EE1002 Introduction to Circuits and Systems

Lab 1 Briefing:

LTSpice Familiarization

Things to note in lab…

- Dress properly – Covered footwear is compulsory

- Sign your attendance

- Switch off all the equipment and tidy up your work area before you leave

- Submit your safety quiz to our lab officer.

Assessment

- Lab 1-5 (Week 3-7): 2% each

- Lab Test (Week 7): 10%

- Project (Week 8-13): 30%

- Total Lab Work: 50%

Lab-1 Activities

- **Simulation and Tools Demo in Parallel**
  - Tables 1-12 – LTSpice
  - Tables 13-24 – Tools Demo by Dingjuan
  - Swap

- **Demonstrate** your simulation results to any GA after you finish **every question** and get him/her to sign on your lab sheet after verification.

- **Note/Sketch** down the results/answers on your lab sheet.

- **Install** LTSpice

Brief Introduction to SPICE

- **_S_**imulation **_P_**rogram for **_I_**ntegrated **_C_**ircuits **_E_**mphasis
- Circuit Simulator Software ➔ Simulates (Predicts) the behavior of electronic circuits

- How it works :
  - Schematics are converted to netlists
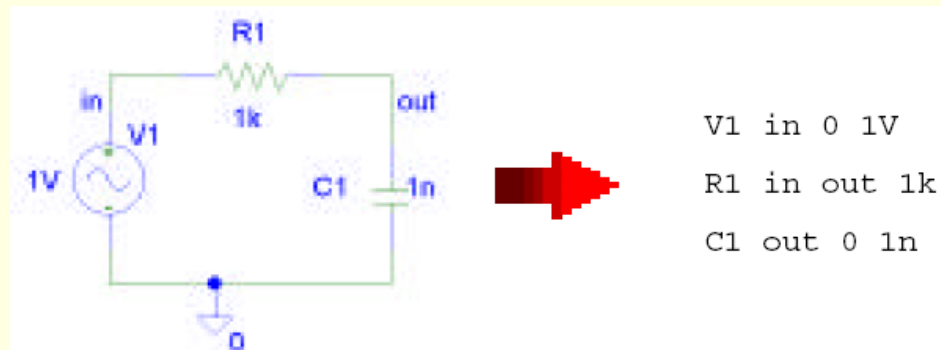
  - Using known models, (e.g. V=IR, KCL, KVL) netlists are converted to matrices which are used to solve for the solutions of the circuit.
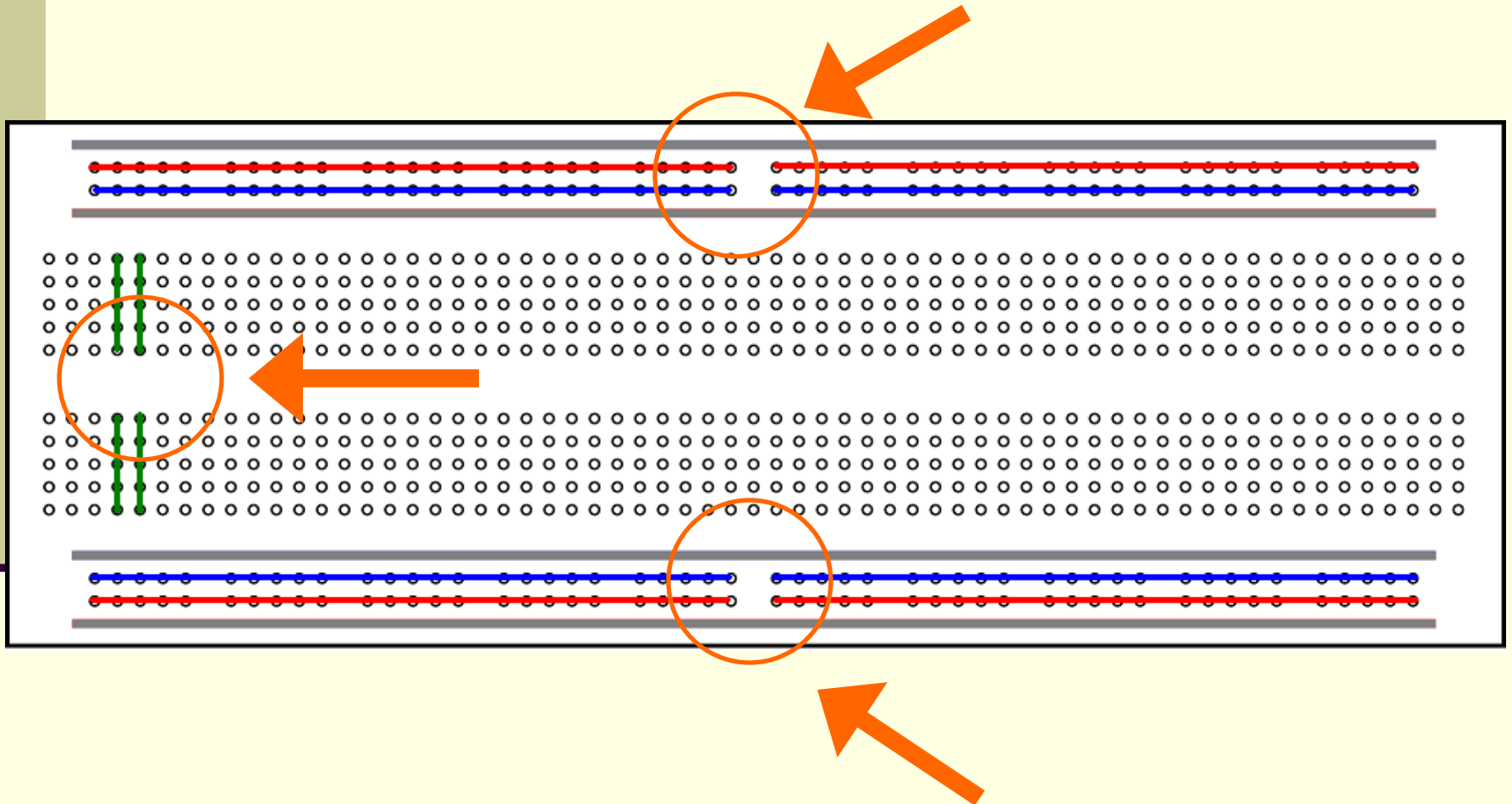
What can SPICE do?

- **DC Voltages / Currents**
- **DC Sweeps, Component value sweeps**
- **Transient Response**

- Frequency response; Bode Plots
- DC bias point/small signal parameters
- Temperature analysis
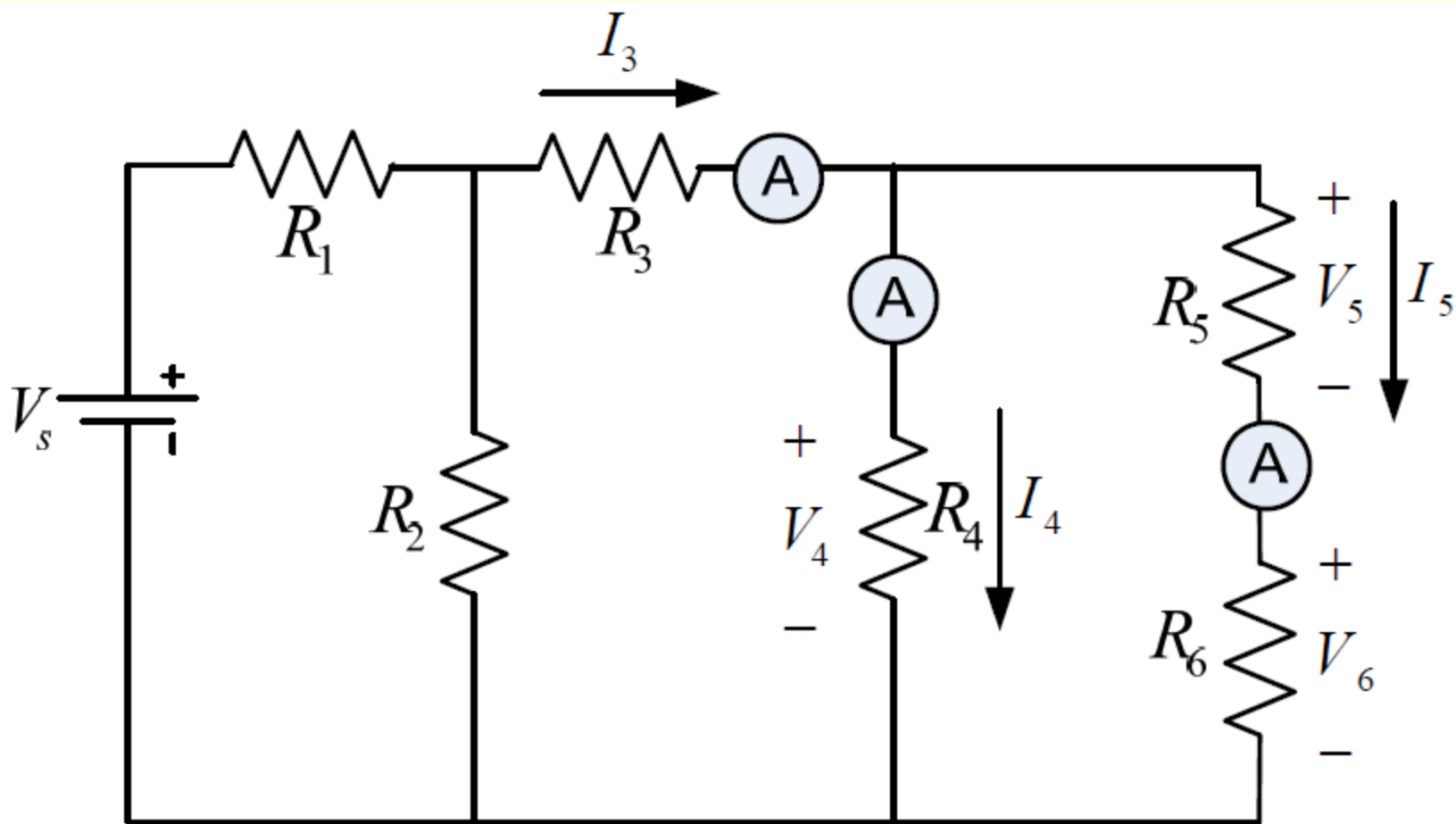- Monte Carlo (for component variations)
- Noise Analysis

Enjoy your Lab!

Bread-Board Connections

Circuit 1

Circuit 2