

# SWITCH COUNTER IMPLEMENTATION

ECE 221

Fall 2002

## I. OBJECTIVE

The objective of this experiment is to implement the switch counter you developed in the first design project using small scale integration (SSI) circuits. The main goal when using SSI circuits is to create a design that uses the minimum number integrated circuits (ICs).

## II. BACKGROUND

Once a digital design has been synthesized into a minimized logic expression there are a number of hardware circuits that can be created that implement the expression. Typically, logic designers try to optimize the implementation by minimizing the number of ICs used or the number of wires that must be connected. Standard SSI ICs (AND, OR, INVERTERS, etc.) are easy to implement but may not always give you a minimized implementation. Typically there are a number of ICs needed and a number of wires that must be connected. SSI implementations may be minimized further if universal gates (NANDs, NORs) are utilized because only one gate type is used and therefore possibly fewer ICs are needed.

## III. PRE-LAB

In the lab you will implement your minimized design of the switch counter from design project #1. To assist in wiring up your design, you will be introduced to another Mentor tool called *Package* which will automatically place pin numbers on your schematic.

*Package* is a part of the Printed Circuit Board (PCB) toolset in Mentor. These tools help designers translate a schematic into a set of documents that can be used by manufacturing to build a prototype system. Follow the steps below access and automatically generate pin numbers for your design.

1. Go to the directory where your minimized design schematic is located and type in:

```
package switch_count
```

This will start the package software. As the software is starting you will see a number of warnings scroll in your command window (it might take a minute). Just disregard these. When the **Report - Startup** window appears just close it.

2. The main window shows the parts in your schematic along with the quantity of each part. On the palette, select the **Build** option. When the Build window appear, put the following location in the **Default Component Geometry** box.

```
/usr/local/mentor/pkgs/parts_lib/bpl_rls_lib/pcb_geoms
```

These geometries tell the tool how many parts are on each IC and what the pin numbers are for the inputs and outputs. Hit OK to start the build process. Once the build has finished another report window will appear. Scroll down to the bottom of the window and you should see the following statement:

```
No errors found in pins or used mapping files
```

If you don't see that message, try working through this step again and then if it fails again contact the instructor. Close the Report window.

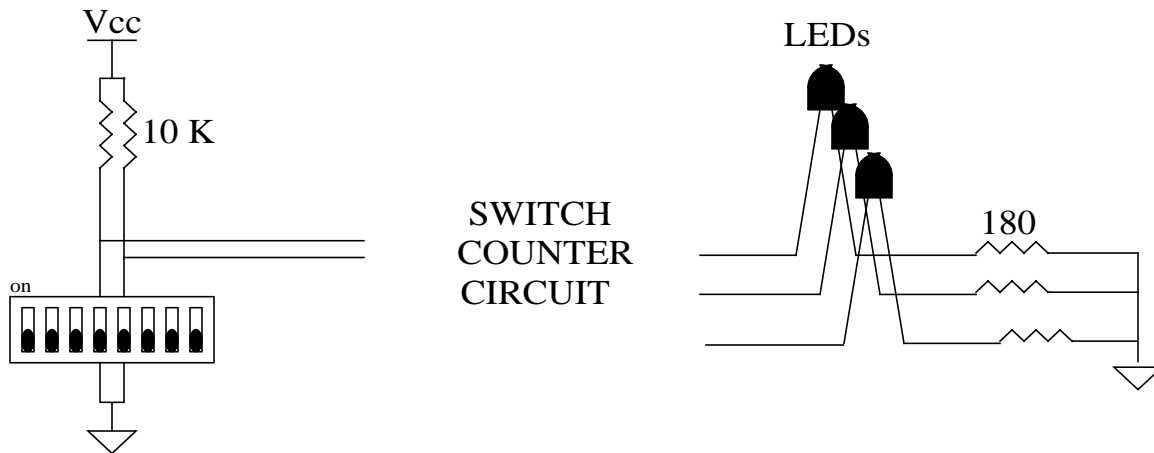
3. Select the **Save->Design All** option under the **File** menu. When the Save window appears, select **Yes** for the **Back Annotate PCB Design Viewpoint** and then hit OK. This will create a viewpoint of your design schematic that has the pin numbers for all of the ICs on it. A *viewpoint* in Mentor is simply a

representation of your schematic that has special properties placed on it so it can be used with other Mentor tools. The viewpoint this tool creates is named **pcb\_design\_vpt** and is found under your **switch\_count** directory. Close the report window that appears and then exit the Package tool by selecting **Session** on the palette followed by **Close Session** on the new palette that appears.

- To open the viewpoint, first start up Design Architect. When the main Design Architect window appears, select **Set Viewpoint** on the palette. When the Set Viewpoint window appears, type in **switch\_count** as the **Component Name** and if it is not already there, type **pcb\_design\_vpt** for the **Viewpoint Name**. Hit **OK** and then when the Open Design Sheet window appears hit **OK** again. You should see your schematic and when you zoom in on a part, the IC number (U#) and the pin numbers for that IC will be shown in red. Print out the schematic and bring it to lab.

#### IV. PROCEDURE

- Create the minimized implementation that you designed. Use 2 switches as your inputs (X, Y) to your circuit and 3 LEDs for outputs (ZERO, ONE, TWO). Connect your switches and LEDs as shown below:



When the switch is open (off) the value going into the circuit is a logical '1'. When the switch is closed, the value going in is a logical '0'. Check all four combinations to test the circuit's operation (put them in your lab notebook). Have the instructor or a lab assistant sign off on your working design.

- You may need to find out pin numbers for the power and ground on the 74LS02. These can be found in the Texas Instruments on-line databook which can be accessed from the **Interesting Links** page on the course web page. Log into the computer at your station and when the TI site comes up, put the part number (example - 74LS02) in the box and do the search. Then go to the datasheet section for that part and open the PDF formatted file.

#### HINTS ON TROUBLESHOOTING YOUR DESIGN

- Check to make sure each IC has power and ground connected properly.
- Use literal analysis to work through your design and verify each pin has the correct value.

#### V. SUMMARY AND ANALYSIS

In a paragraph summarize today's lab and then answer the following questions.

- Why is it important to try and minimize the number of ICs used?
- How many ICs would be needed to implement your design if you used AND, OR, and INVERTER ICs (7408 - Quad 2-input AND, 7404 - Hex inverter, 7432 - Quad 2-input OR)