Using a Bus and designing custom devices in DesignWorks

To design a more complex circuit, the techniques of the previous labs are not sufficient. This tutorial demonstrates:

1. Designing a new device that is readable and manageable. The example used for this is a **2x1 MUX**.
2. Use of a bus.

1. Designing a new device:
   a. Download **mux_2.cct** from the course website.
   b. Open the **mux_2.cct**.
   c. Delete the switches and the probe from the **mux_2.cct**.
   d. To create port connectors: go to "*Options*" menu; choose "*Subcircuit*" ➔ "*New Port Connector*". A new window called "*New Port Connector*" appears, as shown in Figure 1.
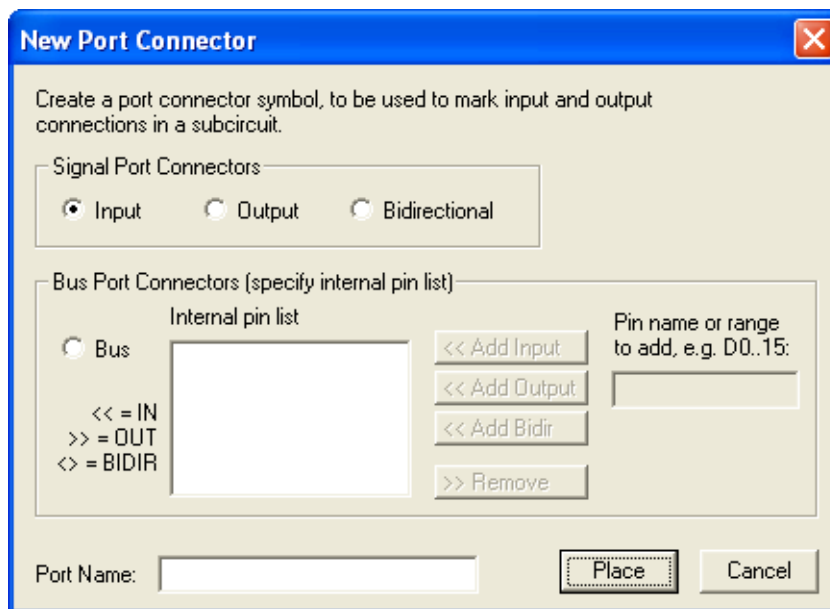

Figure 1

   e. Type "**S0**" at "*Port Name*" field. Click on "*Place*" buttonA symbol like this will appear:

   f. Click on the connector symbol to highlight it and right click on it again, choose "*Flip Horizontal*" to change the orientation of the connector and connect it to the wire named "**S0**". Repeat the process for "**d0**" and "**d1**".
   g. When creating the connector for output "**F**", choose the "*Output*" radio button. (Figure 2 below shows the "*New Port Connector*" window with output "**F**" selected.

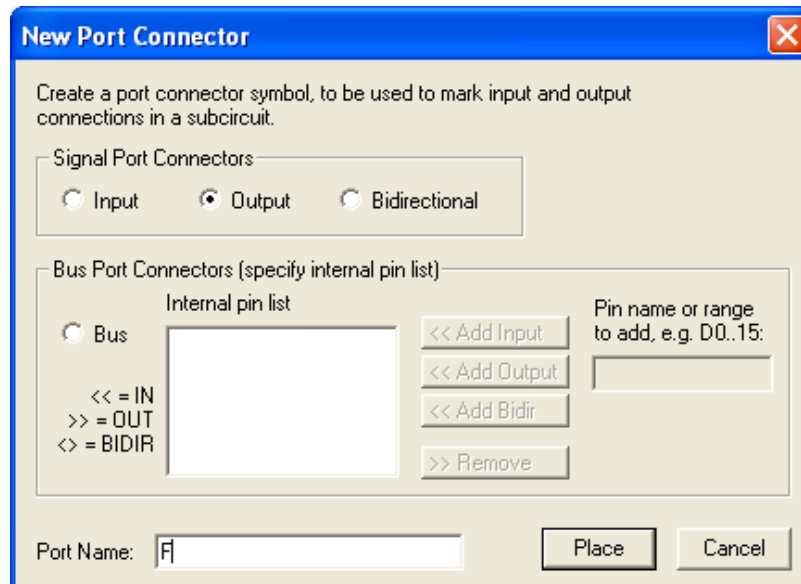Note the output connector symbol is different from the input connector symbol and appears slightly different:

Figure 2

h. Save the file.
i. Create a new file called **`tutorial.cct`** and design a **`2x1 MUX`** device. In **`tutorial.cct`**, go to the "*Options*" menu, choose "*Subcircuit*" ➔ "*Create Subcircuit Block*".
j. Click on the "*An existing circuit file from disk*" radio button. Click on "next". The Subcircuit Wizard window at this point is shown in Figure 3.
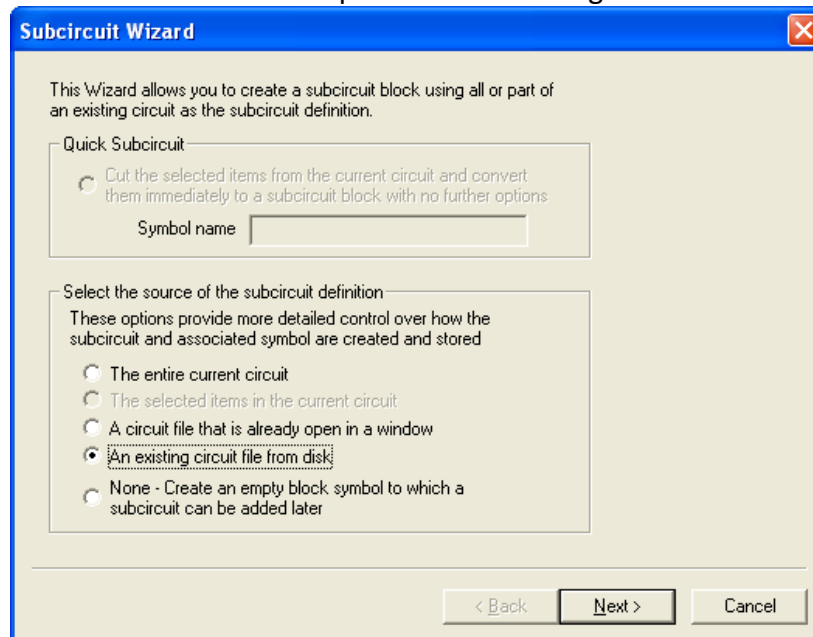

Figure 3

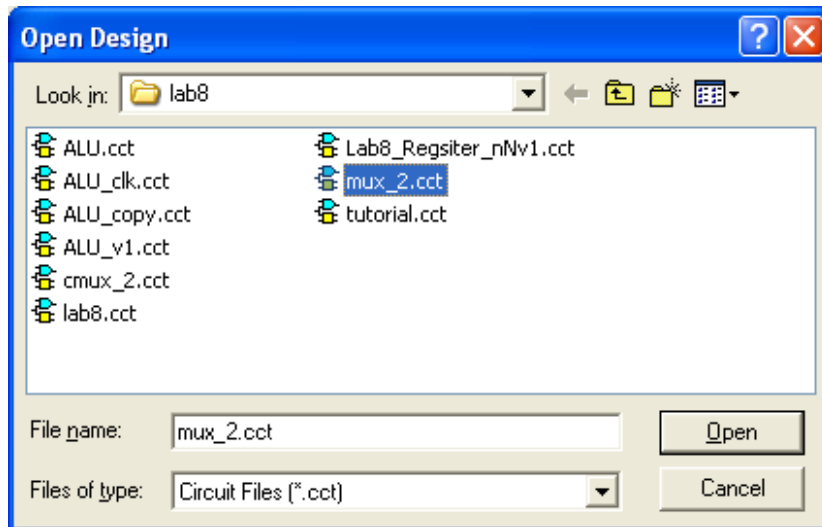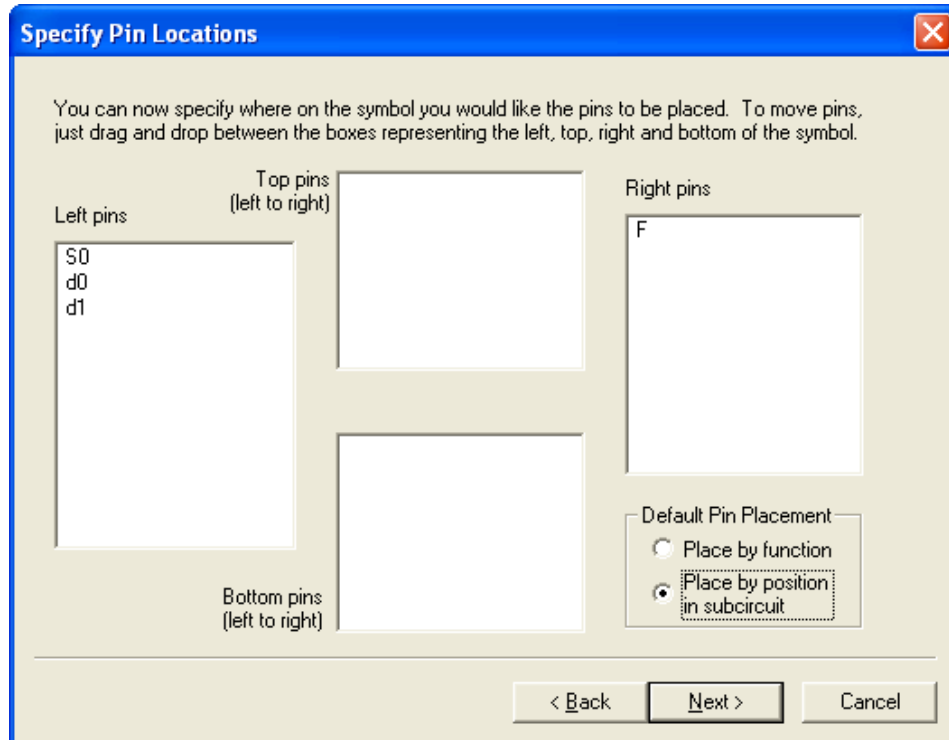k. Select "**`mux_2.cct`**". Click on the "*Open*" button (see Figures 4 and 5).

Figure 4

Figure 5

I. You may re-arrange the positions of the pins: Click on "**S0**" on the "*Left pins*" field and drag it to the "*Top Pins [left to right]*" field, as shown in Figure 6.
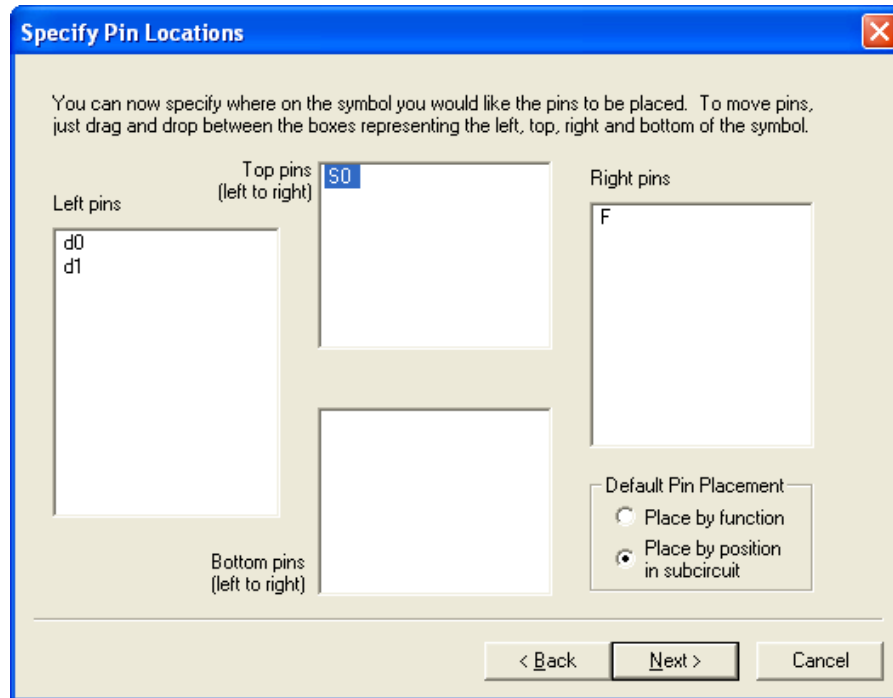
Figure 6

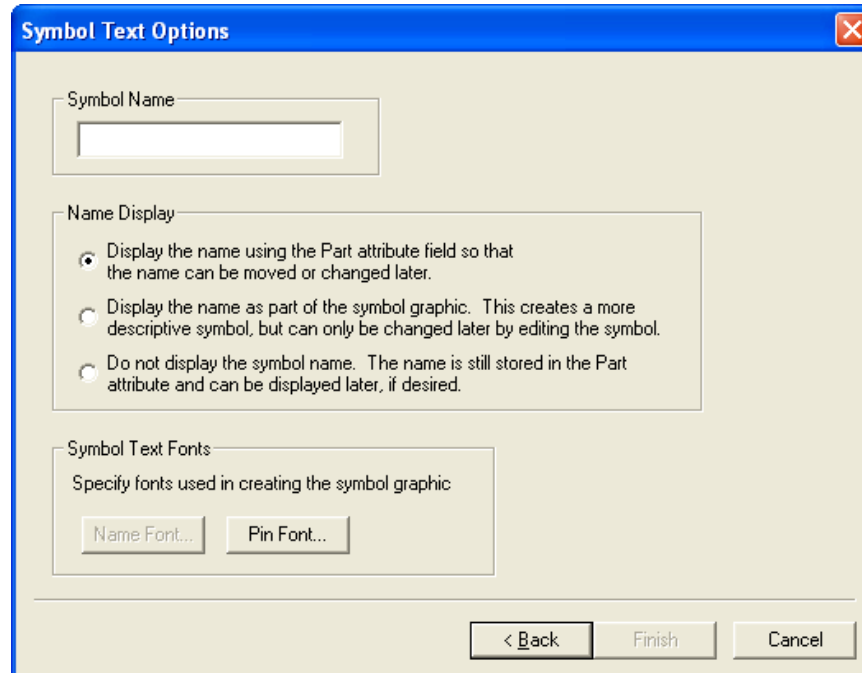**m.** Click on the "*Next*" button, to get the "*Symbol Text Options*" window shown in Figure 7.

Figure 7

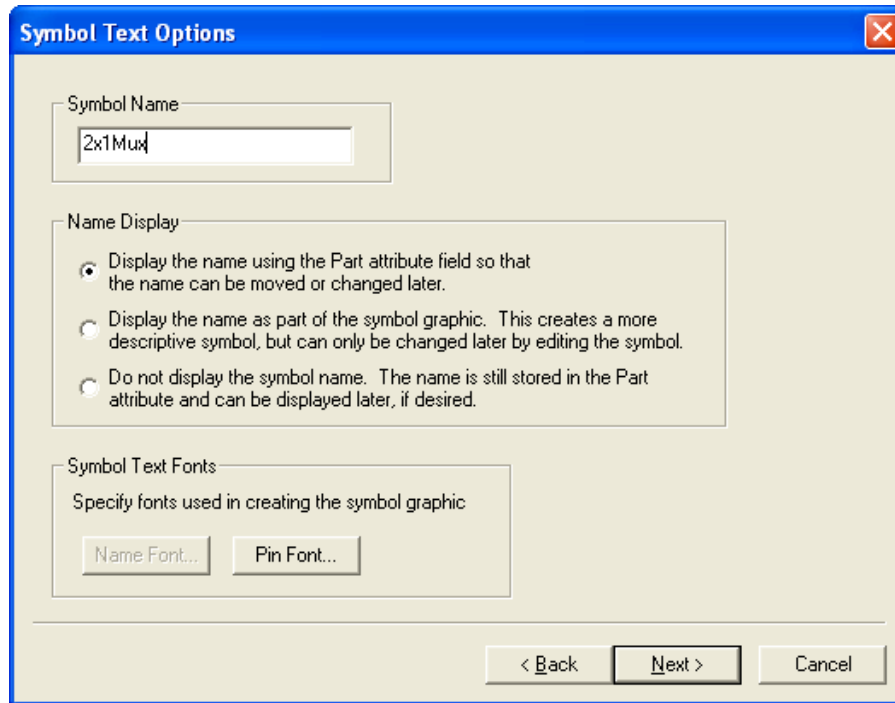**n.** Enter "**2x1Mux**" in the "*Symbol Name*" field, as in Figure 8.

Figure 8

o. Click on the "*Next*" button to get the "*Save the Part*" window of Figure 9.

Figure 9

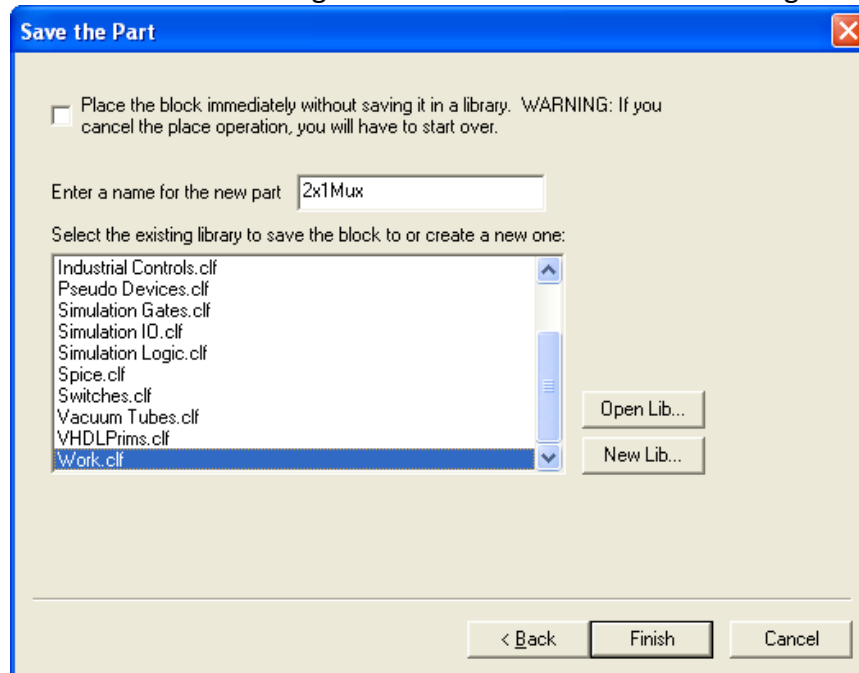**p.** First click on the "*New Lib…*" button, provide a name for your private part library, then "*Save*" and Click on "*Finish*". The **2x1 Mux** just created is now in the parts library and highlighted. You are back to the "**tutorial.cct**" file, with a screen as shown in Figure 10.
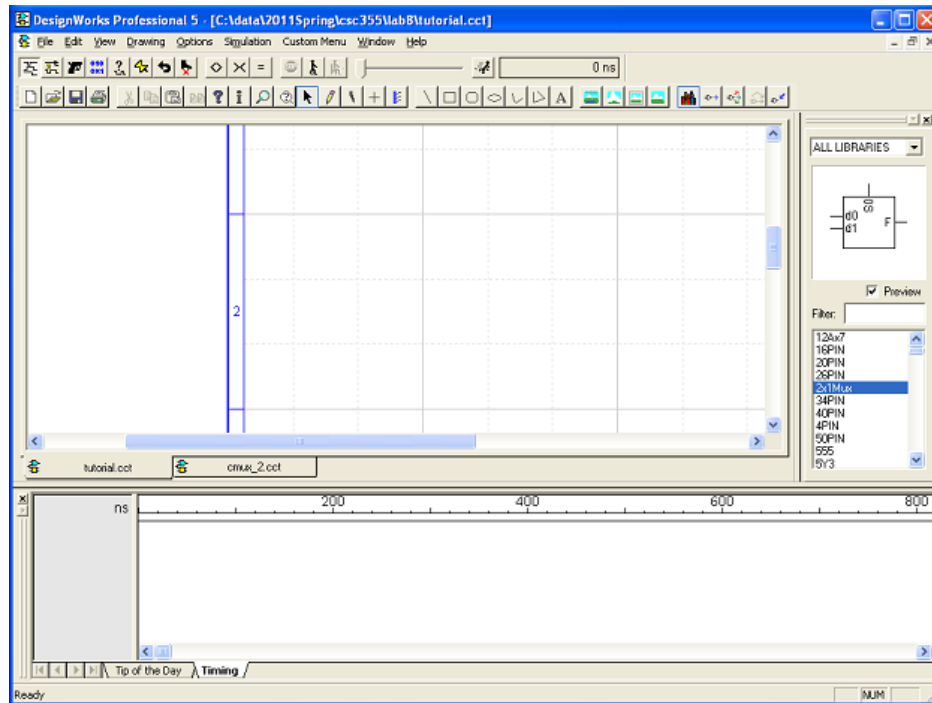
Figure 10

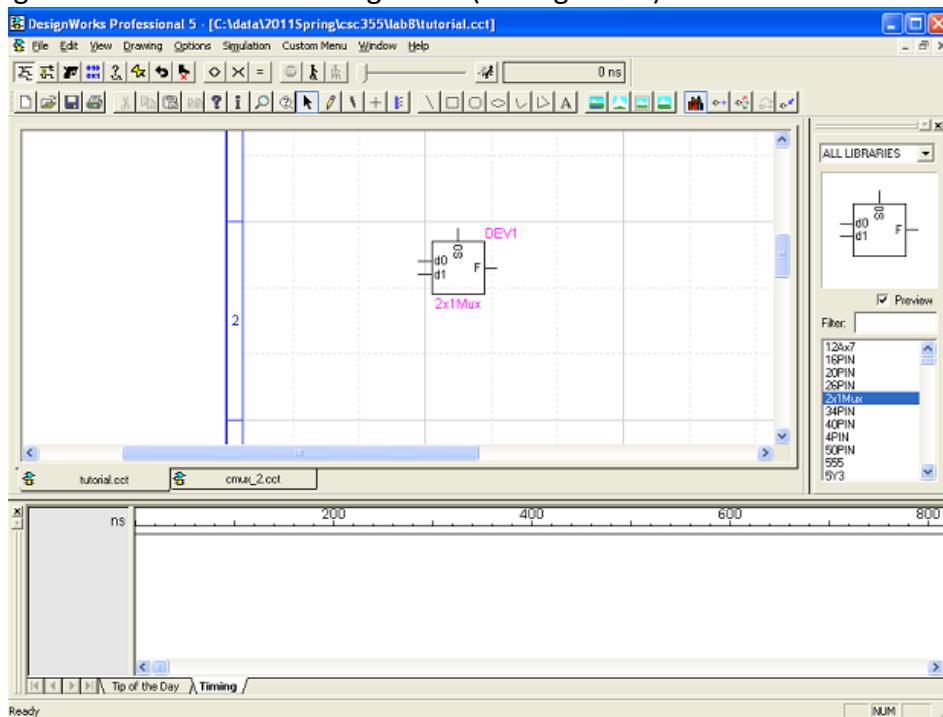q. Drag the **2x1 Mux** to the design area (see Figure 11).

Figure 11

2. Use of a bus. Recall that a 'bus' is simply a group of wires that function in parallel. The bus symbol looks like this:

    a. Click on the bus symbol and place it around the "**2x1 Mux**", as shown in Figure 12.
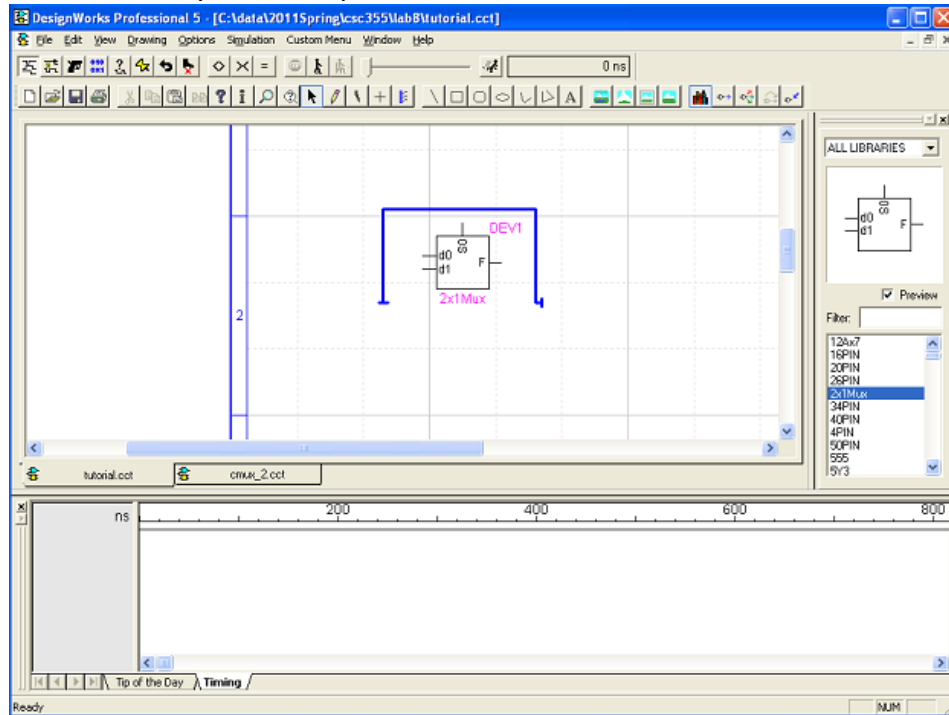

Figure 12

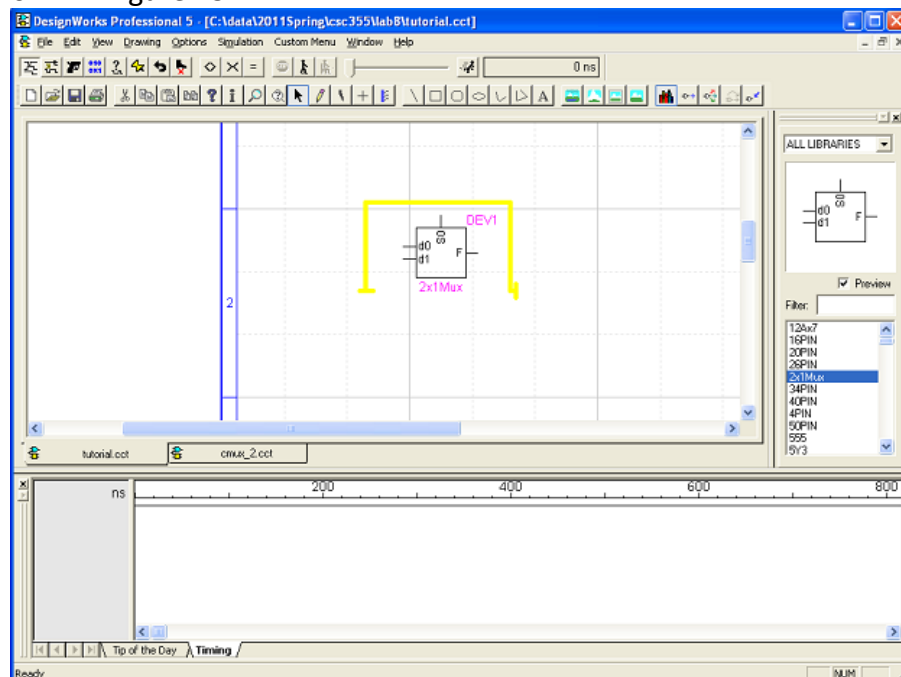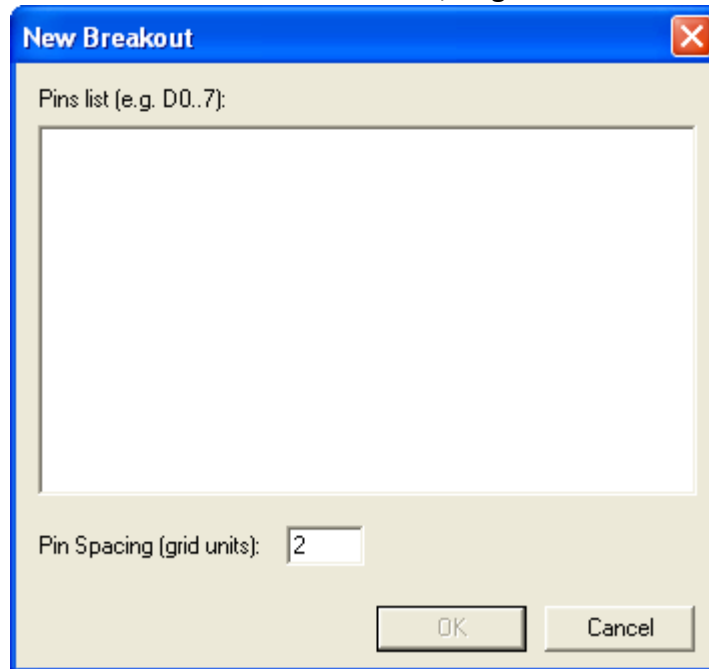   b. Click on the "*Pointer*" symbol (the arrow) and click on the bus, which becomes yellow, as shown in Figure 13.
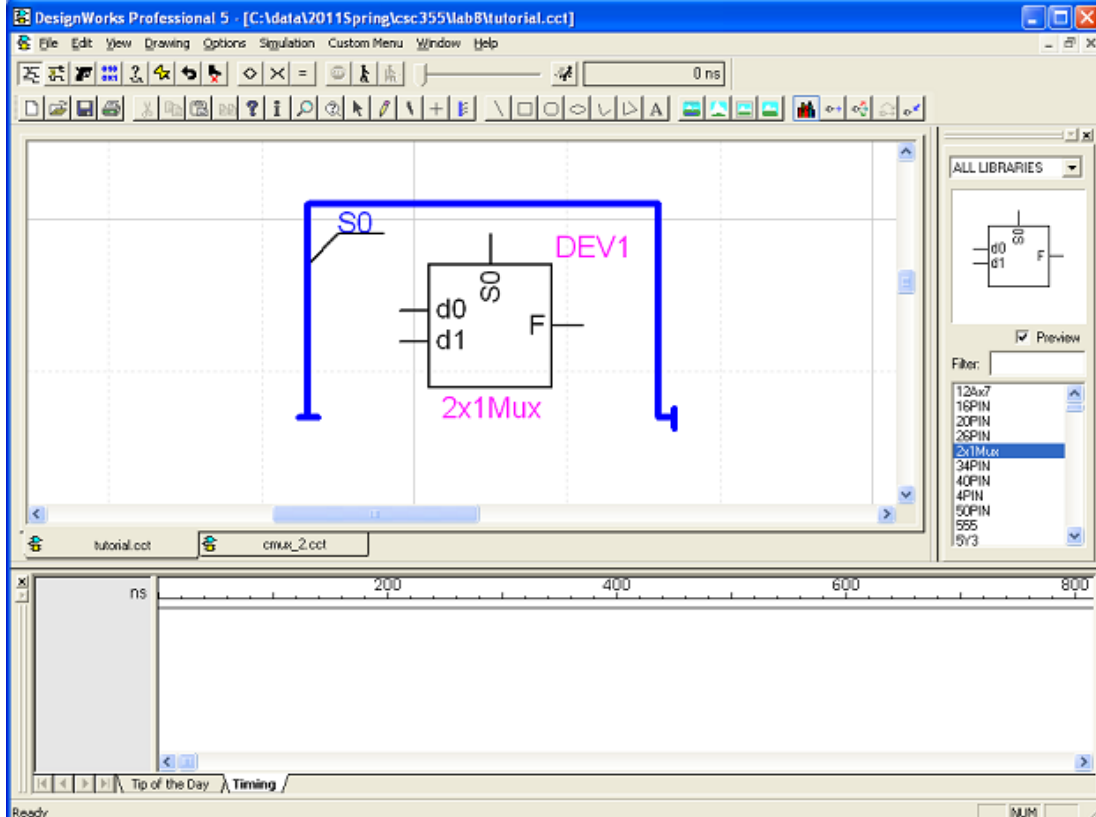
Figure 13

c. Right click on the bus and select "*Breakout…*", to get the window shown in Figure 14.

Figure 14

d. Enter "**S0**" and click on "Ok". Place the pin as shown in Figure 15.

Figure 15

e. Repeat the process for "**F**" and "**d0**" and "**d1**". Note that entering "**d0..1**" and the two pins will be created at once. Also, highlighting the pin allows the use of flip/rotate to get the desired orientation before it is placed on the bus. Figure 16 shows the pin "**F**" highlighted.
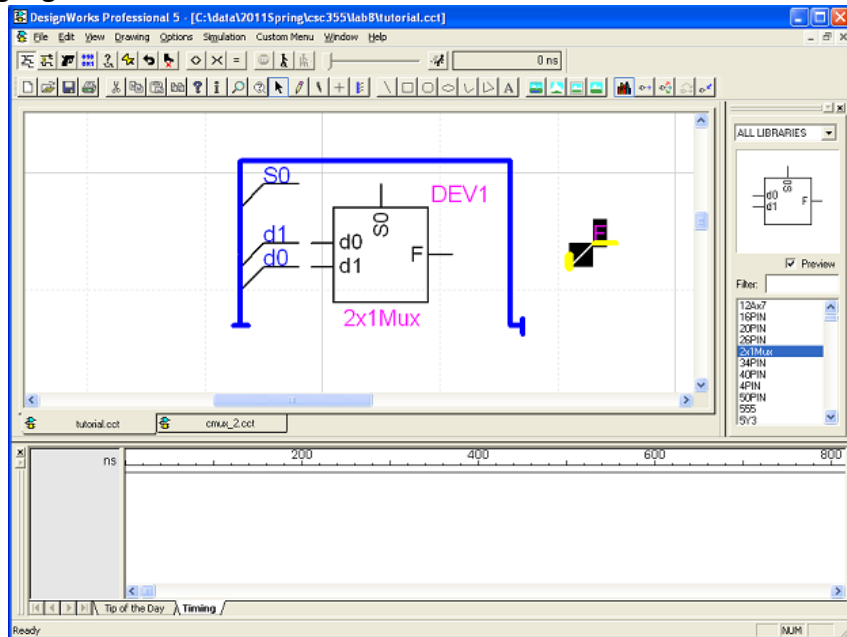

Figure 16

f. Right click on the highlighted pin and select "*Flip Horizontal*" to flip the pin. Note the pins "**d1**" and "**d0**" should be switched before being connected to the pins of the **2x1 Mux**. Add Logic Switches and a probe to demonstrate circuit functionality. The final circuit is shown in Figure 17.
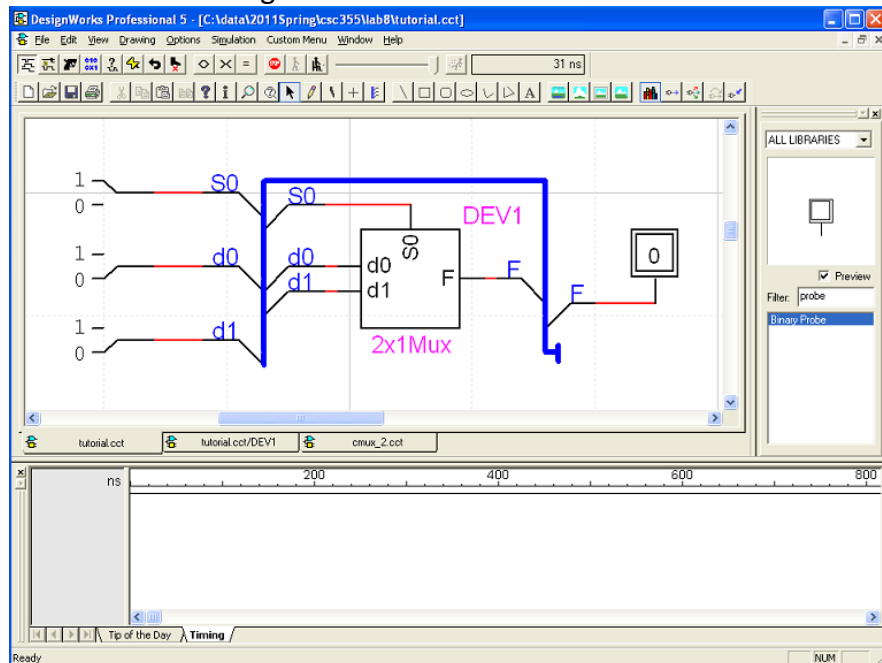

Figure 17