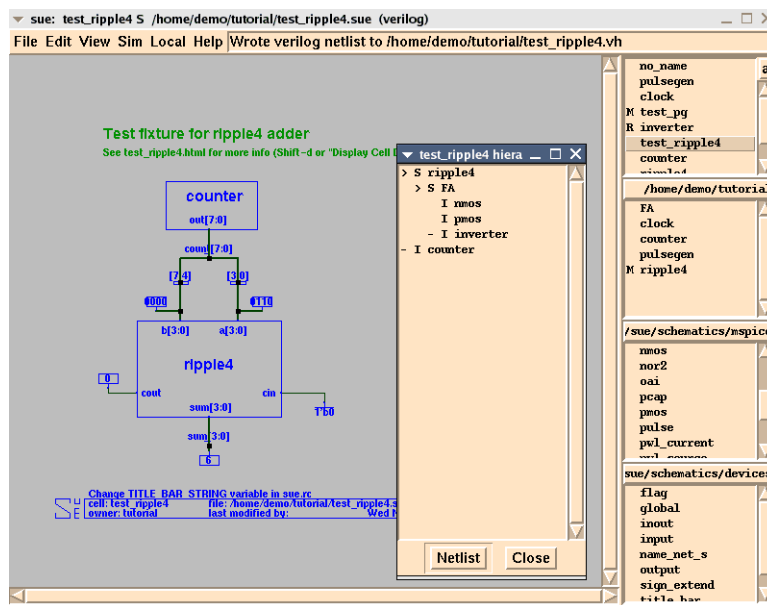
The Design Hierarchy popup tells you that the two icons “ripple4” and “counter” were netlisted with their Verilog properties. Let's tell SUE to netlist the ripple4 schematic and ignore its Verilog property:

- Step 5** ■ Click with Button-1 on “- I ripple4” in the Design Hierarchy popup.

The “1” in - 1 ripple4 should have changed to “S”, meaning that the Verilog property is now ignored for this icon and the *schematic* will be netlisted.

- Step 6** ■ Hit the Netlist button in the **Design Hierarchy** popup. SUE re-netlists this schematic and now shows you that there is a “FA” icon whose Verilog property is being used.
- Step 7** ■ Click with **Button-1** on “- 1 FA” in the **Design Hierarchy** popup and then click on Netlist. You should see the devices contained in the FA cell (see Figure 32). Likewise the `test_ripple4.vh` file will include all of this hierarchy.

Figure 32: Expanded Design Hierarchy



Mixed-Mode Verilog Simulation

When we simulate our circuit in Verilog, SUE uses the following Verilog command line arguments “+libext+.vb+.v -y . -y lib.v”. These instruct Verilog to look for any undefined modules in either the current directory (“.”) or the subdirectory (“lib.v”) and look in files with extensions of “.vb” or “.v”. (Note: you can add to or change these options in the `.suerc` file — see the Micro Magic, Inc. *SUE User Manual* for more information.)

DEFINE VERILOG MODELS

Therefore, we need to make sure that we have defined Verilog modules for any icons whose Verilog properties we use (unless they are included inline in the icon view, like the inverter example above). Let's make sure we have the “.vb” files:

- Step 1** ■ You should be in `test_ripple4`. If not, pop up to it with **Ctrl-e**.
- Step 2** ■ Select the `ripple4` icon and hit **Shift-e** or **Edit Verilog** in the **Sim** menu to view the Verilog. This will bring up the Verilog in an editor.

You should see that we have defined the adder behavioral model with the line:

```
assign {cout, sum[3:0]} = {1'b0, a[3:0]} + {1'b0, b[3:0]} + {4'b0, cin};
```

So if we netlist `test_ripple4` with the `ripple4` Verilog property, our simulation will be very fast since it only executes this one line instead of the underlying hierarchy. Let's do it.

- Step 3** ■ Quit out of the editor.
- Step 4** ■ In the **Design Hierarchy** popup, make sure the you see “- I ripple4”. If ripple4 shows an “S” for *schematic*, click on it to return to “I”. Then hit Netlist.
- Step 5** ■ Look at `test_ripple4.vh` in a text editor. Do you see how short that file is? Look at the `test_ripple4` module.
- Step 6** ■ Hit **Ctrl-i** or **Init Probe** in the **Sim** menu to start up Verilog. Do you see the Verilog command line arguments “+libext+.vb+.v -y . -y lib.v” in the window you started SUE from?
- Step 7** ■ Hit **s** a few times to step the time forward. You should see the flags changing.
- Step 8** ■ Select the `ripple4` icon and push into it (hotkey: **e**). Select one of the **FA** icons and push into it (hotkey: **e**). Hit **Ctrl-o** or **Display Term Values** in the **Sim** menu. You won't get any values. Why? because this schematic is not part of the simulation. The netlist doesn't include all of this hierarchy.

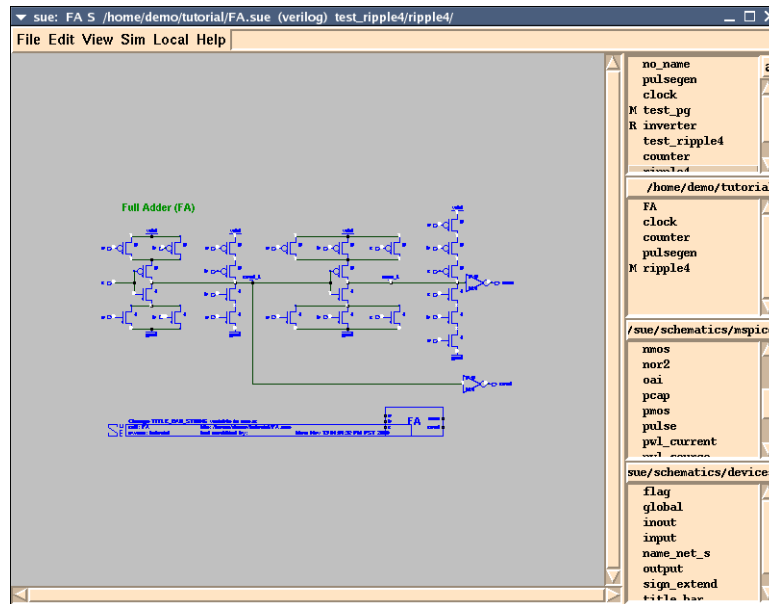
SIMULATE WITH ALL HIERARCHY

Now let's simulate with all of the hierarchy:

- Step 9** ■ Click with **Button-1** on `-I ripple4` in the **Design Hierarchy** popup and then hit Netlist. (This will also bring you to your top cell: `test_ripple4`.)
- Step 10** ■ If you see “-I FA” then click on that line to get the **FA** schematic and hit Netlist. Now we will simulate three levels of hierarchy.
- Step 11** ■ Look at `test_ripple4.vh` in a text editor. Do you see that there are now modules for **FA** and `ripple4`?
- Step 12** ■ Hit **Ctrl-i** or **Init Probe** in the **Sim** menu to start up Verilog.
- Step 13** ■ Hit **s** a few times to *step the time* forward. You should see the flags changing.
- Step 14** ■ Select the `ripple4` icon and push into it (hotkey: **e**). Select one of the **FA** icons and push into it (hotkey: **e**). Hit **Ctrl-o** or **Display Term Values** in the **Sim** menu.

You should now get values since the **FA** schematic IS part of the netlist. Refer to Figure 33.

Figure 33: Term Values in FA Schematic



By changing the **Display Design Hierarchy** in SUE, the designer or verification person can change what is being simulated on the fly. Thus, schematics can be “flattened” to verify that they are correct down to the “transistor” level, or simulations can be done at very high-level behavioral levels to increase speed.

In addition, mixed-level simulations can also be done. That is, simulate high-level behavioral models for some blocks, and flatten others down to the transistor level.

Most importantly, what you see in the **Display Design Hierarchy** window is what you get when you simulate. You never have to wonder what you are actually simulating!

When you are done looking over the adder example go to the next section. You can also run SPICE on this example.

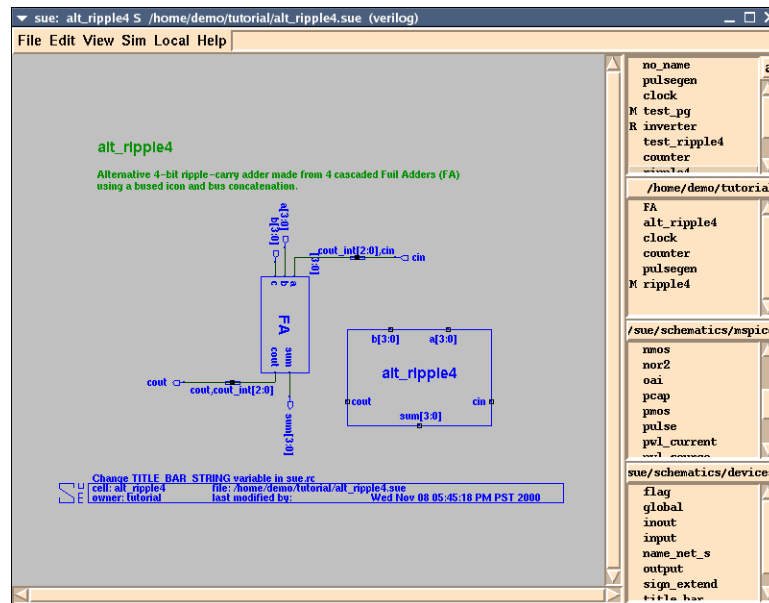
Buses

SCHEMATIC WITH BUSES

While the “**ripple4**” example circuit uses simple bused wires or buses, it doesn't demonstrate bused instances or bus concatenation. An alternative “**ripple4**” schematic was created to demonstrate these features.

- Step 1** ■ Open up the schematic “**alt_ripple4**”. It should look like Figure 34.

Figure 34: "alt_ripple4": Alternative Ripple-Carry Adder Schematic



- Step 2** ■ Notice that the 4 FA icons have been replaced by one icon whose name is "[3:0]". Simply naming an icon with a Verilog bus range will cause it to be duplicated that many times.
- Step 3** ■ Look at the "a" input to the FA icon. We want this to be connected to the "cout" output of the previous FA, or "cin" if it is the first one. This is done by labelling the net with a name_net_s icon with the name "cout_int[2:0],cin". The comma (",") causes a Verilog bus concatenation. "cout_int[2:0]" are the three nets that go from "cout" on one FA to "cin" on the next one. For more examples of buses, see the SUE User Manual.

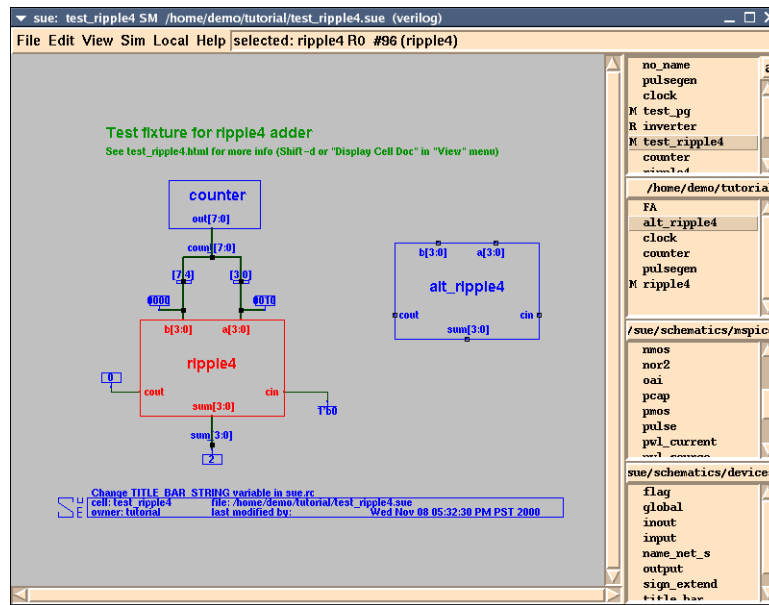
REPLACING AN ICON

You can replace "ripple4" in `test_ripple4` with "alt_ripple4" and simulate it to see that it works the same way as ripple4.

- Step 4** ■ Go back to the `test_ripple4` schematic by clicking the left mouse button on it in the *schematic* list box (the top one).
- Step 5** ■ To replace the ripple4 icon with alt_ripple4, we need to first place an instance of alt_ripple4 into the test_ripple4 schematic. Left click on alt_ripple4 in the *icon* list box (the second one) and place this icon somewhere in the schematic.
- Step 6** ■ Select the ripple4 icon. Your schematic should now look like Figure 35. Select Replace Instance from the Edit menu or type b. Then click with the left mouse button on the alt_ripple4 icon in the schematic.

Notice that ripple4 has now been replaced with alt_ripple4. This works if the icons are the same size and the ports in the same location.

Figure 35: Replacing “ripple4” Icon with “alt_ripple4” Icon in Schematic



- Step 7 ■ Delete (hotkey: q) the extra alt_ripple4 icon.
- Step 8 ■ Netlist test_ripple4 (Shift-n) with initialize Verilog (Ctrl-i).
Hit s a few times to *step the time forward*. You should see the flags changing just as they did when you simulated with ripple4.

Part 5

Cross-probing with MAX

Cross-Probing With MAX

In this section we are going to bring up a schematic in SUE and a layout in MAX, the MMI full-custom layout editor. Using interprocess communication, SUE and MAX will then share data. In this case, you will be able to select any net in a schematic in SUE and tell MAX to highlight that net in the layout, and vice-versa.

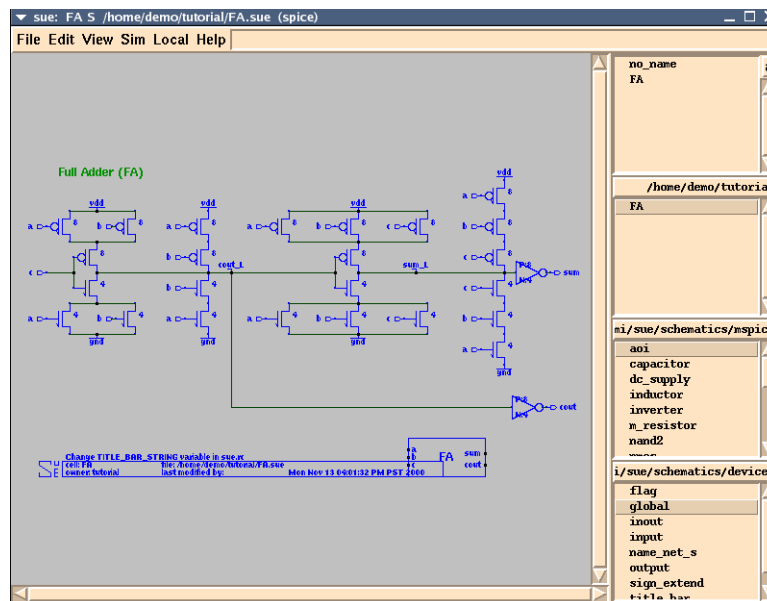


You will need a license for MAX to run this section. If you don't know if you have a MAX license, just type "max" at the UNIX prompt. If MAX loads, you have a license.

Step 1

- If you don't have SUE running at this time, start it, and load the FA cell. If you are continuing from before, click on the FA cell in the Schematic List box. Your screen should look like Figure 36.

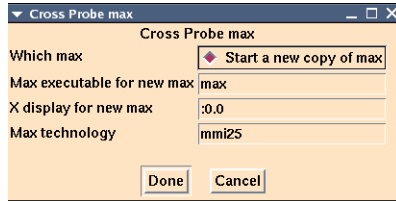
Figure 36: "FA" Schematic



Since cross-probing to layout is a transistor-level operation, you need to be in either spice or sim simulation mode.

- Step 2** ■ If you are not sure that you are in spice or sim simulation mode — look at the SUE Title Bar. If you are unsure, go to Change Simulation Mode... in the Sim menu and change to spice or sim.
- Step 3** ■ Select Max Cross Probe Init in the Sim menu. the Cross Probe pop-up will appear, as shown in Figure 37. Be sure that “Start a new copy of MAX” is selected. Type in `mmi25` for the MAX technology. Then click on Done. This will set up the cross-probing as described below.

Figure 37: MAX Cross-Probe Popup



- A copy of MAX will start, and this will load the corresponding MAX layout file (`FA.max`, for this example).
- Next, MAX and SUE both create sim netlists of their current layouts/ schematics.

SUE doesn't have to be in sim mode for this.

- Finally, LVS using **Gemini** (a netlist compare program from the University of Washington) is run to compare the schematic and layout netlists and determine node equivalents. The nets which have correspondences are selected in both SUE and MAX.

- Step 4** ■ To cross-probe a specific wire in SUE, select one wire as shown in Figure 38 and type the **hotkey k** or choose Max Cross Probe from the Sim menu. The wire that you selected will be highlighted on the layout in the MAX window, as shown in Figure 39.

Figure 38: Net Selected In SUE

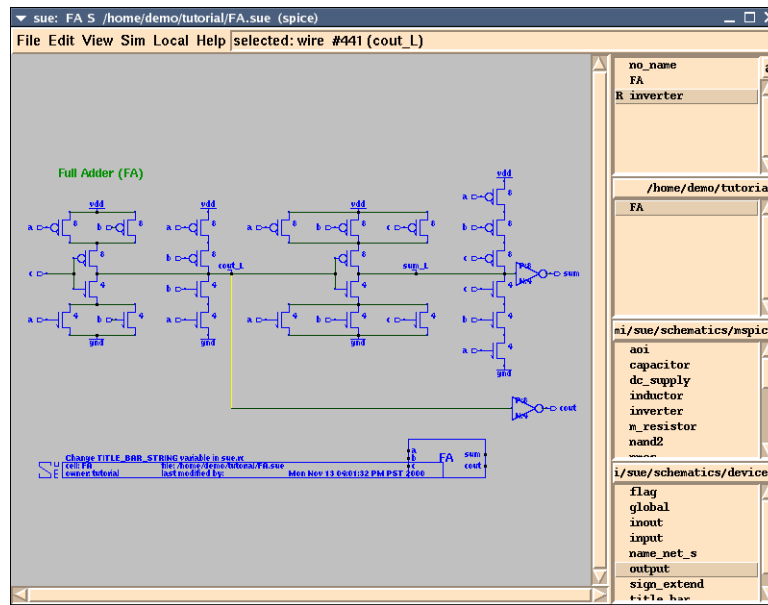
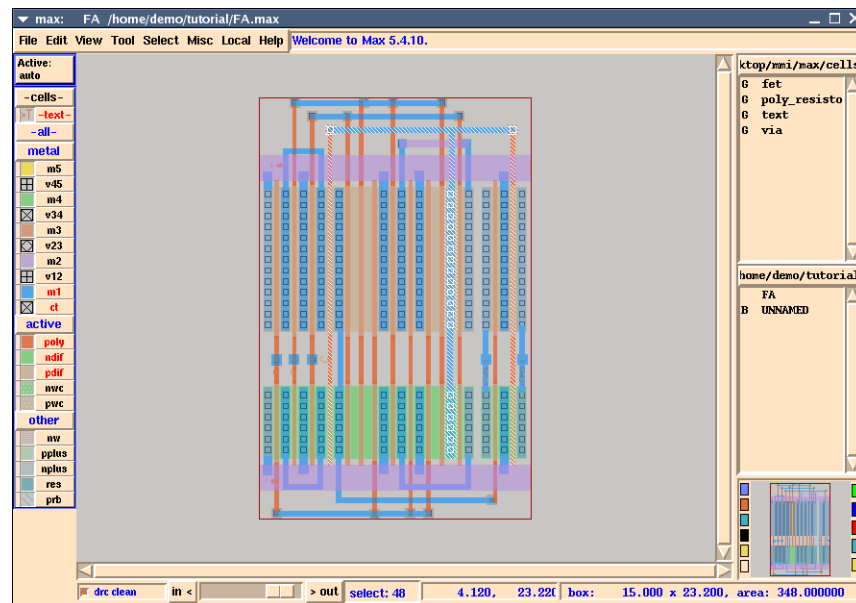


Figure 39: Highlighted Net In MAX



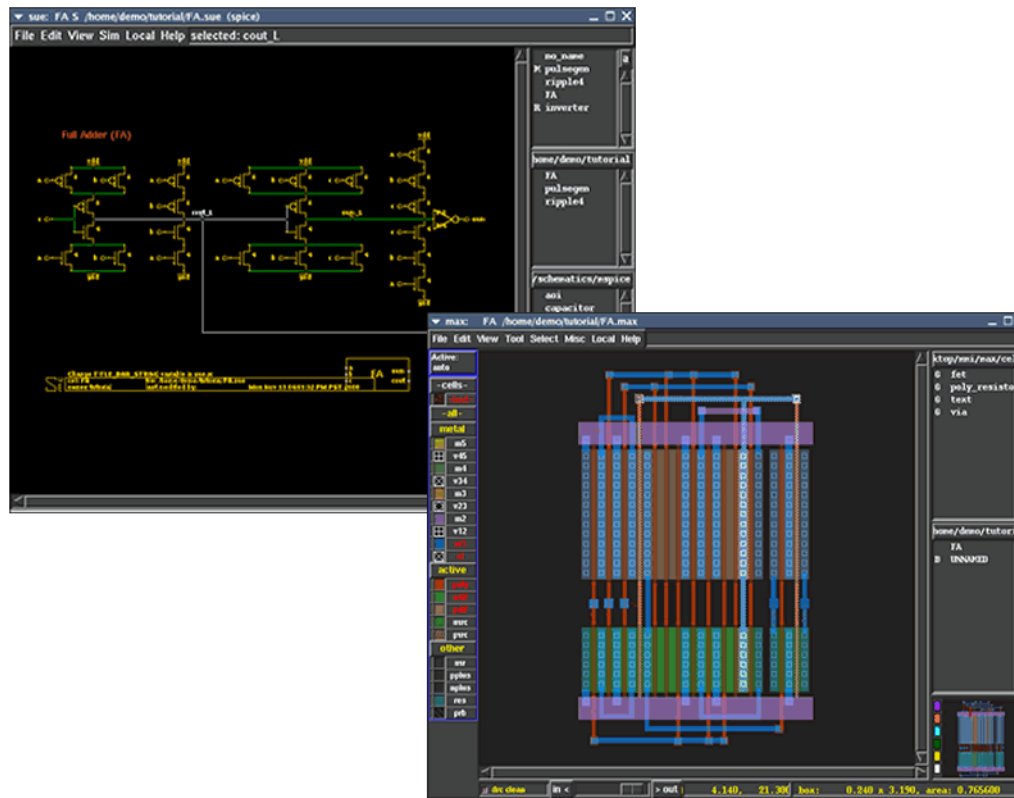
Step 5

- Now select a different net in MAX by moving the cursor over a net and hitting s. You can also select a net in the current hierarchy by simply clicking on it with the left mouse button (Button-1).

- Step 6**
- Once you have selected a net in MAX, hit **k**, or select **SUE Cross Probe Init** from the **Tool** menu. The corresponding net should be selected in the schematic in SUE.

Figure 40 shows what the windows would look like if you were using the black-background color scheme.

Figure 40: Highlighted Nets in SUE and MAX Using the Black-background Color Scheme



In addition to cross-probing, MAX is a very fast and powerful, full-featured, full-custom layout editor with a complete language API. Refer to the Micro Magic, Inc. MAX User Manual and Micro Magic, Inc. MAX Tutorial (accessed from either the **Help** menu in MAX, or from the Micro Magic, Inc. Documentation Guide) for more details.

Now take your left hand and raise it over your head and pat yourself on the back. You survived the SUE tutorial, and hopefully learned a little about SUE. However, the best way to really learn SUE - or any tool - is to try it out with something you designed yourself. So go ahead, and good luck!