

Using Design Architect Schematic Capture with Spice

Hardy J. Pottinger
(With some modifications by Daryl Beetner)
Department of Electrical and Computer Engineering
University of MO - Rolla
hjp@umr.edu

Purpose

The purpose of this tutorial is to familiarize you with Design Architect (or just plain *da*). DA is Mentor Graphic's schematic capture program that runs on the Sun IPX workstations in the Department of Electrical Engineering. During the course of this tutorial, you will use Design Architect and Accusim (Mentor's version of Spice 2g6) to design a two input CMOS NAND gate to meet certain drive and timing specifications. Although this is a fairly simple process, you will go through most of the steps needed to design other, more complex parts. You can use Accusim without da but many of the powerful design capabilities of Accusim are unavailable when using spice files alone and Accusim seems to run much faster with a design in schematic form instead of a spice netlist. There are several advantages to using a schematic over a netlist. Foremost is the ability to modify circuit parameters (properties) without leaving the simulator. Another handy feature is the ability to specify nodes by name or simply pointing to them on the schematic instead of having to remember spice node numbers.

This tutorial will tell you how to do some operation one way and one way only for the most part. Be aware however that there are typically a myriad of ways to do any one thing in particular in Mentor applications. You can use anything from menus, commands, and strokes to function keys and can even write command scripts or macros to customize Mentor applications. Feel free to experiment on your own and develop your own techniques.

This tutorial will by no means make you an expert at using Mentor tools. In particular there is no attempt made to show you how to recover from anything other than the simplest of errors. Should you wish to explore further, you can read through the Design Architect User's Manual or the Design Architect Reference Manual. There is also a much more in depth Mentor supplied tutorial called the Design Architect Training Workbook but be prepared to spend about 40+ hours on that! There are also various Getting Started With... manuals that can prove to be useful. All these and more are stored on cdrom, are kept on-line, and can be read with the Mentor application `bold_browser`.

Notation

Throughout this tutorial, commands you type will be printed in `typewriter font`. The symbol (`↵`) means to press the Enter or Return key. Mouse commands are similar to those in the familiar MS-Windows. The mouse buttons will be referred to as LMB, MMB, and RMB for left, middle, and right mouse button. Menu commands will be shown as **Menu>Item** where Menu refers to the main pull down menu item or the popup menu title and the greater sign is used to separate menu items. Click means to rapidly press and release the indicated mouse button as in 'click the LMB'. Double click means to click twice in rapid succession. Drag means to hold the mouse button down while dragging the cursor to a new position, then releasing the mouse button to complete the command. The terms *window* and *screen* will be used interchangeably. The key Ctrl-D means to hold down the control key while pressing the d key. Alt-F1 and Sh-F1 similarly mean to hold down the Alt (or Shift) key while pressing the F1 function key.

Getting Started

In order to use any Mentor application, you'll need to have an account for the workstations in 106EE. These are Unix™ workstations and, although it helps to be familiar with unix, Mentor applications isolate you to some extent from its idiosyncrasies. This tutorial assumes little or no familiarity with either unix or the Sun workstations.

Before proceeding further, you should have some previous experience with the Mentor Graphics tool Design Architect. The tutorials Digital1 and Digital2 (or CpE 112 lab) will provide you with that experience.

Entering a Schematic with Design Architect

Now we'll bring up Design Architect (called da) which is the schematic capture program. You'll use da to construct a schematic model for a two input CMOS nand gate like that shown in Figure 1. CD to the directory you plan to place your design, type "sul" and "swd" to set up the mentor environment, then start Design Architect by typing the command `da_`. After a few moments the da window will appear. It will look something like that in Figure 1 without the circuit of course.

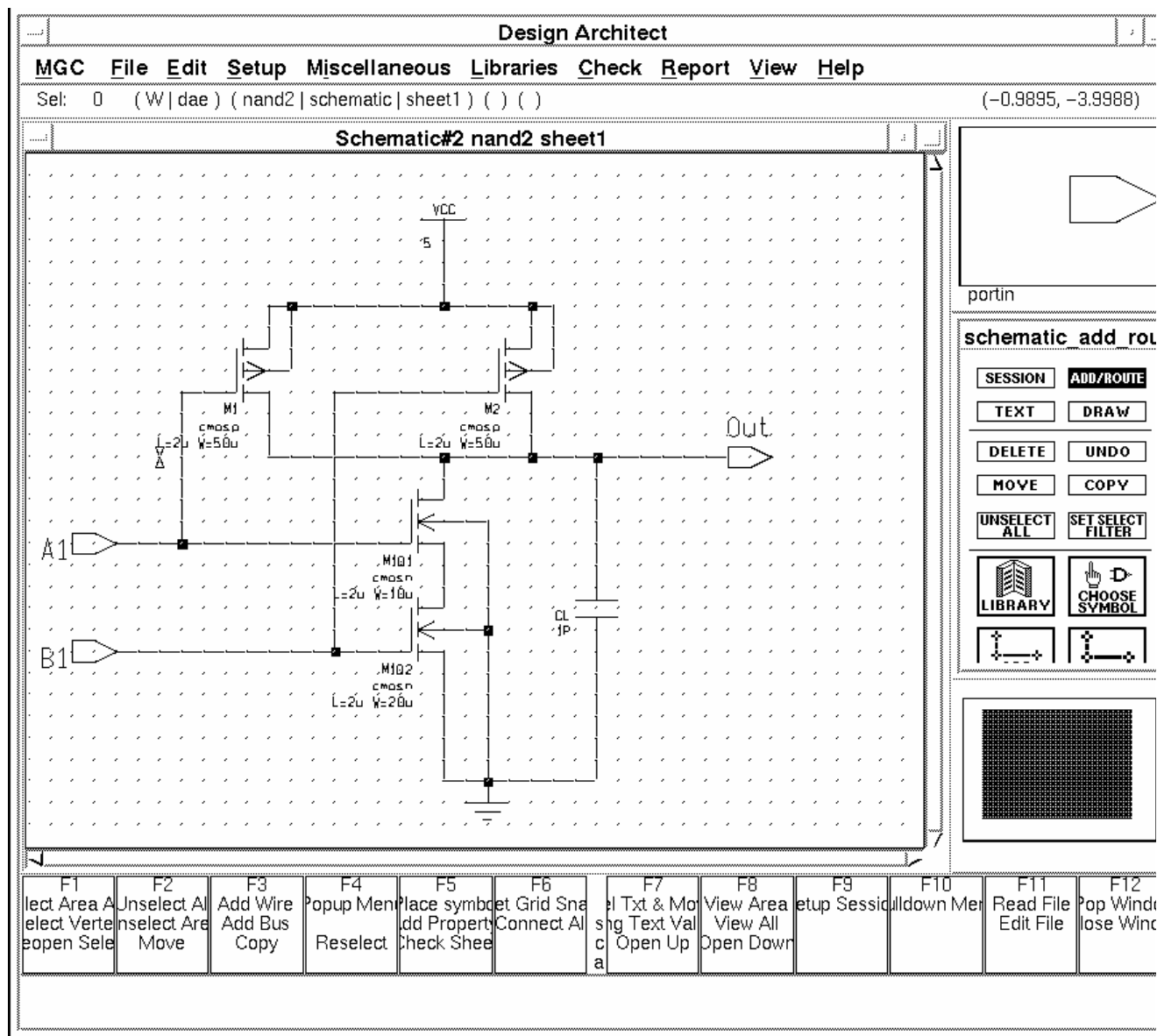


Figure 1 Design Architect Window

Design Architect is one of several Mentor tools that use their Falcon Framework user interface. Along with the main menu bar at top and the softkey legend at bottom, there are three windows along the right hand side of the screen: the active symbol window, palette, and context window. These windows can be turned on and off by the main menu item `Mgc>Setup>Session`. Take a moment right now and click the RMB (menu button) in each of these three windows to see what kind of popup menu is available in each. Make a note of the menu items in your notebook. The symbol window is where part symbols from a library will appear during placement, the palette window contains a variety of command icons that are useful shortcuts for various menu commands, and the context window shows a miniature version of the schematic as a kind of road map.

From the session palette, open a schematic sheet by clicking the LMB on Open Sheet. This will bring up the Open Sheet dialog box. Type in the name *nand2* in the **Component Name** box and OK the dialog. Do not ever change the sheet name! The first time you open a component, da will bring up an empty schematic sheet window. Notice the line just under the main menu bar and just above the schematic sheet. It should read

something like: Sel: 0 (W|dae) (nand2|schematic|sheet1) () (). The Sel field is the *selection count*. We'll get back to that later after you place some parts.

You'll need seven parts for the nand2 schematic: an nMOS fet, pMOS fet, input and output ports, power and ground symbols, and a capacitor. These all come from the generic lib library. Start by selecting the Libraries>MGC Analog Libraries item from the main menu. This will switch the palette to display a list of libraries. Select the Generic Parts entry by clicking the LMB on it. That will display a list of part symbols from the generic analog library. Before going any further, let us navigate through the various palette menus first. Select the Palette>Back or Palette>Root popup items to return to the main analog library selection menu. Click LMB on the Generic Parts entry to return to the generic lib icons. Click the LMB on the session (client) window area (this is the large gray area surrounding the schematic window to bring up the main session palette with the Open Sheet button and click the LMB on the schematic window to bring back the library palette. If you get lost, you can always go back to the main menu and select Libraries>MGC Analog Libraries. Leave the palette with the list of part icons from the library displayed.

M1 and M2 are both PENH4 parts (see Figure 1) which aren't shown so you'll have to scroll the palette window down to see them. Place the mouse cursor in the palette window and press the Page Down key until you see the PENH4 icon. (There's also a PENH3 icon but don't use it.) The names are below the icons. Click the LMB on the PENH4 icon and you should see a Place Parts prompt bar displayed. Move the cursor out over the schematic and you should see a ghost image of the part that follows the cursor. Notice that the cursor snaps to the nearest grid location identified by a white dot. You can turn grid snap off but leave it on for best results. Place the part somewhere by clicking the LMB. The part will remain white and the Selection count will be shown as Sel: 1. This means that the part has been *selected*.

All mentor graphic applications work on a *select an object and do something to it* paradigm. This usually works just fine but occasionally results in unexpected behavior when you have something selected that you didn't know about. Keep your eye on the select count and occasionally hit the F2 key to unselect everything. Try that now to see what happens to the pMOS fet. You should see the part turn blue with small purple diamonds identifying the part's *terminals*. The terminals are the only place on the part where you can make a connection and correspond to pins or leads on a real part. Carefully place the cursor on one of the terminals and click the LMB. Notice that the terminal turns white and the Sel count increments by one. If the whole part turns white, hit F2 and start over. Select each terminal in turn and notice the select count increment. Finally, place the cursor to the upper left of the part, hold down the LMB and drag a rubber band box out to enclose the entire part then release the LMB. Notice that this selects everything and the selection count goes up to 5. Hit F2 to unselect everything.

Place another pMOS fet by making a copy of the first one. First, select M1. Make sure the select count is 1 then press the Ctrl-F3 function key. Notice that the function key template at the bottom of the screen identifies this as the Copy key. Hold down the Ctrl-F3 key while moving the new copy to the right of the first. Release the F3 key and notice that the part remains selected but the first part has been unselected. Hit the backspace key to delete the selected part and repeat the copy operation a couple of times to get comfortable with it. You can move a selected part with the Ctrl-F2 key. Try that a few times. Try selecting both parts and moving them.

While we're at it, this is a good time to introduce *strokes*. Strokes are a handy shortcut for many common operations. They involve holding down the MMB and drawing a simple shape with the mouse much like using a pen. It takes a little hand-eye coordination to get right but is worth the effort. There are at least a half dozen strokes that are very useful and there are others you may find helpful as well. Carefully hold down the MMB and draw a short diagonal line in a north-east direction then release the MMB. This is the zoom out stroke and you should see the schematic image shrink. Zoom in is a short diagonal stroke in the southwest direction. View all is a short diagonal in the north west direction. There is a view area stroke but the F8 key may be easier to use to zoom in on a particular area of the screen. You can also use the scroll bars on the sides of the schematic window to pan the image around. A short horizontal stroke from left to right serves the same function as the Enter key or clicking on an OK button while a short horizontal stroke from right to left serves as the cancel button or Esc key. Finally, a stroke in the shape of the hook part of a question mark brings up quick help on strokes with a picture of the various defined strokes.

Now use the techniques you've just learned to place a copy of two NENH4's, two PORTIN's, a PRTOUT, a VCC, a GROUND, and a CAP in roughly the positions shown in Figure 1. The exact position isn't critical, just leave plenty of room for wiring.

Wiring Parts

After you have placed some parts you can wire them up. We call this *drawing nets*. A *net* is a set of connected wires that circuit theory texts often call a *node* as in *node voltage*. Nets can only be used to join terminals on parts. To draw a net, press the F3 function key that will bring up the Add Wi prompt bar. Move the

cursor to the source terminal (the bottom one) of M101 and click the LMB. Next, drag the rubber band line down to the drain terminal (the top one) of M102 and double click the LMB. This completes the net between these two terminals. Notice that the net remains selected. Hit F2 to unselect the net. Notice also that the prompt bar stays up. You can continue adding nets in this fashion as long as the prompt bar is active. Hit ESC to terminate the prompt bar after you are finished drawing nets.

The autoroute feature of da is sometimes handy. If you hit ESC to cancel the prompt bar, hit F3 again to bring back the Add Wi prompt. Add a diagonal wire from one of the INPORTs to the gate terminal of M1. Do this by simply clicking on the INPORT terminal, dragging the rubber band wire to the gate terminal of M1 and double clicking LMB. Now hold down the RMB to bring up the popup menu and select the Net>Route menu item. You should see the selected net change from a diagonal to a manhattan route (all vertical and horizontal runs). It may not be very pretty however. Da frequently puts the vertical segments in a funny spot. Unselect everything and then select just the vertical segment of the net you just drew. If you have difficulty doing this try using the area select by holding down the LMB and dragging the rubber band box over the item you want to select. After you have the vertical segment of a net selected, move it by holding down the Ctrl-F2 key while moving the mouse. Release the F2 key when you have the segment where you want it. You might want to experiment with this feature awhile to get used to it. Try selecting a vertex instead of a segment and see how they move. Make sure that only those objects you want to move are selected or you'll have a mess on your hands!

Finish wiring the rest of the circuit like that shown in Figure 1. Notice that when you click the LMB you add another vertex to the wire. You terminate a wire by double clicking the LMB. A net that ends at a terminal or another net is connected to it but one that just crosses another net is not connected. Connected nets are indicated as such by a small connection dot or box. Occasionally you may see a *not dot* that looks like a circle with a slash through it over two nets that are close to one another. This frequently happens which you use copy to add a net instead of the usual Add Wi command. If that happens and you really want to make a connection in place of the not dot, hit Sh-F6 to connect all.

At this point in time, it might be well to save your work before going on. It's a good idea to save your file periodically to avoid loss of valuable work. To save your file simply select the File>Save Sheet item from the main menu. Da will probably complain that the sheet has not been checked but just ignore that for now. After saving your file, close it by selecting the schematic's window menu Close item. This should bring you back to the original da session screen with a large empty gray client area. Click on the Open Sheet button on the palette as before to bring up the Open Sheet dialog. This time, click on the Navigator button to bring up a navigator window. This window has basically the same functionality as dmgr. You should see your nand2 component identified as such by a small icon that looks like a file folder with a C inside. Notice the navigator buttons that allow you to navigate through a directory hierarchy. Select the nand2 and OK the box. Notice that the component name has been filled in for you. OK the dialog box and you should see your nand2 schematic again. At this point in time you should be able to create and edit simple schematics.

Adding and Changing Properties

In order for our circuit to simulate properly we will need to add some new attributes (or properties) recognizable by SPICE. In this case, the properties we need have already been added to the library symbols. In some cases you may need to add an entirely new property. This will most likely be true when you create your own component symbols but we won't do that here. The three properties we need to change are: MODEL, INSTPAR, and INST. The MODEL property is currently set to a value of MODEL and that's what you see displayed near each fet symbol. The INSTPAR property is used to add values such as 1k or 10nH to parts and we'll use it to hold the Length and Width values for our fets. Finally, the INST property is used to name each instance such as M1, M2, M101, etc.

We'll start with M1. Unselect everything and then select just M1. Make sure the Sel count is 1. Pick the Report>Object menu item to bring up a printed list of the object's properties as they currently exist. Browse through the list with the page up/down keys then close the report. Now type the command: cha pr va asim model cmosp. This is the command to CHAnge PProperty VALUE of the ASIM_MODEL property to cmosp. You can do this with the popup menu item Instance>Properties>Modify but the command is simpler. Notice that all you need to do is to start typing with the mouse cursor in the schematic window and the prompt bar pops up automatically. If this happens to you accidentally just hit the ESC key to cancel the prompt bar. Notice also that after you type the change property command the ASIM_MODEL value goes away and is replaced by the new model name. This model name is the name that appears in a .MODEL spice record which we'll get to later. Next, enter change property value commands to change the INST property to M1 and INSTPAR property to L=2u W=50u. You'll have to put this last value inside double quotes (") since there is a space between 2u and W. Make sure you include the units (u) or you will end up with a 50 meter wide transistor! Change the properties of the other components in a similar fashion according to Table I below:

Table I Nand2 instance properties

Instance	ASIM_MODEL	INSTPAR	INST
M1	cmosp	L=2u W=50u	M1
M2	cmosp	L=2u W=50u	M2
M101	cmosn	L=2u W=10u	M101
M102	cmosn	L=2u W=20u	M102
CL		1p	CL

Notice that the ASIM_MODEL property for the load cap is blank.

Next we'll change the names on the input and output ports. These names are also properties attached to the terminals of the ports. Properties can be attached to either symbols or their terminals. In this case the property is a net name so logically it belongs to the terminal. In the previous case the properties applied to the part itself so were attached to the instances.

To change the name of the ports simply place the cursor over the port name and hit the Sh-F7 key. This brings up the CHA PR VA prompt bar with the property name NET already filled in and the value NET selected. If NET isn't selected, click the LMB somewhere in the New Value field. You can also use the Tab key to move around inside the prompt bar. With the value NET selected, type in the new name (A1) and OK the bar. Repeat this for the B1 and Out ports. You can use this technique to change any other visible property text as well.

If you don't like the position of some text item you can move it with the F7 key. Place the cursor over the Out netname and press the F7 key. While holding the key down move the Out text so that it appears above the port.

It's usually a good idea to add comment text like titles and the like to your schematic. You can do this a number of ways but we'll use the schematic palette. Select the text button on the palette and notice how the palette buttons change. There are four such buttons at the top of the palette: session, add, text, and route. If you hit the session button you'll unselect the schematic window and return to the session palette. Simply click on the schematic window somewhere to return to the schematic palette. On the schematic_text palette, notice an Add Comment button. Click on that one to bring up the Add TExt prompt bar. Fill in some descriptive text (like CMOS nand2) and OK the bar. Since you haven't finished specifying the location, the bar stays up and the At Location item is selected. Move the cursor and your text moves along with it. Click the LMB to fix the position and the prompt bar will disappear. More than likely the text height won't look right. With the text selected, pick the popup menu item Property Text>Change Height>2.0 x pin spacing.

Now unselect the text by hitting F2. Try to select it again by clicking the LMB on it. What happens? If you said 'nothing' you're correct! This is because the *select filter* has been set to exclude comment text. Hit Ctrl-F to just see what the filter is set to but don't change anything. Notice that comment text isn't included and then hit ESC to cancel the filter dialog. We could have included comment text by clicking on the selector button next to comment text but that might mess things up later on. An easier way is to simply use the F1 (select area all) key. Place the cursor over the comment text and just hit F1 to select the text. You can also drag out a selection rectangle by holding the F1 key down and moving the mouse but it's usually sufficient to use F1 this way to select single items. Hit F2 to unselect everything.

Finishing Up

At this point in time your schematic should look something like that in Figure 1. Make sure everything is connected and your properties are correct according to Table I. Before you can use a schematic with the other tools it must be checked for errors. Pick the main menu item Check>Sheet. If all went well you should see a report that indicates there are no errors. Normally this won't be the case so let's create an error condition to see what happens.

Hit Ctrl-H which will bring up the symbol history dialog. This is a useful command that not only presents you with a list of the parts you've recently used but some common ones as well such as portin and prtout. Chose the CAP part from the history list and OK the box. The active symbol window changes to display the CAP part. Place an extra cap somewhere on the sheet so that it doesn't touch any other part. Hit F2 to unselect the cap. Now run Check>Sheet again to see what happens. This time you should see a warning that you have one or more unconnected pins or *dangles*. Place the cursor over the instance name which will be something like I\$819 and click the LMB to select it. Pop the schematic window by clicking the MMB on it somewhere. The MMB is a safer button to use since it doesn't change the Sel count. Notice that the cap has been selected. This is called *cross selection* and is a common feature across all mentor tools. You can generally select an object in one window and have it selected in all windows. Now pick the View>Selected item from the main menu. This will zoom in on the selected items and, along with View>Centered, is a handy way to find the location of errors in the Check>Sheet report. Now hit Del to erase the extra cap, select Check>Sheet again to verify that there are no errors, and select File>Save Sheet to save your file.

Identifying a Modelfile

Go back to the main analog palette by picking Libraries>MGC Analog Libraries from the main menu. Select Analog M/S Utilities from the palette. From Utilities select Create Viewpoint which will bring up a dialog box. Enter the filename of a model file in the Netlistfile field. We'll use the n14p.spice.level3.model for this example. This is simply the name of an ascii file with spice .MODEL records in it. Actually it can be any legal spice code but most of the time it's just the .MODEL's of your transistors. We'll use a model supplied by MOSIS from a previous fabrication run. Click on the Navigator bar next to the Netlistfile field and navigate to \$UMRLIB. This is a directory that contains a number of files, including the n14p model file. Select it and ok the dialog box. The rest of the fields should be left alone and should contain: default, sim_ba, and AccuSim II. OK the dialog box and be patient while a thing called a design viewpoint is created. Keep in mind that you only need to do this once per design. After you've created a design viewpoint it generally doesn't change. You can edit the schematic to your heart's content and never bother with the DV again.

Apparently most Mentor users are born with the knowledge of what a design viewpoint is because the documentation is fairly vague in that respect. Suffice to say that one is required for simulation and for most other so called *downstream* applications such as printed circuit board layout. A design viewpoint is yet another design object with yet another editor called the DVE or design viewpoint editor that is used to indirectly modify a design such as a schematic. As far as we're concerned there are two important uses for the design viewpoint. One use is to make certain properties visible to downstream applications while hiding others that aren't required. The other use allows you to attach so called *back annotation* objects to your design. You'll see this when we get to simulation but for now back annotation is simply a way to change your schematic without actually changing it. Putting it another way it's a way for downstream applications to feed back information into the schematic. We'll use it to change the L and W properties of our mosfets while in the simulator and that's the most common way we'll use back annotations. Another common use for BA's in industry is to assign actual package identifiers such as U1-2, pin numbers, and so forth to IC's. This assignment is not usually done until board design time, long after the schematic design is finished. Most simulator applications create the design viewpoint for you but since we wanted to attach a modelfile we used this method.

After a few moments the design viewpoint is created, a message indicating Viewpoint created successfully is displayed, and you can leave da. After you're done, save your sheet. Then, before you leave da you may want to make a hard copy of your design. Simply pick File>Print Sheet from the main menu and OK the box. The UMR-ECE printer name is called (appropriately enough) *mentor*. Close the session window by selecting the Close item from the session window menu. After a moment or two the da window will disappear. There may be an extra terminal window on your screen with a title something like des_arch/bin/da. This is the transcript window mentioned earlier and contains a list of all the transactions that took place during your session. Look for things like `$$close_session(void)`. These are *ample functions* which are the low level commands issued by menu picks and so forth. You can do things like type them in directly (who would want to?) or collect several together into an ample script or *dofile* which is something like a macro. You can ultimately customize everything and even make your own menus if you'd like but that's way beyond the scope of this tutorial!

Running Accusim

In this section we will use an analog simulator to analyze our nand2 component. Accusim II is Mentor's version of SPICE which you are most likely already familiar with. We will use Accusim to a) run a DC sweep to verify the nand's transfer function, b) run a transient analysis to measure the gate's rise and fall time performance, and c) adjust the width of one of the mosfets to achieve a desired performance.

Before entering Accusim, get into the mentor shell by typing "`mgc csh`". Invoke Accusim on your circuit by typing the command `accusim nand2`. After a few moments you'll see a window somewhat like that in Figure 2. The specific sizes and positions of the client windows will be different since those can be moved around and resized as you see fit. After the Accusim window comes up, maximize it by typing the command `max win` to make it fill the screen.

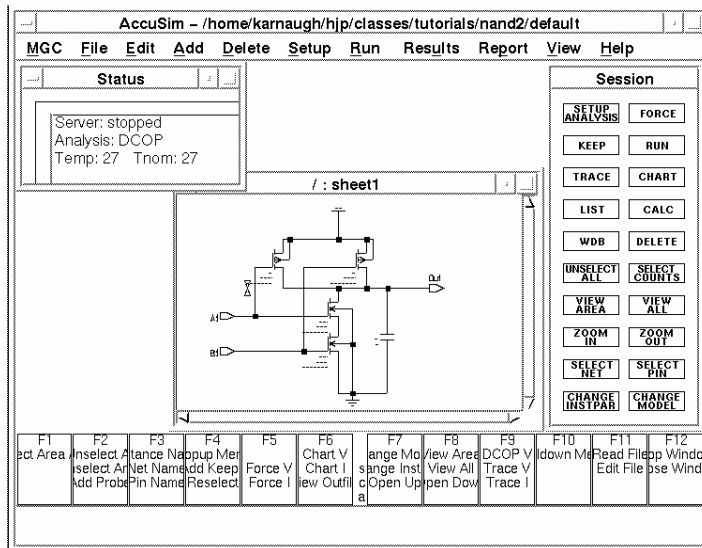


Figure 3 AccuSim Window

Running a DC Sweep

First of all let's run a DC sweep analysis. This is a common simulation step to verify the dc transfer function of a device. First select Add>Keeps, select All and OK the resulting dialog box. This step tells the simulator to save all of the node voltages and branch currents in the simulation. In a large simulation you would want to select particular states to save in order to conserve time and memory. We'll also need to force the B1 to 5v in order to produce an interesting output. Select the B1 input in the schematic window and click on the Add Force button in the session palette. This will bring up the Force dialog box partially filled in with the selected input signal. Complete the dialog by filling in Mode DC and Magnitude of 5 and OK the box. Repeat this step for the input A1. Hit F2 to unselect all and click on the A1 port to select

the A1 input and then click on Setup Analysis on the palette window. Fill out the dialog as follows: DC Sweep, From =0, To =5, Increment value=0.1 Make sure the source net was filled in with /A1 for you and OK the box. Now hit the run button on the palette window. The status window will briefly display Simulation: RUNNING and then revert back to NOT RUNNING.

In order to examine your results, select both A1 and Out in your schematic view then hit the Chart button. In a moment you'll see a chart window appear with two graphs, one of A1 and the other of Out. The chart of A1 is simply the dc sweep. The graph of Out should be a basic inverter characteristic something like the one shown in Figure 3.2 of Smith or Figure 2.17 of Weste and Eshraghian 2nd Ed.

Select the menu item Delete>Forces, select All to delete all forces, then OK the dialog to prepare for the next analysis.

Running a Transient Analysis

Next we'll run a transient analysis and measure the delay characteristics of our nand2 cell. We'll assume that we want a two input nand gate that can drive a 1pF load capacitor with a rise and fall time of less than 3.5nS and a propagation delay of no more than 3nS. Pick setup>analysis from the main menu. This will bring up the same dialog box that you saw earlier. The menu pick does the same thing as the palette selection. Select the transient button and OK the box leaving the default values for the time step and stop values set at 1N and 100N (N=nSec).

Select signal A1 in the schematic window (make sure all other nets are unselected) and press the Add Force button on the palette. Force A1 with a pulse with initial value of 0, a pulsed value of 5, delay of 40N, rise time 1N, pulse width 40N, fall time of 1N, and a period of 80N. What do you think this signal will look like? Select signal B1 and force it with a pulse from 0 to 5v with a delay of 20N, risetime of 1N, pulse width of 40N, falltime of 1N, and period of 80N. Notice that this pulse overlaps the A1 pulse so that both nand inputs will be high at the same time.

After creating the two forces it's a good idea to check things out so select Report>Forces from the main menu and examine the report window. When you are satisfied that all is well, close the report. To run the transient analysis, simply hit the run button on the palette.

In order to examine the transient analysis results, chart A1, B1, and Out just like you did earlier. Select the three signals on the schematic window and click on the Chart button on the palette window. You should see three charts this time: the two input forces and the resulting Out signal. Select the output waveform in the chart window. Click on the Measure button in the palette and click on the Fall Time button. The fall time will be graphically shown on the waveform and reported as something like 4.678N. You should visually verify from the chart of V(/Out) that this is the correct fall time. You'll probably want to use the F8 key to zoom in on the falling edge of Out. Also notice that as you bring the mouse cursor near a waveform a dynamic message is displayed with the current x and y values of the curve (time and voltage in this case). Repeat this step to verify that the rise time is about 2.34 nS. In order to measure the propagation delay you'll have to be a bit more subtle than the previous two measurements.

Hit F2 to unselect everything. When measuring a delay we need to specify which input edge causes which output edge. Select the A1 waveform first followed by the Out waveform, then click Delay in the time group of measurements. Delay is measured from the first edge Accusim encounters on the input to the first edge of the output. This isn't good enough to find TpLH B to Out so you'll need to define the active search area. You can do this by zooming in with the F8 key (or use strokes) and hit delay or by adding two cursors to define the start and stop time. Add a cursor by using the Chart>Add>Add Cursor popup menu item. The delay from a1 to out is 2.54nS and the delay from b1 to out is 1.16 nS. You should verify this. Notice too that these two values are only half of the propagation delays: tpHL(a-Out) and tpLH(b-Out). What will you need to do to measure the other two values?

Adjusting Properties for desired performance

You may have noticed that the fall time didn't quite meet the specification. You may have also noticed that the two pulldown fets had different widths and may have deduced that is the problem. Let's now change the width of the M101 fet.

Select M101 in the schematic window. Make sure everything else is unselected. Pick Edit>Property>Change which will bring up the Change Properties dialog. Select the instpar property and OK the box. This will bring up another change property dialog for the instpar property on /M101. The '/' in /M101, incidentally, means that this is instance M101 in the *root sheet*. In a large design you may have to navigate through several levels of instances to get to the one you want. Change the value of the width to 20u which should be about right. Be sure to put the units 'u' on the 20 or you'll get a 20 meter transistor! After you OK the dialog box notice that the new property value is displayed in Red. This indicates that the new value has come from a *back annotation* object. You haven't actually changed the schematic. Back annotations can be made *persistent* or permanent and design architect as well as other tools can view a schematic through a viewpoint to see the back annotations that may have been applied. You can even merge the BA's into the schematic to make them permanent.

Use what you learned above to rerun the transient analysis and verify that the new fall time is within specification.

Leaving Accusim

Before leaving Accusim, try selecting an axis on one of the charts and examine the popup menu commands Chart>Edit>Change Title. Use help to examine the Charting User's and Reference manual (in the Simview bookcase). Many of the charting commands are common to other analysis tools and you can even use the charting commands in a standalone environment outside of any simulator to analyze any data you might have. Skim through the Analog Simulators User's Manual (in the AccuSim bookcase) or a much longer and in-depth tutorial is available in the Getting Started with Accusim II manual.

You exit AccuSim just like any other Mentor application. Simply pick the Close item from the window menu button in the upper left corner of the session window. When you are finished with this, logoff. Under no circumstances should you leave a workstation logged on.

Going Further

Congratulations! If you've finished this tutorial you are well on your way to becoming proficient with one of the leading state of the art, industrial strength ECAD tools. Several suggestions for additional reading have already been made. You should also watch for other tutorials in this series. If you have suggestions for corrections or improvements to this tutorial or other applications you'd like to see please contact the author.

If you have time, you might want to reinforce what you've just learned by using da and Accusim to do some investigation on your own. You might try examining the effect of reduced supply voltage on timing. Vcc=3.3v is becoming a standard for battery powered applications. Try plotting the charging current that results when a transistor switches.