

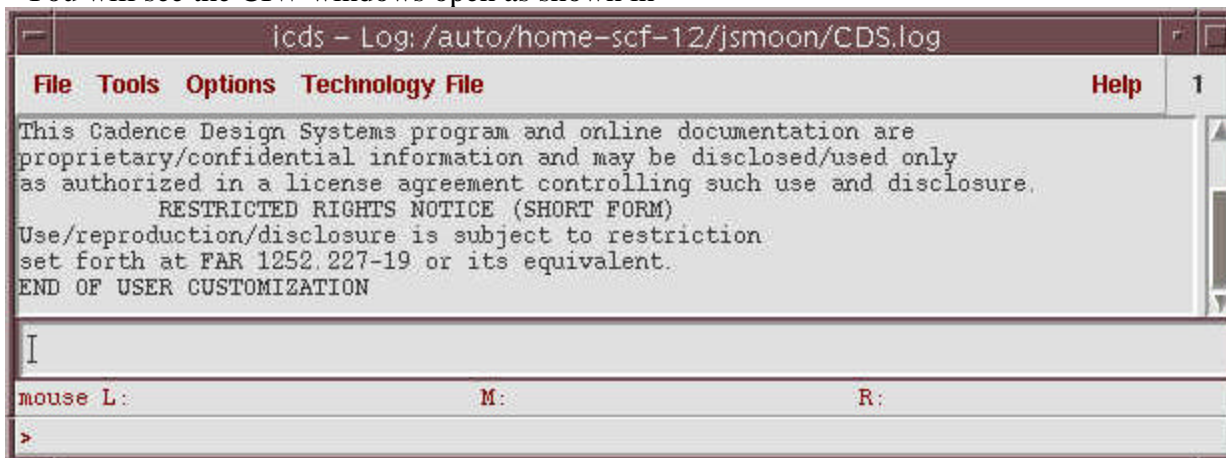
# Cadence Tutorial : 8-bit Ripple Carry Adder Schematic & Symbol

bug or comment to [tugsinav@usc.edu](mailto:tugsinav@usc.edu)

- Library Create
  1. Invoke icfb program.

`%icfb &`

- You will see the CIW windows open as shown in



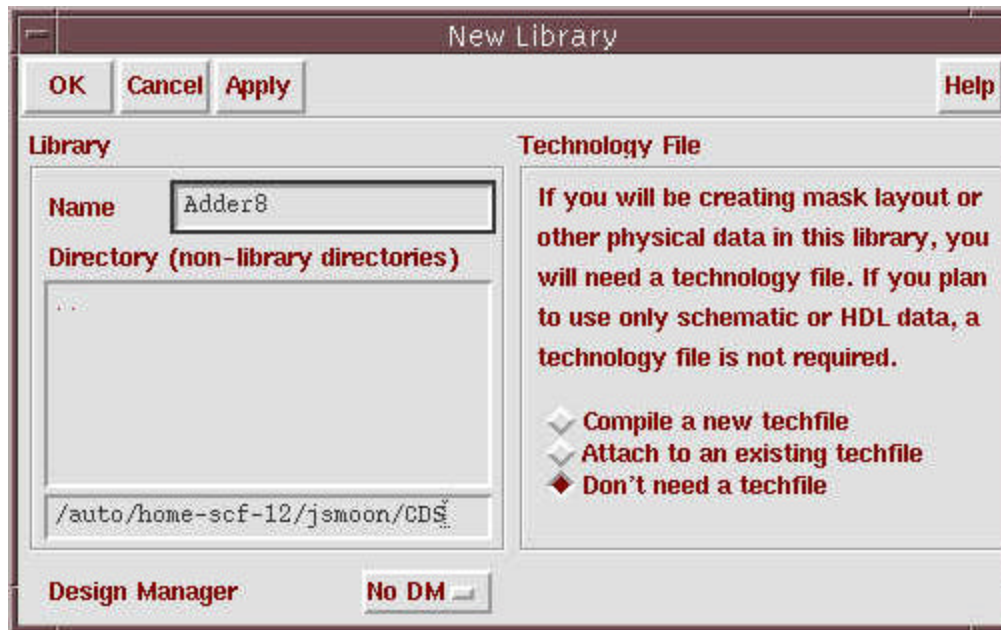
[Fig 1.](#)

2. Create adder8 library.

File->New->Library

In New Library window,

- Name : *Adder8*
- Technology File : *Don't need a techfile* (on the right window)



See  
[Fig 2.](#)

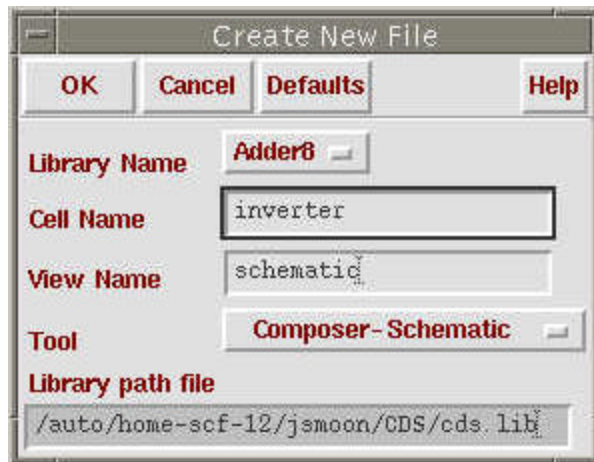
- Inverter Schematic Create

1. **Open schematic window.**

File->New->Cellview

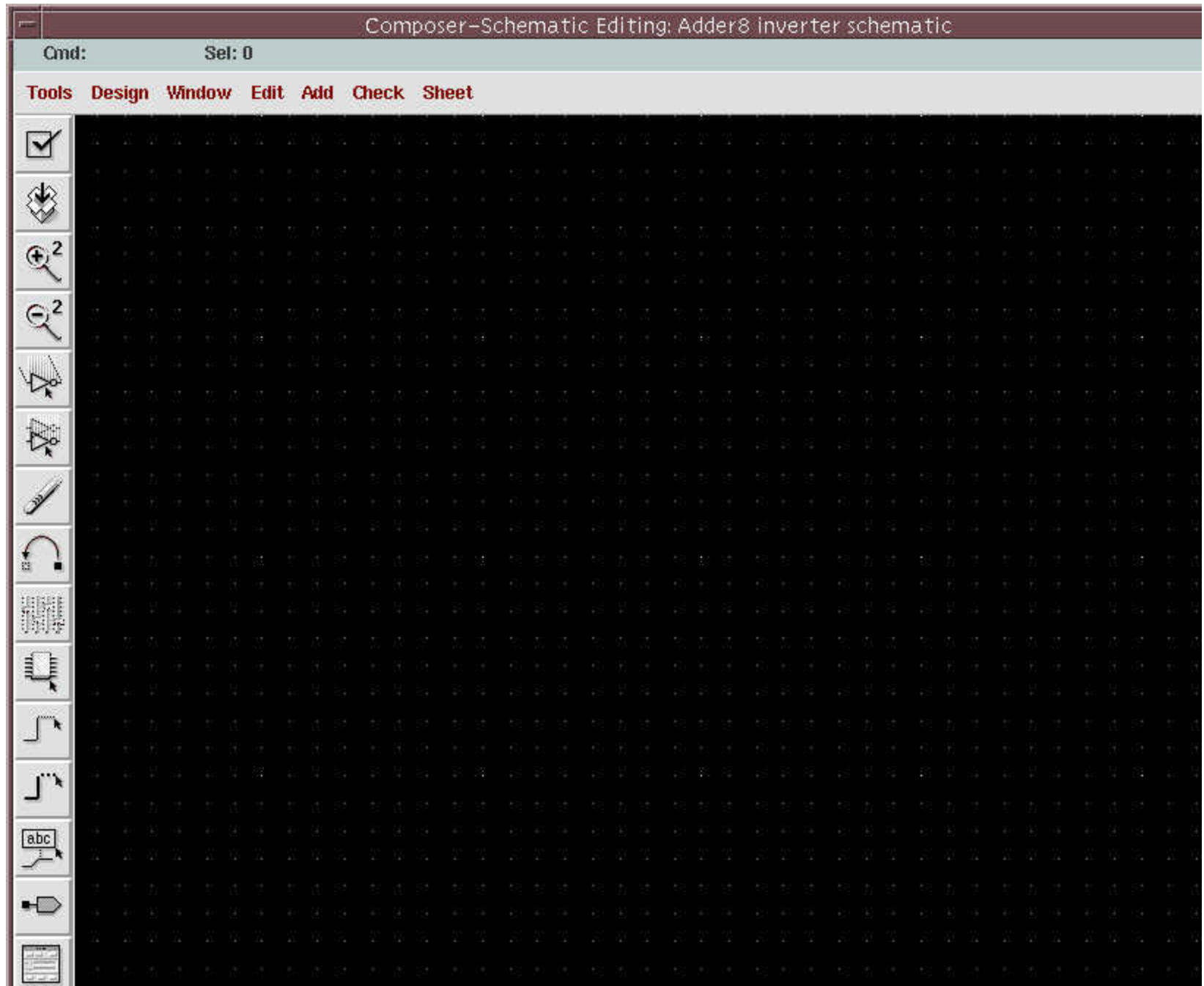
In Create New File window,

- *Library Name : Adder8*
- *Cell Name : inverter*
- *View Name : schematic*
- *Tool : Composer-Schematic*



See  
[Fig 3.](#)

- You will see schematic window open as shown in



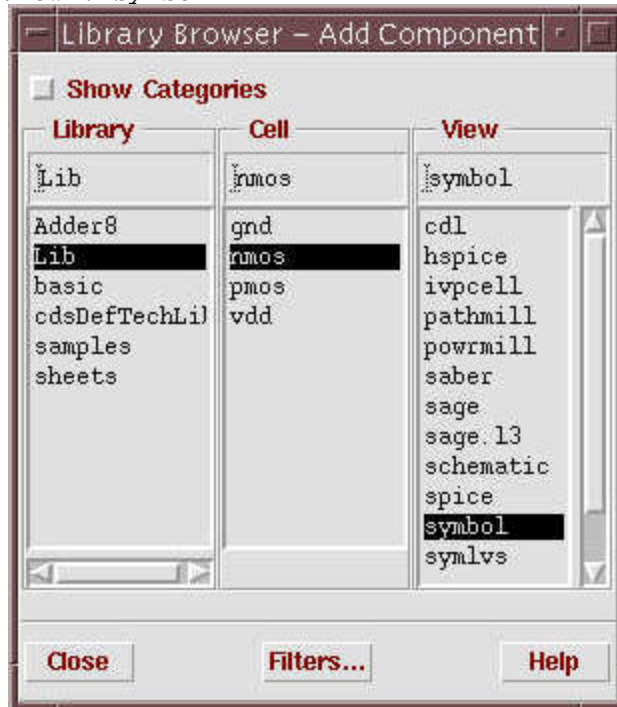
[Fig 4.](#)**2. Place nmos transistor.**

sch:Add->Component (or "i")

Add Component:click Browse

In Library Browser window,

- *Library : Lib*
- *Cell : nmos*
- *View : symbol*



See  
[Fig 5.](#)

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

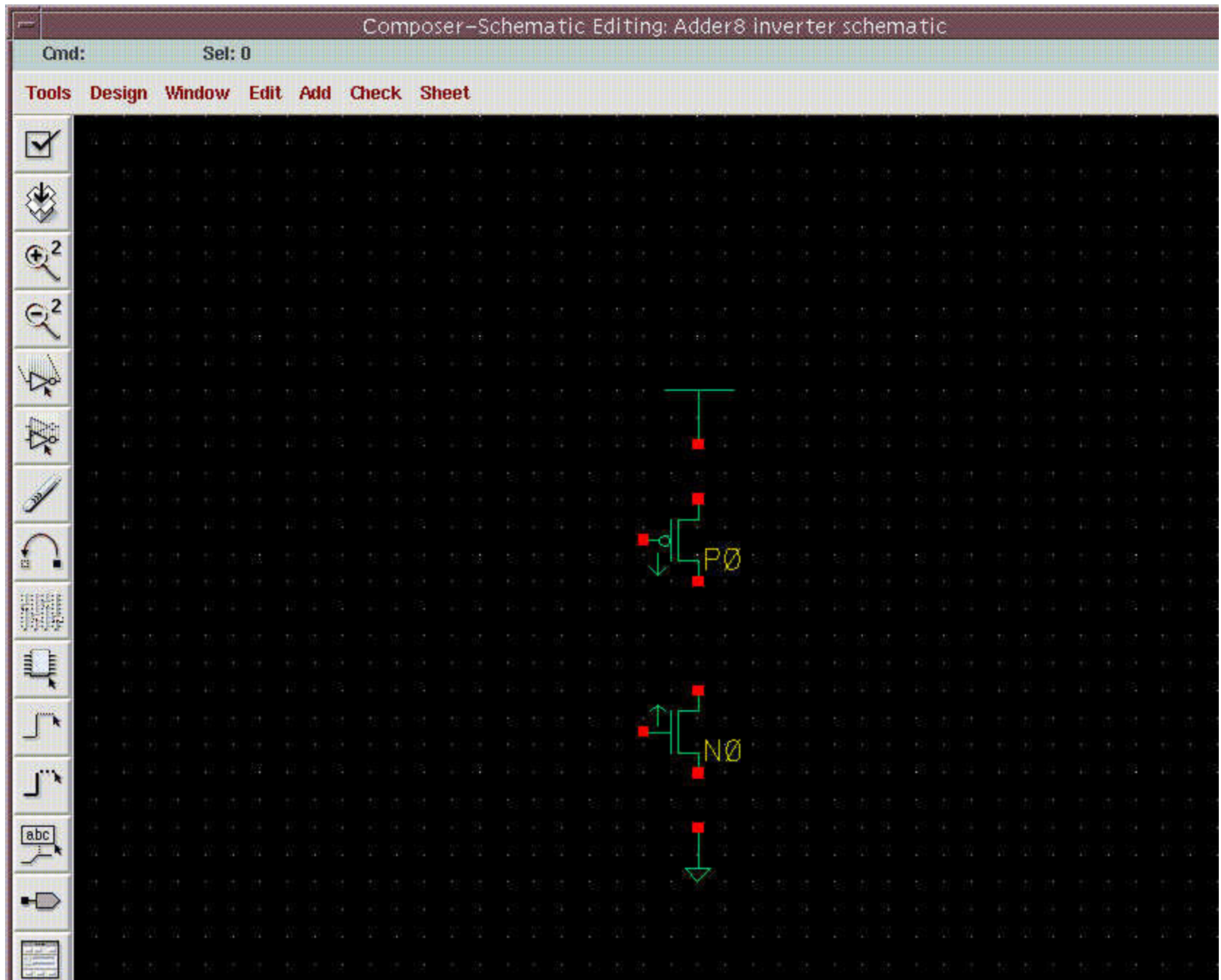
See



[Fig 6.](#)

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). See





[Fig 7.](#)

- While Library Browser is open, clicking pmos, vdd, and gnd in cell field will bring instances. (you don't have to type "Esc" each time.)

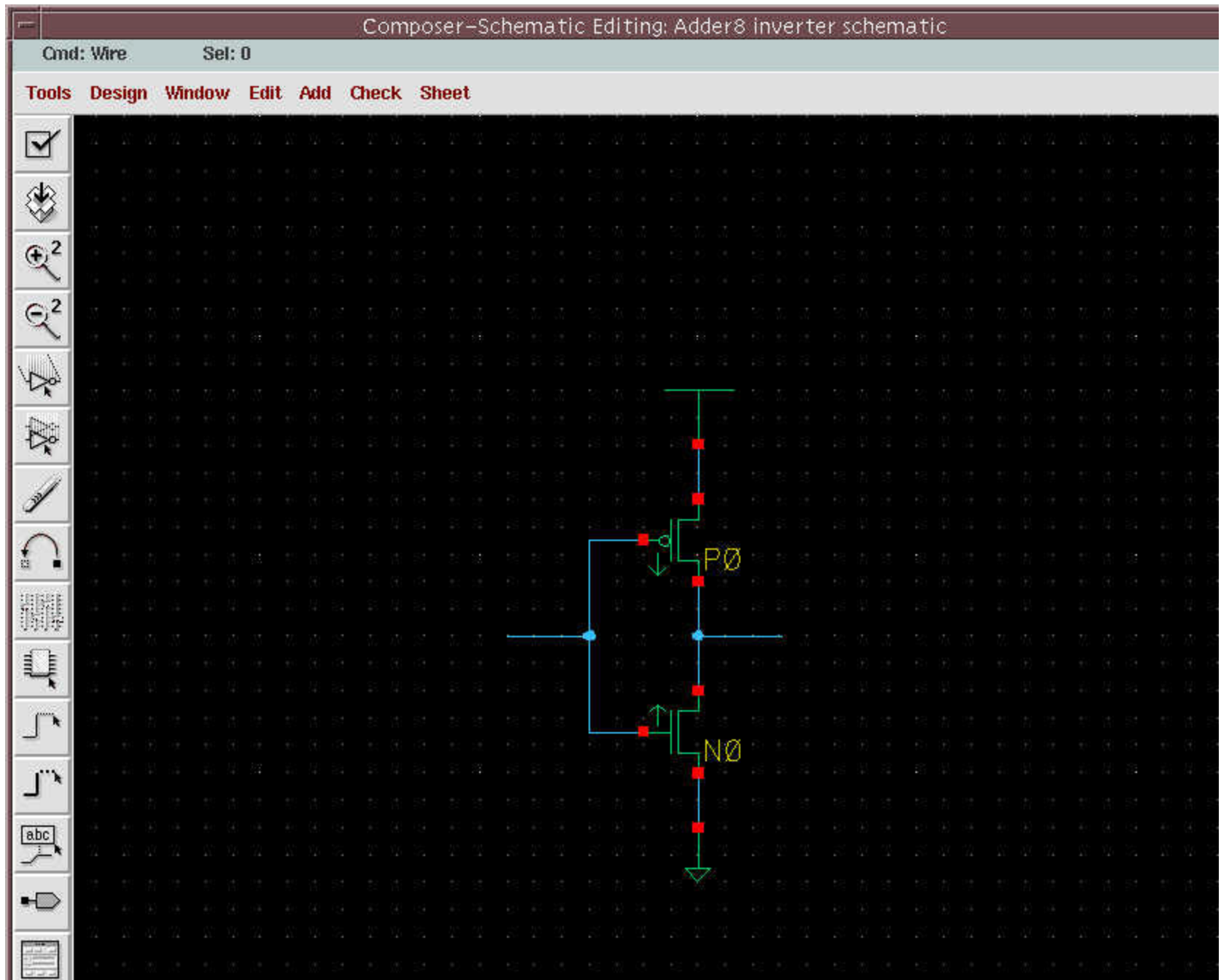
4. **Connect each component using wires.**

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.

- Repeat above procedure to finish wiring as shown in



[Fig 8.](#)

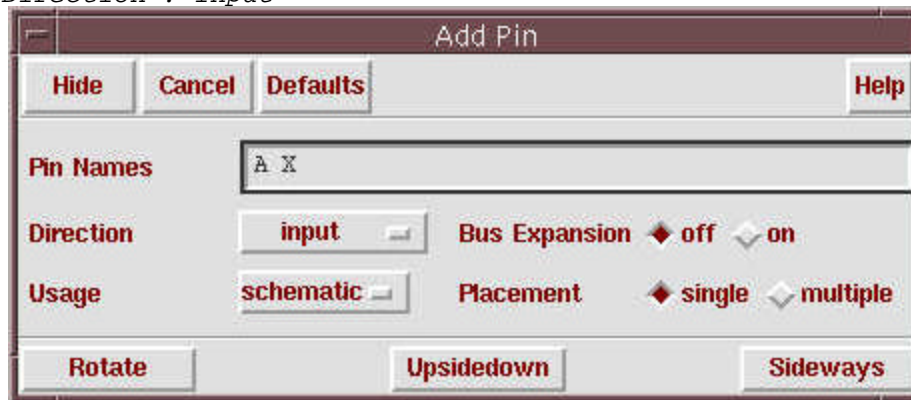
- To make wire open node at one end (like input and output of inverter), double-click "mouse L"

#### 5. Place pins.

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

In Add Pin window,

- *Pin Names* : A X
- *Direction* : input



See

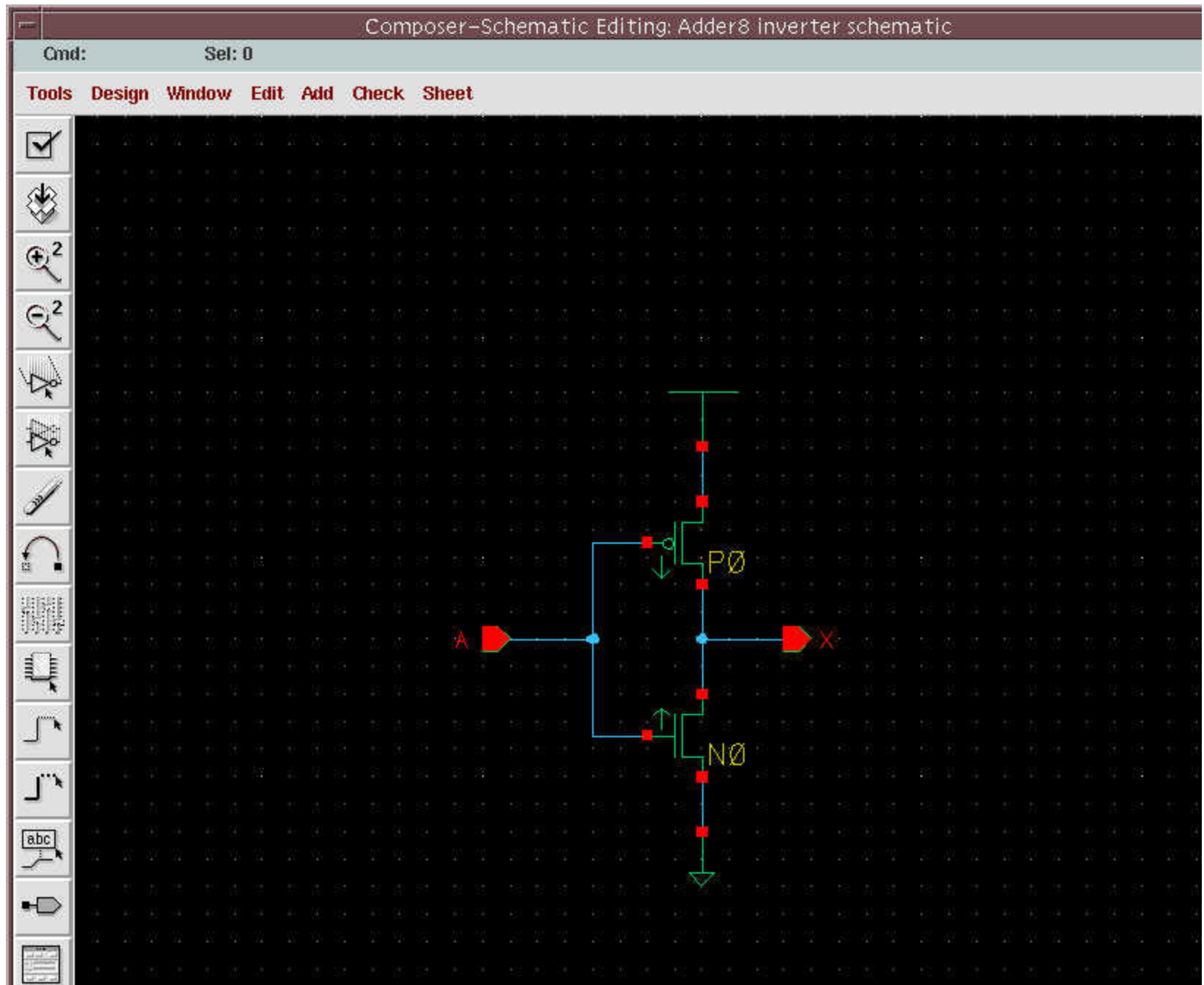
[Fig 9.](#)

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.

See



[Fig 10.](#)**6. Add pmos parameters. (W/L)**

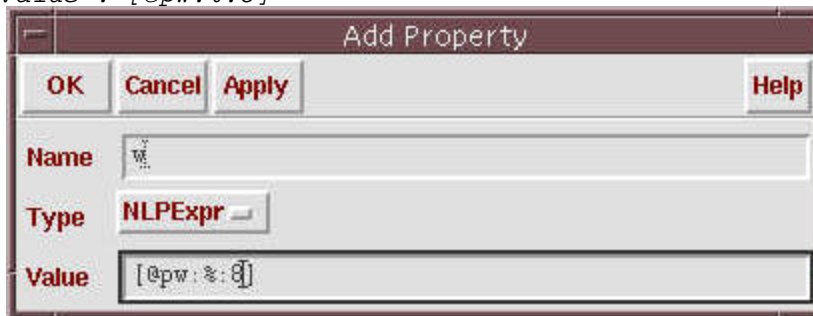
Click pmos

sch>Edit->Properties->Objects (or "q")

In Edit Object Properties window, click Add

In Add Property window,

- Name : w (small letter)
- Type : NLPEXpr
- Value : [*@pw:?:8*]



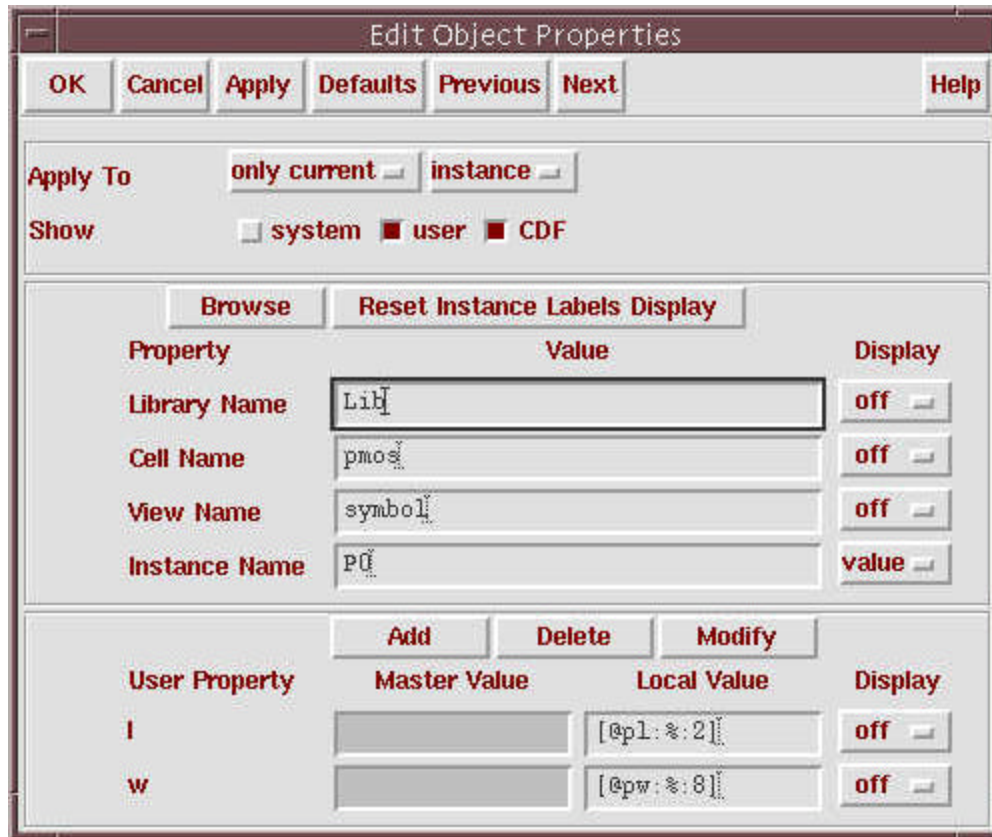
See

[Fig 11.](#)

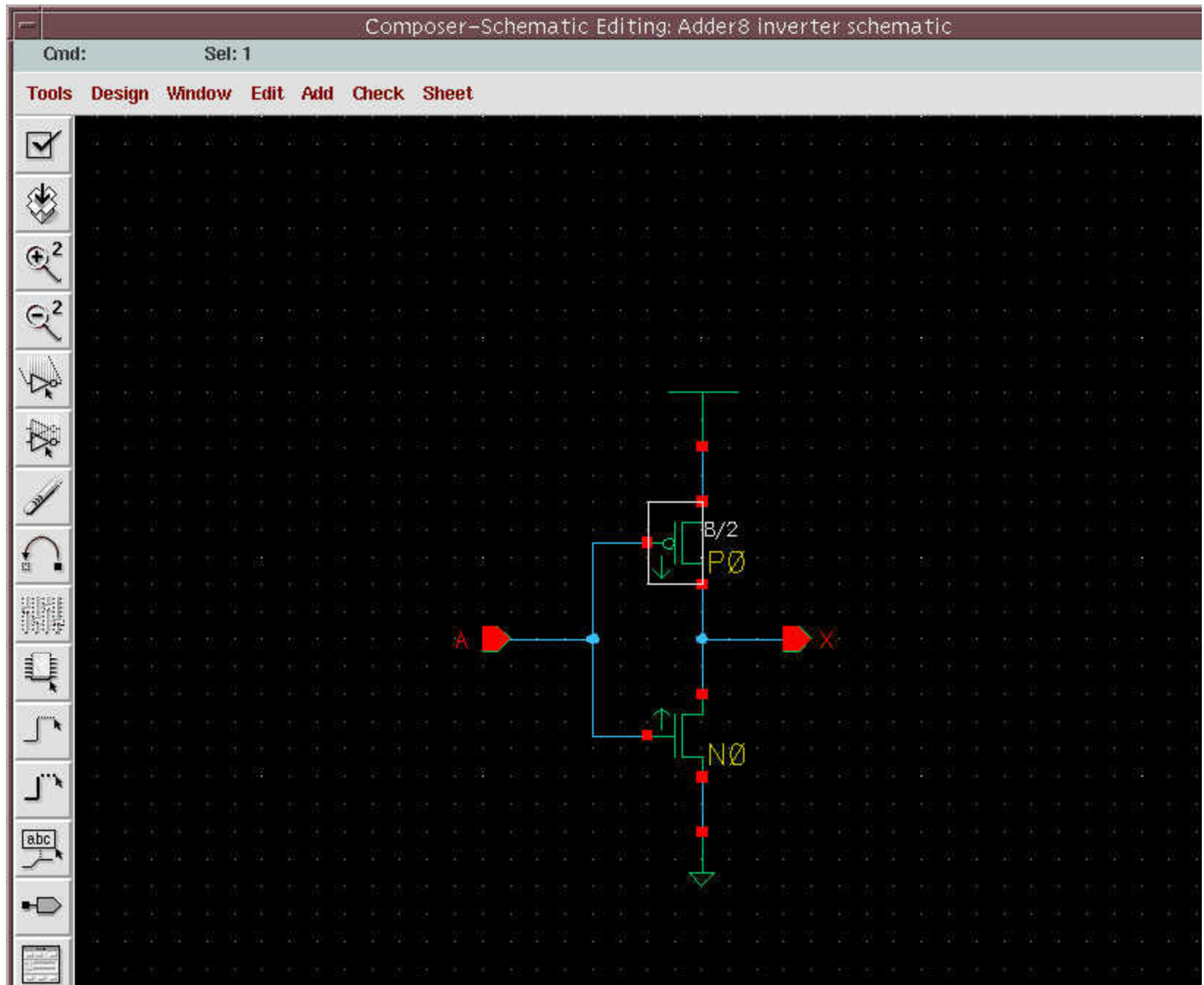
Repeat Add Property for L.

- Name : l (small letter)
- Type : NLPEXpr
- Value : [*@pl:?:2*]

- Your Edit Object Properties window will look like

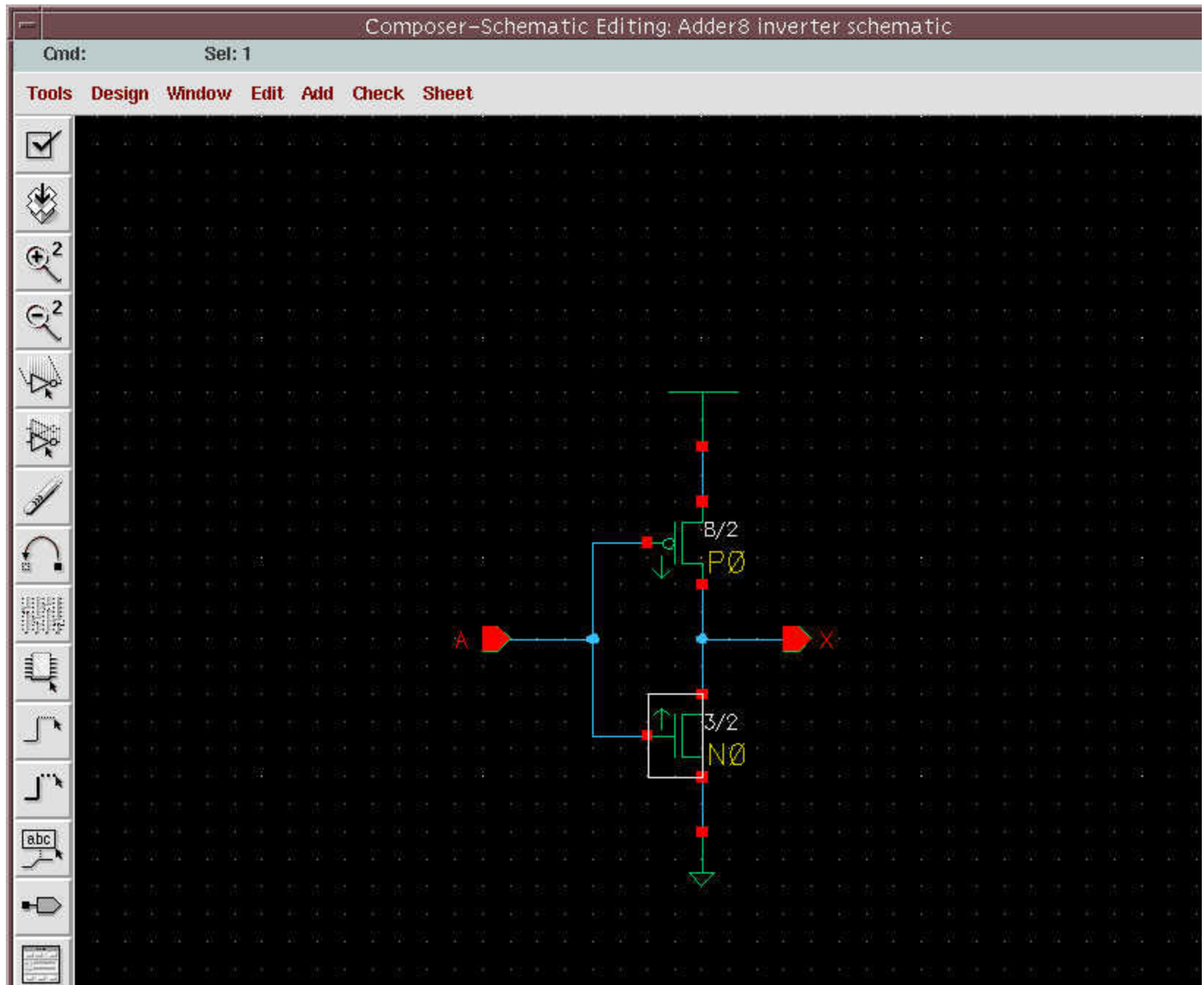


[Fig 12.](#)



- [Fig 13](#) shows the resulting schematic. - w, l of pmos have default values 8 and 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 values will be overuled. This is how we can achieve parameterized schematic.
- Repeat for nmos with w=3, l=2 as default values and @nw, @nl as parameters as shown in





[Fig 14.](#)

## 7. Check and Save

sch:File->Check and Save

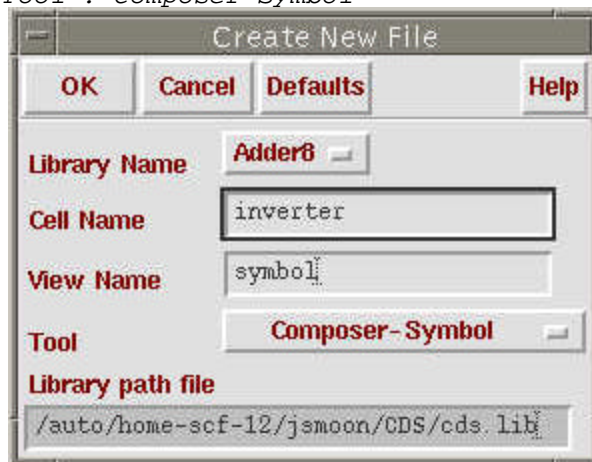
- Inverter Symbol Create

### 1. Open symbol window.

icds:File->New->Cellview

In Create New File window,

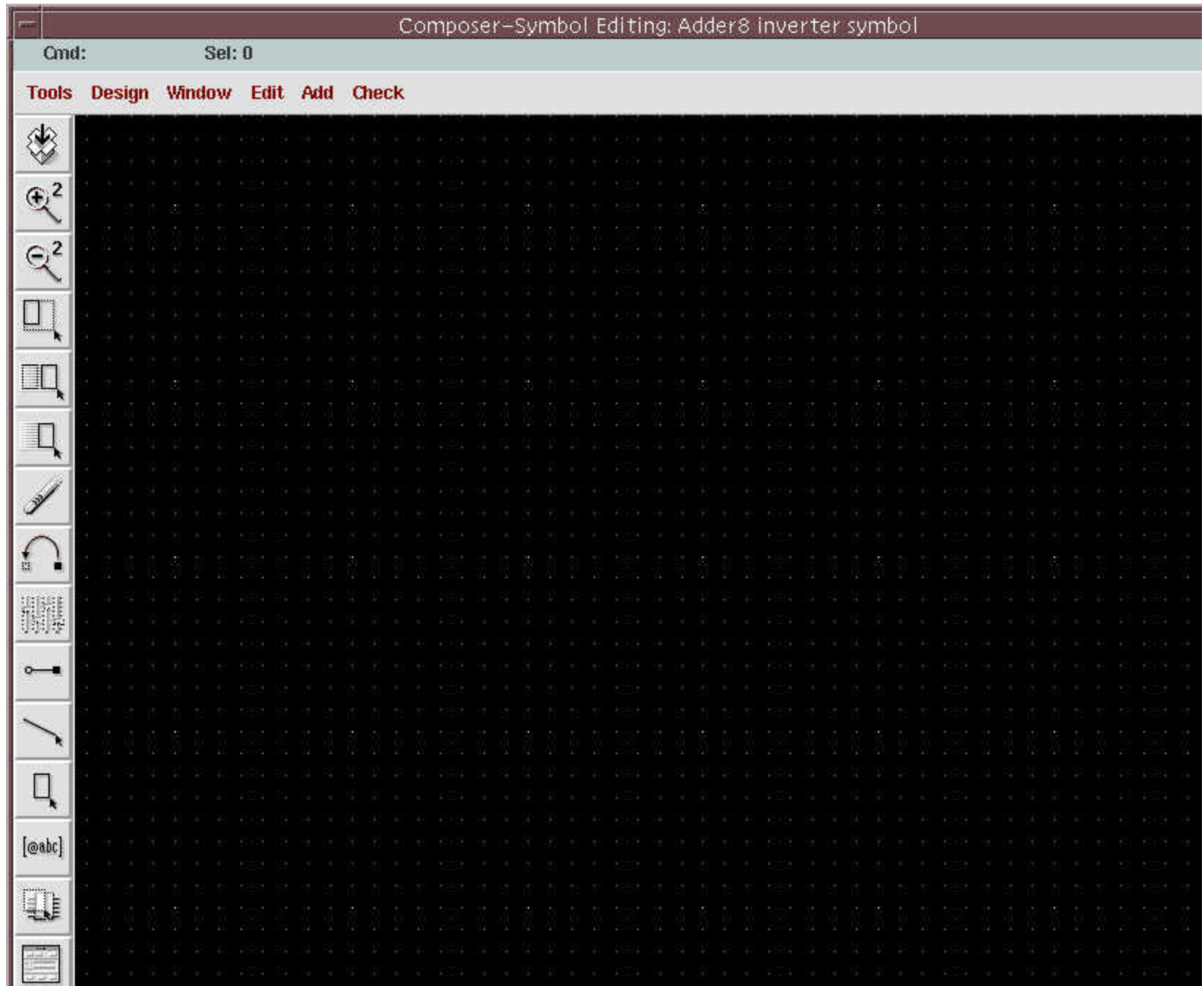
- *Library Name* : *Adder8*
- *Cell Name* : *inverter*
- *View Name* : *symbol*
- *Tool* : *Composer-Symbol*



See

[Fig 15.](#)

- You will see symbol window open as shown in

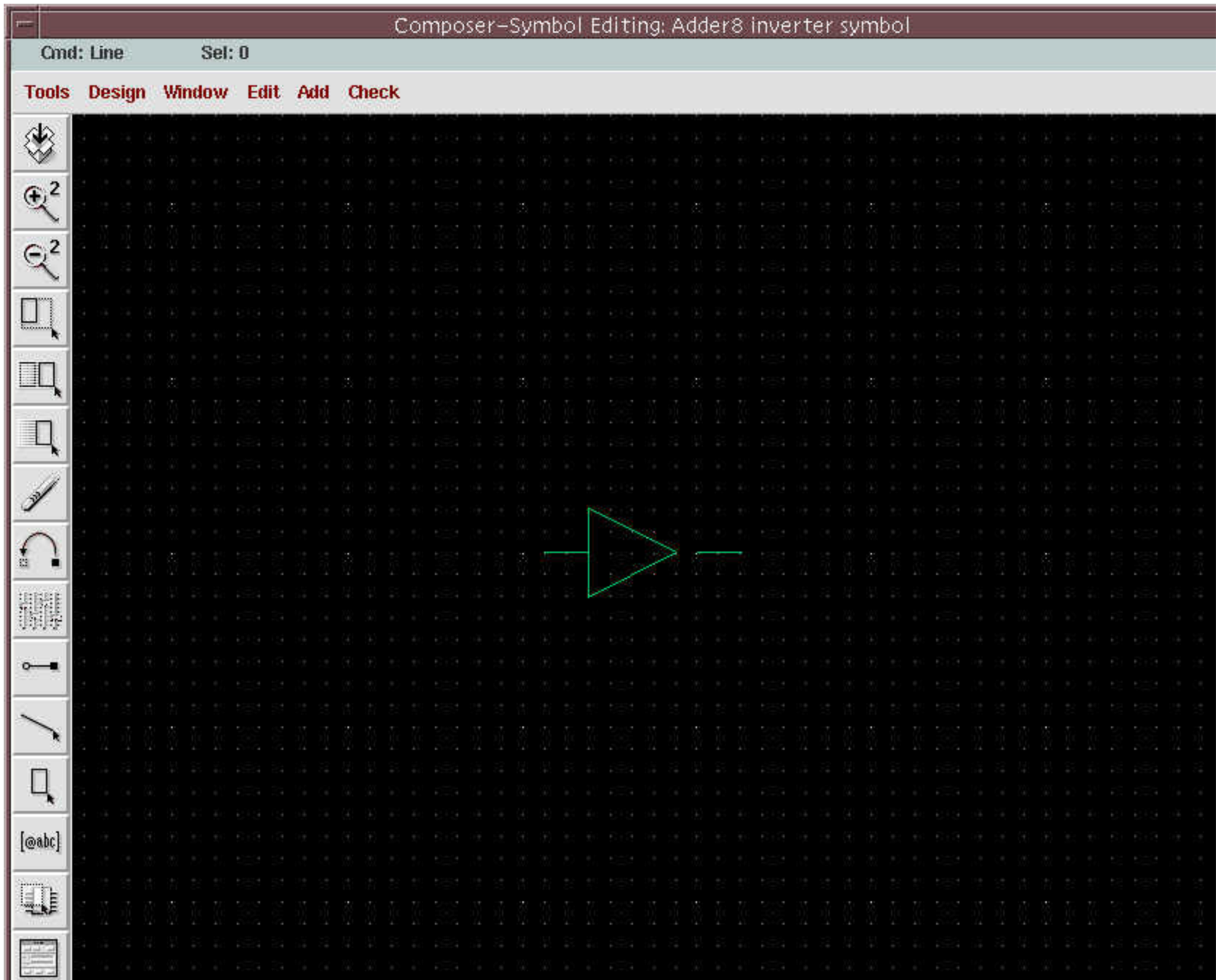


[Fig 16.](#)

2. **Draw outline for inverter symbol.**

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

See



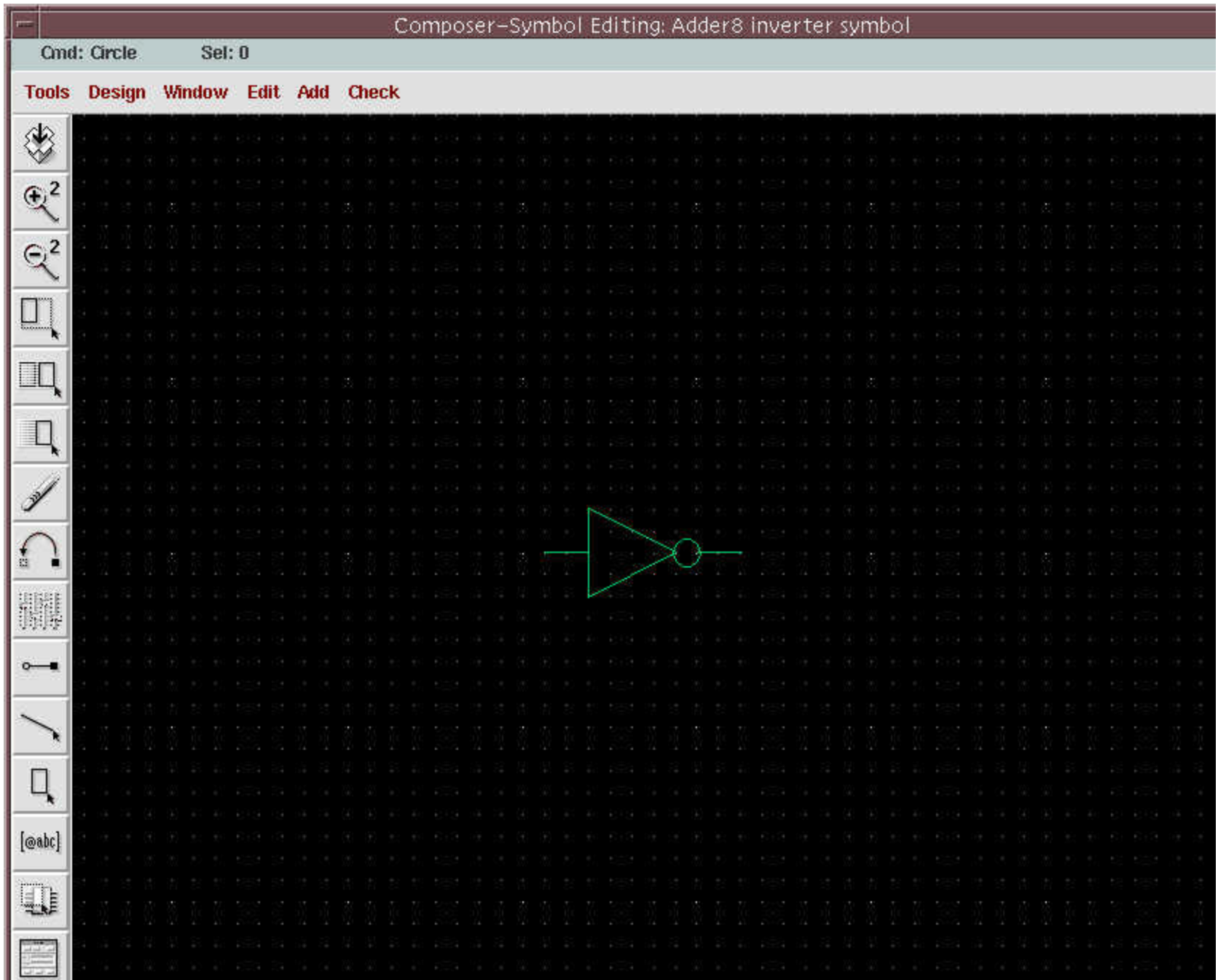
[Fig 17.](#)

3. **Finish outline by adding circle.**

sym:Add->Shape->Circle

Click "mouse L" on the center of circle, drag and click "mouse L" again.

See

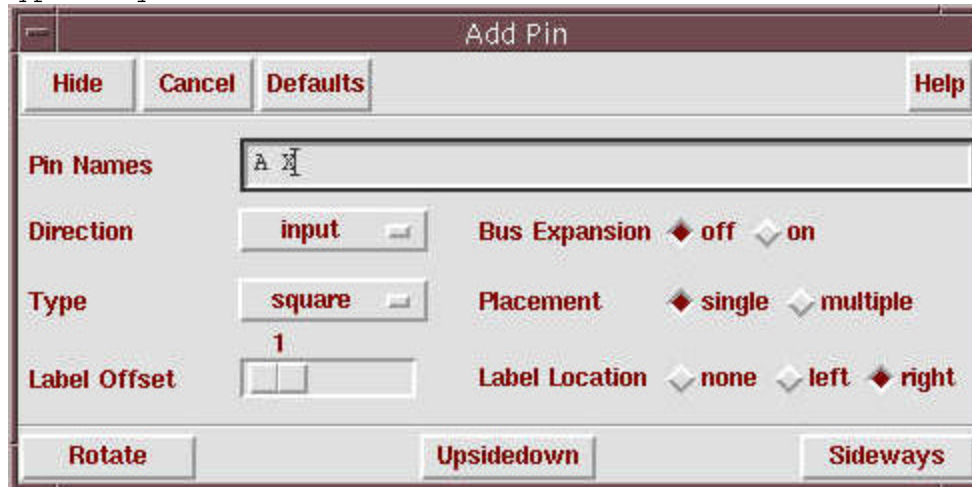


[Fig 18.](#)4. **Place pins.**

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

In Add Pin window,

- Pin Names : A X
- Direction : input
- Type : square



See  
[Fig 19.](#)

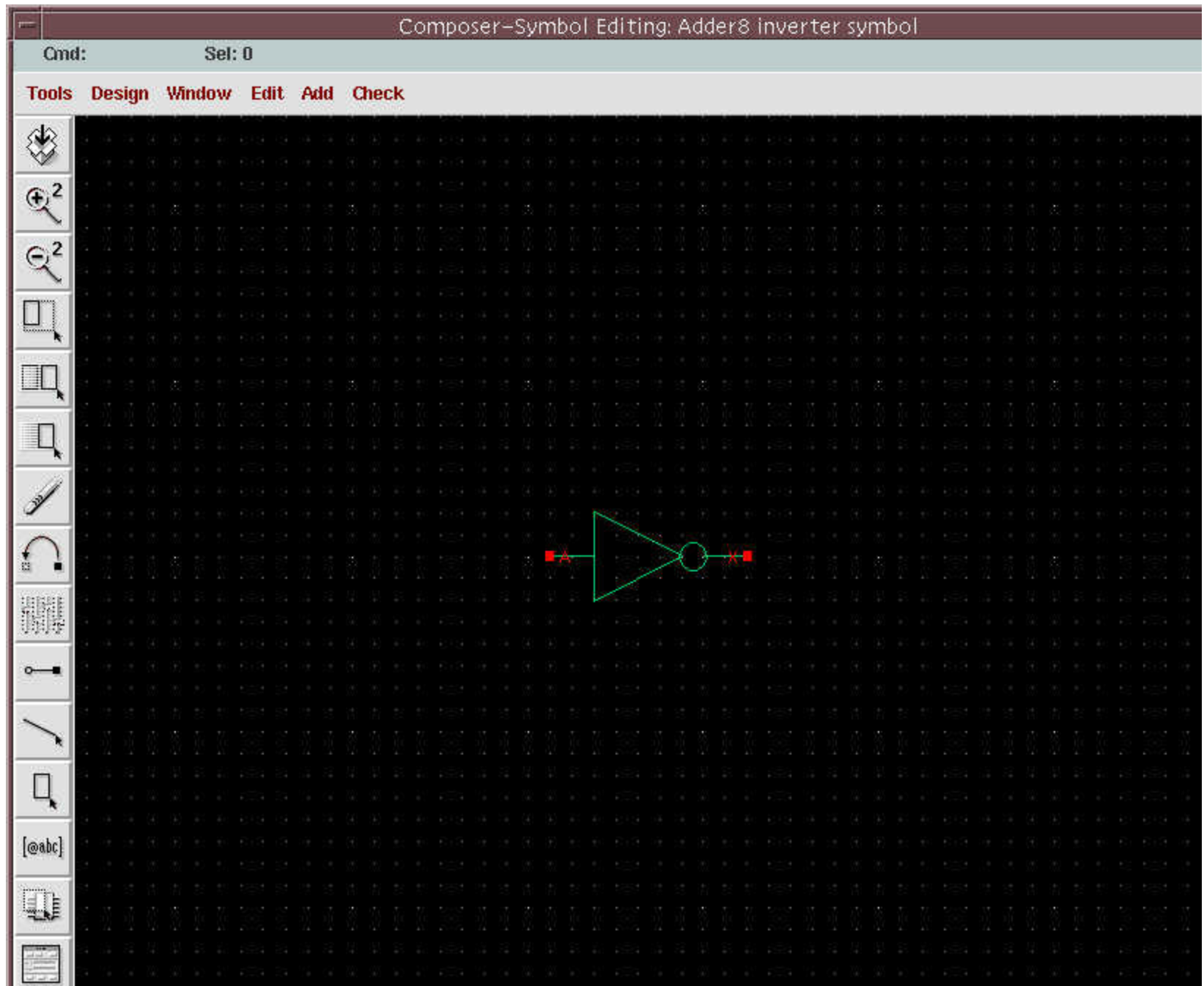
Move mouse to place A pin at input of inverter, then click "mouse L".

In Add Pin window, change direction to output.

Place X pin at output of inverter.

See





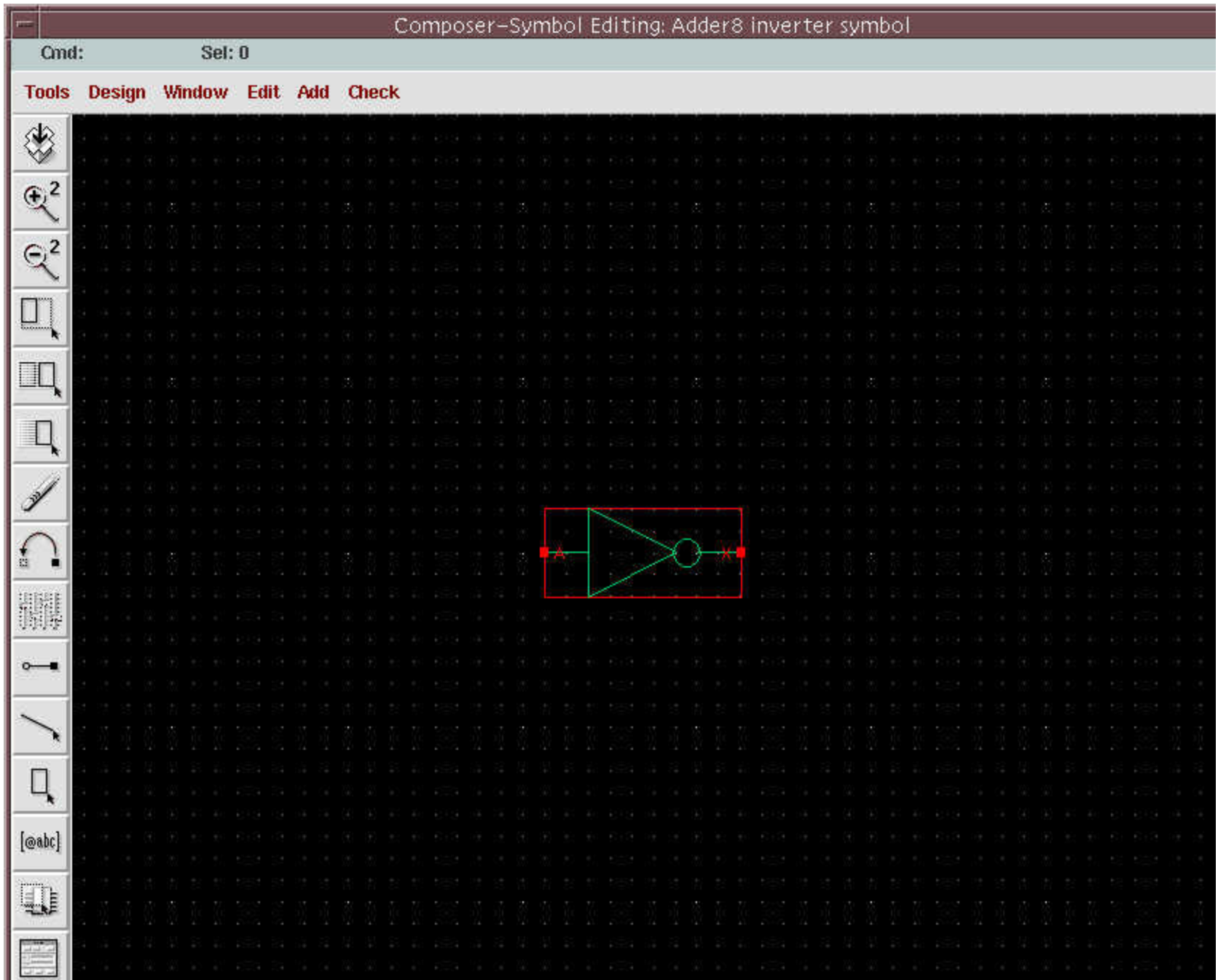
[Fig 20.](#)

**5. Add selection box.**

sym:Add->Selection Box

In Add Selection Box window, click Automatic.

See



[Fig 21.](#)

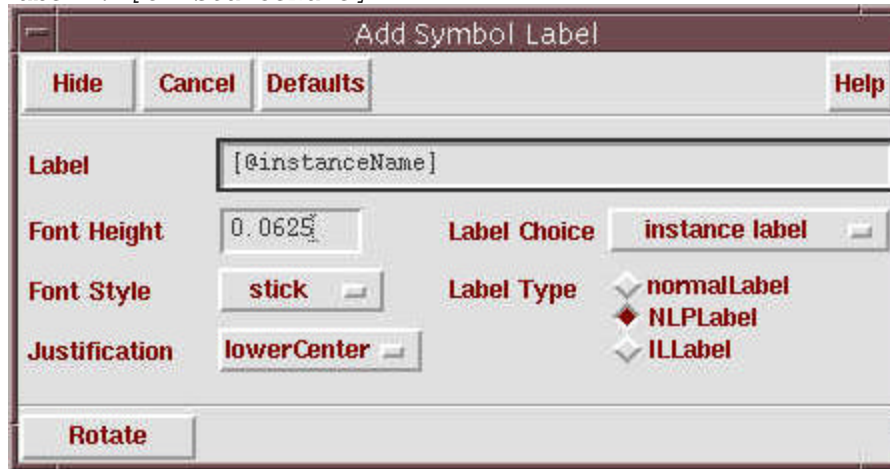
- Selection box is a boundary to select this symbol in a schematic which include the current symbol.

#### 6. Add labels.

sym:Add->Label

In Add Label window,

- Label : [*@instanceName*]

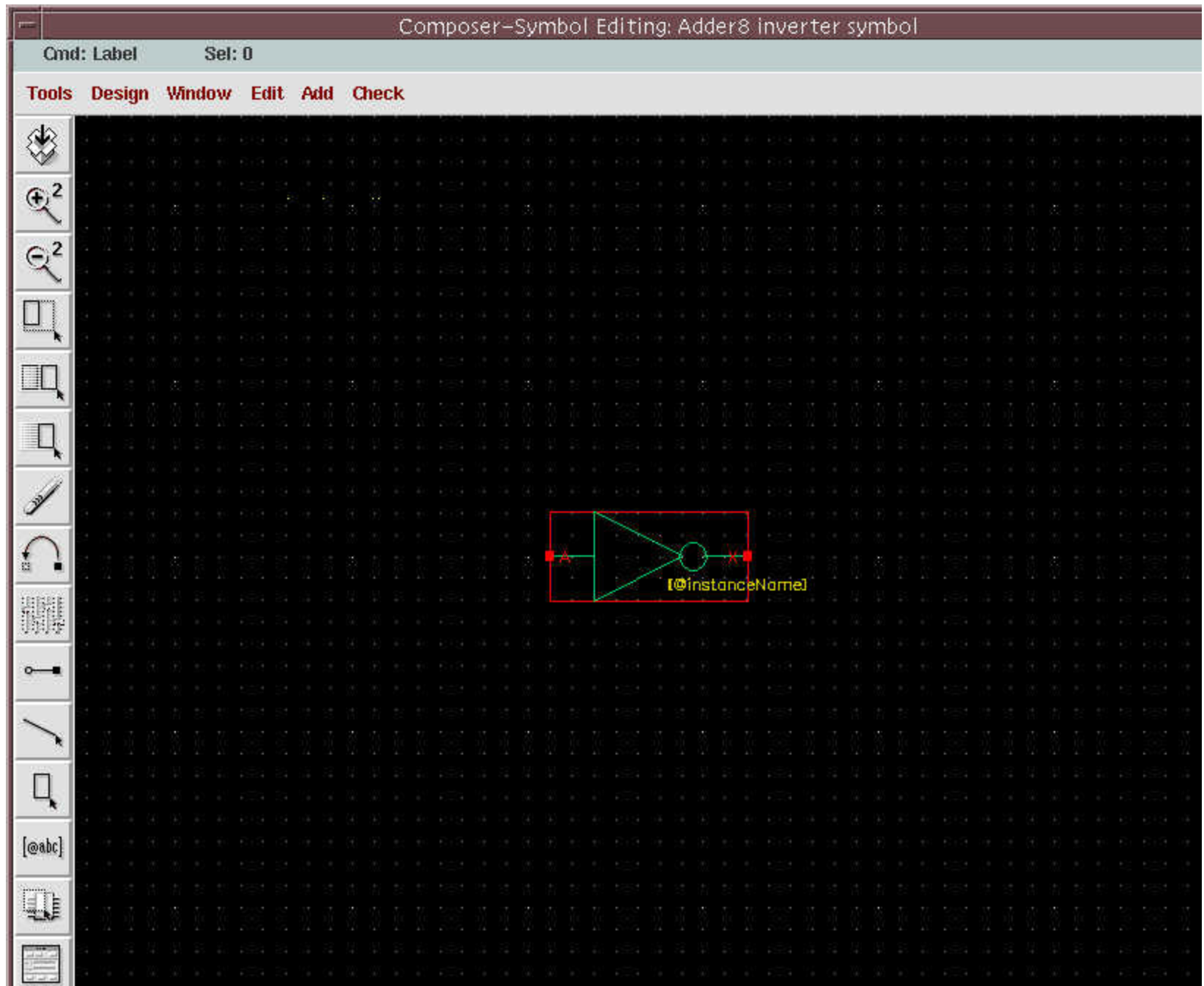


See

[Fig 22.](#)

Place label.

See



[Fig 23.](#)

Repeat Add Label for parameterized attributes.  
In Add Label window,

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



See  
[Fig 24.](#)

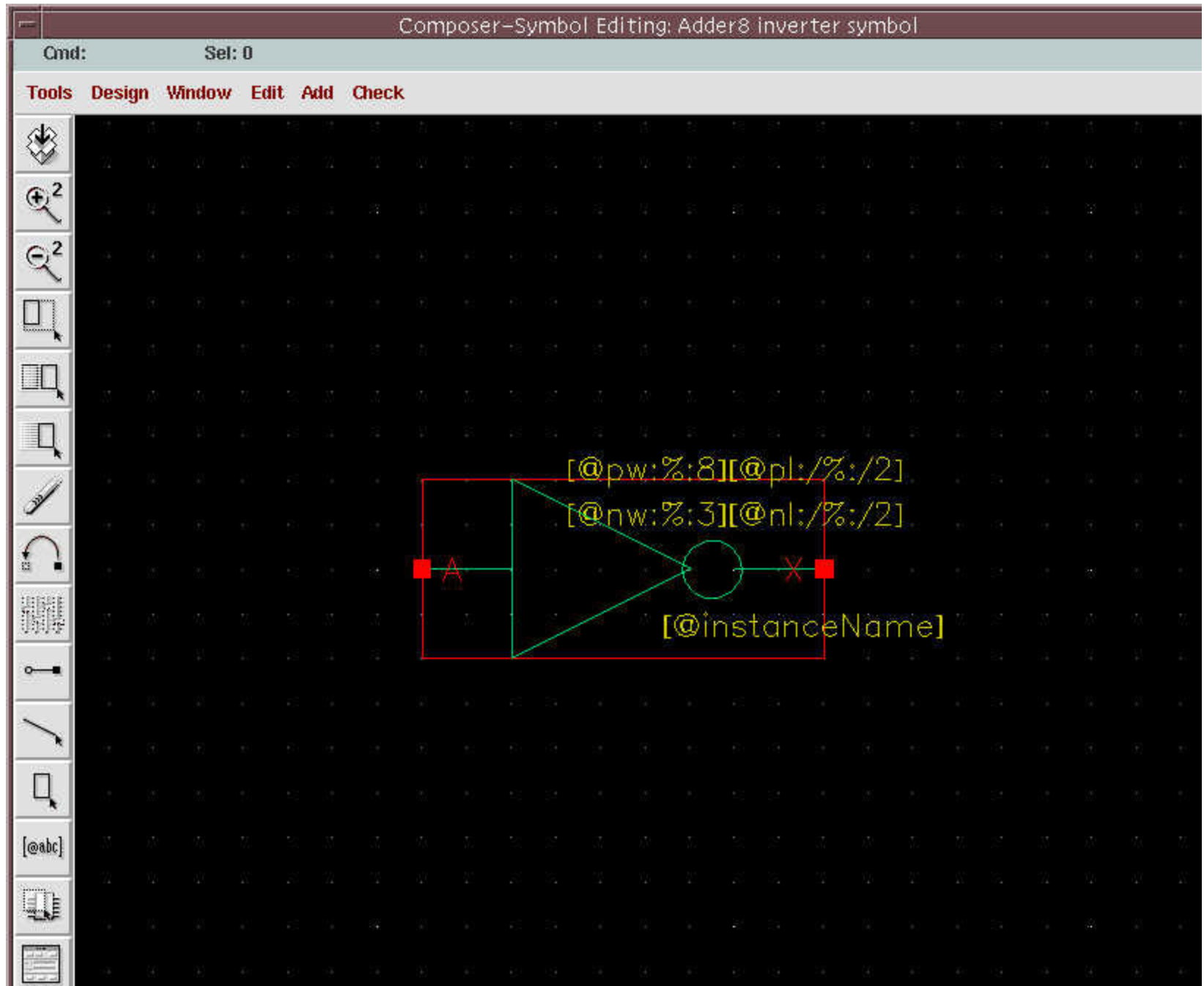
Place label.

Repeat Add Label for parameterized attributes.  
In Add Label window,

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

Place label.

See



[Fig 25.](#)

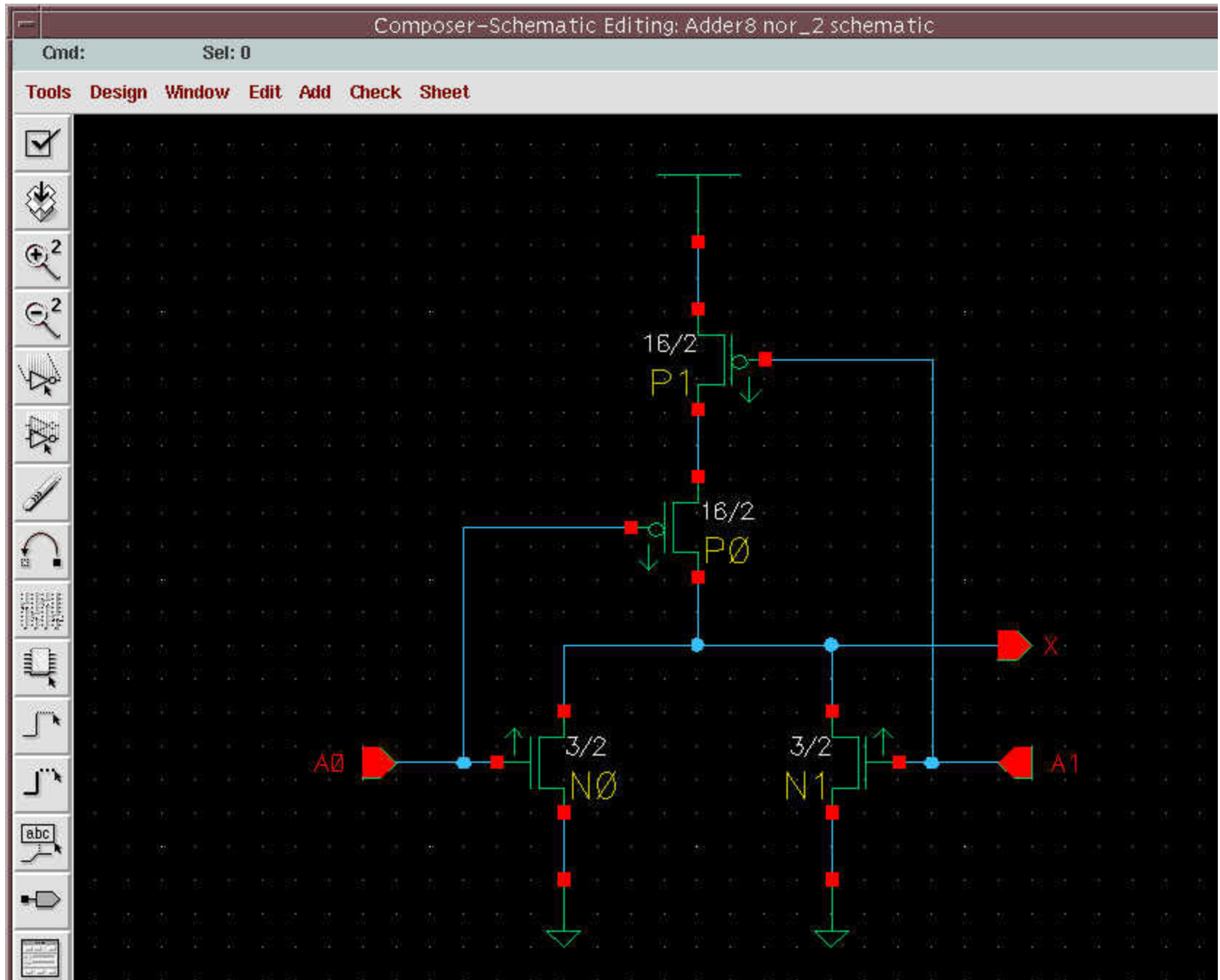
### 7. Check and Save

sch:File->Check and Save

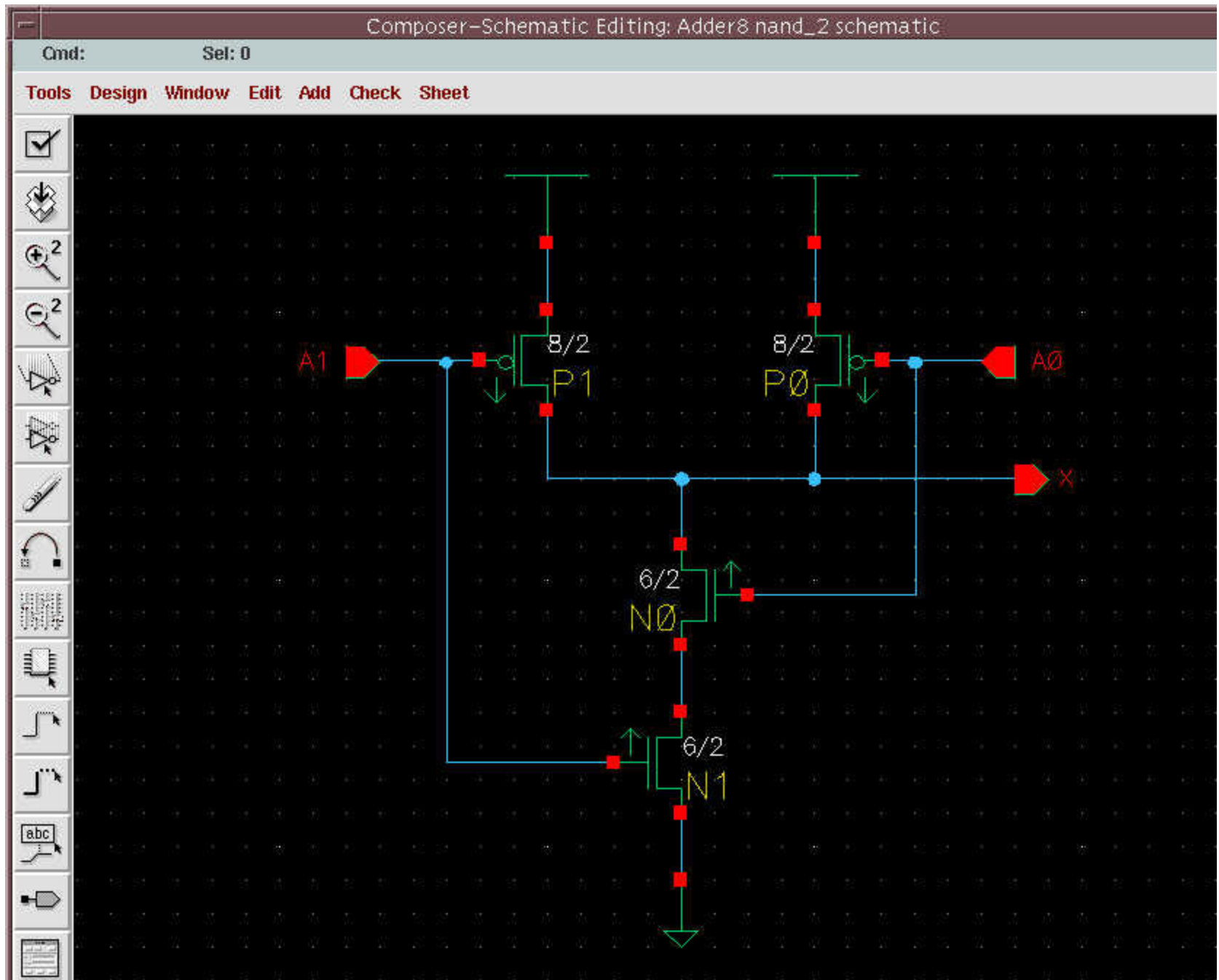
- Create Other Schematics and Symbols

Draw schematic and symbol for the following components.

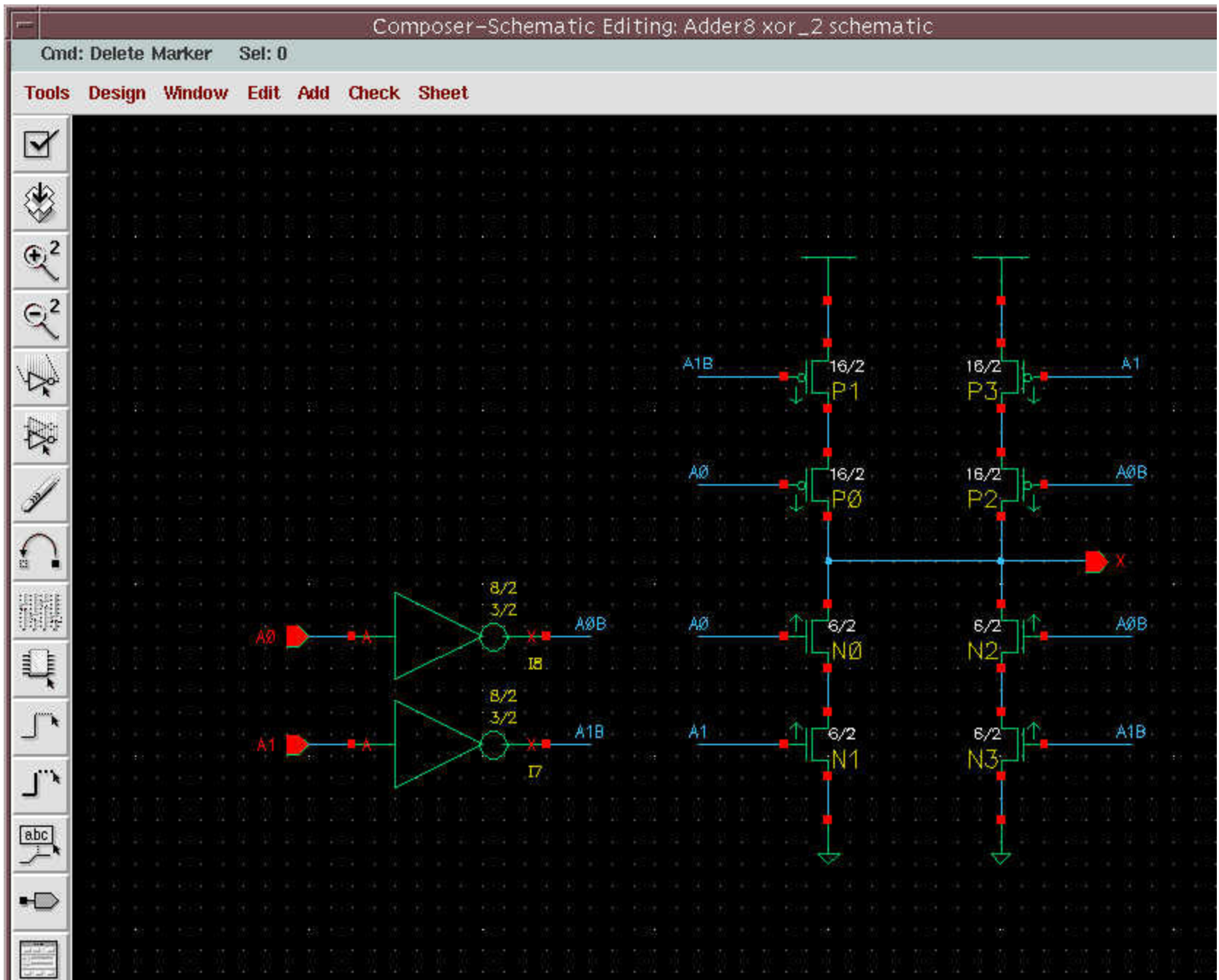




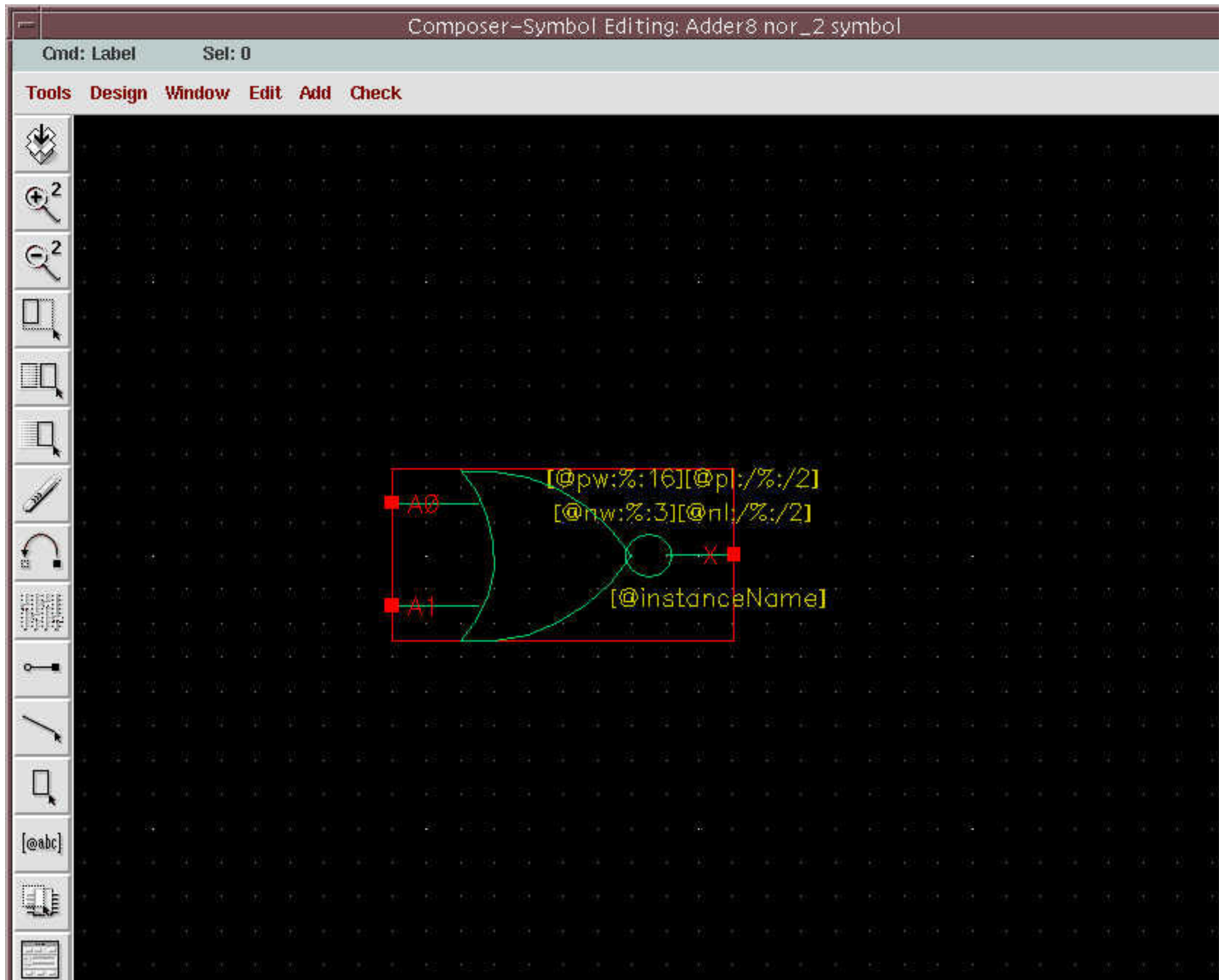
[2-input NOR gate schematic](#) : nor\_2



[2-input NAND gate schematic](#) : nand\_2

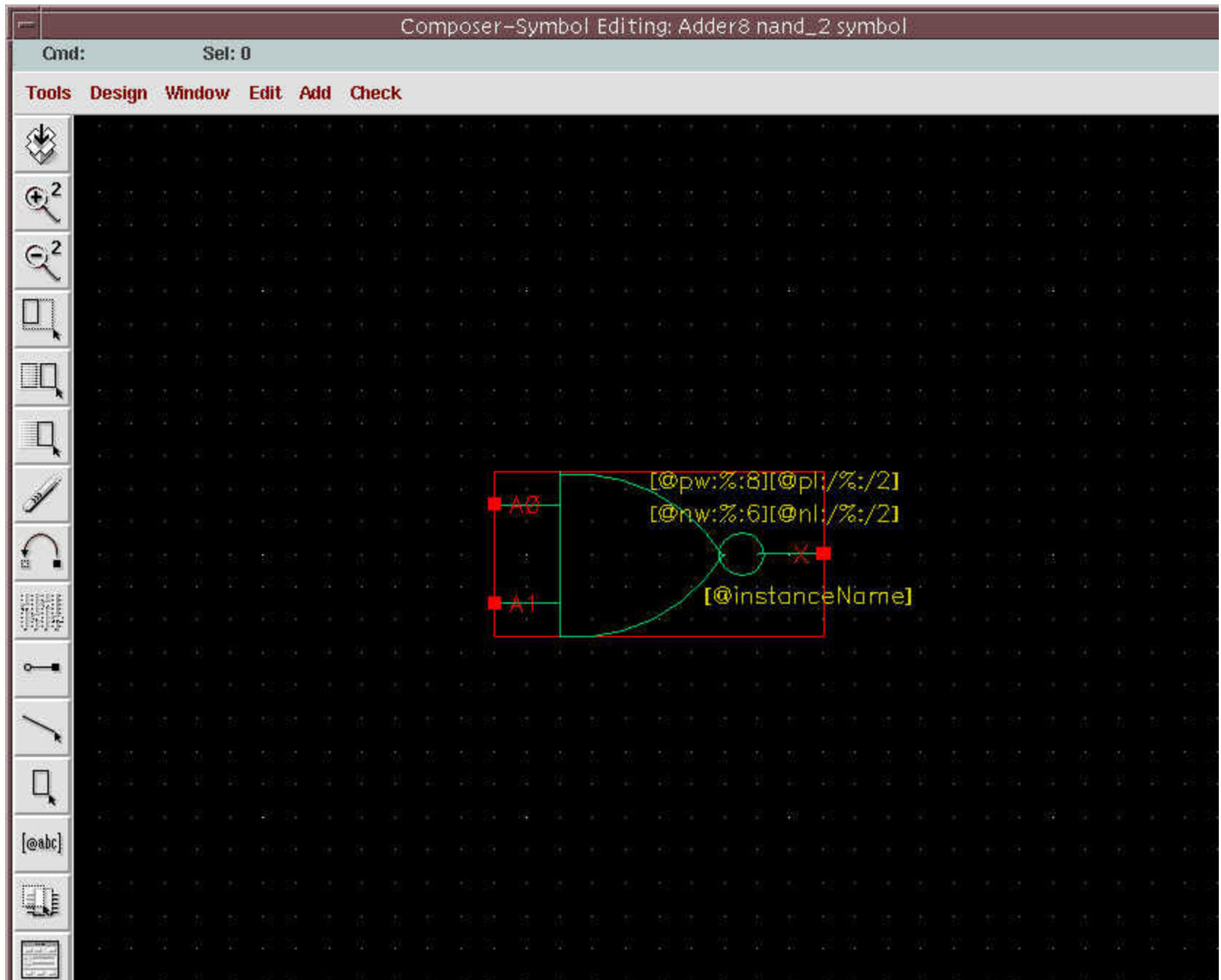


[2-input XOR gate schematic](#) : xor\_2

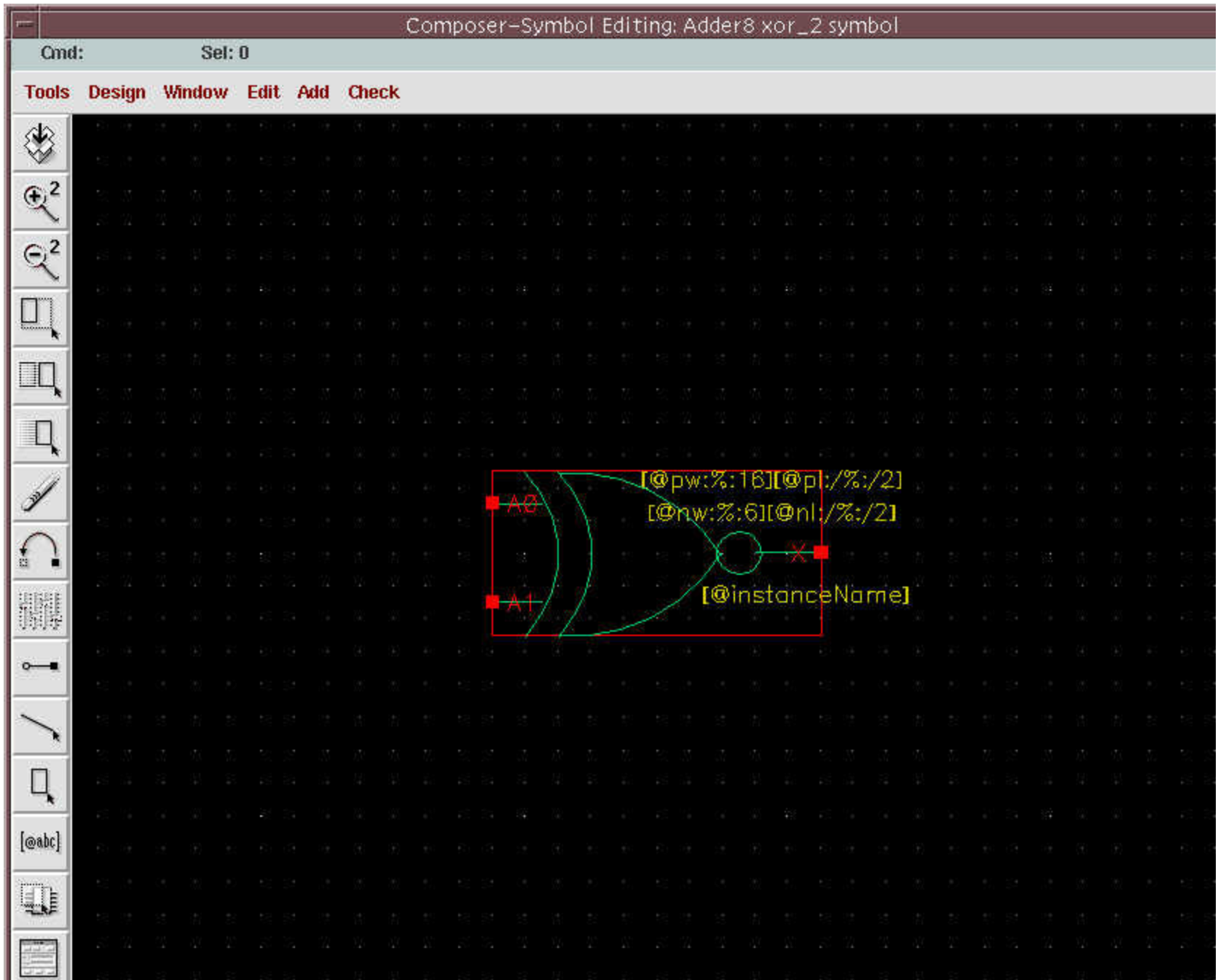


[2-input NOR gate symbol](#) : nor\_2





[2-input NAND gate symbol](#) : nand\_2

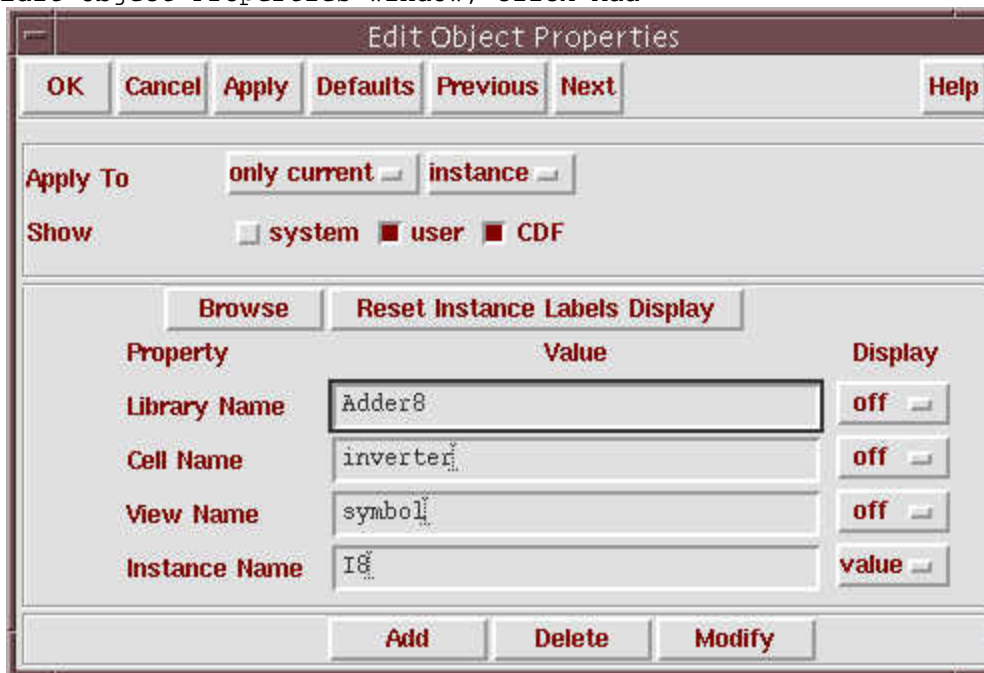


[2-input XOR gate symbol](#) : xor\_2

- In xor\_2 schematic, use wire name to place node name on wire.
- In xor\_2 schematic, use inverter we just created.
- If you want resize inverter in xor\_2 schematic,

Click inverter and type "q"

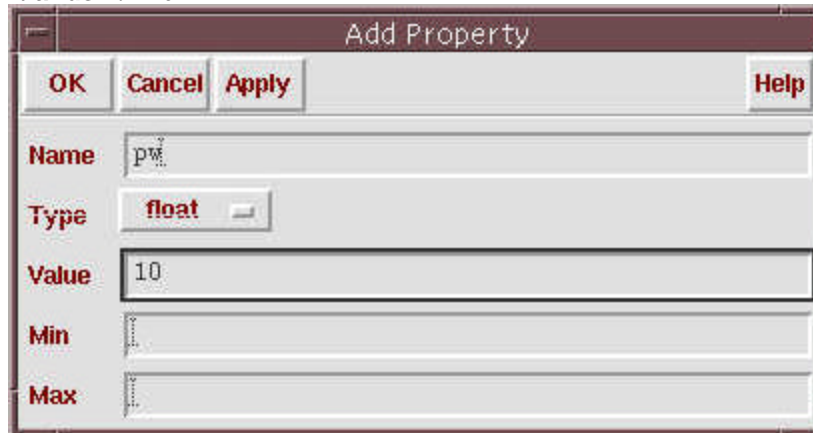
In Edit Object Properties window, click Add



See  
[Fig 32](#).

In Add Property window,

7. Name : *pw*
8. Type : *float*
9. Value : *10*



See

[Fig 33.](#)

You will see the sizing of inverter has been changed to 10/2 for pmos.

- Create Bit-slice of Adder

1. **Create schematic.**

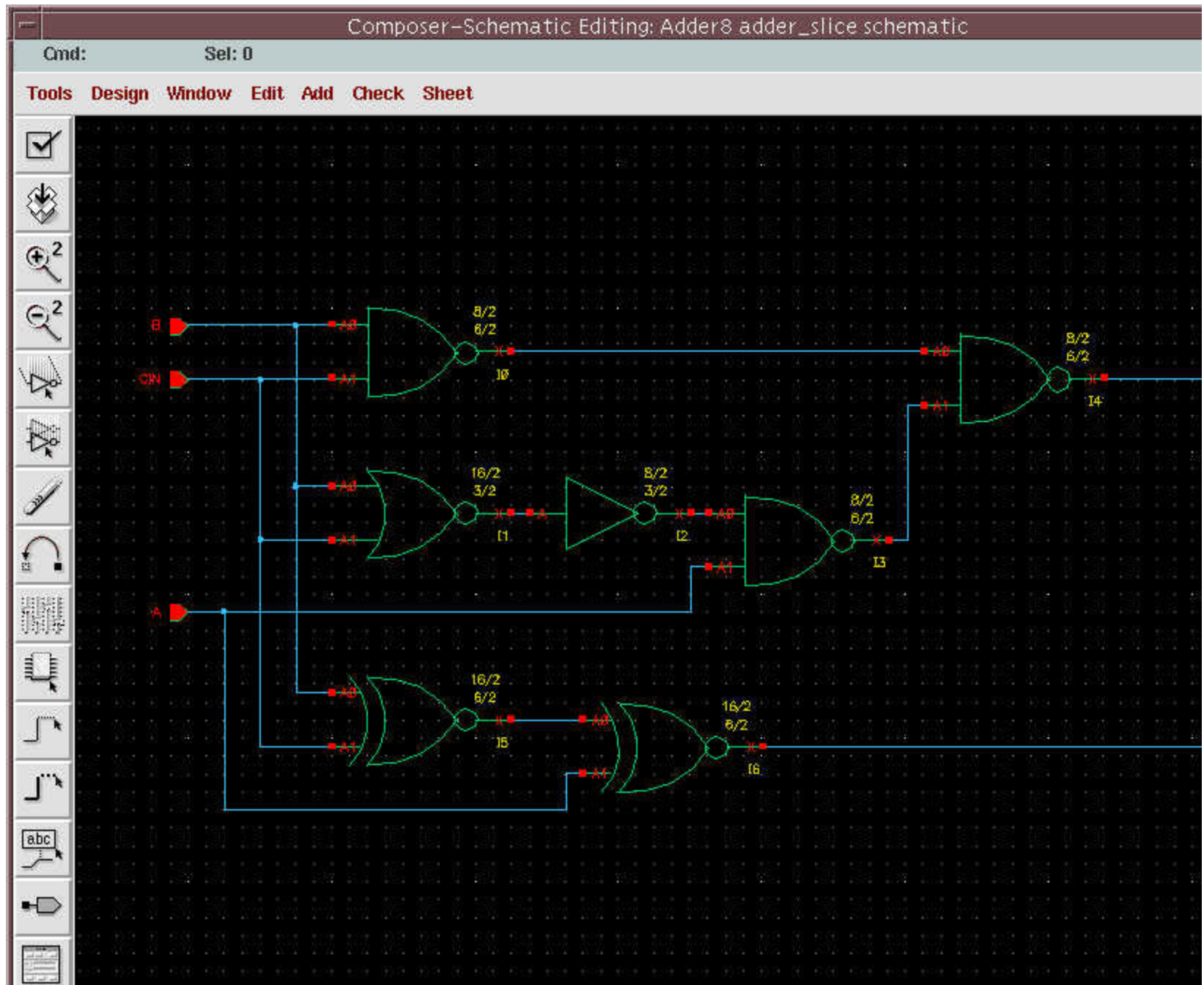
icds:File->New->Cellview

In Create New File window,

- Library Name : *Adder8*
- Cell Name : *adder\_slice*
- View Name : *schematic*
- Tool : *Composer-Schematic*

Complete schematic.

See



[Fig 34.](#)

## 2. Create symbol automatically.

sch:Design->Create Cellview->From Cellview

In Cellview From Cellview window,

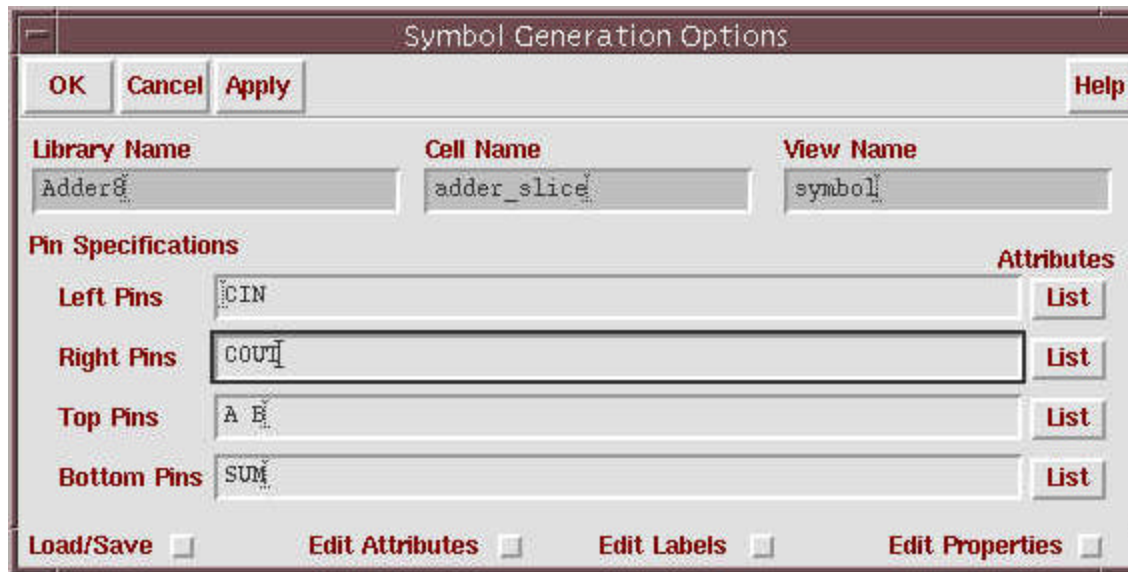
- *Library Name* : Adder8
- *Cell Name* : adder\_slice
- *From View Name* : schematic
- *To View Name* : symbol



See  
[Fig 35.](#)

In Symbol Generation Option window,

- *Left Pins* : CIN
- *Right Pins* : COUT
- *Top Pins* : A B
- *Bottom Pins* : SUM



The image shows a dialog box titled "Symbol Generation Options". At the top, there are buttons for "OK", "Cancel", "Apply", and "Help". Below these are three text input fields: "Library Name" with the value "adder8", "Cell Name" with the value "adder\_slice", and "View Name" with the value "symbol".

The main section is titled "Pin Specifications" and contains four rows, each with a label, a text input field, and a "List" button:

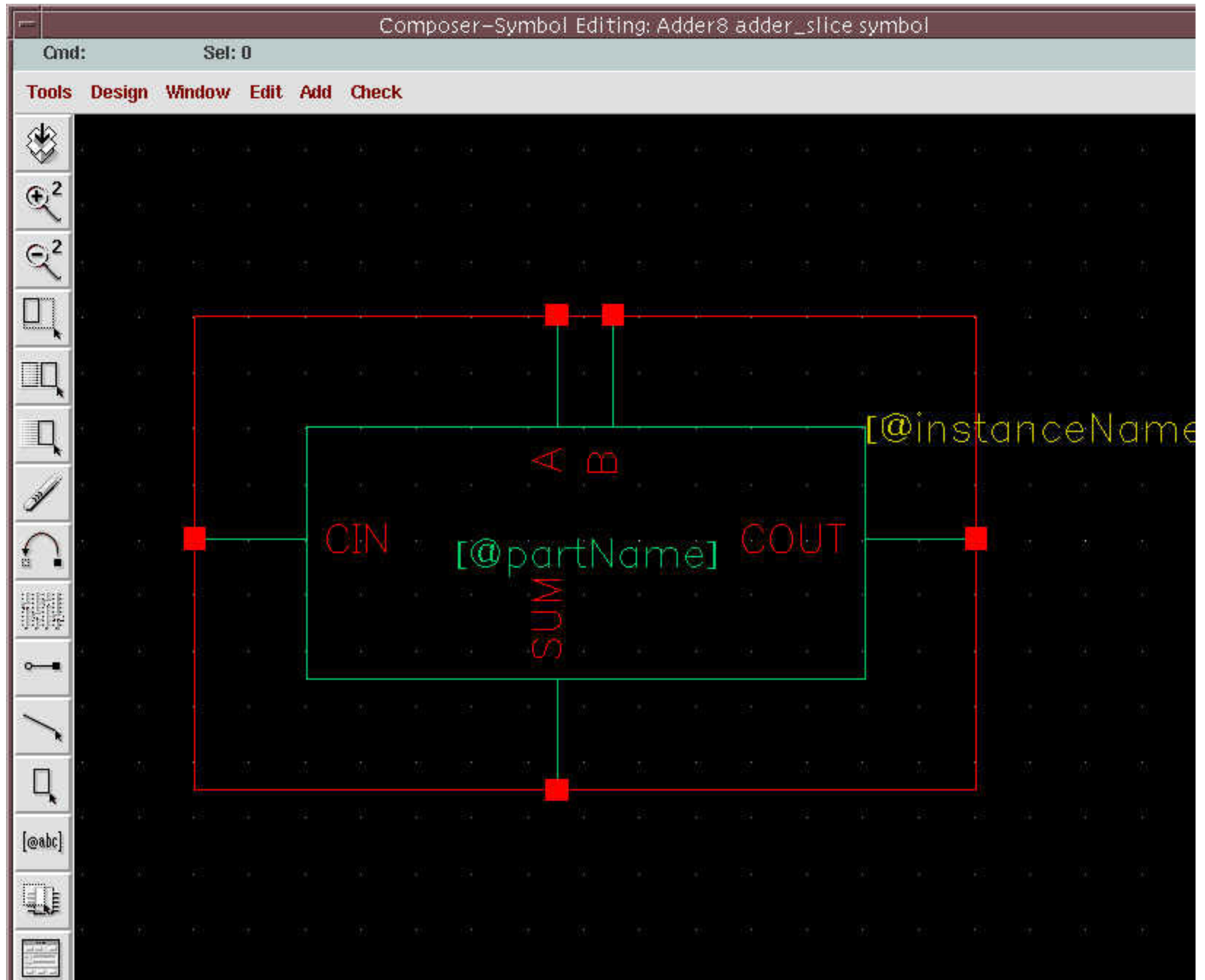
	Pin Specifications	Attributes
Left Pins	CIN	List
Right Pins	COU	List
Top Pins	A B	List
Bottom Pins	SUM	List

At the bottom of the dialog, there are four checkboxes: "Load/Save" (unchecked), "Edit Attributes" (unchecked), "Edit Labels" (unchecked), and "Edit Properties" (unchecked).

See  
[Fig 36.](#)

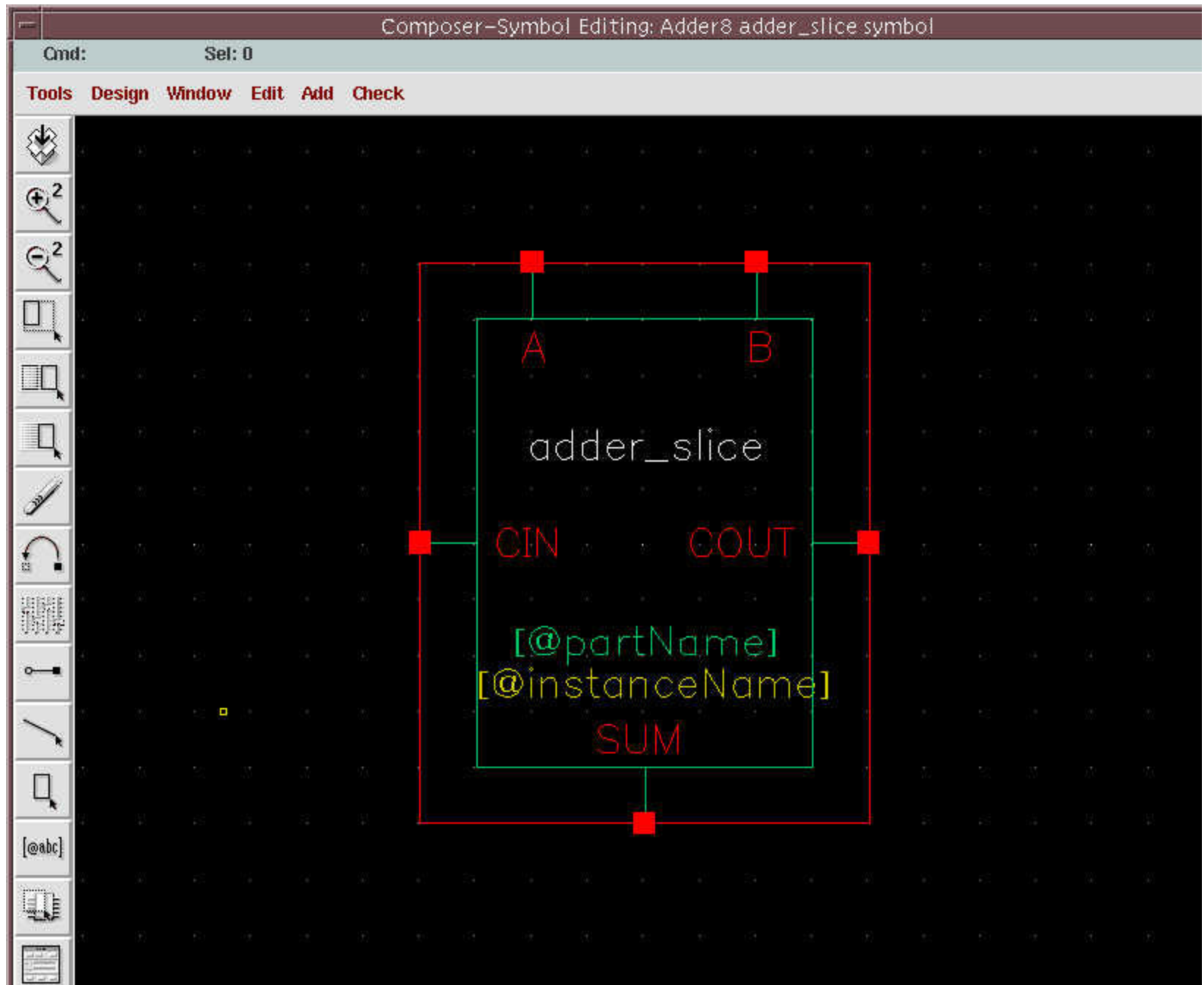
- This will display symbol window as shown in





[Fig 37.](#)

- Edit symbol as shown in



[Fig 38.](#) (Not necessary)

- Create 8-bit Adder Schematic

1. **Create schematic.**

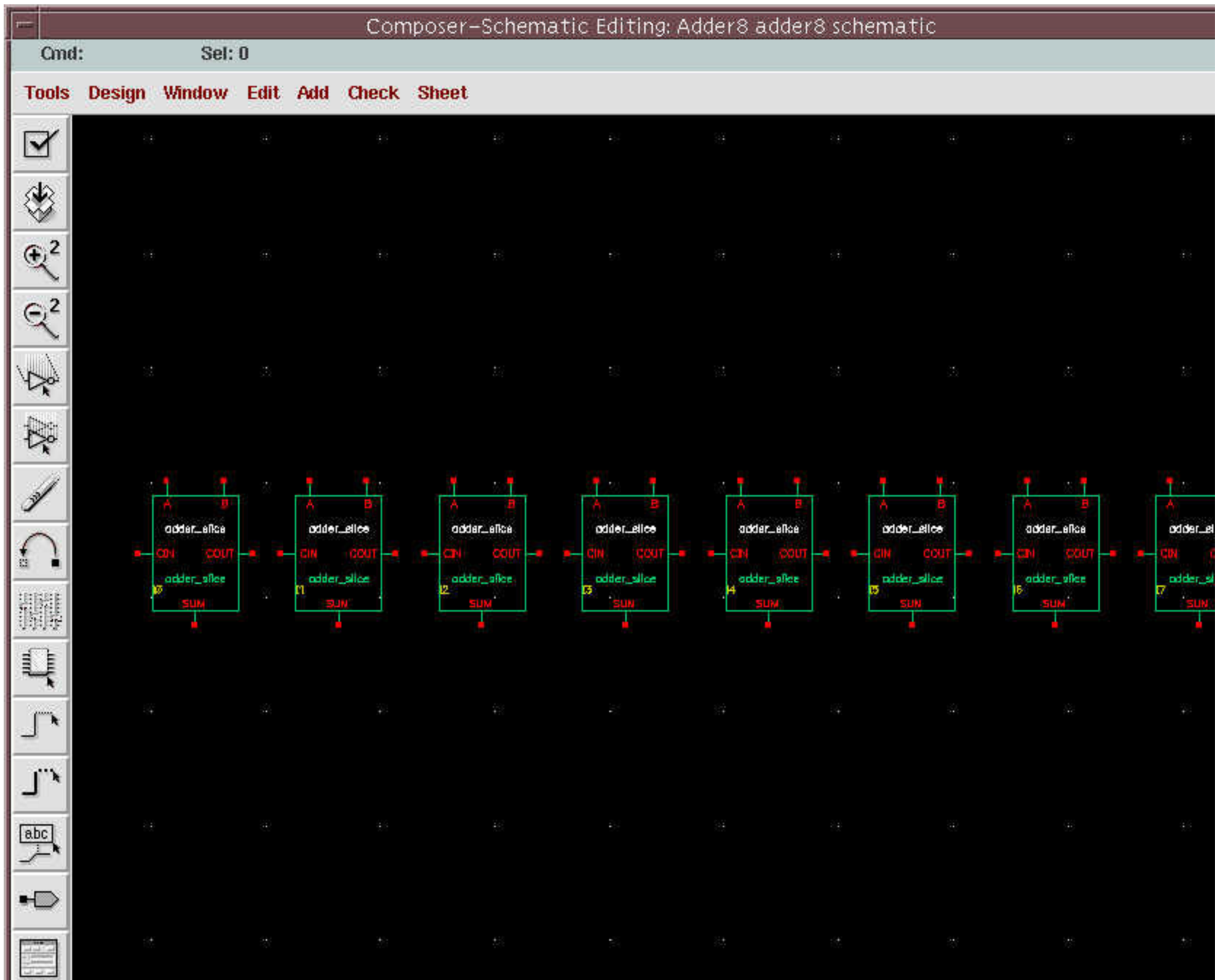
icds:File->New->Cellview

In Create New File window,

- *Library Name : Adder8*
- *Cell Name : adder8*
- *View Name : schematic*
- *Tool : Composer-Schematic*

2. **Place adder\_slice.**

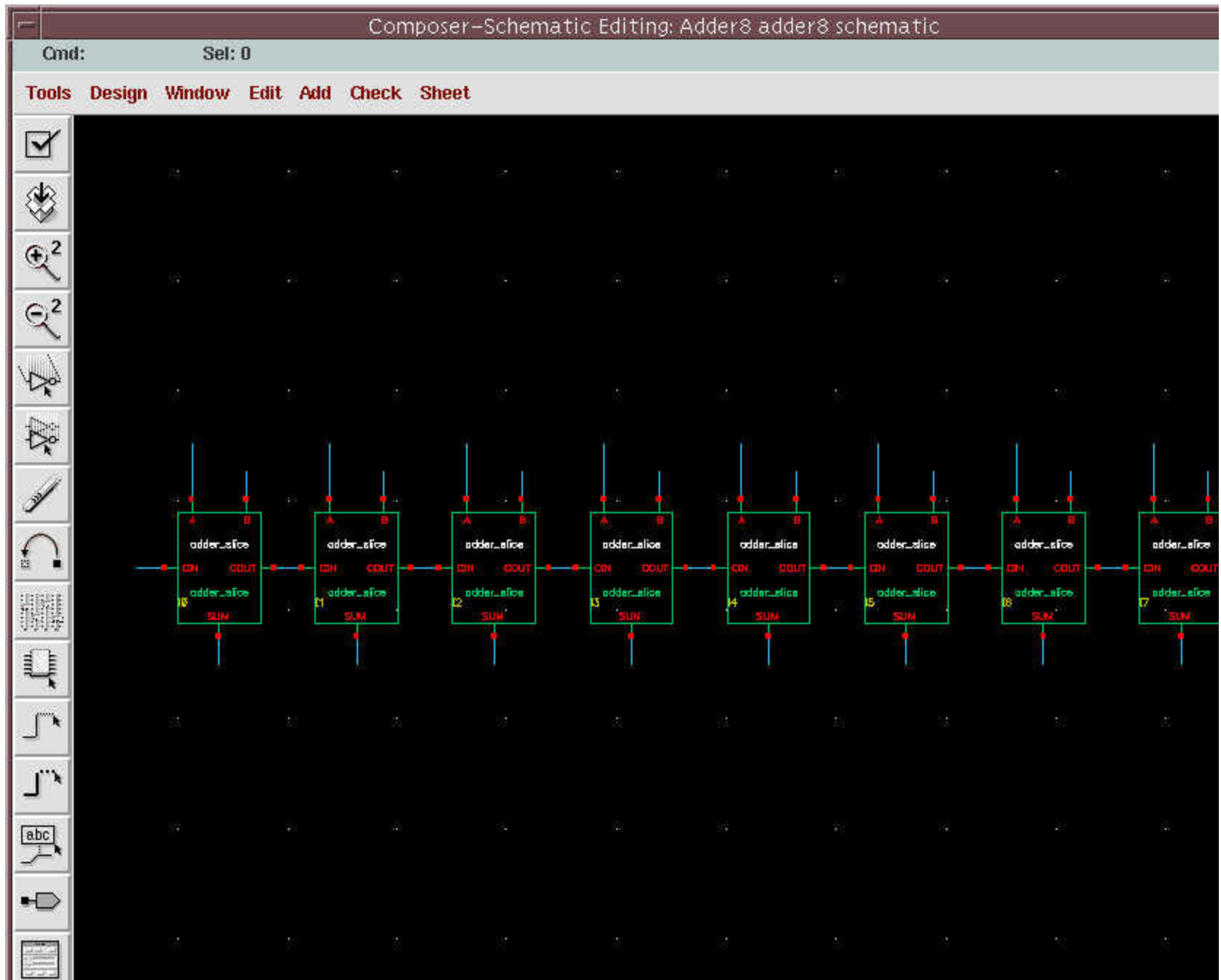
See



[Fig 39.](#)

3. **Draw wires.**

See

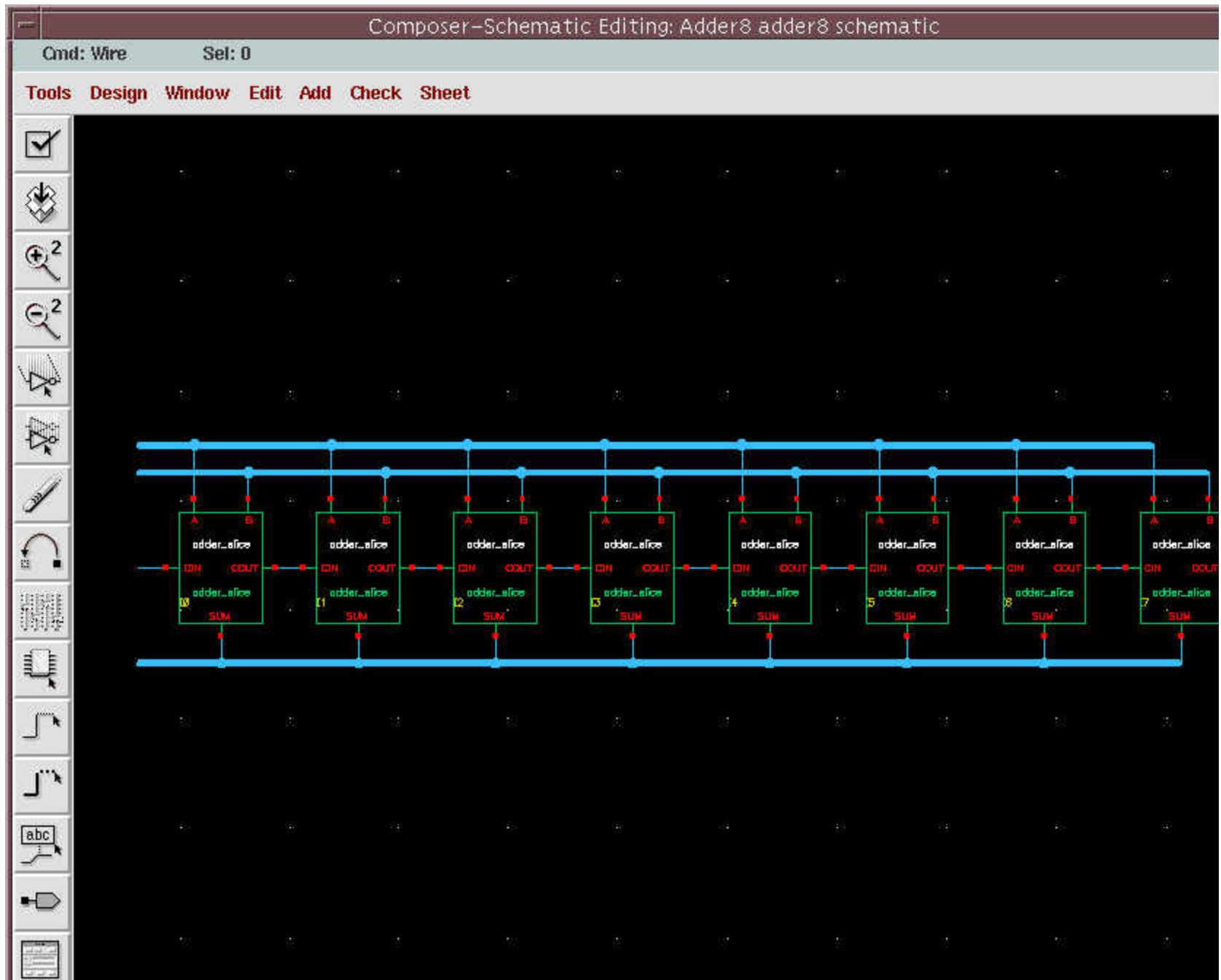


[Fig 40.](#)

4. **Draw buses.**

See





[Fig 41.](#)

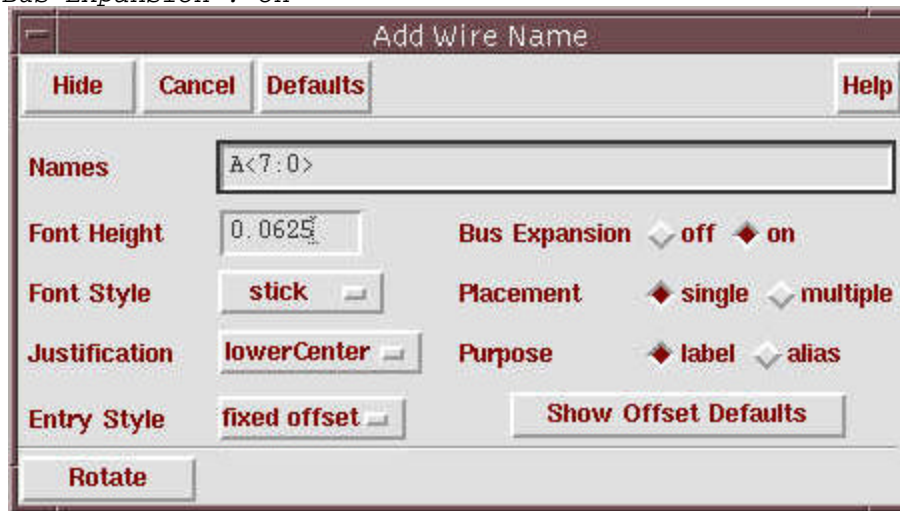
- Use Add->Wire(wide) (or "W")

### 5. Add wire labels on A bus.

sch:Add->Wire Name

In Wire Name window,

- Names : A<7:0>
- Bus Expansion : on

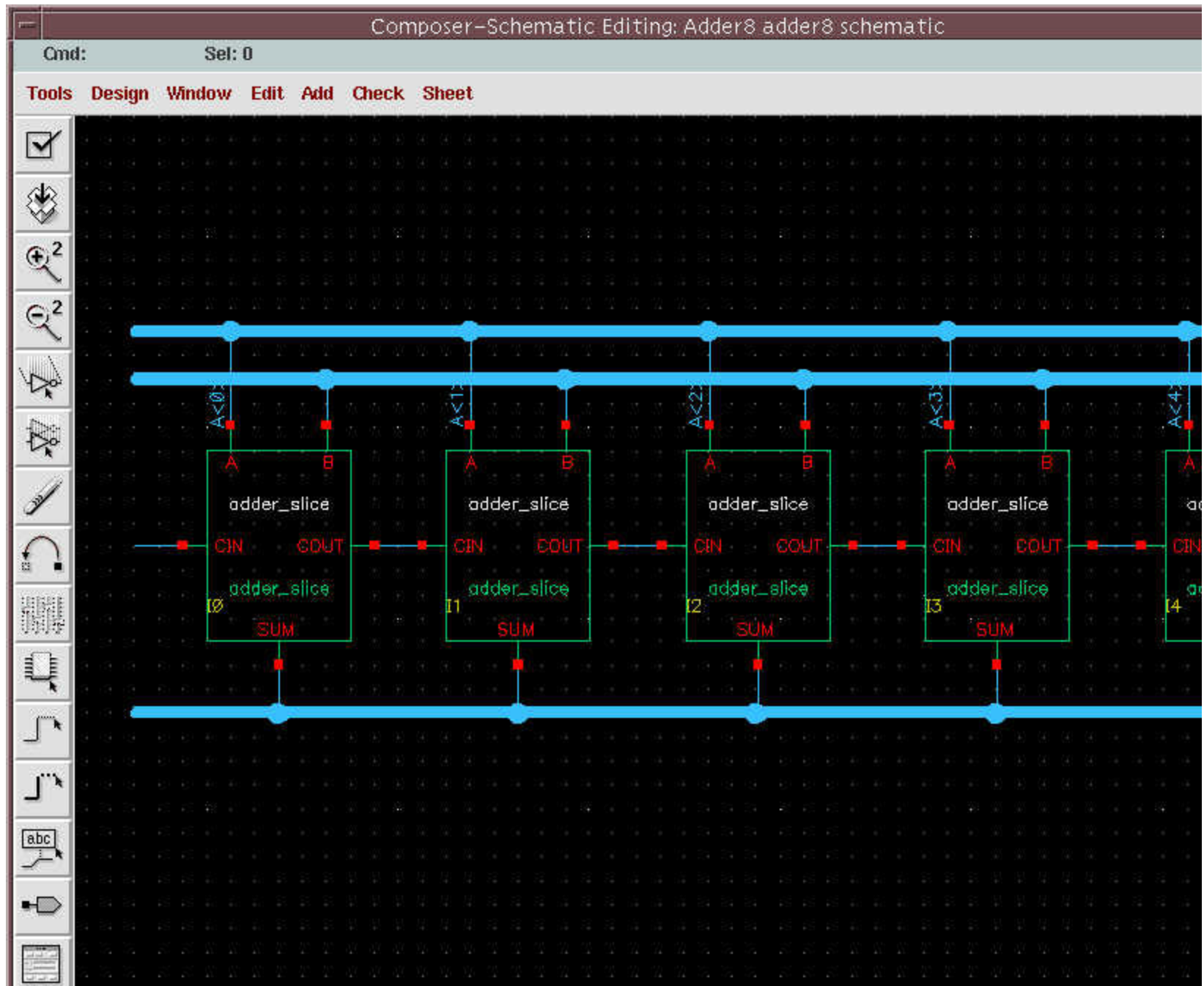


See

[Fig 42.](#)

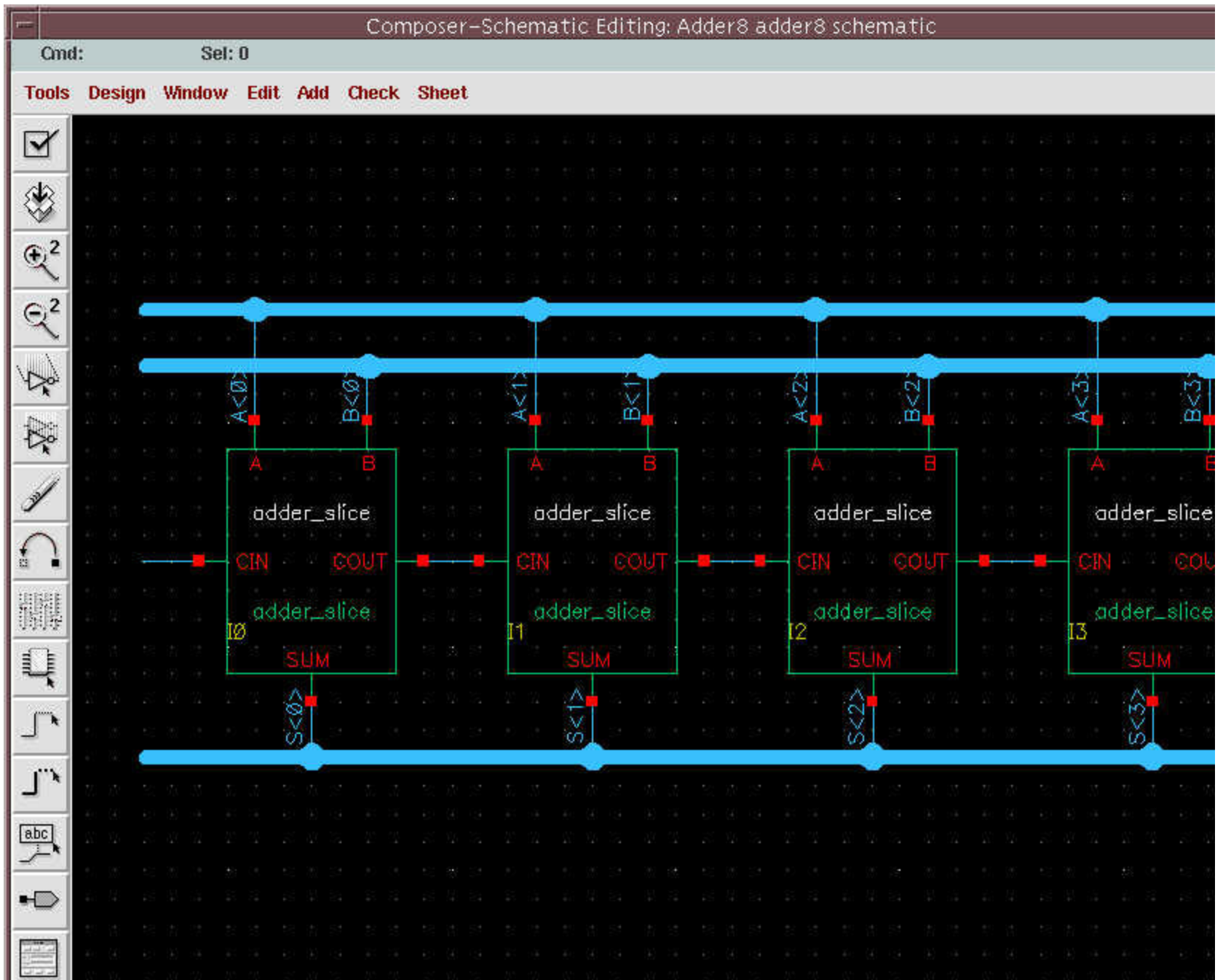
Place label on each wire

See



[Fig 43.](#)

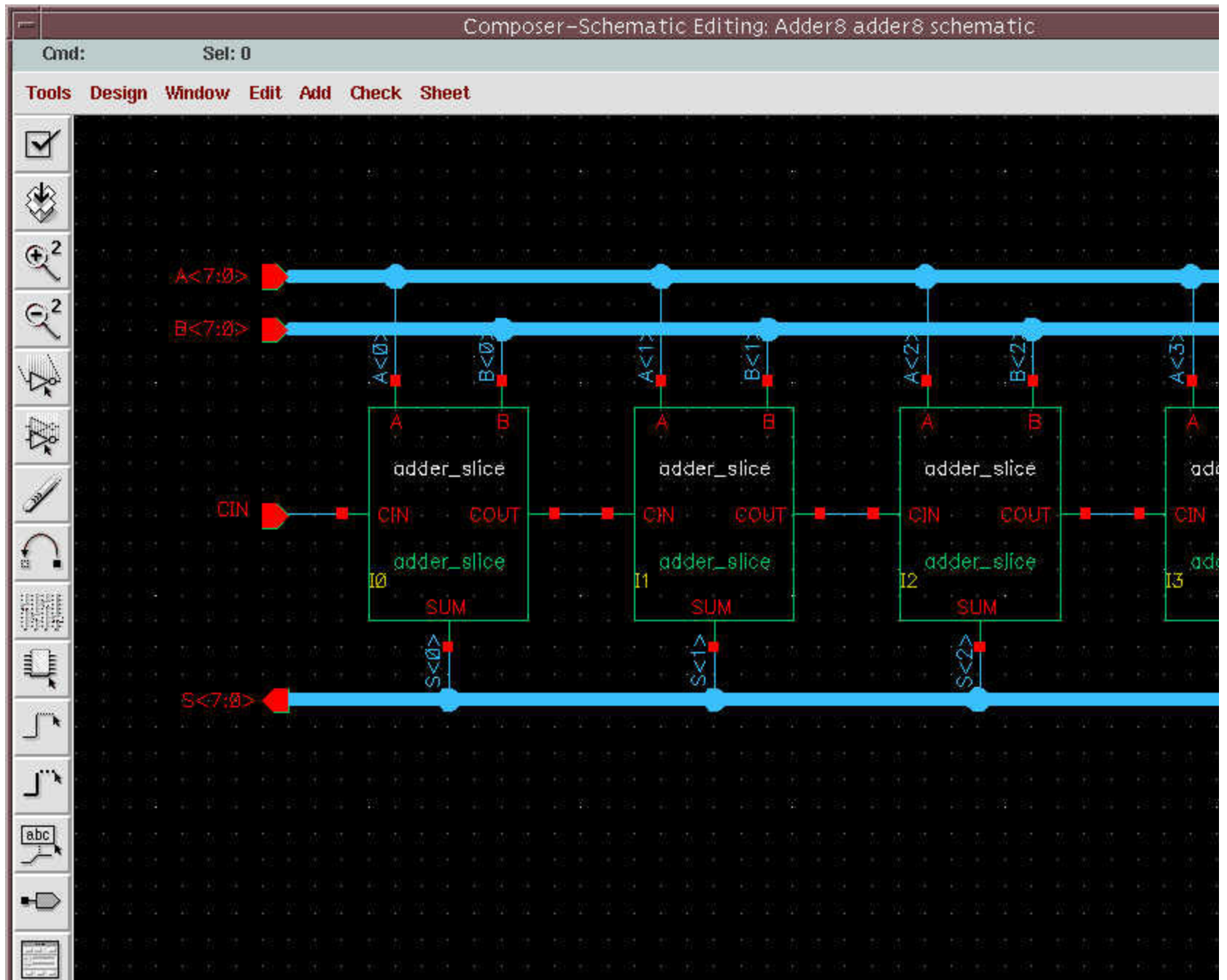
- 6. Repeat for B and S buses.**  
See



[Fig 44.](#)

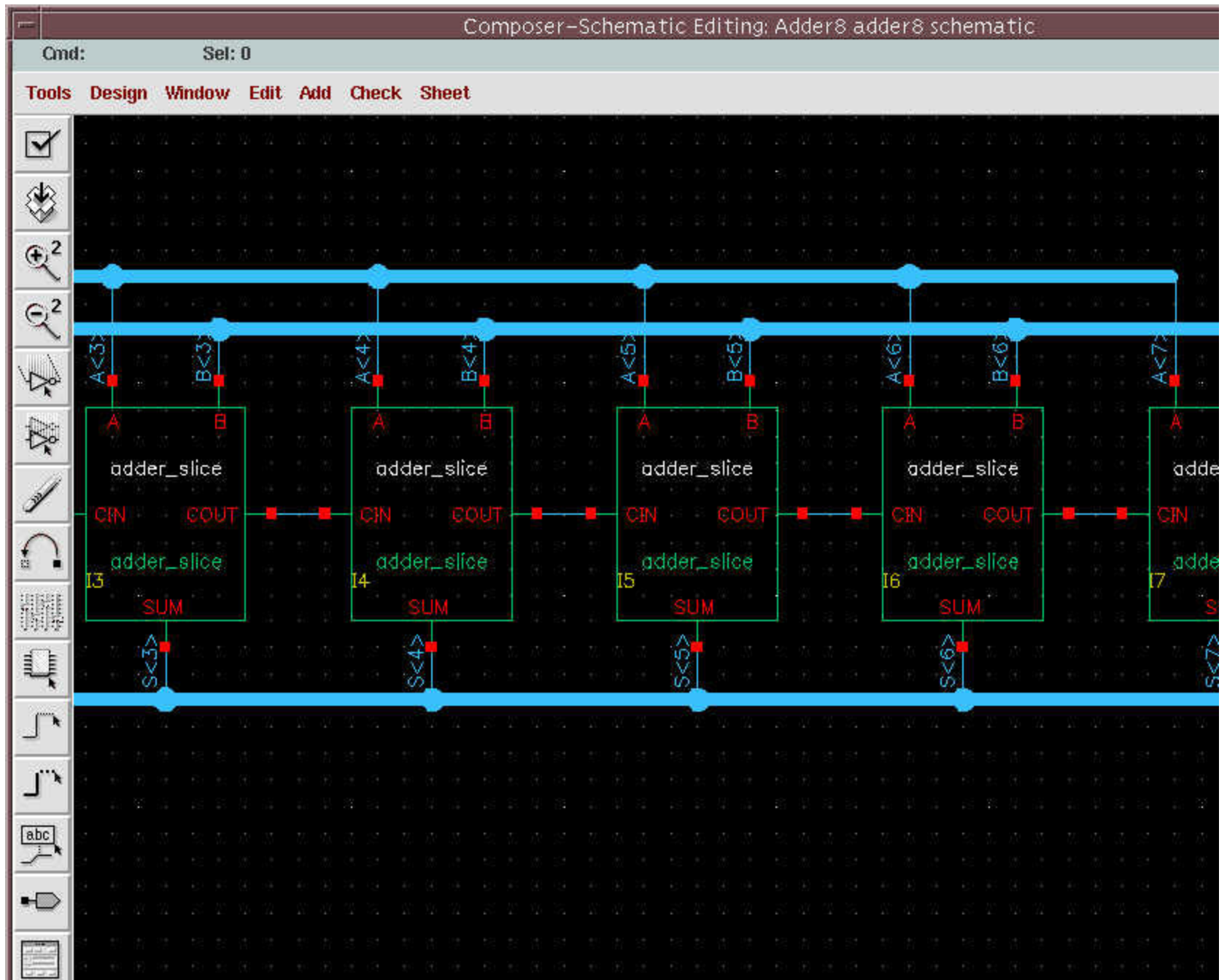
7. **Add pins and complete schematic.**

See



[Fig 45](#) and





[Fig 46.](#)

8. **Check and Save**

sch:File->Check and Save

---

*Copyright [Joong-Seok "Jay" Moon](#)*

The University of Southern California does not screen or control the content on this website and thus does not guarantee the accuracy, integrity, or quality of such content. All content on this website is provided by and is the sole responsibility of the person from which such content originated, and such content does not necessarily reflect the opinions of the University administration or the Board of Trustees