

# Physics 623

## Analog and Digital Simulation

April 28, 2024

The simulations will be done using the commercial SPICE software, National Instruments *MultiSim 12.0*, which is installed on the laboratory computers. With some preparation before the lab, you should be able to do both an analog and a digital simulation. The simulations run the circuits in software and all voltages, currents, and waveforms can be examined and printed out.

The circuits are entered by schematic capture, i.e. you draw the circuit on the computer screen by placing the parts and connecting them with wires. Once the circuit is entered, the simulation is run or an analysis is performed.

### Part 1– Analog Simulation

The analog simulation circuit will be the difference amplifier from the difference amplifier lab. The simulated circuit should have the same component values used in the lab circuit.

There are various type of analyses that can be performed, including DC operating point, Transient Analysis, and AC analysis. A large variety of instruments such as Multimeters, Function Generators, Oscilloscopes, and Bode plotters may be connected to various points in the circuit for obtaining information. However, you don't actually need to use any of these, since transient and AC analysis results can be plotted in the grapher for any point in the circuit - see the simulation hints.

You will get points for having plots of your results that clearly show the important points and are easy to read. You can adjust essentially everything about a plot within the grapher, as described in the simulation hints. **Be sure to have grids and a white background on all plots.**

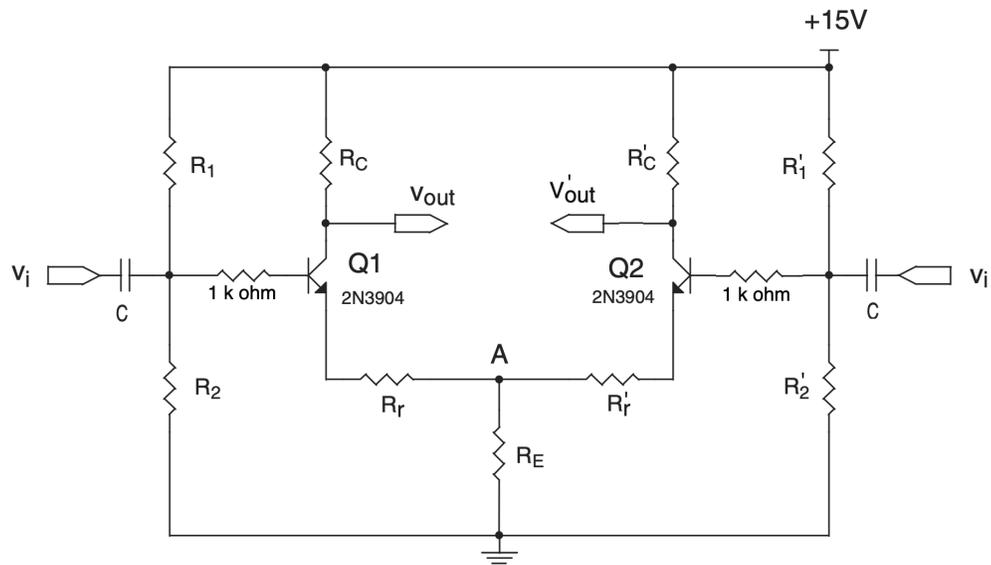


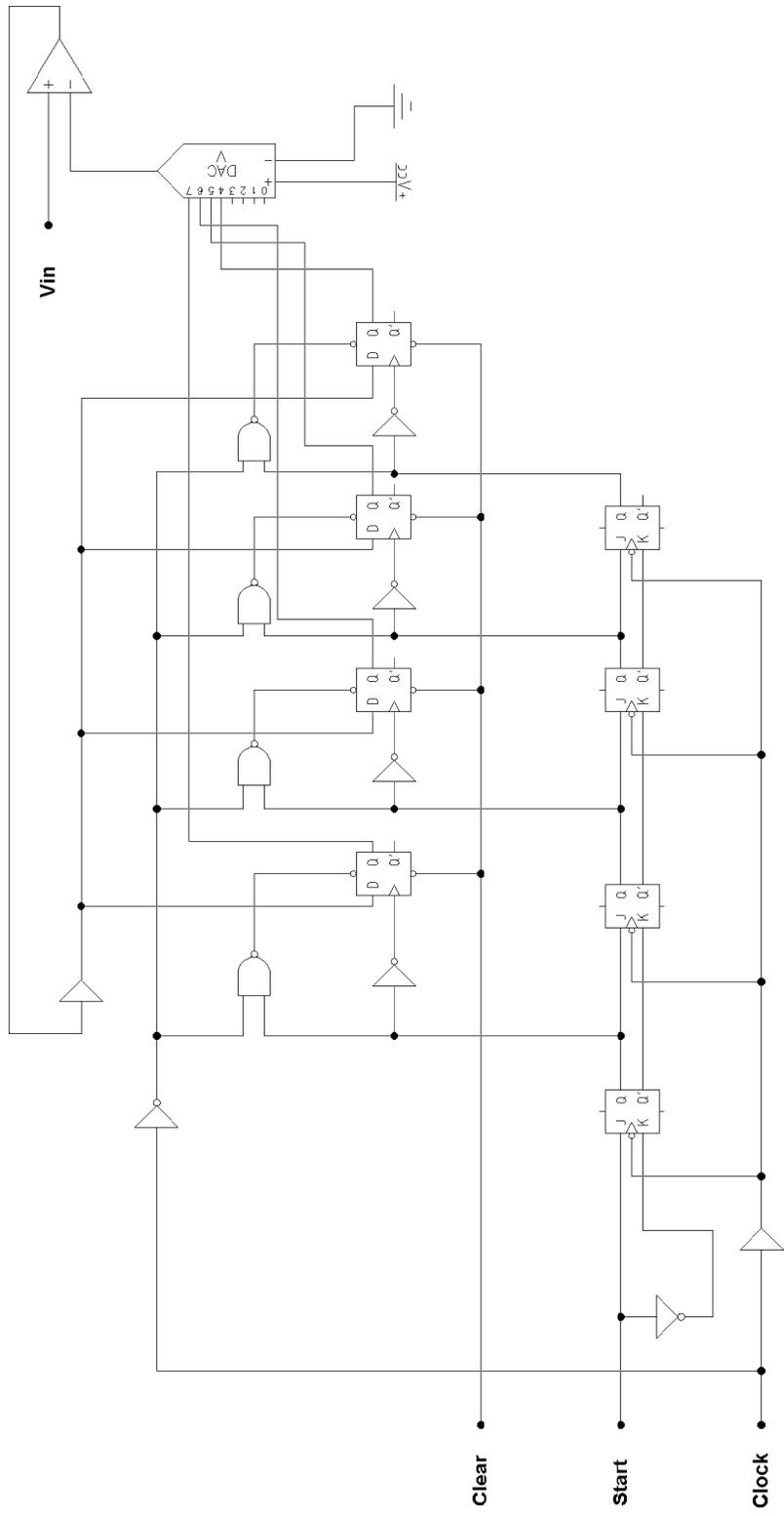
Figure 1: Schematic of the Difference Amplifier

1. Simulate the difference amplifier circuit using the circuit you built for the Difference Amplifier lab. The component values should be identical to those used in the lab. We use the *MultiSim 12.0* application.
2. The first step is to make a schematic diagram. Print a copy for your notebook. (“File” → “print options” → “print sheet setup”, uncheck “in black/white” and set zooms: 75% is about right if your schematic fills the sheet. Send to color printer.)
3. Find the DC operating point of the circuit. Present the results as a three column table: theory and measurement (from your diffamp lab), and the simulated values. Should the latter be the same as the theoretical ones? You blamed differences in your measured values on resistor tolerances, but that’s not a factor here. What assumptions were made in your original calculations that could be causing any differences you see?
4. Perform a Transient analysis for about five cycles. Use the same frequency as you used in the lab. Print out the result for common mode and differential mode inputs. Compute the differential and common mode gains and compare them with the calculated values from your diffamp lab. Then increase the differential input amplitude to cause a moderate amount of saturation of the output and include a plot of that.
5. Perform an AC frequency analysis and determine the low and high rolloff frequencies. Put the Bode plot in your notebook.
6. Most people did not measure the frequency dependence in lab, but you should be able to calculate the expected lower 3 dB point. The upper ones will depend on the stray capacitances in the transistor and the source impedance of your base circuit. See if you can estimate them:  $C_{CB}$  and  $C_{BE}$  are both about 3 pf for the 2N3904. If you use generic (ideal) transistors instead of the 2N3904s, there won’t be any high-frequency drop-offs.

## Part 2– Digital Simulation

You will simulate a 4 bit Successive Approximation ADC made up of a DAC, comparator, D flip flops, J-K flip flops, and NAND gates. You will enter a DC input voltage, run the simulation, and read out the digital state of the circuit under the control of CLK pulses. The circuit also requires START and CLEAR inputs. All three can be generated in the proper sequence using the Word Generator.

1. Enter the circuit via schematic capture. Consult the Multisim Hints sheet for details. You can ask the instructor for a file with the completed schematic if you're running out of time. In any case put a copy of the schematic in your notebook.
2. The initial conditions for the various lines are entered using the Word Generator tool, which allows you to enter the CLK, START and CLEAR pulses and run them as a sequence either step by step or as an automatic sequence at a given frequency. See the multisim hints for details.
3. Use the Logic Analyzer tool to display the circuit output. This allows you to display the timing diagram for up to 16 digital signals. The analog output and  $V_{in}$  can be viewed on the 2-channel Oscilloscope instrument. The DAC output is  $V_{out} = V_{ref} \times D/256$  where  $V_{ref} = 5\text{ V}$ . The binary outputs can also be viewed with LEDs which light up on a logical "1" if you've added them to your schematic.
4. Include a plot of the DAC output and  $V_{in}$  from the oscilloscope along with the logic analyzer output. You should see the DAC output converge to the analog input voltage with an accuracy consistent with the 4 bits of the circuit, i.e. one part in sixteen. The digital result resides in the four D flip flops. Try several different DC input voltages, but you only need to put the plots for one in your notebook.



**Successive Approximation ADC**