INDEX

EXPERIMENT 1: HALF WAVE UNCONTROLLED RECTIFIER

Objective : To design and simulate a Half wave uncontrolled rectifier circuit using diodes.

Required Hardware Components : Step Down (230:30) Transformer

230 V,50 Hz Single phase AC Supply

One IN4007 Diode

Load Resistor (1 kohm)

Connecting Wires

Soldering Iron and wire

Multimeter

Dotted PCB or Breadboard

Software Tool : MATLAB Simulink

Circuit Diagram :

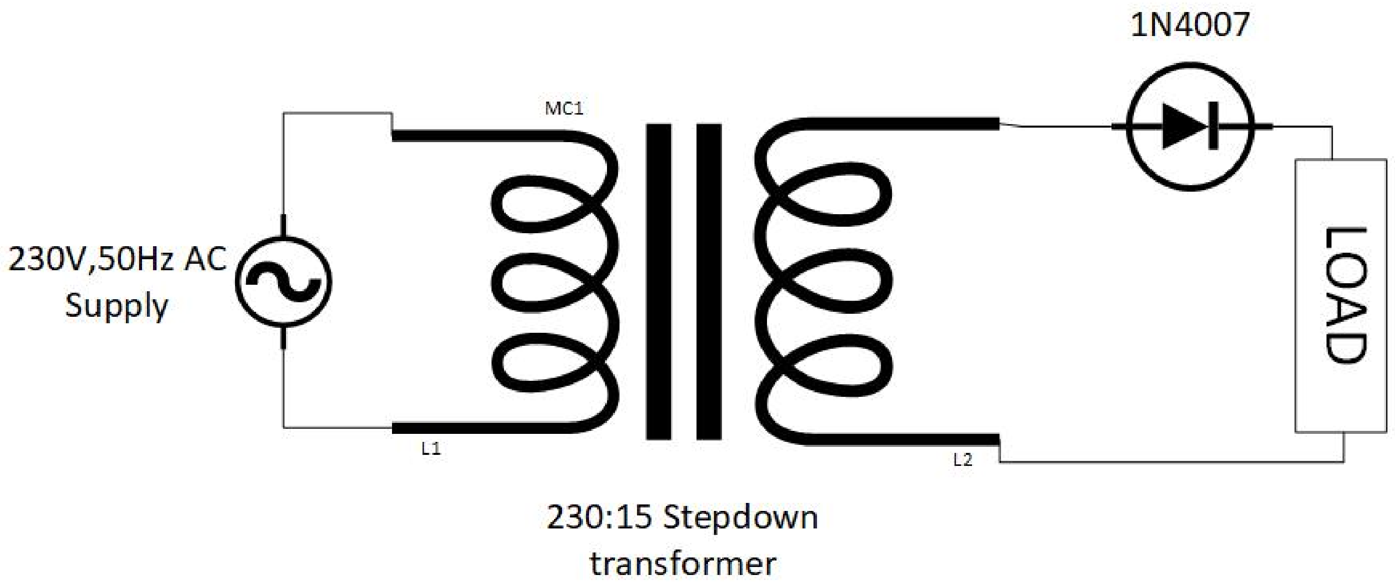


Fig. 1(a) Circuit Diagram of Half Wave Rectifier using Diodes

Part A : Simulation of circuit

Circuit diagram shown in Fig. 1(a) is designed in MATLAB Simulink below,

Note that Snubber circuit of diode is removed.

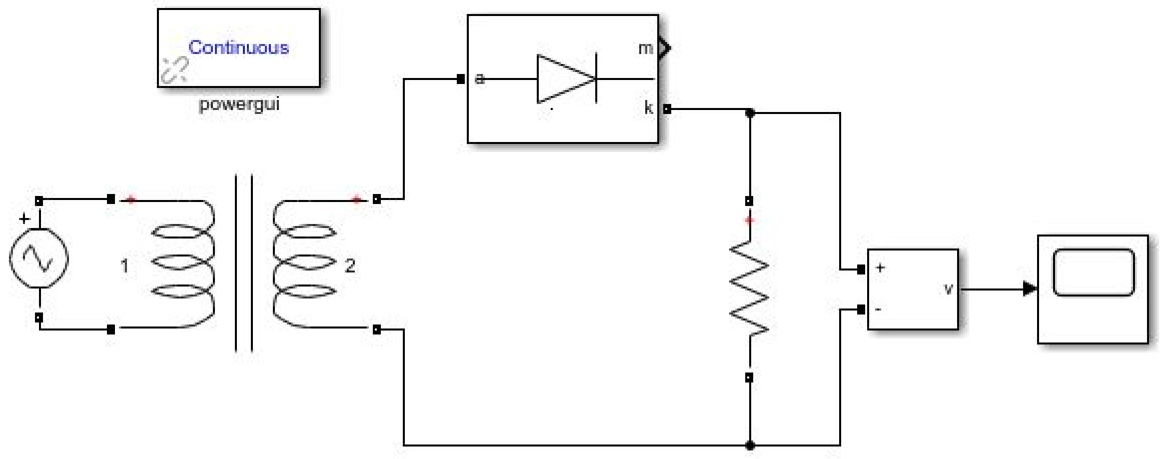


Fig. 1(b) Simulink Model of Half wave rectifier with R load

Output Waveform :

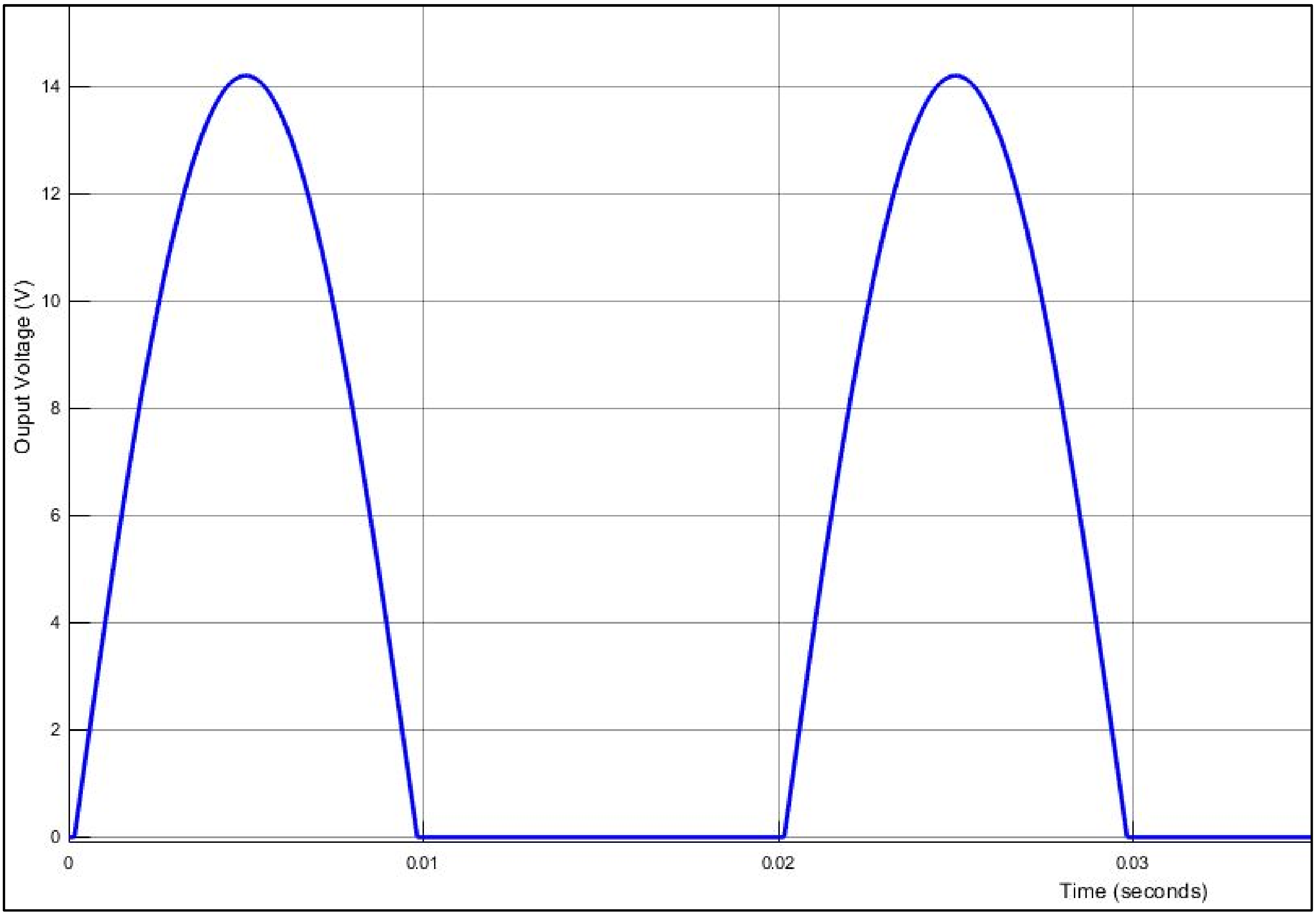


Fig. 1(c) Simulink Output Waveform with Resistive load

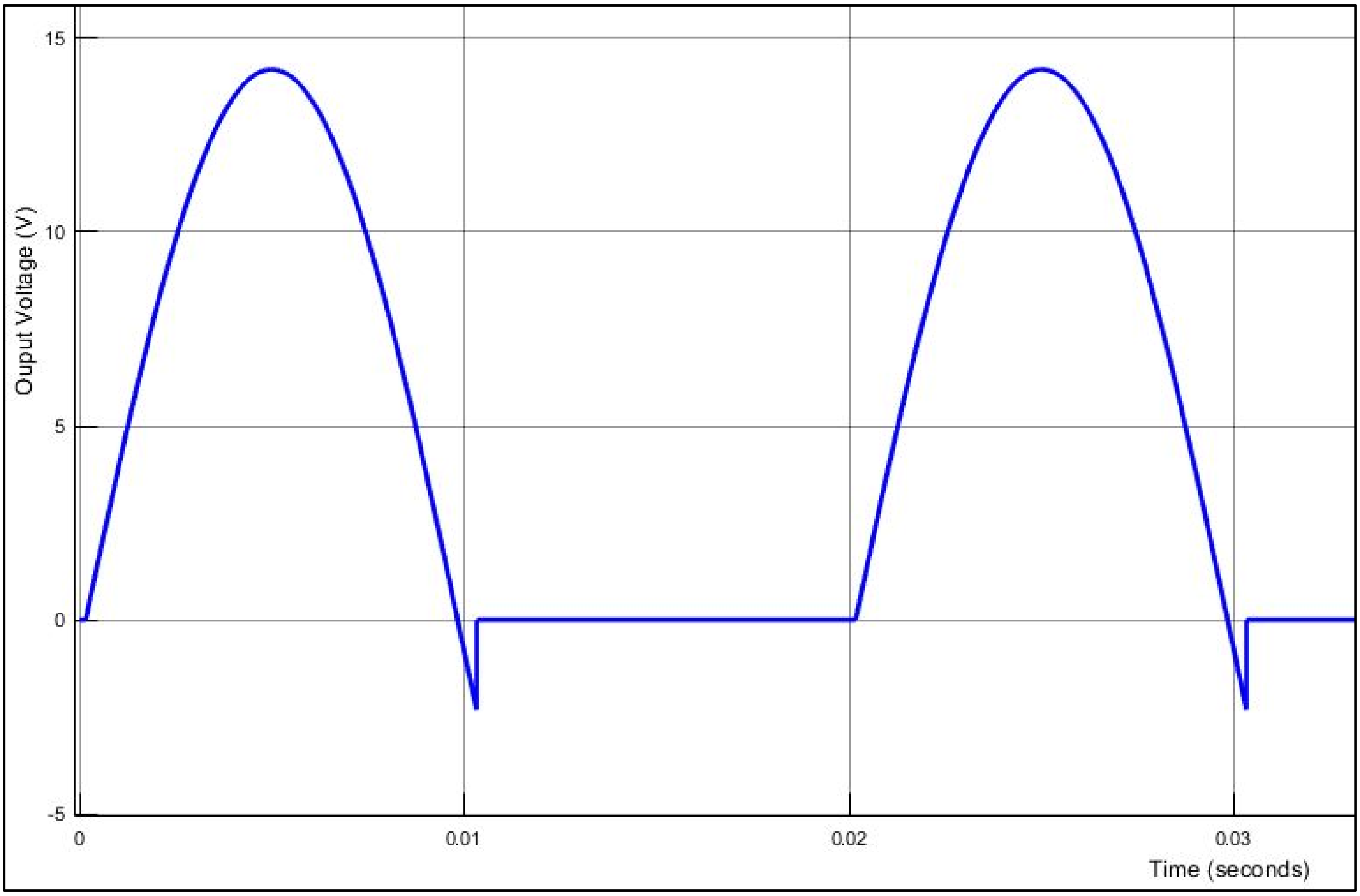


Fig. 1(d) Simulink Output Waveform with R-L load

Note that here L=0.5H is taken.

Observation : Here by comparing both waveforms of Fig. 1(c) & Fig. 1(d) we observe that in R-load there is no output present in negative half cycle while in case of R-L load there is some negative voltage appearing at the starting of negative half cycle due to the fact that inductor opposes change in current.

Result : For R-L load,In positive half cycle, inductor is charged with appropriate polarity. Now when the negative cycle starts, the net effective voltage across diode is such that it will be in forward bias until inductor is discharged. Therefore at starting of negative cycle we observe some negative voltage portion. By changing value of L , we can change this portion. In R-Load, there will not be any negative voltage.

EXPERIMENT 2: FULL WAVE UNCONTROLLED CENTER TAPPED & BRIDGE RECTIFIER

Objective : To design and simulate full wave uncontrolled both center tapped and bridge rectifier circuit.

Required Hardware Components : Step Down (230:30) Transformer

230 V,50 Hz Single phase AC Supply

Six IN4007 Diodes

Two load Resistors 10 kohms

Connecting Wires

Soldering Iron and wire

Multimeter

Dotted PCB or Breadboard

Required Software Tool : MATLAB Simulink

Circuit Diagram :

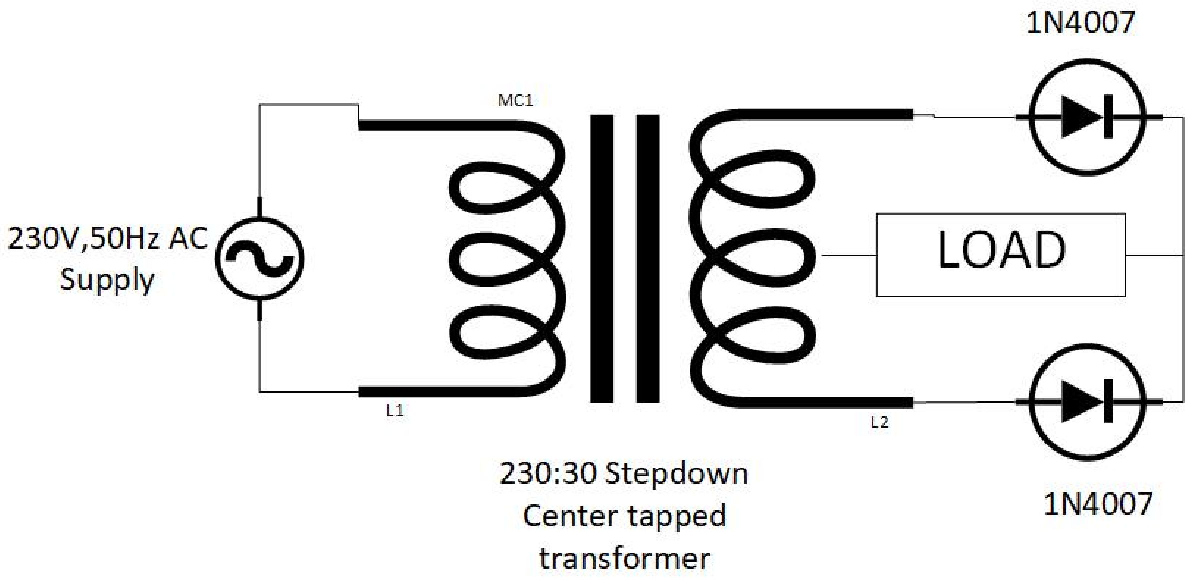


Fig. 2(a) : Circuit Diagram of Center tapped Full Wave Rectifier

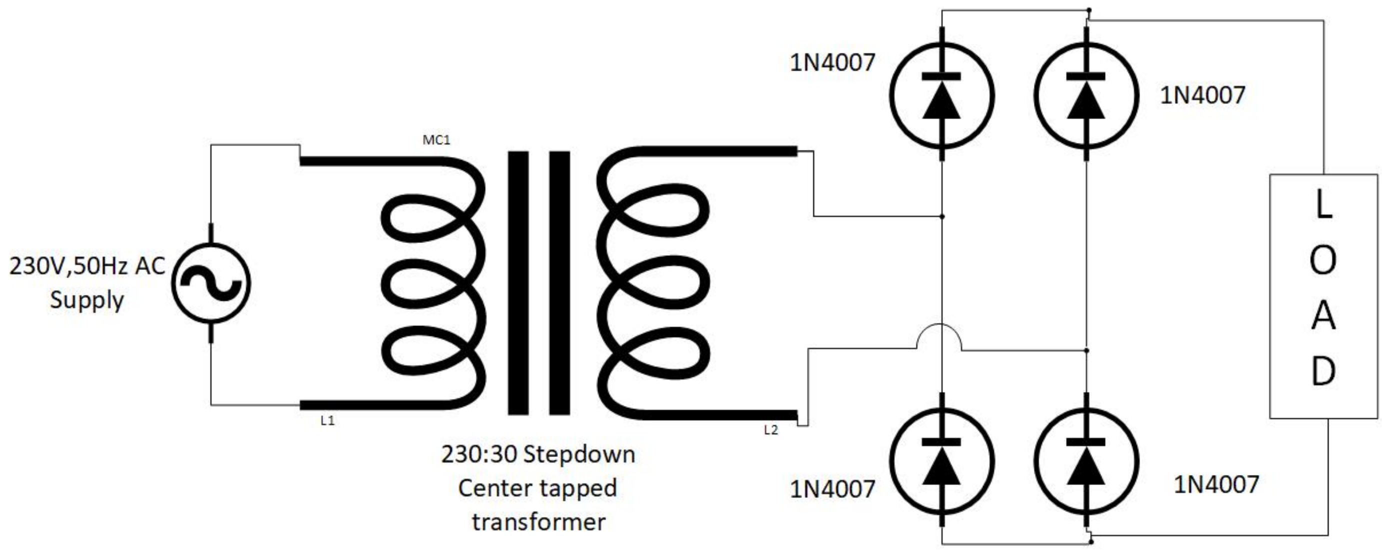


Fig. 2(b) : Circuit Diagram of Bridge Full Wave Rectifier

Part A : Simulation of circuit

Circuit diagram shown in Fig. 2(a) & Fig. 2(b) is designed in MATLAB Simulink below,

Bridge Wave Rectifier Simulation

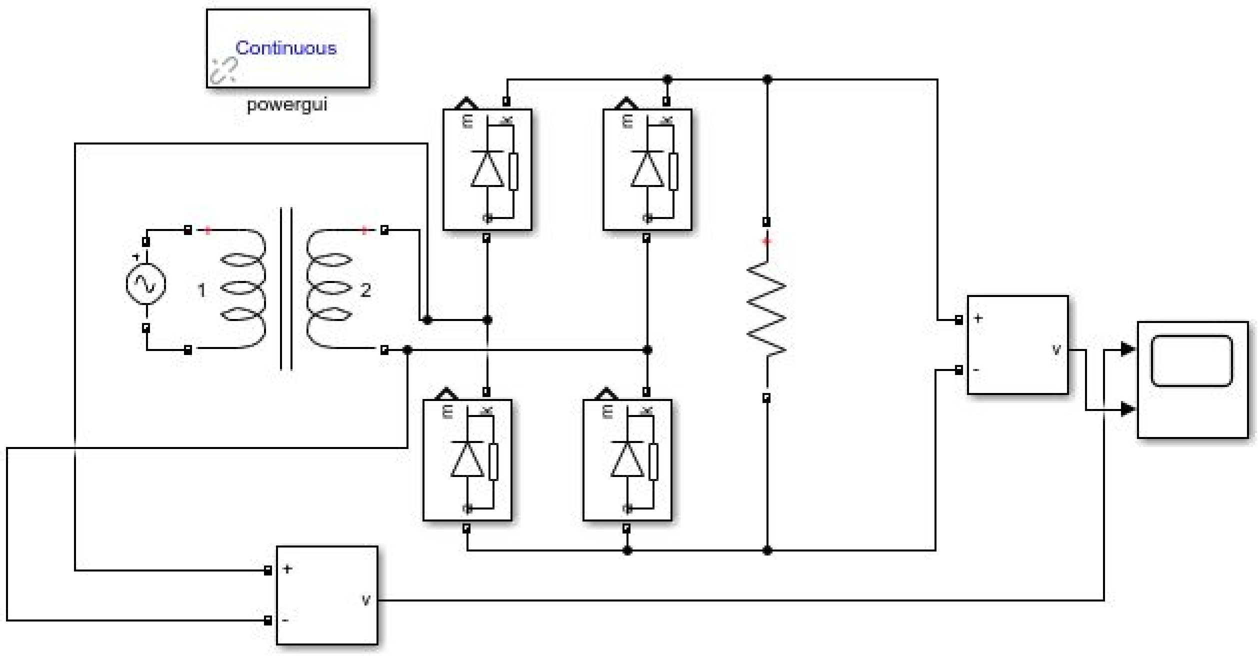


Fig. 2(c) Simulink Model of Bridge Full Wave Rectifier

Output Waveform :

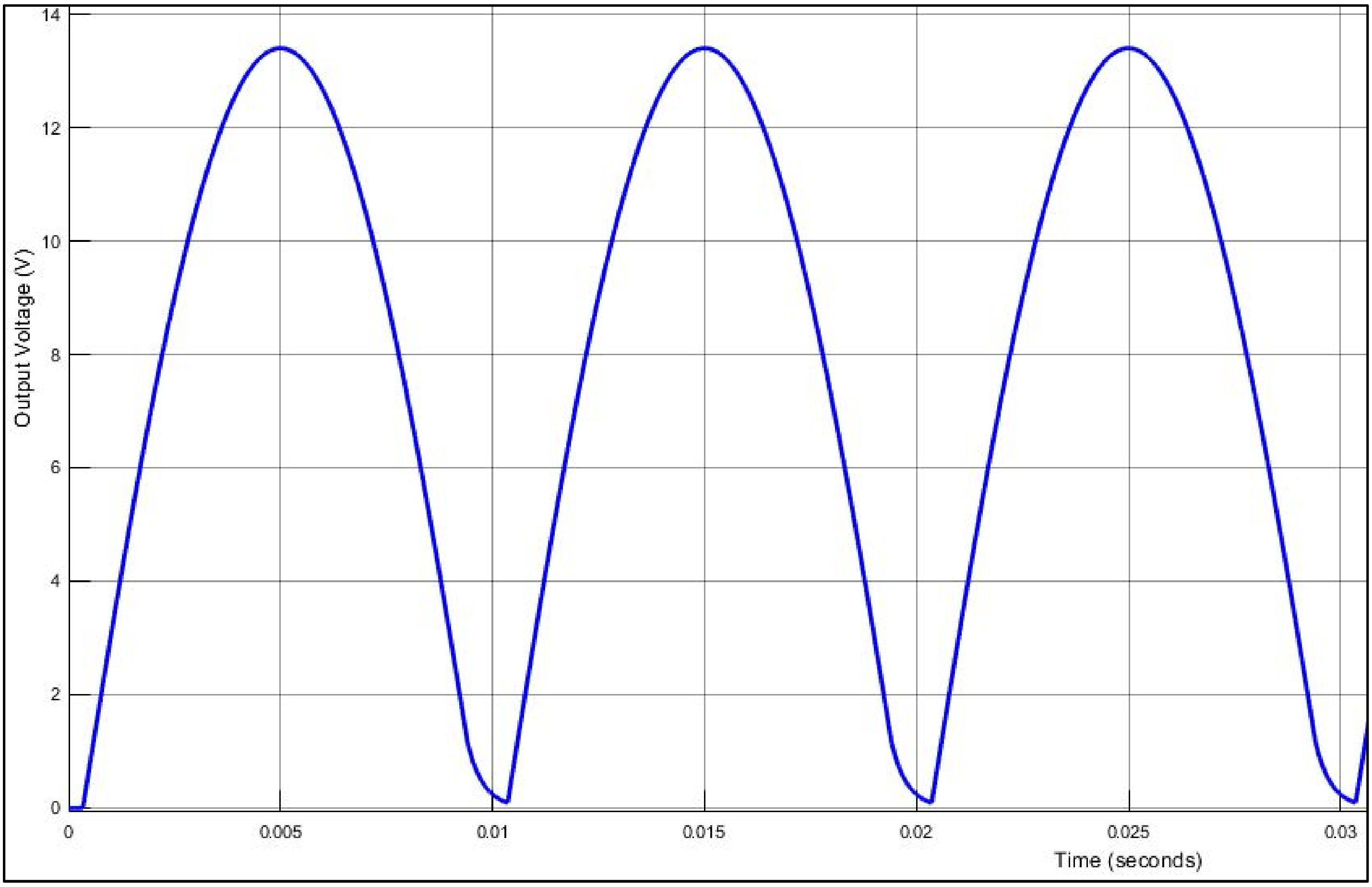


Fig. 2(d) Simulink Output Waveform

Center Tapped Full Wave Rectifier

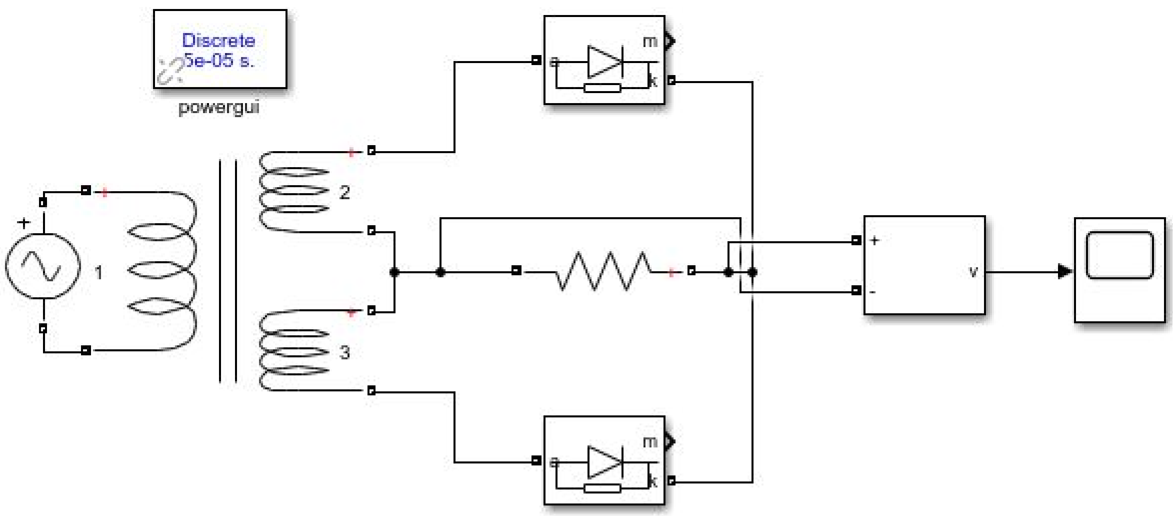


Fig. 2(e) Simulink Model of Center Tapped Full Wave Rectifier

Output Waveform :

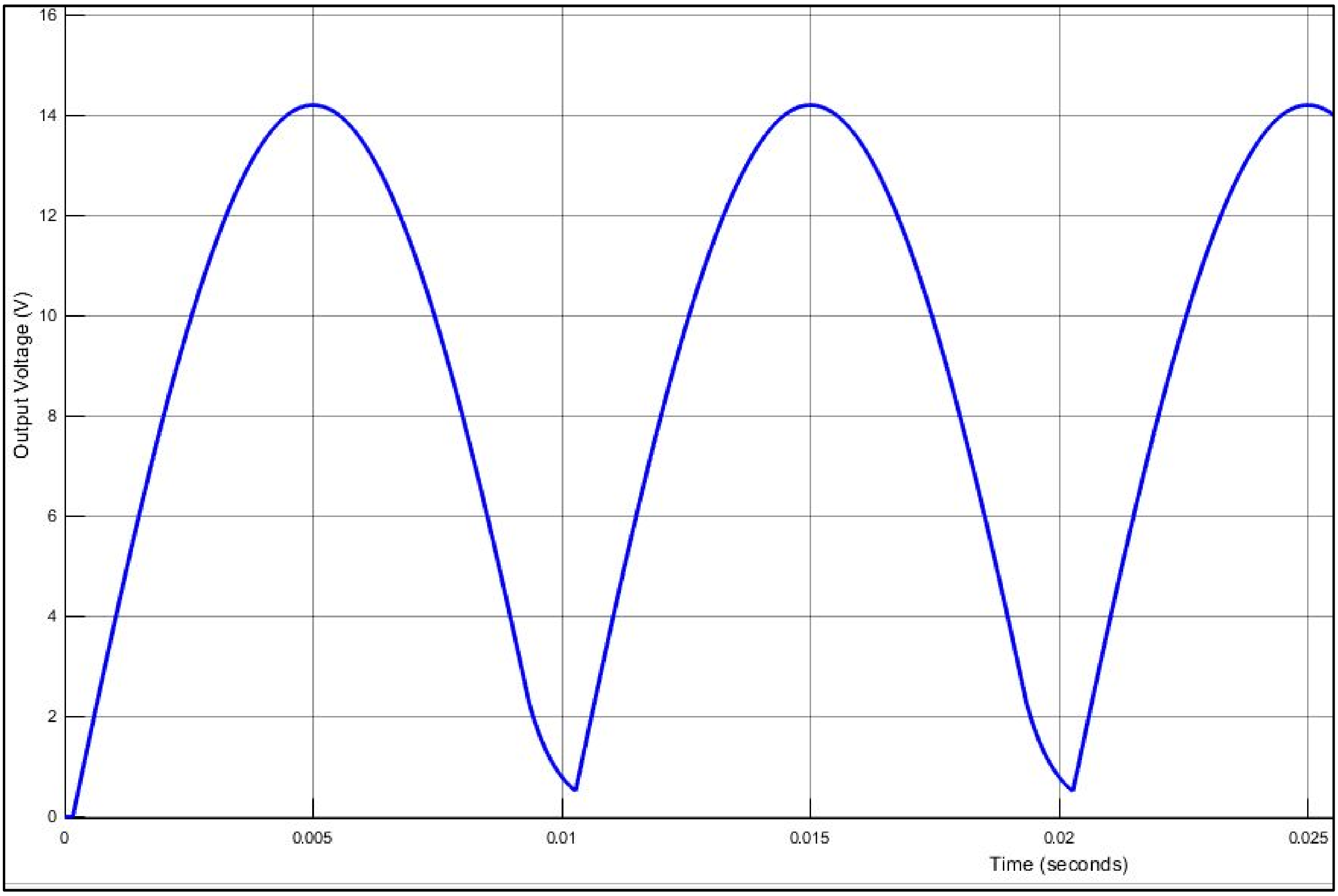


Fig. 2(f) Output Waveform of Center tapped Full wave Rectifier

Observation : We can observe that in Center tapped rectifier, output waveform is slightly shifted above 0V line while in case of Bridge wave rectifier waveform touches to 0V line and then rise. Also the peak voltage is slightly more in case of center tapped compared to bridge wave rectifier.

Part B : Hardware Implementation

Hardware Picture :

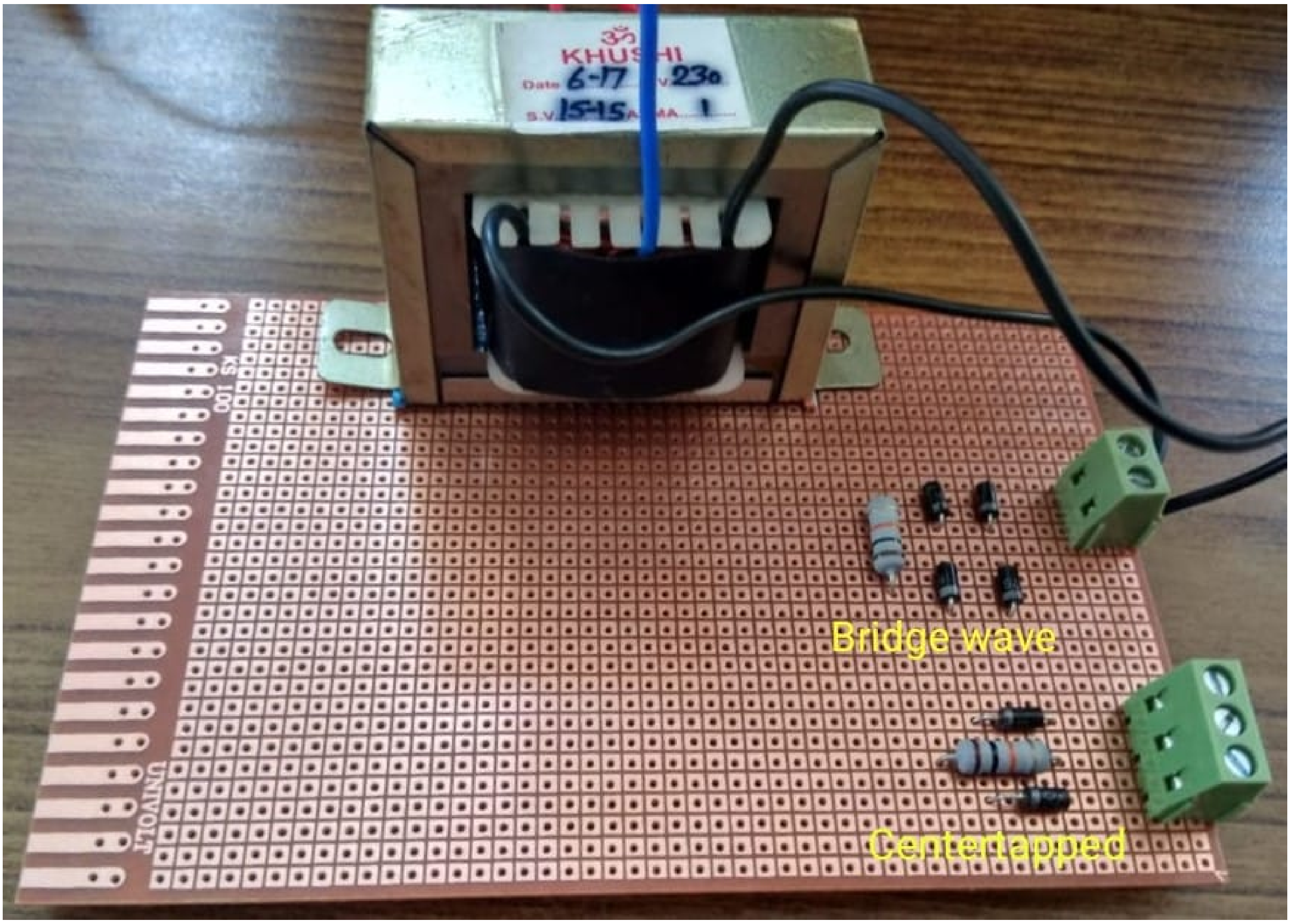


Fig. 2(g) Hardware Picture of Bridge Wave(upper) and Center tapped(lower) full wave rectifier

Result : By performing this experiment, we could get the full wave rectified output at the load which is UNCONTROLLED because diode are used in designing the rectifier hence having output with whole waveform.

EXPERIMENT 3: HALF WAVE SEMICONTROLLED RECTIFIER

Objective : To design and simulate a Half wave semicontrolled rectifier circuit using thyristors.

Required Hardware Components : Step Down (230:30) Transformer

230 V,50 Hz Single phase AC Supply

One TYN612 Thyristor (SCR)

Load Resistor (1 kohm)

Connecting Wires

Soldering Iron and wire

Multimeter

Dotted PCB or Breadboard

Software Tool : MATLAB Simulink

Circuit Diagram :

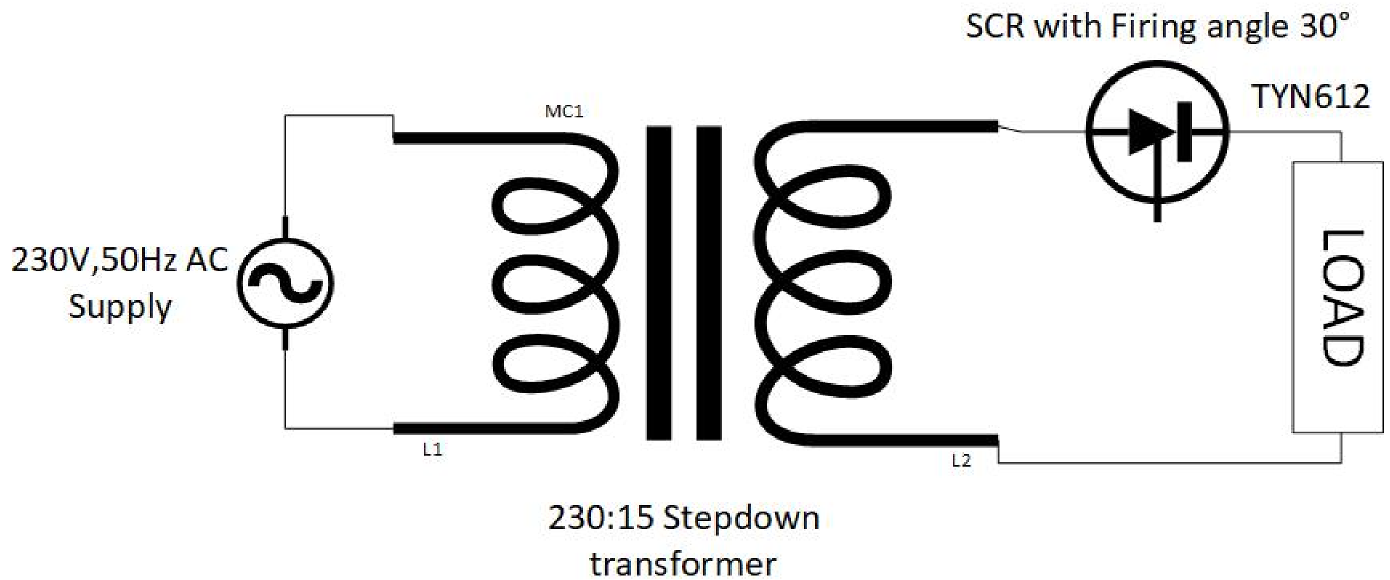


Fig. 3(a) Circuit Diagram of Half Wave Rectifier using Diodes

Part A : Simulation of circuit

Circuit diagram shown in Fig. 3(a) is designed in MATLAB Simulink below,

Note that Snubber circuit of diode is removed and SCR is having firing angle 30°.

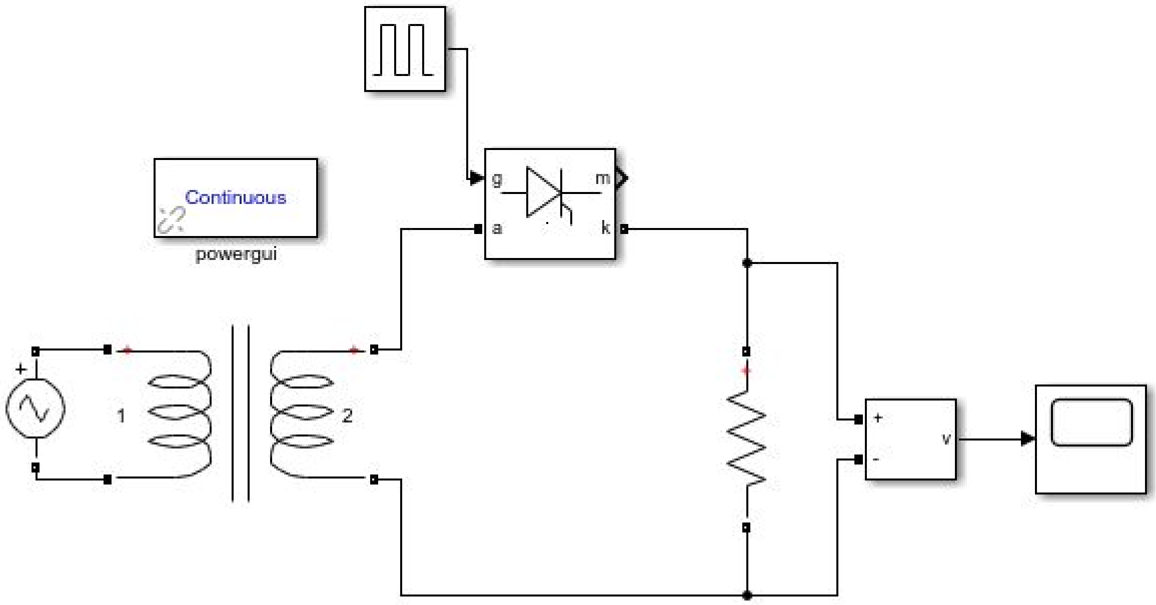


Fig. 3(b) Simulink Model of Half wave rectifier with R load

Output Waveform :

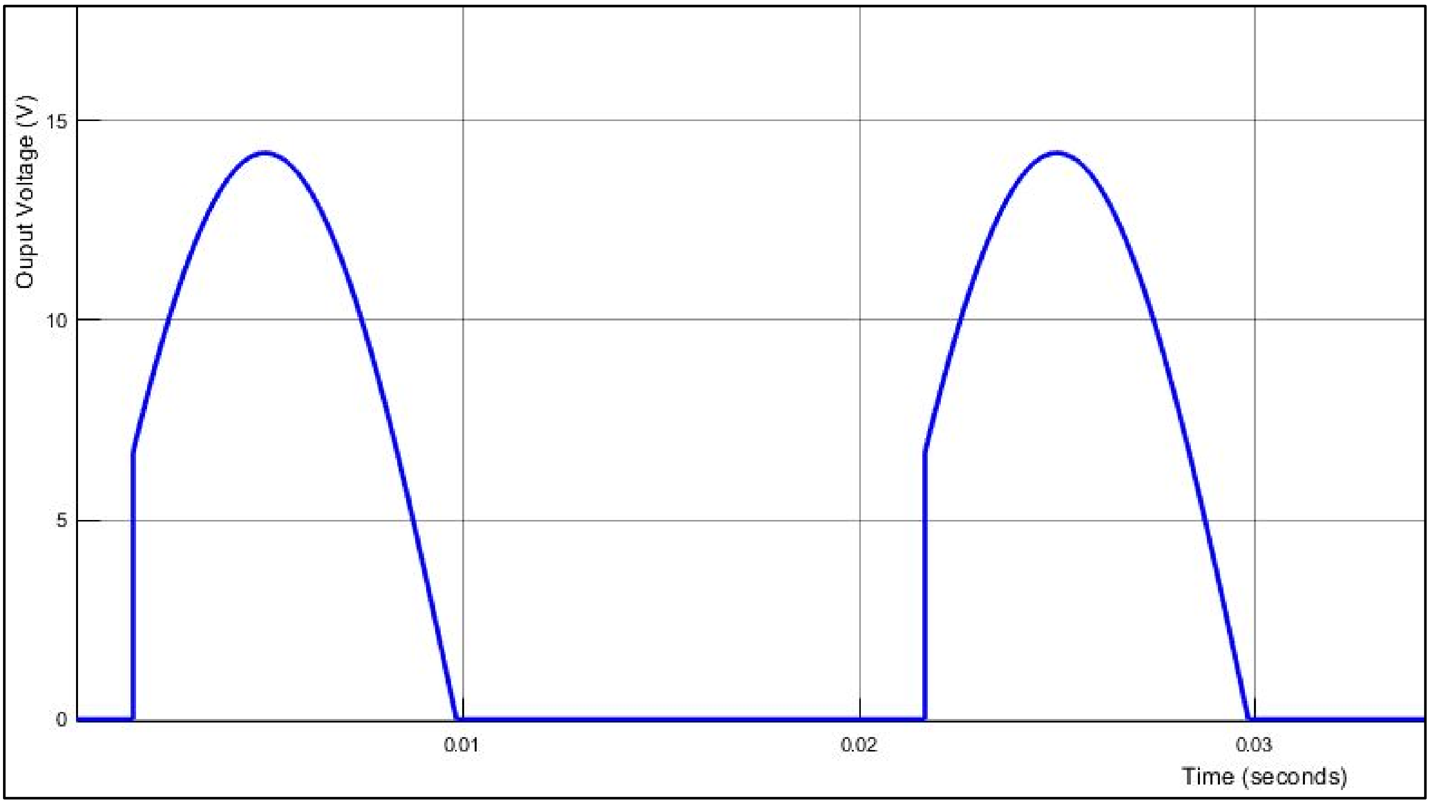


Fig. 3(c) Simulink Output Waveform with Resistive load

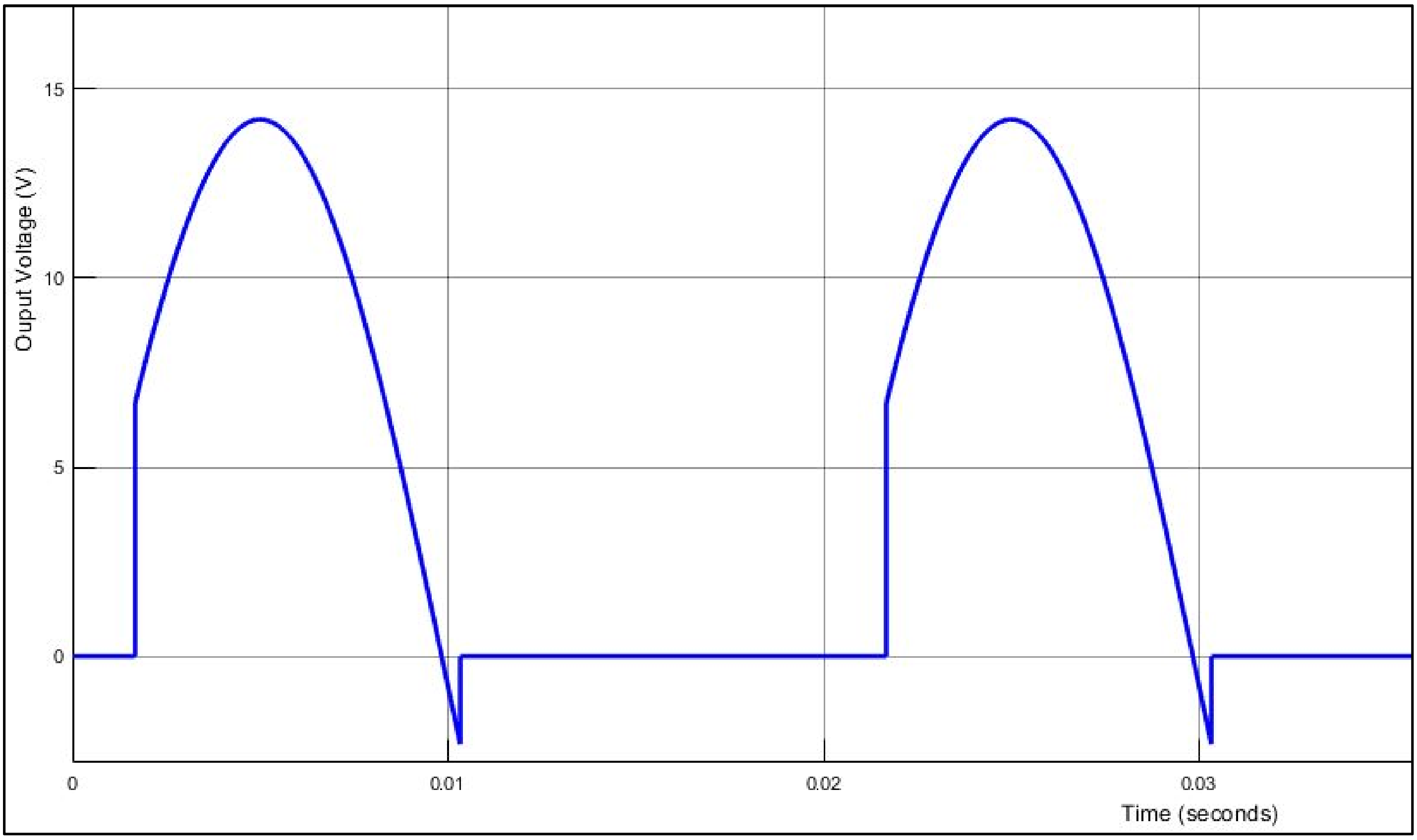


Fig. 3(d) Simulink Output Waveform with R-L load

Note that here L=0.5H is taken.

Observation : We can observe that output voltage waveform doesn't start at t=0sec but it is triggered at 30°, so it will start at 0.02*30/360=0.00166sec.

Here by comparing both waveforms of Fig. 1(c) & Fig. 1(d) we observe that in R-load there is no output present in negative half cycle while in case of R-L load there is some negative voltage appearing at the starting of negative half cycle due to the fact that inductor opposes the change in current.

Result : Output voltage is gained only after firing angle at which it is triggered (here 30°). We can change this firing angle by designing gate Triggering circuit.(In coming experiment)

Also we noted that, For R-L load, In positive half cycle, inductor is charged with appropriate polarity. Now when the negative cycle starts, the net effective voltage across diode is such that it will be in forward bias until inductor is discharged. Therefore at starting of negative cycle we observe some negative voltage portion. By changing value of L , we can change this portion. In R-Load, there will not be any negative voltage.

EXPERIMENT 4 : FULL WAVE SEMICONTROLLED CENTER TAPPED & BRIDGE RECTIFIER

Objective : To design and simulate full wave semicontrolled both center tapped and bridge rectifier circuit.

Required Hardware Components : Step Down (230:30) Transformer

230 V,50 Hz Single phase AC Supply

Six TYN612 Thyristors (SCRs)

Two load Resistors 10 kohms

Connecting Wires

Soldering Iron and wire

Multimeter

Dotted PCB or Breadboard

Required Software Tool : MATLAB Simulink

Circuit Diagram :

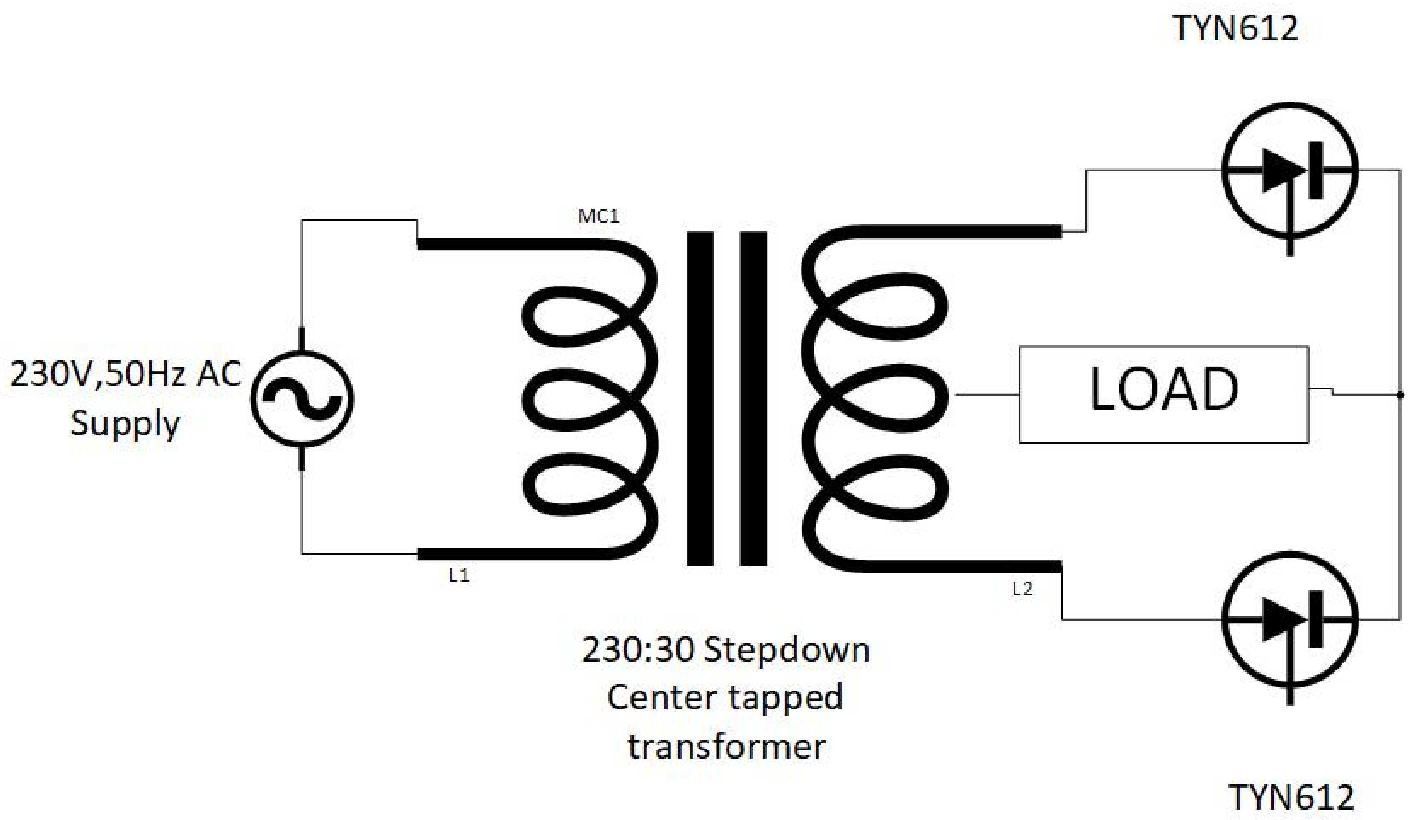


Fig. 4(a) : Circuit Diagram of Center tapped Full Wave Rectifier using SCRs

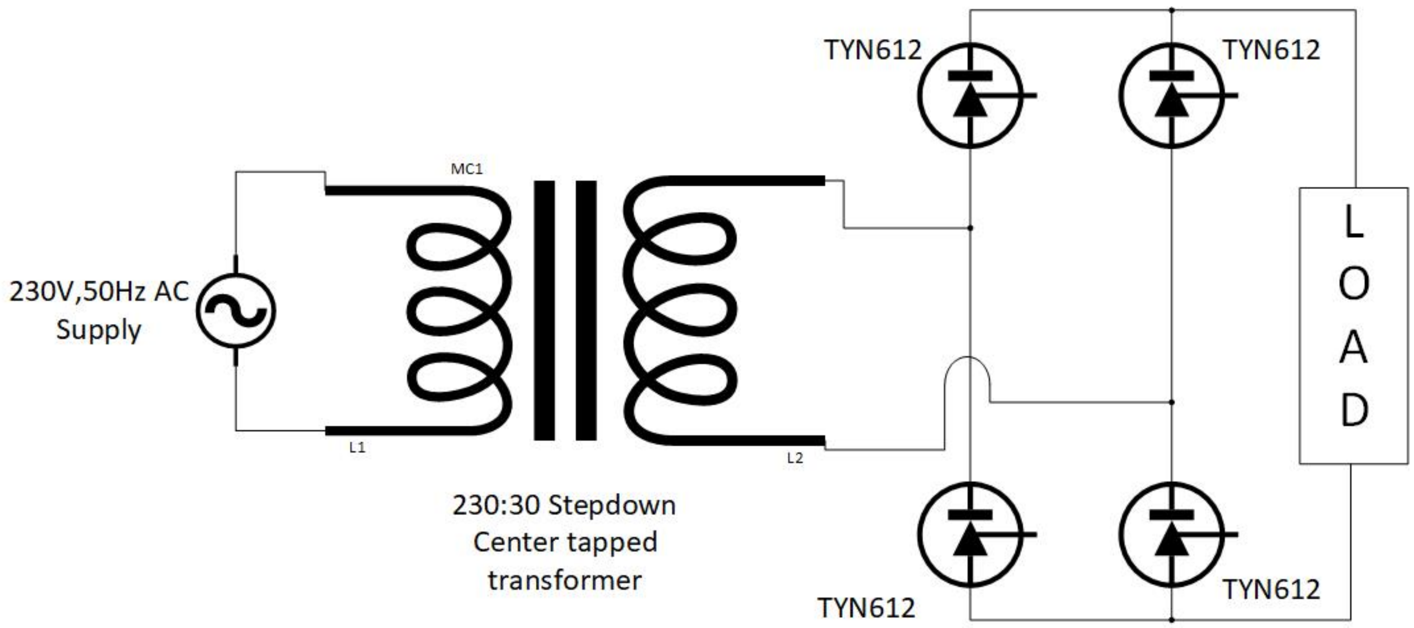


Fig. 4(b) : Circuit Diagram of Bridge Full Wave Rectifier using SCRs

Part A : Simulation of circuit

Circuit diagram shown in Fig. 4(a) & Fig. 4(b) is designed in MATLAB Simulink below,

Bridge Wave Rectifier Simulation

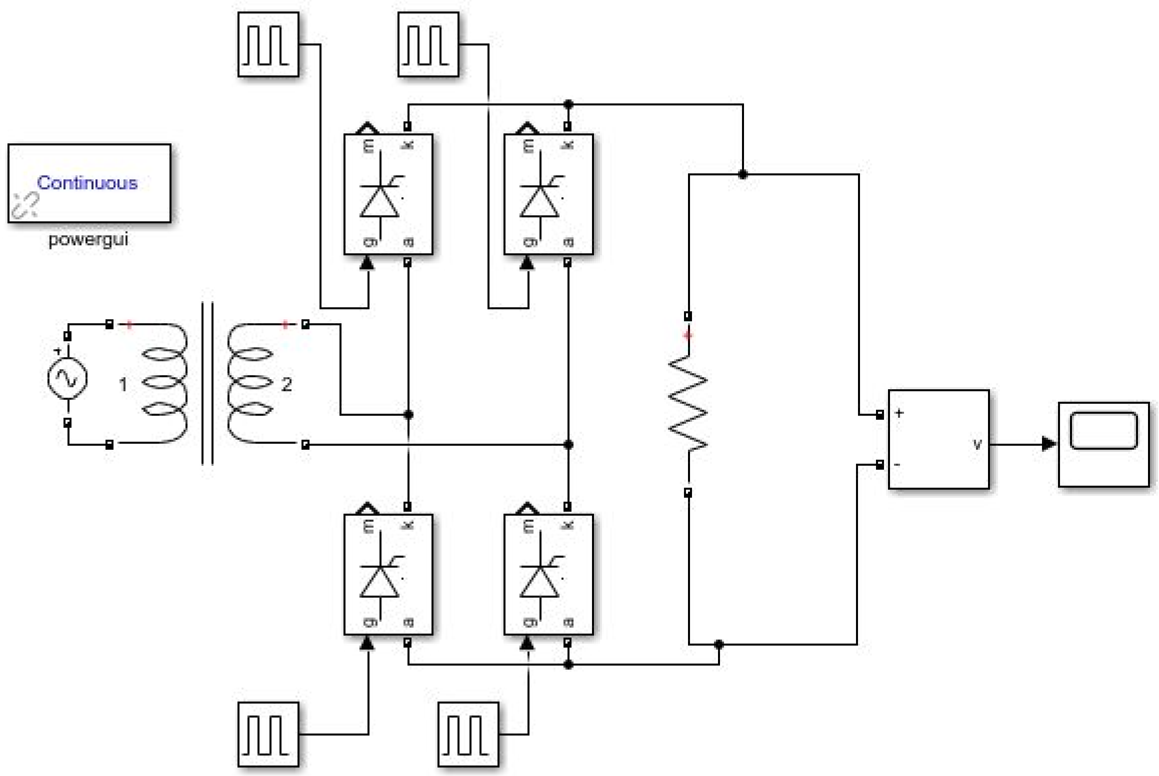


Fig. 4(c) Simulink Model of Bridge Full Wave Rectifier

Output Waveform :

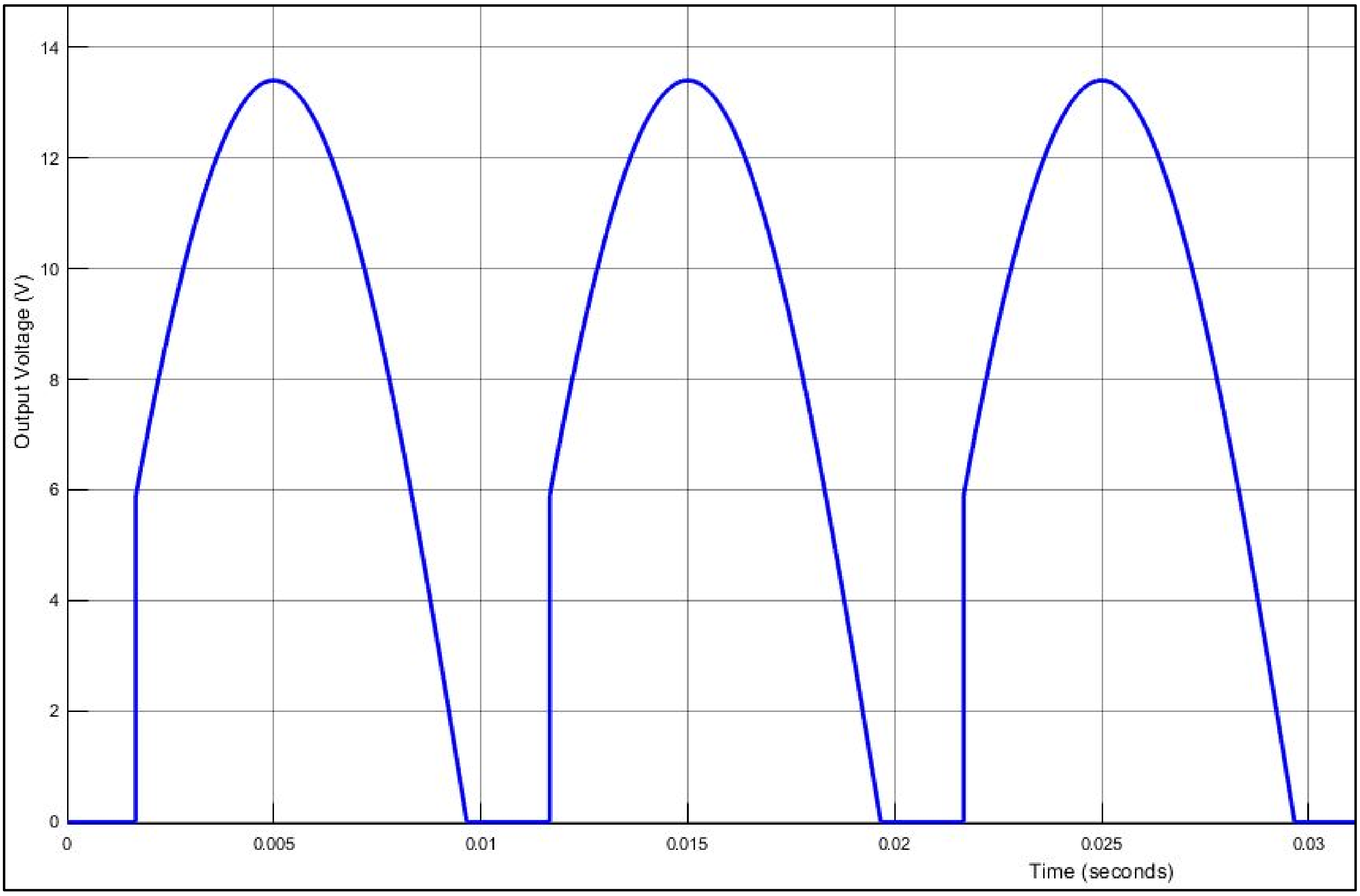


Fig. 4(d) Simulink Output Waveform

Center Tapped Full Wave Rectifier

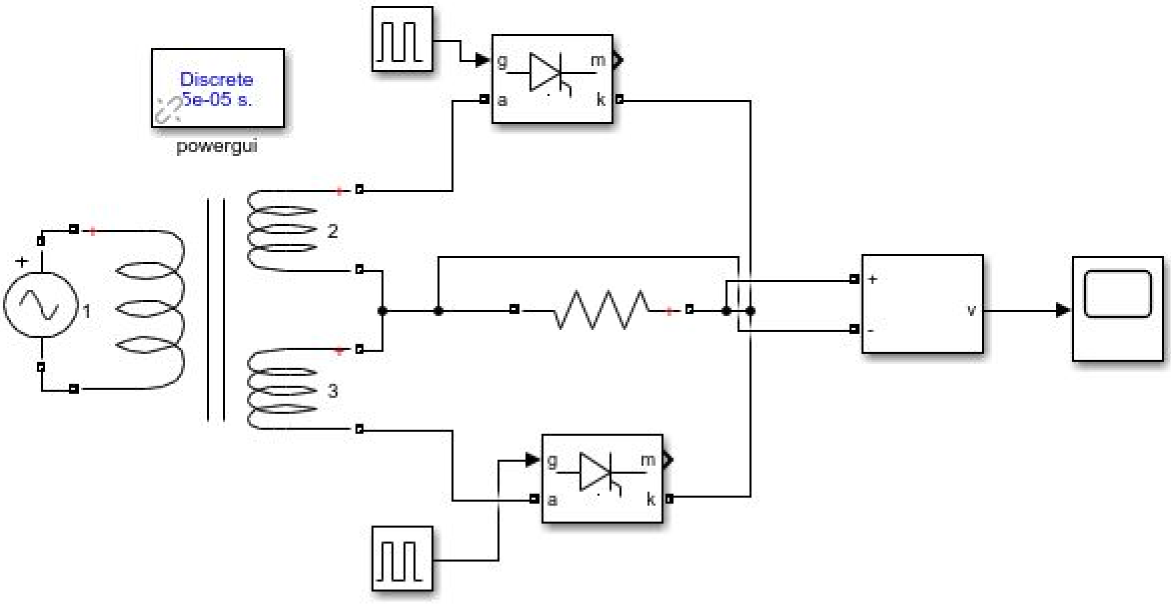


Fig. 4(e) Simulink Model of Center Tapped Full Wave Rectifier

Output Waveform :

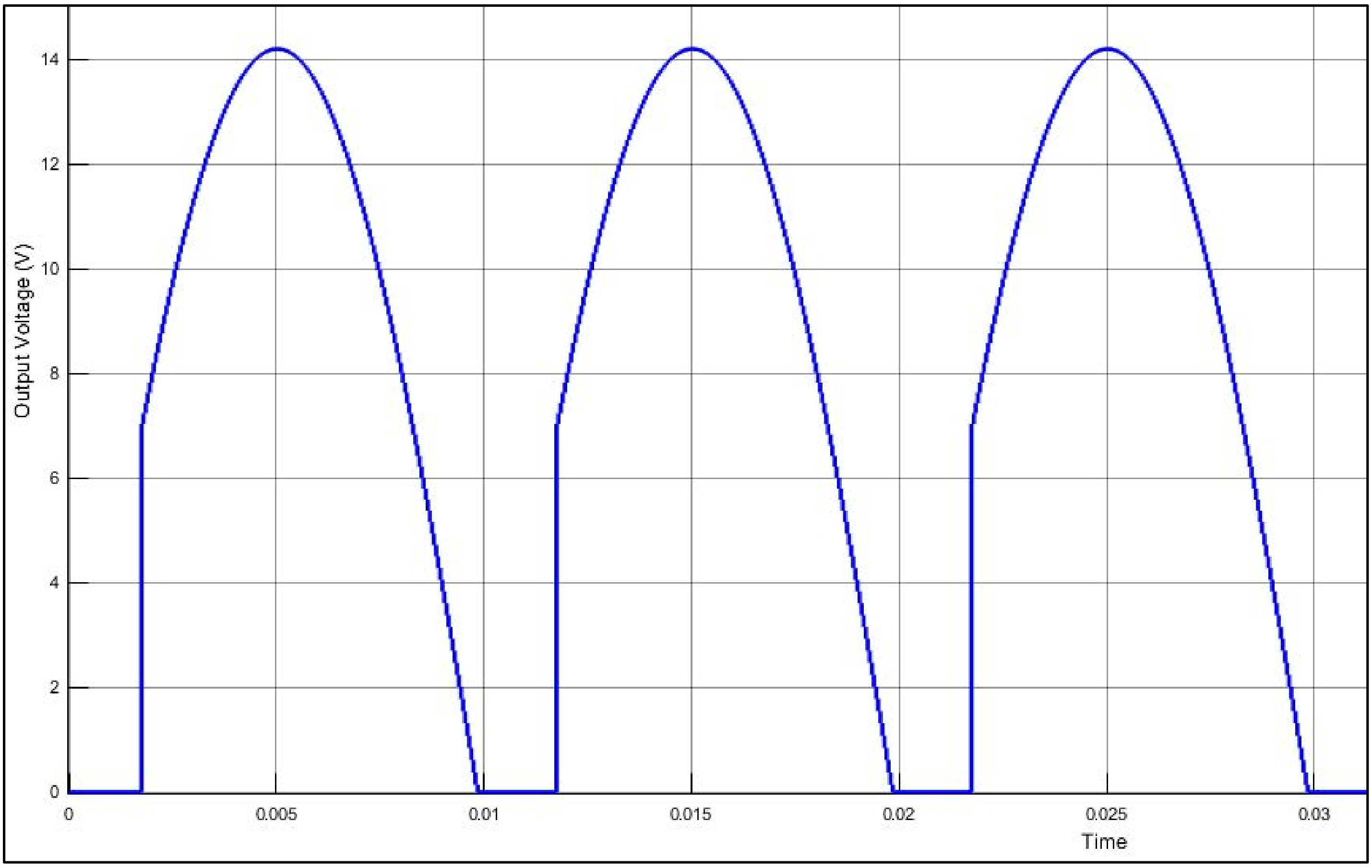


Fig. 4(f) Output Waveform of Center tapped Full wave Rectifier

Observation : We can observe that output voltage waveform doesn't start at t=0sec but it is triggered at 30°, so it will start at 0.02*30/360=0.00166sec. Similarly other half wave will also start according to fire angle set. Here other set of SCR is triggered at 210°.

We can observe that in Center tapped rectifier, output waveform is having the peak voltage slightly more compared to bridge wave rectifier.

Peak Amplitude in Bridge rectifier	13.3V
Peak Amplitude in Center tapped rectifier	14.1V

Result : By performing this experiment, we could get the full wave rectified output at the load which is SEMICONTROLLED because SCRs are used in designing the rectifier hence having output with whole waveform.

EXPERIMENT 5 : THREE PHASE HALF WAVE UNCONTROLLED RECTIFIER

Objective : To study the characteristics of three phase half wave rectifier using diodes in MATLAB.

Theory: For higher power application and where three-phase power supply is available, a three phase bridge rectifier, as shown in figure (1), should be used. One diode is conduct at any instant. It is the diode connected to the phase having the highest instantaneous voltage. The output voltage of the successive phase voltages and varying from Vm/2 to Vm, three times per input cycle.

The average output voltage is:

$$V_{dc} = 1/(2/\pi/3) = \int_{\frac{\pi}{6}}^{\frac{5\pi}{6}} vd(wt) = 3/2\,\pi \int_{\pi/6}^{5\pi/6} V_{s\,m} \sin wt(dt)$$

$$= (3V_{s\,m}/2\pi) \times 2 \times 0.866$$

$$\text{i.e. } V_{dc} = 0.827\, V_{s\,m} \text{ or } 0.827 \times \sqrt{2}V_{s\,rms} \text{ i.e. } 1.17\, V_{s\,rms}$$

$$I_{dc} = V_{dc}/R_{LOAD} = 0.827\, V_{s\,m}/R_{LOAD}$$

Similarly, the rms value of the output voltage can be found as:

$$V_{O(RMS)} = \left[\frac{3}{2\pi}\int_{\frac{\pi}{6}+\alpha}^{\frac{5\pi}{6}+\alpha} V_m^2 \sin^2 \omega t.d(\omega t)\right]^{\frac{1}{2}}$$

and we obtain

$$V_{O(RMS)} = \sqrt{3}V_m\left[\frac{1}{6}+\frac{\sqrt{3}}{8\pi}\cos 2\alpha\right]^{\frac{1}{2}}$$

Circuit Diagram:

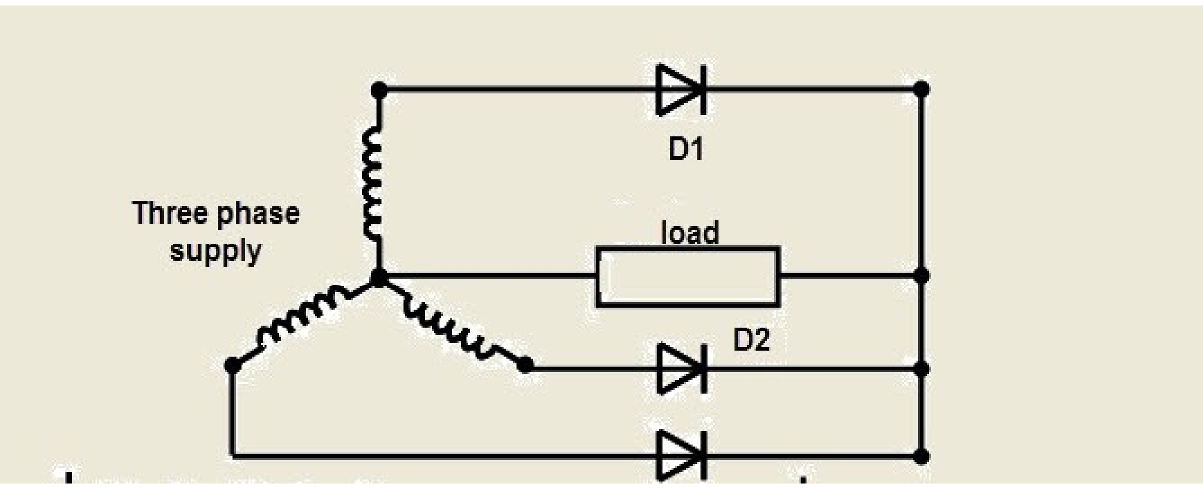


Fig. 5(a) Circuit diagram of three phase half wave rectifier uncontrolled

Part A : Simulation of circuit

Circuit diagram shown in Fig. 5(a) is designed in MATLAB Simulink below,

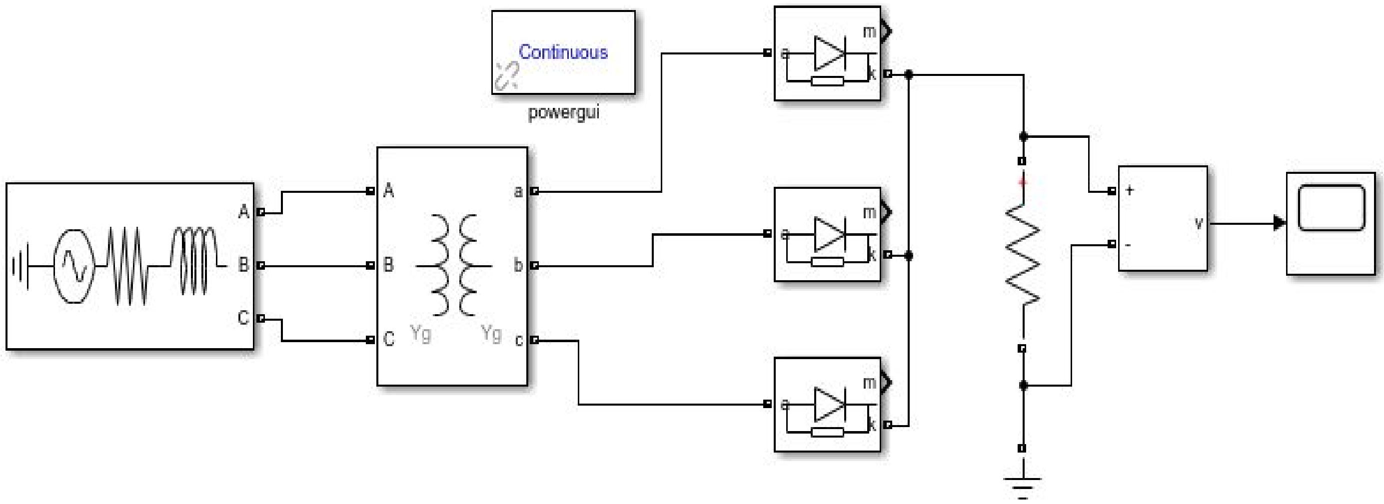


Fig. 5(b) Simulink Model of three phase half wave rectifier uncontrolled

Observation :

We could obtain high resultant magnitude of output voltage because there are three phase present having 120º phase displacement to each other.

Therefore Output Waveform is having some height from the 0V line.

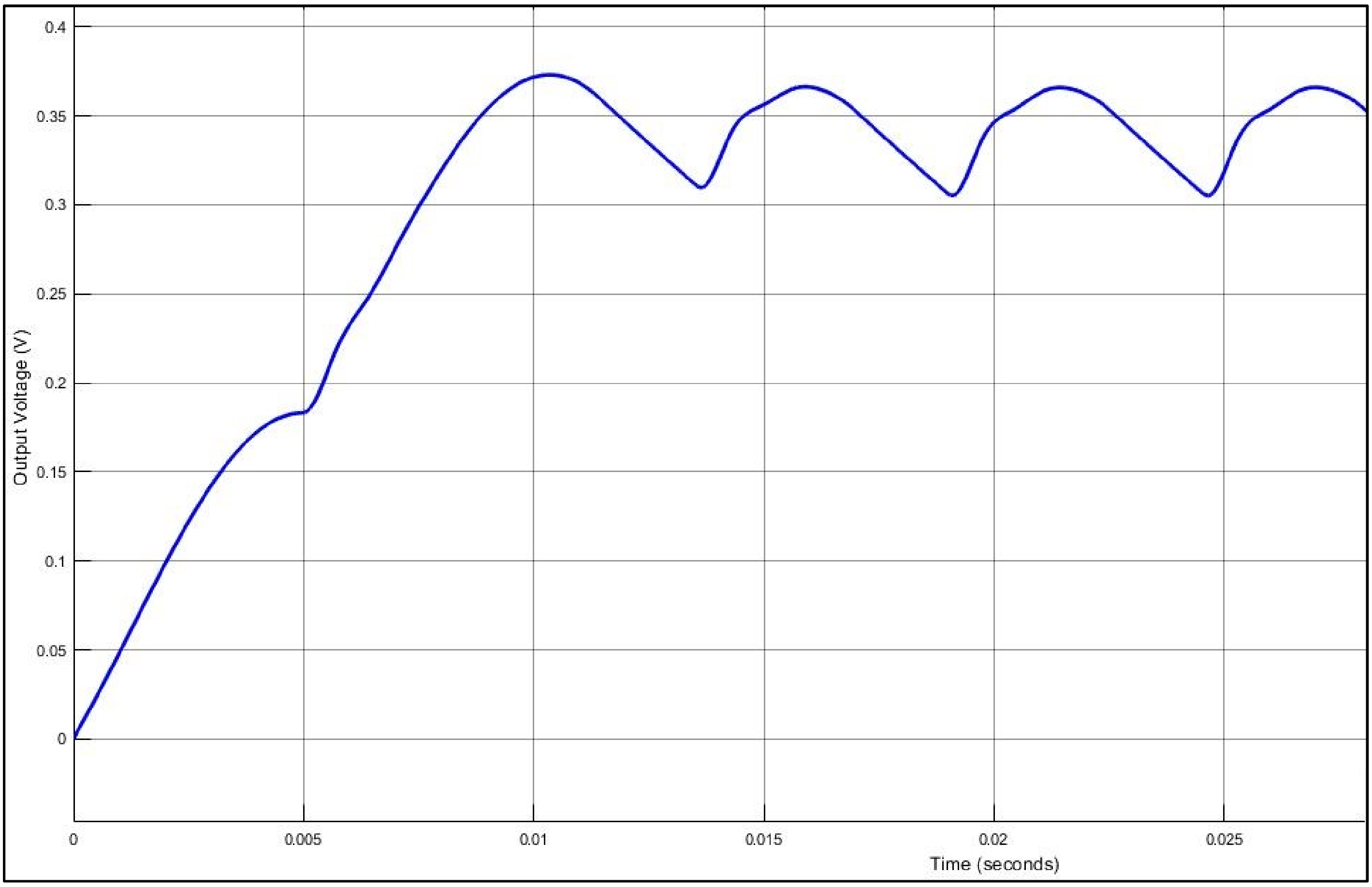


Fig. 5(c) Output Waveform of three phase half wave rectifier uncontrolled

Result: Output Waveform of three phase rectifier circuit is having some height from 0 V line.

EXPERIMENT 6 : THREE PHASE FULL WAVE SEMICONTROLLED RECTIFIER

Objective : To study the characteristics of three phase full wave rectifier using SCRs in MATLAB.

Circuit Diagram:

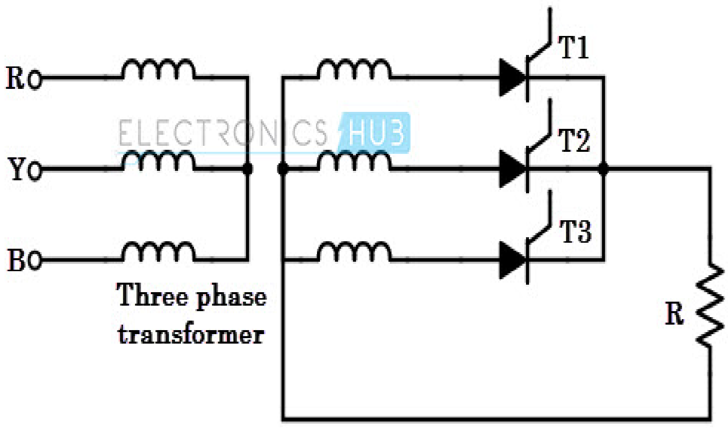


Fig. 6(a) Circuit diagram of three phase full wave rectifier semicontrolled

Part A : Simulation of circuit

Circuit diagram shown in Fig. 6(a) is designed in MATLAB Simulink below,

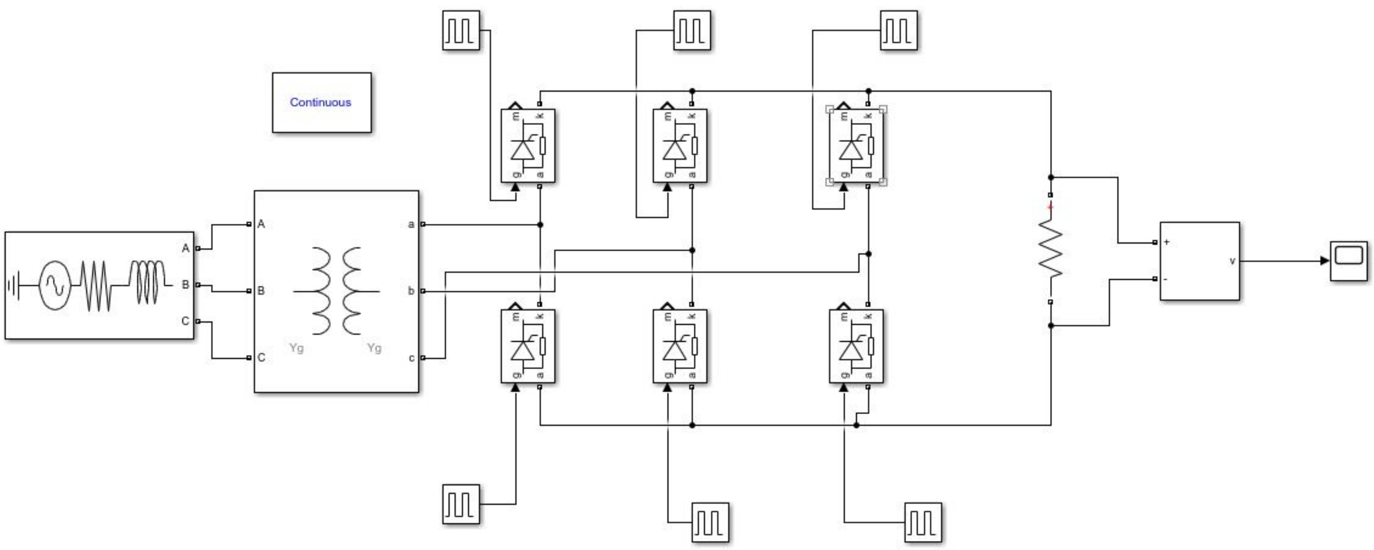


Fig. 6(b) Simulink Model of three phase full wave rectifier semicontrolled

Observation :

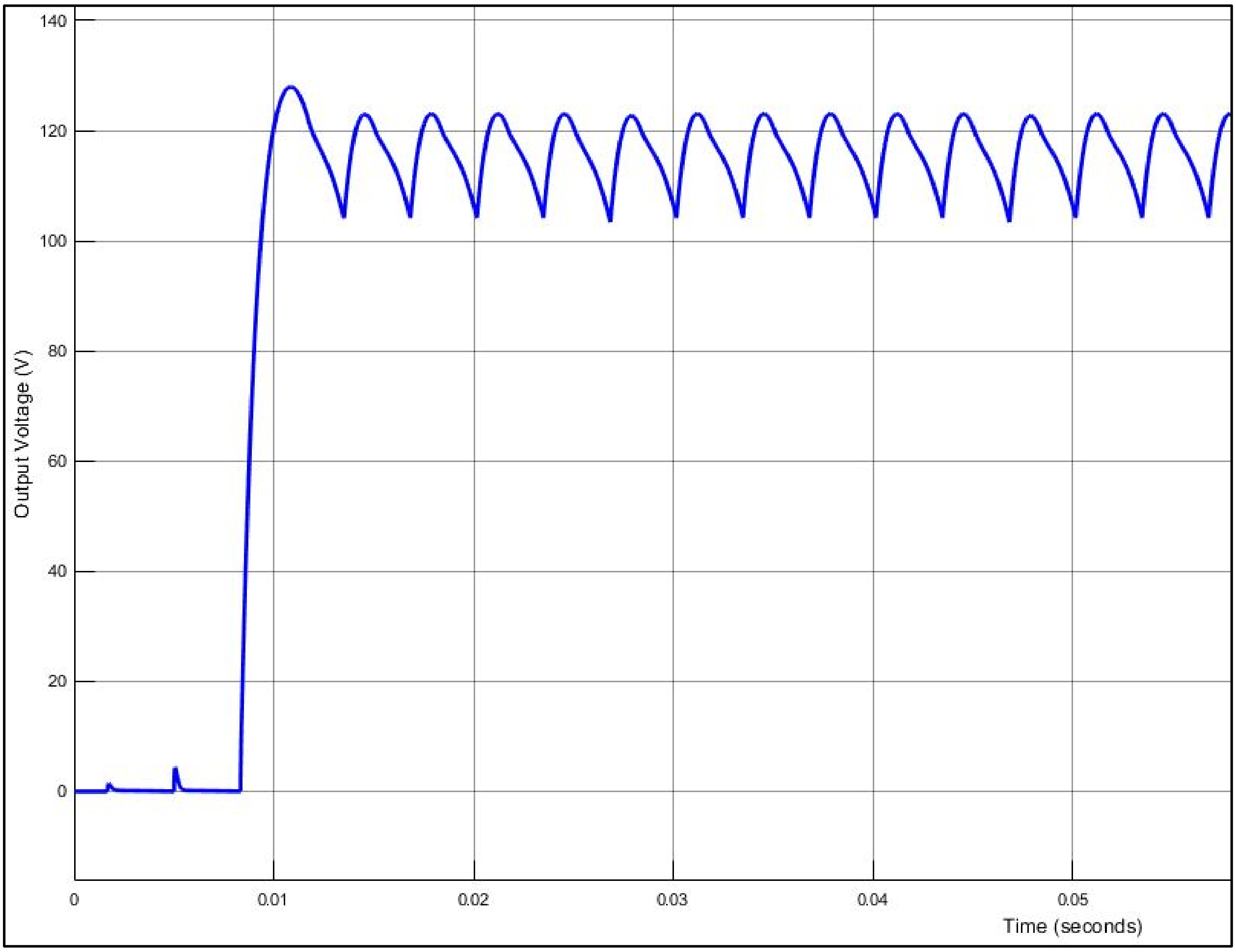


Fig. 6(c) Output Waveform of three phase Full wave rectifier semicontrolled

Result: Output Waveform of three phase rectifier circuit is having some height from 0 V line.

EXPERIMENT 7: DC POWER SUPPLY FOR GATE DRIVER CIRCUIT

Objective : To design and simulate a Regulated 15V DC Power Supply for Gate driver circuit of MOSFET or IGBT.

Required Hardware Components : Step Down (230:30) Transformer

230 V,50 Hz Single phase AC Supply

Four IN4007 Diodes

Four Capacitors (2 nos. 2200µF,25V & 2 nos. 1000µF)

Positive & Negative Voltage Regulator ICs (7815 & 7915 respectively)

Connecting Wires

Soldering Iron and wire

Multimeter

Dotted PCB or Breadboard

Software Tool : MATLAB Simulink

Circuit Diagram :

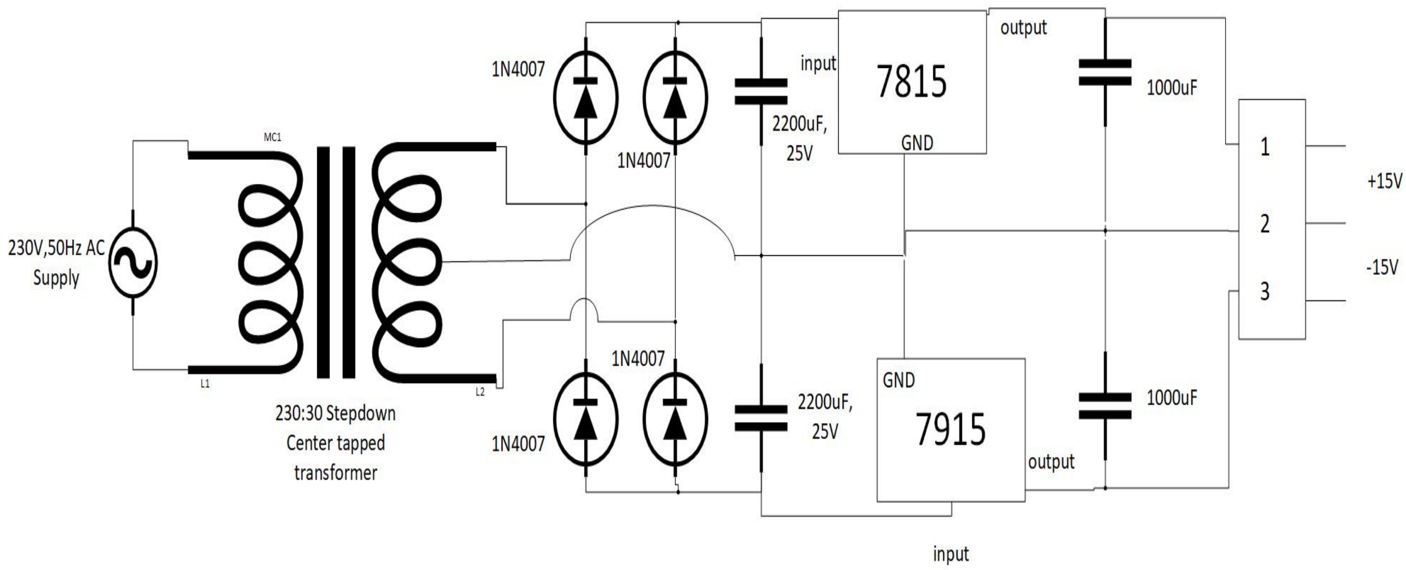


Fig. 7(a) Circuit Diagram of Regulated DC Power Supply

Part A : Simulation of circuit

Circuit diagram shown in Fig. 7(a) is designed in MATLAB Simulink below,

Note that Here 7815 Regulator IC is *not* used in Simulink. But we can verify output waveform for by simulation

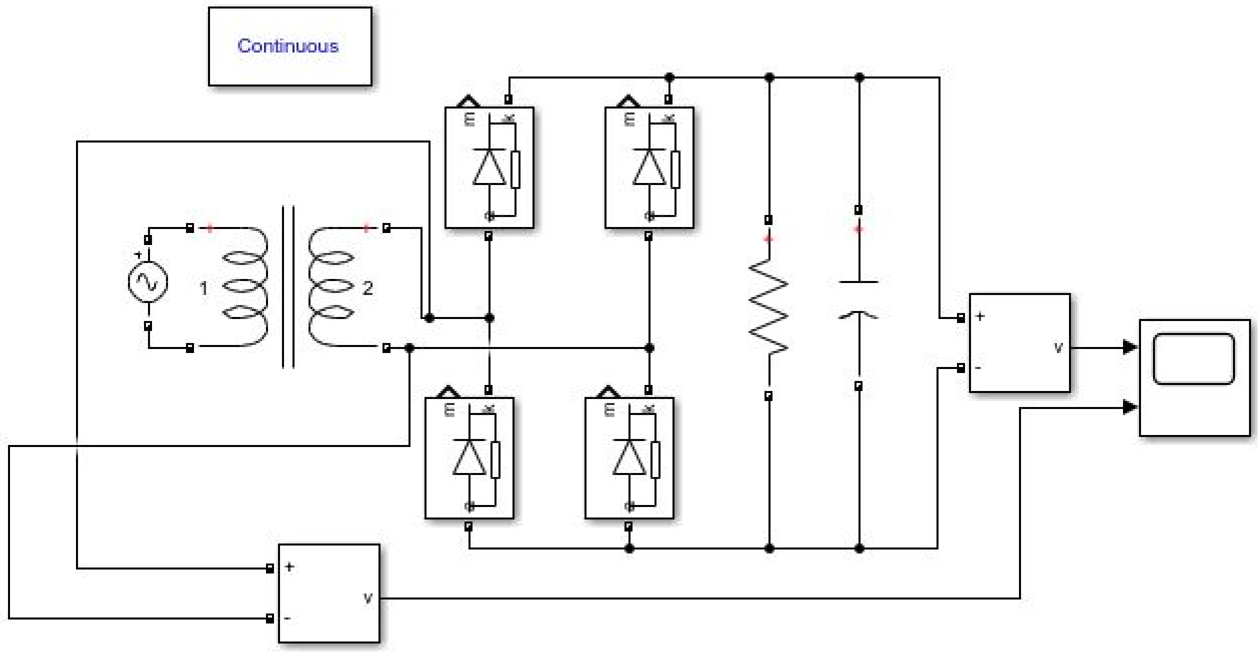


Fig. 7(b) Simulink Model of DC Power Supply without Regulator ICs

Output Waveform :

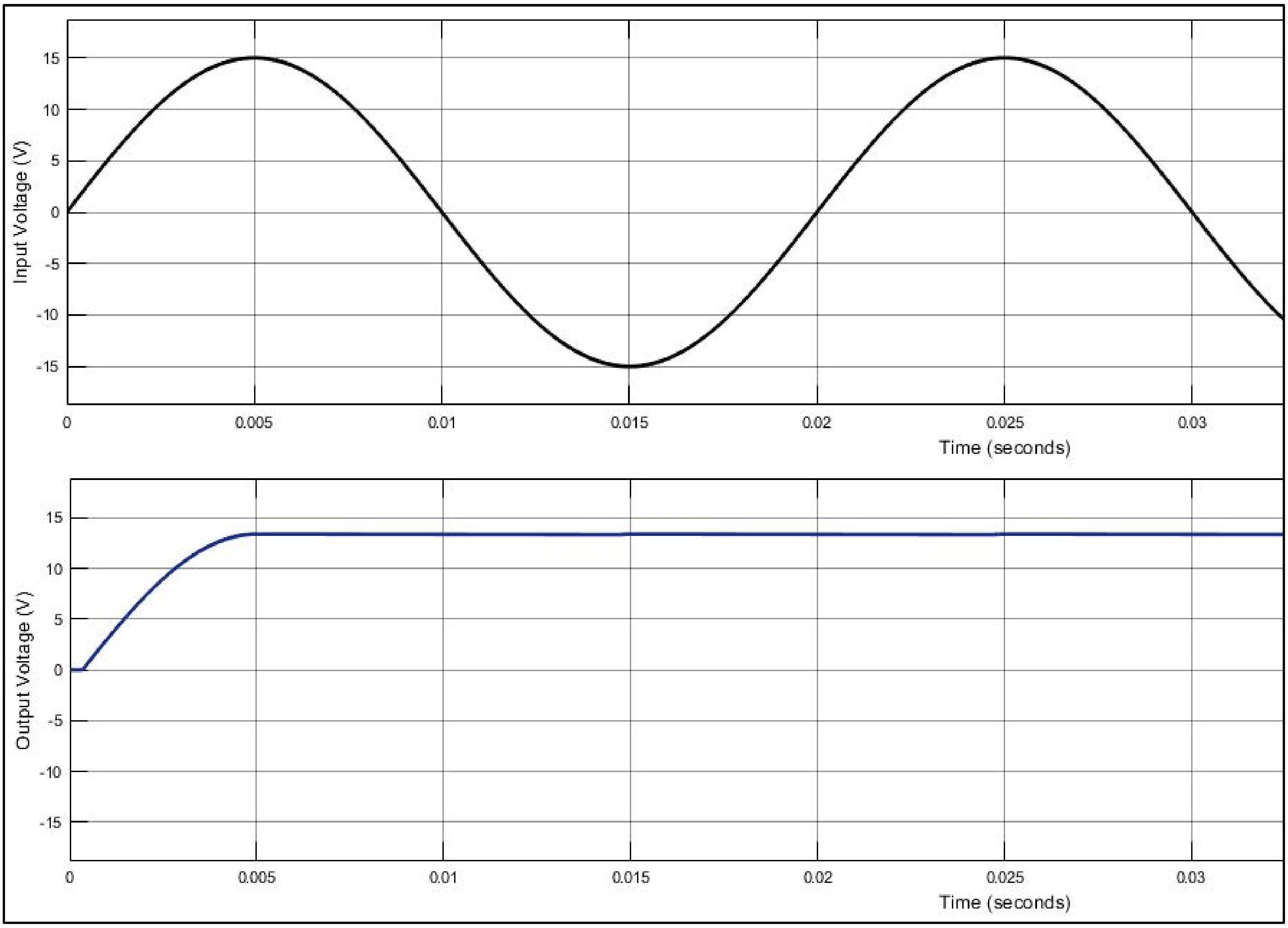


Fig. 7(c) Simulink Output Waveform along with Input Waveform

Observation : Here by simulation we got the DC output of 13.4 V with some ripple. We can regulate it to 15V using regulator ICs and remove ripple using filters.

Part B : Hardware Implementation

Hardware Picture :

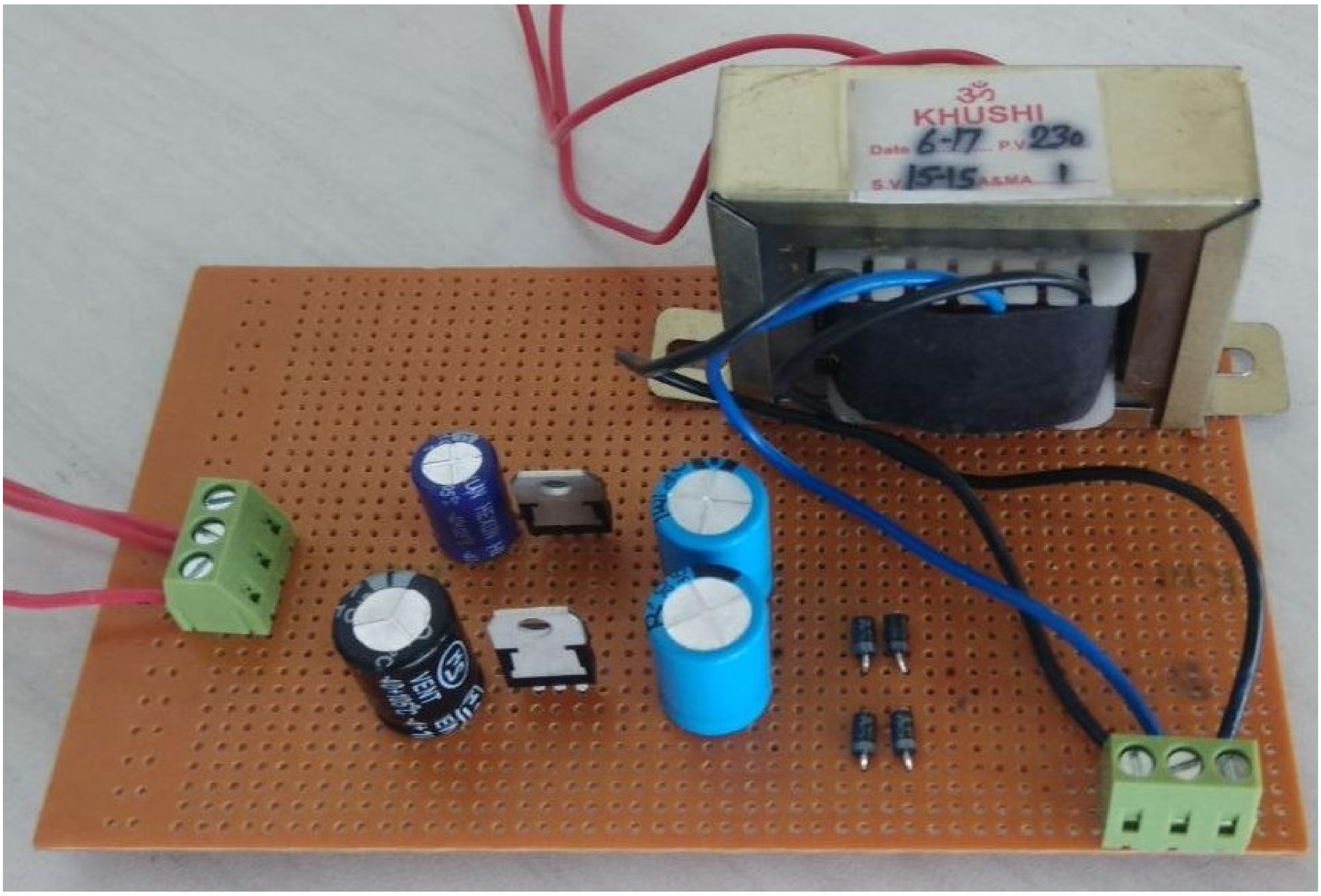


Fig. 7(d) Hardware Picture of Regulated DC Power Supply

Observation : Following observation were taken after implementing circuit as shown in fig.1(b).

Voltage between terminal 1 and 2	+14.8 V
Voltage between terminal 2 and 3	-14.8 V

Result : AC input is rectified by bridge rectifier and then it filtered by capacitive filters. It is further given as input to the Regulator ICs which regulates the voltage and finally the capacitors are connected to remove ripples.

By performing this experiment thus we could design a regulated DC power supply for gate driver circuit of MOSFET or IGBT.

EXPERIMENT 8 : R TRIGGERING METHOD FOR A THYRISTOR

Objective : To design and simulate Resistive triggering circuit for Thyristor(SCR) for triggering.

Required Hardware Components : Step Down (230:18) Transformer

230 V,50 Hz Single phase AC Supply

One IN4007 diode

One TYN612 Thyristor(SCR)

Three Resistors (100 ohm, 9 kohm, Load Resistor 1 kohm, 5W)

Potentiometer(100 kohm)

Connecting Wires

Soldering Iron and wire

Multimeter

Dotted PCB or Breadboard

Required Software Tool : MATLAB Simulink

Circuit Diagram :

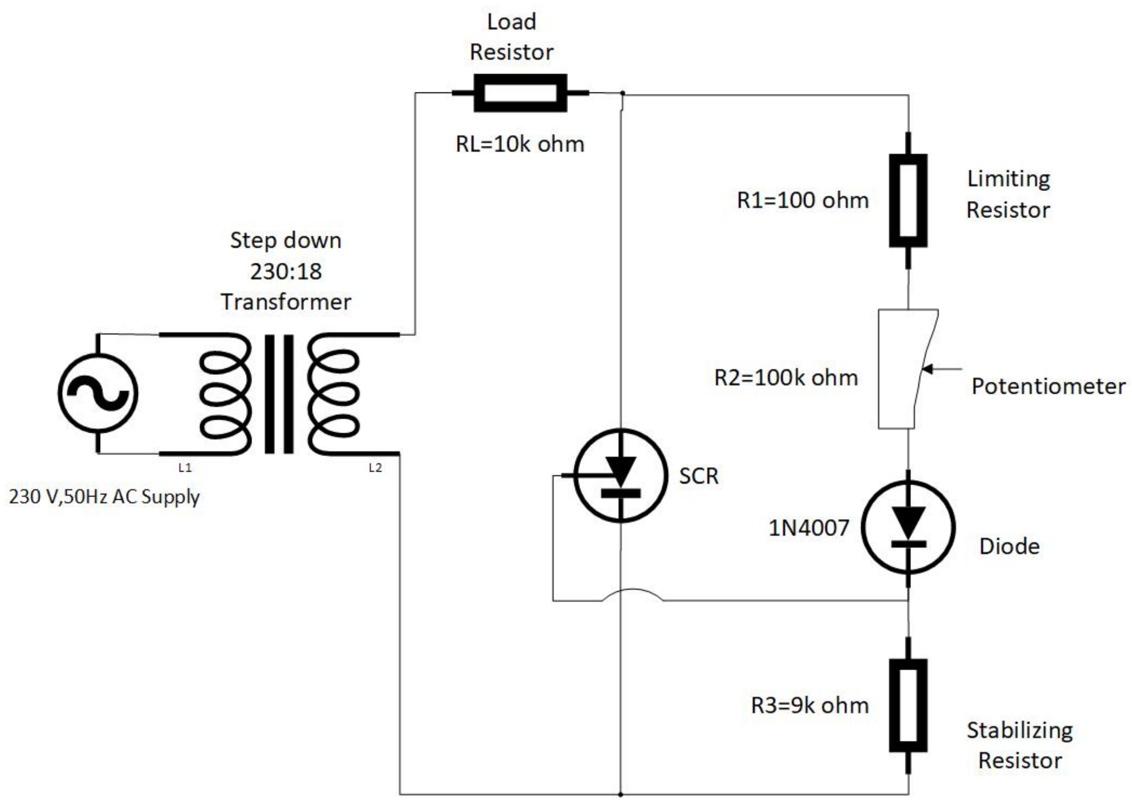


Fig. 8(a) Circuit Diagram of R-triggering of SCR

Part A : Simulation of circuit

Circuit diagram shown in Fig. 8(a) is designed in MATLAB Simulink below,

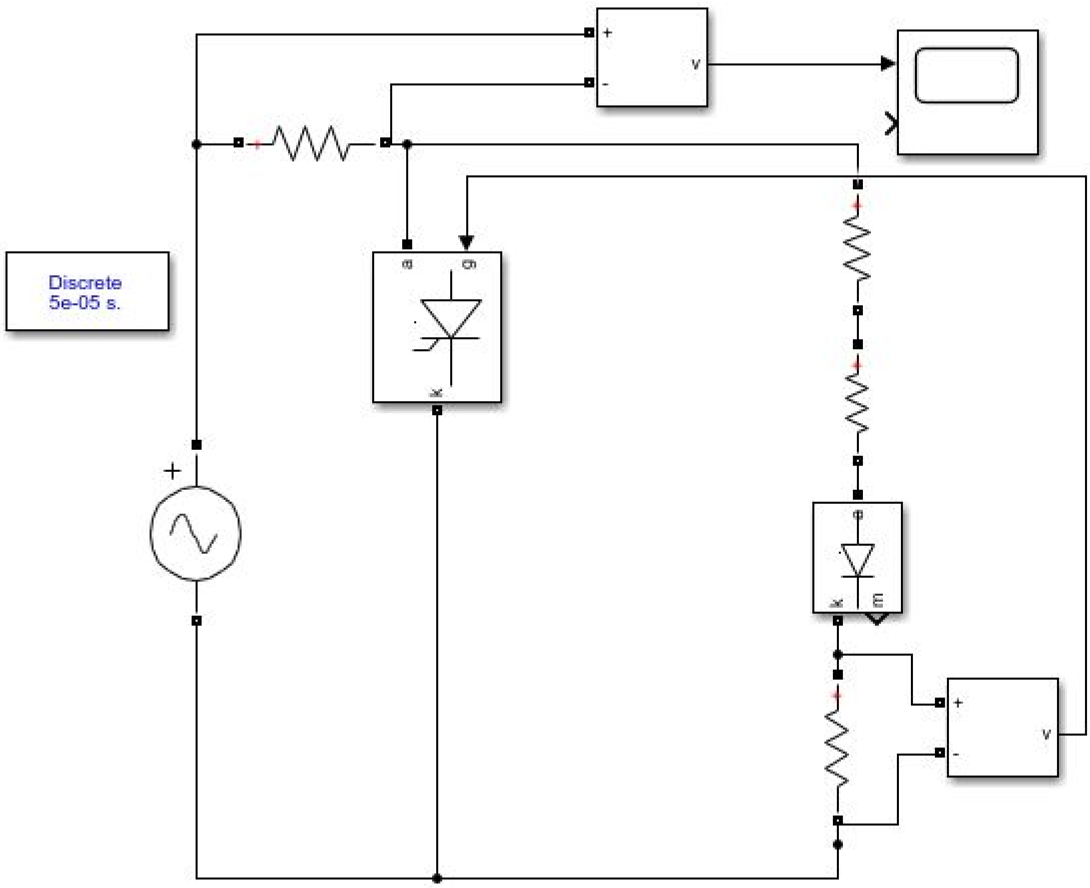


Fig. 8(b) : Simulink Model of R-triggering for SCR

NOTE : AC supply voltage is having peak amplitude of $Vm = 9\sqrt{2}\,V$ in simulink.

Output Waveform :

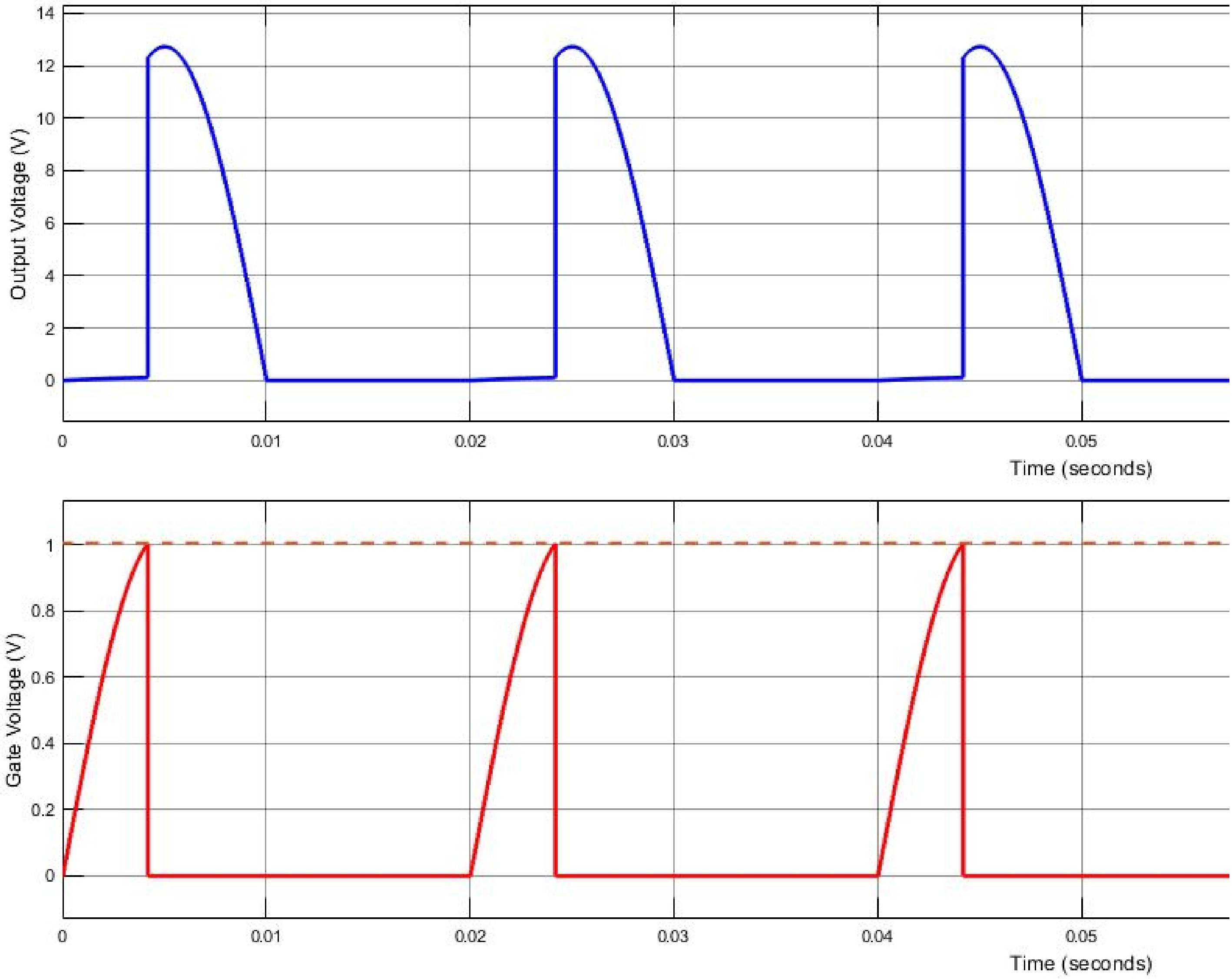


Fig. 8(c) : Output Voltage Waveform across load(Upper) & Voltage Given to Gate of SCR(Lower)

Observation : By inspecting the second waveform(Red) we can say that when Voltage across R3 (i.e Voltage given to Gate of SCR) reaches to Vgt Gate Triggering Voltage of SCR, Thyristor is turned ON or triggered. It is seen from waveform that $Vgt = 1V$ for SCR used in Simulink. By changing R2 (Pot), we can change voltage across R3 and so we can change the firing angle α .

Also we can calculate the value of R2 when the SCR is triggered at 90°, which is $\boldsymbol{R2 = 104450\ ohms.}$

Hardware Picture :

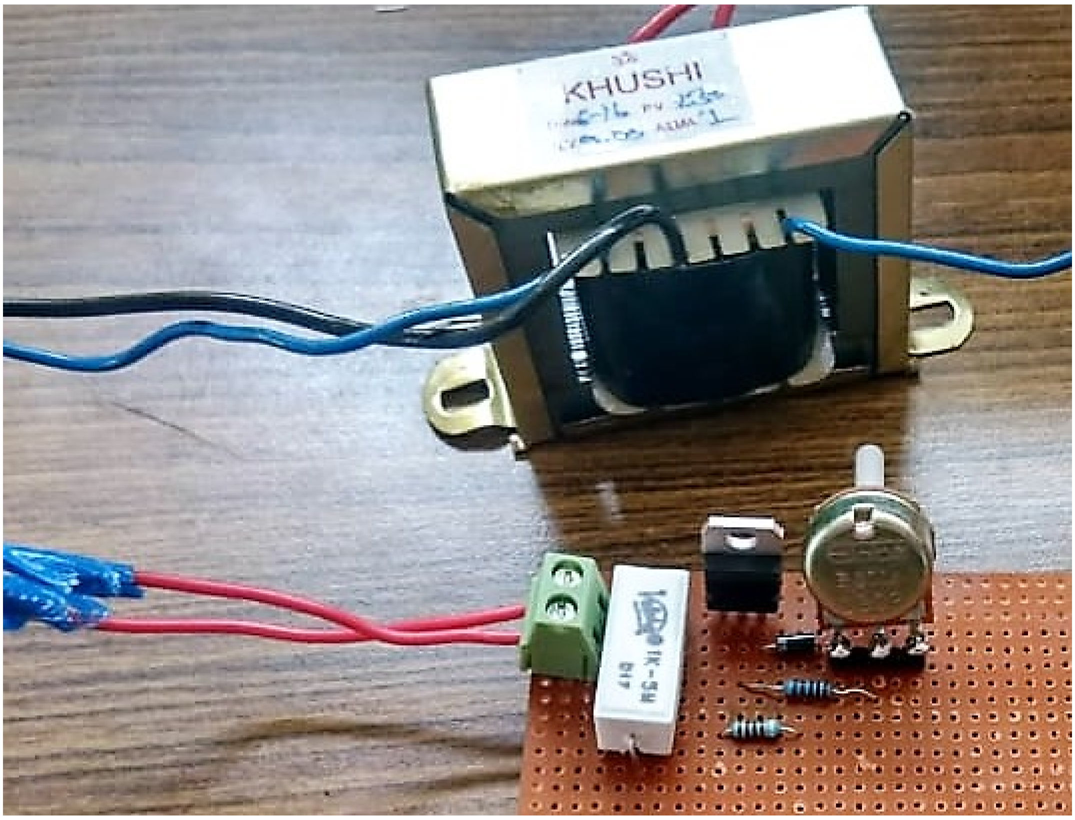


Fig. 8(d) Hardware Picture of Regulated DC Power Supply

DSO Output waveform :

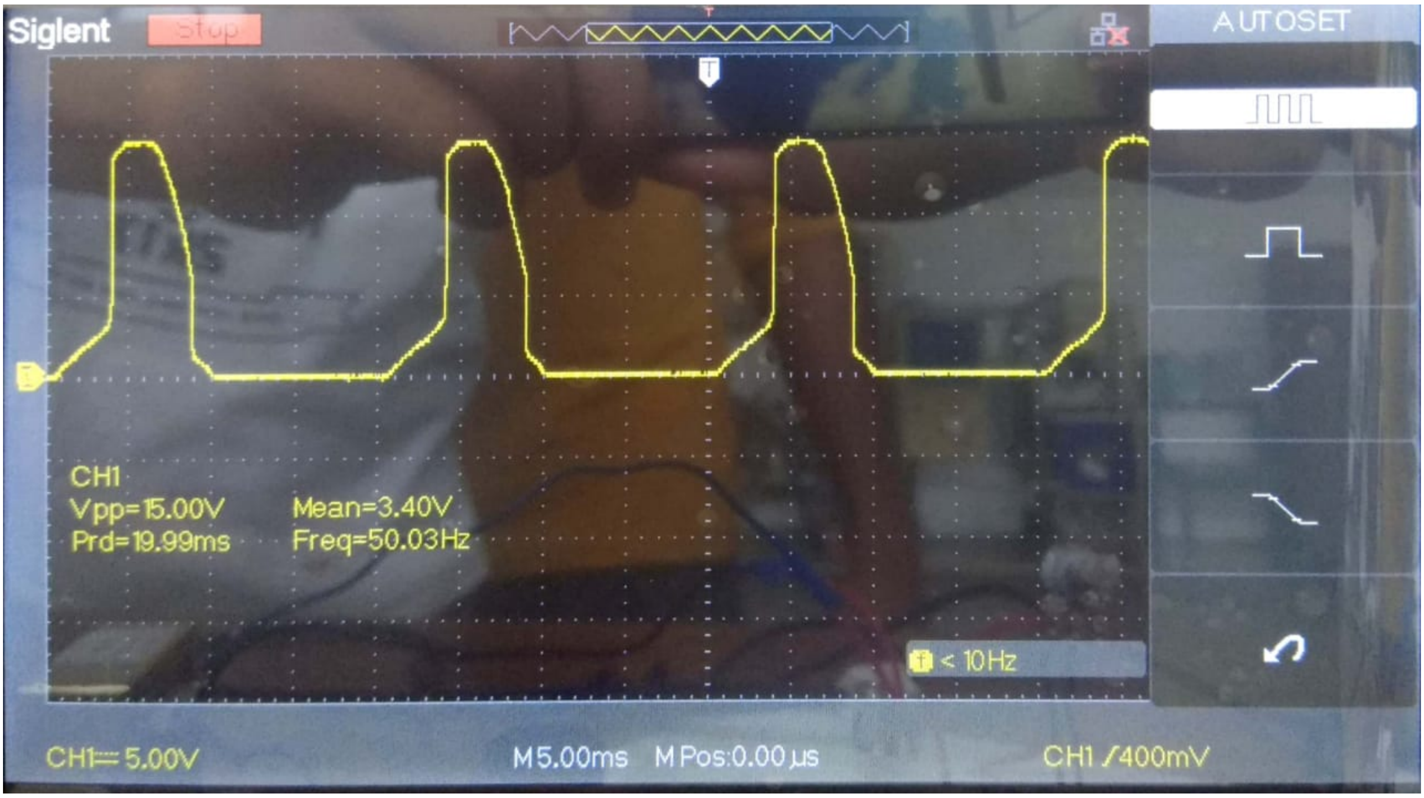


Fig. 8(e) DSO Output Waveform of R Triggering

Result : By performing this experiment, We can say that by Resistive Triggering method, we can trigger SCR between 0° to 90° but can't trigger beyond 90° because of the fact that voltage across R3 resistor is always pure sinusoidal having peak value at 90° and symmetrical about 90° line.

This disadvantage can be overcome by using R-C triggering.

EXPERIMENT 9 : R-C TRIGGERING METHOD FOR A THYRISTOR

Objective : To design and simulate Resistive-Capacitance triggering circuit for Thyristor(SCR).

Required Hardware Components : Step Down (230:18) Transformer

230 V,50 Hz Single phase AC Supply

Two IN4007 diodes

One TYN612 Thyristor (SCR)

One Capacitor (200uF)

One Load Resistors (1 kohm, 5W)

Potentiometer(100 kohm)

Connecting Wires

Soldering Iron and wire

Multimeter

Dotted PCB or Breadboard

Required Software Tool : MATLAB Simulink

Circuit Diagram :

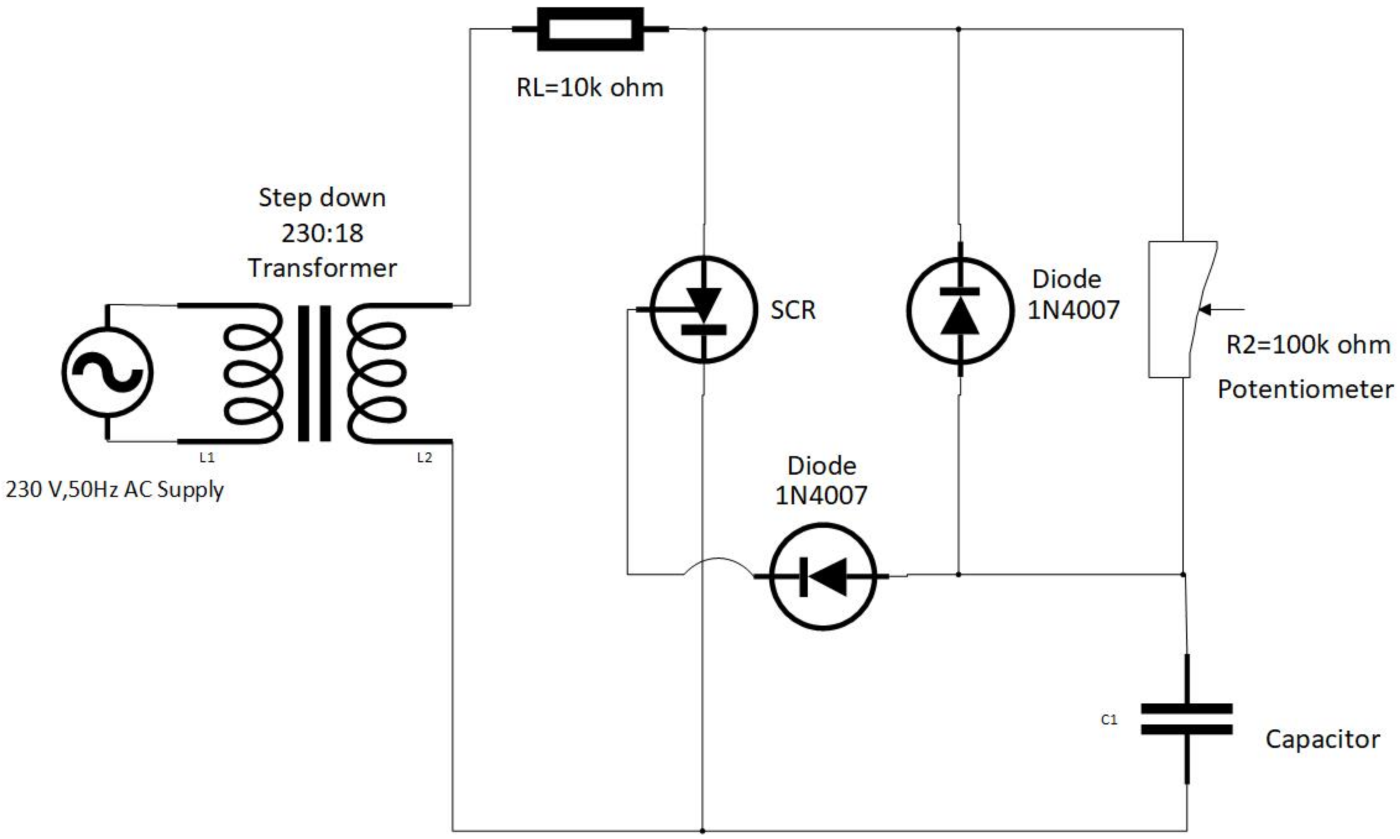


Fig. 9(a) Circuit Diagram of R-C triggering of SCR

Part A : Simulation of circuit

Circuit diagram shown in Fig. 9(a) is designed in MATLAB Simulink below,

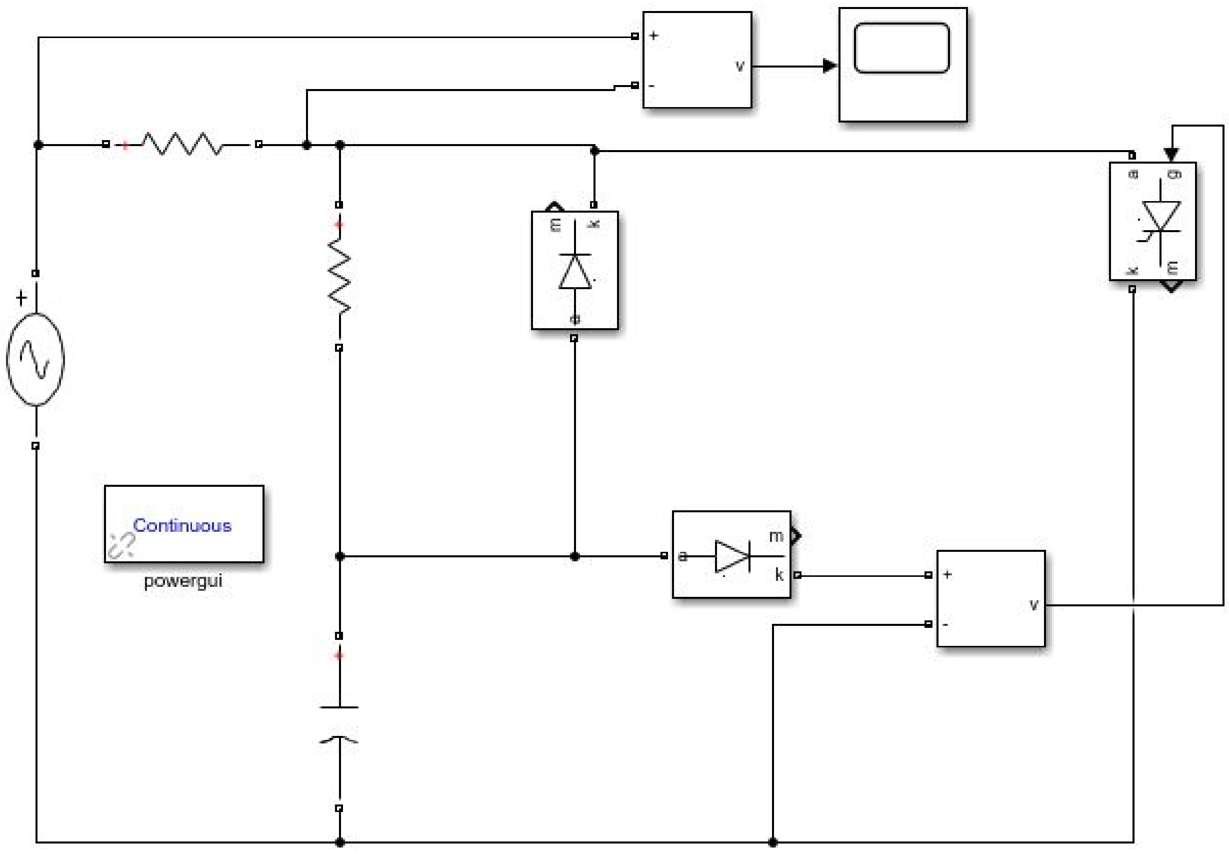


Fig. 9(b) : Simulink Model of R-C triggering for SCR

NOTE : AC supply voltage is having peak amplitude of $Vm = 9\sqrt{2}\,V$ in simulink.

Observation : By changing R2 (Pot), we can change voltage across capacitor and so we can change the firing angle α .

Output Waveform :

Fig. 9(c) : Output Voltage Waveform across load

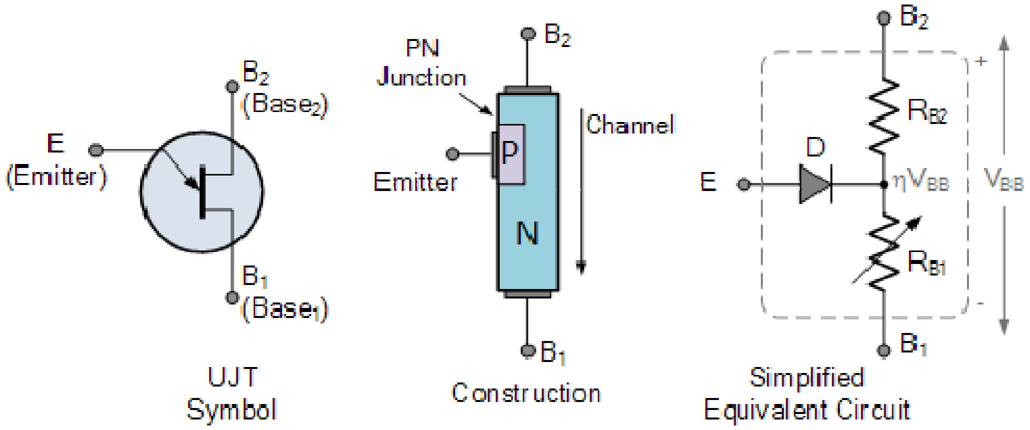
Result : By performing this experiment, We can say that by Resistive-Capacitance Triggering method, we can trigger SCR between 0° to 180°.

EXPERIMENT 10 : Unijunction Transistor (UJT) TRIGGERING METHOD FOR A THYRISTOR

Objective : To design and simulate UJT triggering circuit for Thyristor(SCR).

Theory : A **Unijunction transistor** (**UJT**) is an electronic semiconductor device that has only one junction. The **UJT** has three terminals: an emitter (E) and two bases (B1 and B2). The base is formed by lightly doped n-type bar of silicon. Two ohmic contacts B1 and B2 are attached at its ends. The emitter is of p-type and it is heavily doped. The resistance between B1 and B2, when the emitter is open-circuit is called interbase resistance.

Initially the capacitor charges through variable resistance R whose voltage is applied to the emitter of **UJT**. When the capacitor voltage reaches peak point voltage of **UJT**, the **UJT** will switch to on condition. Now the capacitor discharges through the output resistance. UJT acts as **Relaxation Oscillator.** Thus the pulse is generated in the circuit.



Required Hardware Components :

S No.	Name of component	Specification	Rating	Quantity
1.	Transformer	15 volt	(230:15)	1
2.	Transformer	4503	(1:1:1)	1
3.	Diode	1N4007	0.7volt	4
4.	Zener Diode		15V	1
5.	Resistor	1606	12A/600-1000V	1
6.	UJT	2N2646	50mA/30V	1
7.	Thyristor	TYN612	12A/600-1000V	2

Required Software Tool : Multisim

Circuit Diagram :

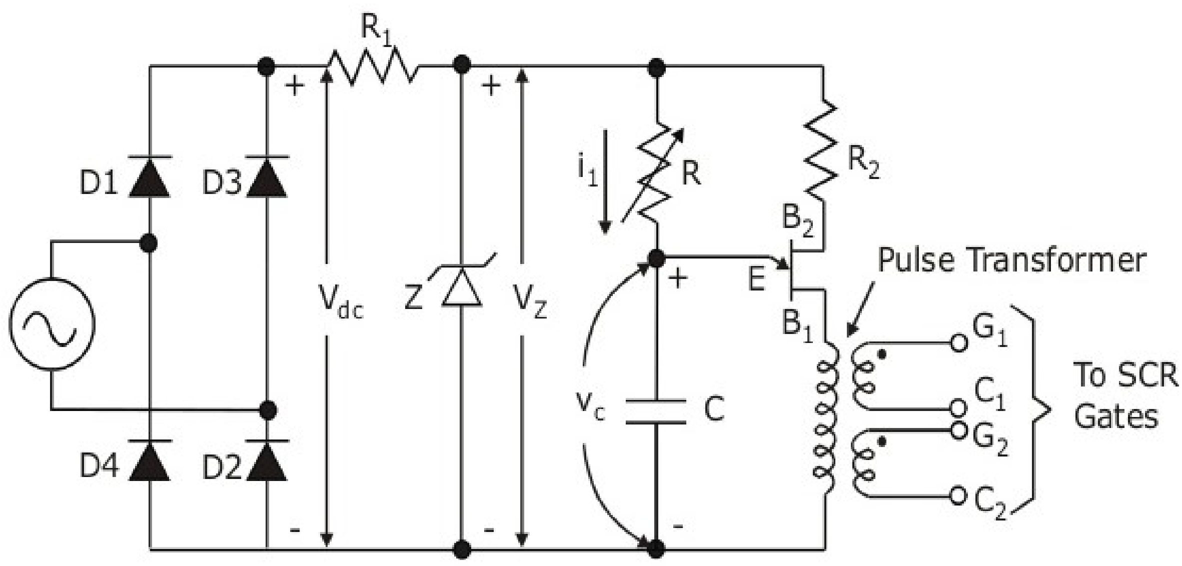


Fig. 10(a) Circuit Diagram of UJT triggering of SCR

Part A : Simulation of circuit

Circuit diagram shown in Fig. 10(a) is designed in Multisim below,

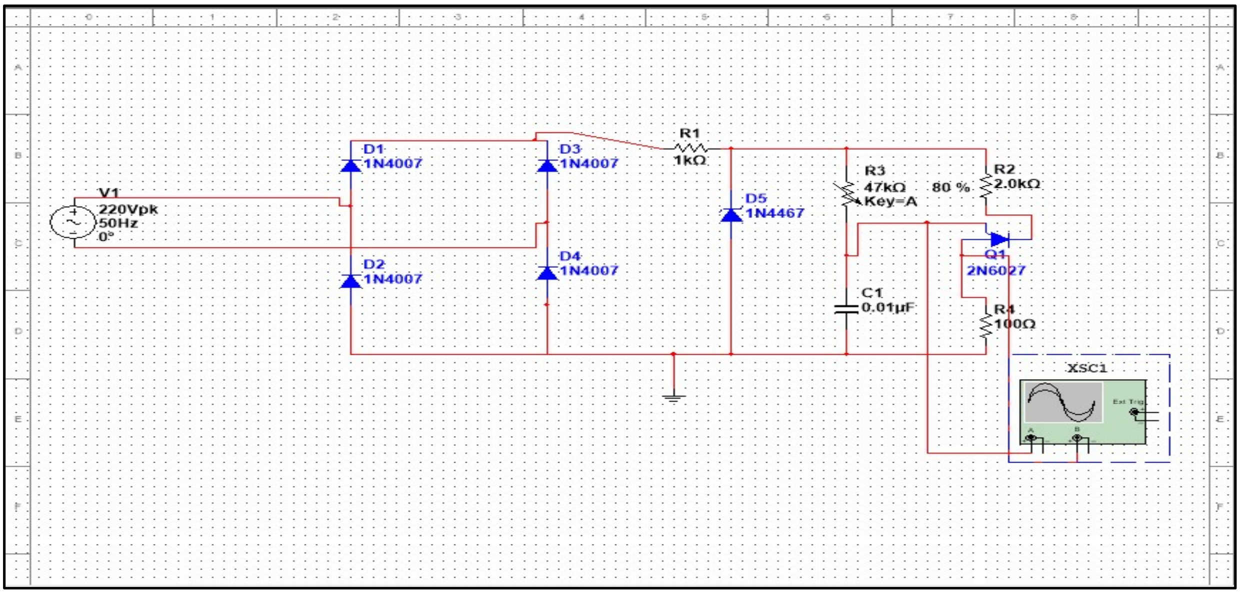


Fig. 10(b) : Multisim Model of UJT triggering for SCR

Observation : By changing R (Pot), we can change voltage across capacitor and so we can change the firing angle α .

Output Waveform :

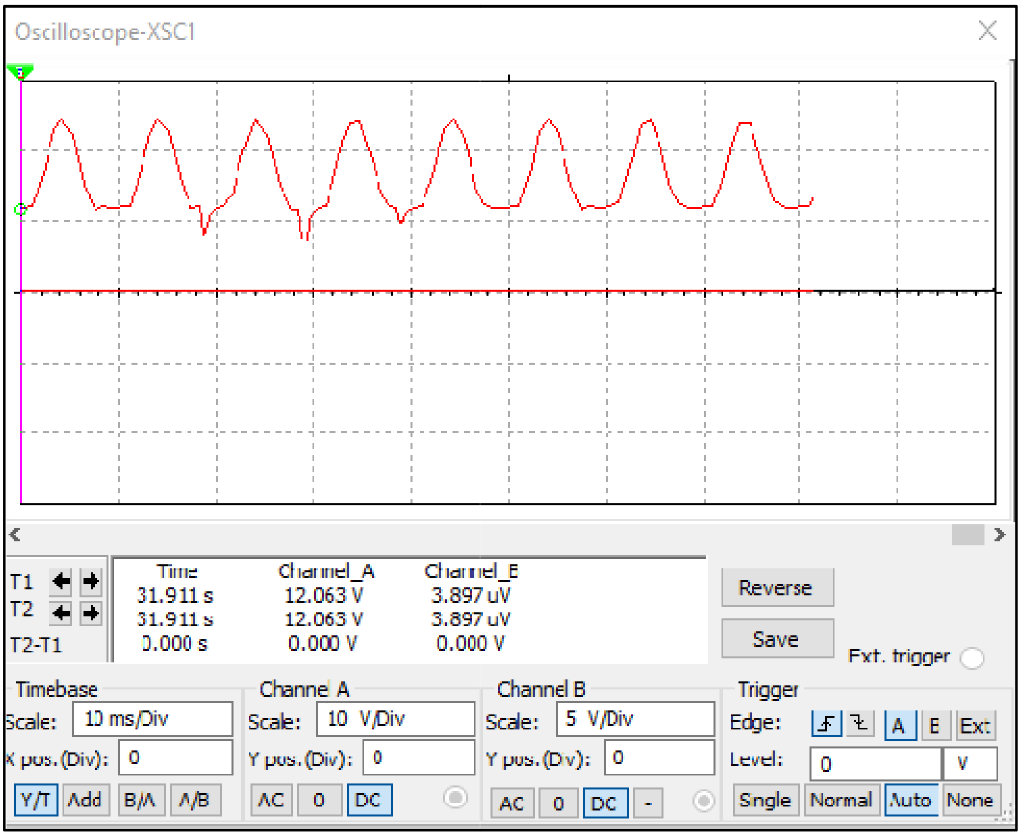


Fig. 10(c) : Simulation Output Voltage Waveform across load

Part B : Hardware Implementation

Hardware Picture :

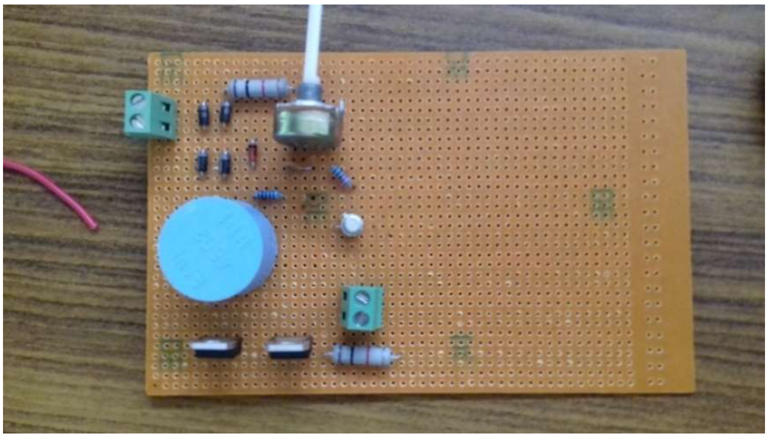


Fig. 10(d) : UJT Triggering circuit for a Thyristor on PCB

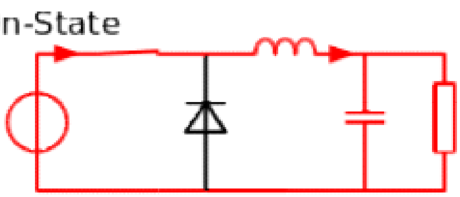
Result : By performing this experiment, We can say that by UJT Triggering method, we can trigger SCR between 0° to 180° same as RC Triggering but additionally this method has better control on triggering because it consist UJT and also it has very less power loss compared to RC Triggering method.

EXPERIMENT 11 : BUCK CONVERTER

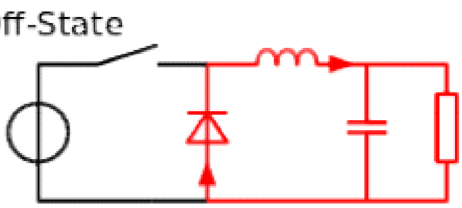
Objective : To design and simulate Buck Converter (Step down DC-DC Converter) circuit.

Theory : A Buck Converter steps down the DC voltage from the input to the output. The circuit operation depends on the conduction state of the MOSFET:

•**On-state**: The current through the inductor increases and the diode blocks.



•**Off-state**: Since the current through the inductor cannot abruptly change the diode must carry the current so it commutates and begins conducting. Energy is transferred from the inductor to the capacitor resulting in a decreasing inductor current.



During steady state the circuit is said to operate:

- in *discontinuous conduction* mode if the inductor current reaches zero and
- in *continuous conduction* mode if the inductor current never reaches zero.

The circuit has two limits of operation. For a PWM duty cycle $D \rightarrow 0$ the output voltage equals zero, and for $D \rightarrow 1$ the output voltage equals V_{in}. In between those limits the output voltage in continuous conduction mode is given by: $V_{out} = D \cdot V_{in}$.

Note that the parallel combination of inductor and capacitor as shown above acts as a second order low pass filter reducing the voltage ripple at the output.

By doing network analysis of Buck Converter circuit, we can derive output voltage formula as below :

$$\boldsymbol{V0 = D \cdot Vs}$$

Where D= Duty Cycle
Vs=Input Supply Voltage
V0=Output Voltage of buck-boost converter

It is seen from this equation that as the Duty cycle can vary between $0 < D < 1$, Output voltage Vo will always be less than input voltage Vs.

As it is step down DC-DC Converter hence the name Buck Converter.

Required Hardware Components :

S No.	Name of component	Specification	Rating	Quantity
1.	Transformer	15 volt dc	(230:15)	1
2.	Transformer	9 volt dc	(230:09)	1
3.	Power Supply	IC 7815	15 volt dc	1
4.	Power Supply	IC 7915	15 volt dc	1
5.	Power Supply	IC7812	12 volt dc	1
6.	Mosfet	IRF460	500V,6A	1
7.	Inductor		10mH	1
8.	Power Diode			1
9.	Capacitor		400V,2.2μF	1
10.	Resistor		100Ω	1
11.	Arduino Uno			1
12.	Gate Driver	TLP250	10-35V,11mA	1
13.	Multimeter			1
14.	PCB			4

Required Software Tool : MATLAB Simulink

Circuit Diagram :

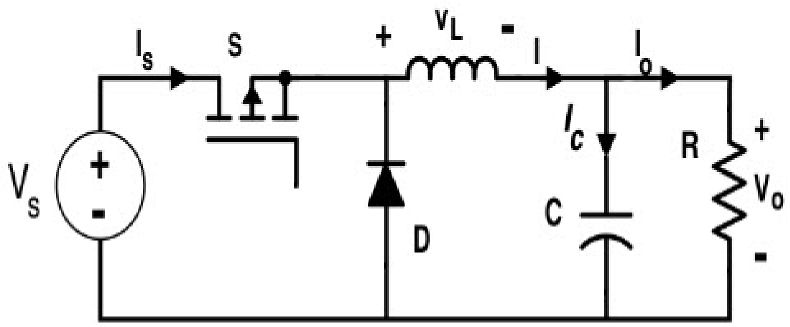


Fig. 11(a) Circuit Diagram of Buck Converter

Part A : Simulation of circuit

Circuit diagram shown in Fig. 11(a) is designed in Simulink below,

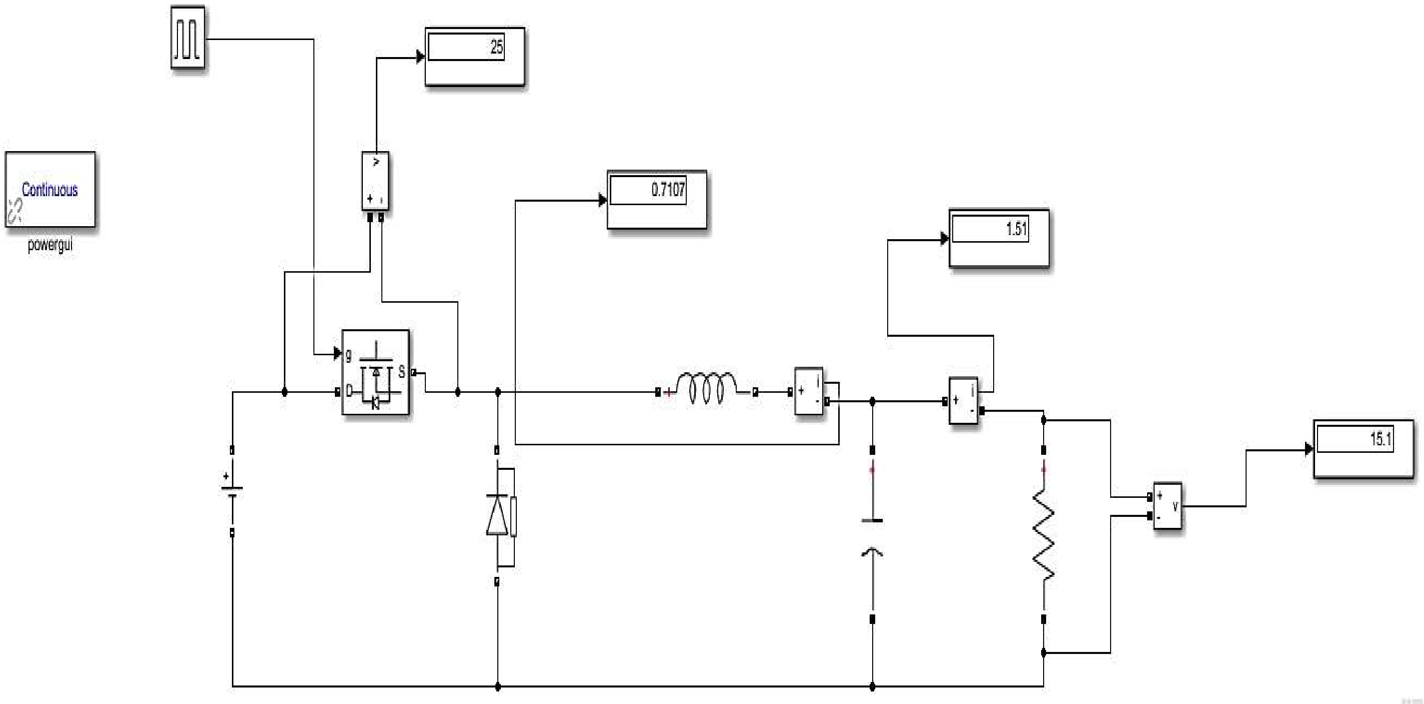


Fig. 11(b) : Simulink Model of Buck Converter

❖ Calculations for Simulink model

- Input voltage, $V_{in} = 25V$
- Output voltage, $V_{out} = 15.1V$
- f = 20 KHz L = 0.4mH R=10ohm

we know, $V_{out} = D \cdot V_{in}$

So, $D = (15.1/25) = 0.6$

$I_0 = I_L = V_{out} / R = (15.1/10) = 1.51A$

$\Delta I_L = (D(1 - D)V_{in}) / (f * L) = 0.7A$

Part B : Hardware Implementation

Hardware Picture :

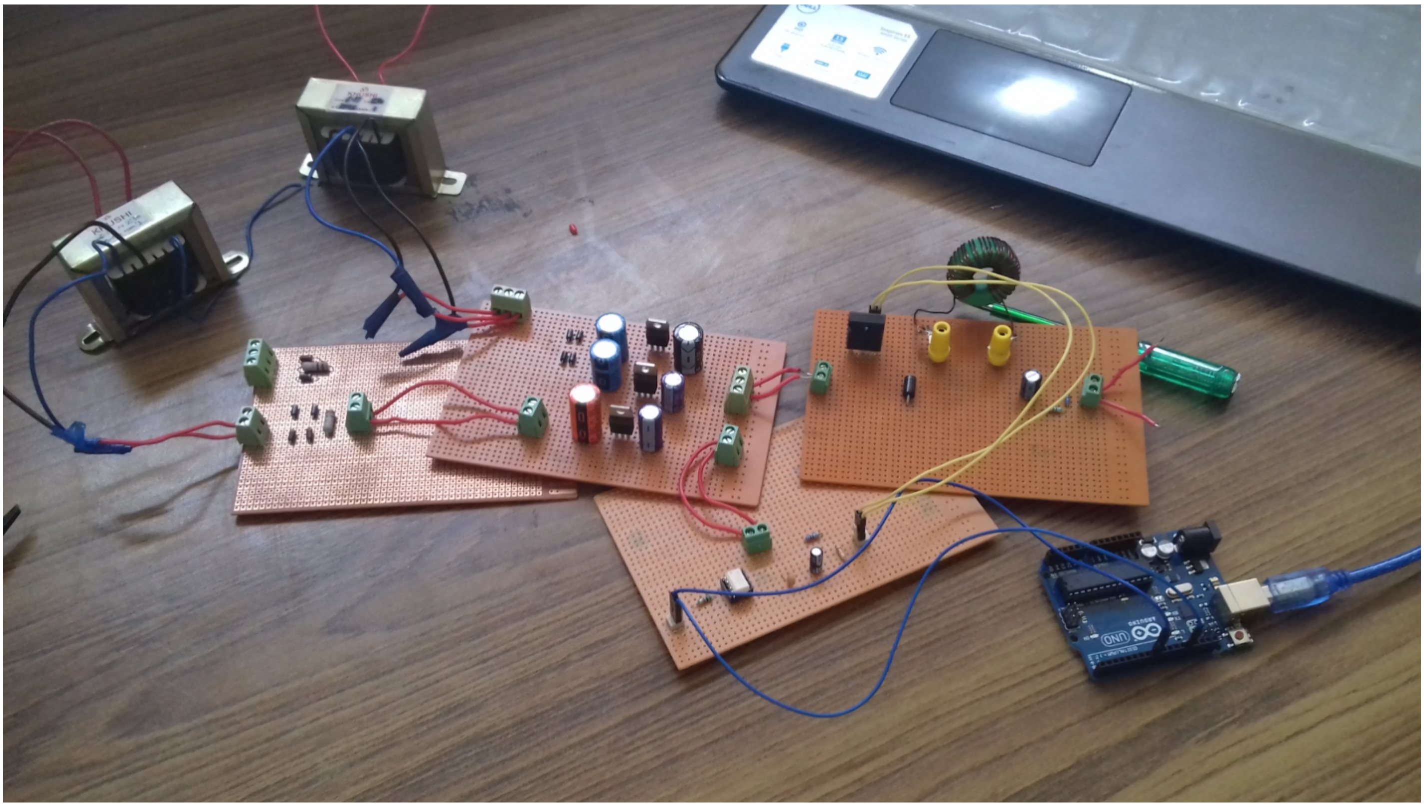


Fig.11(c) Circuit of Buck Converter on Hardware PCB

Observation : After implementing circuit as shown in fig. 11(a) in hardware, we could get following readings by changing duty cycle controlled by Arduino UNO.

Reading 1.) For Duty Cycle D=0.4, Output Voltage **V0= 5.7 V**

Reading 2.) For Duty Cycle D=0.7, Output Voltage **V0= 10.2 V**

Reading 3.) For Duty Cycle D=0.9, Output Voltage **V0= 13.2 V**

Result : By changing Duty Cycle and after checking the readings and comparing to that of theoretical respectively, we can conclude that buck converter is correctly working.

EXPERIMENT 12 : BOOST CONVERTER

Objective : To design and simulate Boost Converter (Step up DC-DC Converter) circuit.

Theory : A Boost Converter steps up the DC voltage from the input to the output. The circuit operation depends on the conduction state of the MOSFET:

By doing network analysis of Boost Converter circuit shown in fig(a), we can derive output voltage formula as below :

$$\boldsymbol{V0} = \frac{\boldsymbol{Vs}}{\boldsymbol{1 - D}}$$

Where D= Duty Cycle
Vs=Input Supply Voltage
V0=Output Voltage of buck-boost converter

It is seen from this equation that as the Duty cycle can vary between $0 < D < 1$, Output voltage Vo will always be greater than input voltage Vs.

As it is step up DC-DC Converter hence the name Boost Converter.

Required Hardware Components :

S No.	Name of component	Specification	Rating	Quantity
1.	Transformer	15 volt dc	(230:15)	1
2.	Transformer	9 volt dc	(230:09)	1
3.	Power Supply	IC 7815	15 volt dc	1
4.	Power Supply	IC 7915	15 volt dc	1
5.	Power Supply	IC7812	12 volt dc	1
6.	Mosfet	IRF460	500V,6A	1
7.	Inductor		10mH	1
8.	Power Diode			1
9.	Capacitor		400V,2.2μF	1
10.	Resistor		100Ω	1
11.	Arduino Uno			1
12.	Gate Driver	TLP250	10-35V,11mA	1
13.	Multimeter			1
14.	PCB			4

Required Software Tool : MATLAB Simulink

Circuit Diagram :

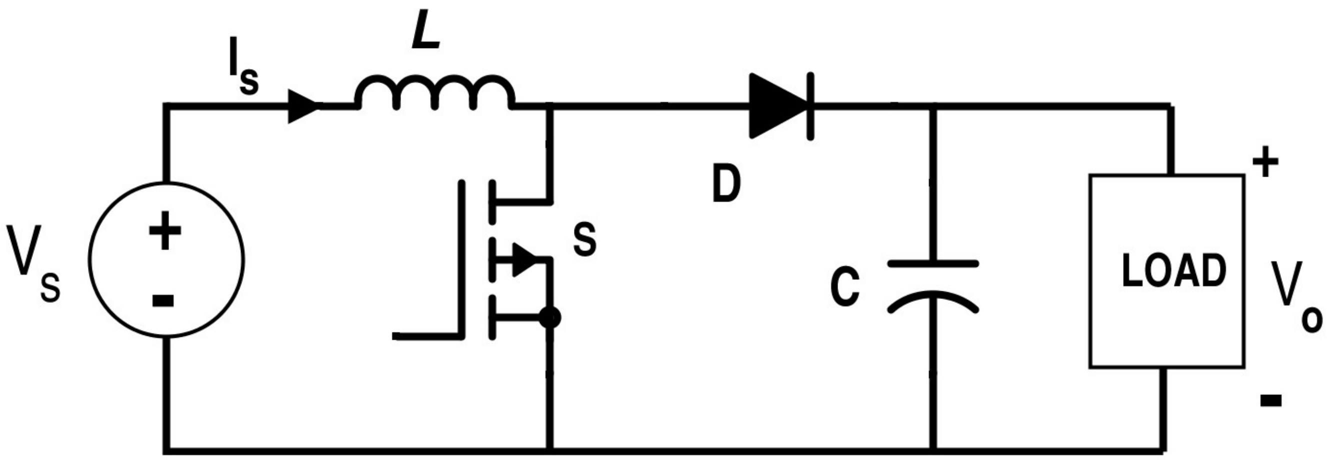


Fig. 12(a) Circuit Diagram of Boost Converter

Part A : Simulation of circuit

Circuit diagram shown in Fig. 12(a) is designed in Simulink below,

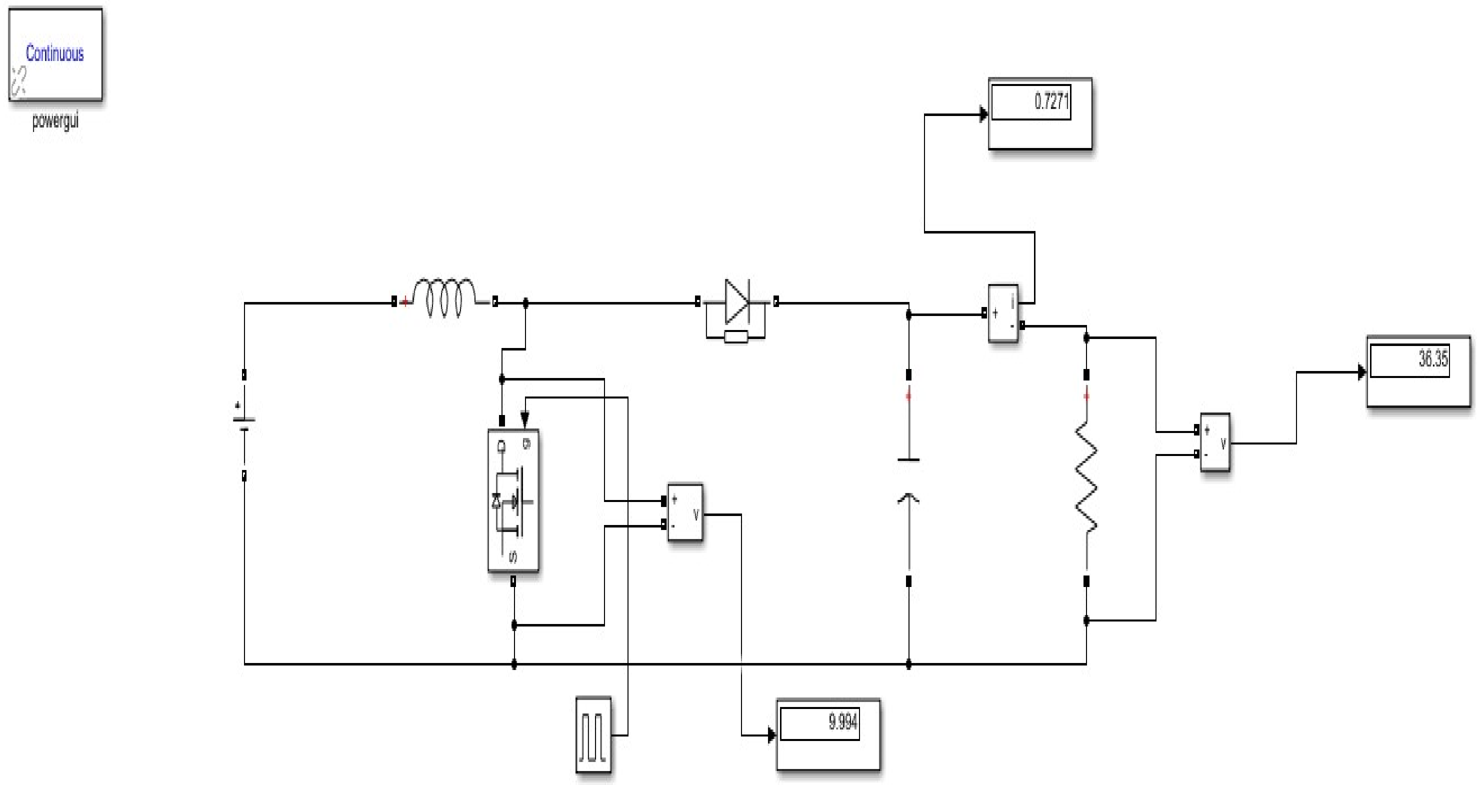


Fig. 12(b) : Simulink Model of Boost Converter

❖ **Calculations for Simulink model :**

- Input voltage, V_{in} = 10V
- Output voltage, V_{out}= 36.35V
- f = 50KHz L= 96µH R=50ohm

we know, $V_{out} = V_{in}/(1-D)$.

So, $D = (1 - V_{in} / V_{out}) = 0.73$

$I_0 = V_{out} /R = (36.35 / 50) = 0.727A$

$\Delta I_L = (D*V_{in}) /(f * L) = 1.52A$

Part B : Hardware Implementation

Hardware Picture :

Circuit of Boost Converter on Hardware PCB is similar as Fig.11(c)

Observation :

After implementing circuit as shown in fig(a), we could get following readings by changing duty cycle controlled by Arduino UNO.

Reading 1.) For Duty Cycle D=0.4, Output Voltage **V0= 24.4 V**

Reading 2.) For Duty Cycle D=0.6, Output Voltage **V0= 36.8 V**

Reading 3.) For Duty Cycle D=0.7, Output Voltage **V0= 46.7 V**

Result : By changing Duty Cycle and after checking the readings and comparing to that of theoretical respectively, we can conclude that boost converter is correctly working.

EXPERIMENT 13 : BUCK-BOOST CONVERTER

Objective : To design and simulate Buck-Boost Converter (Step up/down DC-DC Converter) circuit.

Theory : A Buck-Boost Converter steps up or down the DC voltage from the input to the output. The circuit operation depends on the conduction state of the MOSFET:

By doing network analysis of Buck-Boost Converter circuit shown in fig(a), we can derive output voltage formula as below :

$$\boldsymbol{V0} = \frac{-\boldsymbol{D}}{\mathbf{1} - \boldsymbol{D}} \boldsymbol{Vs}$$

Where D= Duty Cycle
Vs=Input Supply Voltage
V0=Output Voltage of buck-boost converter

1) It is seen from this equation that for duty cycle $D < 0.5$, it will act as a buck converter and will step down the voltage according to duty cycle set.

2) For duty cycle $D > 0.5$, it will act as a boost converter and will step up the voltage according to duty cycle set.

3) For duty cycle $D = 0.5$, it will act as a buffer and will give output voltage same as input voltage.

Required Hardware Components :

S No.	Name of component	Specification	Rating	Quantity
1.	Transformer	15 volt dc	(230:15)	1
2.	Transformer	9 volt dc	(230:09)	1
3.	Power Supply	IC 7815	15 volt dc	1
4.	Power Supply	IC 7915	15 volt dc	1
5.	Power Supply	IC7812	12 volt dc	1
6.	Mosfet	IRF460	500V,6A	1
7.	Inductor		10mH	1
8.	Power Diode			1
9.	Capacitor		400V,2.2μF	1
10.	Resistor		100Ω	1
11.	Arduino Uno			1
12.	Gate Driver	TLP250	10-35V,11mA	1
13.	Multimeter			1
14.	PCB			4

Required Software Tool : MATLAB Simulink

Circuit Diagram :

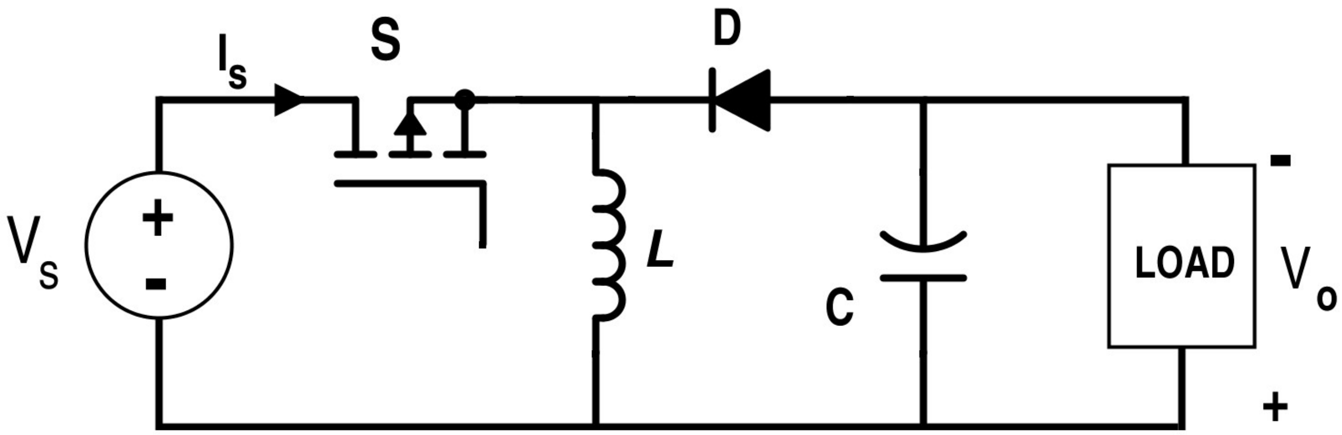


Fig. 13(a) Circuit Diagram of Buck-Boost Converter

NOTE: Output voltage polarity is shown as reversed in Circuit diagram of Fig. 13(a) Buck-Boost Converter.

Explaination : When MOSFET is in ON condition, inductor gets charged by supply and diode remains in reverse biased(open circuited). As the Inductor gets charged by polarity with positive at lower end(i.e Ground) and negative at upper end, It will force diode to come in forward bias(short circuit) and inductor provides current to load through diode. Thus the current flowing in opposite direction(from lower to upper end) explain the reason of opposite output voltage polarity at load.

Also this is the reason, we inspect negative sign (-) in the formula of Output voltage of Buck-Boost converter as stated earlier because in Analysis conventional polarity is opposite then actual one.

Part A : Simulation of circuit

Circuit diagram shown in Fig. 13(a) is designed in Simulink below,

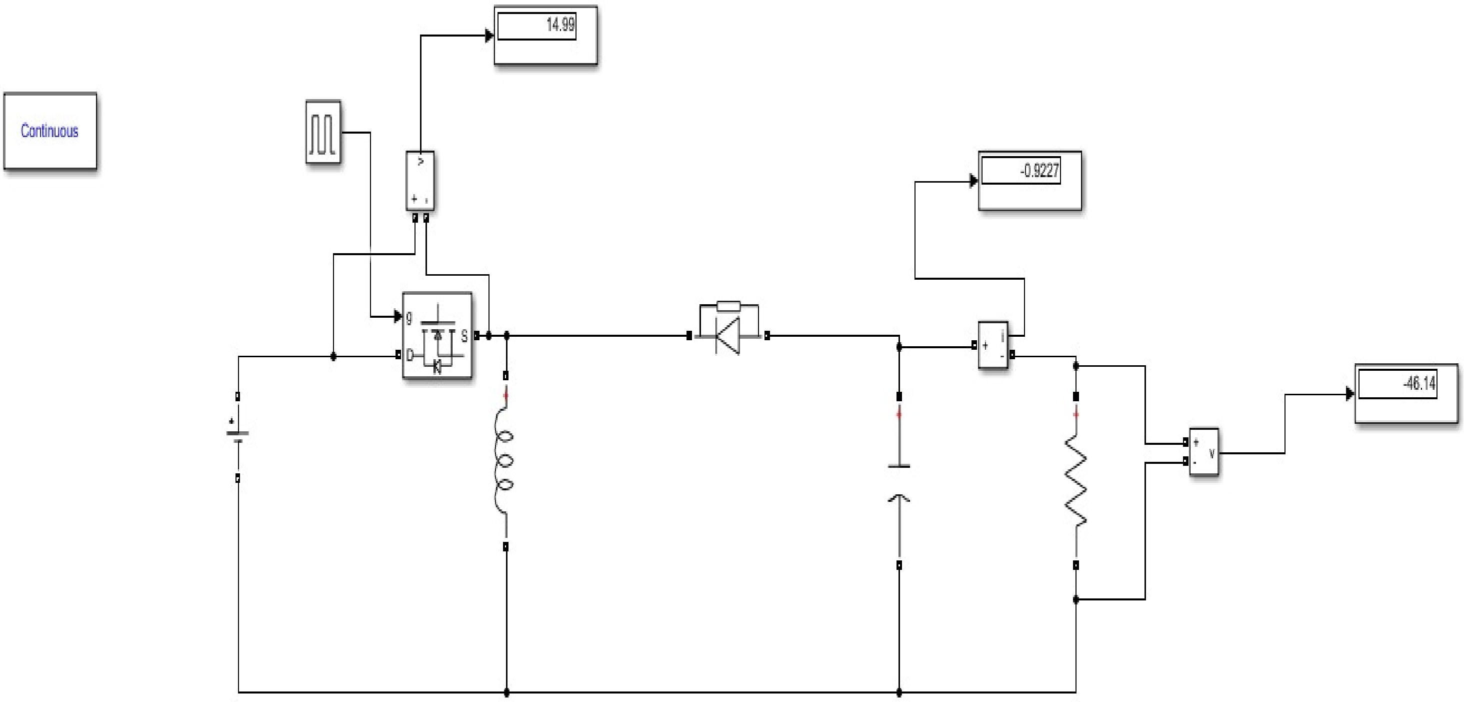


Fig. 13(b) : Simulink Model of Buck-Boost Converter

❖ **Calculations for Simulink model :**

- Input voltage, V_{in} = 15V
- Output voltage, V_{out}= 46.14V
- f=50kHz L = 96µH R=50ohm

we know, $V_{out} = -DV_{in}/(1-D)$.

So, $D = V_{out} / (V_{out} - V_{in}) = 0.75$

$I_0 = V_{out} /R = (46.14 / 50) = 0.9228A$

$I_L = (D*V_{in}) /(f * L) = 0.0225A$

Part B : Hardware Implementation

Hardware Picture :

Circuit of Buck-Boost Converter on Hardware PCB is similar as Fig.11(c)

Observation :

After implementing circuit as shown in fig(a), we could get following readings by changing duty cycle controlled by Arduino UNO.

Reading 1.) For Duty Cycle D=0.4, Output Voltage **V0= 10.2 V**

Reading 2.) For Duty Cycle D=0.6, Output Voltage **V0= 23 V**

Reading 3.) For Duty Cycle D=0.5, Output Voltage **V0= 14.8 V**

Result : By changing Duty Cycle and after checking the readings and comparing to that of theoretical respectively, we can conclude that buck-boost converter is correctly working.