

EECS 312: Digital Integrated Circuits
Hierarchy Tutorial – Creating Hierarchical Components in Schematic Capture

Written by David Bild

1 Introduction

This tutorial explains how to create and use hierarchical components in Schematic Capture. You have already used components (e.g., NMOS, PMOS, Cap, and Vdc) in your previous assignments. However, when working on the project, you will be creating components (e.g., TGs, inverters, NAND gates) that you want to use multiple times. Rather creating multiple copies from transistors, you can create one copy of the device and then instantiate multiple instances of it in your higher-level schematics.

2 Tutorial

This tutorial walks you through the creation of a simple minimum-size inverter.

1. Create and initialize a folder and library named “**InvTutorial**” for this tutorial. See Lab 1 for instructions.
2. Create a new schematic cell view for the inverter.
3. Add the “**nmos**”, “**pmos**”, “**vdd**”, and “**gnd**” symbols and connect them as shown in Figure 4. Because we want to body terminals to be tied to the default values of ground and Vdd, use the “**nmos**” and “**pmos**” transistors instead of the four-terminal “**nmos4**” and “**pmos4**” transistors. This keeps the schematic looking cleaner and reduces the potential for wiring mistakes.
4. We will now specify the input and output ports that will appear on the inverter symbol. In this case, we have one input (“**A**”) and one output (“**F**”). We do not need to include inputs for “**Vdd**” or “**Gnd**”. The “**Vdd**” and “**Gnd**” symbols indicate that these are *global* nets. All nets connected to “**Vdd**” or “**Gnd**” symbols are considered connected, even if they are not explicitly connected by wires on the schematic. In your top-level schematic, you will assign specific voltages to these global nets by placing a “**Vdc**” voltage source between “**Vdd**” and “**Gnd**”.
 - (a) Click on the “**Pin**” button on the left-side toolbar. An “**Add Pin**” window will appear.
 - (b) Set the direction to “**Input**” and the Pin Name to “**A**”.
 - (c) Click on the schematic to place the input pin.
 - (d) Change the direction to “**Output**” and the Pin Name to “**F**”.
 - (e) Click on the schematic to place the output pin.
 - (f) Close the “**Add Pin**” window.
5. Connect the input and output pins as shown in Figure 4.
6. “**Check and Save**” the schematic, correcting any errors and verifying that any warnings are OK.

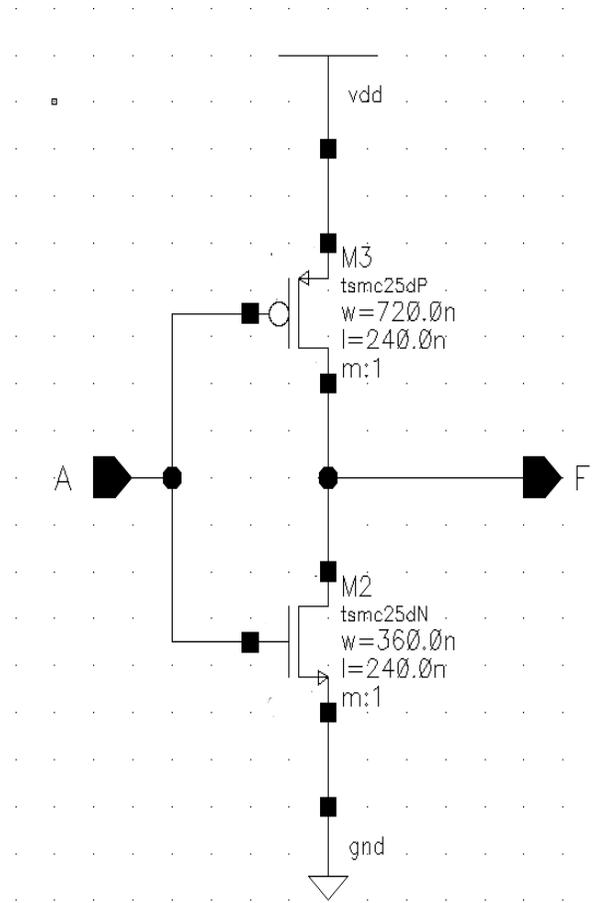


Figure 1: Inverter schematic.

7. We will now create a *symbol* for the inverter. This symbol is what will be shown in schematics which instantiate the inverter.
 - (a) From inside the Schematic Capture window, click on “Design → Create Cellview → From Cellview...”.
 - (b) In the pop-up, keep the default values and select OK. For reference, “From Cell View” should be set to “schematic” and the “To View Name” should be “symbol” and the “Tool / Data Type” should be “Composer-Symbol”.
 - (c) The symbol editor should open, showing an auto-generated symbol with the following parts:

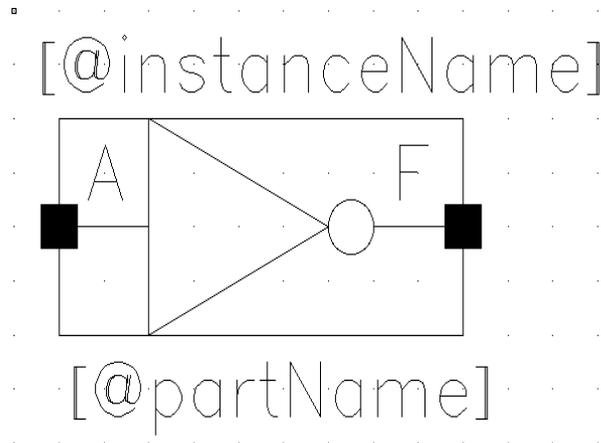


Figure 2: Inverter symbol.

Component	Description
Red rectangular box	Bounding box of the symbol. This box will be highlighted when an instantiated symbol is selected.
Two small filled red squares	Input and Output pins. These are the locations where wires will be connected.
A and F	Names for the pins. Note that each name is attached to a specific pin. When you drag the pin, the corresponding name will also move.
Tags <code>[@partName]</code> and <code>[@instanceName]</code>	These display the name of the part (e.g., INV) and the name of instance. There is no need to modify them, although you may move them around.
Green box and lines	This is the symbol that will appear when the part is instantiated.

- (d) The auto-generated symbol is just a simple rectangle, which does not look much like the standard inverter symbol. We must draw our own instead. Select and delete the green rectangle.
 - (e) Click on the “Line” button and draw the triangular inverter symbol.
 - (f) Add the “bubble” by selecting “Add→ Shape→ Circle”. In order to draw a small circle, you may need to decrease your “Snap-to” size. Do this by selecting “Options→ Display” and the setting the “Grid Controls Spacing” to 0.0625 and the “Grid Controls Snap Spacing” to 0.03125.
 - (g) Move the pins and labels and resize the bounding box so that your symbol looks like Figure 7.
 - (h) “Check and Save” the symbol. Correct any errors that are reported.
 - (i) Close the symbol editor.
8. Close the schematic capture window.
 9. The inverter can now be used in other designs just like the other parts with which you are familiar: Select “Add Instance”. In the “Component Browser”, set the library to “InvTutorial” and select the inverter.
 10. If you make changes the input/output pins of your schematic, you will need to update or recreate the symbol. You do not need to update the symbol, however, for changes which do not modify the inputs and outputs.