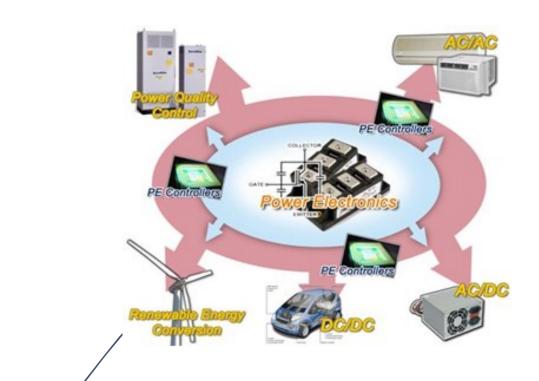2018/2019

POWER ELECTRONICS AND DRIVES LAB
LABORATORY MANUAL

By:

Osama Fuad

Nazmi Ashour

Table of Contents

INTRODUCTION TO THE LAB

This manual includes a collection of experiments that introduce the students to the principles of basic switching topologies used in power electronics and Drives systems. The primary objective of this laboratory is to provide the students with basic tools for analysis, building, and testing of power electronics converters by practical experimentation. The integration of power electronic converters with electric machines and control is one of the main aims of this laboratory. The laboratory also enhances the skills of building circuits on PCB boards through practicing and hands on experimentation, it also adds experience with common components such as motors, variable frequency drives, and power semiconductors.

LABORATORY SAFETY RULES

Laboratory safety is of paramount importance. That's exactly why this safety rules have been made, to encourage and promote safe and efficient working practices in the lab. On going attention and reinforcement of safety procedures is required when working in the lab for any reason. Without the right precaution taken, there's the significant risk of endangering yourself and others around you.

Emergency Response

1. It is your responsibility to read safety and fire alarm posters and follow the instructions during an emergency.
2. Know the location and function of all lab safety equipment including First Aid box and fire extinguisher.
3. Notify your instructor immediately after any injury, fire or explosion.

General Safety Rules

1. Read all directions for an experiment and follow the directions exactly as they are written, if in doubt, ask the teacher.
2. Always obtain permission before experimenting on your own.
3. Never handle any equipment unless you have specific permission.
4. Pay particular attention to moving or rotating objects like motors, gears and pistons.
5. Never eat, drink, or smoke while working in the laboratory.
6. You have long hair or loose clothes, make sure it is tied back or confined.

Electrical Safety

1. Obtain permission before operating any high voltage equipment.
2. Make sure electronic equipment is OFF when plugging or unplugging from an outlet.
3. Make sure the working area for electrical equipment is clean and dry.
4. Avoid using extension cords whenever possible. If you must use one, obtain a heavy- duty one that is electrically grounded, with its own fuse, and install it safely. Never, ever modify, attach or otherwise change any high voltage equipment.
5. Always make sure all capacitors are discharged before touching high voltage leads or the "inside" of any equipment even after it has been turned off. Capacitors can hold charge for many hours after the equipment has been turned off.

End of Laboratory activity rules

1. Clean all laboratory equipment and return to their locations
2. Unplug and store properly any electrical device.
3. Wash your hands after every experiment.
4. Clean up your work area before leaving.

EXPERIMENT 1: INTRODUCTION TO SOLDERING

OBJECTIVES

In this experiment, students will be introduced to basic Soldering equipment and they will practice basic soldering techniques of components onto printed circuit boards PCB. In addition, students will learn how to inspect a soldered connection and printed circuit connections.

INTRODUCTION

Soldering is an old technique for creating permanent electrical and mechanical connections between metals. In contrast to ordinary glueing, which makes only a physical connection between materials (although electrically conducting glue also exists) soldering causes a chemical reaction with other materials, creating a new alloy. There are several soldering techniques, which all have a few things in common: basic metals, soldering flux, solder and heating.

It is recommended in this lab to understand how soldering techniques are used and to get familiar with the materials used during the soldering process, and we have to learn how to inspect a soldered connection.
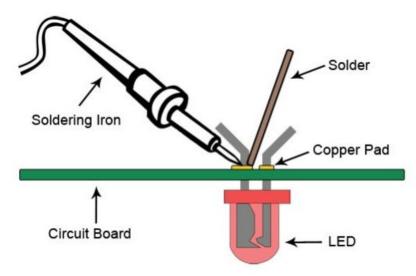

Figure 1: Through hole soldering.

SOLDERING TOOLS

Soldering Iron: soldering iron is a hand tool that plugs into a standard 120v AC outlet and heats up in order to melt solder around electrical connections.

Figure 2: Soldering Iron

Soldering Iron Stand: A soldering iron stand is very basic but very useful and handy to have. This stand helps prevent the hot iron tip from coming in contact with flammable materials or causing accidental injury to your hand.

Figure 3: Soldering Iron Stand

Solder: Solder: is a metal alloy material that is melted to create a permanent bond between electrical parts. It comes in both lead and lead-free variations with diameters of .032″ and .062″ being the most common. Inside the solder core is a material known as flux which helps improve electrical contact and its mechanical strength.

Figure 4: Soldering Iron Stand

Most of the soldering done in this Lab is through-hole. A through-hole joint is a type of soldering joint in which the component joins with the PCB pad through a physical hole in the board. The following steps will illustrate how to make a proper through hole solder joint on a PCB.

Reference: Soldering Guidelines, Revision 01, EE400/1 Design

1. Ensure that the printed circuit board and all components are clean. Cleaning can be achieved with a mild abrasive and/or the application of flux.

2. Plug in the soldering iron, turn it on, and let it warm up for 2–3 minutes.

3. Wet the soldering station sponge with the water provided in the lab. Do not wet the sponge in the bathroom or the water fountain.

4. Clean the tip of the soldering iron and tin it with solder.

5. Insert the component into the holes. Ensure that the component is secure by taping the component or by using a third hand. Optionally, the component leads can be clinched as shown in Fig. 1.1. This technique, however, is not recommended for two-sided boards as the flow of solder to the component side is restricted.
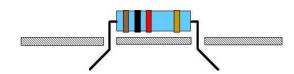

Figure 5: Through hole soldering of resistor.

6. Apply the soldering iron tip to one side of joint making contact with the component lead and the board copper foil, ensuring that both are heated up to the same temperature as shown in Fig. 1.2 and Fig 1.3. Notice the tinned tip in Fig. 1.3.

7. Slowly add a few millimetres of solder to the other side of the joint. DO NOT apply solder to the soldering iron tip. If enough heat was applied to the PCB pad and component wire, the solder will flow freely onto the joint.
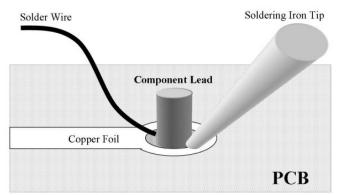

Figure 6: Method of Soldering Through Hole Joints

8. Remove the solder when the joint is suitably covered as shown in Fig. 1.4. The goal is to get the joint to be a "fillet": a curve as shown in Fig. 1.4.

9. If the PCB is double-sided, the solder should flow through the hole around the component lead and make a bond on the component side of the board (opposite to the side that the solder was applied). If this "wicking" does not occur, the hole may be undersized, clinching could be blocking the solder's path, or the component lead is not clean.

10. Remove the soldering iron and allow the joint to cool naturally.
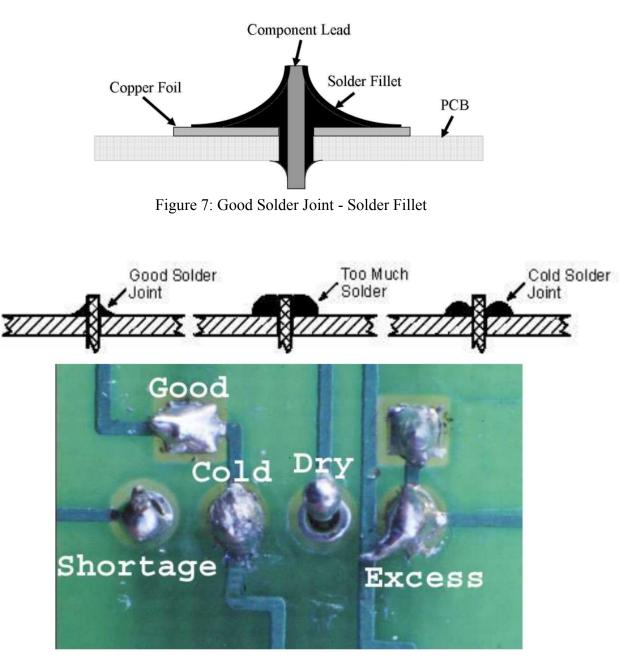
11. Cut the lead of the component, if necessary.

Figure 7: Good Solder Joint - Solder Fillet

Figure 8: Joint Examples,

JOINING TWO WIRES

1. Slide shrink wrap on one of the wires.
2. Tin or coat each wire with some solder.
3. Join the wires in a way that provides a good mechanical connection (such as twisting together) and apply the soldering iron tip and some solder.
4. Let the joint cool.
5. Slide shrink wrap over the joint and heat the shrink wrap with a heating gun or a heat source until it the the wrap shrinks around the joint.

Figure 9: Soldering two wires.

TESTING CONNECTIONS

After completely soldering a component to a PCB, it is good practice to ensure connectivity between the component wire leads and the PCB pads they are soldered to. A DMM (Digital Multi-Meter) is sufficient to deter-mine connectivity; many DMMs include an audible connectivity setting, but failing this, measure the joint resistance.

EXPERIMENT 2: HALF WAVE UNCONTROLLED RECTIFIER

OBJECTIVES

This experiment aims to study the operation of single phase uncontrolled rectifier., and understand the effect of filters on the output voltage of rectifier. It also provides you with basics of the power supply topologies.
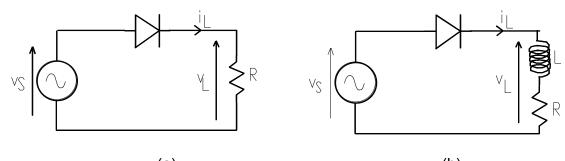
THEORY

Rectifiers specially refer to power electronic converters where the electrical power flows from the ac side to the dc side. During this process, an ac voltage or current source is converted to dc voltage and current. Rectifier is the heart of power supply circuit where an ac voltage source is firstly stepped down using transformers before it is converted to dc using rectifiers, after this the produced dc signal is filtered using capacitors.

Single phase rectifiers have two configurations: the half wave rectifiers and the full wave rectifiers, in the case of single phase half wave rectifier configuration only the positive half of the AC input waveform is passed while the negative one is eliminated. The circuit of the half wave rectifier is shown in Fgiure 1 for R and RL load.

The circuit of full wave rectifier is shown in Figure 2, its configuration of diodes provides full wave rectification because for both positive and negative swings of the supply two diodes are forward biased while the other two are reverse biased. Thus instead of the single diode in the half wave rectifier, there are two diodes in the conduction path.

During the positive half cycle, diodes D1 and D4 are forward biased while diodes D2 and D3 are reverse biased. Then for the positive half cycle of the input waveform, current flows along the path of: D1 –RL –D4 and back to the supply.
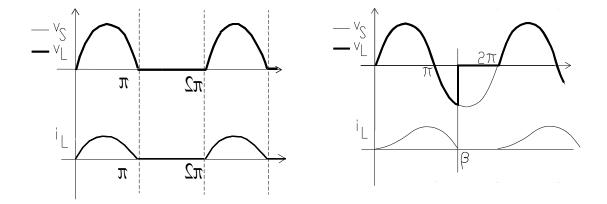
Figure 1: Single-phase, half-wave diode rectifier for R and RL load.
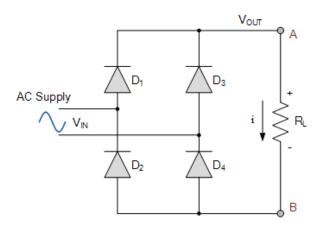

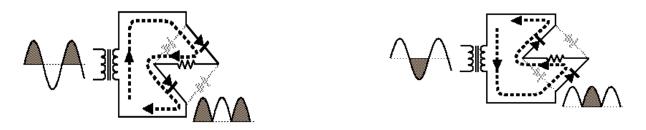Figure 2: Single-phase, full-wave uncontrolled rectifier.

Figure 3: Both conduction paths through the load resistor

APPARATUS:

- AC voltage regulator VARIAC
- Transformers.
- Oscilloscope
- Multimeter
- PCB Board
- Soldering iron.
- Power Diodes
- Universal Motor
- Resistors: Various
- Capacitors: Various

PROCEDURE

CAUTION! – High voltages are present in this Experiment. DO NOT make any connections with the power supply ON. Get in the habit of turning OFF the power supply after every measurement.

Unfiltered Rectifier

1. Connect the circuit as shown in the Figure 2.
2. Verify that the transformers given to you are well working. You should do this with the AC voltage setting on the multimeter.
3. Using Oscilloscope obtain the output voltage of the bridge and measure the ripple. It is a good idea to AC couple the oscilloscope so that the DC component is removed and the ripple is clearly visible.
4. Measure the DC output voltage.
5. Compare your results with the theoretical values.
6. Connect your circuit to operate the universal motor.

Filtered Rectifier

1. Add the capacitor to the circuit, make sure to switch the power supply off before any changes to the circuit.
2. Try to measure the ripple voltage. It may be below the resolution of the oscilloscope.
3. Measure the DC output voltages with a Multimeter or on the scope.
4. Connect your circuit to operate the universal motor. What do you notice compared to the previous part.

EXPERIMENT 3: THREE PHASE UNCONTROLLED RECTIFIER

OBJECTIVES

This experiment aims to study the operation of three phase uncontrolled rectifier, and understand the effect of filters on the output voltage of rectifier.

THEORY

In this experiment, we will build and study three phase rectification circuits which are commonly used in many industrial applications such as motor control or battery charging where higher power is required.

The three phase bridge rectifier shown in Figure 1, the conduction process always occurs in the most positive diode that is connected to the highest instantaneous voltage and the corresponding one with the most negative voltage. Thus conduction is passed from diode to diode as the three phases rotate across the terminals, the diodes conduction sequence is given as: D_{1-2} D_{1-6} D_{3-6} D_{3-6} D_{3-4} D_{5-4} D_{5-2} and D_{1-2}.
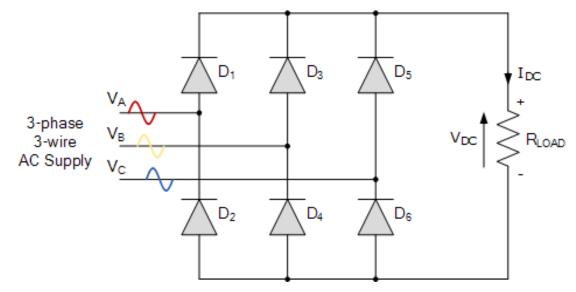

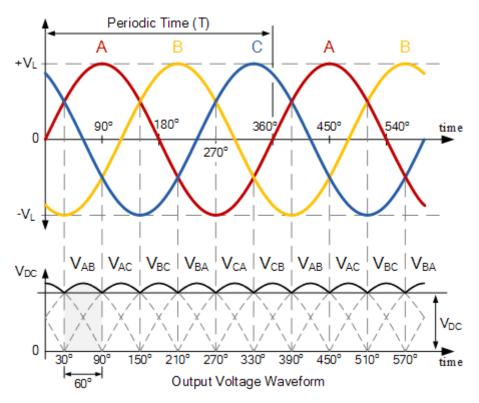Figure 1: Circuit diagram for three-phase full wave rectifier.

Figure 2: input/output waveforms of the three-phase full wave rectifier.

The output voltage of 3 phase full wave rectifiers is maintained between the peak voltage and 86.6% of that value that is why the output voltage appears regulated. The ripples in three phase rectifiers have a high frequency. Thus they can be easily filtered.

The average value of the output voltage may be got by averaging over a single output pulse:

$$V_{avg} = \frac{\int_{-\pi/6}^{\pi/6} \sqrt{3}V_m \cos(\omega t) d\omega t}{2\pi/6} = \frac{3\sqrt{3}}{\pi} V_m$$

Where V_m is the Peak line to neutral voltage, also we can calculate the RMS output voltage by integrating over a single pulse:

$$V_{avg} = \sqrt{\frac{\int_{-\pi/6}^{\pi/6} (\sqrt{3}V_m \cos(\omega t) d\omega t)^2}{2\pi/6}} = V_m \sqrt{\frac{3}{2} + \frac{9\sqrt{3}}{4\pi}}$$

APPARATUS:

- AC voltage regulator VARIAC
- Transformers.
- Oscilloscope
- Multimeter
- PCB Board
- Soldering iron.
- Power Diodes
- Universal Motor
- Resistors: various
- Capacitors: various

PROCEDURE

CAUTION! – High voltages are present in this Experiment. DO NOT make any connections with the power supply ON. Get in the habit of turning OFF the power supply after every measurement

Unfiltered Rectifier

7. Connect the circuit of the 3 phase rectifier on PCB board as shown in the Figure 1. The input voltage V_{in} is a 100 V_{rms}, 50 Hz. V_{in} is stepped down from line voltage (50 Hz and 380 V_{rms}) using a transformer.

8. Using Oscilloscope obtain the output voltage of the bridge and measure the ripple.

9. Find the frequency of the ripple in the output waveform.

10. Measure the DC output voltage.

11. Compare your results with the theoretical values.

12. Plot the waveforms you obtained in the oscilloscope.

13. Run the universal motor using the rectifier you have built.

Filtered Rectifier

5. Add the capacitor to your circuit, make sure to switch the power supply off before any changes to the circuit.

6. Try to measure the ripple voltage. It may be below the resolution of the oscilloscope.

7. Measure the DC output voltages with a Multimeter.

8. Connect your circuit to operate the universal motor. What do you notice compared to the previous part.

	The University of Jordan School of Engineering Mechatronics Engineering Department Power Electronics & Drives Lab	

EXPERIMENT 4: HALF WAVE CONTROLLED RECTIFIER Using SCR

OBJECTIVES

This experiment aims to build and study the operation of single phase controlled rectifier for R load using RC firing circuit. You also have to know the effects of delay/firing angle on the performance of the controlled rectifier.

THEORY

Phase controlled AC-DC converters employing thyristors are extensively used for changing constant ac input voltage to controlled dc output voltage. In phase controlled rectifiers, a thyristor is tuned off as AC supply voltage reverse biases it, provided anode current has fallen to level below the holding current. Controlled rectifiers have a wide range of applications, from small rectifiers to large high voltage direct current (HVDC) transmission systems. They are used for electrochemical processes, many kinds of motor drives, traction equipment, controlled power supplies, and many other applications.
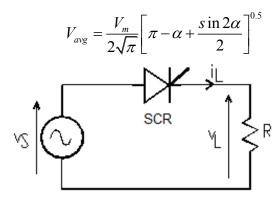
Single-Phase Half-Wave Controlled Rectifier As shown in Figure 1, the single-phase half wave rectifier uses a single thyristor to control the load voltage. The thyristor will conduct, ON state, when the voltage V_T is positive and a firing current pulse I_G is applied to the gate terminal. Delaying the firing pulse by an angle α does the control of the load voltage.

In Figure 2, the output voltage is plotted at two different values of angle α. The load in Figure 2 is resistive and thus the current has the same waveform as the load voltage. The thyristor goes to the non-conducting condition, OFF state, when the load voltage and, consequently, the current try to reach a negative value.

The load average voltage is given by:

$$V_{avg} = \frac{V_m}{2\pi}\left[1 + \cos\alpha\right]$$

The output voltage RMS value is given by:
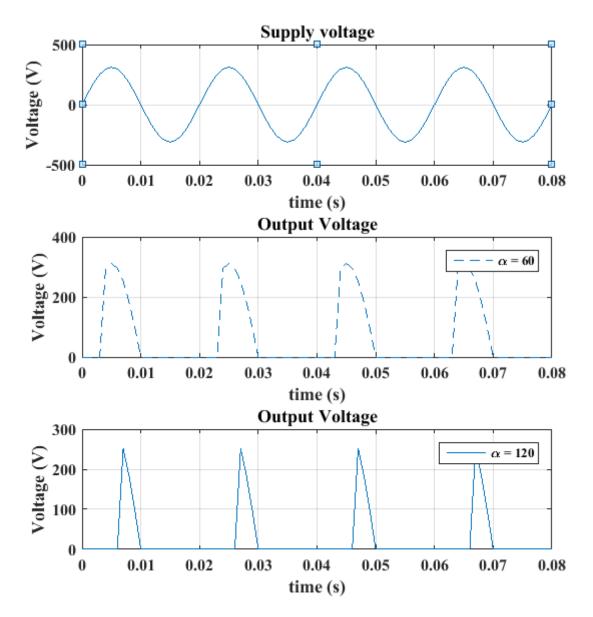
$$V_{avg} = \frac{V_m}{2\sqrt{\pi}} \left[\pi - \alpha + \frac{\sin 2\alpha}{2} \right]^{0.5}$$

Figure 1: half wave controlled rectifier using SCR.

Figure 1: output voltage waveform at different firing angles.

RC FIRING CIRCUIT

The firing angle α can be changed using the variable resistance VR1 and the capacitor C1. The capacitor C1 charges up at the start of each cycle until its voltage is sufficient to trigger SCR into conduction i.e. the capacitor voltage equals to the gate threshold voltage. Diode D2 is used to prevent the negative voltage to gate.

Figure 3: Half wave controlled rectifier using RC firing angles.

APPARATUS:

- AC voltage regulator VARIAC
- Oscilloscope
- Multimeter
- PCB Board
- Soldering iron.
- Thyristor BT151
- Potentiometer 20 KΩ
- Universal Motor
- Resistors: 47KΩ , 1 KΩ
- Capacitors: 10μF, 2μF

PROCEDURE

CAUTION! – High voltages are present in this Experiment. DO NOT make any connections with the power supply ON. Get in the habit of turning OFF the power supply after every measurement

1. Complete the connections as per the circuit diagram in Figure 3. The pin assignment of the BT151 thyristor is shown below in Figure 4.
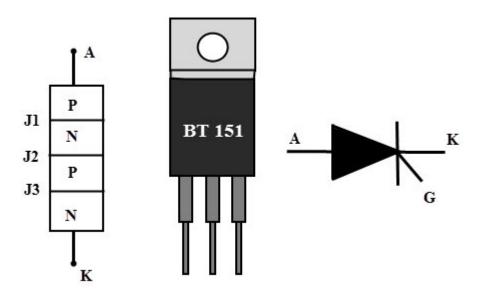

Figure 4: BT151 Thyristor pin assignment.

2. Keep the potentiometer VR1 in minimum position (maximum resistance in the circuit)
3. After your instructor permission, turn on the power switch.
4. Using the Oscilloscope, observe the output voltage waveform, SCR voltage and gate signals for various firing angles.
5. Measure the average output voltage for the corresponding firing angle.
6. Calculate the theoretical output voltage for the corresponding firing angle.
7. Fill the table below, and plot the control characteristic of the rectifier (the curve V_o as function of α is called the control characteristic curve).

The firing angle α	Measured output voltage V_{avg}	Theoretical output voltage

EXPERIMENT 5: AC Voltage Control Using TRIAC

OBJECTIVES

This experiment aims to study and understand the characteristics of AC-AC converter using TRIAC-DIAC combination. The output will be observed by the variation of intensity of AC lamp light with reference to changing the firing angle. The obtained output voltages will be compare to the theoretical values.

THEORY

Phase controlled AC-AC converters employing TRAICS are commonly used for changing constant ac input voltage to controlled ac output voltage. In AC voltage control, the RMS value of AC voltage applied to a load circuit is controlled by controlling the triggering angle of the TRIAC in the ac voltage controller circuits. TRIAC speed control circuit is suitable for AC motor speed control, lamp dimmers and electrical heater control.

In the circuit shown in Figure 1, the amount of voltage applied to the load is controlled by the variable resistor VR1 which is used to change the amount of phase shift on the gate of the TRIAC. The VR1 – C1 combination determines the TRIAC's triggering voltage, the value of VR1 and C1 are selected to give a firing angle range of nearly 0 & 180. The DIAC is a bidirectional semiconductor device that helps provide a sharp trigger current pulse to fully turn ON the TRIAC.

The capacitor C1 charges up at the start of each cycle until its voltage is sufficient to trigger the DIAC into conduction, as a consequence, capacitor discharges into the gate of the TRIAC turning it "ON". Once the TRIAC is triggered into conduction at firing angle α, the source voltage Vs appears across the load. At the end of the half-cycle, the TRIAC turns-OFF automatically and the RC triggering process continues on the next half cycle.

R1 resistor is added to protect the DIAC and TRIAC gate from getting exposed to the full supply voltage when VR1 is zero, while R2 limits the current in the DIAC and TRIAC gate when DIAC turns ON.
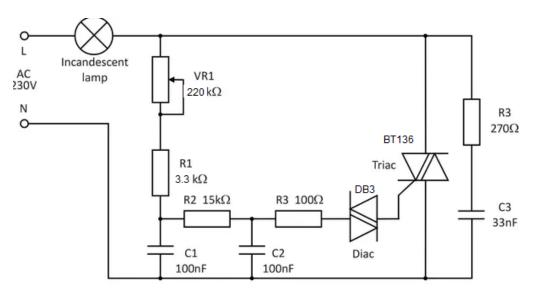

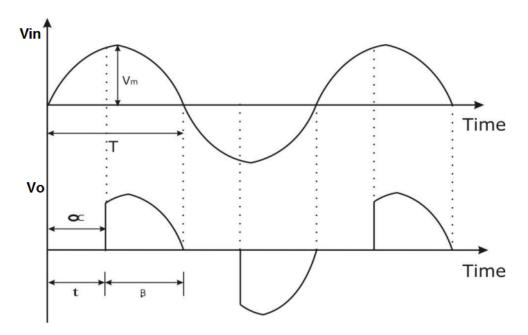Figure 1: AC Voltage Control circuit using TRIAC-DIAC combination.

Figure 2: input output waveforms for the AC voltage control circuit.

The output voltage RMS value is given by:

$$\frac{V}{\sqrt{2}}\sqrt{\pi - \alpha + 2\sin(2\alpha)}$$

APPARATUS:

- AC voltage regulator VARIAC
- Oscilloscope
- Mulitimeter
- PCB Board
- Soldering iron.
- TRIAC BT13
- DIAC DB3
- Potentiometer 220 KΩ
- Resistive Load (Lamp)
- Resistors: 3.3KΩ , 15 KΩ, 100Ω
- Capacitors: 100nF

PROCEDURE

 CAUTION! – High voltages are present in this Experiment. DO NOT make any connections with the power supply ON. Get in the habit of turning OFF the power supply after every measurement

1) Connect the circuit as shown in Figure 1. The pin assignment of the BT136 TRIAC is shown below.

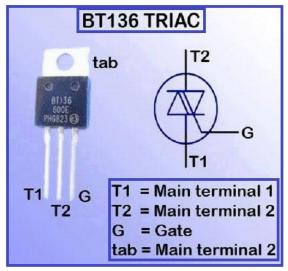
Figure 3: pin assignment of the BT136 TRIAC

Figure 4: DB3 DIAC

2) Using the potentiometer, change the firing angle and observe the intensity of the lamp light.

3) Vary the firing angle from 0 to 180.

4) Measure the average output voltage for the corresponding firing angle.

5) Calculate the theoretical output voltage for the corresponding firing angle.

6) Fill the table below:

The firing angle α	Measured output voltage V_{avg}	Theoretical output voltage

7) Plot a graph for the output voltage versus firing angle also plot the theoretically calculated results on the same graph.

EXPERIMENT 6: Buck Converter

OBJECTIVES

This experiment aims to study and understand the characteristics of DC-DC buck converter. The circuit will be operated in open loop conditions (no feedback). Our main goal will be to compare the theoretical results with the experimental results.

THEORY

The buck converter is the most widely used DC-DC converter topology in power management and microprocessor voltage-regulator (VRM) applications. Those applications require fast load and line transient responses and high efficiency over a wide load current range. They can convert a voltage source into a lower regulated voltage. For example, within a computer system, voltage needs to be stepped down and a lower voltage needs to be maintained. For this purpose the Buck Converter can be used. Furthermore buck converters provide longer battery life for mobile systems that spend most of their time in "stand-by". Buck regulators are often used as switch-mode power supplies SMPS for baseband digital core and the RF power amplifier.

The converter itself consists of one active controlled switch, a rectifier and filter elements. This great simplicity allows for cost effective high efficient power distribution throughout the application. The buck converter has the filter inductor on the output side, which provides a smooth continuous output current waveform to the load. This could be considered a qualitative benefit but requires special considerations for big load transients.

Figure 1 shows the circuit diagram of step down DC-DC Buck converter. When switch S (which in practice can be, e.g., a ***MOSFET***) is ON, diode D is reverse biased and voltage across inductor will be Vs-Vo (Figure 3). So inductor current increases and attains a peak. When switch S is OFF, diode D is forward biased and voltage across inductor will be -Vo. So inductor current decreases. Hence inductor with diode ensures an uninterrupted current flow and hence a constant output voltage with capacitor for removing voltage ripples.

Output voltage is given by: **_Vo=D Vs_**, Where D is duty cycle and V_S is input voltage
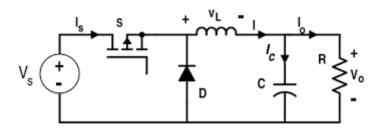

Figure 1: Circuit Diagram for Buck Converter.

The MOSFET switch is pulse width controlled at a constant clock-pulse rate by varying the width of the switch on pulses (operating interval).

Figure 2: Inductor Voltage in Continuous Mode

$$i_{L,max} = i_{L,avg} + \frac{\Delta I}{2}$$

$$i_{L,avg} = I_{out}$$

$$i_{L,min} = i_{L,avg} - \frac{\Delta I}{2}$$

Figure 3: Inductor Current Waveform for Continuous Mode
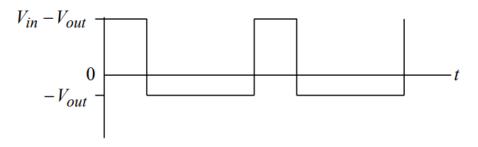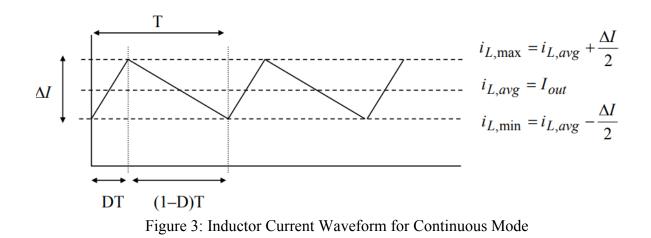
The inductor ripple ΔI is given by $\Delta I = \dfrac{V_{in} - V_{out}}{L} DT = \dfrac{V_{in} D(1-D)}{f_s L}$

Where: f_s is the switching frequency. Taking the derivative of with respect to D and setting it to zero shows that ΔI is maximum when D = 50%. Thus,

$$\Delta I_{max} = \frac{V_{in}}{4 f_s L}$$

The buck converter can operate in different modes; continuous conduction mode (CM) and discontinuous mode (DCM). In CM, Current flows continuously in the inductor during the entire switching cycle in steady state operation. In most Buck regulator applications, the inductor current never drops to zero during full-load operation. Overall performance is usually better using continuous mode, and it allows maximum output power to be obtained from a given input voltage and switch current rating.

In the DCM the current in the inductor falls to zero and remains at zero for some portion of the switching cycle. It starts at zero, reaches a peak value, and returns to zero during each switching cycle. In applications where the maximum load current is fairly low, it can be advantageous to design for discontinuous mode operation. In these cases, operating in discontinuous mode can result in a smaller overall converter size (because a smaller inductor can be used). Often the output capacitor must be large to keep the voltage constant.
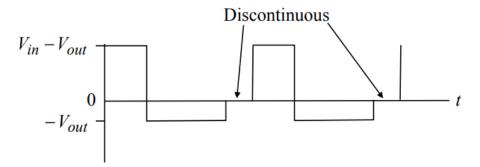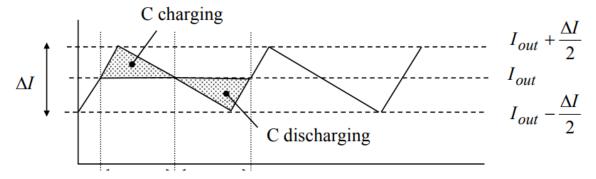
Figure 4: Inductor Voltage in Discontinuous Mode.

Figure 5: Capacitor Charging and Discharging Intervals in Continuous Mode

Figure 6: Capacitor Voltage in Continuous Mode

$$i_{L,\max} = i_{L,avg} + \frac{\Delta I}{2}$$

$$i_{L,avg} = I_{out}$$

$$i_{L,\min} = i_{L,avg} - \frac{\Delta I}{2}$$

Figure 7: Diode Current Waveform for Continuous Mode

DESIGN OF CIRCUIT COMPONENTS

❖ Input voltage, V_S=25V

❖ Output Voltage required V_O=12 V

❖ Output current I_O= 0.5 A

❖ Switching frequency, f_S=10 kHZ

❖ Ripple allowed in inductor current, ΔI = 30%

Note: A good estimation for the inductor ripple current is 20% to 40% of the output current.

❖ Ripple allowed in capacitor voltage, ΔV_c= 150 mV

inductor $L = \dfrac{V_0(1-D)}{f_s \Delta I}$

capacitor $C = \dfrac{V_0(1-D)}{8 f_s^2 \Delta V_c L}$

Load Resistance $R = \dfrac{V_o}{I_o}$

The Diode forward current rating needed is equal to:

$I_D = I_o(1-D)$

26

PROCEDURE

Connect the circuit in Figure 1 and take the following measurements:

1) VARYING DUTY RATIO

1. Set the duty ratio at 50%, switching frequency at 10 kHz and R_L= 24Ω.
2. Observe and plot the output voltage and voltage across the diode.
3. Vary the duty ratio from 10 % to 90 % (in steps of 10%).
4. Measure the average output voltage for the corresponding duty ratio.
5. Calculate the theoretical average output voltage for the corresponding duty ratios.
6. Plot a graph for the output voltage versus duty ratio, also plot the theoretically calculated results on the same graph. Compare the two plots and comment about how the buck converter works as a variable dc step down transformer.
7. Plot the obtained inductor current and the capacitor current waveforms. Explain the relation between the two currents. Comment on the ripple in the inductor current.

2) VARYING SWITCHING FREQUENCY

1. Set the duty ratio to 50 %.
2. Measure the peak-peak output ripple voltage and the peak-peak ripple in inductor current.
3. Repeat the above procedure for different switching frequencies (5 kHz, 10 kHz, 40 kHz). Make sure that output voltage is maintained at 12V.
4. Observe and plot the inductor current and capacitor current waveforms at 10 kHz .
5. Plot the peak-peak ripple in the output voltage versus switching frequency.

3) VARYING LOAD

1. Set the switching frequency at 10 kHz and duty ratio at 50%.
2. Set the load resistance R_L=24Ω.
3. Increase the load resistance and observe the inductor current waveform.
4. Keep increasing the load resistance, until the buck converter enters discontinuous current mode operation.
5. Note down the average inductor current value and make a copy of the inductor current waveform when the converter starts entering discontinuous current mode of operation.

4) EFFICIENCY OF BUCK CONVERTER:

Determine the efficiency of the buck converter at frequency of 10 kHz.

1. Set duty ratio at 50%.
2. Set load resistance RL=24Ω.
3. Measure the average output voltage.
4. Measure the average output current Io.
5. Measure the average input voltage V_{in}.
6. Measure the average input current I_{in}.
7. Calculate the efficiency of the buck converter for the above frequency.
8. Comment on the difference in efficiency results obtained for two switching frequencies.

EXPERIMENT 7: Variable Frequency Drive

OBJECTIVES

In this experiment you will understand the principle of operation of the variable frequency drive and how it is used to control the speed of a squirrel cage induction motor (SCIM). In addition you will understand how the frequency and the voltage to the induction motor should be varied in order to control the speed and to prevent the motor from saturating.

THEORY

The variable frequency drive comprises two main sections: the uncontrolled six pulse rectifier (built using 6 diodes) that produces the dc voltage on the dc link capacitor, and the inverter section (built using 6 IGBT and parallel diodes) that converts the dc to an ac voltage at the induction motor terminals.

The 6 IGBTs are switched using the principle of pulse width modulation (PWM). This allows the inverter to achieve the required frequency to change the speed of the motor by changing the output switching frequency. The method of triangulation is used in order to decide the switching times of the six IGBT's. The modulation index is changed in order to change the output voltage.

When changing the output frequency of the variable frequency drive, it is necessary to change the output voltage as well. This is due to the fact that the equivalent circuit of the squirrel cage induction motor comprises inductors. The reactance of an inductor is proportional to frequency. Thus, if the frequency is reduced without reducing the voltage, the current in the inductor will increase; and for the induction motor, the current will increase and saturate the core (leading to an excessive increase in the heat of the motor).

Figure 1 below shows the procedure that should be followed in changing the frequency and the voltage when varying the output frequency of the variable frequency drive.
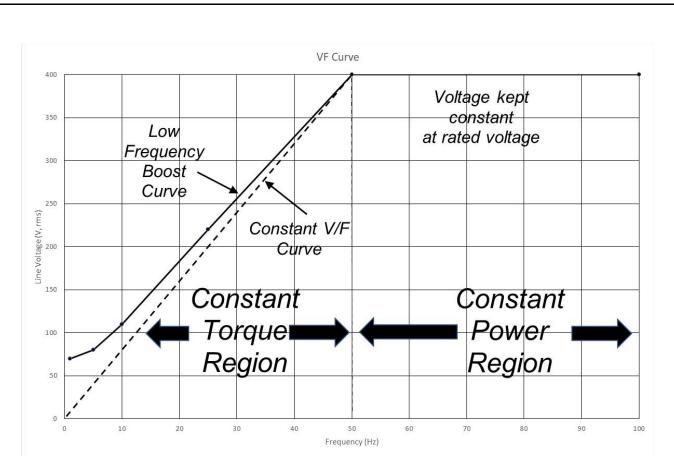
Figure 1: The theory of the VF curve as applied in variable frequency drive systems.

However, the low frequency boost in the Danfoss Variable frequency drive that is used in the lab works in a slightly different way. This is shown in Figure 2. It is explained as follows:

1. Find the rated voltage of the motor (380 V in this case).
2. Decide on the amount of the low frequency boost voltage percentage (e.g., 2%).
1. The maximum allowable boost is 40%.
2. Calculate the amount of boost in volts at zero Hz. For example, if the low
3. frequency boost percentage is 2%, then the number of volts at a frequency of zero Hz is: 2% x 380= 7.6 V.
4. A straight line is then drawn from the rated voltage and frequency point (i.e., 50 Hz and 380 V) to the 0 Hz point (i.e., 0 Hz and 7.6 V).

APPARATUS:
- Three phase induction Motor.
- Variable Frequency Drive.
- Mulitimeter
- Brake
- Speed and Torque Measurement Unit

PROCEDURE

CAUTION! – High voltages are present in this Experiment. DO NOT make any connections with the power supply ON. Get in the habit of turning OFF the power supply after every measurement

In the connections for this experiment, you will connect a three phase supply to the input of the VF drive (see Figure 3 for a general overview). The three output lines will be connected from the output of the variable frequency drive to the delta connected three phase induction motor Couple the induction motor to the power brake in order to apply torque to it an measure the speed.

Figure 3: General overview of the connections of the experiment.

1. Now set the frequency to 50 Hz and run the system. Start gradually increasing the load torque

2. Measure the output voltage from the variable frequency drive (should be 380 V AC).
   ➢ **f = 50 Hz.**

#	Torque (N.m)	Speed (rpm)	Line Current (A)
1	0		
2	0.25		
3	0.5		
4	0.75		
5	1		
6	1.25		
7	1.5		

3. Now low frequency voltage boost
4. Measure the output voltage from the variable frequency drive (should be 76 V AC)

> **f=10 Hz**

#	Torque (N.m)	Speed (rpm)	Line Current (A)
1	0		
2	0.25		
3	0.5		
4	0.75		
5	1		
6	1.25		
7	1.5		

5. Now add the low frequency boost at a value of 2%

6. Measure the output voltage from the variable frequency drive (should be approximately 82 V AC).

#	Torque (N.m)	Speed (rpm)	Line Current (A)
1	0		
2	0.25		
3	0.5		
4	0.75		
5	1		
6	1.25		
7	1.5		

7. Plot the speed torque curves for the induction motor at 50 Hz, and 10 Hz under two conditions: zero voltage boost and 2% voltage boost.

8. Disucss the reasons for the difference of the speed torque curve under the two different voltage boosts.

EXPERIMENT 8: Regulated Power Supply

INTRODUCTION

The principles of negative feedback are employed to keep the output voltage constant (almost constant) despite changes in the load current and line voltage.

Operation of Series Feedback Voltage Regulator

As you can see in the circuit diagram, we feed the unregulated d.c. supply to the voltage regulator. The function of the circuit is to maintain constant output voltage despite variations in load or input voltage.

Case 1 : Output Voltage Increases

Say, for any reason, the output voltage increases resulting . As a result the voltage across KL (*i.e.*, $R2$) increases. This in turn means that more VF is fed back to the base of transistor $Q1$; producing a large collector current of $Q1$. This causes base voltage of $Q2$ to decrease as most of this collector current flows through $R3$. Hence we get less output voltage . In other words the increase in voltage is offset, keeping the output voltage constant.

Case 2: Output Voltage Decreases

Say, for any reason, the output voltage tries to decreases. As a result the feedback voltage VF also decreases. Hence the current through $Q1$ and $R3$ reduces. Which actually means that we get more base voltage at $Q2$ and more output voltage. Consequently, keeping the output voltage at the original level.
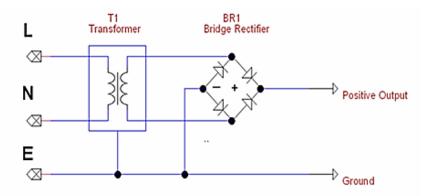
APPARATUS:

- VARIAC
- Single phase transformer 0-12V.
- Bridge rectifier, 5A
- Capacitor 1000UF, 25V
- Transistor, tip 41
- Transistor, BC 536
- VR =1KΩ, R=470Ω, 1.2kΩ, 820Ω,1kΩ
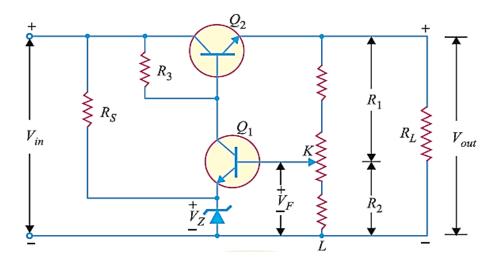- Zener Diode .5.6V.
- PCB or Breadboard

PROCEDURE

CAUTION! – High voltages are present in this Experiment. DO NOT make any connections with the power supply ON. Get in the habit of turning OFF the power supply after every measurement

Connect the circuit diagram of series feedback voltage regulator in the figure below.

Voltage Regulation Circuit:

All the load current passes through the transistor $Q2$ and that's why it is called a pass transistor. The voltage divider consisting of $R1$ and $R2$ is the sample and adjust circuit. The function of the voltage divider is to sample the output voltage and then deliver a negative feedback voltage to the base of $Q1$. The collector current of $Q1$ is controlled by the feedback voltage VF.

Output Voltage. The voltage divider $R_1 - R_2$ provides the feedback voltage.

$$\text{Feedback fraction, } m = \frac{V_F}{V_{out}} = \frac{R_2}{R_1 + R_2}$$

$$\text{Closed loop voltage gain, } A_{CL} = \frac{1}{m} = \frac{R_1 + R_2}{R_2} = 1 + \frac{R_1}{R_2}$$

Now $$V_F = V_Z + V_{BE}$$

or $$m V_{out} = V_Z + V_{BE} \qquad (\because V_F = m V_{out})$$

or $$V_{out} = \frac{V_Z + V_{BE}}{m}$$

or $$V_{out} = A_{CL} (V_Z + V_{BE}) \qquad (\because 1/m = A_{CL})$$

Therefore, the regulated output voltage is equal to closed-loop voltage gain times the sum of zener voltage and base-emitter voltage.

Results:

....
....
....
....
......

APPENDIX

The appendix contains datasheets for various components used during our experiments in the lab. For example BT136 TRIAC, IRF540 MOSFET Transistor and others.