Electric CAD Tool

We will be using the Electric VLSI Design System for schematic entry, which can be downloaded from

http://www.staticfreesoft.com/productsFree.html

Download the latest version of the GNU Electric Binary Release. The software will be in the form of a jar that you can run on your laptop.

You can also access this on the CETS computers by running:

```
Electric
```

There is a complete online user's manual:

http://www.staticfreesoft.com/documentsUser.html

We will point you at specific parts of the manual to get started, but you may find it useful to read other parts as you get started or want to learn how to do more with the tool.

Getting Started

When running electric for the first time, you may the display color settings difficult to see. To change them, go to

File -> Preferences -> Display -> Layers

From the Technology dropdown menu, select schematic

Layer colors you may want to change are

Special:BACKGROUND, Arc, Special:HIGHLIGHT

In this tutorial, we will walk you through the process of creating your own inverter.

Start a new library and name it “ese370”

File -> New Library

Create a new cell to hold your schematic.

Cell -> New Cell

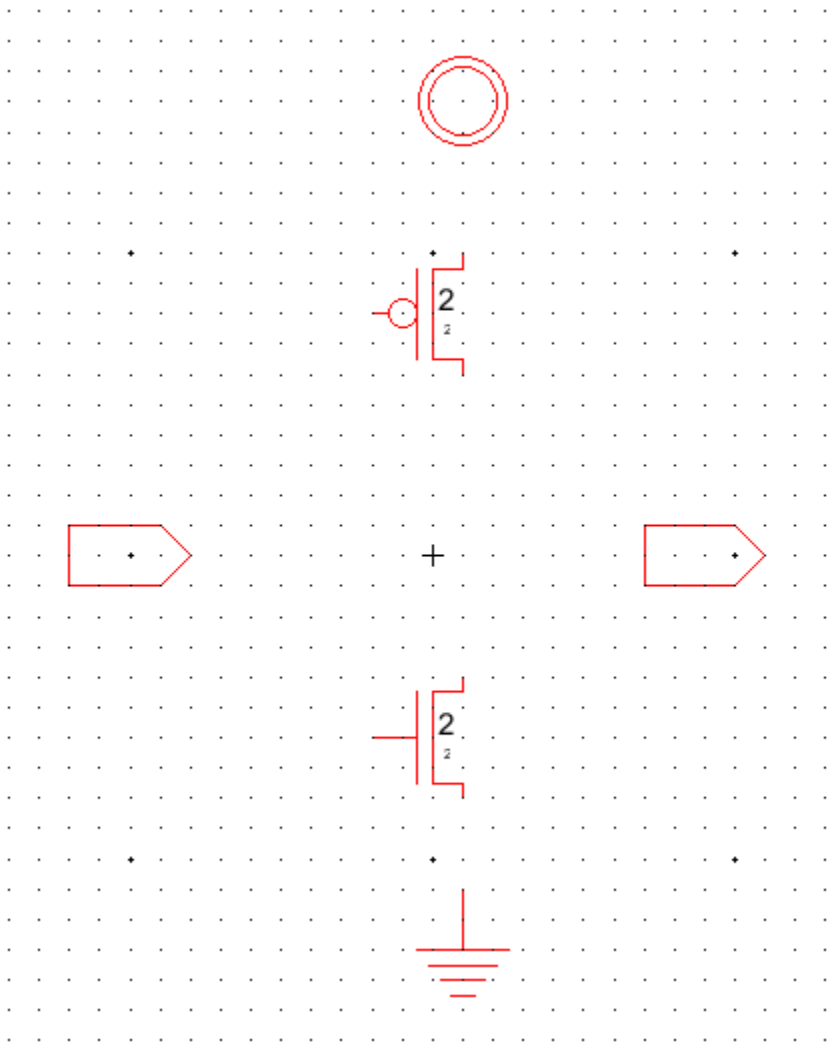
Select “schematic” as your view and name your cell “inv”

Under the components tab, switch the dropdown menu to **schematic** to access the MOSFET symbols

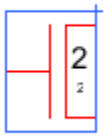
Left click on the component you want to place, and left click again in the workspace pane to place the component.

To move around on the workspace pane, use your middle mouse button or the scroll bars on the sides to pan, and “CTRL-middle” or “CTRL-0” and “CTRL-7” to zoom in and out.

Place the components like below. To make it easier to align the components, use the **Toggle Grid** command from the **Window** menu or CTRL-G



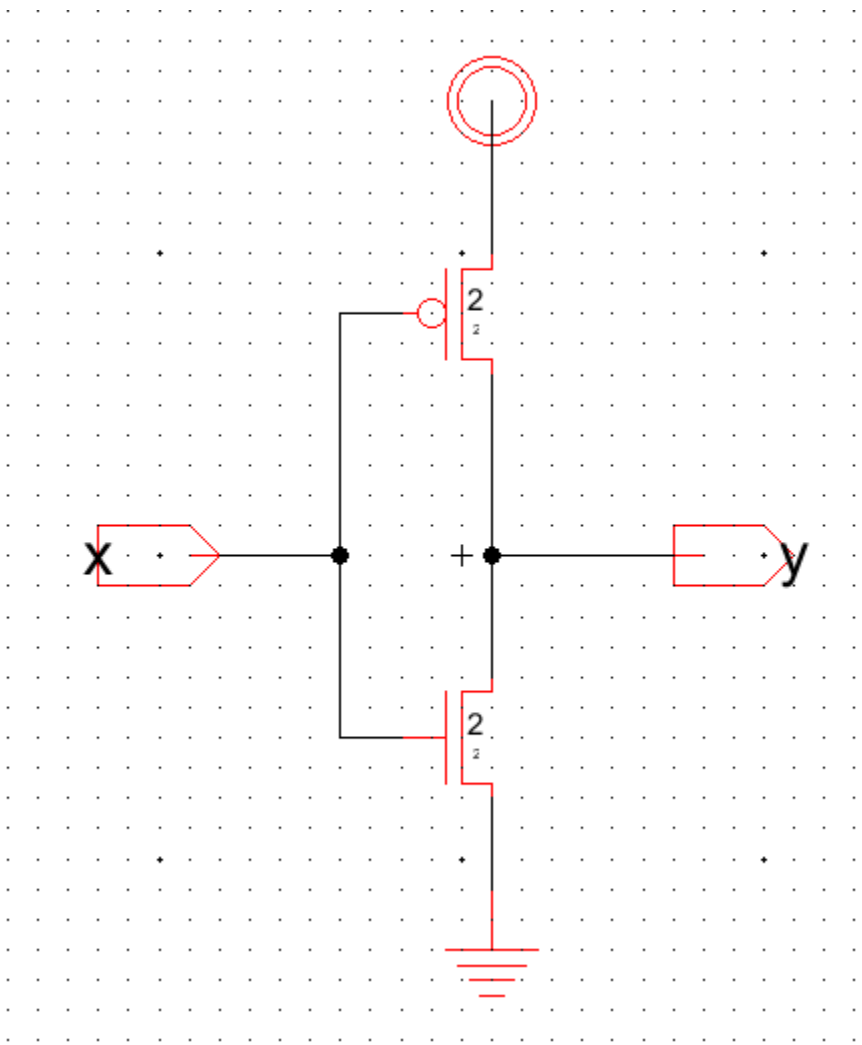
To draw wires between components, hover over a node until the component is highlighted and the node is marked with a cross. Use your left mouse button to select the component and node.



Hold down your right mouse button and drag to begin drawing a wire, called arcs in Electric. Connect all the components like below.

To edit the wires, click and drag to move them, or click to select them before using your delete button to remove them.

When the schematic is wired, you will need to create exports which define inputs and outputs of the cell. Select a port and use the **Create Export** command (in menu **Export**), or just type Ctrl-E. Name the input pin export "x" and define its characteristic as "input". Similarly, name the output pin export “y” and define its characteristic as “output”



Now is a good time to save your library. Use the **Save Library** command (in menu File), or just type Ctrl-S. Get into the habit of saving your library regularly. Also, learn the keyboard shortcuts for the commands you use frequently.

To access a more detailed tutorial, go to

http://www.staticfreesoft.com/jmanual/mchap01-12-01.html#mchap01-12-01.html