A

Report

of

Self – Project

on

Simulation & PCB Design of single phase Voltage Source Inverter (VSI) using LTspice & Eagle software.

Prepared

by

Ranjeet Kumar

(Roll No. – 234102105)

M. Tech. in Power Engineering

भारतीय प्रौद्योगिकी संस्थान गुवाहाटी • Indian Institute of Technology Guwahati •

Department of Electronics & Electrical Engg.
Indian Institute of Technology, Guwahati

Abstract

This report discusses the simulation and PCB design of a single-phase Voltage Source Inverter (VSI) using two popular software tools: LTspice for circuit simulation and Eagle for Printed Circuit Board (PCB) design. The VSI is a critical component in various power electronic applications, including motor drives and renewable energy systems. This report outlines the design process, simulation setup, and PCB layout for a single-phase VSI.

Introduction:

Single Phase Full Bridge Inverter is basically a voltage source inverter, a power electronic device that converts DC voltage into a controlled AC voltage. It is widely used in applications where variable AC voltage is required, such as adjustable speed drives and renewable energy systems.

Unlike Single Phase Half Bridge Inverter, this inverter does not require three wire DC input supply. Rather, two wire DC input power source suffices the requirement. The output frequency can be controlled by controlling the turn ON and turn OFF time of MOSFET.

The output or load voltage can be resolved into Fourier series:

$$V_0 = \sum_{n=1,3,5.....}^{\infty} \frac{4V_s}{(n\pi)} sinn\omega t$$

Design Parameters:

❑ Design of Snubber Circuit:

Snubber circuits provide protection against transient voltages that occur during turn-off. Generally, a simple RC snubber uses a resistor R in series with a capacitor C. The RC snubber is connected in parallel with a power MOSFET. Cutting off a current in a circuit causes its voltage to increase sharply due to stray inductances. The snubber damps this surge voltage to protect the power MOSFET as well as components in its vicinity.

The value of capacitance:

$$\boldsymbol{C = I_{load\ current} * \frac{T_{off}}{V_{DC}}}$$

The value of Resistance:

$$\boldsymbol{I_{(load\ max)} + \frac{V_{DC}}{R} < I_{peak\ cap\ device}}$$

$$\boldsymbol{R > \frac{V_{DC}}{I_{peak\ cap\ device} - I_{load\ max}}}$$

Simulation steps using LTspice:

1. Circuit Design: The single-phase VSI circuit consists of power switches (usually insulated-gate bipolar transistors - IGBTs), gate drivers, and control circuitry. The circuit topology typically includes a full-bridge configuration.

2. Component Selection: Choose appropriate IGBTs, diodes, gate driver ICs, and other passive components. The datasheets of these components provide necessary information like switching characteristics, voltage and current ratings.

3. Gate Drive Signals: Design the gate drive circuitry to provide precise and synchronized gate signals to the IGBTs. This involves voltage level shifting, dead-time management, and protection against shoot-through currents.

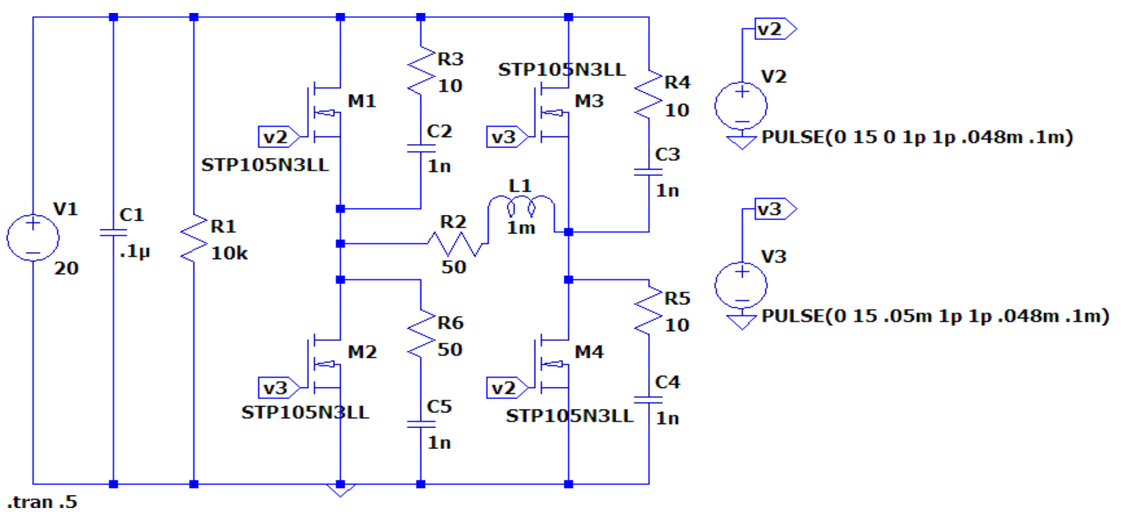
4. Control Strategy: Implement a control strategy such as Pulse Width Modulation (PWM) to generate the desired AC output voltage. The control strategy helps in achieving the required output frequency and magnitude.

5. Simulation Setup: Construct the complete circuit in LTspice, including the gate drive signals and control logic. Define simulation parameters, initial conditions, and load characteristics.

6. Transient Analysis: Run transient simulations to observe the transient response of the VSI under different load conditions, switch timings, and control strategies. Analyse waveform outputs, voltage ripples, and efficiency.

Use of component:	Specification
MOSFET: STP105N3LL	MOSFET N - Ch. 30V 2.7mOhm 150A STripFET VI
Gate driver IC: HCPL3120	2.5A Gate Driver Optical Coupling 3750Vrms 1 Channel 8-DIP Gull Wing
Snubber-resistance: ERJP06J100V	10 Ohms ±5% 0.5W, 1/2W Chip Resistor 0805 (2012 Metric) AutomotiveAEC-Q200, Pulse Withstanding Thick Film
Snubber Capacitance: C1206C104K5RACTU	0.1 µF ±10% 50V Ceramic Capacitor X7R 1206 (3216 Metric)
Film capacitor: B32672L8102	1000 pF Film Capacitor 700V 2000V (2kV) Polypropylene (PP), Metallized radial
Wire wound resistor: HS25KJ	Chassis mount 25W 1000 Ohms 5%
TWO PIN: 1715721	

Simulation model of LT Spice:



Simulation Waveform (Input voltage and MOSFET switching)

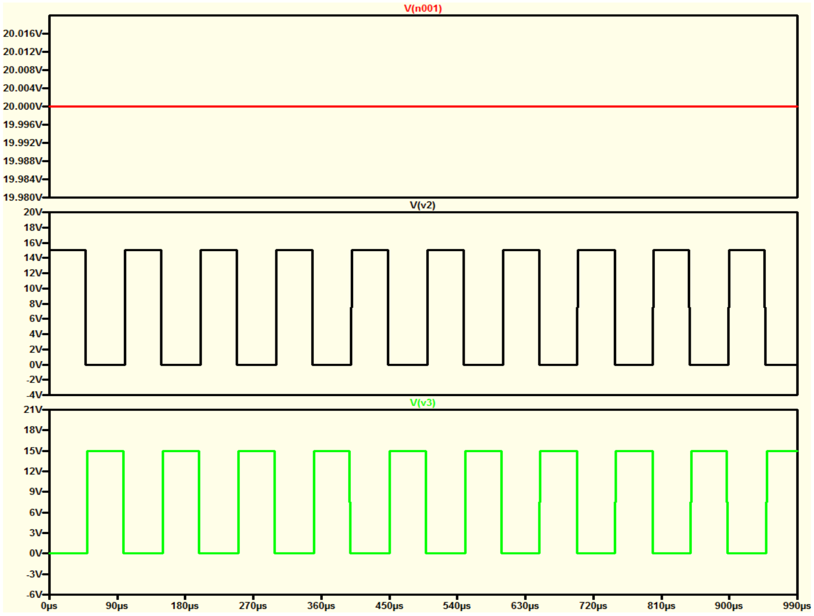
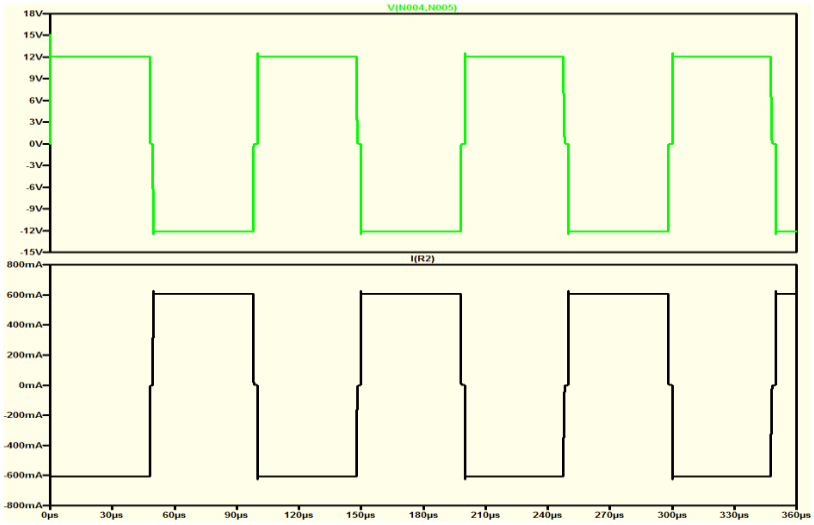
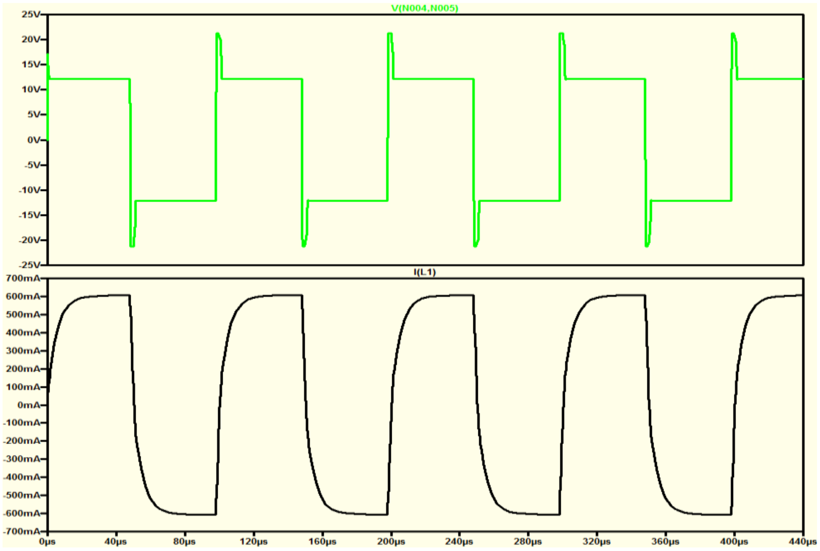


Fig. 1:

Simulation Waveform (output Voltage and output current for R load)



Simulation Waveform (output Voltage and output current for R-L load)



PCB Designing steps using Eagle:

1. Schematic Design: Create a schematic diagram of the VSI circuit using Eagle software. Place the components, connect them, and label the nets properly.

2. Component Footprints: Select or create appropriate component footprints in Eagle's library. Ensure that the footprints match the physical dimensions of the actual components.

3. PCB Layout: Transfer the schematic design into the PCB layout editor. Position the components on the PCB, considering factors like component heat dissipation, signal integrity, and space constraints.

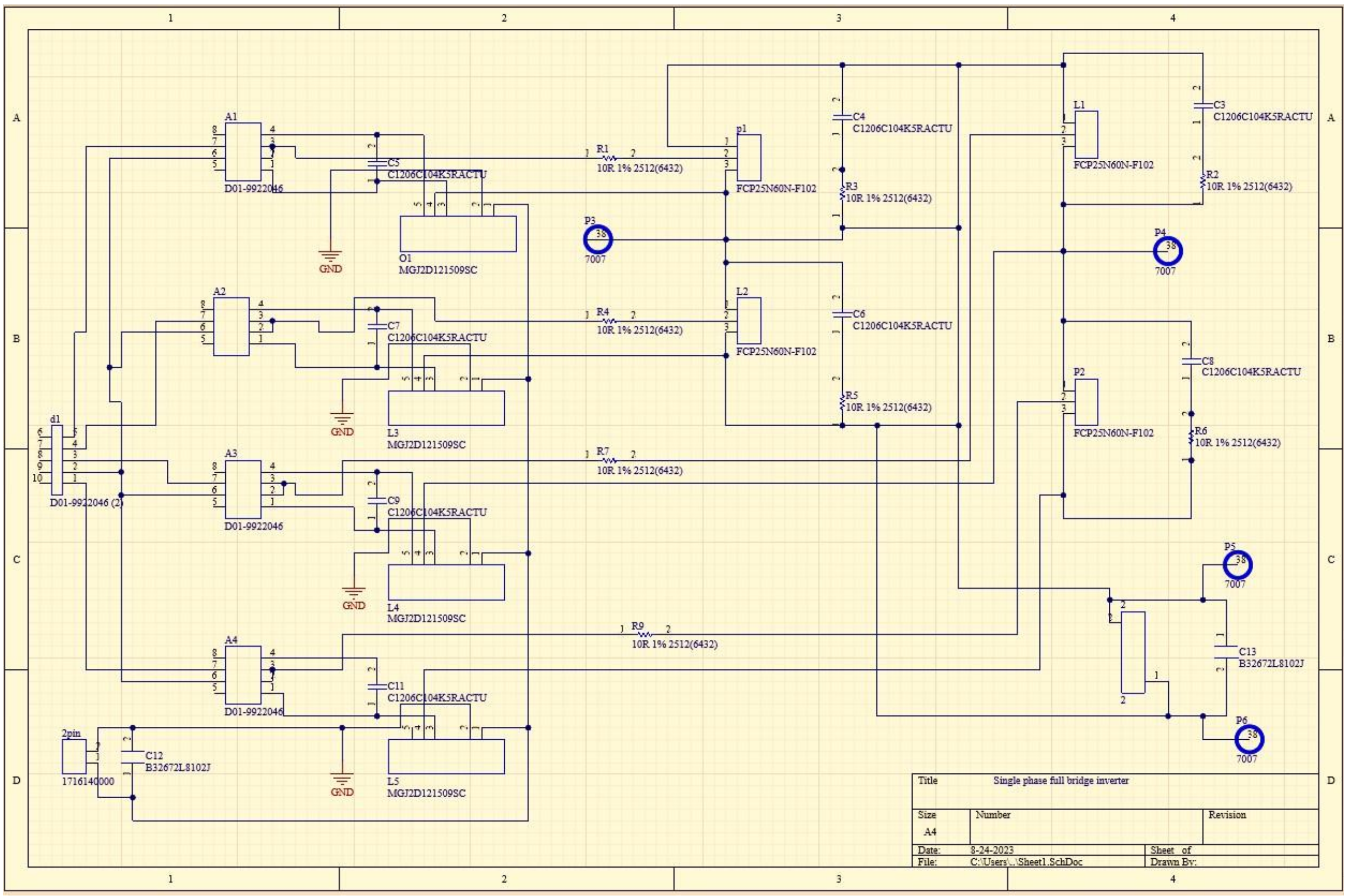
4. Routing: Route the traces carefully, adhering to best practices for power and signal routing. Separate high-current paths from sensitive signal paths to minimize interference.

5. Grounding: Implement a proper grounding scheme to reduce noise and ensure proper functioning of the circuit. Connect ground planes and minimize ground loops.

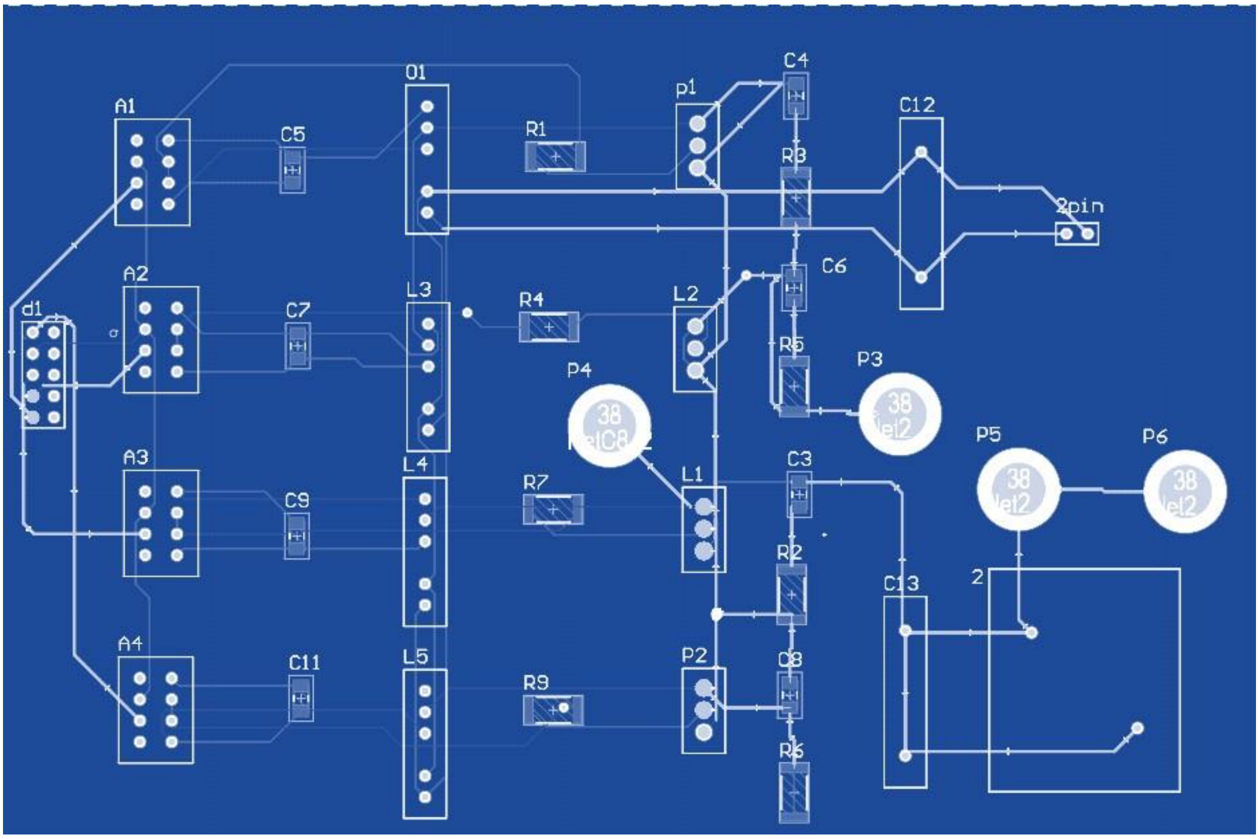
6. Design Rule Check (DRC): Run a DRC to identify and correct any design rule violations such as trace width violations, clearance issues, or unconnected nets.

7. Gerber Files Generation: Once the PCB layout is complete, generate Gerber files that contain the necessary information for manufacturing the PCB.

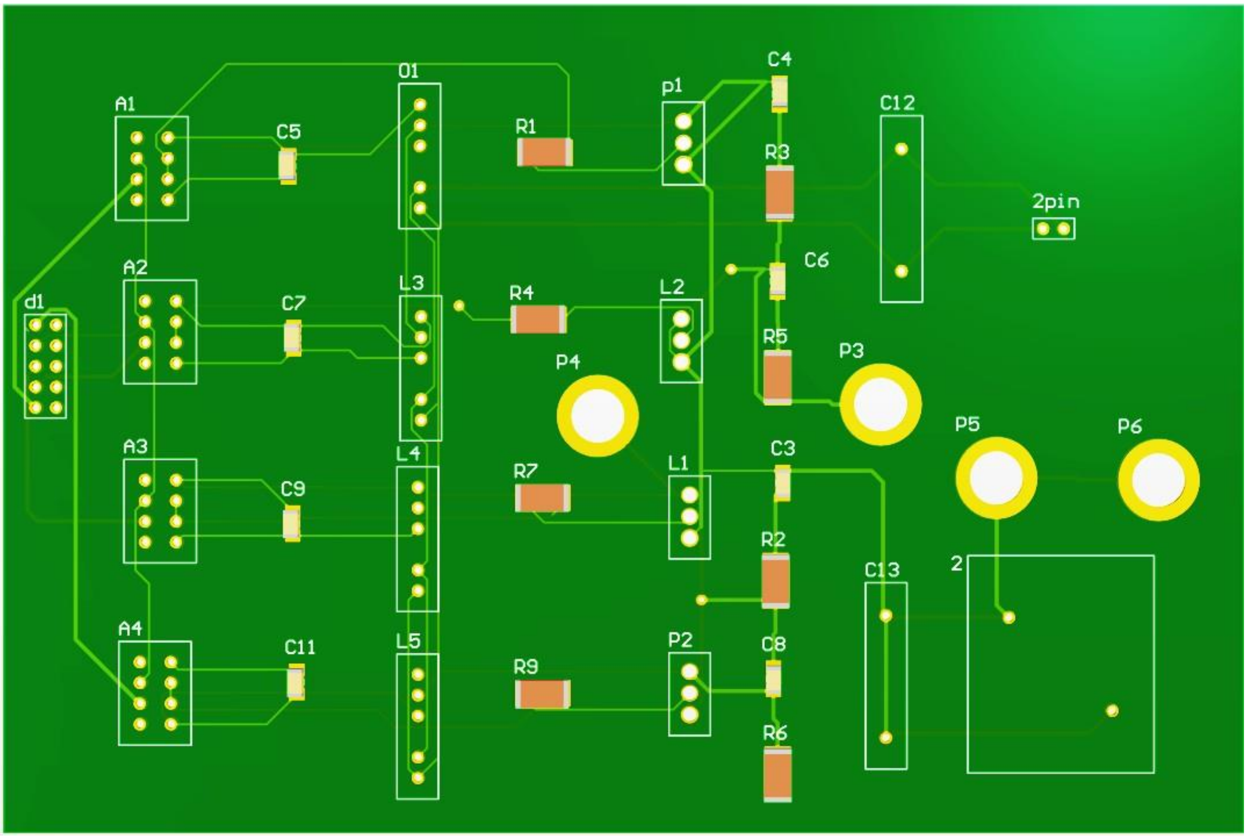
PCB schematic circuit diagram:



Routed Board:



Gerber File:



Conclusion:

This report discussed the simulation and PCB design of a single-phase Voltage Source Inverter (VSI) using LTspice and Eagle software. The simulation phase involves creating a functional circuit in LTspice, implementing a control strategy, and analysing the transient response. The PCB design phase involves schematic creation, component placement, trace routing, grounding, and adherence to design rules. The integration of simulation and PCB design tools allows engineers to design and validate complex power electronic circuits efficiently before physical implementation.

- ☞ Gate driver HPCL3120 is considered for giving switching pulses to the MOSFET.