

# Cadence Tutorial 1: Schematic Entry Basic Gates Drawing

## EE577b Fall98

Refer [http://www-scf.usc.edu/~ee577b/tutorial/cadence\\_tut.html](http://www-scf.usc.edu/~ee577b/tutorial/cadence_tut.html)

*jsmoon@ISI.EDU*

### 1. Tool Setup

1. Make sure you have "source ~ee577/setup.csh" entry in your .cshrc file or .login file.

2. Create cds directory at your home by  
`%mkdir cds`

3. Copy cadence\_ee577b.tar to your cds directory.  
`%cp ~ee577/cadence_ee577b.tar ./cds`

4. Extract tar file at your cds directory.  
`%cd cds`  
`%tar xvf cadence_ee577b.tar`

5. Make sure the following files are located at your cds directory.

```
1 drwxr-xr-x  4 ee577      512 Nov 15 14:33 ./
4 drwxr-xr-x 32 ee577      4096 Nov 15 14:33 ../
9 -rw-r--r--  1 ee577      8220 Oct 25 11:17 .cdsinit
1 -rw-r--r--  1 ee577       402 Nov 15 14:32 cds.lib
10 -rw-r--r--  1 ee577     9612 Nov 12 00:30 schBindKeys.il
```

6. Make sure you can run cadence tool by typing  
`%which icds`  
`/usr/usc/cadence/97a/tools/dfII/bin/icds`

*jsmoon@ISI.EDU*

## 2. Create Library

1. Invoke "icds" program at cds directory. (icds is front-end cadence tools integration.)

```
%icds &
```



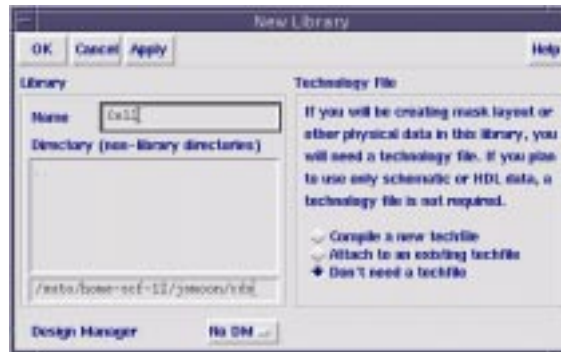
2. Create Cell Library.

```
%File->New->Library
```

In [New Library] window,

Name : Cell

Technology File : Don't need a techfile



*jsmoon@ISI.EDU*

## 3. Create Inverter Schematic

1. Open Schematic Window.

```
%File->New->Cellview
```

In [Create New File] window,

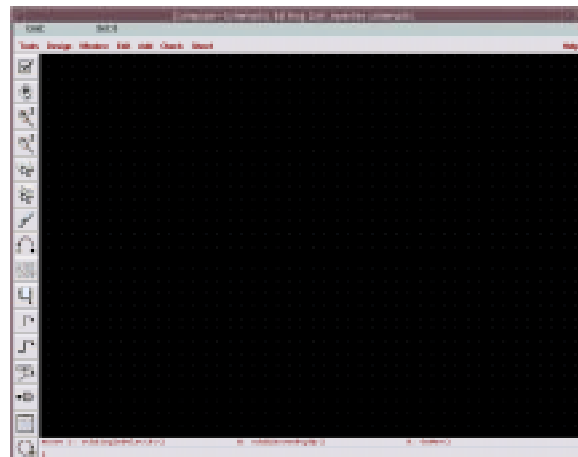
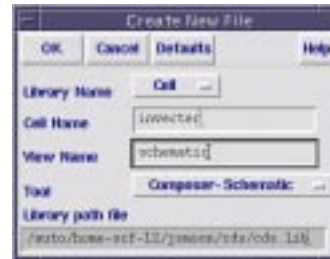
Library Name : Cell

Cell Name : inverter

View Name : schematic

Tool : Composer-Schematic

Click OK



*jsmoon@ISI.EDU*

### 3. Create Inverter Schematic (continued..)

#### 2. Place nmos transistor

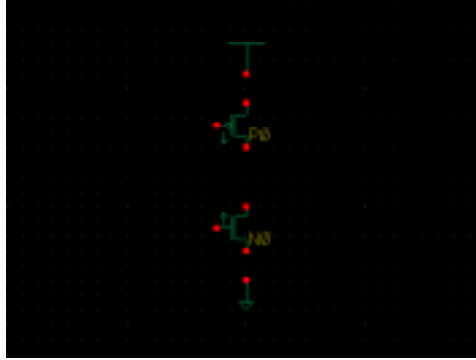
sch:Add->Component (or "i")  
 Add Component : click Browse  
 In [Library Browser] window,  
 Library : Lib  
 Cell : nmos  
 View : symbol



When you move mouse into schematic window, nmos symbol will follow your pointer. Click "mouse L" to place nmos. Type "Esc" to exit adding component action.

#### 3. Place pmos, vdd, and gnd.

-Repeat above procedure for pmos, vdd and gnd placement.  
 -All components can be found at the same library (Lib).  
 -While Library Brower is open, clicking pmos, vdd and gnd in cell field will bring instances. (You don't have to type "Esc" and "i" each time).



*jsmoon@ISI.EDU*

### 3. Create Inverter Schematic (continued..)

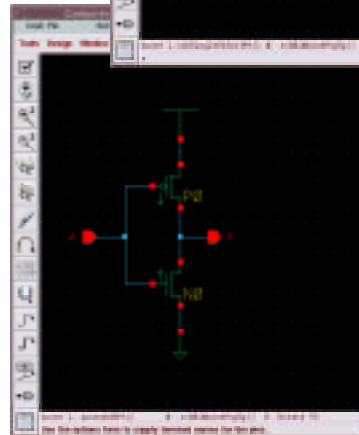
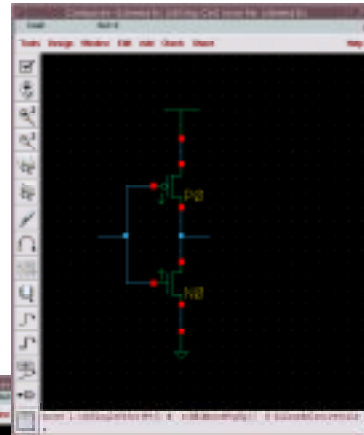
#### 4. Connect each component using wires.

Place mouse pointer on one of the node you want to connect.  
 sch:Add->Wire(narrow) (or "w")  
 Click "mouse L", drag to other node to connect, and click "mouse L" to finish.

-To make wire open node at one end (like input and output of inverter), double-click "mouse L".

#### 5. Place pins.

sch:Add->Pin (or "p")  
 In [Add Pin] window,  
 Pin Names : A X  
 Direction : input  
 Move mouse to place A pin at input of inverter, then click "mouse L".  
 In [Add Pin] window, change direction to output.  
 Place X pin at output of inverter



*jsmoon@ISI.EDU*

### 3. Create Inverter Schematic (continued..)

#### 6. Add transistor parameters. (W/L)

Click pmos  
 sch:Edit->Properties->Objects (or "q")  
 In [Edit Object Properties] window, click Add  
 In [Add Property] window,  
 Names : W  
 Type : NLPEXPR  
 Value : [@pw:%:8]



Repeat for L parameter.

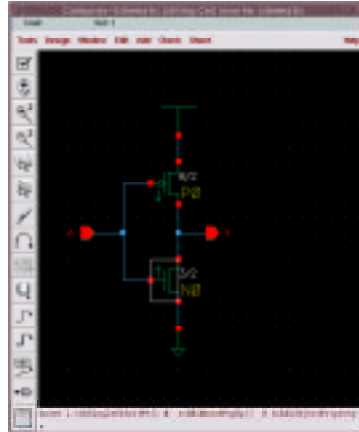
Names : L  
 Type : NLPEXPR  
 Value : [@pl:%:2]

Repeat for nmos with W=[@nw:%:3] and L=[@nl:%:2]

-@pw and @pl stand for parameterized attributes. If we assign pw=10 and pl=4 for inverter at schematic which includes inverter symbol, default value (W/L = 8/2 for pmos and W/L=3/2 for nmos) will be overruled.

#### 7. Check and Save

sch:File->Check and Save

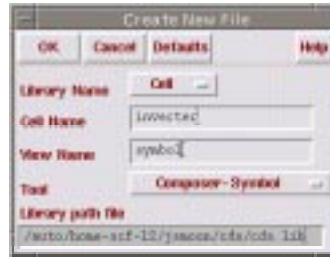


*jsmoon@ISI.EDU*

### 4. Create Inverter Symbol

#### 1. Open Symbol Window.

%File->New->Cellview  
 In [Create New File] window,  
 Library Name : Cell  
 Cell Name : inverter  
 View Name : symbol  
 Tool : Composer-Symbol  
 Click OK

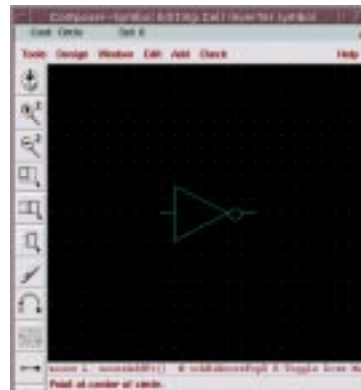
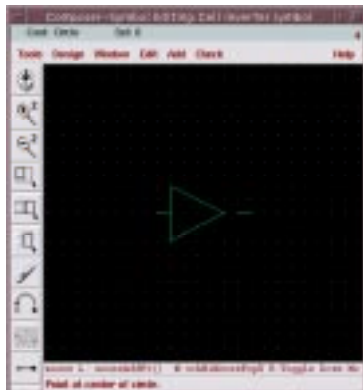


#### 2. Draw outline for inverter symbol.

Click line icon on left toolbox of symbol window and draw outline of inverter.

#### 3. Finish outline of inverter by adding circle.

sym:Add->Shape->Circle



*jsmoon@ISI.EDU*

#### 4. Create Inverter Symbol (continued..)

##### 4. Place pins.

sym:Add->Pin (or "p")

In [Add Pin] window,  
Pin Names : A X  
Direction : input  
Type : square

Move mouse to place A pin at input of inverter, then click "mouse L".

In [Add Pin] window, change direction to output.

Place X pin at output of inverter.

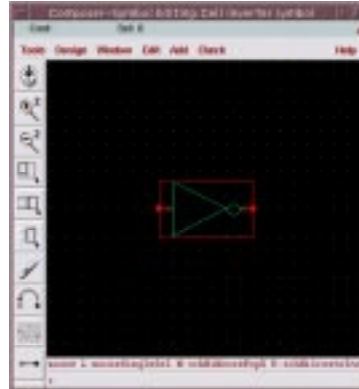
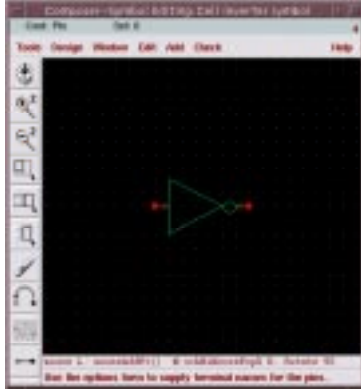


##### 5. Add selection box.

sym:Add->Selection Box

In [Add Pin] window, click Automatic

- Selection box is a boundary to select this symbol in a schematic which include the current symbol.



*jsmoon@ISI.EDU*

#### 4. Create Inverter Symbol (continued..)

##### 6. Add labels.

sym:Add->Label

In [Add Pin] window,

Label : [@instanceName]

Place instanceName label

Repeat for parameter attributes.

In [Add Pin] window,

Label : [@pw:%:8][@pl:%:/2]

Place.

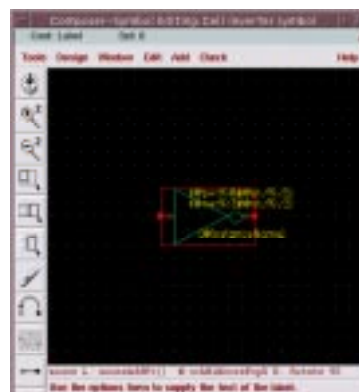
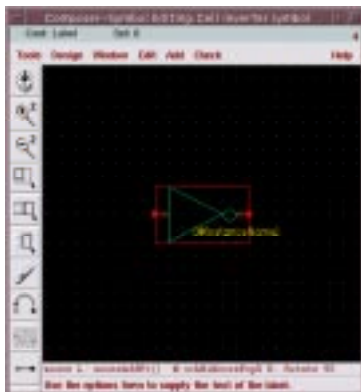
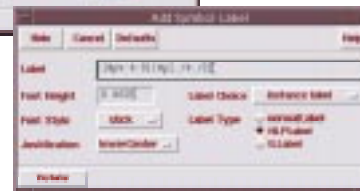
In [Add Pin] window,

Label : [@nw:%:3][@nl:%:/2]

Place.

##### 7. Check and Save

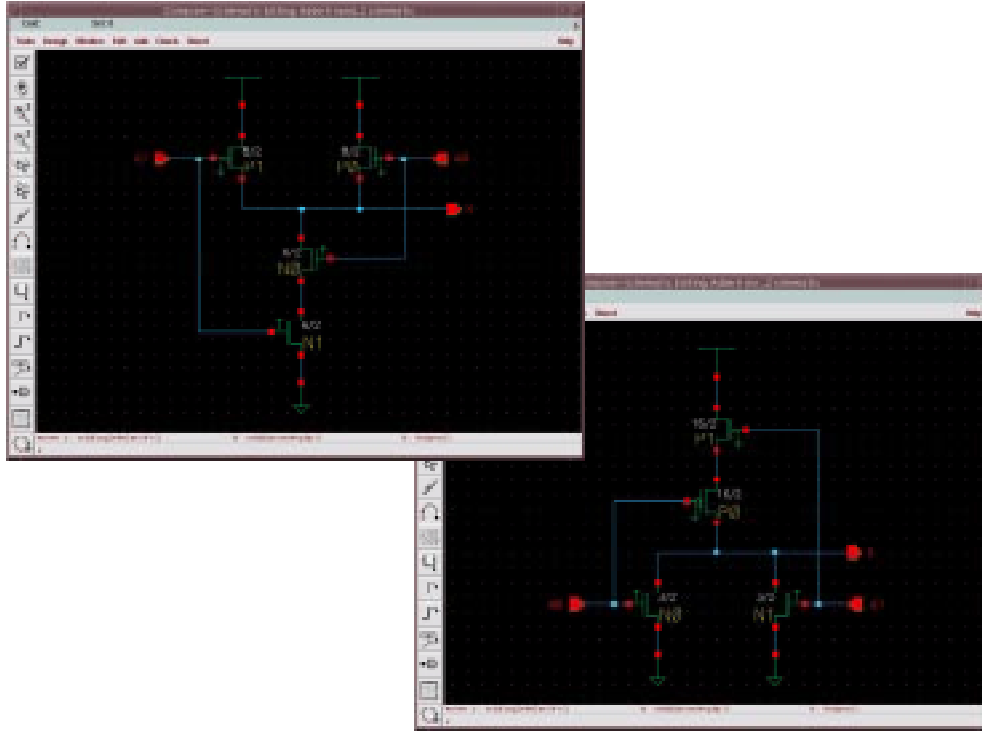
sym:File->Check and Save



*jsmoon@ISI.EDU*

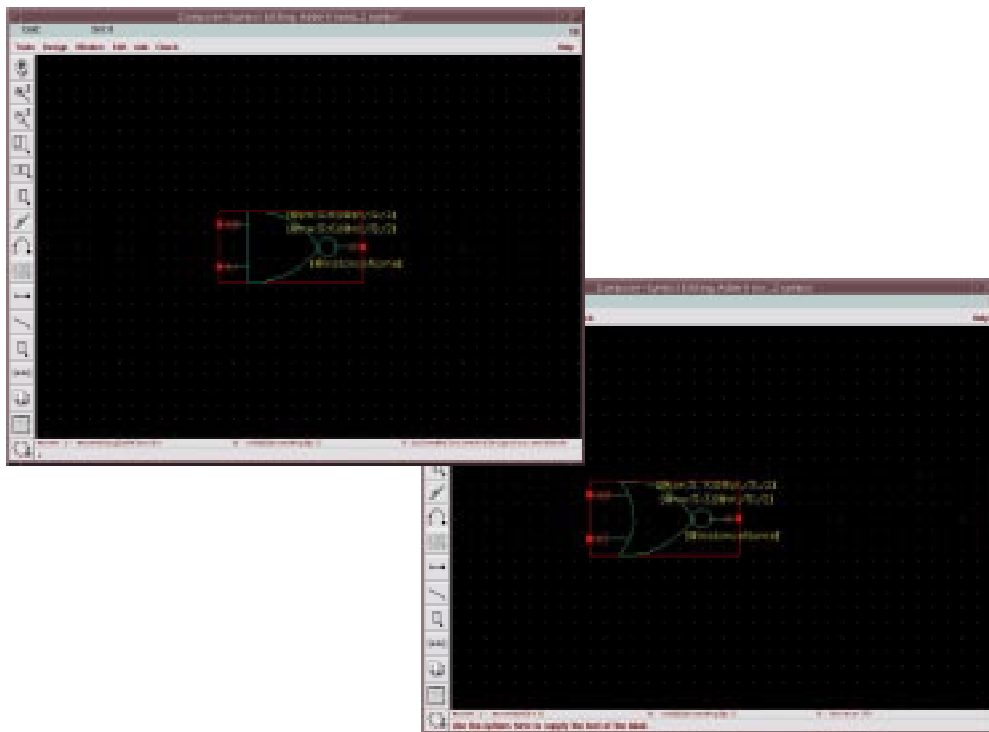
### 5. NAND2, NOR2 Schematic & Symbol

Complete nand\_2, nor\_2 schematic and symbol with the same steps above.



*jsmoon@ISI.EDU*

### 5. NAND2, NOR2 Schematic & Symbol (Continued..)



*jsmoon@ISI.EDU*