

Part IA Computer Science Tripos

Hardware Practical Classes

Year: 2013 – 2014

Dr. I. J. Wassell,

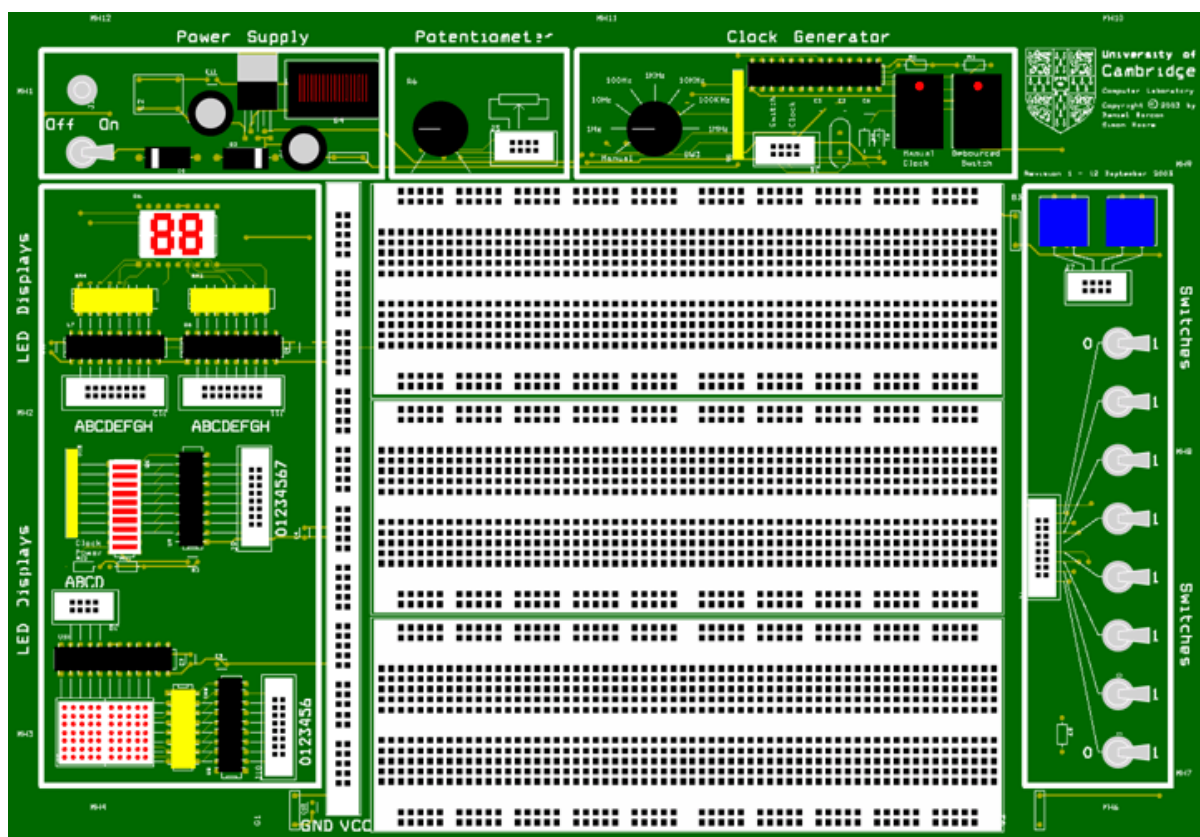
Mr. N. Batterham.

Digital Hardware Labs - Introduction

Many materials are available on which to build prototype circuits. The material chosen will depend on the required life and use of the circuit. For laboratory use or other short-term applications where a permanent circuit is not needed and the operating frequency is below 10 MHz, solderless breadboards such as that used in the prototyping box shown below are suitable.

The prototyping box comprises:

- 3 breadboards on which your circuits will be constructed
- A 5 volt power supply required for the logic integrated circuits (chips)
- A variable frequency clock which operates in either manual or free running mode
- 8 switches, which can provide inputs to your circuit
- 2 push button switches (not debounced)
- A debounced push button switch
- Two 7-segment light emitting diode (LED) display
- A 10x7 LED array
- 8 user input LEDs
- A potentiometer



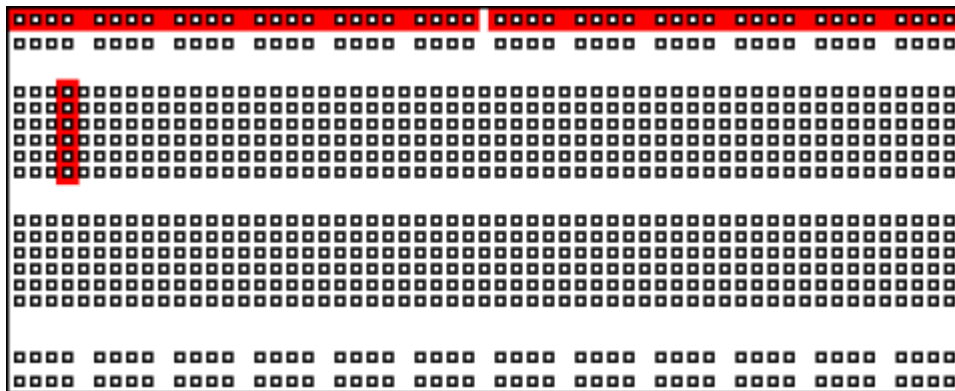
There are three banks of LEDs on the left hand side of the board for signal output:

- At the top there is a pair of 7-segment LED displays which are connected via PALs. The PALs have been configured to do binary to 7-segment hexadecimal character conversion. Inputs A to D provide the binary inputs.
- In the middle there is an LED bar graph. 8 of the LEDs are connected via a buffer. The other two indicate when the power is on and the state of the clock.

- At the bottom you will find an LED matrix. The column decoding is provided by a 4-to-16 line decoder (inputs A to D) and the row value is supplied via a buffer (inputs 0 to 6).

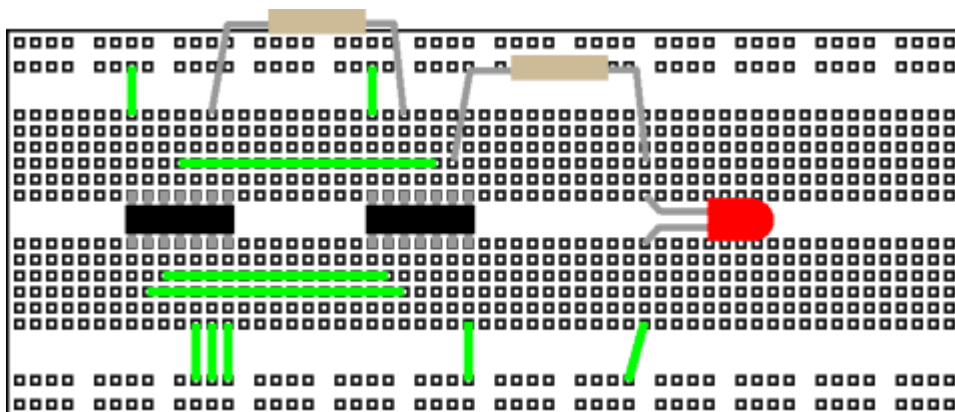
Using the Breadboard

A breadboard has many strips of metal which run underneath the board. The metal strips are laid out as shown in red below, horizontally along the two lines at the top and bottom, and vertically to join six holes together in the central section.



These strips connect the holes on the top of the board. This makes it easy to connect components together to build circuits. To use the breadboard, the legs of components are placed in the holes (the sockets). The holes are made so that they will hold the component in place. Each hole is connected to one of the metal strips running underneath the board.

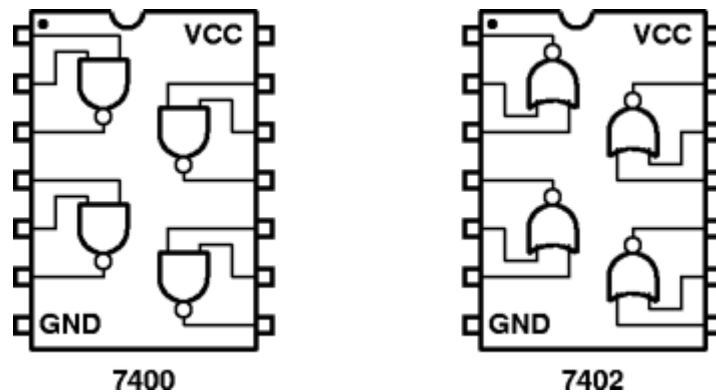
The long top and bottom row of holes are usually used for power supply connections. The rest of the circuit is built by placing components and connecting them together with jumper wires. Chips can be placed in the middle of the board so that half the legs are on one side of the middle line and half are on the other side. A completed circuit might look like the following.



Careful planning of the circuit layout simplifies wiring, minimizes errors and makes debugging easier. Try to arrange the circuit for a logical signal flow, usually inputs on the left and outputs on the right. This helps anyone looking at the board to find easily sections of the circuit and trace signals through it. Wherever possible, all ICs should be pointed in the same direction to reduce the chance of one being put in backwards. This also makes it easier to keep track of pin numbers when wiring and debugging. Colour coded wiring is an easy way to reduce wiring errors and aid the process. If possible, build and test one

section of the circuit at a time to simplify debugging before connecting the sections together. Keep connecting wires as short as possible and route them around ICs so that a defective IC can be replaced without removing and often incorrectly replacing the wires. While all this may seem laborious, the time spent in careful construction and checking is well repaid by having more circuits work first time.

The pin outs of all the ICs used in the hardware labs are available as a booklet in the lab. Never take the pin out of an IC for granted! Below are two similar chips: the 7400 and 7402, are quad NAND and quad NOR gates respectively. Note the different pin out.



More details concerning the chips you will be using are available in the lab booklet and also on the course web page.

Common Ticking Criteria

All of the workshops should be written up in full in the style of a laboratory log book. You may use an actual log book or else loose sheets with page numbers. From your notes it should be possible for somebody else to be able to reproduce your work. This is good scientific practise. When an exercise is complete and written up, including answers to the questions, the final page will be signed by a demonstrator and a tick entered in the tick sheet.

We recommend that you draw out circuits in your log book before wiring them up. Put pin numbers for the connections to each chip.

VERY IMPORTANT: you need to hand in this assessed exercise as part of your portfolio of work at the end of the year (see the Head of Department's Notice), SO YOU MUST KEEP YOUR WORK!

Appendix 1 – Resistor Colour Code

Resistors are normally coded with coloured bands to enable quick identification of their value. The colour code specifies the value of the resistor, in ohms, and the maximum deviation from the stated value (the tolerance). Most resistors have a manufacturer's tolerance with is adequate for most electronic applications.

Interpreting the Colour Code

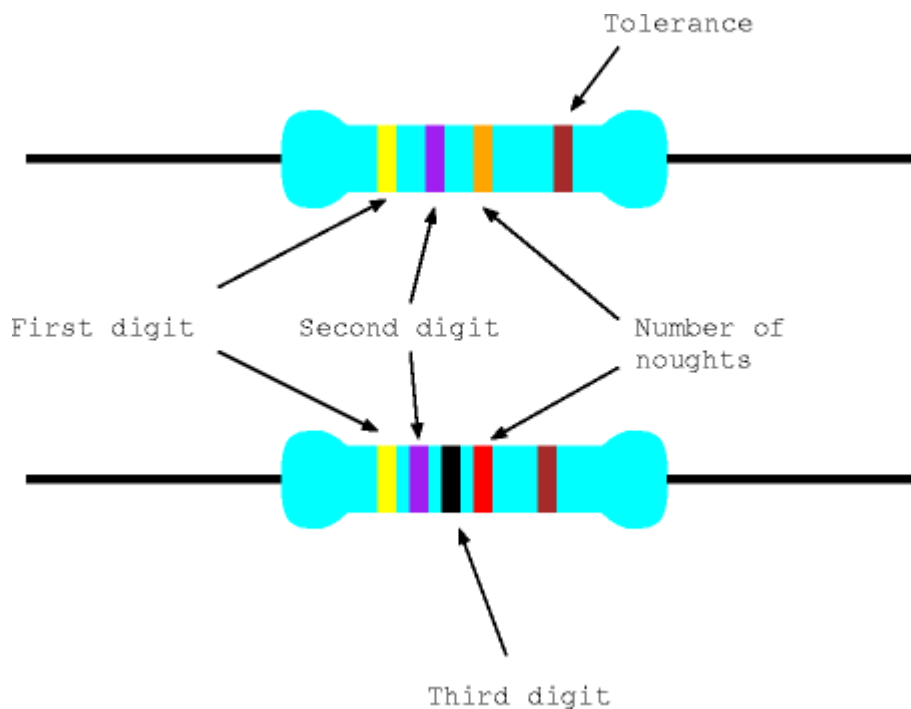
The four colour code bands are at one end of the component. Counting from the end, the first three (or sometimes four) bands give the resistance value and the last the tolerance.

The significance of the colours is shown in the table below:

First 3 (or 4) bands		Tolerance band	
Black	0	Brown	1%
Brown	1	Red	2%
Red	2	Gold	5%
Orange	3	Silver	10%
Yellow	4	No band	20%
Green	5		
Blue	6		
Violet	7		
Grey	8		
White	9		

Reading the value (three band)

The first two bands are used to specify the first two digits of the resistor's value. The third gives the number of noughts to be added. Thus in this example, yellow and violet give the first two digits as 4 and 7 and the number of zeros is 3 (orange) giving a value of 47000 ohms.



Four Band Resistors

More modern resistors use four bands to specify the resistance, the first three giving the three most significant digits and the fourth the number of zeros. Yellow, violet, black, red decodes as 4, 7, 0, 00 i.e., 47000 ohms.

© 2011 DJ Greaves, DL Gordon, SW Moore and IJ Wassell

Page last updated on 11-Aug-2011

Workshop One – Electronic Die

Introduction

The purpose of this lab is to build a simple electronic die using light emitting diodes (LEDs) as the display device. Digital logic chips will be used to implement the required functionality. You will construct the die on a prototyping breadboard that is housed in the 1A workshop prototyping box. Before you start wiring up the circuits, make sure you understand how the holes in the breadboard are connected and also how the pins are numbered on the logic chips – see the description in Appendix A.

Equipment and Components

1A Workshop Prototyping Box

Connecting wire

Red LEDs (7)

220 Ω resistors (7)

10k Ω resistor

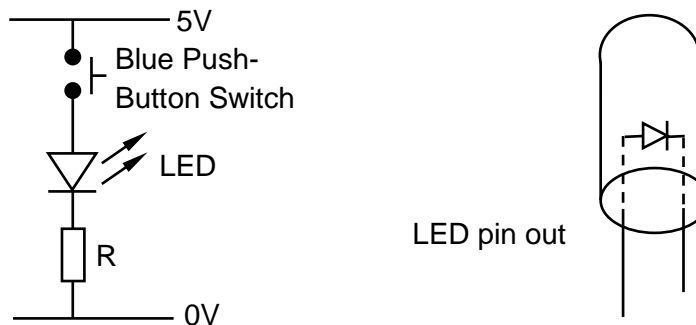
74HC00 – Quad 2 input NAND gates (2)

74HC193 – Synchronous 4 bit up/down counter (1)

Step 1 – Wiring up one LED

First we need to see how to turn on one LED; note there will be seven in the completed die. If a LED is connected directly across the 5V power supply used to power the digital logic chips, it may well draw enough current to destroy itself. To prevent this happening we need to connect a current limiting resistor in series with the LED. The required circuit is below, followed by the calculation to select an appropriate value for the current limiting resistor, R.

Please make sure you use the correct value resistor, otherwise you may destroy the LED.



When the LED is lit correctly, the voltage across it will be about 1.6V and the current flowing through it should be about 15mA. To work out the appropriate value for R we need to use Ohm's law, i.e., $V = IR$. In this case the voltage across R is (5 - 1.6)V, so,

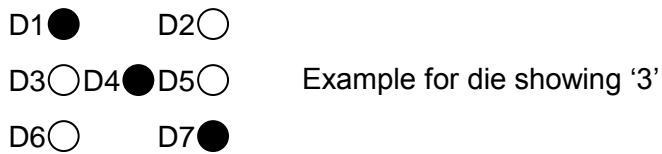
$$R = V/I = (5 - 1.6)/0.015 = 226\Omega.$$

Note that the nearest available value (known as the preferred value) is 220 Ω .

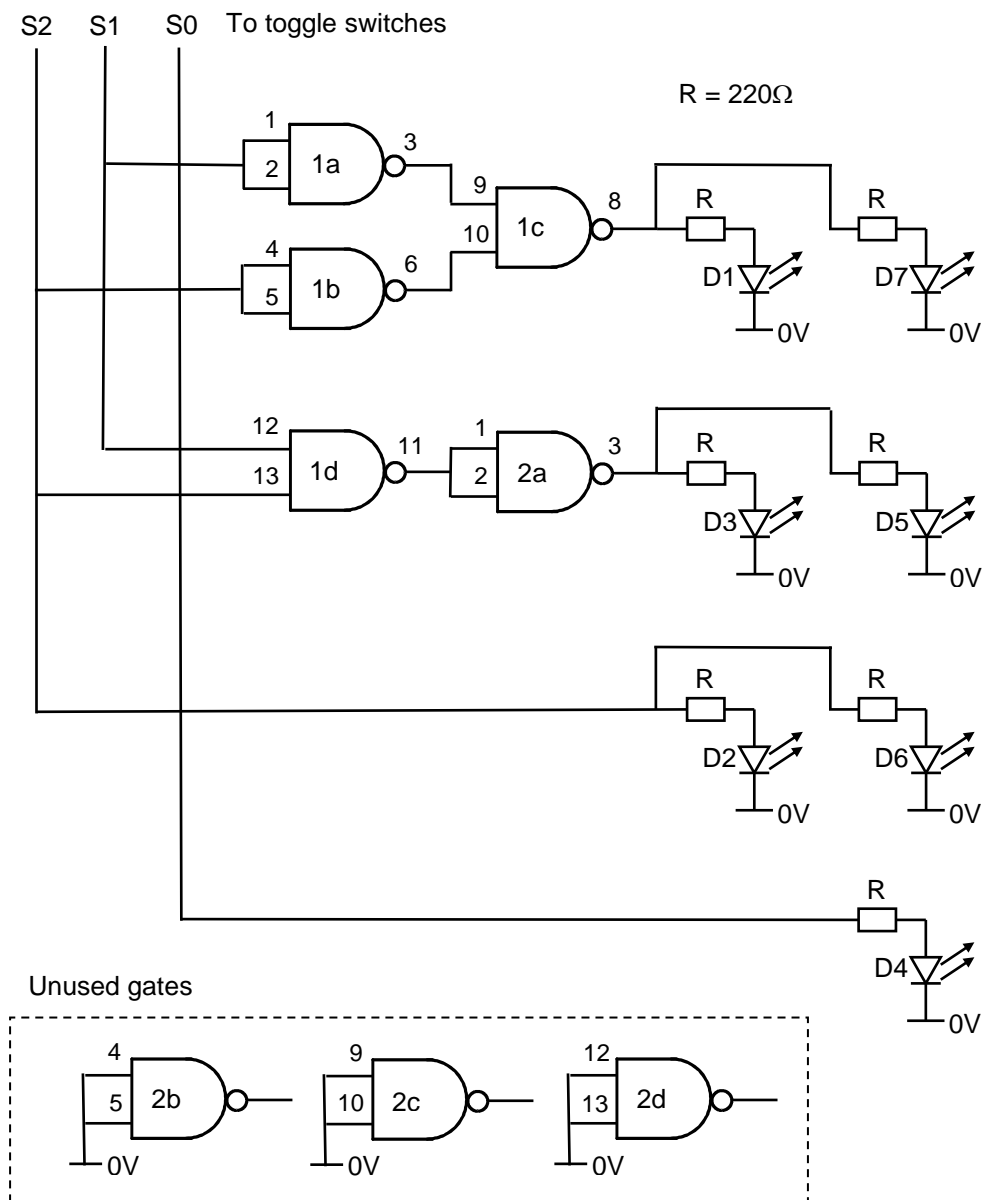
Task: Construct the above circuit and check that the LED lights when the button is pressed.

Step 2 – Binary to Die Decoder

To construct the die you will need to arrange the 7 LEDs in the following pattern to emulate the spots on a die.

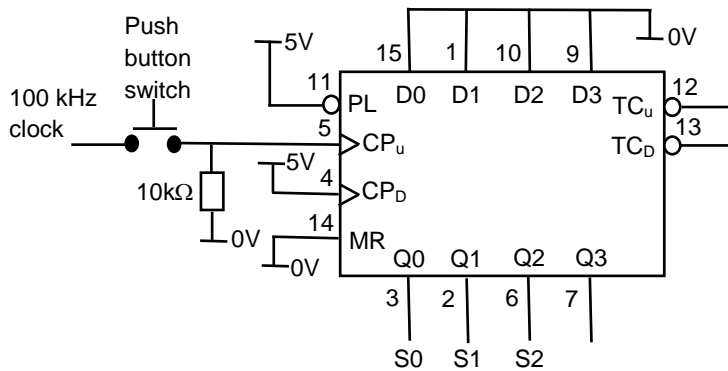


Task: Arrange the 7 LEDs on your breadboard to make the above pattern; making sure that each LED has a 220Ω resistor in series with it. Then construct the following circuit using a 74HC00 quad 2-input NAND chip, i.e., each chip contains 4 NAND gates. Test the circuit by connecting the 3 inputs to the toggle switches in the prototyping box. Remember to connect V_{cc} (pin 14) to 5V and GND (pin 7) to 0V. Also unused gate inputs must be connected to 0V to prevent unstable operation of the chips.



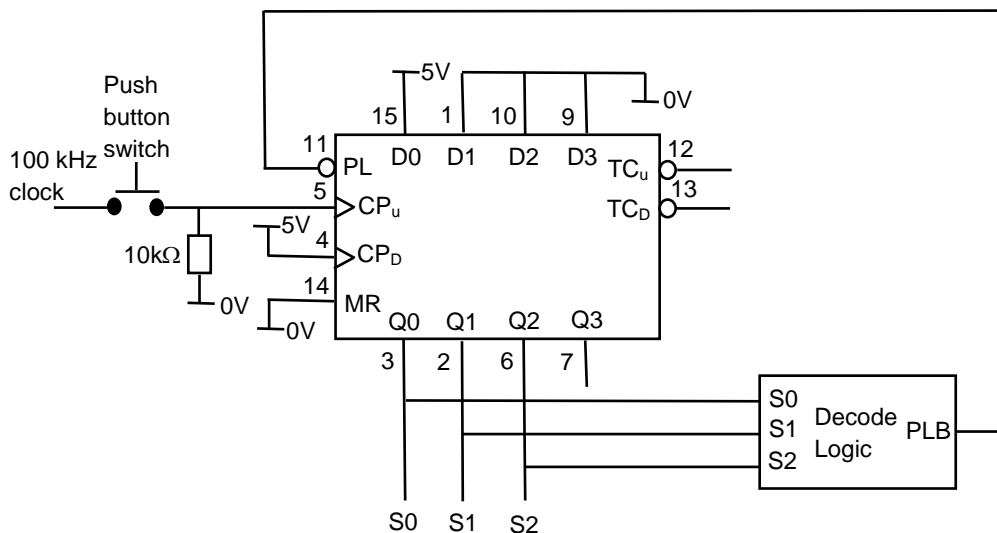
Step 3 – Make the die spin

Task: Use a 74HC193 counter chip as a replacement for the toggle switches. Wire up the circuit as shown below. Set the clock generator in the prototyping box to 100kHz. Note that to make the counter pausable, the 100 kHz clock is connected through one of the blue push button switches. Remember to disconnect S0, S1, S2 from the toggle switches and to connect them to the appropriate outputs from the counter, namely Q0, Q1, Q2. Also remember to connect V_{cc} (pin 16) to 5V and GND (pin 8) to 0V. You should now have a usable 8-sided die!

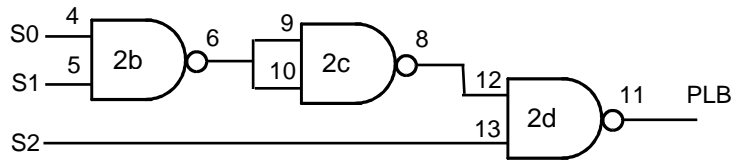


Optional Bonus Step

Task: Modify the previous circuit as shown below so that the die counts from 1 through 6. The decode logic detects the count of '7' and forces the counter to parallel load with the count of '1'.



The decode logic has the following circuit diagram. Note that it uses the previously unused NAND gates from the original die design – remember to disconnect their inputs from 0V!



Assessment

Ticking criteria: Demonstrate that your 8-sided die works and answer the following questions.

Once your work has met the Common Ticking Criteria (see Introduction), get your work ticked by an assessor. Remember that you need to hand-in this assessment exercise as part of your portfolio of work (see Head of Department's notice).

In future workshops, it will be often be convenient to monitor the logical value of a signal using LEDs. Rather than wiring up discrete LEDs with serial resistors as we have done in this workshop, it will be more convenient to use the bank of 8 pre-wired LEDs which is available on the left hand side of the prototyping box.

Questions

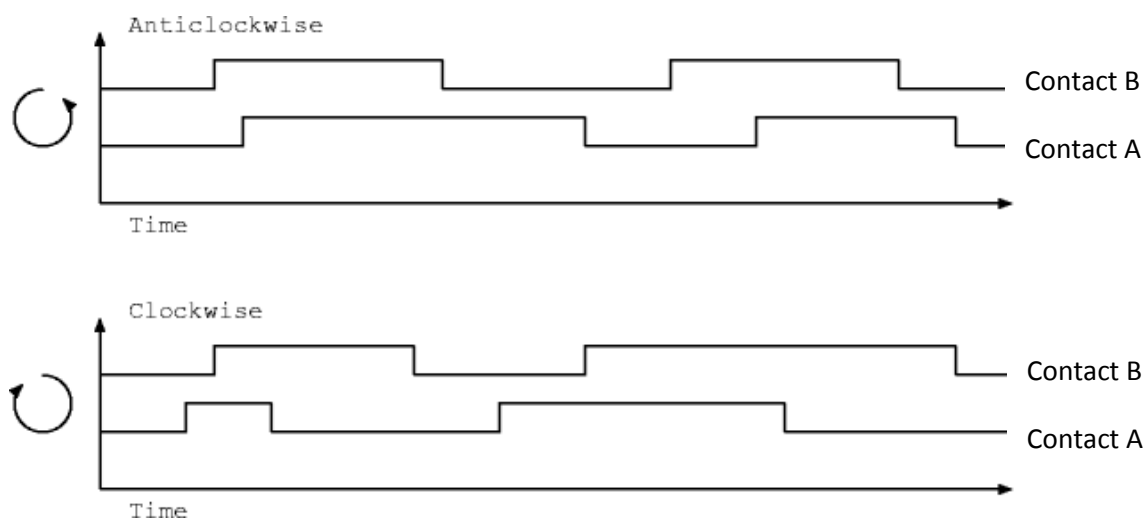
1. What die patterns are displayed for binary inputs 000 and 111?
2. How much current can each NAND gate on the 74HC00 drive?
3. How would you arrange an LED (and resistor) so that it turns on when the output of a NAND gate is low?
4. How could this reduce the number of gates you have used?

Workshop Two – Shaft Position Encoder

Introduction

Some industrial automation applications require control systems which know the rotational position of a shaft. Similar devices are also used for digital volume controls, etc., on domestic appliances. A shaft encoder is used to record the rotation typically in the form of a digital Gray code. In this workshop you need to build a decoder for such an encoder.

The shaft encoder you will be using produces a 2-bit Gray code. Such encoders are usually optical (e.g., those commonly used in a mouse), but in this case, we will be using a mechanical encoder. A disk is connected to the shaft which rotates with it. Electrical contacts are made with this disk to produce one of two Gray code sequences shown below depending upon the direction of rotation.



A shaft decoder module is required to convert this 2-bit Gray code (B,A) into a 4-bit position count. The 4-bit position count should be *incremented* every time the input from the encoder changes owing to a *clockwise* rotation (e.g., from 00 to 01, or from 10 to 00). Similarly, the 4-bit position count should be *decremented* every time the input from the encoder changes owing to an *anticlockwise* rotation (e.g., from 00 to 10, or from 01 to 00). Firstly, a Finite State Machine (FSM) must be designed to generate the clockwise and anticlockwise rotation detection signals. These signals are then used to control a separate 4-bit binary up/down counter that generates the current position (i.e., the counter output value) that is then displayed on one of the 7-segment LED displays (with built-in hexadecimal to 7-segment display decoder) housed in the prototype box.

Components

- 1a prototyping box
- rotary shaft encoder and two 4.7k Ω pull-down resistors
- a PAL - GAL16V8
- 74HC193 - 4-bit up/down counter

Note: The recommended circuit for the shaft encoder is given in the lab booklet - remember to use the 4.7k Ω pull-down resistors to turn the switch open state into a firm logic value. Also note that the encoder provided steps through **all four** of the output combinations for each 'click' position.

Step 1 - shaft decoder to generate clockwise/anticlockwise signals

Design task 1: Draw a state diagram to decode this movement. Generate two independent outputs, one that goes high for one clock cycle when the shaft rotates one output combination change clockwise, namely CW, and the other for one output combination change anticlockwise, namely AW. Assume 4 states, i.e., 00, 01, 11 and 10, and **make the next state equal to the current input** values B and A. For example, assume we are in state 00 and the shaft rotates one output combination *anticlockwise* giving current input values, B=1, A=0, the next state will be 10 and the outputs are AW=1 and CW=0. Now complete the state diagram for all 4 states and for all possible inputs per state (3 per state).

Design task 2: From the state diagram, create a state transition table. Note that this table will have columns for the Current State, the Next State (remember that is the same as the current inputs) and for the clockwise (CW) and anticlockwise (AW) outputs. Now determine expressions for the AW and CW outputs. Remember that it is possible to use don't care conditions to simplify the expressions for AW and CW. For example, one don't care condition for both AW and CW is for the 00 to 11 state transition that we know does not ever occur. We now wish to implement a synchronous (clocked from an independent free-running clock) FSM inside a PAL.

The PAL has 8 possible output pins labelled P19 to P12 and 8 inputs labelled P2 to P9. Connect P2 to Contact A and P3 to Contact B.

This exercise can be completed by designing a four-state FSM, where the D-type flip-flops (FFs) having outputs labelled P16 and P17 hold the states. Your equations should look something like this:

```
P19 = p2
P18 = p3
P17 = p19
P16 = p18
P15 = <Some function of p16, p17, p18 and p19>
P14 = <Some function of p16, p17, p18 and p19>
```

This means that FF output P19 is a registered version of its input P2 (that is connected to A) and FF output P18 is a registered version of its input P3 (that is connected to B). The state of the FSM is held in the FFs having outputs P16 and P17 and these are registered versions of the FF outputs P18 and P19 respectively. Effectively, the Next State (that we have made equivalent to the current inputs) is FF outputs P18 and P19, and the Current State are FF outputs P16 and P17. The outputs labelled P15 and P14 are the Clockwise (CW) and Anticlockwise (AW) signals respectively and are therefore some function of the state (P16 and P17) and the registered inputs (P18 and P19).

Very Important Point Concerning the 74HC193 4-bit up/down Counter

Fundamentally, the CW and AW signals are used to control whether the counter increments or decrements its current count value respectively, i.e., whether it up or down counts. The up/down counter has 2 separate clock inputs, one for up counting (CPU) and one for down counting (CPD). For example to make the counter increment, then a rising edge has to be applied to CPU input (essentially a pulse signal created by changing the CPU input signal from a binary '1' to a binary '0' and then back again), while the CPD input is held at binary '1'. The reverse situation applies concerning the clock inputs to make the counter decrement

its value. Consequently, when generating the P15 and P14 clockwise and anticlockwise signals, we actually require the complement of the CW and AW signals (as defined previously) to provide the CPU and CPD clock signals respectively. Consequently if we are using K-maps to determine the simplified Boolean expressions for P15 and P14, we should group the '0's and any don't care states (rather than grouping the '1's and the don't care states) in the map.

NOTE: As indicated, we make the outputs from the shaft encoder (B and A) synchronous by using 2 FFs before these signals are applied to the 2 FFs that implement the FSM. To do this, make sure that the PAL compiler is in **registered mode**.

Implementation task 1: Now run your equations through the PAL compiler. Read the Appendix on PALs and the PAL compiler if you are unfamiliar with them. The compiler should be running on a stand-alone PC connected to the programmer.

Implementation task 2: Now connect your PAL to a rotary encoder and your two outputs to LEDs. Connect the clock to pin 1 and make sure that the output enable (pin 11) is connected to ground. To test your design start by setting the clock to 10Hz. Make sure that each LED flashes once as the rotary encoder is moved one output combination clockwise or anticlockwise.

Step 2 - count clockwise/anticlockwise pulses

Implementation task: Use the clockwise/anticlockwise signals as up/down count signals for the 74HC193 counter. Connect the output of the counter to the 7-segment display. Make sure the unused inputs of the 74HC193 are tied off appropriately.

Optional Bonus Step

Task: How could you make use of two 74HC193 chips to provide an 8-bit up/down counter?

Assessment

Ticking criteria: Write up your design and answer the following questions. Demonstrate that your rotary position counter works. Once your work has met the **Common Ticking Criteria** (see Introduction), get your work ticked by an assessor. Remember that you need to hand in this assessed exercise as part of your portfolio of work (see the Head of Department's notice).

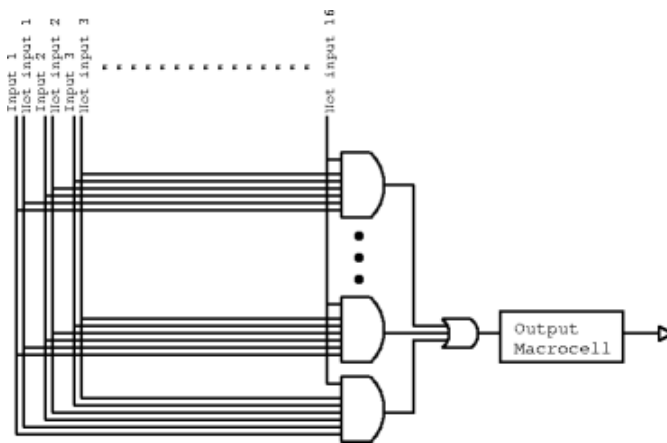
Questions

1. Why didn't we ask you to debounce the inputs from the rotary encoder?
2. How fast can the rotary encoder be rotated before your circuit fails to count correctly?

Appendix – PALs and the PAL Compiler

A PAL (programmable array logic) is a circuit which can be configured by the user to perform a logic function. It consists of an AND array followed by an OR array, the former being programmable. Inputs are fed into the AND array, which performs the desired AND functions and generates product terms. The various product terms are then fed into the OR array. In the OR array, the outputs of the various product terms are combined to produce the desired outputs.

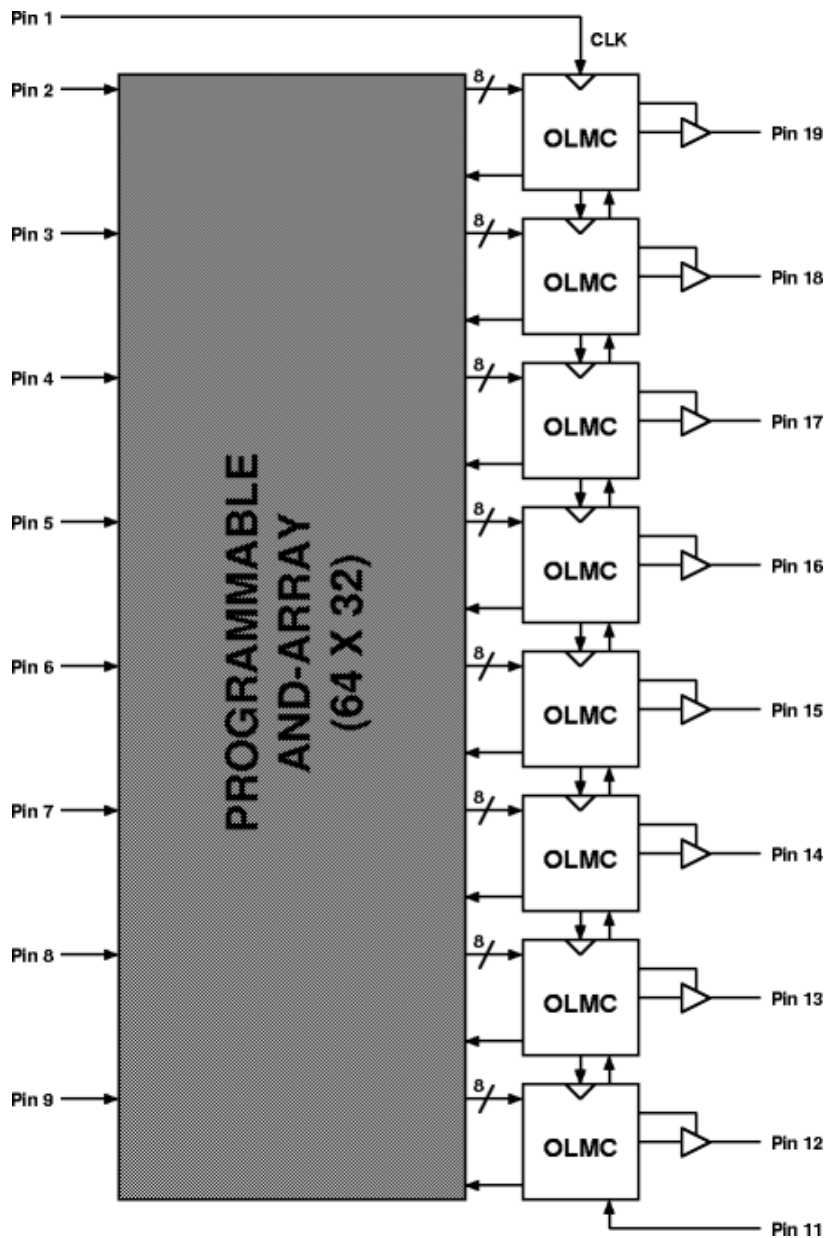
You will be using a small, relatively modern PAL called a GAL16V8. It has between 0 and 16 inputs and 8 to 0 outputs respectively, a common configuration is for up to 8 inputs and up to 8 outputs. Each output is driven by a single 8-input OR gate, which in turn is driven by eight 32-input AND gates. These 32 inputs can be programmed to be connected to the 16 possible inputs to the chip and their inverses. Note that when used to implement an FSM up to 8 of these inputs are actually outputs from the D-type FFs in each of the 8 output macrocells.



The output from the each OR gate is connect to an output pin through an output macrocell. This can be programmed to latch (using a D-type FF) the output or feed the output signal back into AND array. For a more detailed description of this PAL, read the datasheet.

PAL Compiler

To program the PAL with the functions necessary to implement a state machine, you will need a PAL compiler. This will produce a JEDEC file which is needed by the programmer. You will be using a web-based compiler. You need to type in equations using variables p2 to p9 on the input side (representing the input pins) and p12 to p19 on the output side (representing output pins).



When using the PAL Compiler you should have the above diagram in mind. The compiler expects 8 equations, one to drive each output logic macrocell (OLMC). The OLMCs in turn drive pins 19 to 12. What the 16 inputs are depends on the configuration mode of the PAL. Pins 2 to 9 are always inputs, but in **registered** mode pins 12 to 19 make the other 8, in **complex** mode pins 1, 11 and 13 to 18 are used and in **simple** mode pins 1, 11 to 14 and 17 to 19. The mode can be selected using the check boxes.

The equations for each output (called P19 to P12, in that order) can consist of inputs, operators and brackets. The input must be referred to by the "P" followed by a number. Operators are &, |, ^ and ! for AND, OR, XOR and NOT respectively

Examples:

P19 = p2	A simple assignment latch
P19 = p2 ^ p3 P18 = p2 & p3	A half adder
P19 = !p19	A 1-bit counter

When you have typed in your equations, click the **compile** button.

© 2011 DJ Greaves, DL Gordon, SW Moore and IJ Wassell

Page last updated on 17-Aug-2013

Workshop Three – Debouncing a Switch

Introduction

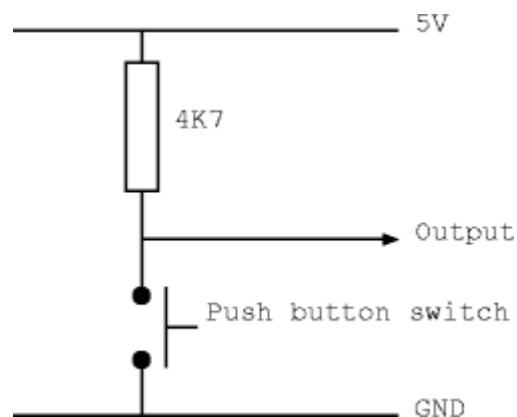
When using a switch as an input to a digital circuit, several pit-falls may be encountered. The problem of switch `bounce' is the subject of this experiment, what it is, and ways (including an all-digital method) of overcoming it.

Components

- 1a prototyping box
- connecting wire
- 2 x 4K7 resistor
- 74HC00 - NAND gates
- 74HC193 counter
- A PAL - GAL16V8

Step 1 - Watch a switch bouncing

Task: Build the following circuit on a breadboard using the blue non-debounced push-switch. Connect a scope probe to the output. Look at the output on the scope as the contact is made and broken. This is best done by triggering off the switch and varying the timebase until the appropriate detail can be seen.

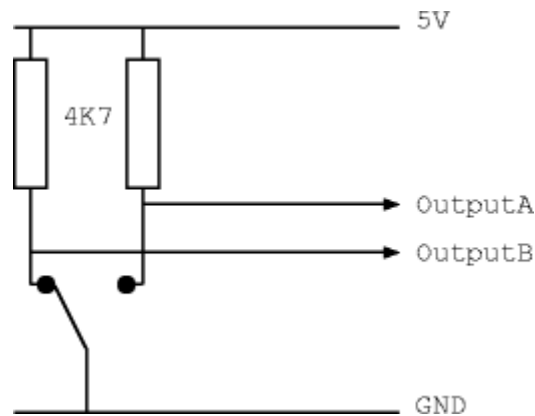


Step 2 - Count bounces

Task: Connect the output of the circuit in Step 1 to the clock input of the counter (make sure you connect the unused inputs appropriately). Connect the output to the 7-segment display. What should happen? Why doesn't it work? The object of the next step is to debounce the switch so that counter increments just once as the switch contact is made.

Step 3 - Debouncing a double-throw switch

You may be surprised to find that a double-throw switch is easier to debounce than single-throw switch, so we'll start with a double-throw switch first. The circuit below illustrates a simple circuit with a double-throw switch and two pull-up resistors. The switch you will be using is a break before make switch, i.e., it is mechanically incapable of connecting both switch terminals together. What is the relationship between the two outputs if bounces only appear on one contact at a time?



Task: Build the above circuit. Design and build a circuit which takes OutputA and OutputB as input and produces a debounced output. Hint: this only requires one latch.

Step 4 - Debouncing the push button switch

Debouncing a switch can be done in many ways, both using analogue and digital techniques. For this workshop, use a PAL to implement a shift register with the switch as its input. You should not use the output of the first flip-flop since it may go metastable if the input changes too near the clock. When all of the other flip-flops in the shift register become high, the output of your debouncer should go high. When they're all low, the output should go low.

Task: Design, build and test this digital switch debouncer.

Optional Bonus Step

How could you use an RC (resistor + capacitor) filter to remove bounce and what will this do to the slew rate (i.e., rate of change) of the signal?

Assessment

Ticking criteria: Write up your experimental data and final design, and then answer the following questions. Demonstrate that your debouncer works.

Once your work has met the **Common Ticking Criteria** (see Introduction), get your work ticked by an assessor. Remember that you need to hand in this assessed exercise as part of your portfolio of work (see the Head of Department's notice).

Questions

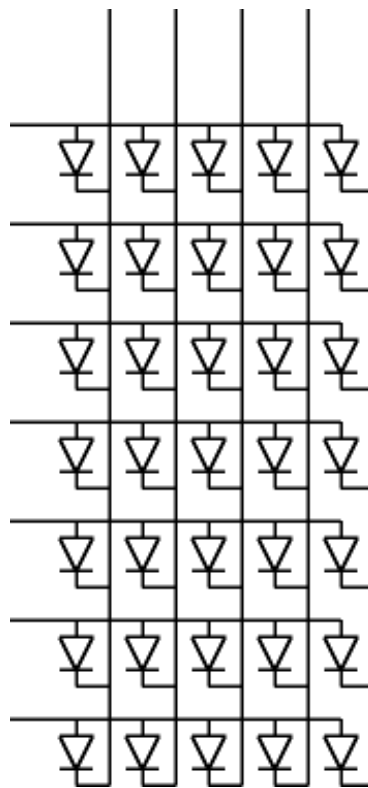
1. Why is a switch debouncer necessary?
2. How should you choose the clock frequency for the PAL based debouncer?

Workshop Four - Framestore for an LED Array

Introduction

This experiment is designed to show the operation of a framestore and introduce the use of a Static Random Access Memory (SRAM).

The design uses the SRAM as a framestore for two 5x7 light emitting diode (LED) arrays. The LED display is made up from individual LEDs which are connected in a grid. Connecting a row (through a current limiting resistor) to the power rail and a column to ground lights the LED where they cross. This means that although each LED can be lit individually, an arbitrary pattern cannot be displayed merely by applying constant voltages to the row and column inputs. Instead, the array must be scanned, preferably at a rate at which the display does not appear to flicker. In this experiment, the pattern to be displayed is to be read from an SRAM.



Components

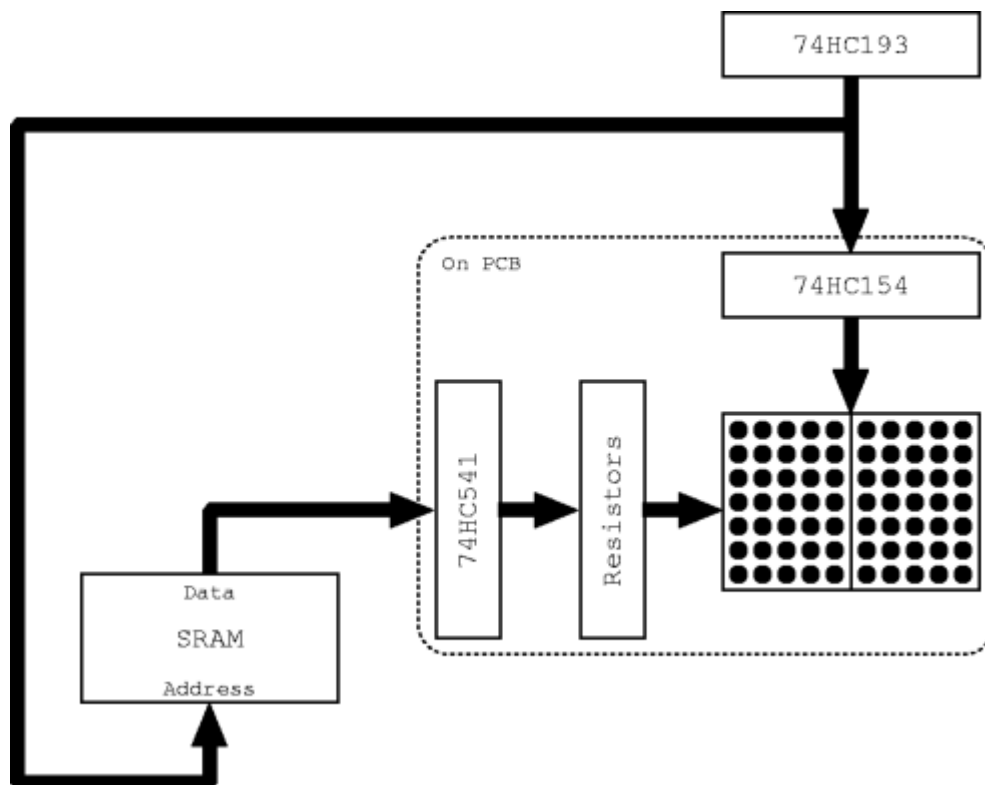
- 1a prototyping box
- connecting wires
- Holtek HT6116-70 SRAM
- 74HC541 octal tri-state buffer
- 74HC00 quad NAND gates
- 74HC193 4-bit synchronous binary counter
- 74HC74 dual D flip-flop
- Binary-coded rotary switch

Step 1 - Make the LED matrix light up

Task: Use the 74HC193 IC to produce a 4-bit counter. Make sure that the unused input pins are tied appropriately. Use this 4 bit value as the column address for scanning the LED array. Tie the row inputs to logic 1 (the supply voltage).

Step 2 - Connect the SRAM to the LED matrix

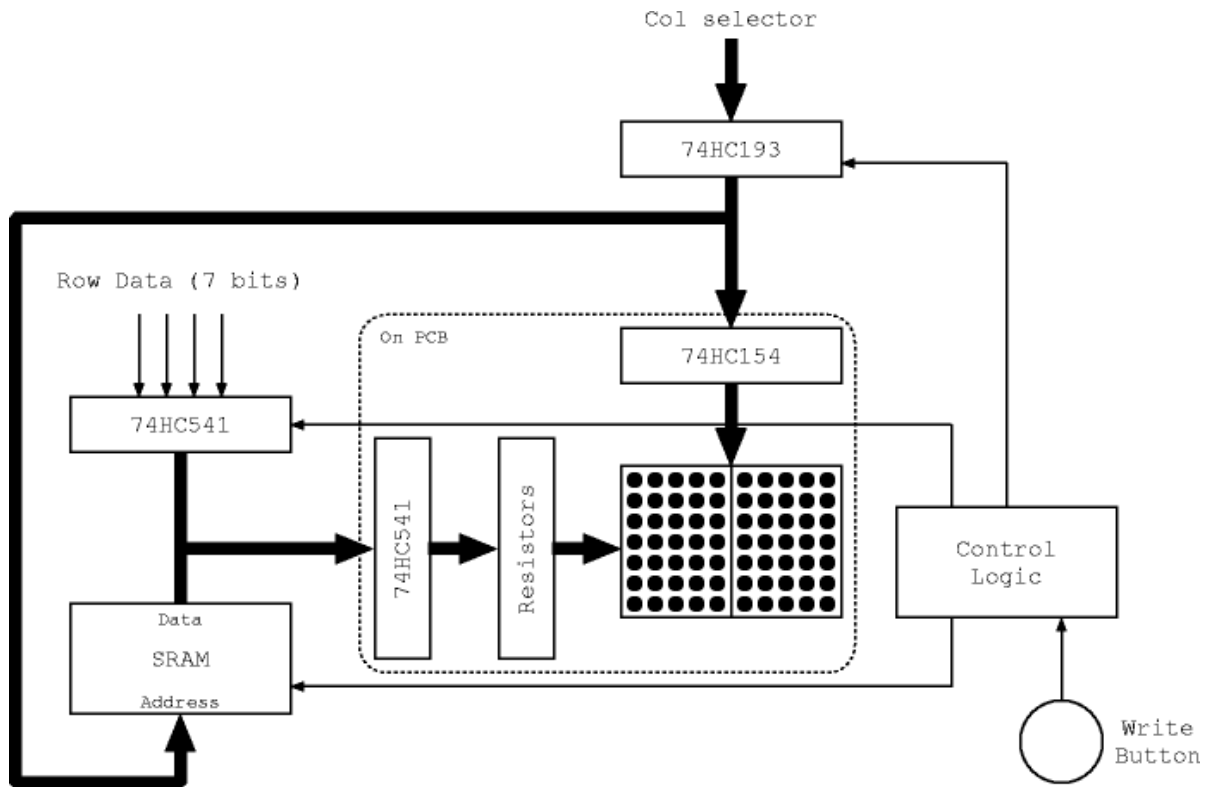
Task: Use the same 4-bit value as the address for the SRAM and connect the data output from the SRAM to the rows of the array, making sure that the CS, OE and WR pins are tied appropriately to fix the chip in read mode. Unused address pins should be connected to ground.



Your circuit should now display the random contents for SRAM on the array. Turn your power supply on and off several times to see if the pattern changes.

Step 3 - Add an SRAM write circuit

Task 1: The next part of this experiment is to add some circuitry to allow values to be written into the SRAM. The complete circuit should look something like this:



Connect the output of the 74HC541 octal buffer to the data bus, and use seven toggle switches as its input. Now connect the column selector (binary coded rotary switch) to the input of the binary counter. When complete, you should be able to write the data from the toggle switches into the SRAM at the address supplied by the column selector. This should only happen when a write button is pressed.

Task 2: Design the control circuit using a 74HC74 and a 74HC00 chip. The behaviour is as follows: as the write button (use the debounced switch) is pressed, the value from the rotary switch should be loaded into the binary counter. The value from the row switches should be driven onto the data bus, and the write pin on the SRAM chip should be pulsed. Inputs to the control circuit will be the clock and the write button. It needs to produce three outputs: an enable signal for the octal buffer, a write pulse for the SRAM and a load signal for the counter. Start by producing a timing diagram showing the clock oscillating and the switch being pressed. Fill in what you want the outputs to do as the switch is pressed and goes high. You might like a demonstrator to check before proceeding to the final design.

Tips:

- The write button should be synchronised with the clock before it is used. This can be done by passing it through a latch.
- The SRAM data bus now has another driver, a 74HC541 octal buffer. Care should be taken to ensure that the SRAM and the buffer do not drive this bus at the same time.
- In order to write into the SRAM, its write pin should be pulsed low. Remember to ensure that the address inputs do not change during the write operation and that the data input must be valid at the end of write operation but do not need to be valid at the start.
- The load of the binary counter is asynchronous. The output value changes immediately, and does not wait for the next rising edge of the clock.

- The column selector needs 4 x 4K7 pull-down resistors on its outputs.

Optional Bonus Step

Implement a "clear display" button which writes zeros into the SRAM.

Assessment

Ticking criteria: Write up your experimental data and final design, and then answer the following questions. Demonstrate that your circuit works correctly: you should be able to specify a column on the LED array with the column selector and the data pattern for that column with the toggle switches. Pressing the write button should make the pattern permanent.

Once your work has met the **Common Ticking Criteria** (see Introduction), get your work ticked by an assessor. Remember that you need to hand in this assessed exercise as part of your portfolio of work (see the Head of Department's notice).

Note: because this workshop is a little tricky, one tick is awarded for completing steps 1 and 2 and question 1, and the other tick for step 3 and question 2.

Questions

1. Each LED in the matrix can draw a maximum current of 5mA. What is the maximum current that the two LED arrays can draw during the operation of the frame store?
2. What would happen if the write button was not synchronised?

© 2011 DJ Greaves, DL Gordon, SW Moore and IJ Wassell

Page last updated on 17-Jul-2013

Workshop Five – nMOS, pMOS and CMOS Inverters

Introduction

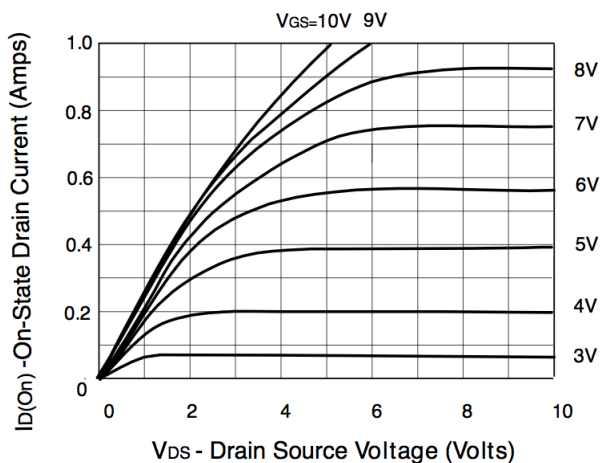
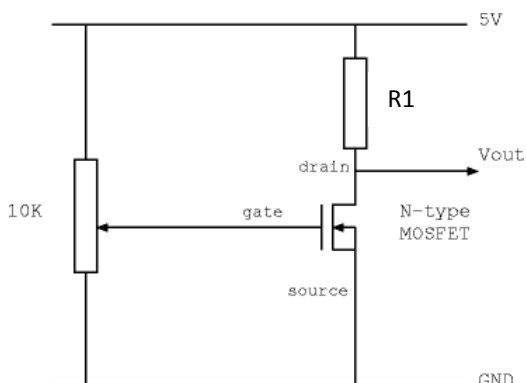
In this workshop you will build nMOS, pMOS and CMOS inverters and then measure their characteristics.

Components

- oscilloscope
- 1a prototyping box
- connecting wire
- pair of matched nMOS (ZVN3306A) and pMOS (ZVP2106A) FETs
- 27 Ω , 470 Ω , 4.7k Ω resistors
- 330nF capacitor

Step 1: Characterise nMOS Inverters

Task 1: For the nMOS inverter circuit shown below with $R1 = 27\Omega$, use the adjacent transistor characteristics (a larger version of which is reproduced in Appendix 1) to estimate V_{out} for $V_{GS} = 0V, 3V, 4V$ and $5V$. Build the circuit on the breadboard and measure V_{out} at the specified values of V_{GS} using the oscilloscope (and its voltage cursor function) and compare them with the estimated values. Make sure you properly identify which of the FET leads is the source (S), drain (D) and gate (G) before inserting it into the breadboard. Connect the potentiometer and resistor R1 as shown.



Task 2: To characterize this inverter at a more realistic value of drain current, now make $R1 = 4.7k\Omega$. You will need to measure how the output voltage V_{out} varies with the input voltage V_{GS} . Connect one of the oscilloscope probes to the drain of the transistor and the other to the gate. Vary the gate voltage in steps from 0 to 5V using the potentiometer. At each step, measure the voltage at the drain. Note that over wide ranges of gate voltage the drain voltage will be constantly near to either 5V or 0V, consequently you do not need to use constant step sizes over the entire range and you only need to use small step sizes (say 0.2V) where the drain voltage changes rapidly. Plot a graph of drain vs gate voltage showing the transistor switching.

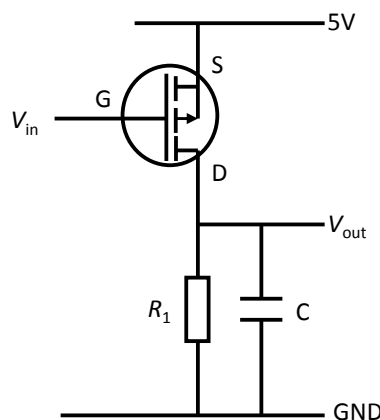
For the nMOS inverter, how does the current flowing through the transistor vary as the gate voltage increases?

Task 3: Investigate the effect of capacitive loading. To do this include a 330nF capacitor (C) at the output of the nMOS inverter, i.e., connect it between V_{out} and GND (0V). Disconnect the potentiometer from the input V_{GS} and instead connect V_{GS} to the Clock Generator output. Set the operating frequency to say 100Hz initially, but be prepared to change it if required later on when performing measurements of output transition times. Sketch the input and output signals observed using the oscilloscope. Specifically, for the output rising edges measure the time it takes for the signal to rise from 0V to $0.63 \times 5V = 3.15V$ and compare this with the time constant CR_1 . Now for the output falling edges, measure the time it takes for the signal to fall from 5V to $0.37 \times 5V = 1.85V$. Why is this much shorter than the rise time? Now estimate the ON resistance of the FET.

Now make $R_1 = 470 \Omega$ and then repeat the previous measurements. What is the advantage and what is the disadvantage of increasing R_1 ?

Step 2: Investigate pMOS Inverters

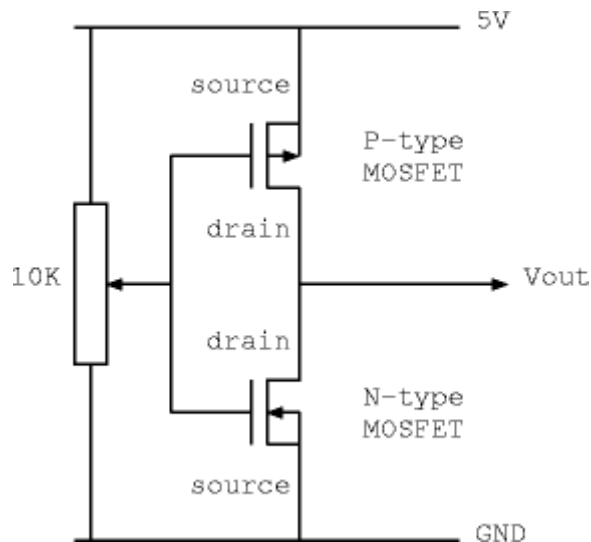
Task 1: Investigate the effect of capacitive loading. Build the pMOS inverter circuit shown below with $R_1 = 4.7k \Omega$, load capacitor $C = 330nF$ and connect V_{in} to the Clock Generator output. Sketch the input and output signals observed using the oscilloscope and perform similar measurements to that undertaken in Step 1, Task 3 concerning the output rise and fall times. Compare this waveform with that recorded previously in Task 3, highlighting the significant differences. Calculate the theoretical waveform falling edge time (as defined in Task 3) and use the rising edge measurement to estimate the ON resistance of the pMOS FET.



Step 3: Characterise a CMOS inverter

Task 1: The CMOS inverter circuit shown in the following figure effectively combines the nMOS and pMOS inverters shown previously. First characterise the input-output characteristic by repeating the measurement procedure in Step 1, Task 2. For the CMOS inverter, how does the current flowing through the transistor vary as the input voltage increases?

WARNING: ensure you connect the source and drains of the transistors as shown since if you get them the wrong way around the transistors will conduct and burn out!



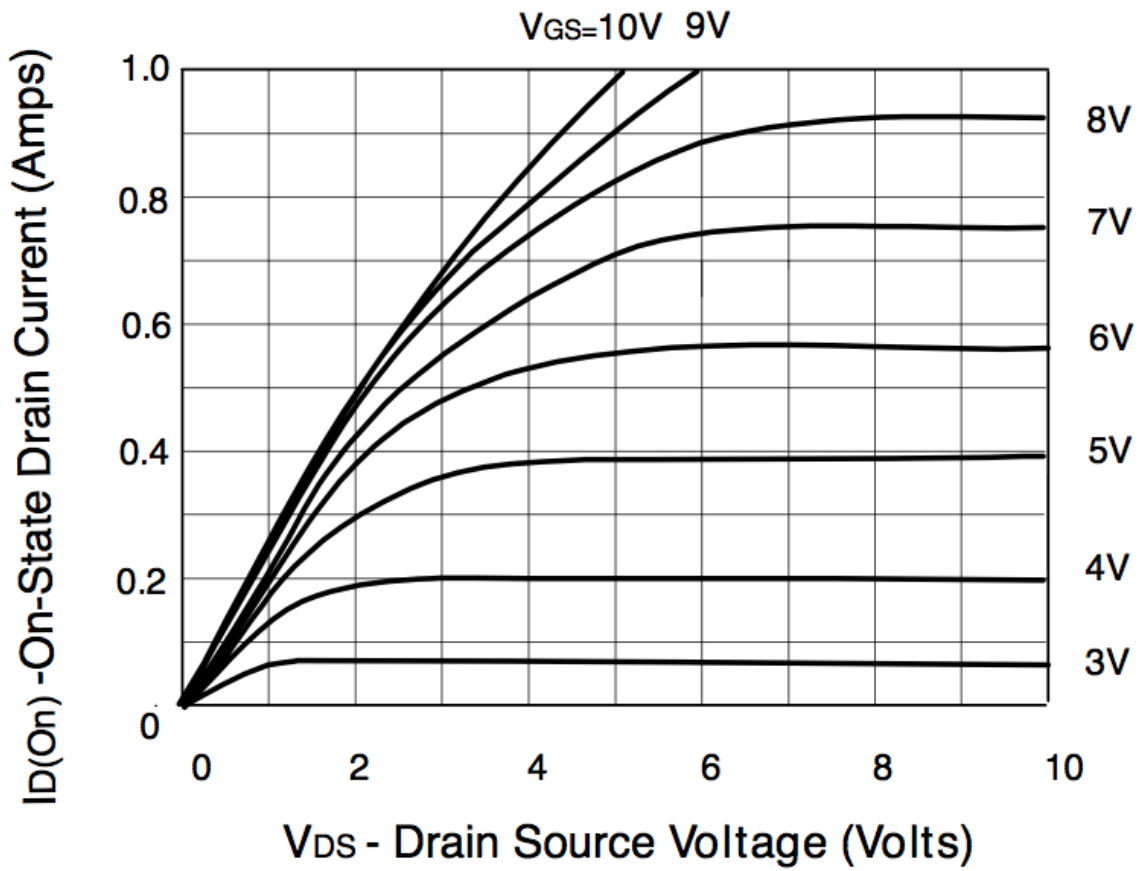
Task 2: Investigate the effect of capacitive loading. To do this add a 330nF capacitor (C) at the output, i.e., connect it between V_{out} and GND (0V). Disconnect the potentiometer from the inverter input and instead connect the inverter input to the Clock Generator output. Set the operating frequency to say 100Hz initially, but be prepared to change it if required later on when performing measurements of output transition times. Sketch the input and output signals observed using the oscilloscope. Compare this waveform with those observed previously for the nMOS and pMOS inverters and note down the significant differences. What other advantage does the CMOS inverter have over the nMOS and pMOS inverters?

Assessment

Ticking criteria: Write up your experimental data and answer the questions.

Once your work has met the Common Ticking Criteria (see Introduction), get your work ticked by an assessor. Remember that you need to hand in this assessed exercise as part of your portfolio of work (see the Head of Department's notice).

Appendix 1 – nMOS Transistor Characteristics



© 2013 DJ Greaves, DL Gordon, SW Moore and IJ Wassell

Page last updated on 30-Jul-2013

Workshop Six - Digital-to-Analogue Conversion

Introduction

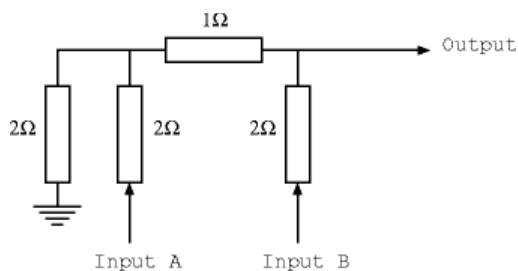
In this workshop, you will build two different kinds of digital-to-analogue converter (DAC). The first uses a resistor network called an R2R ladder, and works in parallel, i.e., a binary value is fed into the resistor network in parallel and a corresponding analogue signal is output. The second DAC (known as a 1-bit DAC) is all digital and uses a clock and a state machine to produce a single output. The output fed through an analogue low pass filter which removes the high frequency digital component leaving an average of the ones and zeros as the analogue output. The more ones that are in the sequence will cause the output voltage to rise, while the more zeros will cause the output voltage to fall.

Components

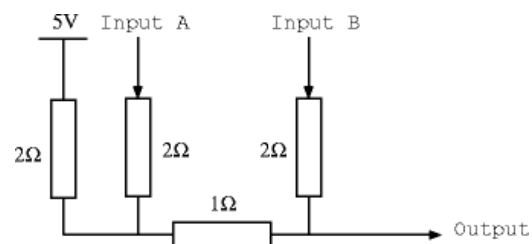
- 1a prototyping box
- 1 x 110Ω resistor
- 6 x $4k7\Omega$ resistors
- 1 x 74HC283 adder
- 1 x 74HC374 octal D-type flip-flop
- 1 x $0.1\mu\text{F}$ capacitor
- 1 x light bulb and holder
- nMOS FET (ZVN3306A)
- Binary-coded rotary switch (optional)

Step 1: R2R ladder DAC

Task 1: Consider the following resistor network and answer questions 1 and 2.



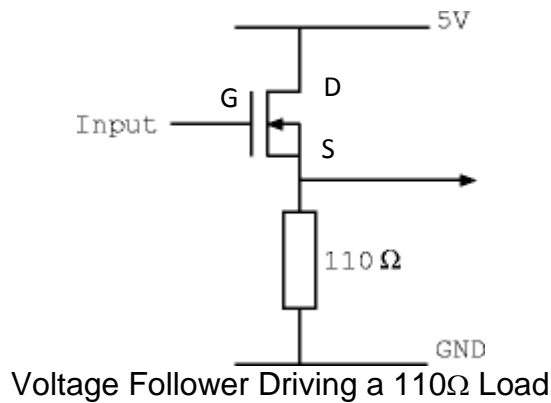
Task 2: Now consider the inverted version of this resistor network and answer question 1 again.



Step 2: Build the R2R DAC

Task 1: Construct the inverted DAC using $4.7k\Omega$ resistors to replace the 2Ω resistors, and two $4.7k\Omega$ resistors in parallel to replace the 1Ω resistors. Use the toggle switches as input and check that the output voltages are correct as the inputs are changed.

Task 2: Now connect an nMOS transistor and resistor load to form a voltage follower as shown in the following figure. Answer questions 3, 4 and 5.



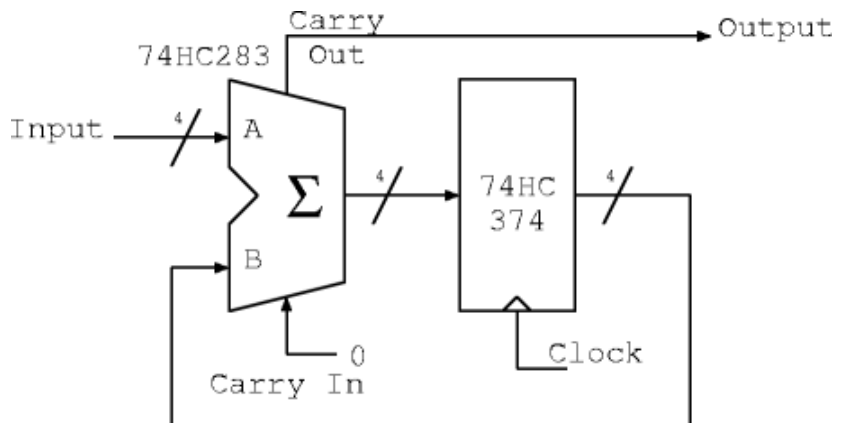
Step 3: Build a 1-bit DAC (Bitstream DAC)

Use either four of the switches on the right-hand side of the breadboard or else the binary coded rotary switch to generate the desired input value.

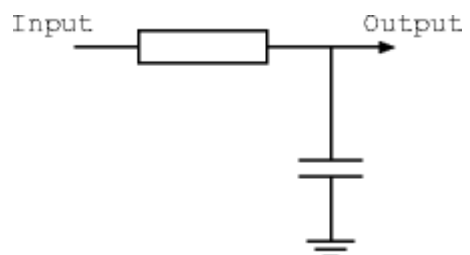
Task 1: Consider the waveforms and answer question 6.



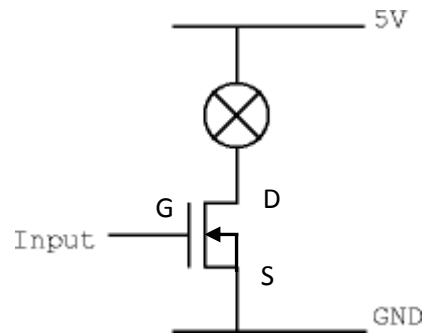
Task 2: Consider the circuit diagram below. The 74HC283 is a 4-bit adder with carry-in and carry-out. The 74HC374 contains 8 D-type flip-flops (although this circuit only uses 4 of them). Answer question 7. Construct this circuit and verify that it works.



Task 3: One way to convert this waveform into a steady voltage (the average voltage) is to filter the output through a low-pass filter. This will remove the high frequency component in the waveform. Set the clock to 100kHz, and try a resistor value of 4.7kΩ and a 0.1μF capacitor. Using an oscilloscope, verify that the output is filtered and corresponds to the binary input selected. Answer question 8.



Task 4: Replace the output filter with a light bulb and nMOS driver transistor as shown below. Since the temperature of the light bulb filament has inertia, it provides a filtering effect on the light output. Answer question 9.



Assessment

Ticking criteria: Write up your experimental data and answer the following questions.

Once your work has met the **Common Ticking Criteria** (see Introduction), get your work ticked by an assessor. Remember that you need to hand in this assessed exercise as part of your portfolio of work (see the Head of Department's notice).

Questions

1. Using Ohm's law and the formula for resistors in parallel, what is the output voltage of the R2R DAC when inputs A and B are connected to each combination of ground and 5 volts? Please display the results as a table.
2. How much current will flow through this resistor network when input B is connected to 5 volts and A to ground?
3. How much current flows through the load when the gate voltage is at its maximum? Give the values of two parameters of the circuit, apart from the supply voltage, that limits the current?
4. Why is it a good idea to use a voltage follower on the output of the R2R DAC resistor network?
5. Why is the inverted version of the R2R DAC used with the voltage follower rather than using the non-inverted form?
6. What is the long-term average voltage of each of the 1-bit DAC waveforms?
7. For the 1-bit DAC, if the input has a fixed value 7 (0111 in binary), and the flip-flops all start at zero, what values do the flip-flops attain in the next 16 clock cycles and what value does carry-out take in each cycle?
8. If you were going to design a 16-bit DAC for audio purposes (e.g., CD player output), how would the resistor tolerances affect the errors in the output for R2R and 1-bit DAC implementations?
9. How does the power efficiency of the R2R DAC with voltage follower compare with the 1-bit DAC with driver transistor?

© 2011 DJ Greaves, DL Gordon, SW Moore and IJ Wassell

Page last updated on 13-Jul-2012