NAPIER UNIVERSITY
EDINBURGH

ASSIGNMENT COVER SHEET

MATRICULATION NUMBER: | 4 | 0 | 2 | 9 | 2 | 3 | 0 | 2 |

If this is a group assignment, please provide the Matriculation Numbers of all group members.

MATRICULATION NUMBER(S): | | | | | | | |

| | | | | | | | | | | | | | | | | | | | | | | | |

Please ensure that you have removed your name from your assignment
– don't forget to check both the header and the footer.
Please **do** include your Matriculation Number, though.

MODULE NUMBER: **ELE08110**

MODULE TITLE: **Circuit Realisation**

ASSESSMENT TITLE: Report On/Off Light Circuit _____

NAME OF MODULE LEADER: **A Edgar**

DATE OF SUBMISSION: _____

DECLARATION

I agree to work within Napier University's Academic Conduct Regulations[1] which require that any work that I submit is entirely my own[2]. The regulations require me to use appropriate citations and references in order to acknowledge where I have used any materials from any sources.

I am providing my student Matriculation Number (above) - in place of a signed declaration – in order to comply with Napier University's assessment procedures.

To be completed by Marker

Mark   **Marker's signature**

NAME(S)__Olivier Chaligne_____

---

[1] These form part of the Student Disciplinary Regulations - www.napier.ac.uk/registry/regulation.htm.
A useful website on Academic Conduct requirements and how you can ensure that you meet them may
be accessed through the Student Portal, via the Plagiarism icon.
Please note that breaches of Student Disciplinary Regulations, such as Plagiarism and Collusion, may be investigated and penalised.
[2] If the assignment brief specifies this is a group assignment, the Matriculation Numbers for all group members must be included on this coversheet. The work must then be entirely the work of the group members, who agree collectively to the statement in the declaration.

Report: ON/OFF Light Circuit

40292302

B.Eng Mechatronics Year 2   March 2018

Table of Content

Introduction

This report details the steps taken to produce a working prototype for an On/Off light circuit. Receiving the hand-written schematic from the Engineering department, the first step was to implement it in OrCAD using the Capture module of the programme. Then, a simulation was realised, using PSpice, to verify the circuit was responding to the requirement of the design. Then, using the PCB editor module, components were placed and routed to obtain the design data used to produce the PCB. Finally, the circuit was constructed and tested. The report concludes on a review and possible improvement of the process.

I. Circuit Capture

1. Hand written design to digital format.

The first step was receiving the hand-written design (Fig I.1.1) from the engineers. It was assumed the choice of components and relating values would provide a working design answering the requirements from the design brief agreed by the customer. The circuit would be simulated, using PSpice, before being manufactured. This would allow to check if it indeed would fulfil its function.
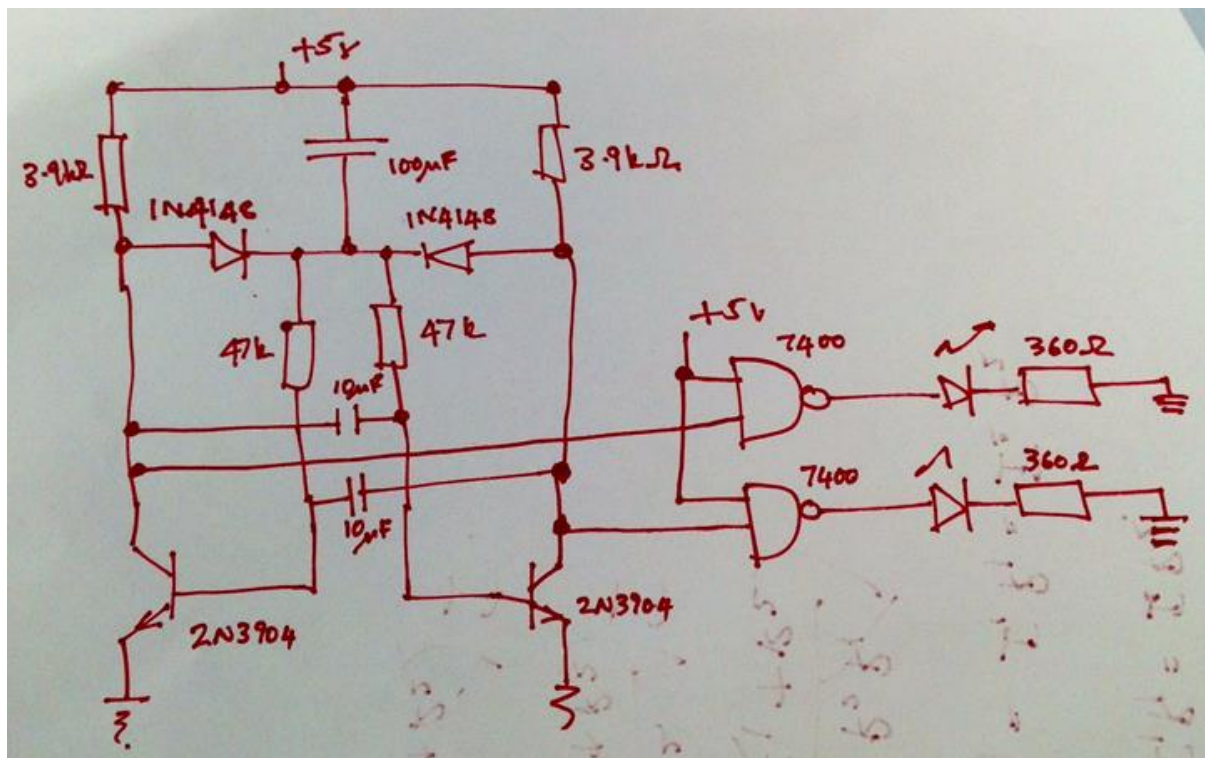

Fig I.1.1. Hand written On/off light circuit design.

Next, using the provided design and the Capture module from the OrCAD suite, the circuit was implemented digitally. A few points were to take into consideration to complete the task.

First, it is noticeable from Fig I.1.1, the components attached to the collector of the transistors have tracks going over other tracks hence not creating connexions. It was also noted most components (Electrolytic capacitors, diodes, transistors, integrated circuit) were connected in a specific way due to their polarity and/or due to their function in the circuit. The components would also have to be picked from the PSpice library. Failure to do so would not allow to the required simulation of the circuit. Indeed, the PSpice library provides the mathematical function describing the behaviour of the component to simulate its reaction in the circuit.

It is worth mentioning the presentation of the circuit did not need to be exactly the same as on the provided design, but it would keep the project consistent and easier to troubleshoot in the event of a failure.
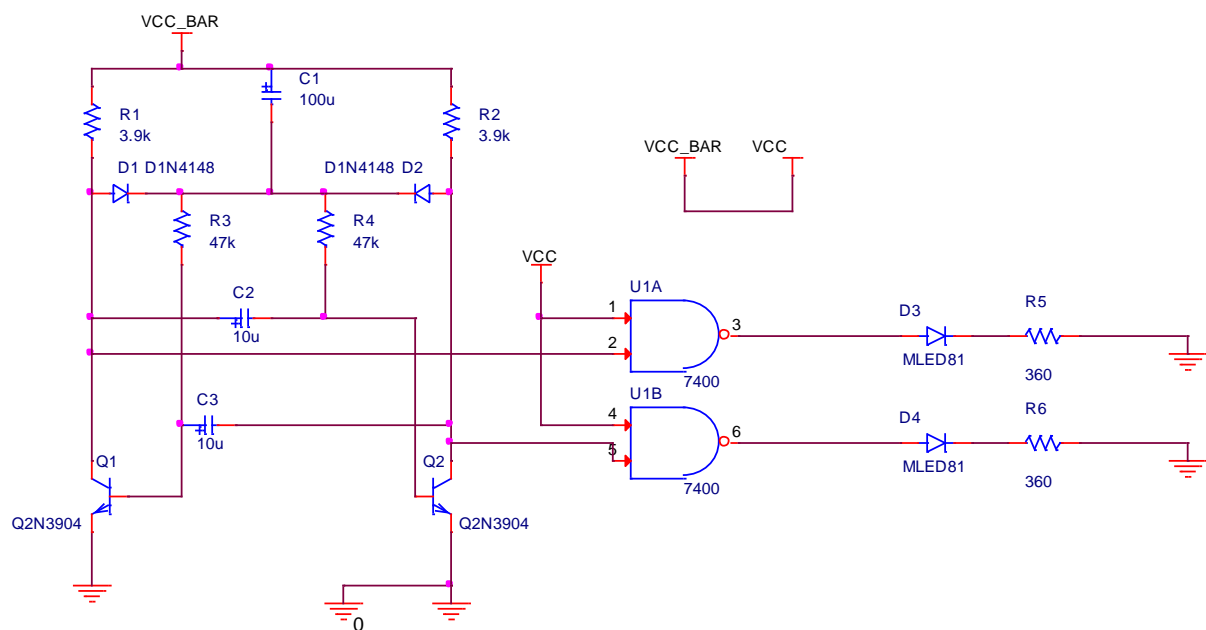

Fig I.1.2 OrCAD Capture: On/off light circuit design.

2. Special Considerations.

From Fig I.1.2, it is noticeable two components were added to the design from Fig I.1.1:

VCC_BAR   VCC

Fig I.2.1 Grounds   Fig I.2.2 Supplies.

These symbols were used to order to make the circuit clearer and more understable. It avoids having to run a supply track and a ground track connecting the various points of the design.

Fig I.2.1 signifies that any ground symbol will be associated to the common ground GND.

Fig I.2.2 means any VCC point is connected to the supply through VCC_BAR.

The gates from the IC needed to receive current and to be connected to a ground to fulfil their function. The pins could be displayed and connected manually. But keeping in mind the need for the circuit to be as clear as possible the pins would receive "GND" for the negative and "VCC" for the positive thus avoiding unecessary cabling on the schematics. Fig I.2.3 demonstrates the procedure for one of the gates.
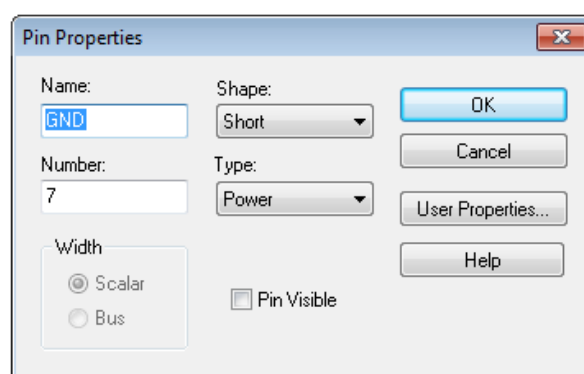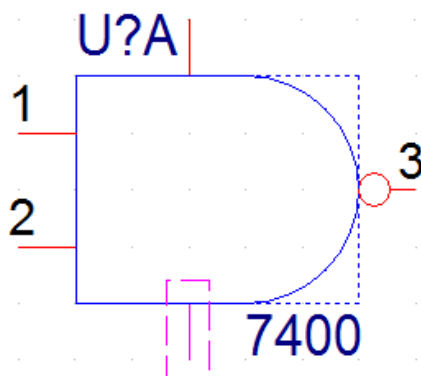
Fig I.2.3 Gate pins.

The capacitors, that were advised for use, being electrolytics are polarised therefore their pins needed to be too. Fig I.2.4 demonstrates the procedure for the positive pin 1. The procedure was repeated on the other pin for the negative pin 2.
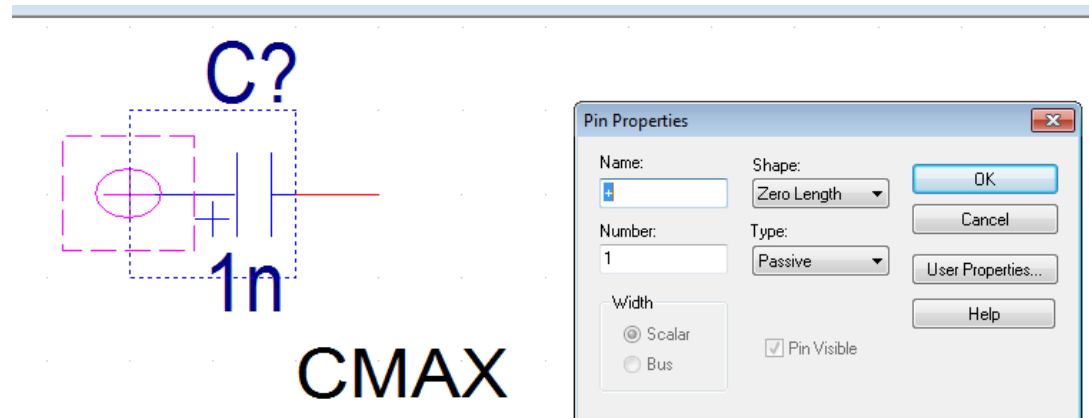

Fig I.2.4 Electrolytic Capacitor pins.

Finally, each component received a footprint which was used later for placing the components on the PCB. This defines the physical space occupied by the component on the board.
Then, through the Property Editor, in the parts section, it was possible to edit the PCB Footprint for each component using the "PCB Editor packages Default" (Cadence, 2009). The documentations for the transistors (Multicomp, 2012) provided the package code.
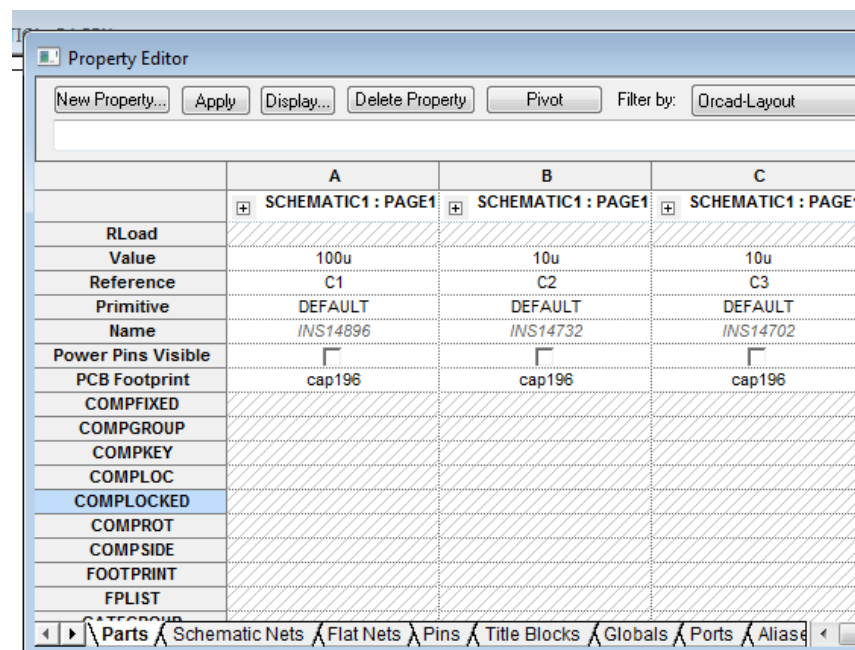

Fig I.2.4 Component Footprint

3. Preparation for simulation.

To be able to realise simulation it was necessary to add two modules that were not present on the original schematics. Although they will not be present on the final design. This section explains their purpose.

First, Fig I.3.1, this component is a current pulse generator. Its functions are to be tuned to operate correctly:

I1 (initial current) = 0

I2 (max current) = 20 mA

TD (Time delay before pulse) = 0

TR (Time Rise) = 1ms

TF (Time Fall) = 1µs

PW (Pulse Width) = 0.0001s

PER (Period) = 1000s

This generator was attached at the base of one of the transistors. The purpose was to create an in balance in between the two transistors to create an alternating oscillation. In applications the two transistors would be slightly different due to manufacturing and oscillate in an alternating manner. PSpice uses a standard formula to simulate the reaction of the components hence the two transistors would be oscillating at the same time and not create an alternating pattern. The pulse saturated the transistor essentially shutting it "off" when the other transistor was "on", creating and alternating pattern.

I1 = 0
I2 = 20m
TD = 0
TR = 1m
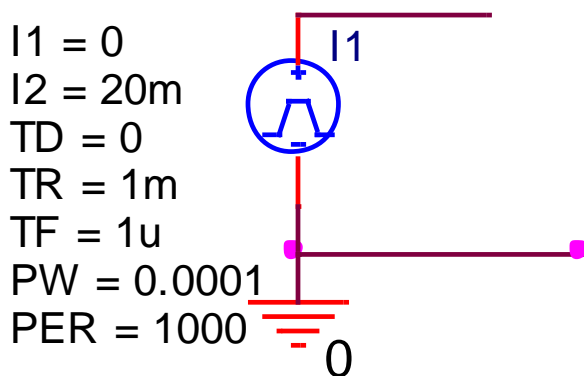TF = 1u
PW = 0.0001
PER = 1000

Fig I.3.1 iPulse Component.

Secondly, a supply module was added. The supply for our device was to be external therefore to obtain simulation result the programme must assume the circuit is connected to a supply. Here, the header connector was connected to a supply (V). This supply is connected to the circuit using VCC_BAR and GND.
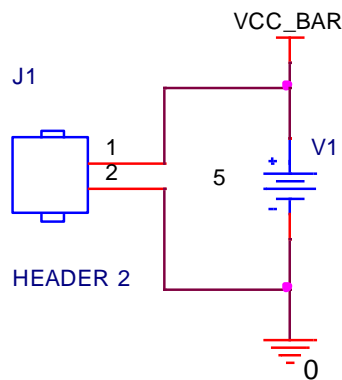

Fig I.3.2 Supply Module.

4. Final Circuit and Summary

Fig I.4.1 shows how the modules from the previous section were integrated to the design. Now was the good time to realise a "Design Rule Check" (DRC) to verify the design follows the basic principles of circuitry. Selecting the oscillator.dsn in the file manager and then using Tools> Design Rules Check would allow to setup the programme to realise electrical and physical checks. Errors were dealt with (explanations provided in the last part of this report)
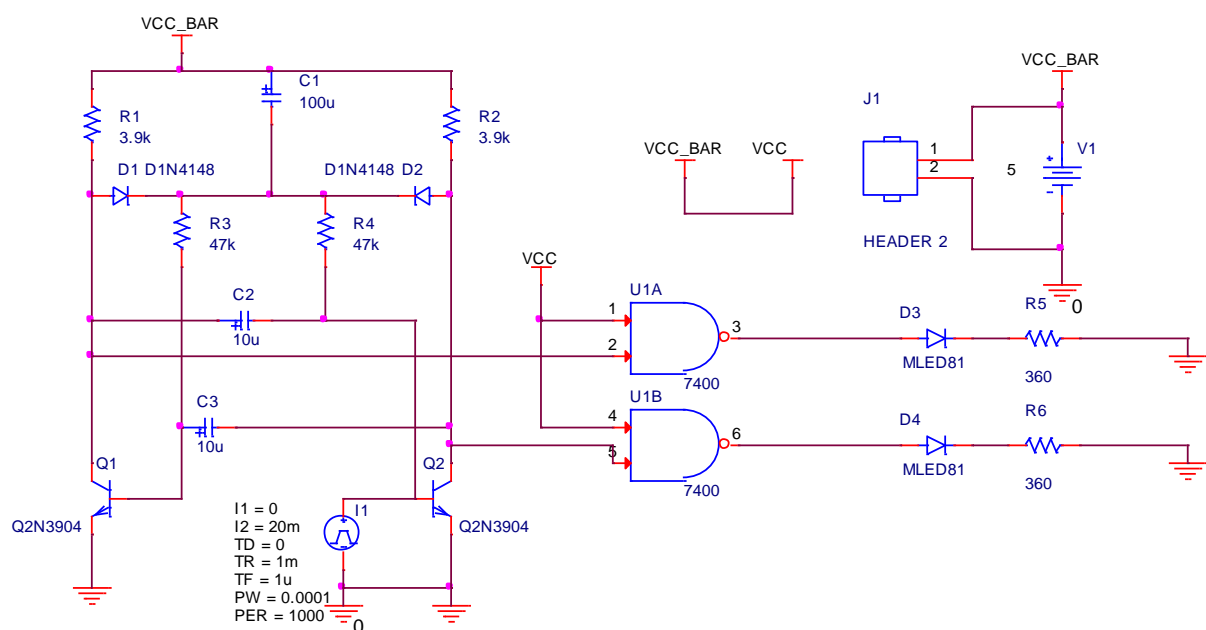

Fig I.4.1 Final Circuit.

Summary of the Capture section:

1. The hand drawn design was implemented in OrCAD.
2. Characteristics (Polarity, function) of the components were considered and applied.
3. "Grounds" and "Supplies" were added to keep the design clear and readable avoiding unnecessary cabling.
4. Modification had to be added (Pulse and power supply) to provide a working circuit for the simulation.
5. The circuit passed a Design Rules Check and error were corrected.

II. Simulation

1. Simulation Setup.

Logically, the interest of this circuit is to have the output LEDs blinking alternatively. Therefore, as a simulation, it was necessary to measure the signal going to the LEDs. The measurements for each LED would reported on a graph which would show the signals alternating. The probes were placed as per Fig II.1.1
PSpice is the simulation module included in OrCAD. The first step, to realising the simulation, was creating a simulation profile using from the menu PSpice> New Simulation Profile. The configuration of the profile was a transient time domain analysis ran over 2 seconds (sufficient to notice an alternating signal pattern) of the signal from the probes red and green.
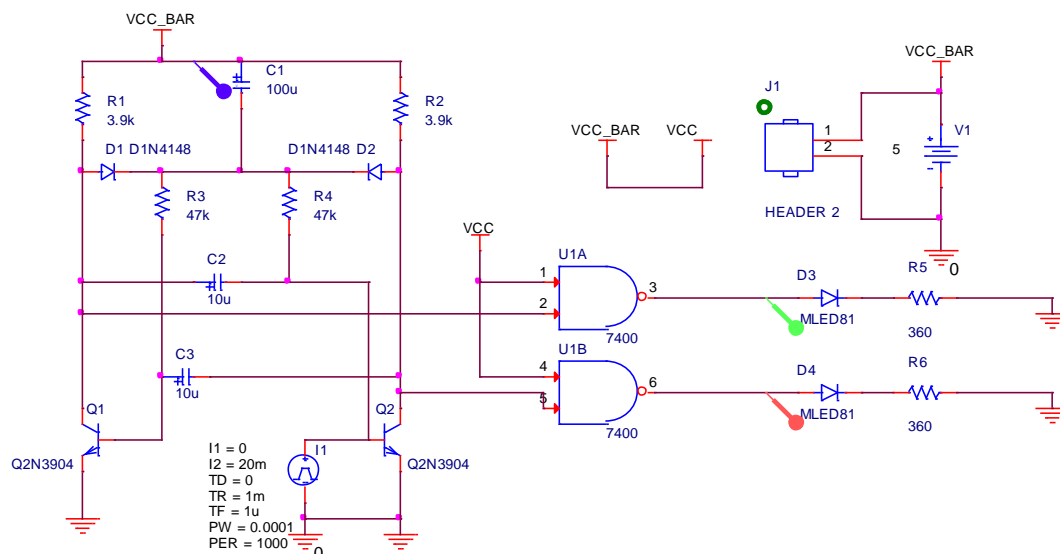

Fig II.1.1 Probe Placement.

2.Simulation Results.

This section presents the output of the simulation.

Fig II.2.1 offers the votlage continuity of circuit. It allows to verify the different parts were power supplied and taken in account in the simulation.

Fig II.2.2, clearly displays the signals of each LED alternating. When one signal was high the other was low and the oscillation continues over time.
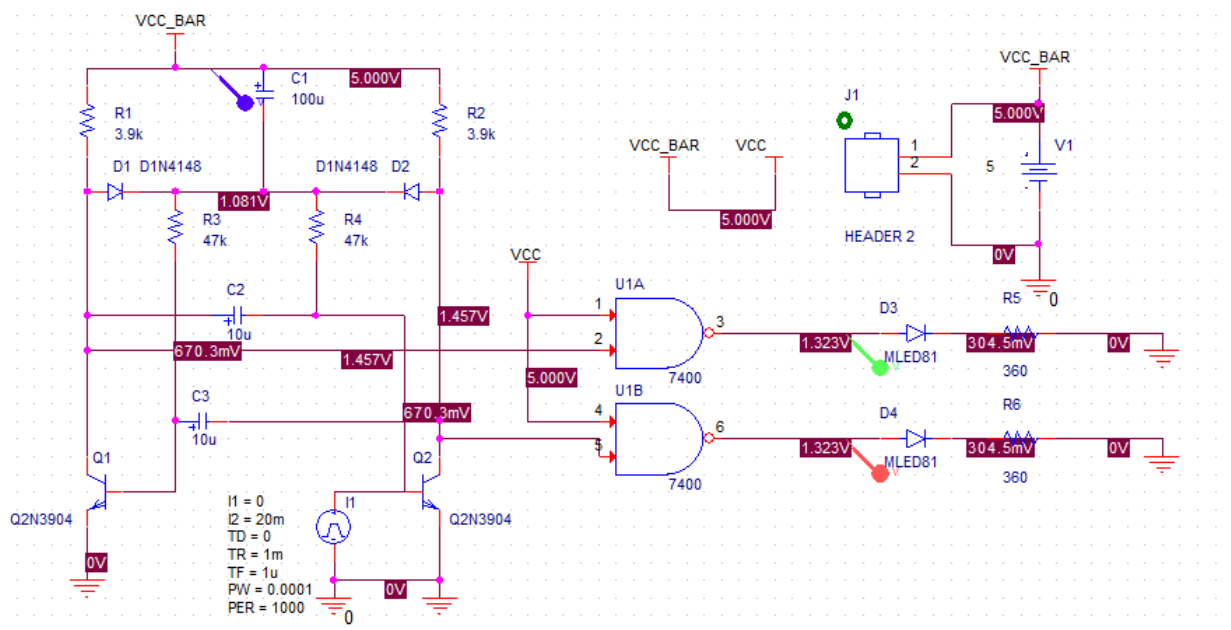
Fig II.2.1 Voltage Continuity.

Fig II.2.2 Graphical representation of each LED output.

III. PCB Placement and layout.

1. Printed Circuit Board template creation.

In this part of the design process, the PCB Editor module from OrCAD was used. First, it was necessary to create the template of the PCB. This essentially defines the physical properties of the board on whichthe components will be placed. The creation of the PCB was done using the wizard utility tool from the PCB Editor.

The parameters used in the wizard were:
- Measurement units: mil.
- Drawing size: A.
- Grid spacing: 100 mils.
- Two Layers: Top/Bottom.
- Minimum line width: 25 Mils (In the brief the track width and spacing should be at least 20 mils; the ground and power tracks at least 25 mils. To simplify all tracks were set at 25 mils).
- Padstack (pin soldering geormetrical surface): 60C85c35d.
- Rectangular board 3000x2000 mils.
- Route keepin distance (Safe placement of tracks fron the edge): 100 mils.
- Package keepin distance (Safe palcement of the components from the edge): 250 mils

The template was saved and ready to receive the components. Using the design file in Capture, it was possible to create the Netlist using the function of the same name in the menu bar.
In this window some options had to be verified and confirmed before moving to the next stage:
- "PCB Footprint" had to be present.
- "Create PCB Editor Netlist" selected.
- The Netlist Files Directory should be displayed.
- "Create or Update PCB Editor Board" selected.
- "Input board file" is the templated created previously.
- In "Board Launching Option" the option "Open Board in OrCAD PCB Editor" was selected.

2. Component placement and ratsnest.

Once the steps from the previous section completed the PCB Editor opened on a view of our board template. Using the "Create Netlist" allowed to import all the information regarding our design and components from the design file to the PCB. Using place or autoplace it was possible to move our components on the board template. Using the ratsnest allowed to see the relations between the components and help to place them to avoid track conflicts in the next stage. Fig III.2.1, is an example of component placement. In reality this step was repeated several times before a workign design was possible.
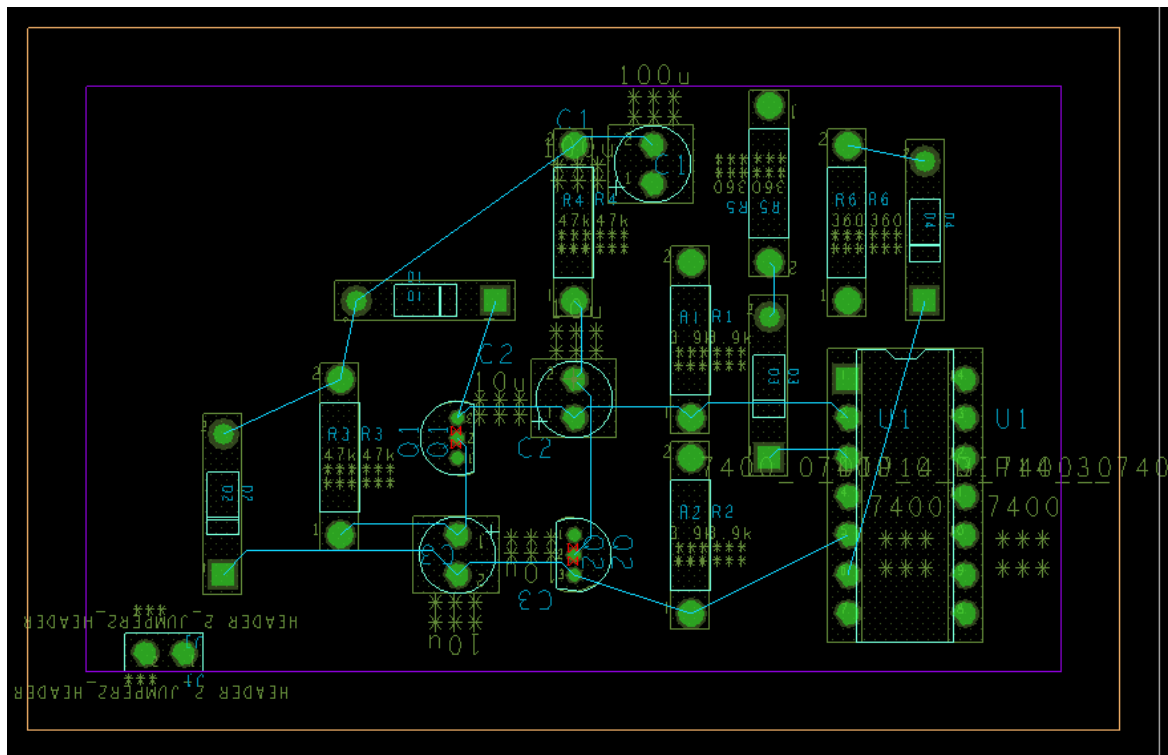

Fig III.2.1 Placement of the components on the PCB and ratsnest.

3. Board routing and track improvement.

Once the components were placed it was possible to route the components using the autoroute function. As mentionned in the previous section this had to be done several times to correct tracking errors and obtain a routing with no crossing tracks. Fig III.3.1 shows the Final PCB Routing (comments will be available in the conclusion) used to generate the Design data.

Fig III.3.1 Final PCB routing.
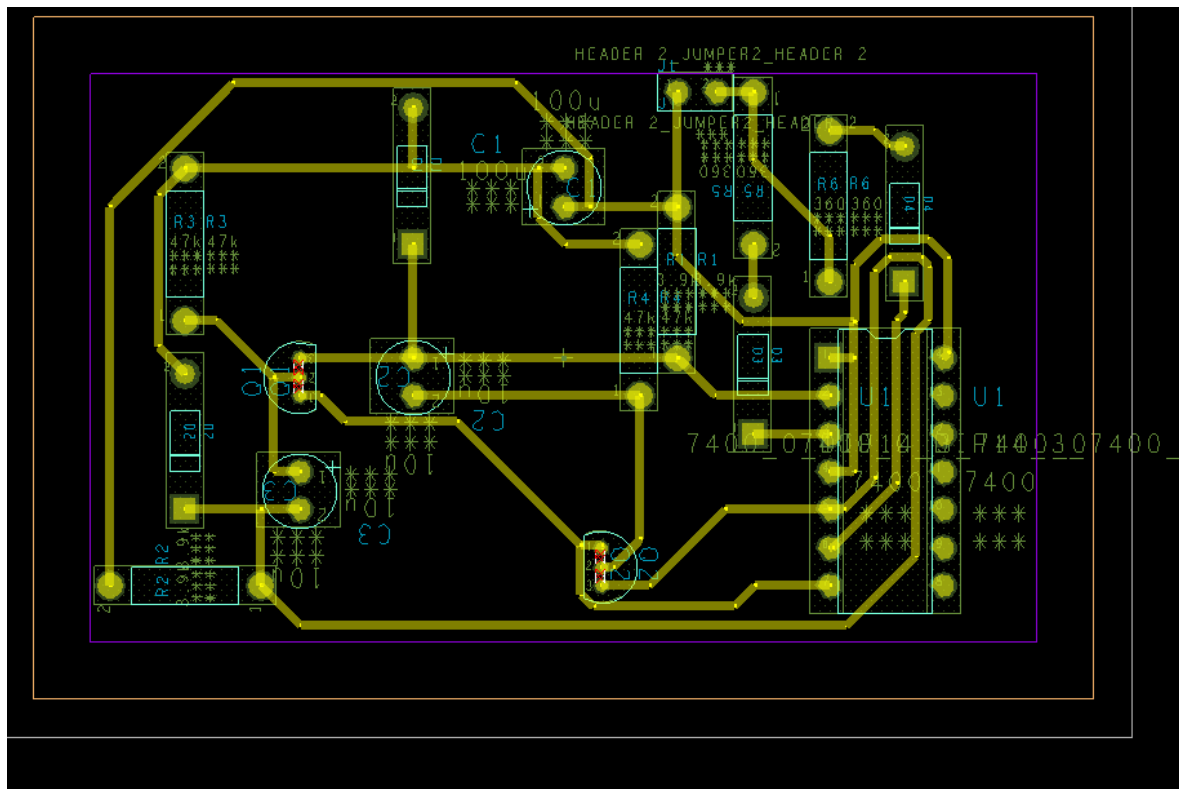
IV. Design Data

This section describes how the manufacturing files were obtained and their use. These files were sent to the technician who has the expertise and knowledge to use the manufacturing equipment to produce the physical PCB.

1. Gerber File

The Gerber file displays the track routing and the pin hole placements. The file was obtained using Manufacture> Bottom from the PCB Editor Menu bar. "Bottom" was selected in the available films (This is the side of the board with the routing and soldering pads. The top is the face through which the component legs are passed). Then clicking on "Create Artwork" produces the BOTTOM.art files (Fig V.1.1).
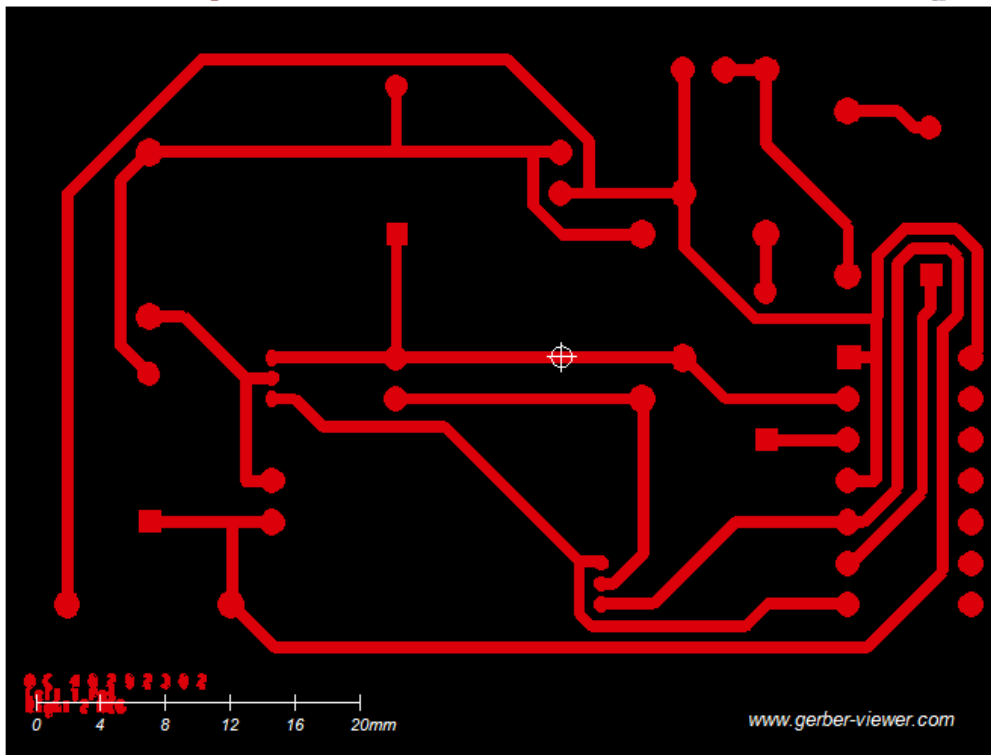

Fig V.1.1 Gerber File: Bottom.art

2. Drill file

This file creates a map, placing the component pin holes location. It can used in a machine to automatically drill the PCB.

It was obtained following these steps:

1. Selecting from the menu bar Manufacturing>NC>NCDrill parameters.
2. Selecting "Enhanced Excellon Format" then closing the window.
3. Selecting from the menu bar Manufacturing>NC>NCDrill.
4. Checking the drill file has a name.
5. Setting the scale ratio to 1.00.
6. Selecting Auto Tool Select.
7. Clicking on "Drill" to produce the drill file (Oscillator.drl).

3. Bill of Materials

The bill of material (Fig IV.1.1) was obtained in Capture. It lists the different components, their quantity, reference and part name or value.

Bill Of Materials April 24,2018 8:57:41 Page1			
Item	Quantity	Reference	Part
1	1	C1	100u
2	2	C2,C3	10u
3	2	D1,D2	D1N4148
4	2	D3,D4	MLED81
5	1	J1	HEADER 2
6	2	Q1,Q2	Q2N3904
7	2	R1,R2	3.9k
8	2	R3,R4	47k
9	2	R5,R6	360
10	1	U1	7400

Fig IV.1.1 Bill of Material.

V. Construction

The manufactured PCB (Fig V.1.1) and electronic components required for the On/Off Light circuit were prepared, by technician, and ready to be soldered.
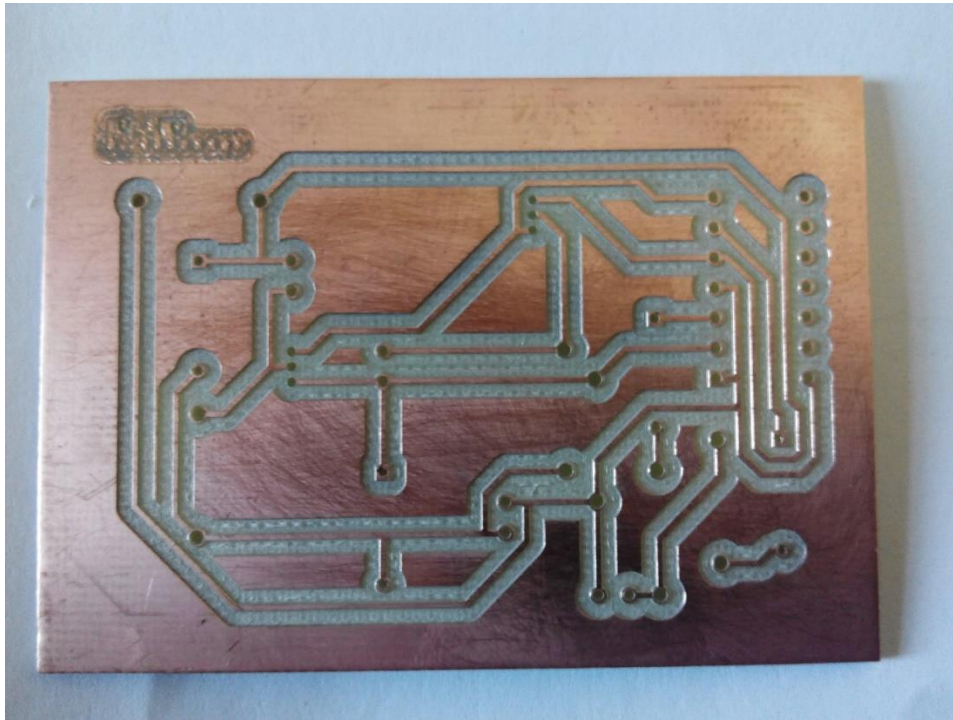

Fig V.1.1 PCB On/Off Light Circuit.

Using a soldering station set at 360°C, tools, and personal protections the PCB was assembled:

First, the IC holder was placed. Pin 1 and 14 were soldered first to hold it in place while the other pins received solder.

The resistors pins were bend using a pair of pliers. They then placed according to the schematics, BOM and measured values (using a multimeter).

In a similar manner the LEDs, diodes capacitors were placed and soldered. For these components special care had to be taken to respect the polarity of the component.

Then, the transistors were placed taking of the collector and emitter leg placement.

Two cables were soldered in the place of the jumper to be able and connect the board to a supply source.

Finally, the IC was mounted in its socket.

The circuit was connected to a supply and ready to be tested.

Conclusion

This report has shown the various steps taken to go from a hand-written schematic to a finish soldered functioning device using the OrCAD suite.
There were a lot of issue using OrCAD. The programme, when saving in the Capture module, seemed to be using split locations. My project file and design file were saved in two different locations. This created a few errors when coming back to the files. It was possible to determine if it was an issue with the user or with OrCAD.

When realising the simulation some component errors were found. First, the voltage from the supply would not accept "5Vdc" but only "5". Another error was regarding the header. This is not a component with PSpice equivalent but the component having no direct effect on the circuit purpose the error was ignored.

OrCAD is a programme requiring a lot of learning time spent on it to fully understand it's possibilities. OrCAD being a professional programme, it is not a requirement for it to be user-friendly. Possibly more recent versions offer a better user experience.
I believe there are cheaper alternatives to OrCAD available offering the same level of professional design tools (For example: Upverter by Altium).

Unfortunately, it was not possible for the technician to offer a demonstration of the manufacturing of a PCB. It would useful to spend time with the manufacturing equipment and understand the process.
The assembly on the board proved to be tedious with such small soldering pads. More soldering practice is required to do a neat job. Two resistors were of the wrong values and two others were missing. Replacements were found.

Overall, the experience has proven to be a good one despite the setbacks. It was an eye-opener on how to obtain relatively simple schematics, implement them in a CAD package to obtain manufacturing data files that would be used by a third party to produce a PCB that could then be assembled. The possibilities are now limitless.

Bibliography

Multicomp, "NPN Transistor TO-92", 2N3904 datasheet, Dec. 2012.