Exp. No.: 04 Date:03/04/23	PROTOTYPE DESIGN FOR LOW NOISE PREAMPLIFIER CIRCUIT

OBJECTIVE:

To construct a prototype design for a low noise pre amplifier circuit.

AIM:

To design a low noise high performance frequency amplifier circuit using NE5534 IC.

SOFTWARE REQUIRED:

ALTIUM DESIGNER 2022.

THEORY:

In a communication system, the receiver section needs amplification for the weak signal which is received from the antenna. So this amplification can be accomplished through the main component called Low Noise Amplifier or LNA. The characteristics of this amplifier can be described by certain parameters like gain, noise figure, chip area, linearity, power consumption & bandwidth.

The block diagram of the low noise amplifier is shown below. The designing of a low noise amplifier can be done by using a common gate amplifier, active inductor & common drain stage. Generally, the common gate amplifier is mainly used at the input stage whereas the common drain amplifier is used at the output stage to provide the best input as well as output matching. The low noise amplifier is bound with particular characteristics like gain & noise figure but the selection of LNA mainly depends on some specific parameters like power supply, bandwidth, chip area & linearity.

PCB DESIGN:

PCB (Printed Circuit Board) design is the Signal Integrity Theory. Signal integrity refers to the quality of electrical signals as they travel through a PCB. It focuses on minimizing noise, interference, and distortion to ensure that signals remain accurate and reliable. According to the

Signal Integrity Theory, there are several factors that need to be considered during PCB design to maintain signal integrity:

Trace Routing: Proper routing of traces is crucial to minimize signal degradation. High-speed signals should be routed as short and direct as possible to reduce signal reflections and crosstalk. Differential pairs should be routed in close proximity and with equal length to maintain signal balance.

Impedance Matching: Impedance matching ensures that the impedance of the transmission line matches the impedance of the source and load components. This helps to minimize signal reflections, which can cause signal distortion and loss.

Grounding and Power Distribution: Proper grounding and power distribution are essential for maintaining a stable reference voltage and reducing noise. Ground planes should be used to provide a low impedance return path for signals and to minimize ground loops. Decoupling capacitors should be placed near high-speed components to provide clean power and suppress voltage fluctuations.

EMI/EMC Considerations: Electromagnetic interference (EMI) and electromagnetic compatibility (EMC) issues can arise due to high-speed signals. Careful PCB layout and component placement can help reduce EMI/EMC problems. Techniques like shielding, signal isolation, and controlled impedance routing can be employed to minimize electromagnetic emissions and susceptibility.

Thermal Considerations: High-power components can generate heat, which needs to be managed to prevent thermal issues. Proper placement of heat sinks, thermal vias, and consideration of the PCB's thermal conductivity are important to dissipate heat effectively and prevent component failure.

Design for Manufacturability: PCB designs should be manufactural within the constraints of the chosen fabrication and assembly processes. DFM considerations involve optimizing component placement, trace widths, and clearances to ensure ease of manufacturing, inspection, and testing. These are some of the key aspects of the Signal Integrity Theory in PCB design. By understanding and implementing these principles, designers can improve signal quality, minimize errors, and ensure reliable operation of electronic devices.

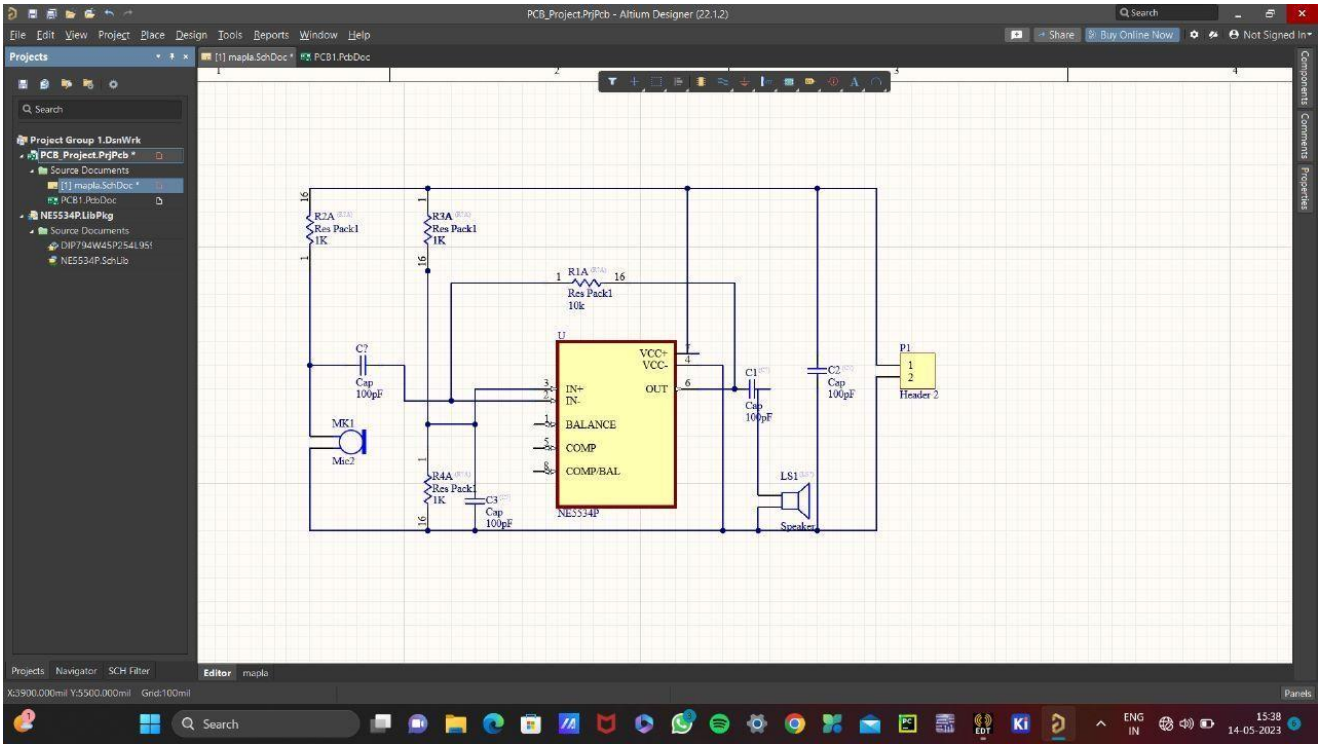

Figure 4.1: Schematic diagram of Low noise pre amplifier circuit.

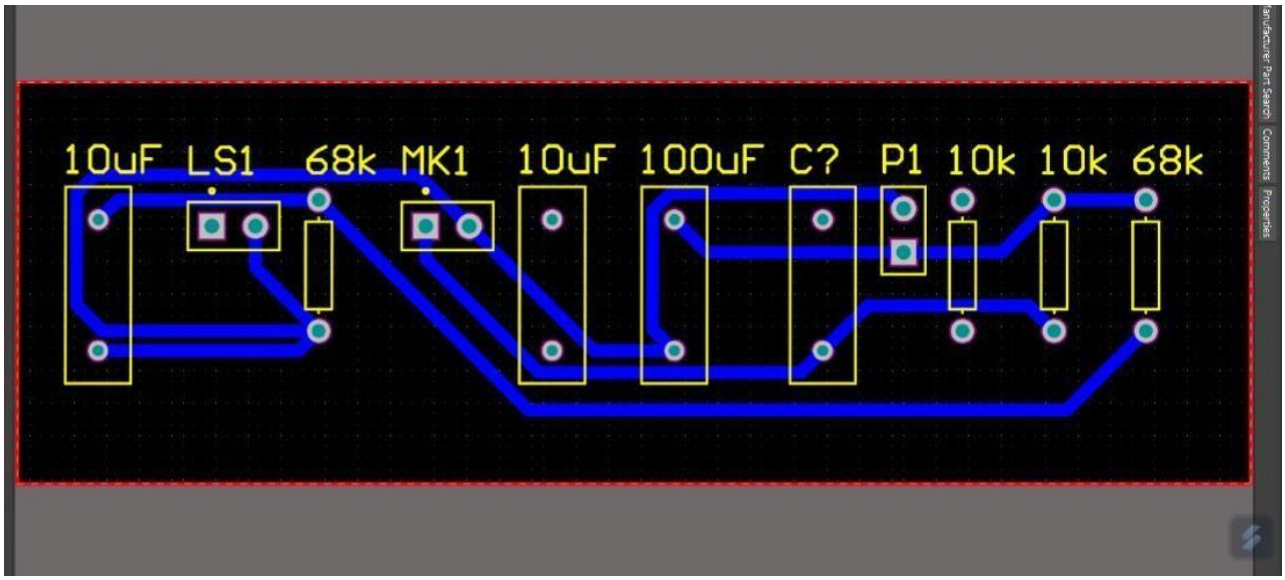

Figure 4.2: PCB layout diagram of Low noise pre amplifier circuit.

PROCEDURE:

The procedure for PCB design for a bipolar voltage regulator involves several steps. Here's a general outline of the process:

1. Schematic Design: Start by creating a schematic diagram of the bipolar voltage regulator circuit. This schematic will serve as a blueprint for the PCB layout. Include all the necessary components such as transistors, resistors, capacitors, and voltage references.
2. Component Selection: Choose appropriate components for your voltage regulator circuit based on your requirements, such as input/output voltage levels, current ratings, and power dissipation. Consider factors like package size and availability for ease of PCB layout.
3. PCB Footprint Creation: Create or select the appropriate PCB footprints for each component in your schematic. The footprint defines the physical dimensions and pin arrangement of the component on the PCB.
4. PCB Layout: Begin the PCB layout process by placing the components on the PCB. Consider factors like component orientation, signal flow, and thermal considerations. Group related components together to minimize trace lengths and optimize signal integrity.
5. Power and Ground Planes: Designate power and ground planes on the PCB. Power planes provide a low impedance path for power distribution, while ground planes serve as a stable reference for signals and minimize noise. Properly connect power and ground planes to the regulator's input and output pins.
6. Trace Routing: Route the traces on the PCB to connect the components according to the schematic and layout requirements. Pay attention to trace widths, spacing, and impedance matching for high-frequency signals. Minimize trace lengths and avoid crossing sensitive analog signals with noisy digital signals.
7. Thermal Considerations: Ensure that components generating significant heat, such as power transistors, have sufficient thermal relief and are adequately connected to heat sinks or thermal vias to dissipate heat effectively

8. Design Rule Check (DRC): Perform a design rule check to ensure that your PCB layout adheres to manufacturing and assembly constraints. Check for clearance violations, minimum trace widths, and other design rules specified by the manufacturer or your PCB design software.
9. Gerber File Generation: Once the layout is complete and error-free, generate Gerber files. These files contain the necessary information for the PCB manufacturer to fabricate your PCB.
10. PCB Prototyping and Testing: Send your Gerber files to a PCB manufacturer for fabrication. Once you receive the fabricated PCB, assemble the components and perform thorough testing to verify the functionality and performance of the bipolar voltage regulator circuit.
11. Remember, this is a general procedure, and the specific details may vary depending on the complexity of your voltage regulator circuit and the tools and software you use for PCB design. It's important to consult the datasheets of your components and follow best practices for PCB layout to ensure a successful design.

PRELAB QUESTIONS:

1. How to design a Low noise amplifier?

A low noise amplifier can be designed using either a negative feedback topology or one without any feedback. The former approach is more commonly used because it can provide better performance when compared to an equivalent amplifier without feedback.

2. What is PCB design?

PCB design is the process of creating a layout for a printed circuit board that connects and supports electronic components, facilitating their proper functioning in an electronic device.

3. What is IC NE 5534?

The NE5534, NE5534A, SA5534, and SA5534A devices are high-performance operational amplifiers combining excellent dc and ac characteristics. Some of the features include very low noise, high output-drive capability, high unity-gain and maximum-output-swing bandwidths, low distortion, and high slew rate.

4.What is Gerber diagram?

A Gerber file is a standard file format used in the PCB manufacturing industry to describe the artwork of each layer of a printed circuit board. It contains the information necessary to produce the copper traces, pads, vias, and other features on the PCB during the manufacturing process. The Gerber file acts as a blueprint for the PCB manufacturer, guiding them on how to fabricate and assemble the board accurately.

POSTLAB QUESTIONS:

1. What is the purpose of a low noise amplifier?

- To increase the signal-to-noise ratio
- To increase the gain of an amplifier
- To remove unwanted noise from an electromagnetic wave
- To attenuate the input of an amplifier.

2.What is the difference between noise and interference?

Noise is unwanted signal that can interfere with the operation of a circuit. It is usually caused by thermal agitation, voltage fluctuations, or other electrical effects. Interference is an unwanted signal that interferes with the operation of another circuit.

3. What is the advantages and disadvantages of PCB design?

ADVANTAGES:

- Compact and Space-Efficient.
- Reliable and Consistent Connections.
- Ease of Manufacturing and Assembly.
- Signal Integrity and Noise Reduction.
- Scalability and Upgradability.

DISADVANTAGES:

- Complexity.
- Cost.
- Time-consuming.

- Manufacturing Limitations.

RESULT:

Thus I have completed a prototype design for low noise preamplifier circuit, verified and tested its output successfully.

MARK ALLOCATION:

S.No	Parameters	Marks allotted	Marks awarded
1	Circuit design / Code developing and debugging / Trouble shooting	**0-3**	
2	Implementation / Demonstration	**0-3**	
3	Discussion	**0-3**	
4	Report writing and presentation	**0-3**	
5	Contribution and Team dynamics	**0-3**	
Total		**0-15**	

Signature of Lab In-charge