LABORATORY MANUAL

of

AEL332
POWER ELECTRONICS LAB(S6)

DEPARTMENT OF ELECTRONICS &
COMMUNICATION ENGINEERING

COLLEGE OF ENGINEERING, TRIVANDRUM

DEPT. OF ELECTRONICS & COMMUNICATION ENGINEERING

COLLEGE OF ENGINEERING TRIVANDRUM

CERTIFICATE

This is a controlled document of **Department of Electronics & Communication of College of Engineering, Trivandrum.** No part of this can be reproduced in any form by any means without the prior written permission of the Head of the Department, Electronics & Communication, College of Engineering, Trivandrum. This is prepared as per **2019 KTU B.Tech** Electronics & Communication scheme.

AEL332	POWER ELECTRONICS LAB	CATEGORY	L	T	P	CREDIT
		PCC	0	0	3	2

Preamble:

This course aims to

i. Familiarize the students with different types of power electronics circuits and application

ii. Familiarize students with simulation of basic power electronics circuits

Prerequisite: ECL202 - ANALOG CIRCUITS AND SIMULATION LAB

(i)Course Outcomes: The student will be able to

CO1	Determine the characteristics of SCR and design triggering circuits for SCR based circuits
CO2	Design, set up and analyse single phase AC voltage controllers
CO3	Design, set up and test suitable gate drivers for MOSFET/IGBT
CO4	Design, set up and test basic inverter topologies
CO5	Design and set up dc-dc converters
CO6	Develop simulation models of dc-dc converters, rectifiers, and inverters using modern simulation tools

(ii) CO-PO/PSO matrix showing level of correlation (1-Low, 2-Medium, and 3-high)

Course: AEL332 Power Electronics Lab

Course Outcome Designation	PO-1	PO-2	PO-3	PO-4	PO-5	PO-6	PO-7	PO-8	PO-9	PO-10	PO-11	PO-12	PSO-1	PSO-2	PSO-3
CO1	3	3	2	2	2			2	3	2		3	3	2	2
CO2	3	3	2	2	2			2	3	2		3	3	2	2
CO3	3	3	2	2	2			2	3	2		3	3	2	2
CO4	3	3	2	2	2			2	3	2		3	3	2	2
CO5	3	3	2	2	2			2	3	2		3	3	2	2
CO6	3	3	2	2	3			2	3	2		3	3	2	2

Assessment

Mark distribution

Total Marks	CIE	ESE	ESE Duration
150	75	75	2.5 hours

Continuous Internal Evaluation Pattern:

Attendance	:	15 marks
ContinuousAssessment	:	30 marks
Internal Test (Immediately before the second series test	:	30 marks

End Semester Examination Pattern:

The following guidelines should be followed regarding award of marks

1. Preliminary work :15 Marks
2. Implementing the work/Conducting the experiment :10 Marks
3. Performance, result and inference (usage of equipments and troubleshooting)

   : 25 Marks
4. Vivavoce :20 Marks
5. Record :5 Marks

Continued from previous page

PART A

EXPERIMENT NO. 1

POWER BJT DRIVE CIRCUITS

Aim

To design and setup a basic Power BJT drive circuit.

Components and Equipments Required:

2N3055, Resistors, Breadboard, Power supply, DSO

Theory

Usually, a base drive circuit should provide electrically isolated base signal to turn – ON and turn – OFF a transistor. A power BJT has a current gain about 5 to 10. The power handling capability of driver circuit should be 20% higher than the power circuit. This circuit should have very high impedance; therefore, a very low power driver circuit is carefully designed to drive a BJT. A good BJT base drive circuit must be able to:

1. Supply adequate positive base current during turn-ON
2. Supply adequate base current to maintain the transistor in the steady-ON state
3. Supply negative base current for proper turn-OFF of the transistor

Design:

Driver circuit Design

$$I_C = 1\text{ A}$$

$$h_{FE} = 20$$

$$I_B(\text{on}) = \frac{2I_C}{h_{FE(\text{sat})}} = \frac{2\times 1}{20} = \frac{1}{10} = 0.1\text{ A}$$

$$V_B = 5\text{ V},\quad V_{BE(\text{sat})} = 1\text{ V}$$

$$R_2 = \frac{V_{BE(\text{sat})}}{I_B(\text{on})} = \frac{V_{BE(\text{sat})}}{\frac{2I_C}{h_{FE(\text{sat})}}} = \frac{V_{BE(\text{sat})}\cdot h_{FE(\text{sat})}}{2I_C}$$

$$R_2 = \frac{1}{0.1} = 10\ \Omega,\quad 0.1\text{ W}$$

$$P = I^2R = (0.1)^2\cdot 10 = 0.1\text{ W}$$

$$R_1 = \frac{V_B - V_{BE(\text{sat})}}{2I_B(\text{on})} = \frac{5-1}{2\cdot 0.1} = \frac{4}{0.2} = 20\ \Omega,\quad 0.8\text{ W}$$

$$P = I\cdot V = 2\cdot 0.1\cdot(5-1) = 4\cdot 0.2 = 0.8\text{ W}$$

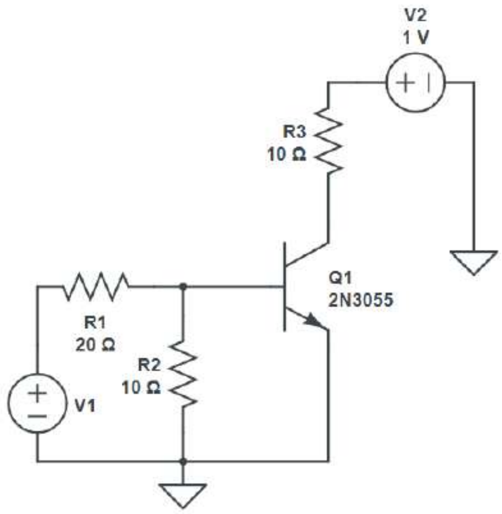
$$V_{CC} = 11\text{ V}$$

$$I_C = \frac{V_{CC}-V_{CE}}{R_C}$$

$$R_C = \frac{11-1}{1} = 10\ \Omega\quad(10\text{ W})$$

$$P = 10\text{ V}\cdot 1\text{ A} = 10\text{ W}$$

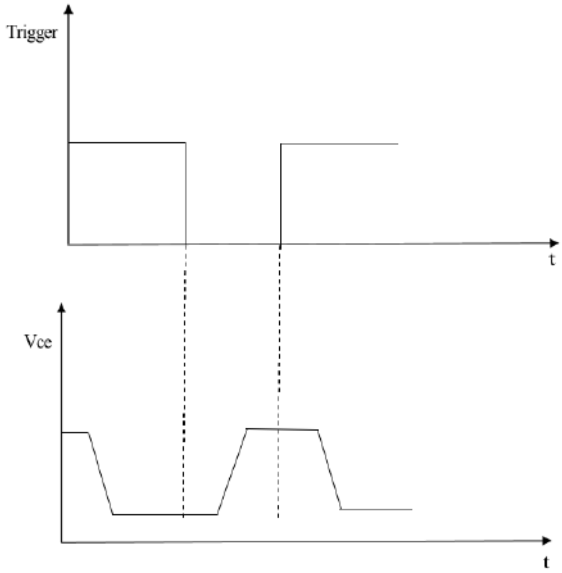
Circuit diagram:



Procedure

1. Setup the circuit as shown in the figure.
2. Observe the output across the load for the given input.

Expected Waveforms:



Results

A drive circuit for power BJT is designed and setup and the corresponding waveforms are observed

EXPERIMENT NO. 2

Power MOSFET Drive Circuits

Aim

To setup a basic power MOSFET drive circuit.

Components and Equipments Required:

IRF840, BC177, !N4148, Resistors, Power supply, DSO, Breadboard, etc

Theory

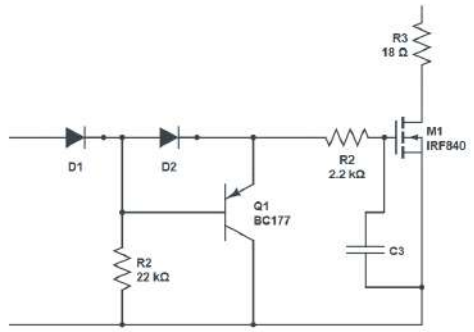
In the MOSFET drive circuit, the gate current required to maintain MOSFET in steady ON condition is zero. The gate circuit energy is used only to turn ON and turn OFF the MOSFET. During turn ON, a positive peak gate current is applied which turns ON the MOSFET. During turn OFF, a negative peak gate current is provided for fast turn OFF of the MOSFET. To turn ON a MOSFET, a specific amount of gate charge has to be supplied to the gate of the MOSFET (which is given in the datasheet).

Design:

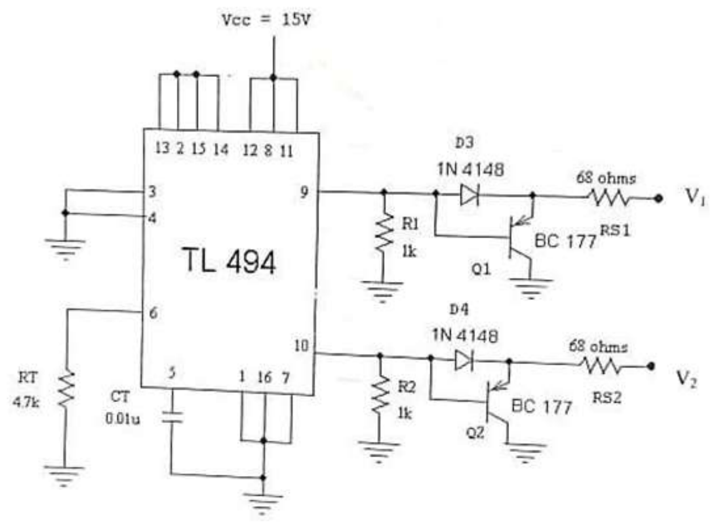
$$
\begin{aligned}
V_{\text{in}} &= 15\ \text{V} \\
V_{\text{diode}} &= 0.7\ \text{V} \\
V_g &= V_{\text{in}} - 2 \times V_{\text{diode}} = 15 - 2 \times 0.7 = 13.6\ \text{V} \\
i_{gm} &= \frac{Q_g}{t_{\text{on}}} \\
t_{\text{on}} &= 15\ \mu\text{s}, \quad Q_g = 42\ \text{nC} \\
i_{gm} &= \frac{42\ \text{nC}}{15\ \mu\text{s}} = 2.8\ \text{mA} \\
i_g^+ &= 2i_{gm} = 5.6\ \text{mA} \\
R_1 &= \frac{V_g}{i_g^+} = \frac{13.6}{5.6\ \text{mA}} \approx 2.2\ \text{k}\Omega \\
R_2 &= 10R_1 = 22\ \text{k}\Omega \\
R_D &= \frac{V_{DD}}{i_0} = \frac{15\ \text{V}}{0.83\ \text{A}} \approx 18\ \Omega \\
f_{\text{osc}} &= \frac{1.2}{R_t \times C_t} = 25\ \text{kHz} \\
\text{Let } C_t &= 0.01\ \mu\text{F} \Rightarrow R_t = \frac{1.2}{25 \times 10^3 \times 0.01 \times 10^{-6}} = 4.8\ \text{k}\Omega \quad (\text{take } R_t = 4.7\ \text{k}\Omega) \\
t_{\text{on}} = t_{\text{off}} &= 1\% \text{ of period} = 1\% \text{ of } \frac{1}{f_{\text{osc}}} = \frac{0.01}{25\ \text{kHz}} = 0.4\ \mu\text{s}
\end{aligned}
$$

Circuit diagram:

Driver Circuit :



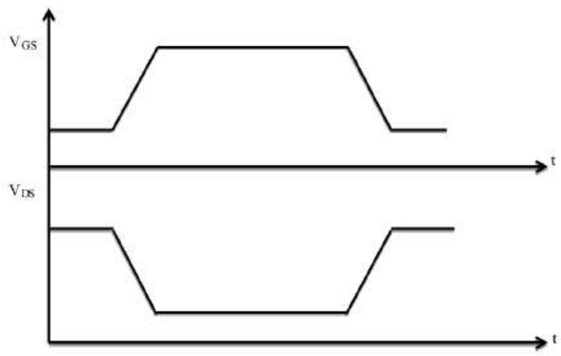
MOSFET Triggering Circuit :



Procedure

1. Make the connections as per the circuit diagram.
2. Connect CRO probes across terminals where required output is taken.
3. Observe the waveforms in CRO and plot the graphs.

Expected Waveforms:



Results

A power MOSFET drive circuit is set up and waveforms are obtained.

EXPERIMENT NO. 3

REALIZATION OF SIMPLE SMPS

Aim

To design a simple switched mode power supply

Components and Equipments Required:

Sl No	Components and Equipments	Specification	Quantity
1	PWM IC	TL 494	1
2	Transformer	230V / 12V	1
3	Transistor	BC 197	2
4	MOSFET	IRF 480	4
5	Diode	1N4148, 1N4007	4
6	Resistor	68Ω, 1kΩ, 4.7kΩ	2,2,1 (5)
7	Capacitor	0.01µF, 100µF	2
8	Breadboard	-	2
9	Probes	-	2
10	DSO	-	1
11	Power Supply	230 V 60 Hz	1

Theory

SMPS is based on the chopper principle. The output DC voltage is controlled by varying the duty cycle of the chopper by PWM or FM techniques. The circuit arrangement for a full bridge SMPS as shown in the figure. When power supply is given to the MOSFETs M_1 and M_2 and they are turned ON, voltage Vs appears across the transformer primary ($V_1 = V_s$) and secondary voltage $V_2 = V_3$ because (N_2/N_1) = aVs Diode D1 gets forward biased $V1 = -Vs$and $V_2 = -Vs(N_2/N_1)$ = -aVs. Diode D2 now begins to conduct and the output voltage is now V0=aVs. The open circuit voltage across each MOSFET is Voc = Vs

Design:

$V_{cd} \equiv \frac{2V_2}{\pi}$

Let $V_2 \equiv 12$ V $\Rightarrow V_{cd} = 7.63$ V

Output ripple $\equiv \left(\frac{\sqrt{2}}{3}\right) V_{dc} \left(\frac{X_C}{X_L}\right)$

$X_C \equiv \frac{1}{2\omega C}, \quad X_L = 2\omega L$

Ripple factor $\equiv \frac{1.195}{LC}$

$C \equiv 0.1\ \mu$F, $\quad R = 10\ \Omega$ (10 W)

Alternatively, $C \equiv 100\ \mu$F and $R = 1$ kΩ

MOSFET Bias Circuit Design:

$t_{\text{ON}} = t_{\text{OFF}} \equiv 1\%$ of period = 0.4 μs

$f_{\text{osc}} \equiv \frac{1.2}{R_T C_T}$

Take $C_T \equiv 0.01\ \mu$F $\Rightarrow R_T = 4.8$ kΩ hence we use 4.7 kΩ

Gate charge for turn ON $\equiv$ 60 nC

$I_c \equiv \frac{60\text{ nC}}{t_{\text{ON}}} = 150$ mA

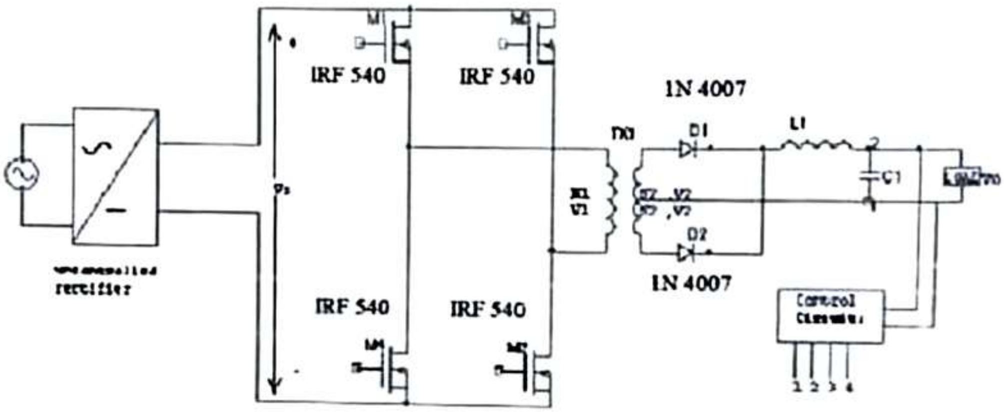
$R_s \equiv \frac{V_{cc}}{I_c} = \frac{10}{150\text{ mA}} = 68\ \Omega$

$R_{s1} \equiv R_{s2} = 68\ \Omega$

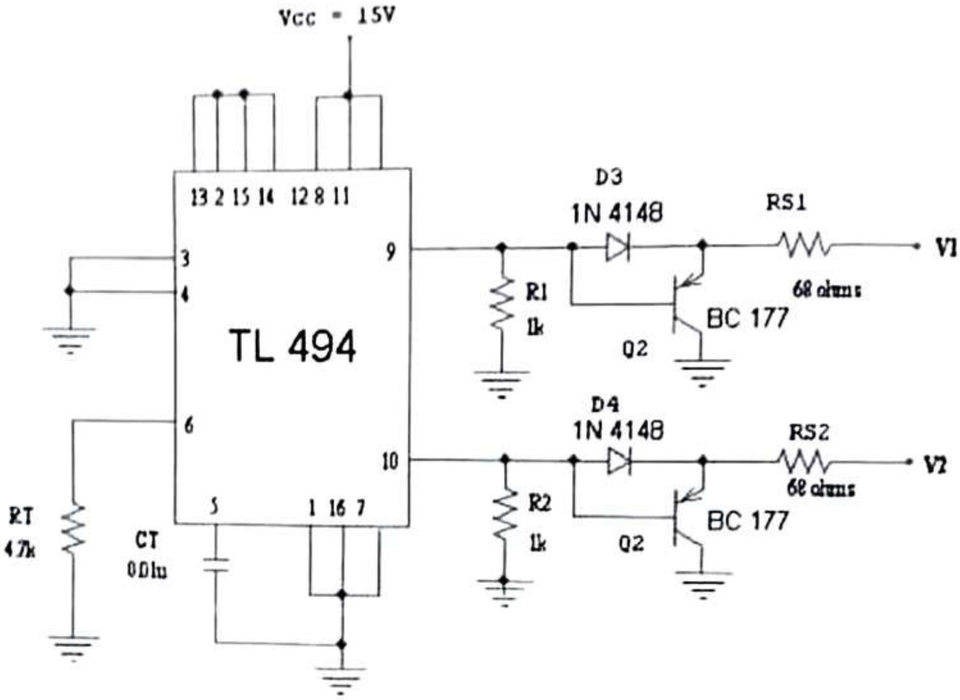
Circuit diagram:

Circuit Diagram:

Switched Mode Power Supply



MOSFET Triggering Circuit



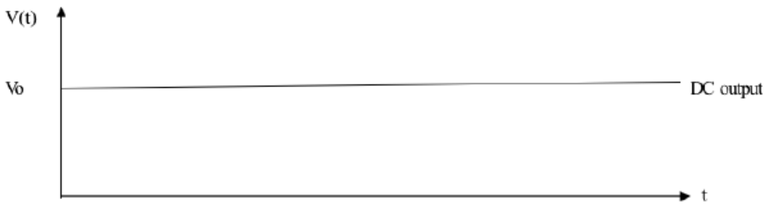
Procedure

1. The circuit is setup according to the given circuit diagrams

2. Connect the trigger circuit output to the appropriate points in the SMPS

3. Observe the output across the load for the given input

Expected Waveforms:



Results

SMPS circuit is obtained and the output was observed to be ________ mV

EXPERIMENT NO. 4

SINGLE PHASE HALF CONTROLLED RECTIFIERS WITH R, RL LOADS

Aim

To design, implement and study the waveforms of single phase half controlled rectifiers with R, RL loads.

Components and Equipments Required:

Sl No	Components and Equipments	Specification	Quantity
1	2P4M SCR	V_{gm} = 1.7V, I_{gm} = 50μA	1
2	24V transformer	Input Voltage: 230V AC Output Voltage: 24V or 0V Output Current: 2 Amp	1
3	Resistors	100Ω, 1MΩ, 100KΩ pot	1, 1, 1
4	Inductor	-	1
5	230V AC power supply	-	1
6	Connecting wires, probes, breadboard, CRO	-	-

Theory

Unlike diode rectifiers, phase controlled rectifiers has an advantage of controlling the output voltage. The diode rectifiers are called uncontrolled rectifiers when these diodes are replaced with thyristors, then it become phase controlled rectifiers. The output voltage can be controlled by varying the firing angle of the thyristors. A phase control thyristor is activated by applying a short pulse to its gate terminal. Single phase controlled rectifiers work from single phase AC input power supply.

Design:

$$V_{CC} = 230\,\text{V}$$

$$V_m = 230\sqrt{2} = 325.27\,\text{V}$$

$$\text{Maximum Gate Current (for 2P4M SCR), } I_{G(\max)} = 0.2\,\text{A}$$

$$\text{Maximum On-State Gate Voltage, } V_{GM} = 1.7\,\text{V}$$

$$R_1 \geq \frac{V_{gm}}{I_{gm}}$$

$$R_1 \geq \frac{1.7V}{50\mu A}$$

$$R_1 \geq 0.03M\Omega$$

$$\text{Assume } R_1 = 1M\Omega$$

$$\text{Also, } R_2 \leq \frac{R_1 V_{gm}}{(V_m - V_{gm})}$$

$$R_2 \leq \frac{10^6 \cdot 1.7}{(230\sqrt{2} - 1.7)}$$

$$R_2 \leq 5.25k\Omega$$

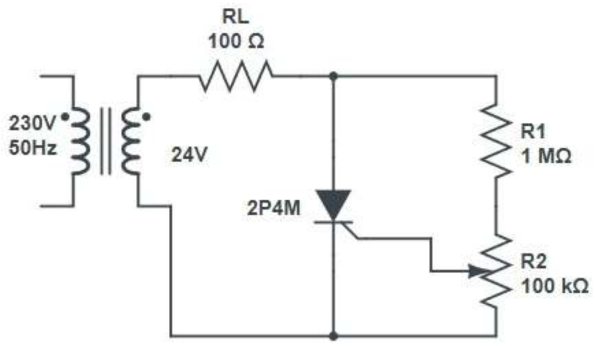
$$R_2 = 100\,\text{k}\Omega \quad \text{(Potentiometer)}$$

$$R_L = \frac{24}{0.2} = 120\,\Omega$$

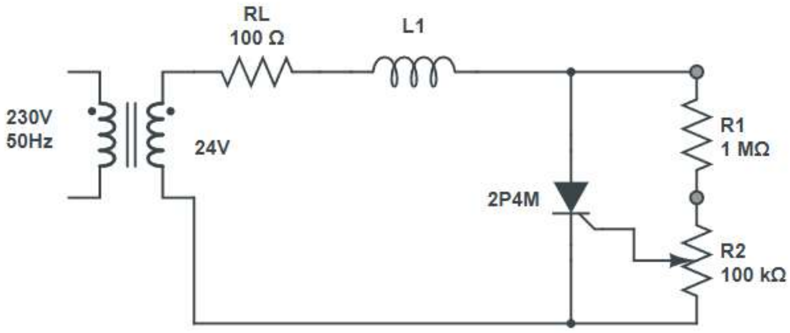
$$\text{Take } R_L = 100\,\Omega$$

Circuit diagram:

Half wave controlled rectifier with resistive load



Half wave controlled rectifier with RL load

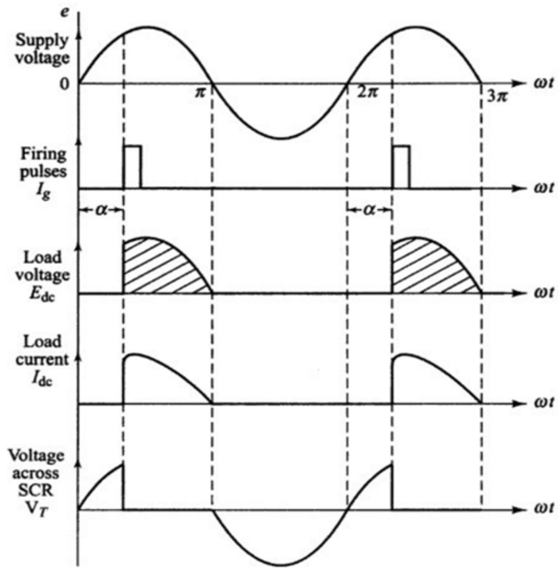


Procedure

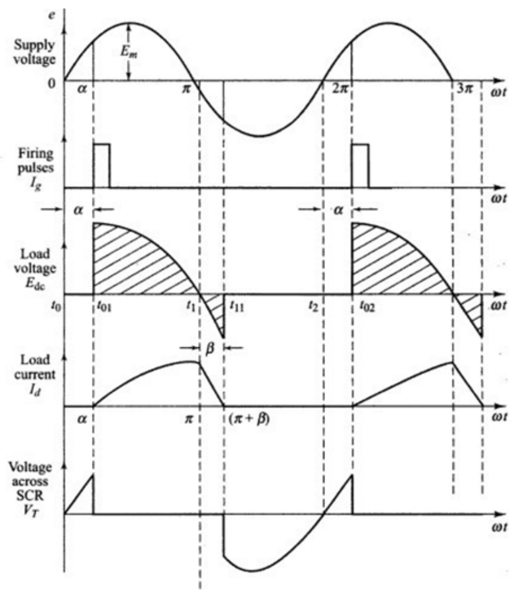
1. Connect the circuit as shown in the figure.
2. Connect the CRO probes across the load terminals and across the SCR.
3. Observe the waveforms on CRO and plot the waveforms.

Expected Waveforms:

Half wave controlled rectifier with resistive load



Half wave controlled rectifier with RL load



Results

Single phase half controlled rectifiers with R, RL loads are designed and implemented. The waveforms are plotted for the same.

EXPERIMENT NO. 5

REALIZATION OF BASIC BUCK, BOOST AND BUCK-BOOST CONVERTERS

Aim

To realize fundamental Buck, Boost and Buck-Boost converters

Components and Equipments Required:

Sl No	Components and Equipments	Specification	Quantity
1	PWM IC	TL494	1
2	Inductors	0.4 mH, 0.53 mH	1, 1
3	Capacitors	7.5 μF, 24 μF, 2.4 μF, 0.01 μF	1, 1, 1, 1
4	DSO	Keysight (DSO1052B)	1
5	Connecting wires	Insulated wires	20
6	Diode	1N4007	1
7	Diode	1N4148	2
8	Resistor	1 kΩ, 4.7 kΩ, 68 Ω	3, 1, 2
9	Transistor	BC177	2

Theory

A buck or step down converter is a DC to DC switch mode power supply which is intended to lower the input voltage of an unregulated DC supply to a stabilized lower output voltage. It has a higher efficiency which can easily exceed 95A boost converter is a DC-to-DC switch mode power supply which is intended to boost the input voltage of an unregulated DC supply to a stabilized higher output voltage. Similar to a buck converter, a boost converter relies on an inductor, diode, capacitor and power switch which regulates the output voltage but they are arranged differently. It is also class of switched-mode power supply. The controlled switch is turned on and off by using Pulse Width Modulation (PWM). A buck-boost converter can supply a regulated DC output from a power supply delivering a voltage either below or above the regulated output voltage of a buck-boost converter circuit. It combines the elements of both a buck converter and a boost converter. It is also a class of switched-mode power supply. The controlled switch is turned on and off by using Pulse Width Modulation (PWM).

Design:

Buck Converter

$$\text{Let } f = 50 \text{ kHz}, \quad T = 0.02 \text{ ms}$$

$$\text{For IN4007, } I_d = 1 \text{ A}, \quad I_d = I_L = 1 \text{ A}$$

$$I = 30\%, \quad I_L = 0.3 \text{ A}$$

$$\text{Let } V = 100 \text{ mV} = 0.1 \text{ V}$$

$$V_S = 24 \text{ V}, \quad V_D = 12 \text{ V}$$

$$V_O = V_{SD}, \quad D = 0.5 \ (50\%)$$

$$L = \frac{V_O(V_S - V_O)\Delta I}{V_S} = 0.02 \times 10^{-3} \times \frac{12 \times 12}{24 \times 0.4} = 0.4 \text{ mH}$$

$$C = \frac{T\Delta I}{8\Delta V} = \frac{0.02 \times 10^{-3} \times 0.3}{8 \times 0.1} = 7.5 \ \mu\text{F}$$

Boost Converter

$$\text{Let } f = 50 \text{ kHz}, \quad T = 0.02 \text{ ms}$$

$$I_d = I_L = 1 \text{ A}$$

$$I = 0.3 \text{ A}$$

$$V = 0.1 \text{ V}$$

$$V_O = V_S(1 - D), \quad D = 0.5$$

$$L = \frac{T\,V_S(V_O - V_S)}{\Delta I\,V_O} = 0.4 \text{ mH}$$

$$C = \frac{i_O}{\Delta V}$$

$$i_O = V_O R$$

$$\text{Let } R = 1 \text{ k}\Omega, \quad t_{on} = DT = 0.01 \text{ ms}$$

$$C = 0.01 \times 10^{-3} \times 240.1 \times 12 = 24\ \mu\text{F}$$

Buck-Boost Converter

$$\text{Let } f = 50 \text{ kHz}, \quad T = 0.02 \text{ ms}$$

$$I_d = I_L = 1 \text{ A}, \quad I = 0.3 \text{ A}$$

$$V = 0.1 \text{ V}$$

$$V_S = 12 \text{ V}, \quad V_O = 24 \text{ V}$$

$$D = 0.5$$

$$L = \frac{T\, V_O V_S\, \Delta I}{V_O + V_S} = 0.53 \text{ mH}$$

$$t_{on} = DT = 0.01 \text{ ms}$$

$$i_O = \frac{V_O}{R}$$

$$\text{Let } R = 1\text{k}\Omega, \quad C = \frac{i_O t_{on}}{\Delta V} = 24 \times 0.01 \times 10^{-3} \times 0.1 = 2.4\ \mu\text{F}$$

MOSFET Bias Circuit Design

$$f_{\text{osc}} = \frac{1.2}{R_t C_t} = 25 \text{ kHz}$$

$$\text{Let } C_t = 0.01\ \mu\text{F} \Rightarrow R_t = 4.8\ \text{k}\Omega\ (\text{take } R_t = 4.7\ \text{k}\Omega)$$

$$t_{\text{on}} = t_{\text{off}} = 1\% \text{ of period} = 1\% \text{ of } \frac{1}{f_{\text{osc}}} = 0.01 \div 25 \text{ kHz} = 0.4\ \mu\text{s}$$

$$\text{Gate charge to turn ON} = 60 \text{ nC}$$

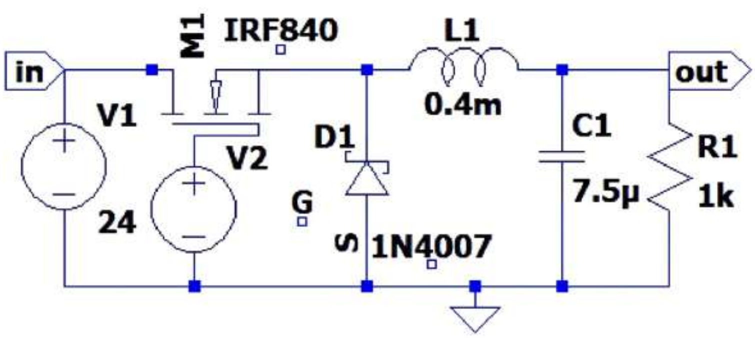
$$I_c = \frac{60 \text{ nC}}{t_{\text{on}}} = 150 \text{ mA}$$

$$R_s = \frac{V_{CC}}{I_c} = \frac{10}{150 \text{ mA}} = 68\ \Omega$$

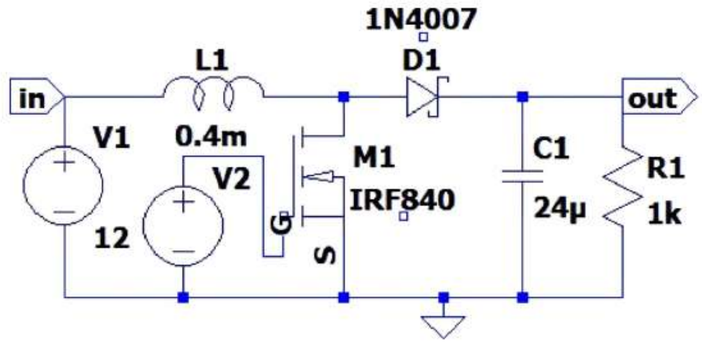
$$\therefore R_{s1} = R_{s2} = 68\ \Omega$$

Circuit diagram:

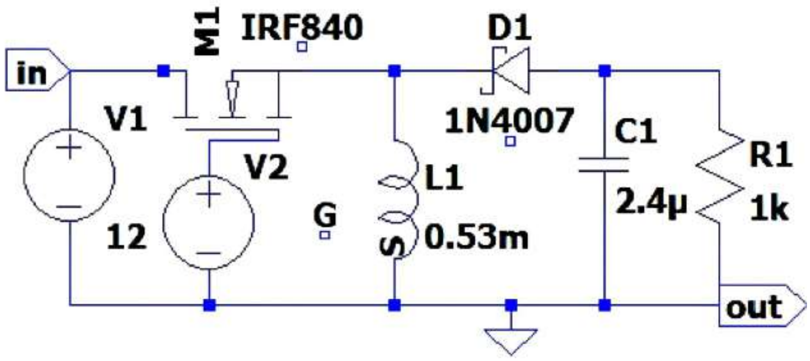
Buck Converter



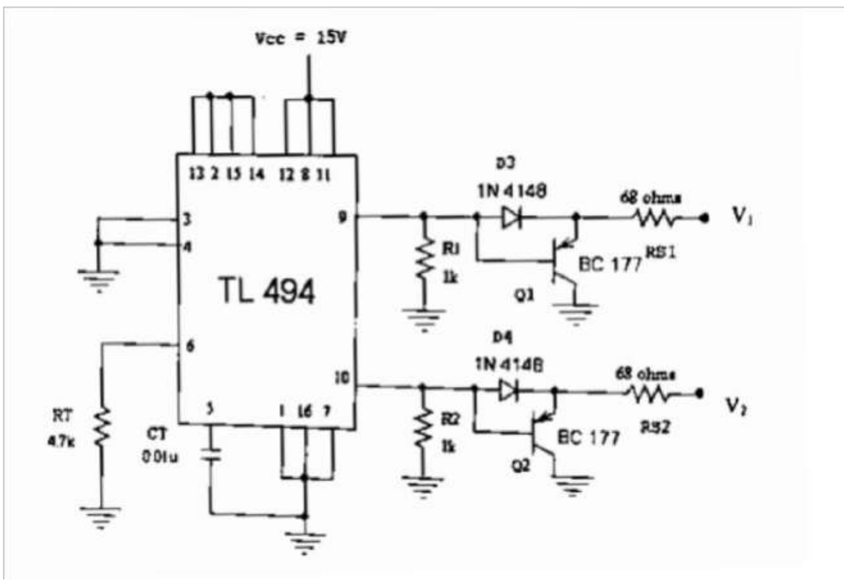
Boost Converter



Buck-Boost Converter



MOSFET Triggering Circuit

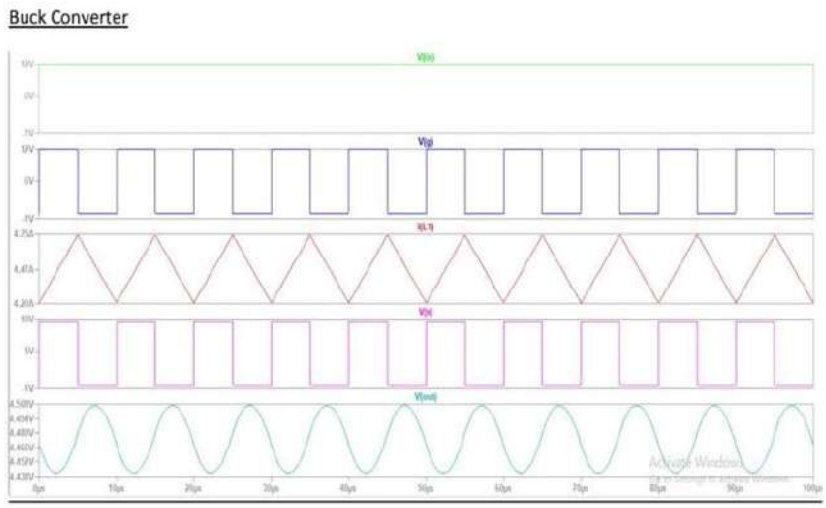


Procedure

1. Connect the circuits as per the figures.
2. Give a triggering signal to the circuits.
3. Connect the CRO probes across the load and observe the waveforms.

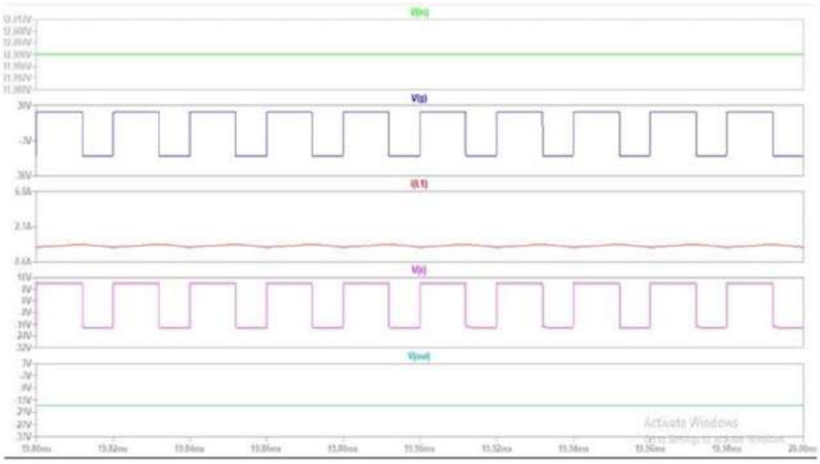
Expected Waveforms:

Buck Converter



Boost Converter

Buck-Boost Converter



Results

A buck converter was designed and setup and output waveforms were obtained with:

- Peak-to-peak voltage, V_{pp} = ...
- Average voltage, V_{avg} = ...
- Maximum voltage, V_{max} = ...
- Minimum voltage, V_{min} = ...

A boost converter was designed and setup and output waveforms were obtained with:

- Peak-to-peak voltage, V_{pp} = ...
- Average voltage, V_{avg} = ...
- Maximum voltage, V_{max} = ...
- Minimum voltage, V_{min} = ...

EXPERIMENT NO. 6

REALIZATION OF HALF BRIDGE CONVERTER

Aim

To design a half bridge converter

Components and Equipments Required:

Sl. No.	**Components and Equipments**	**Specification**	**Quantity**
1	PWM IC	TL494	1
2	MOSFET	IRF840	4
3	Transistor	BC 177	2
4	Resistor	100Ω, 1kΩ, 4.7kΩ, 56kΩ	2, 2, 1, 1
5	Diode	1N4007	2
6	Capacitor	0.01μF	1

Theory

Half-Bridge Inverter

A half-bridge DC to AC converter is a type of inverter that converts a DC voltage into an AC output using two power switches and a centre-tapped transformer or a capacitor divider. The circuit operates by alternately switching the two power MOSFETs, creating a square wave or modulated AC waveform across the load. The TL494 pulse width modulation (PWM) controller is commonly used to generate the required control signals for the switching devices, ensuring stable and efficient operation.

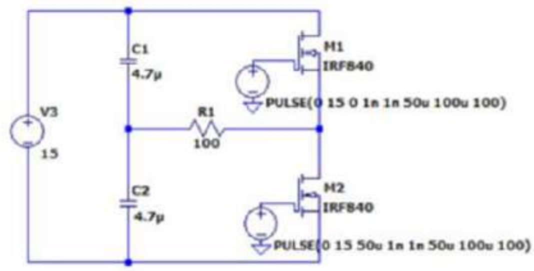
The half-bridge configuration includes two IRF840 MOSFETs, two capacitors connected in series across the DC supply, and a resistor as the load. The midpoint of the capacitors serves as a reference point, resulting in an output voltage that swings between $+V/2$ and $-V/2$. When the first MOSFET turns on, the load sees a positive voltage, and when the second MOSFET turns on, the load sees a negative voltage. The TL494 generates complementary PWM signals to control the switching sequence, regulating the output waveform's frequency and amplitude.

Half-bridge inverters are widely used in medium-power applications such as motor drives and uninterruptible power supplies (UPS) due to their efficiency and cost-effectiveness.

Design:

$$
\begin{aligned}
f_{\text{out}} &= 12.5\ \text{kHz} \\
\text{Therefore, } f_{\text{osc}} &= 2 \times f_{\text{out}} = 25\ \text{kHz} \\
f_{\text{osc}} &= \frac{1.1}{R_T \times C_T} \\
\text{Let } C_T &= 0.01\ \mu\text{F} \\
\text{Therefore, } R_T &= \frac{1.1}{f_{\text{osc}} \times C_T} \\
&= \frac{1.1}{25 \times 10^3 \times 0.01 \times 10^{-6}} \\
&\approx 4.7\ \text{k}\Omega
\end{aligned}
$$

Circuit diagram:



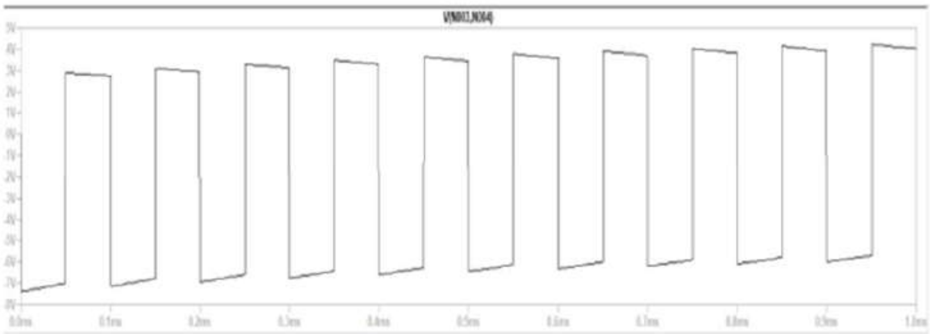
Procedure:

1. Connect the half-bridge inverter circuit as shown in the provided circuit diagram using two IRF840 MOSFETs, two series-connected capacitors across the DC supply, and a resistive load.

2. Interface the TL494 PWM controller circuit as per the diagram to provide complementary gate drive signals to the MOSFETs with the desired switching frequency.

3. After verifying all connections, power the circuit and observe the output waveform across the load using an oscilloscope to confirm the AC output swing between $+\frac{V}{2}$ and $-\frac{V}{2}$.

Expected Waveforms:



Results

Designed and implemented half bridge converter.

EXPERIMENT NO. 7

APPLICATION OF PWM IC TL 494

Aim

To design a pulse width modulated control circuit using TL 494.

Components and Equipments Required:

Sl No	Components and Equipments	Specification	Quantity
1	PWM IC TL494	Supply Voltage: 7V to 40V Number of Outputs: 2 Output Switching Frequency: 300 kHz	1
2	Resistors	Carbon Film Resistor	1
3	Capacitors	Ceramic Capacitor	1
4	CRO	Bandwidth: 20MHz - 50MHz Sample rate: 250,000 samples/sec	2
5	Power supply	+15V, -15V	2

Theory

TL494 is a fixed frequency, pulse width modulation control circuit designed primarily for SWITCH MODE power supply control.

Features:

- Complete Pulse Width Modulation Control Circuitry
- On-Chip Oscillator with Master or Slave Operation

- On-Chip Error Amplifiers
- On-Chip 5.0V Reference
- Adjustable Dead-time Control
- Uncommitted Output Transistors Rated to 500 mA Source or Sink
- Output Control for Push-Pull or Single-Ended Operation
- Under-voltage Lockout

TL494 is a fixed-frequency pulse width modulation control circuit, incorporating the primary building blocks required for the control of switching power supply. An internal linear saw-tooth oscillator is frequency programmable by two external components. The approximate oscillator frequency is determined by:

$$f_{\text{osc}} = \frac{1.1}{R_T \times C_T} \tag{1}$$

Output pulse width modulation is accomplished by comparison of the positive saw-tooth waveform across the capacitor to either of the two control signals. The dead time control comparator has an effective 120 mV input offset which limits the minimum output dead time to approximately the first 4% of the saw-tooth-cycle time. This would result in a maximum duty cycle with the output control grounded, and 48% with it connected to the reference line.

The pulse width modulator comparator provides a means for the error amplifiers to adjust the output pulse width from the maximum percent on-time, established by the dead-time control by setting the dead time-control input, down to zero, as the voltage at the feedback pin varies from 0.5 V to 3.5 V.

Both error amplifiers have a common mode input ranging from -0.3 V to (V_{CC} - 2 V), and may be used to sense power-supply output voltage and current. When capacitors discharge, a positive pulse is generated on the output of the dead time comparator, which clocks the pulse-steering flip-flop and inhibits the output transistors. With the output-control connected to the reference line, the pulse-steering flip-flop directs the modulated pulses to each of the two output transistors alternately for push-pull operation.

Design:

$$f_{\text{osc}} = \frac{1.2}{R_T \times C_T}$$

$$\text{Let } C_T = 1 \text{ nF}$$

$$\text{Let } V_{CC} = 15 \text{ V}, \; I_C = 15 \text{ mA}, \; R_C = \frac{15 \text{ mA}}{15 \text{ mA}} = 1 \text{ k}\Omega$$

To Design R and C,

The voltage at pin 3 is $V_{CC} \times e^{\left(\frac{-t}{R \times C}\right)}$

$$\text{Let } V_3 = 2.8 \text{ V}; \; V_{CC} = 15 \text{ V}; \; R \times C = \tau$$

$$2.8 = 15 \times e^{\left(\frac{-t}{R \times C}\right)}$$

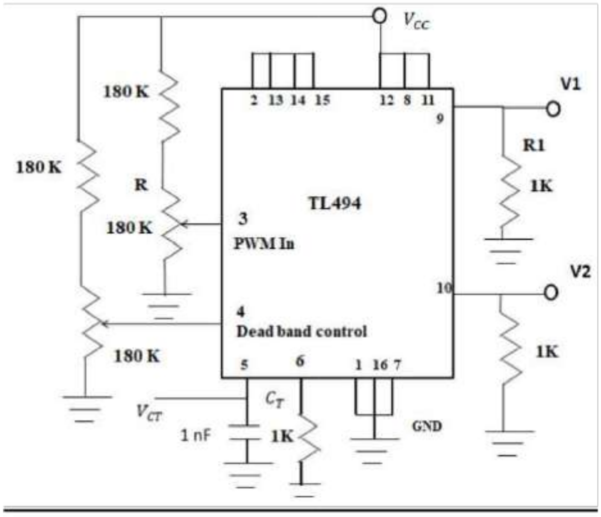
$$\text{Let } t = 5 \text{ s}; \; \tau = 3 \text{ s}$$

$$\text{Let } C = 47\mu\text{F}; \; R = 64 \text{ k}\Omega$$

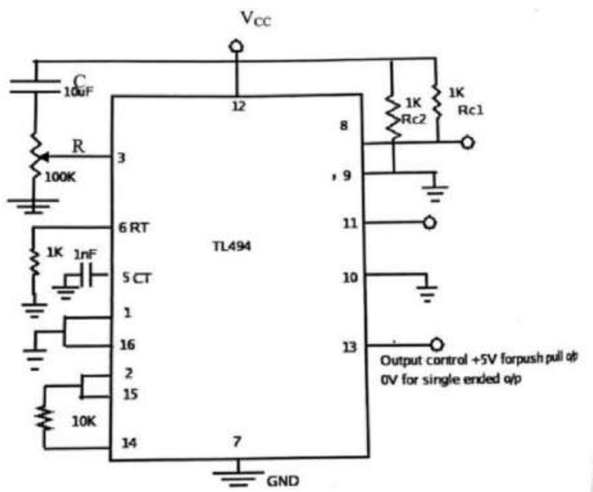
$$R = 100 \text{ k}\Omega \text{ pot.}$$

Circuit diagram:

Push Pull Configuration



Soft Start Configuration



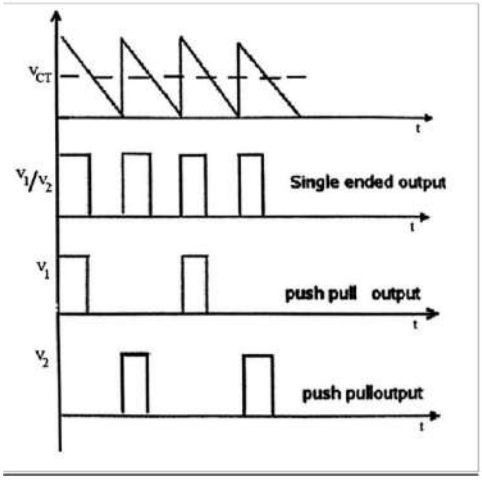
Procedure

1. Set up the circuit as shown in figures.
2. Observe the different waveforms for single-ended and push-pull output.

3. Adjust dead band control and notice the variations in output.

NOTE: Connect pin 13 to GND for single-ended output and connect pin 13 to pin 14 for push-pull output.

Expected Waveforms:



Results

A pulse width modulated control circuit was designed using TL 494 and the following modes were observed with waveforms: A single ended output of 100kHz and a push pull output of 100kHz.

EXPERIMENT NO. 8

DC TO AC INVERTER USING MOSFET & IC

Aim

To design a DC to AC inverter using MOSFET & IC

Components and Equipments Required:

SL.NO.	Components and equipments	Specifications	Quantity
1	MOSFET	IRF540	2
2	PWM IC	TL494	1
3	Transformer	12-0-12/230V	1
4	Transistor	BC 177	2
5	Diode	1N4007	2
6	Resistor	1kΩ, 100Ω, 100kΩ, 56kΩ	2, 2, 1, 1
7	Capacitor	0.1 μF	1

Theory

DC to AC inverters are one of the most important elements in power electronics. There are a variety of applications based on such conversions, i.e., inverters convert direct current input (DC) into alternating current output (AC). The TL494 generates a pulse width modulated signal to control the switching of the MOSFETs, ensuring efficient conversion. The H-bridge arrangement allows current to flow in both directions through the transformer, producing an AC output. The transformer steps up the low voltage AC generated by the MOSFETs to 230V AC, making the circuit suitable for powering household appliances such as UPS systems, renewable energy applications and emergency power supplies.

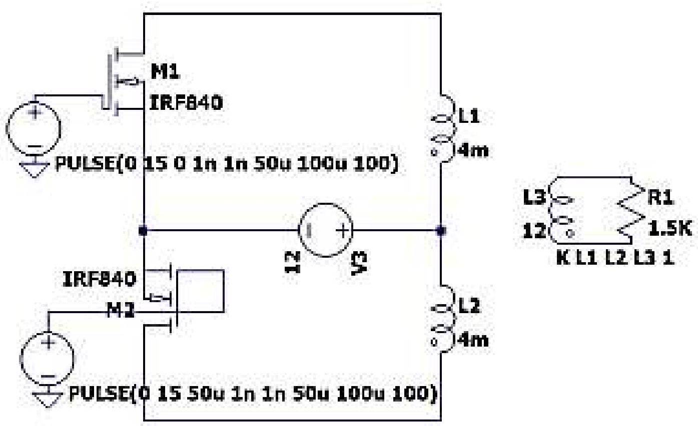
Procedure:

1. Connect the DC to AC inverter circuit as shown in the provided circuit diagram.
2. Configure the TL494 to generate pulse width modulated (PWM) signals that drive the MOSFETs in an H-bridge configuration for bidirectional current flow through the transformer.
3. After ensuring proper connections, apply the DC supply and observe the stepped-up 230V AC output at the transformer secondary using a multimeter or oscilloscope.

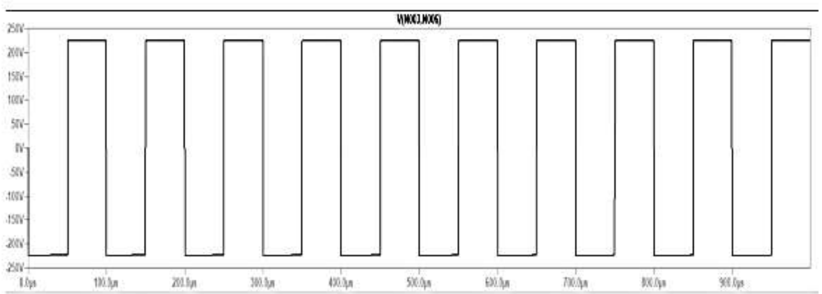
Design:

$$
\begin{aligned}
F_{\text{out}} &= 50\ \text{Hz} \\
\therefore F_{\text{osc}} &= 2 \times F_{\text{out}} = 100\ \text{Hz} \\
100 &= \frac{1.1}{R_T \times C_T} \\
\text{Let } C_T &= 0.1\ \mu\text{F} \\
R_T &= \frac{1.1}{100 \times 0.1 \times 10^{-6}} = 100\ \text{k}\Omega
\end{aligned}
$$

Circuit diagram:



Expected Waveforms:



Results

Designed and implemented a DC to AC inverter using MOSFET & IC

PART B - SIMULATION EXPERIMENTS

EXPERIMENT NO. 1

DRIVER CIRCUITS FOR POWER BJT

Aim

To design and simulate a power BJT circuit

Components and Equipments Required:

Sl No	Components and equipments	Specification	Quantity
1	Power supply	10V	1
2	Power supply	5V	1
3	Power supply	15V	1
4	Pulse generator	0-5V, 50 s time period	1
5	Resistor	10KΩ	2
6	Resistor	1KΩ	2
7	PNP transistor	hfe=200	1
8	NPN transistor	hfe=200	2

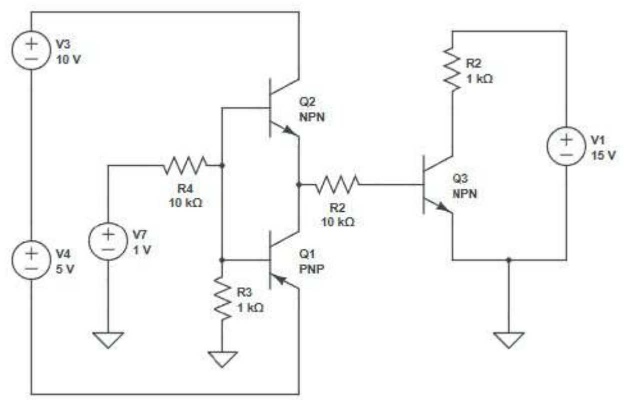
Theory

In BJT circuit, during turn on, the base current should be fast rising and must overshoot the final steady on value[cite: 319]. During turn off, a negative base current should be provided to quickly remove stored charges in BJT[cite: 320]. A good BJT drive circuit must be able to:

1. Supply adequate positive base current during turn on.
2. Supply adequate base current to maintain BJT in steady on state.
3. Supply negative base current for proper turn off of the transistor.

Circuit diagram:

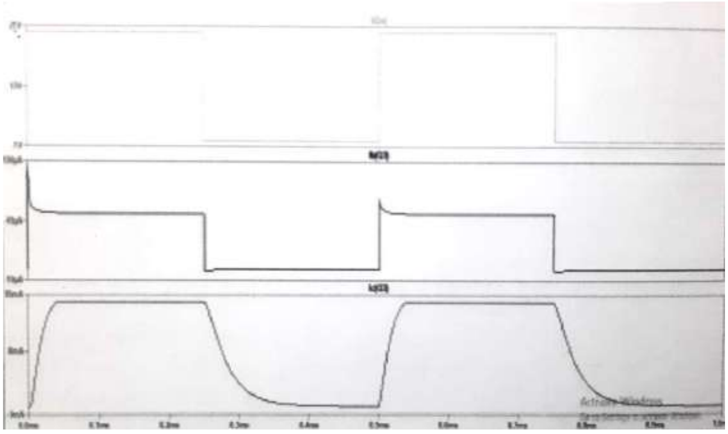
Power BJT Driver Circuit for Simulation



Procedure

1. Assemble the circuit as shown in the the figure above in LT Spice.
2. Simulate and obtain the required waveforms.

Expected Waveforms:



Result

A power BJT driver circuit is setup and simulated and corresponding waveforms are obtained.

EXPERIMENT NO. 2

DRIVE CIRCUITS WITH POWER MOSFET

Aim

To design and simulate power MOSFET dive circuit.

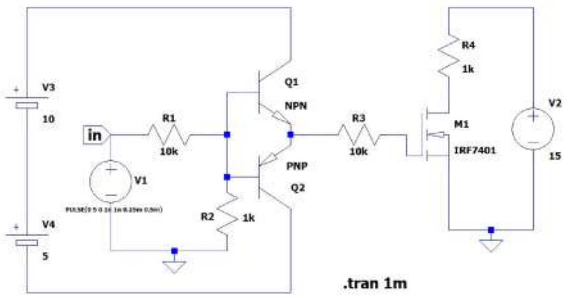
Components and Equipments Required:

Sl No	Components and equipments	Specification	Quantity
1	Power MOSFET	IRF7401	1
2	Resistors	$10k\Omega$, $1k\Omega$	2, 2
3	BJT	PNP, NPN	1, 1

Theory

The MOSFET drive circuit, the gate current required to maintain MOSFET in steady ON condition is zero. The gate circuit energy is used only to turn ON and turn OFF the MOSFET. During turn ON, a positive peak gate current is applied which turns ON the MOSFET. During turn OFF, a negative peak gate current is provided for fast turn off of the MOSFET. To turn ON a MOSFET, a specific amount of gate charge has to be suppliedto the gate of the MOSFET (which is given in the datasheets).

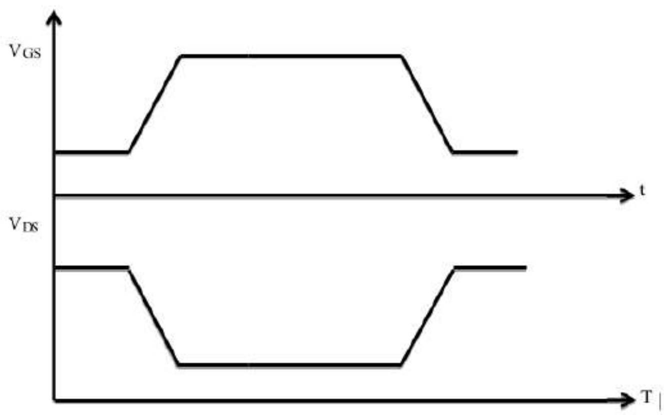
Circuit diagram:



Procedure

1. Connect the circuit as shown in circuit diagram.
2. Simulate and obtain required waveforms

Expected Waveforms:



Results

A power MOSFET drive circuit is set up, simulated and waveforms are obtained.

EXPERIMENT NO. 3

SHUNT AND SERIES SNUBBER CIRCUITS

Aim

To model and simulate MOSFET shunt and series snubber circuits.

Components and Equipments Required:

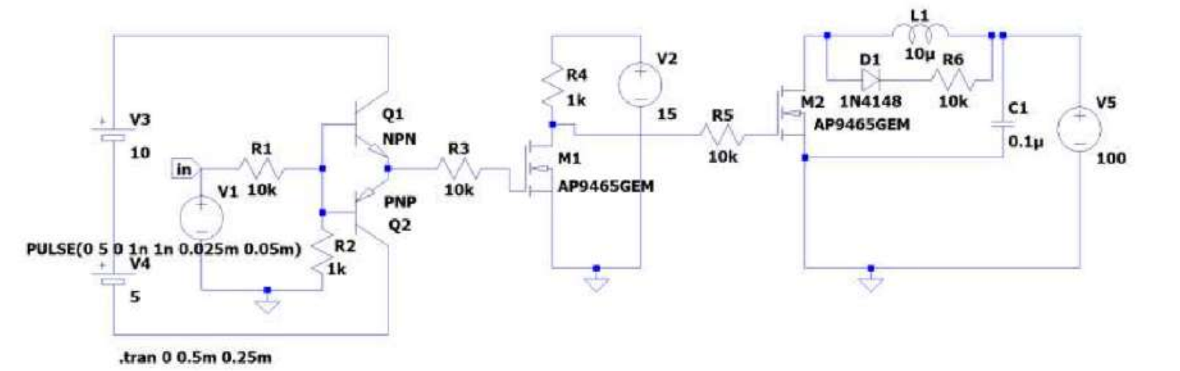
Sl No	Components and equipments	Specification	Quantity
1	Diodes	1N4148	2
2	Capacitors	0.1μF	1
3	D.C Power supply variable	0 - 30 Volts / 4 A, 0-5V / 1A	1
4	Inductors	10μH	1
5	Resistors	$1k\Omega$, $10k\Omega$	2,4
6	BJT	SL100, BC177	1,1
7	Power MOSFET	IRF540	2
8	Connecting wires	~	~

Theory

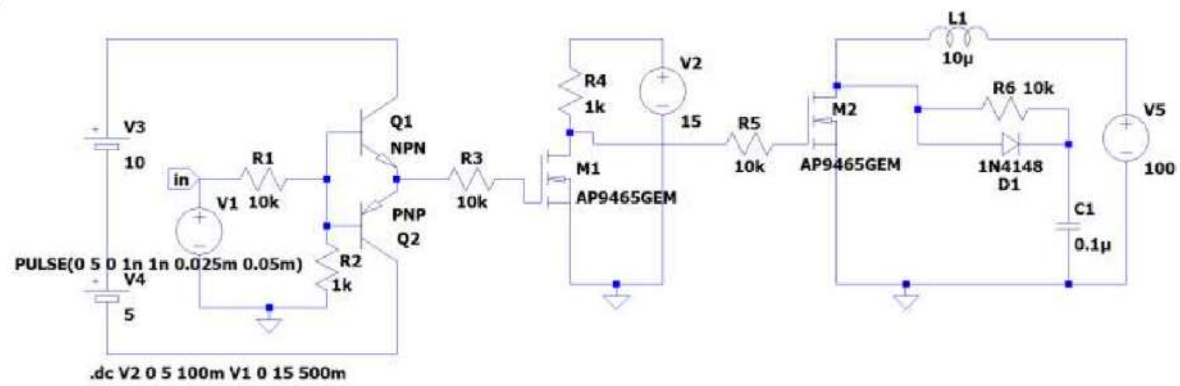
Snubber circuits modify the device switching characteristics, thus reducing transient stress. The snubber circuit action involves energy storage in either an inductor or capacitor. In resetting these passive components its usual to dissipate the stored energy as heat. Thus, circuits with snubber will be less efficient. Instead of dissipating snubber energy stored in the inductor and capacitor, alternately one may recover the energy either back into the supply or into the load. In most power semiconductor devices, there is a danger of exceeding voltage and current ratings of devices during turn off and turn on instants. For example, when BJT is being turned off, the current through device falls rapidly to zero, thus a large voltage spike due to L will occur across device and cause large spike during fall time. Thus, voltage stresses are reduced by turn off snubber circuits or shunt snubber.

On the other hand, during turn on, there will be huge surge current through the devices which could damage the device. The turn on current stress is reduced by using circuits called turn on snubber circuits or series snubber.

Circuit Diagram:



Shunt Snubber Circuit.



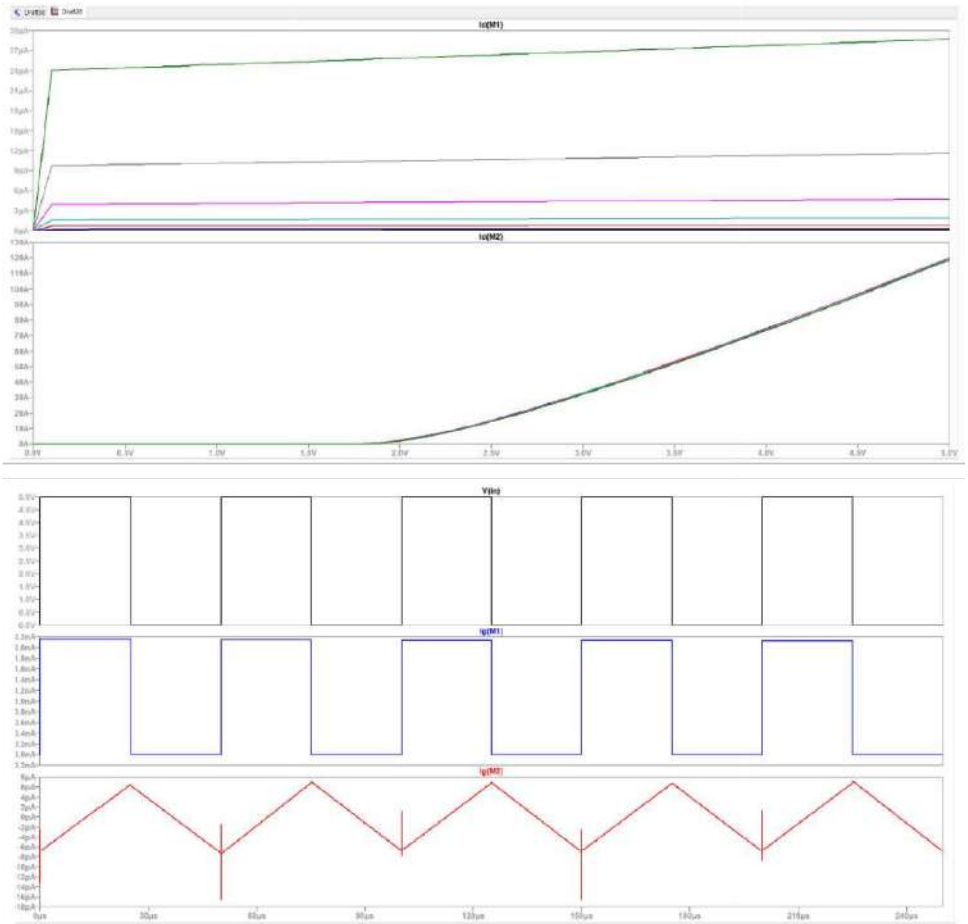
Series Snubber Circuit.

Procedure

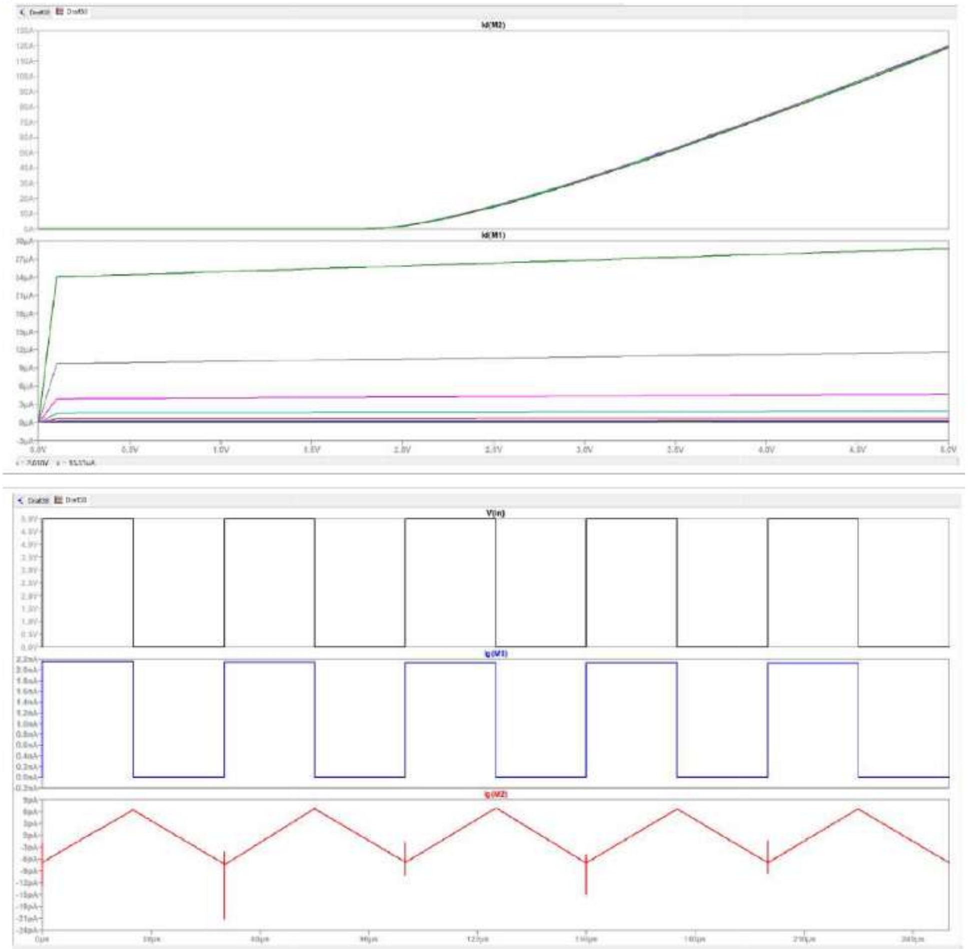
1. Connect the circuit according to the circuit diagram.

2. Simulate and obtain the waveform.

Expected Waveforms:



Expected Waveform for Shunt Snubber.



Expected Waveform for Series Snubber.

Result

Shunt and series snubbers are setup, simulated and corresponding waveforms are obtained.

EXPERIMENT NO. 4

SINGLE PHASE CONTROLLED RECTIFIERS WITH R,R_L LOADS

Aim

To design, implement and study the waveforms of a single phase controlled rectifier with R,R_L loads.

Components Required:

SL. No.	Components and equipments	Specifications	Quantity
1.	Rectifier	2P4M SCR	1
2.	Transformer	230-24V	1
3	Resistor	100 Ω, 1M Ω	1, 1
4	Potentiometer	100k Ω	1
5	Inductor	10mH	1

Theory

A single-phase controlled rectifier is a circuit that converts an AC supply into a controlled DC output using silicon controlled rectifiers (SCRs). Unlike uncontrolled rectifiers, where diodes dictate the conduction period, SCR allows precise control over the firing angle, enabling regulation of the DC output voltage. By delaying the turn on of the SCRs within each AC cycle, the average output voltage can be adjusted, making the circuit suitable for applications requiring variable DC power, such as motor speed control and industrial heating.

The output voltage and current characteristics depending on whether the load is purely resistive (R load) or contains inductance (R_L load).

For a resistive (R) load, the output waveform closely follows the input AC cycle whenever the SCR is triggered. Since there is no energy storage element, the current drops to zero immediately when the AC cycle crosses zero, turning OFF the SCR naturally. However, with an inductive (R_L) load, the stored energy in the inductor causes the current to continue flowing even after the AC voltage has reversed polarity. This results in a more continuous current waveform, but also introduces the possibility of commutation issues, requiring careful control of the firing angle to prevent undesired conduction. The ability to control the output voltage and power makes this rectifier ideal for DC motor drives, heating control and battery charging applications, where precise voltage regulation is essential.

Design:

$V_{cc} \equiv 230\,\text{V}$

$V_m \equiv 230\sqrt{2}\,\text{V}$

For 2P4M SCR:

Maximum current of gate terminal $\equiv 0.2\,\text{A}$

Maximum on-state voltage $(V_{gm}) \equiv 1.7\,\text{V}$

Maximum gate trigger current $(I_{gm}) \equiv 75\,\mu\text{A}$

Firing angle $(\alpha) \equiv 45^\circ$

$V \equiv V_{\text{peak}} \times \sin 45^\circ = 20 \times \sin 45^\circ = 14.1\,\text{V}$

$R \equiv V/I = 14.1/(75 \times 10^{-6}) = 180\,\text{k}\Omega$

$R_2 \equiv 1\,\mathrm{M\Omega}$ POT

$R_L \equiv 1\,\mathrm{k\Omega}$ load

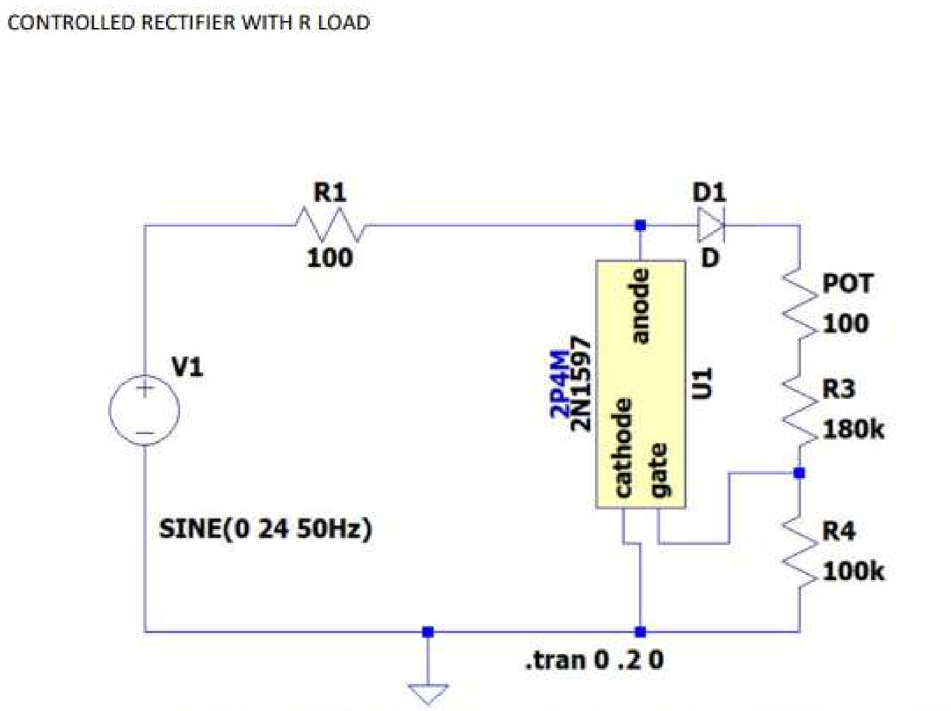
$f \equiv 50\,\mathrm{Hz}$

$\alpha \equiv 45°$

$T \equiv 0.02\,\mathrm{s} = 20\,\mathrm{ms}$

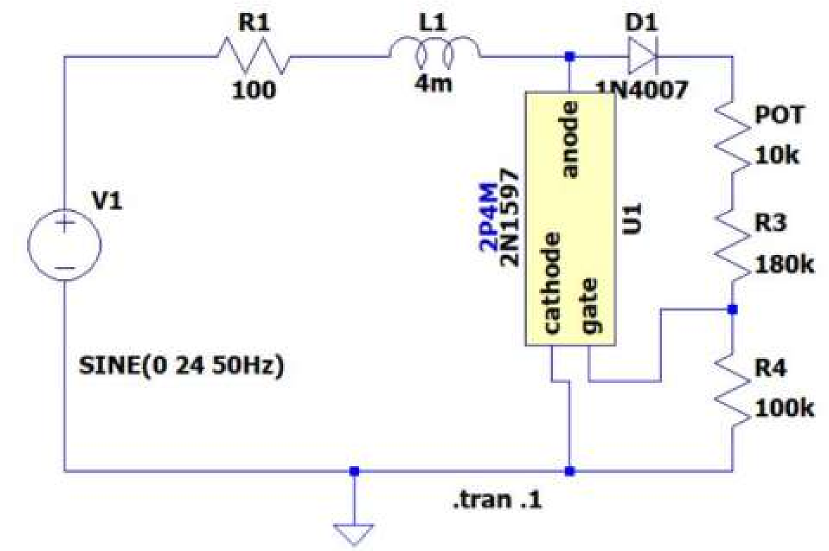
$D \equiv (\alpha \times T)/360 = 2\,\mathrm{ms}$

Circuit Design:



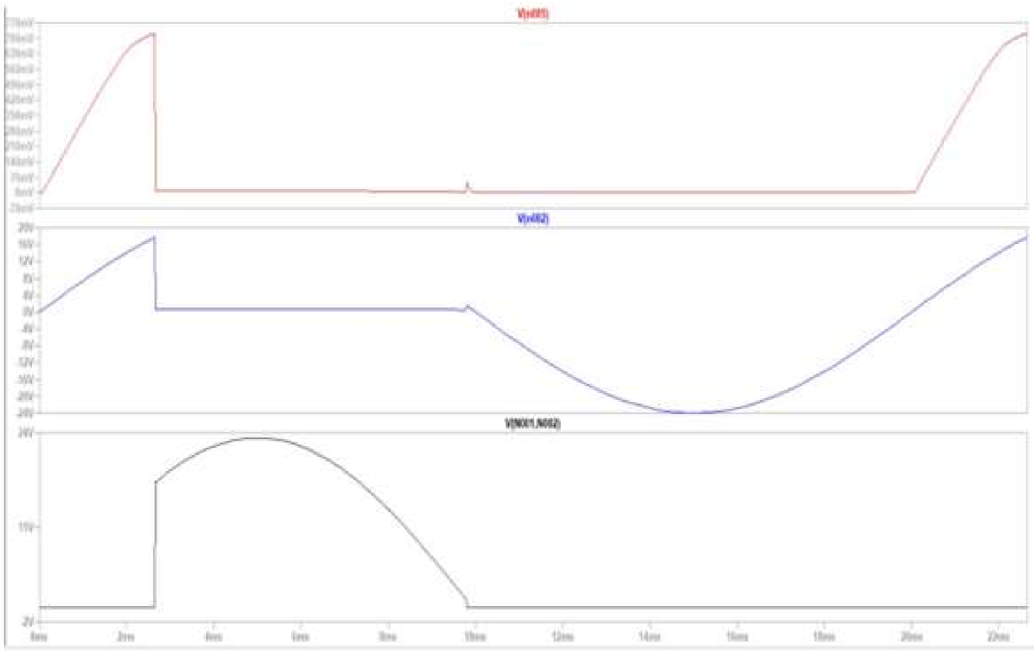
Controlled Rectifier with R Load.

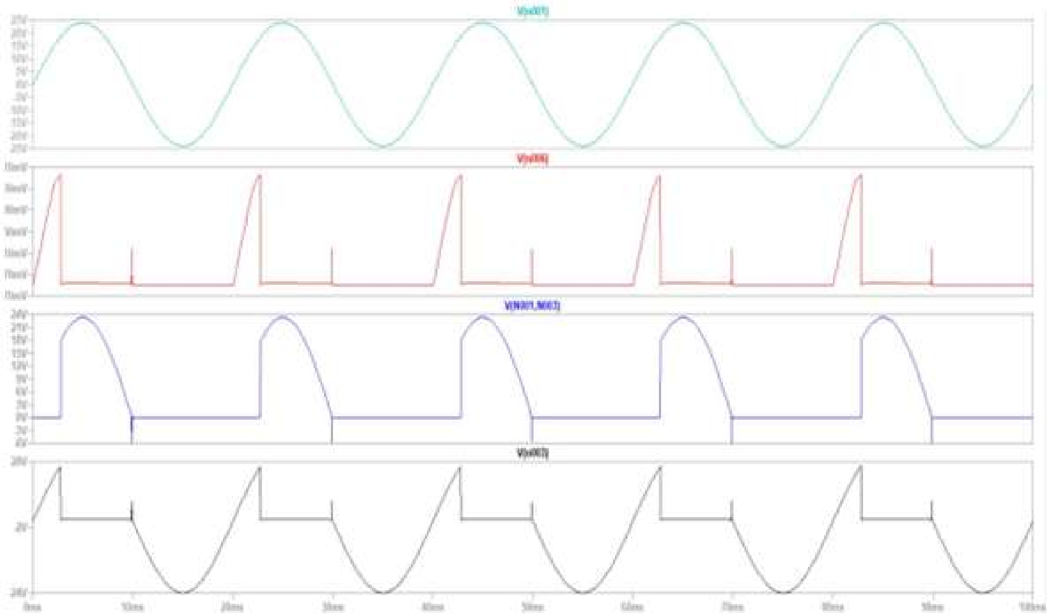
CONTROLLED RECTIFIER WITH RL LOAD



Controlled Rectifier with R_L Load.

Expected Waveform:





Result:

Designed and implemented controlled rectifier with R and R_L load.

EXPERIMENT NO. 5

Realization of Buck, Boost, and Buck-Boost converters

Study its Continuous and Discontinuous mode

Aim

To realize and simulate buck, boost, and buck-boost converters in LTspice and study its continuous and discontinuous conduction mode.

Components and Equipments Required:

Sl No	Components and equipments	Specification	Quantity
1.	Mosfet	BSZ036NE2LS	1
2.	Diode	MBR735	1
3.	Inductor	470μ	1
5.	Capacitor	48μ	1
6.	Resistor	50Ω	1

Theory

A buck converter (step-down converter) is a DC-to-DC power converter that steps down voltage (while stepping up current) from its input (supply) to its output (load).

$$V_o/V_{in} = D$$

D varies between 0 and 1.

A boost converter (step-up converter) is a DC-to-DC power converter that steps up voltage (while stepping down current) from its input (supply) to its output (load).

$$V_o/V_{in} = 1/(1 - D)$$

D varies between 0 and 1. But if D = 1 then the ratio of the output voltage to the input voltage at a steady state goes to infinity, which is not physically possible. In a practical boost converter the duty cycle, if kept at a value greater than 0.7 will lead to instability.

The buck-boost converter is a type of DC-to-DC converter that has an output voltage magnitude that is either greater than or less than the input voltage magnitude.

$$V_o/V_{in} = D/(D - 1)$$

D varies between 0 and 1. If D ¿ 0.5, the output voltage is larger than the input; and if D ¡ 0.5, the output is smaller than the input. But if D = 0.5 the output voltage is equal to the input voltage.

During continuous conduction mode, the inductor current in the energy transfer never reaches zero value. In discontinuous conduction mode, the inductor current falls to zero level which is common in DC-to-DC converters.

In the discontinuous conduction mode, the inductor current is not persistent throughout the complete cycle and reaches zero level earlier even before the end of the period. Discontinuous conduction mode inductance is less than the minimum value of the inductance for the continuous conduction mode, $L_{DCM} < L_{CCM}$. In the case of the discontinuous conduction mode the $L_{DCM} = \xi L_{CCM}$ where $0 < \xi < 1$ for the discontinuous conduction.
Thus, this condition generally arises for the light-load condition.

Design:

Buck Converter

Let $T \equiv 0.02$ms

$V_{in} \equiv 12$V

$D \equiv 0.5$

$V_o/V_{in} \equiv D$

$V_o \equiv 6$V

$C \equiv T\Delta I/8\Delta V$

Let $C \equiv 48\mu$F

$L \equiv 470\mu$H

$R \equiv 50\Omega$

Boost Converter

Let $T \equiv 0.02$ms, $V_{in} \equiv 12$V

$D \equiv 0.5$

$V_o/V_{in} \equiv 1/(1 - D)$

$V_{out} \equiv 24$V

$T_{on} \equiv 0.01$ms, $I_o \equiv V_o/R$

Let $R \equiv 50\Omega$

$C \equiv T\Delta I/8\Delta V$

Let $C \equiv 48\mu$F and $\Delta V = 100$mV, $L \equiv 470\mu$H

Buck-Boost Converter

Let $T \equiv 0.02$ms, $V_{in} \equiv 12$V

$D \equiv 0.5$

$V_o/V_{in} \equiv D/(D - 1)$

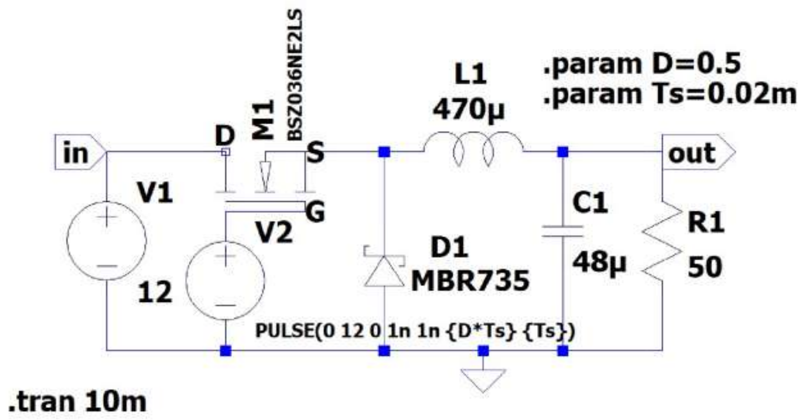
$V_o \equiv -6$V

$C \equiv T\Delta I/8\Delta V$

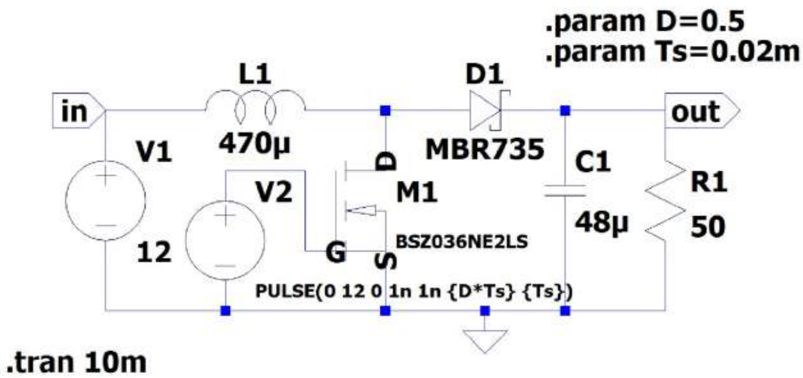
Let $C \equiv 48\mu$F, $L \equiv 470\mu$H

$R \equiv 50\Omega$

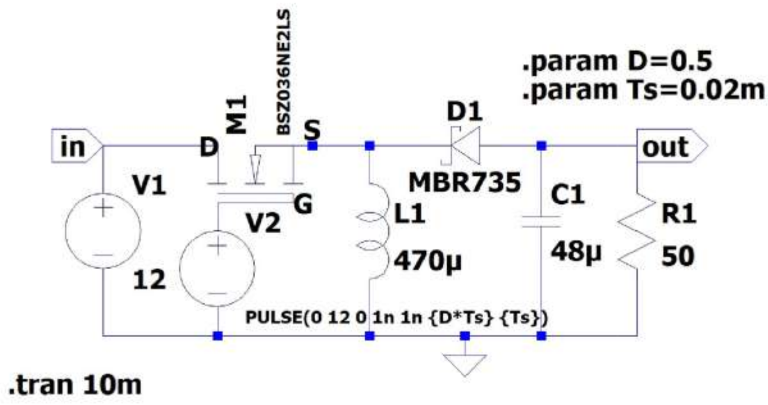
Circuit Diagram:



Buck Converter.



Boost Converter.



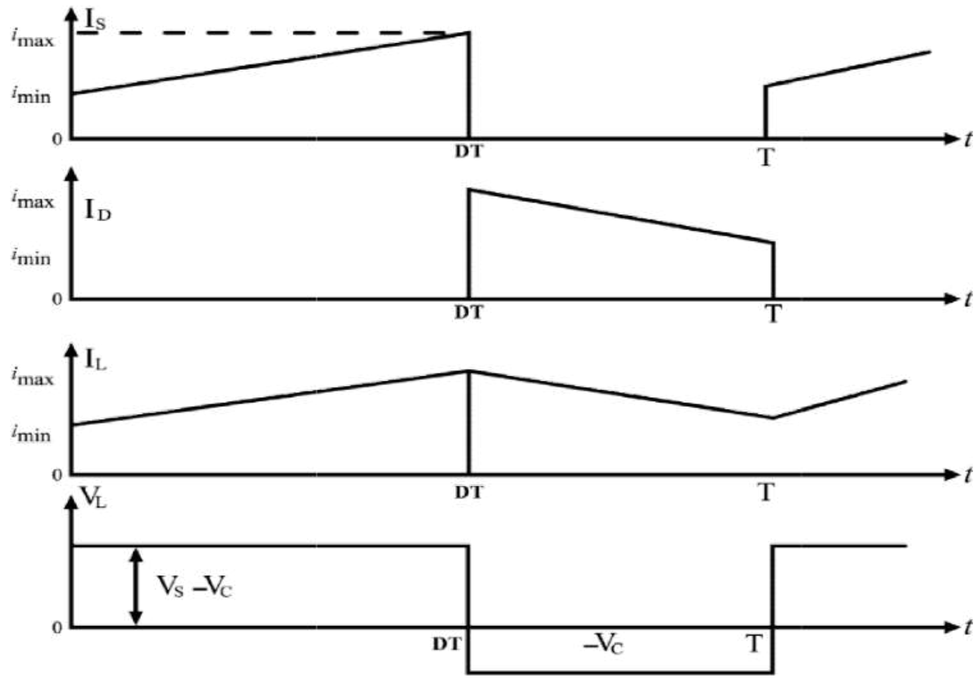
Buck-Boost Converter.

Procedure

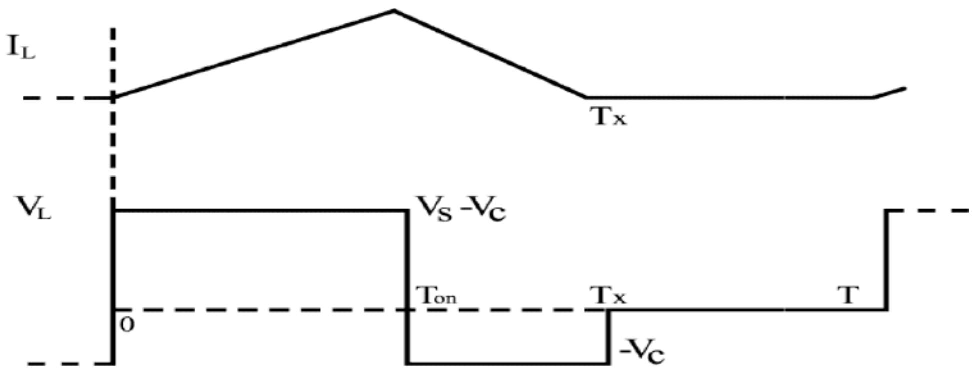
1. Connect the circuit as per the circuit diagram in LTspice.

2. Simulate and obtain the corresponding waveform.

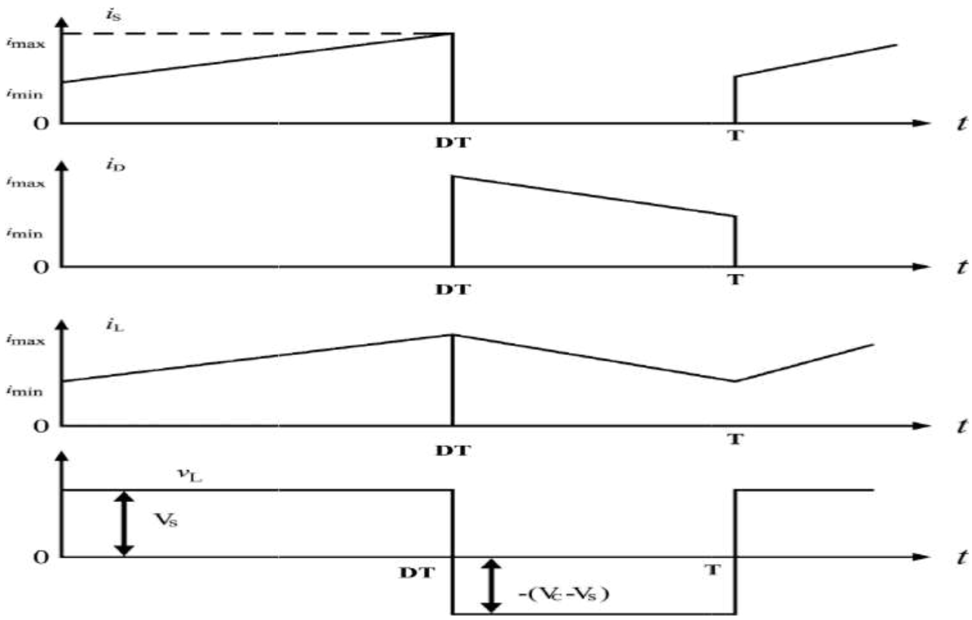
Expected Waveforms



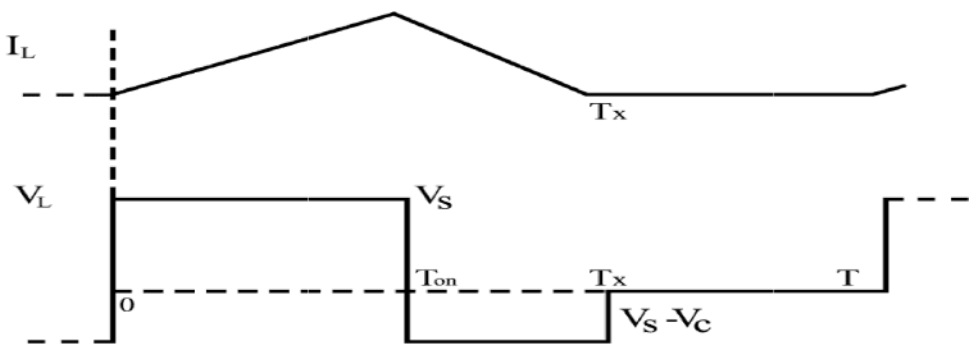
Supply current, diode current, inductor current, and inductor voltage waveforms for a buck converter in continuous mode.



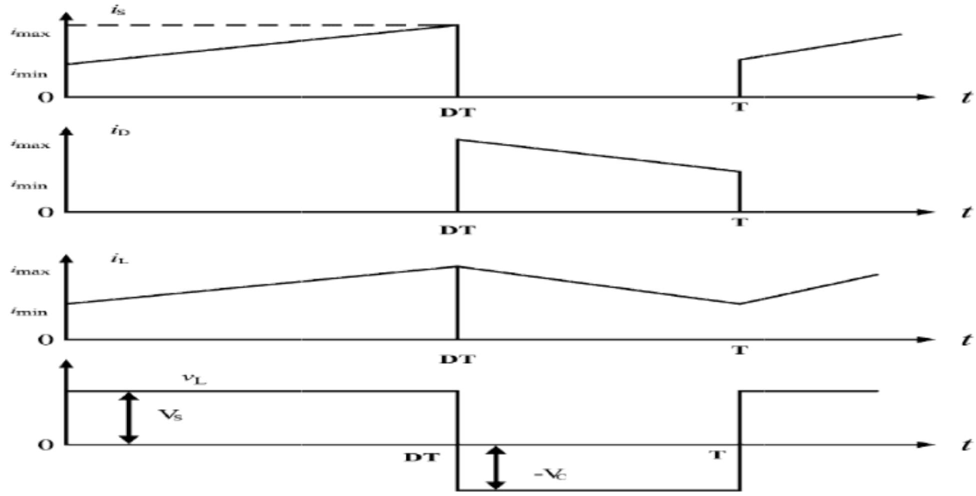
Inductor current and voltage for the discontinuous conduction mode of a buck converter.



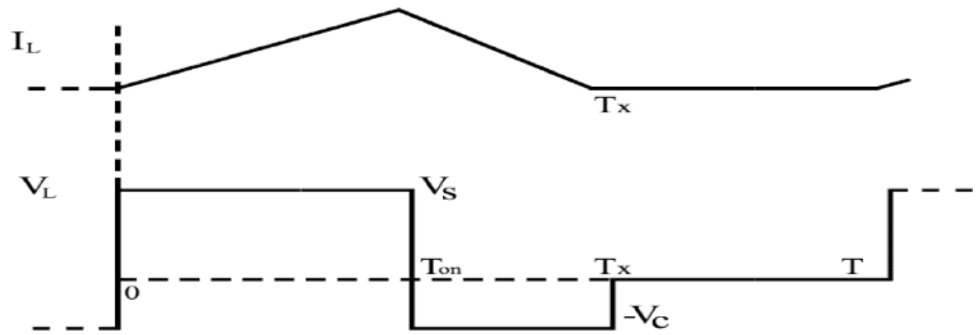
Supply current, diode current, inductor current, and inductor voltage for a boost converter in continuous conduction mode.



Inductor current and voltage for the discontinuous conduction mode of a boost converter.



Supply current, diode current, inductor current, and inductor voltage for a buck-boost converter in continuous conduction mode.



Inductor current and inductor voltage for the discontinuous conduction mode of the buck-boost converter.

Result:

The buck, boost, and buck-boost converter circuits are simulated and waveforms are obtained.

Buck Converter: V_{out} = ________

Boost Converter: V_{out} = ________

Buck-Boost Converter: V_{out} = ________

EXPERIMENT NO. 6

REALIZATION OF ISOLATED CONVERTERS: PUSH-PULL, HALF BRIDGE AND FULL BRIDGE CONFIGURATION.

Aim

To design a half bridge and full bridge converters.

Components Required:

Sl.No.	Components and equipments	Specification	Quantity
1	PWM IC	TL494	1
2	MOSFET	IRF840	4
3	Transistor	BC 177	2
4	Resistor	100Ω, 1kΩ, 4.7kΩ, 56kΩ	2, 2, 1, 1
5	Diode	1N4007	2
6	Capacitor	0.01μF	1

Theory

Half bridge inverter :

A half-bridge DC to AC converter is a type of inverter that converts a DC voltage into an AC output using two power switches and a centre tapped transformer or a capacitor divider. The circuit operates by alternately switching the two power MOSFETs, creating a square wave or modulated AC waveform across the load. The TL494 pulse width modulation (PWM) controller is commonly used to generate the required control signals for the switching devices, ensuring stable and efficient operation.

The half-bridge configuration includes two IRF840 MOSFETs, two capacitors connected in series across the DC supply, and a resistor as the load. The midpoint of the capacitors serves as a reference point, resulting in an output voltage that swings between +V/2 and –V/2. When the first MOSFET turns on, the load sees a positive voltage and when the second MOSFET turns on, the load sees a negative voltage. The TL494 generates complementary PWM signals to control the switching sequence, regulating the output waveform's frequency and amplitude. Half-bridge inverters are widely used in medium-power applications such as motor drives and uninterruptible power supplies (UPS) due to their efficiency and cost-effectiveness.

Full Bridge Inverter

A full-bridge DC to AC converter, also known as an H-bridge inverter, is an advanced topology that improves upon the half-bridge configuration by using four power MOSFETs instead of two. This setup allows the full utilization of the DC supply voltage, thereby increasing the power and voltage output across the load. The TL494 PWM controller is used to generate switching signals that ensure the correct operation of the bridge circuit, enabling the conversion of DC to a well-controlled AC waveform.

In the full-bridge configuration, the four IRF840 MOSFETs operate in pairs. M1 and M4 conduct together in one halfcycle, while M2 and M3 conduct in the next halfcycle. This alternating switching pattern allows the output voltage to swing from +V to -V, effectively doubling the voltage compared to the half-bridge design. The TL494 ensures precise control over the PWM duty cycle, influencing the output waveform's shape and reducing harmonic distortion. A resistor is used as load in this circuit. Due to its superior efficiency and power handling capability, the full-bridge inverter is widely used in high-power applications such as industrial motor drives, electric vehicle power conversion and renewable energy systems like solar inverters.

Design:

$f_{out} \equiv 12.5\,\text{kHz}$

Therefore, $f_{osc} \equiv 2 \times f_{out} = 25\,\text{kHz}$

$f_{osc} \equiv \frac{1.1}{R_T \times C_T}$

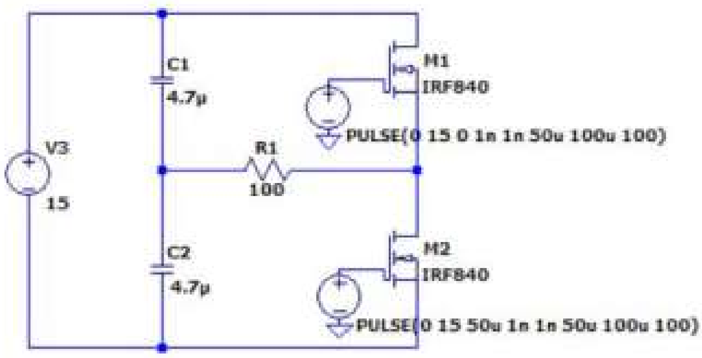
Let $C_T \equiv 0.01\,\mu\text{F}$

Therefore, $R_T \equiv \frac{1.1}{f_{osc} \times C_T}$

$\equiv \frac{1.1}{25 \times 10^3 \times 0.01 \times 10^{-6}}$

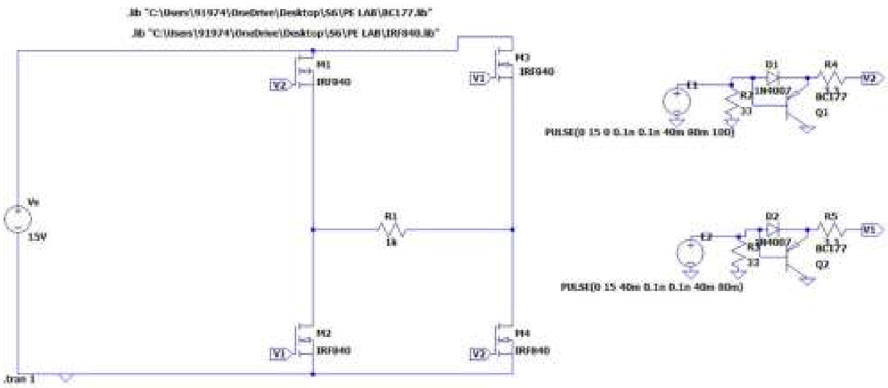
$\approx 4.7\,\text{k}\Omega$

Circuit Diagram



Half Bridge Converter Circuit.

Full Bridge Converter

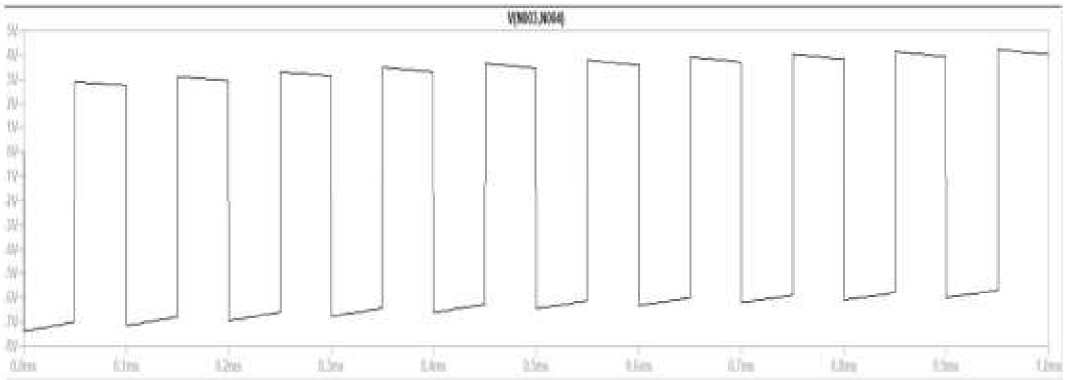


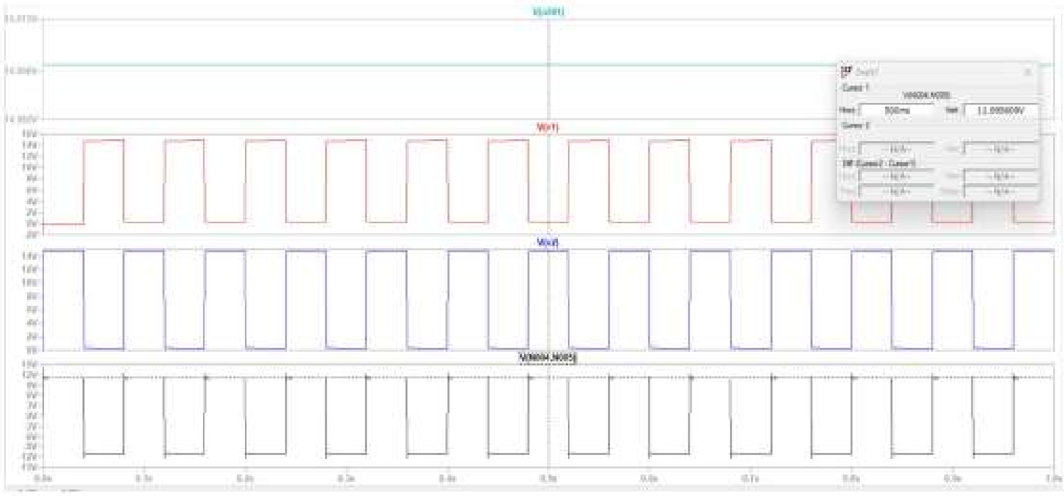
Full Bridge Converter Circuit.

Procedure:

1. Connect the converter circuits as shown in the provided diagrams for push-pull, half-bridge, and full-bridge configurations using the TL494 IC.
2. Apply the DC input after checking all connections and observe the AC output waveforms across the load using an oscilloscope.
3. Compare the waveforms of the three configurations and note the differences in voltage and shape.

Expected Waveforms:





Result:

Designed half bridge and full bridge inverters using MOSFET.

EXPERIMENT NO. 7

DC TO AC INVERTER USING MOSFET

Aim

To design a DC to AC inverter using MOSFET.

Components Required:

SL.NO.	Components and equipments	Specifications	Quantity
1.	MOSFET	IRF540	2
2.	PWM IC	TL494	1
3.	Transformer	12-0-12/230V	1
4.	Transistor	BC 177	2
5.	Diode	1N4007	2
6.	Resistor	1kΩ, 100Ω, 100kΩ, 56kΩ	2, 2, 1, 1
7.	Capacitor	0.1 μF	1

Theory

DC to AC inverters are one of the most important elements in power electronics. There are a variety of applications based on such conversions, i.e., inverters convert direct current input (DC) into alternating current output (AC). The TL494 generates a pulse width modulated signal to control the switching of the MOSFETs, ensuring efficient conversion. The H-bridge arrangement allows current to flow in both directions through the transformer, producing an AC output. The transformer steps up the low voltage AC generated by the MOSFETs to 230V AC, making the circuit suitable for powering household appliances such as UPS systems, renewable energy applications and emergency power supplies.

Design:

$F_{out} \equiv 50\,\text{Hz}$

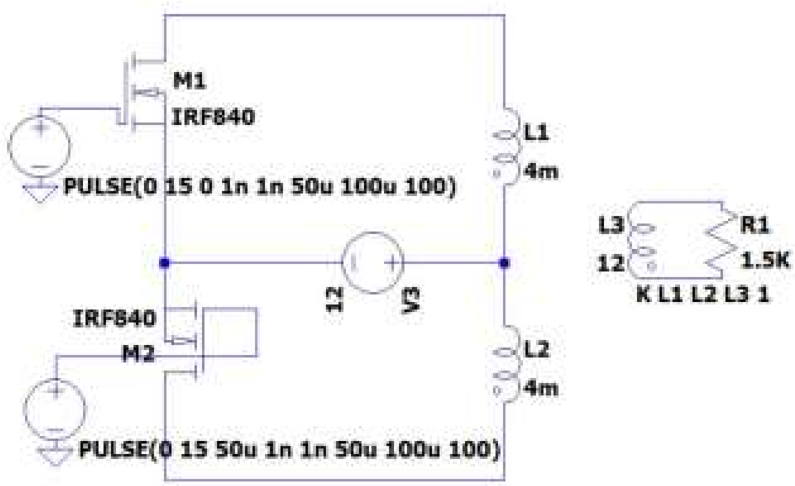
$\therefore F_{osc} \equiv 2 \times F_{out} = 100\,\text{Hz}$

$100 \equiv \frac{1.1}{R_T \times C_T}$

Let $C_T \equiv 0.1\,\mu\text{F}$

$R_T \equiv \frac{1.1}{100 \times 0.1 \times 10^{-6}} = 100\,\text{k}\Omega$

Circuit Design:



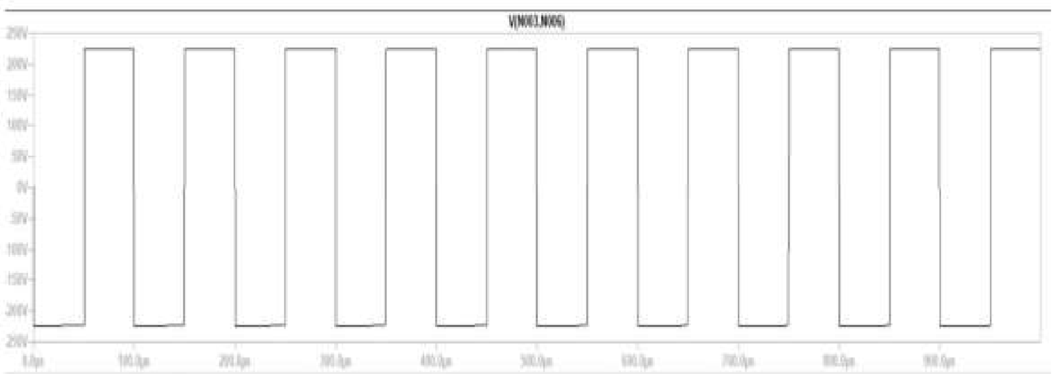
Procedure:

1. Connect the DC to AC inverter circuit as shown in the provided circuit diagram using the TL494 IC, IRF540 MOSFETs, and the 12-0-12/230V transformer.

2. After verifying all connections, power the circuit with the DC supply and observe the AC output waveform across the transformer secondary using an oscilloscope.

3. Measure the output voltage and confirm the conversion of DC input to 230V AC output.

Expected Waveforms:



Result:

Designed and implemented DC to AC inverter using MOSFET.

EXPERIMENT NO. 8

DC MOTOR SPEED CONTROL

Aim

To model and simulate speed control for DC motor using SCR.

Components and Equipments Required:

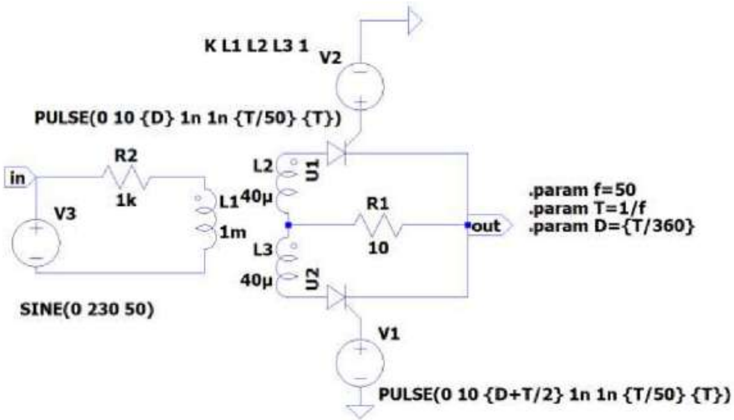
Sl No	Components and equipments	Specification	Quantity
1	Step-down transformer	230/24 Volt /50 Hz/1 A	1
2	SCR	Choose 2P4M	1
3	D.C Power supply variable	0 - 30 Volts / 4 A, 0-5V / 1A	1
4	Resistors	1kΩ, 40μΩ	1,2
5	Connecting wires	~	~

Theory

A DC motor is a rotary electrical motor that converts direct current electrical energy into mechanical energy. The speed of DC motor can be controlled by using SCR as shown in the circuit diagram. The SCR rectifier, the output voltage of the transformer and apply this pulsating DC voltages to armature winding (here a load resistance is used to represent DC motor load).

The output of SCR rectifier can be controlled by controlling gate current of SCR. So, this way SCR can operate at different levels of conduction and apply various voltages to the motor armature winding and speed of motor increases. When voltage at gate of SCR decreases, the gate current decreases, thus the output voltage at the load (armature coil) decreases thus speed of motor decreases. Firing angle can be varies to control on turn of SCR.

Circuit Diagram



Procedure

1. Connect the circuit according to the circuit diagram.
2. Simulate and obtain the waveform.

Design:

$V_{in} \equiv 230\,\text{V}, 50\,\text{Hz}$

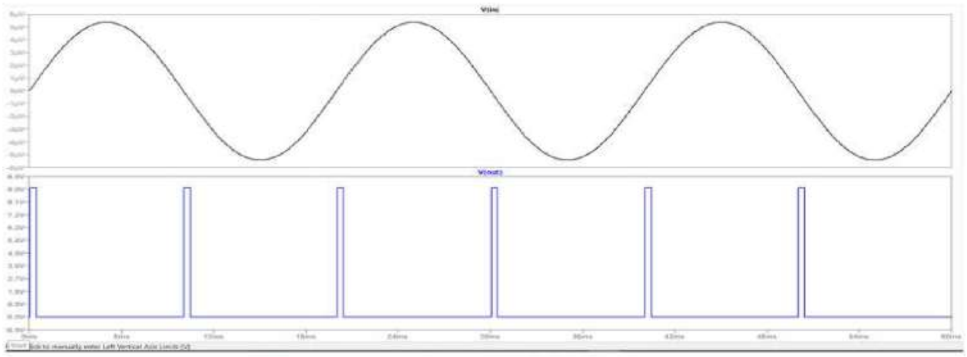
When Gate voltage $V_g \equiv 5\,\text{V}$, $V_{out} = 4.2\,\text{V}$

When Gate voltage $V_g \equiv 10\,\text{V}$, $V_{out} = 9.1\,\text{V}$

That is $V_g \propto V_{out} \propto$ Speed of DC motor

Take Load Resistance (DC motor), $R \equiv 10\,\Omega\,(10\,\text{W})$

Expected Waveforms:



Result:

DC motor speed control circuit using SCR is modelled and simulated waveforms are obtained.