



The Islamic University of Gaza

Faculty of Engineering

Department of Computer Engineering

Electronics I LAB

[LAB 01]

Student Name

Ahmed Maher Al-Shaikh Khalil

Student ID

120220263

Supervisor

Eng. Nael Dokhan

Objectives

- To be familiar with electronic components.
- To be familiar with lab instruments and its functions.
- To be familiar with the LTSpice simulation.

Theory

Electronic circuits rely on fundamental components that are essential for their operation. Without these components, the circuit would be non-functional. These key elements include resistors, capacitors, diodes, integrated circuits, and others. When designing circuits, the following factors must be considered

1. Basic Electronics Components
2. Power sources
3. Measurement and analysis instruments

➤ Basic Electronics Components:

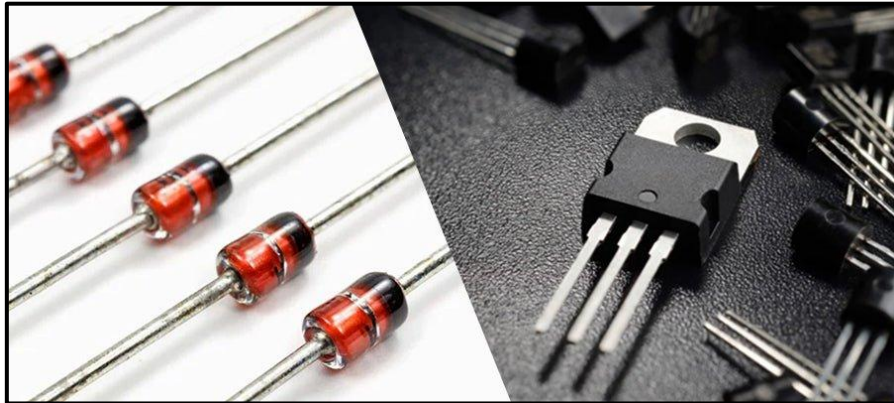
a) passive Electronic Components

These components can store or maintains energy either in the form of current or voltage. Some of these components are discussed, Like Resistors, Capacitors and Inductors.

RESISTOR	INDUCTOR	CAPACITOR
		
		
$I = V/R$	$I = \frac{1}{L} \int_{-\infty}^t V dt$	$I = C \frac{dV}{dt}$
$V = IR$	$V = L \frac{dI}{dt}$	$V = \frac{1}{C} \int_{-\infty}^t I dt$

b) Active Electronic Components

These components rely on a source of energy and are able to control the electron flow through them. Some of these components are semiconductors like diodes, transistors, integrated circuits, various displays like LCD, LED, and CRTs.



➤ Power sources:

a) The Power Supply

A power supply is an essential component in electronic systems that converts input electrical energy (AC or DC) into a stable and regulated output suitable for powering circuits. It typically consists of a transformer (for voltage adjustment), rectifier (AC-to-DC conversion), filter (to smooth voltage ripples), and regulator (to maintain constant output). Power supplies can be linear (simple but less efficient) or switching-mode (SMPS, more efficient but complex). They ensure proper voltage and current delivery while protecting devices from fluctuations, making them critical in applications ranging from consumer electronics to industrial equipment. Advances in design continue to improve efficiency, size, and reliability.



b) The Function Generator

A function generator is an electronic test instrument that produces various types of periodic waveforms, including sine, square, triangular, and sawtooth waves, with adjustable frequency, amplitude, and offset. It is widely used in electronics labs for circuit testing, signal simulation, and system troubleshooting. The generator operates by shaping and amplifying signals from an oscillator circuit, allowing precise control over waveform characteristics. Modern function generators often include digital synthesis (DDS) for high accuracy and additional features such as modulation (AM/FM/PWM) and arbitrary waveform generation. By providing stable and repeatable signals, function generators help verify circuit behavior, calibrate equipment, and analyze frequency responses in communication, audio, and control systems.



➤ **Measurement and analysis instruments:**

A Digital Multimeter (DMM) is an essential electronic measuring instrument that combines multiple measurement functions into a single device, including voltage (AC/DC), current (AC/DC), resistance, continuity, capacitance, frequency, and temperature. Unlike analog multimeters, DMMs use digital signal processing (DSP) to provide precise numerical readings on an LCD or LED display, minimizing human error. Modern DMMs incorporate features such as auto-ranging, data logging, True RMS measurement (for accurate AC readings), and overload protection to ensure safety and reliability. They are widely used in electronics, electrical engineering, and industrial maintenance for troubleshooting circuits, verifying component values, and diagnosing faults. Advanced models may also include Bluetooth connectivity for remote monitoring and high-resolution sampling for dynamic signal analysis.

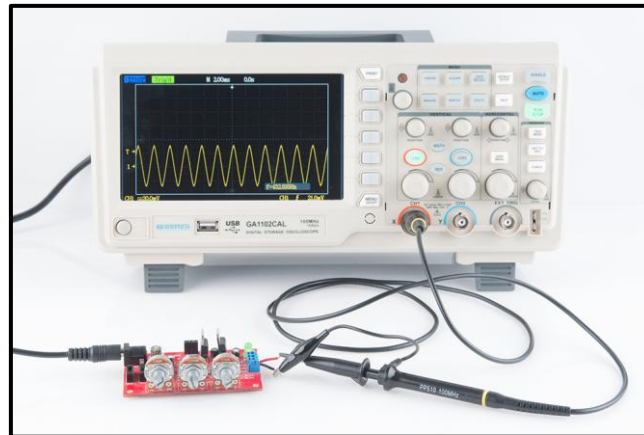


b) The Oscilloscope

An oscilloscope is a vital electronic test instrument used to visualize and analyze time-varying electrical signals in terms of voltage (Y-axis) versus time (X-axis). It captures waveforms in real-time, allowing engineers to measure key parameters such as amplitude, frequency, phase, rise/fall times, and distortion. Modern oscilloscopes are categorized into:

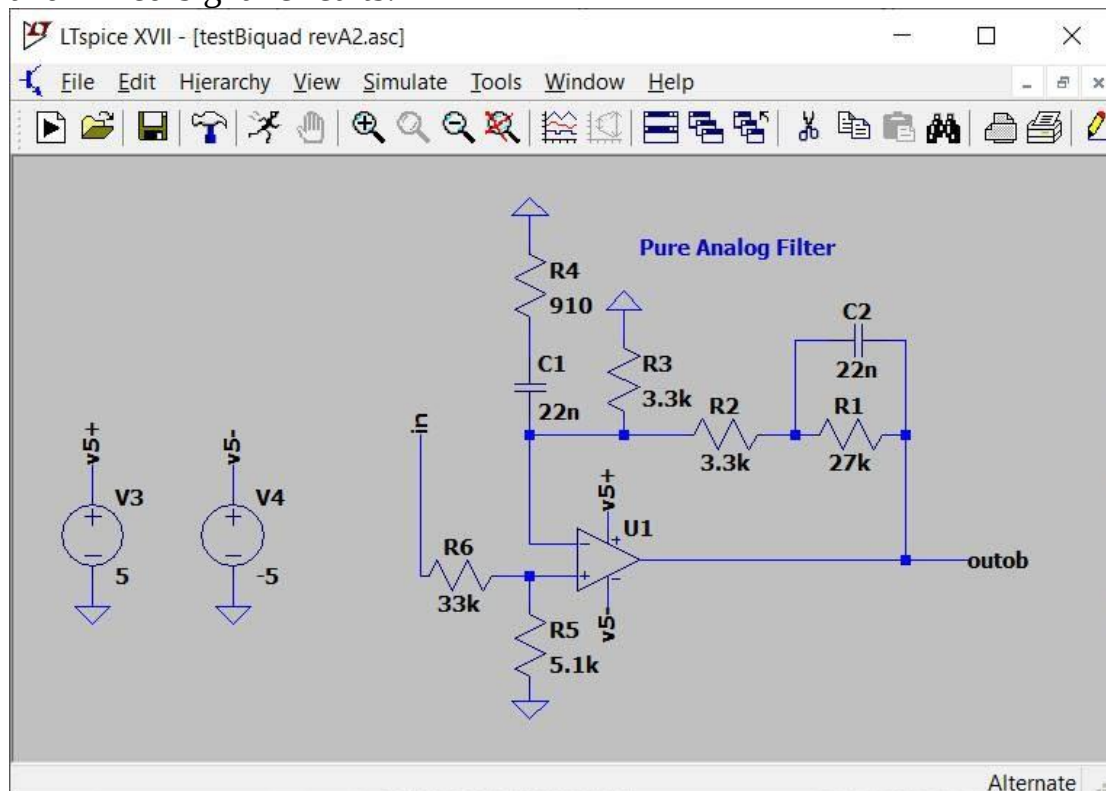
1. Analog Oscilloscopes – Use cathode-ray tubes (CRT) to display continuous signals.
2. Digital Storage Oscilloscopes (DSO) – Digitize signals using ADCs, enabling storage, zoom, and advanced triggering.

3. Mixed-Signal Oscilloscopes (MSO) – Combine analog and digital channels for embedded system debugging.



➤ LTSpice Simulation

LTSpice is a high-performance, free SPICE (Simulation Program with Integrated Circuit Emphasis) software developed by Analog Devices (formerly Linear Technology). It is widely used for circuit simulation, waveform analysis, and design verification of analog, digital, and mixed-signal circuits.



In this lab, we will learn three methods of analysis, which are:

- ✚ DC Operation Points
- ✚ Transient Analysis
- ✚ DC Sweep Analysis

DC Operation Points:

The DC operating point (also called bias point or quiescent point) is a fundamental analysis in circuit design that determines the steady-state voltages and currents in a circuit when no time-varying signals are applied. It represents the equilibrium condition of transistors, diodes, and other active devices, ensuring they operate in the desired region (e.g., saturation for MOSFETs, active mode for BJTs). SPICE-based tools like LTspice compute the DC operating point by solving Kirchhoff's laws and device equations numerically. This analysis is critical for validating biasing networks, checking power dissipation, and avoiding distortion in amplifiers before proceeding to transient or AC analyses. Without a proper DC operating point, circuits may fail to function or exhibit nonlinear behavior.

Transient Analysis:

Transient analysis is a time-domain simulation technique used to study how a circuit responds to changing signals over time, such as step inputs, pulses, or switching events. It solves the circuit's differential equations numerically to compute voltages and currents at each time step, capturing dynamic behavior like rise/fall times, settling periods, and oscillations.

This analysis is essential for evaluating:

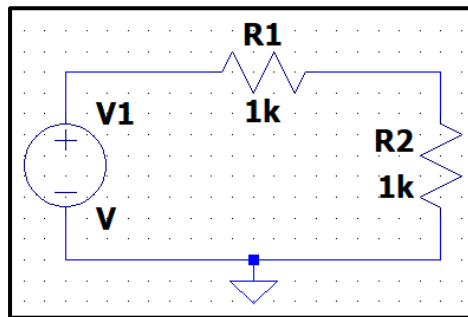
- Switching circuits (e.g., power converters, logic gates),
- Transient responses (e.g., RC/RL time constants),
- Distortion in amplifiers,
- Start-up behavior of power supplies.

In tools like LTspice, users define simulation duration (e.g., 0 to 10ms) and time steps (e.g., 1 μ s). Transient analysis complements DC operating point and AC analyses by revealing real-world signal interactions and nonlinear effects.

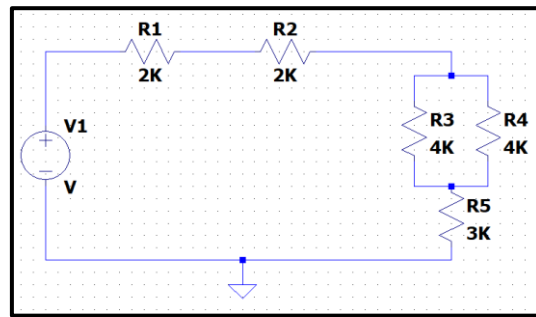
DC Sweep Analysis:

DC Sweep Analysis is a simulation method used to study how a circuit's voltages and currents respond to gradual changes in a DC input voltage or current source. It systematically varies a specified parameter (e.g., power supply voltage, resistor value, or transistor bias) over a defined range and records the resulting steady-state operating points.

We will apply these methods to the two circuits shown for us.



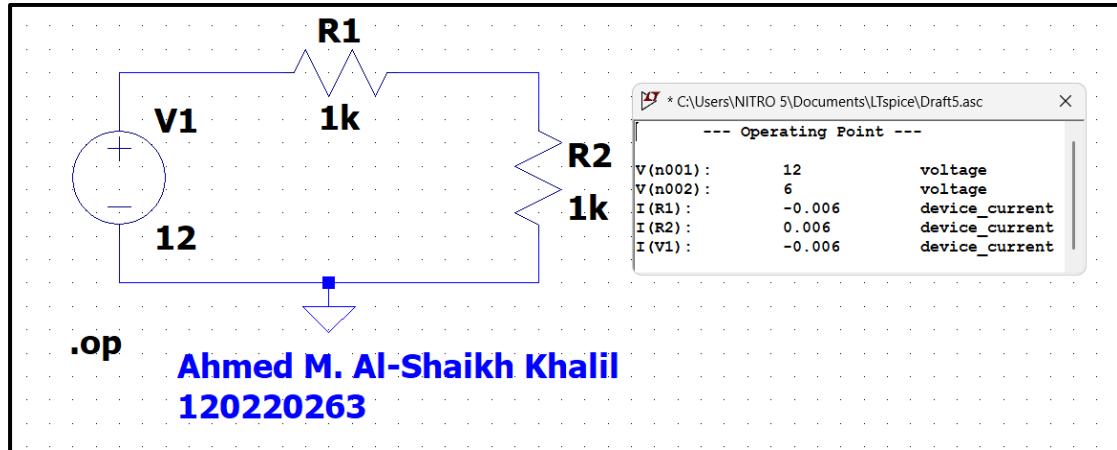
Circuit 1



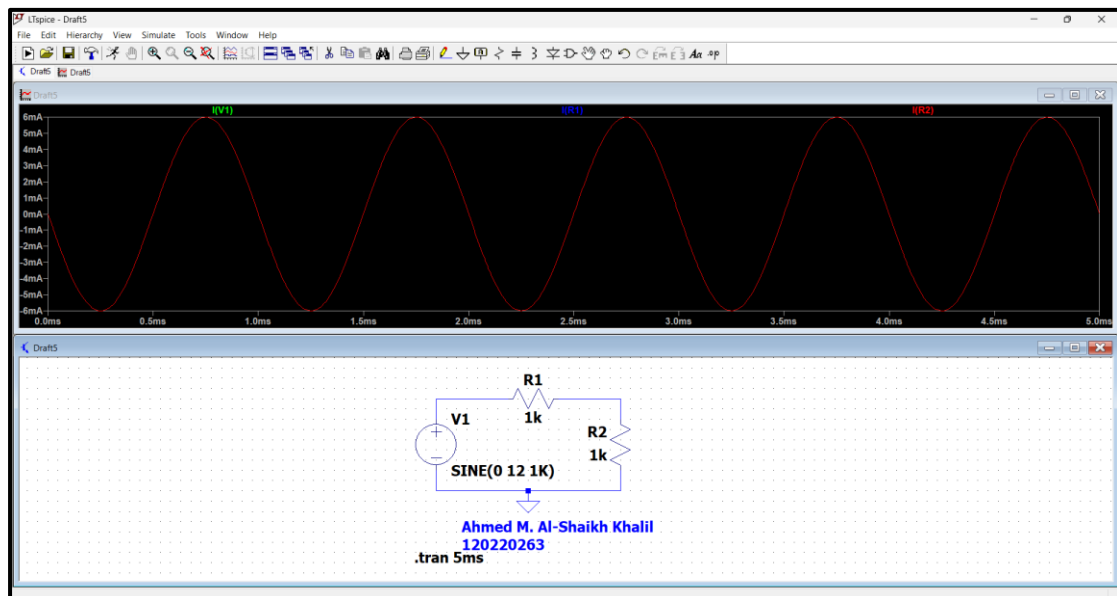
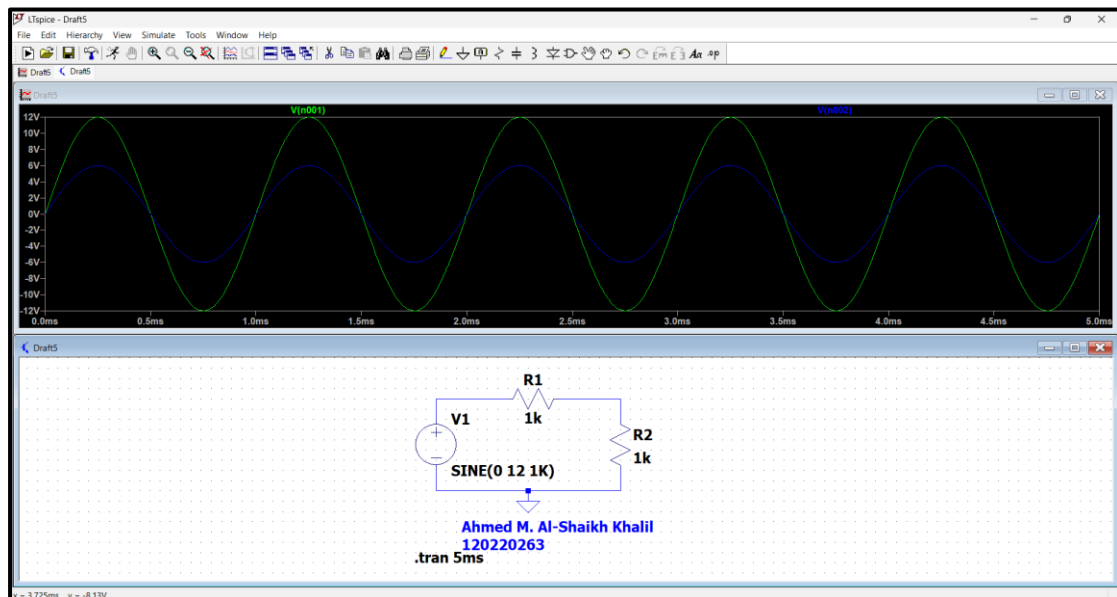
Circuit 2

Circuit 1:

DC Operation Points

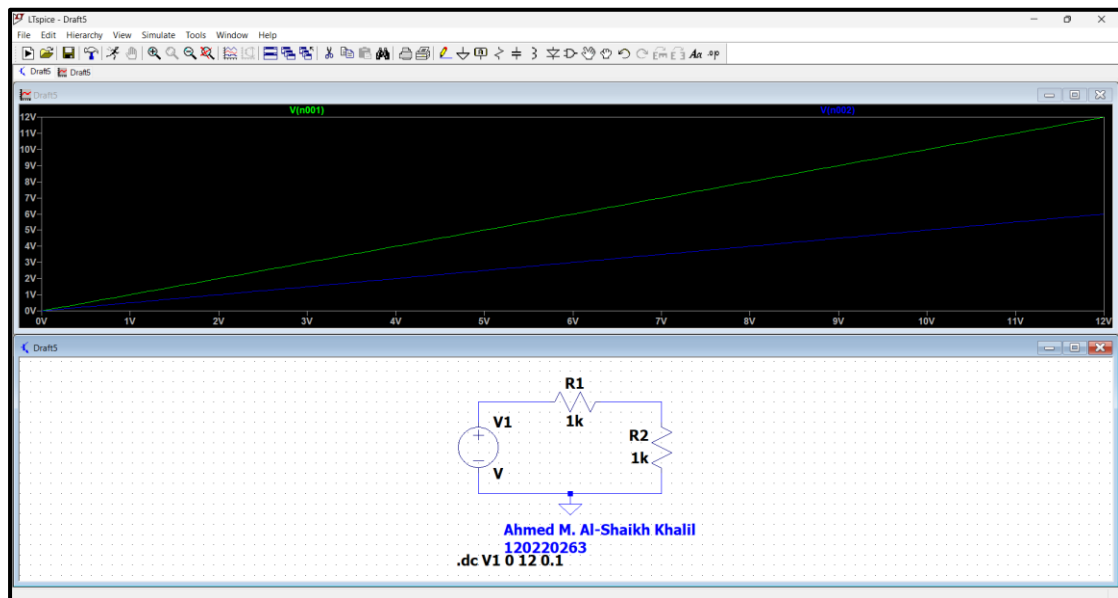
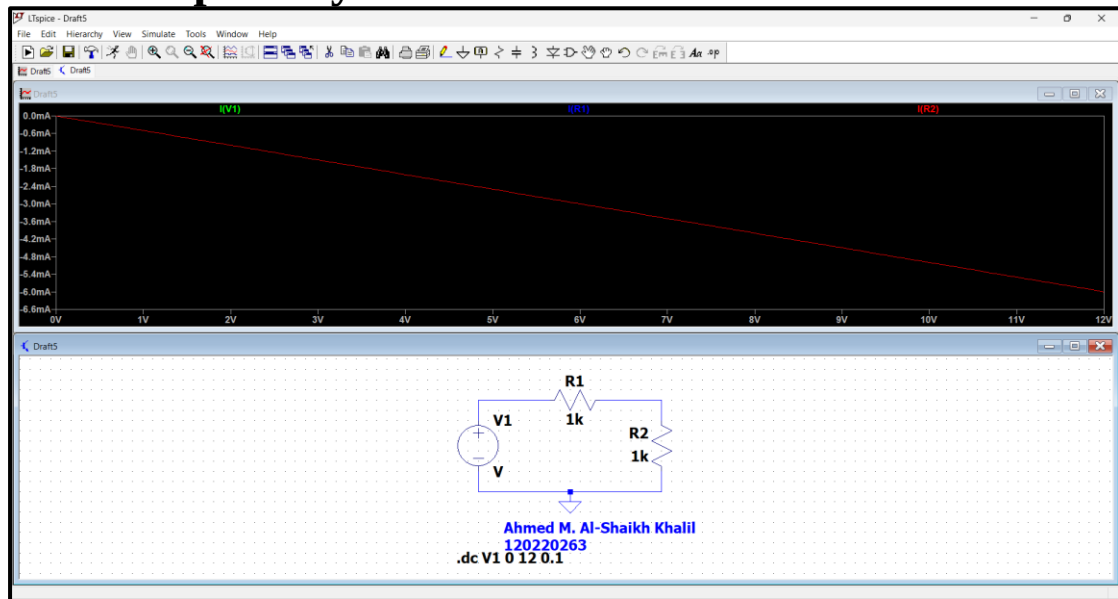


Transient Analysis



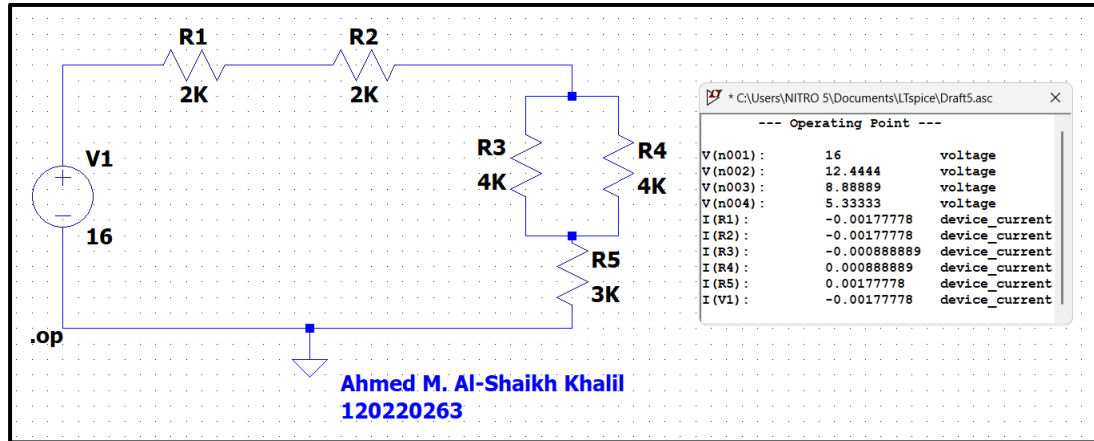
$$I_{R1} = I_{R2} = I_{V1}$$

DC Sweep Analysis

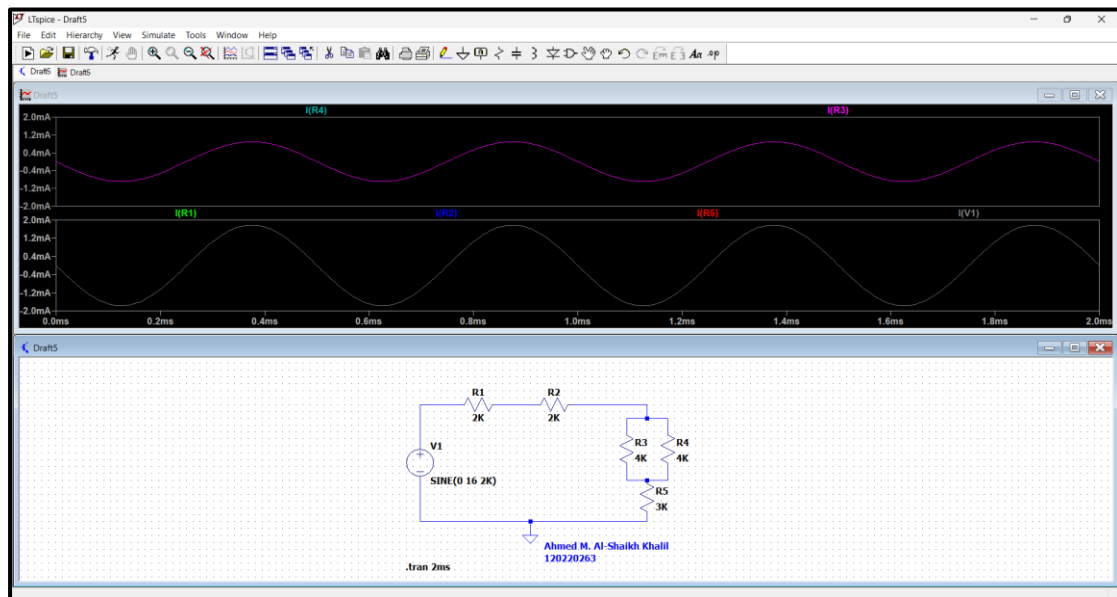


Circuit 2:

DC Operation Points

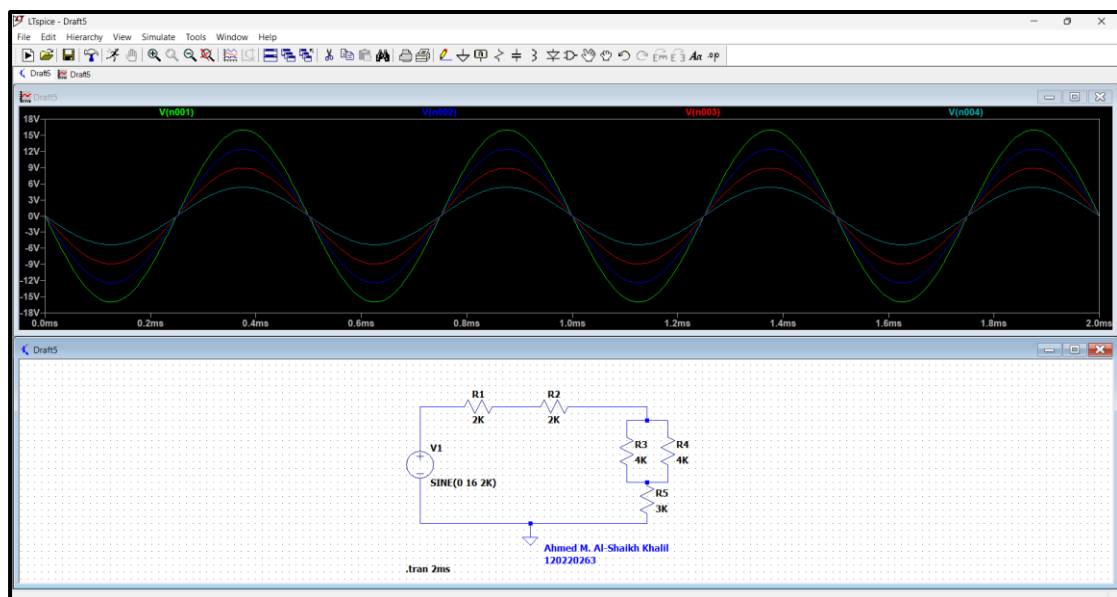


Transient Analysis



$$I_{R1} = I_{R2} = I_{R5} = I_{V1}$$

$$I_{R3} = I_{R4}$$



DC Sweep Analysis

