ELC1010

Electric Circuits (I) – Circuit Theory

Introduction to Circuit Simulators

Dr. Omar Bakry

omar.bakry.eece@cu.edu.eg

Department of Electronics and Electrical Communications

Faculty of Engineering

Cairo University

Fall 2023

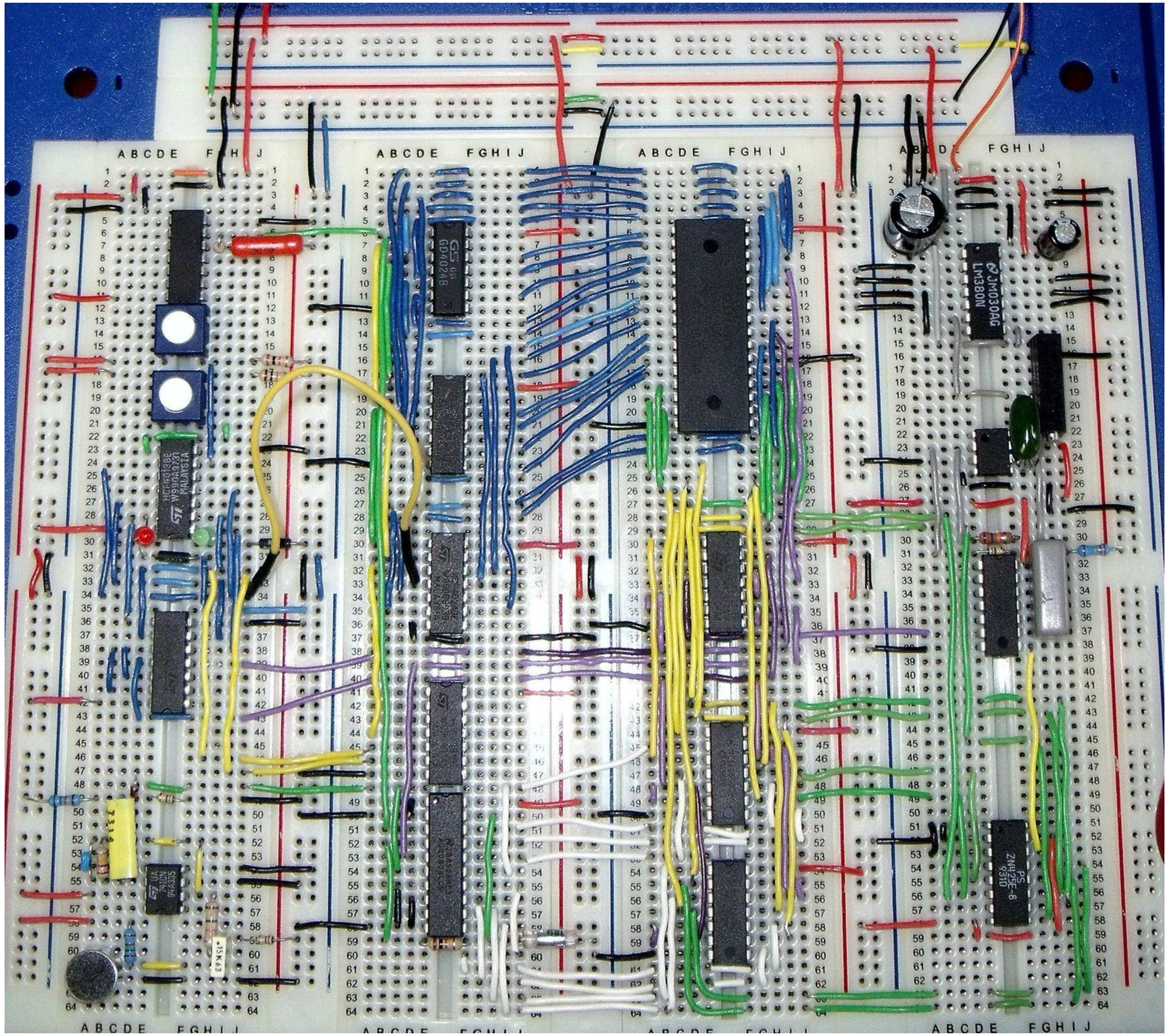
Acknowledgment: Dr. Ahmed Farghaly

Outline

- Introduction to Circuit Simulation and Basic Concepts
- Roles and Functions of Circuit Simulators
- Circuit Design Flow
- Popular Circuit Simulators
- SPICE Simulator: An In-Depth Look
- Netlist
- Demonstration: Set up and run a basic circuit (Hands-On Example)
- Monte Carlo Simulation
- The Future of Circuit Simulators
- Conclusion

Expectations vs Reality

- Circuit is working properly ☺
- Circuit is not working at all ☹
- Circuit is not working as expected ☹
- I need to try another circuit ☹
- Capacitor exploded ☹
- Resistor burned out ☹
- I got a wrong value for an inductor ☹
- I spent 10 hours building the circuit ☹



Purpose of Circuit Simulations

- Provide engineers and designers with a powerful tool to design, analyze, and test electronic circuits without the need for constructing physical prototypes.
- Here's how circuit simulation fulfills this purpose:
  - **Virtual Environment for Circuit Design:**
    - Circuit simulation software offers a virtual environment where engineers can design and assemble electronic circuits.
    - Users can select and place various electronic components, such as resistors, capacitors, transistors, and integrated circuits, into a virtual workspace.
  - **Analysis and Testing:**
    - Once the circuit is designed, the simulation software can be used to analyze its behavior.
    - It allows testing under various conditions (excitations, initial conditions, temperature, component tolerance, …etc.) without the need to physically build the circuit each time.
    - Simulations can include different types of analyses like DC, AC, transient, and more, to understand how a circuit responds over time or to different frequencies.

Purpose of Circuit Simulations

- **Time and Cost Efficiency:**
  - Simulation saves time and resources as it eliminates the need for building multiple prototypes for testing.
  - It helps in identifying and rectifying errors at an early stage, reducing the cost and time involved in rework.
- **Safety and Feasibility:**
  - Testing circuits with high voltages or currents can be dangerous. Simulation allows for safe testing of such circuits.
  - Simulations can test the feasibility of circuits that would be challenging or impractical to physically create.
- **Iterative Design Process:**
  - Engineers can quickly make adjustments to the virtual circuit and immediately see the effects of those changes.
  - This iterative process leads to optimization and refinement of the circuit design before any physical model is built.

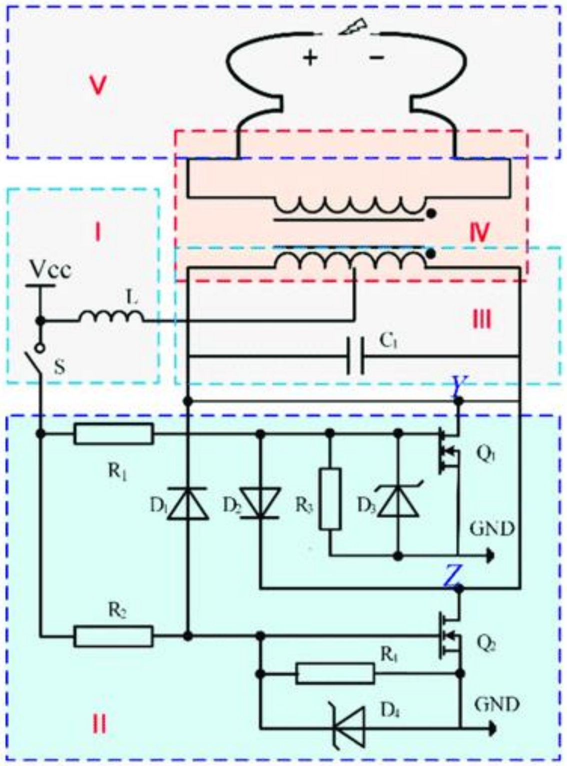
Purpose of Circuit Simulations

- **Realistic and Accurate Results:**
  - Modern simulation software is capable of providing highly accurate and realistic results, closely mimicking the behavior of actual circuits.
  - These tools take into account various real-world factors like temperature effects, component tolerances, and more.
- **Educational and Experimental Tool:**
  - For students and beginners, circuit simulation is an invaluable educational tool, providing a risk-free platform to learn and experiment.
  - It allows for exploration and understanding of complex circuitry without the need for advanced lab setups.

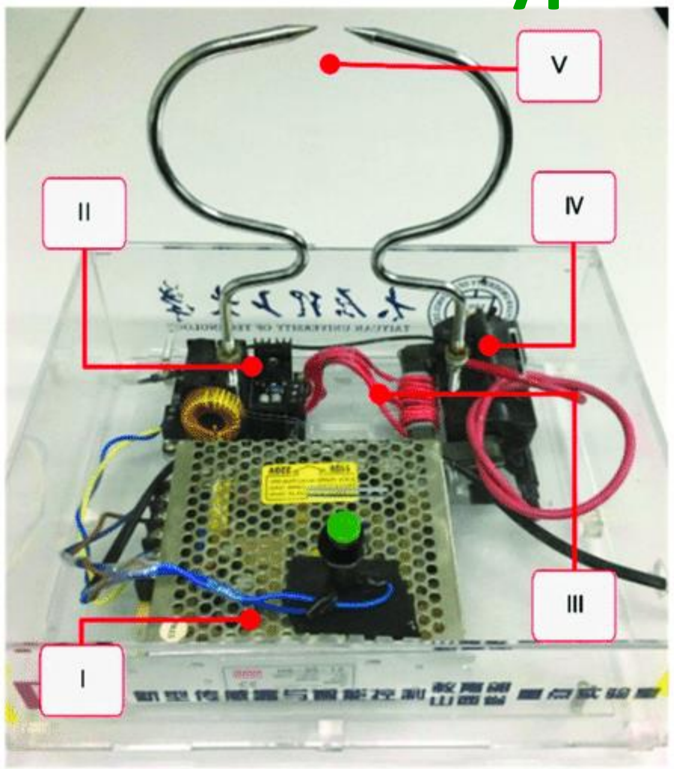
Purpose of Circuit Simulations

- Circuit simulation is an essential aspect of electrical engineering, enabling efficient, safe, and cost-effective design and testing of electronic circuits, which accelerates the development process and enhances the learning experience.

Circuit Schematic



Circuit Prototype



Source: https://www.researchgate.net/figure/PD-simulator-device-a-circuit-diagram-and-b-physical-diagram-in-the-laboratory-Part_fig2_324988716

Roles of Circuit Simulators

- **Design Validation and Optimization:**
  - Circuit simulators allow engineers to validate the design of electronic circuits, ensuring they function as intended before physical prototypes are built.

- **Performance Analysis:**
  - They are used to analyze the performance of circuits under various conditions, including extreme environments that might be difficult or impossible to replicate physically.

- **Fault and Safety Analysis:**
  - Simulators help in identifying potential faults and safety issues, enabling designers to make necessary adjustments early in the design process.

Functions of Circuit Simulators

- **Modeling Components:**
  - Circuit simulators contain libraries of component models that represent real-world electronic components like resistors, capacitors, transistors, and integrated circuits. These models are mathematical representations that mimic the actual behavior of the components.
- **Setting Up Circuit Parameters:**
  - Engineers can set up and modify circuit parameters within the simulator. This includes applying voltages and currents, changing component values, and configuring the circuit topology.
- **Running Simulations:**
  - Once the circuit is set up, the simulator runs calculations based on the laws of physics and electronics, such as Ohm's Law, Kirchhoff's Laws, and others. It calculates the response of the circuit to various inputs over time.

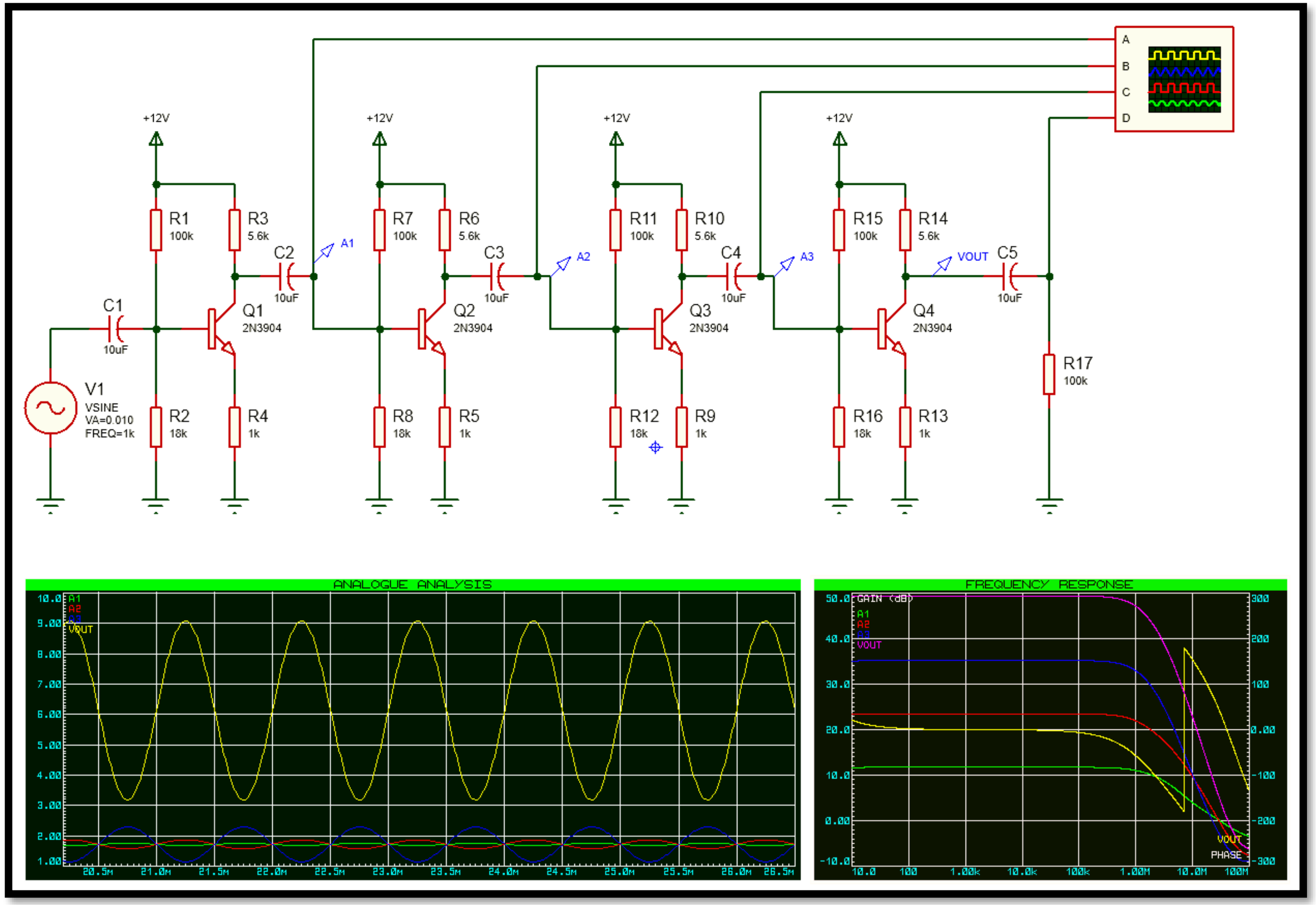
Simulation Types

- Circuit simulators are software tools used by engineers and designers to replicate and analyze the behavior of electronic devices and circuits.

1. **DC Analysis:** Determines the steady-state behavior when a constant voltage or current is applied.
2. **AC Analysis:** Evaluates the response to sinusoidal inputs, useful for understanding frequency response.
3. **Transient Analysis:** Shows how the circuit responds over time to changing inputs, essential for understanding dynamic behavior.
4. **Noise Analysis:** Assesses the impact of noise on circuit performance, crucial for sensitive electronic applications.
5. Advanced simulation analysis can also be performed: SP/HB/PSS/ENV/STB/…etc.

Visual Output and Analysis

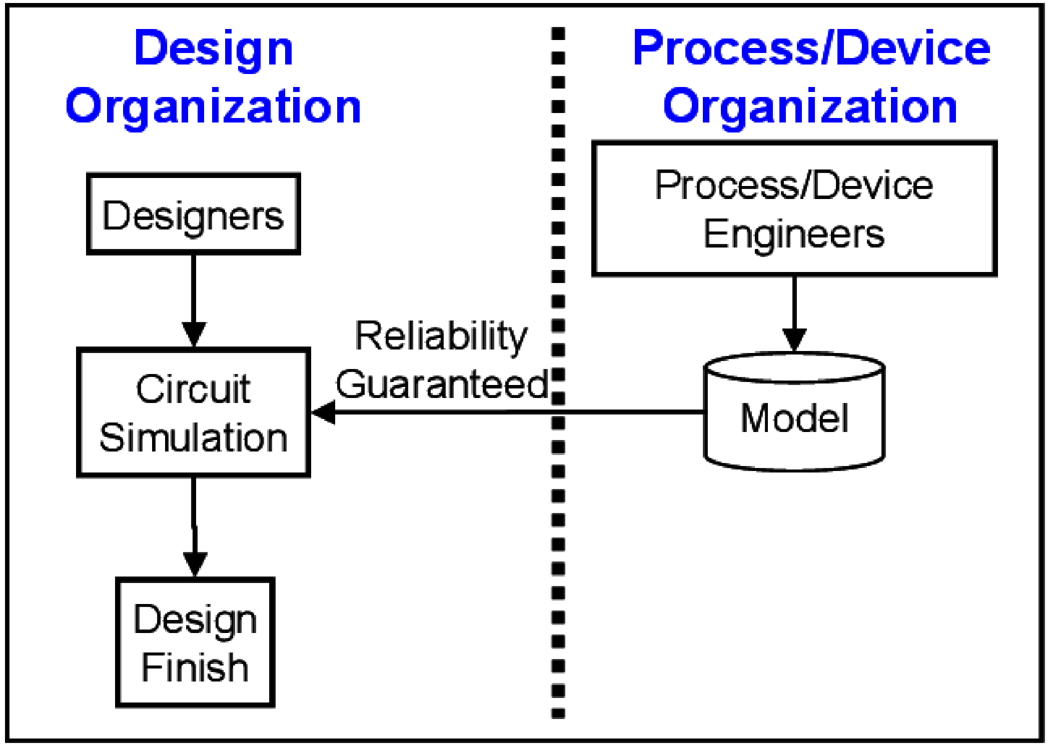
- Circuit simulators provide visual outputs such as waveforms, signal spectra, and other graphical representations. This helps engineers in analyzing the behavior of the circuit under different scenarios.



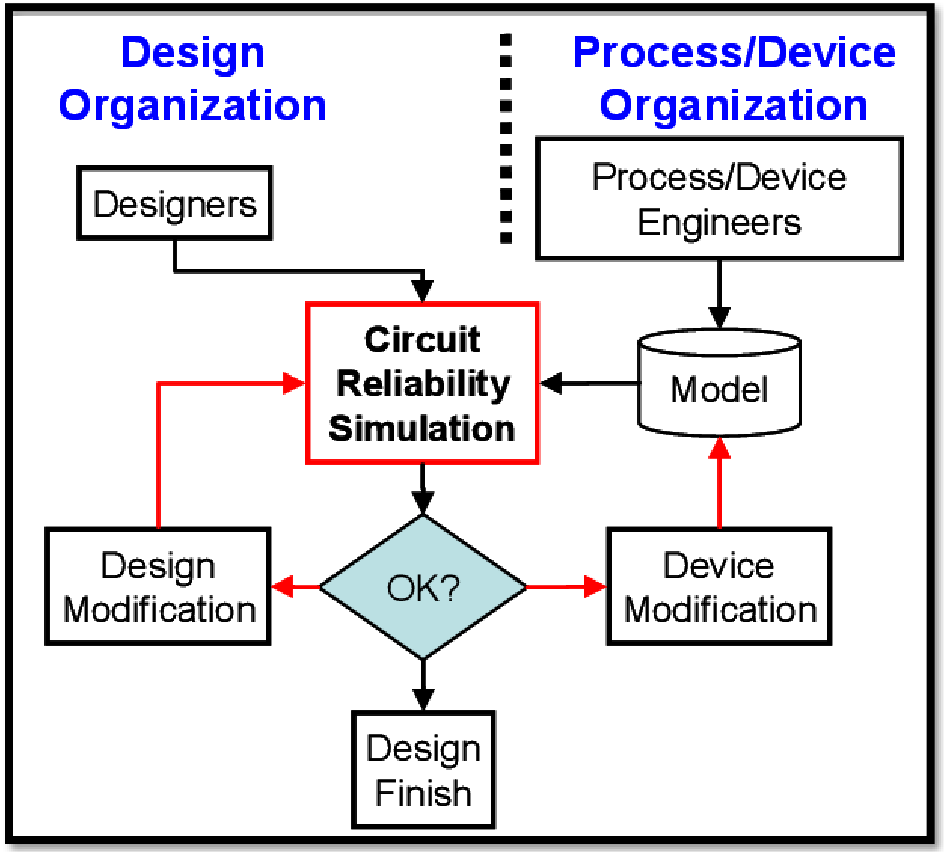
Note:
Simulators usually offer you the option to show the results in table forms and save the output signals as CSV files

Circuit Design Flow

Models are constantly updated and released based on the available Model – Hardware Correlation Data



Design flow when designers and process/device engineers can work separately



Design flow when designers and process/device engineers needs to work together to find a holistic optimized solution

Source: https://www.semanticscholar.org/paper/Compact-modeling-for-simulation-of-circuit-and-Lee/f5a444a6121eee1427f736719e5b046fb0aea83f

Popular Circuit Simulators

- **SPICE (Simulation Program with Integrated Circuit Emphasis)**
  - Overview: Originally developed at UC Berkeley, SPICE is one of the most widely used circuit simulation programs. It is an open-source software that laid the foundation for many other simulators.
  - Key Features: Offers DC, AC, and transient analyses, and has a vast library of electronic components. It’s known for its accuracy and is used primarily for analog circuits.
- **LTspice**
  - Overview: Developed by Linear Technology (now part of Analog Devices), LTspice is a derivative of SPICE with enhancements for faster simulation of switching regulators.
  - Key Features: It’s free, user-friendly, and includes a library of Linear Technology components. It’s popular for power supply and integrated circuit designs.

Popular Circuit Simulators

- **PSpice (Cadence)**
  - Overview: Originating as a derivative of SPICE, PSpice has become a standard in the industry, particularly in the realm of PCB design and simulation.
  - Key Features: Known for its robustness and extensive model libraries, PSpice includes features like MonteCarlo analysis and parametric plots.
- **HSPICE (Synopsys)**
  - Overview: HSPICE is another variant of SPICE, designed for both analog and digital circuits. It is widely used in the semiconductor industry.
  - Key Features: Known for its accuracy in simulating complex circuits, HSPICE is commonly used for IC design, especially in high-frequency applications.

Popular Circuit Simulators

- **Multisim (National Instruments)**
  - Overview: Known for its interactive interface, Multisim combines SPICE simulation with schematic capture.
  - Key Features: It’s ideal for education and research. It offers easy-to-use tools for circuit design and analysis, making it popular in academia.

- **Cadence Virtuoso**
  - Overview: Part of the Cadence Design Systems suite, Virtuoso is a high-end tool mainly used for designing integrated circuits.
  - Key Features: : It provides a comprehensive environment for schematic entry, layout design, and verification. Its advanced simulation capabilities make it a go-to tool for professional IC designers.

Popular Circuit Simulators

- **ADS (Advanced Design System)**
  - <u>Overview:</u> Developed by Keysight Technologies, ADS is widely used for RF, microwave, and high-frequency circuit design.
  - <u>Key Features:</u> It offers integrated design guidance and is popular for its capabilities in signal integrity and RF system design. It also offers the most powerful optimization tools for designers.

- **Circuit Lab (Simulation and Schematics).**
  - <u>Overview:</u> Build and simulate circuits right in your browser.
  - <u>Key Features:</u> easy, free, online, open source.

- **And much more !!**

Integration Between Circuit Simulators

- Each of the circuit simulators has its own set of strengths and challenges.
- Specific simulator is chosen based on:
  - Specific requirements of the project, such as the complexity of the circuit, the type of analysis needed.
  - The user's familiarity with the tool.
  - Online documentation and tool customer support.
- Typically, designers need to utilize the various features of different simulators to complete their design.
- Integration with Matlab for advanced computational capabilities
- Integration with Electromagnetic Simulators (HFSS, CST, Analyst, … etc.) for passives design
- Coding and scripting skillset for designers → design automation, optimization, signal processing, and data analysis and visualization.

Types of Solvers

- Generally, mathematical problems can be solved analytically and numerically.
- An analytical solution involves framing the problem in a well-understood form and calculating the exact solution.
- A numerical solution means making guesses at the solution and testing whether the problem is solved well enough to stop.
- We prefer the analytical method in general because it is faster and because the solution is exact. Nevertheless, sometimes we must resort to a numerical method due to limitations of time or hardware capacity.

SPICE Simulator: An In-Depth Look

History and Development of SPICE (Simulation Program with Integrated Circuit Emphasis)

- Origin:
  - SPICE was developed in the early 1970s at the University of California, Berkeley.
  - It was created by Laurence Nagel under the guidance of Professor Donald Pederson.
- Evolution:
  - Initially developed as a class project, SPICE quickly evolved.
  - By 1973, SPICE2 had been released, offering improved algorithms and models.
  - In the 1980s, SPICE3 added capabilities for handling more complex circuits and different types of analyses.
- Impact on Industry:
  - SPICE set the standard for circuit simulation and has been the foundation upon which many commercial and open-source simulators are built.
  - It revolutionized the way electronic circuits are designed and tested.

SPICE Simulator: An In-Depth Look

- Component Modeling:
  - SPICE can simulate a wide range of components, from simple resistors and capacitors to complex integrated circuits.
  - The models used in SPICE range from basic to very sophisticated, encompassing nonlinear and time-dependent effects.
- Numerical Methods and Algorithms:
  - SPICE uses various numerical methods for solving circuit equations, including Newton-Raphson iteration for nonlinear equations and Gear's method for stiff differential equations.
  - This allows for accurate and efficient simulation of circuits.
- User-Friendly Interface:
  - While the original SPICE was command-line-based, modern incarnations often feature graphical interfaces, making them more accessible to a broader range of users.

SPICE Simulator: An In-Depth Look

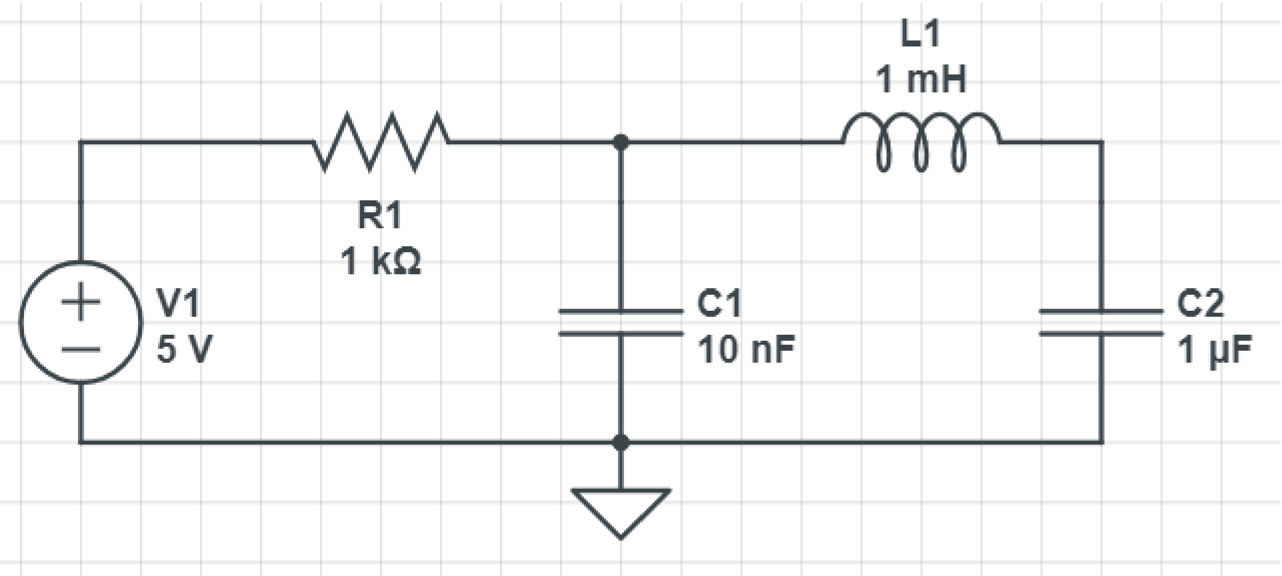
- Extensions and Variants:
  - Various versions of SPICE exist, each offering unique features.
  - Some focus on specific types of circuits, such as digital or power electronics, while others enhance usability or simulation speed.
- Educational and Professional Use:
  - SPICE is widely used in both educational settings for teaching circuit design and analysis.
  - It is also used professionally for circuit design and verification in the electronics industry.
- SPICE's development marked a significant milestone in electronic design automation (EDA), and its legacy continues to influence the field of circuit simulation. Its comprehensive analysis capabilities and ongoing adaptations ensure its relevance in both academic and industrial applications.

Netlist of a Schematic: Overview and Syntax

- Definition:
  - A netlist is a description of an electronic circuit in terms of a list of the components and the nodes they are connected to.
  - It is a textual representation of the circuit schematic.
- Purpose:
  - Netlists are used by circuit simulation programs like SPICE to understand and analyze the circuit without needing a physical diagram.
- Syntax:
  - Basic Structure:
    - The syntax typically includes the component type, a unique identifier for each component, the nodes or terminals it connects to, and relevant values like resistance, capacitance, voltage, etc.
  - Nodes:
    - Nodes in the netlist represent the electrical connection points. They are usually labeled numerically or with a combination of letters and numbers.
  - Component Representation:
    - Each component type has a specific prefix (e.g., 'R' for resistors, 'C' for capacitors, 'L' for inductors, 'Q' for transistors

Example of a Netlist

```
V1 1 0 DC 5
R1 1 2 1k
C1 2 0 10n
L1 2 3 1m
C2 3 0 1u
```



- In this example:
  - "V1 1 0 DC 5" indicates a DC voltage source named V1, between node 1 and ground, with a voltage of 5 volts.
  - "R1 1 2 1k" represents a resistor named R1, connected between nodes 1 and 2, with a resistance of 1 kilohms.
  - "C1 2 0 10n" denotes a capacitor named C1 between node 2 and ground (node 0), with a capacitance of 10 nano-Farads.
  - "L1 2 3 1m" is an inductor named L1, connecting node 2 to node 3, with an inductance of 1 milli-Henry.
  - "C2 3 0 1u" denotes a capacitor named C1 between node 3 and ground (node 0), with a capacitance of 1 micro-Farads.

Example of a More Complex Netlist!

```
THIS CIRCUIT IS A MOS LEVEL 1 MODEL CMOS INVERTER
.TRAN 20ns 30us 0 5ns
.PRINT tran v(vout) v(in) v(1)
.options timeint reltol=5e-3 abstol=1e-3


* Continuation Options
.options nonlin continuation=mos


VDDdev  VDD 0 5V
RIN IN 1 1K
VIN1  1 0  5V PULSE (5V 0V 1.5us 5ns 5ns 1.5us 3us)
R1     VOUT  0  10K
C2     VOUT  0  0.1p
MN1    VOUT  IN 0   0   CD4012_NMOS  L=5u W=175u
MP1    VOUT  IN VDD VDD CD4012_PMOS  L=5u W=270u
.MODEL cd4012_pmos PMOS
.MODEL cd4012_nmos NMOS
.END
```

Source: https://www.researchgate.net/figure/MOSFET-continuation-netlist-example-This-is-a-usage-example-the-circuit-itself-does_fig19_330156898

Example of a More Complex Netlist!

Example 3 : The input file of the circuit of Fig. 8 is:

```
BJT CE Amplifier
vsig 1 0 AC 1 sin(0 5m 100k)
VCC 6 0 DC 5V
Rs 1 2 1K
C1 2 3 1uF
.op
Qamp 4 3 5 Q2N3904
.model Q2N3904 NPN(Is=6.734f Xti=3
+Eg=1.11 Vaf=74.03 Bf=416.4 Ne=1.259
+Ise=6.734f Ikf=66.78m Xtb=1.5 Br=.7371
+Nc=2 Isc=0 Ikr=0 Rc=1 Cjc=3.638p Mjc=.3085
+Vjc=.75 Fc=.5 Cje=4.493p Mje=.2593 Vje=.75
+Tr=239.5n Tf=301.2p Itf=.4 Vtf=4 Xtf=2
+Rb=10)
*Note use "+" at the start of each new line
*when you continue writing the model
*parameters.
RB2 3 0 30K
RB1 6 3 10K
RE 5 0 1.3K
CE 5 0 10uF
RC 6 4 4.3K
C2 4 7 0.15uF
RL 7 0 100K
.ac dec 100 10 40meg
.tran 0.01u 20u
.probe
.END
```

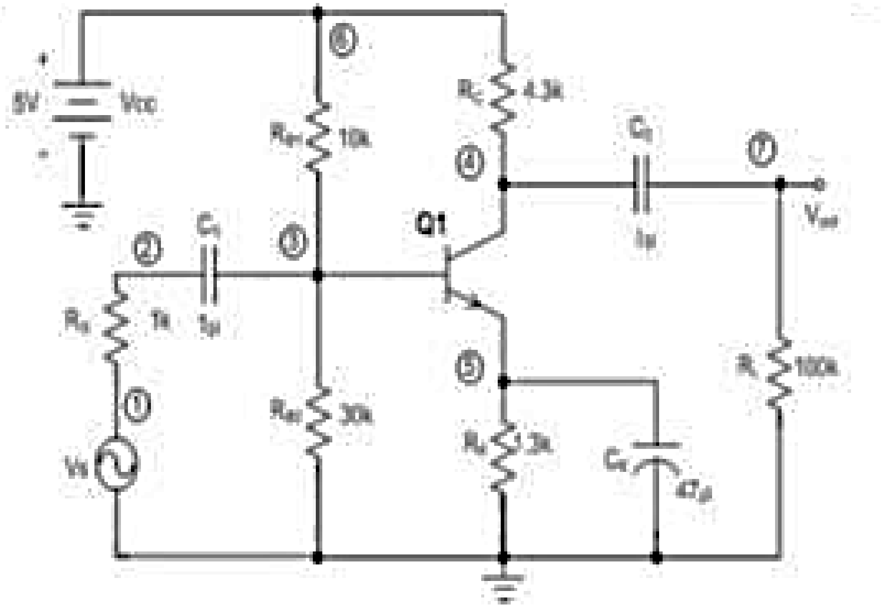


Fig. 8: Circuit for example 3.

Requirements:

1. Label the nodes according to the input file.
2. Run the program using default mode first.
3. Find A_M (midband gain).
4. Determine f_H and f_L where A_M reduces to $0.707A_M$.
5. Then add the given practical model and run the program.
6. Observe the differences.

Additional Remarks

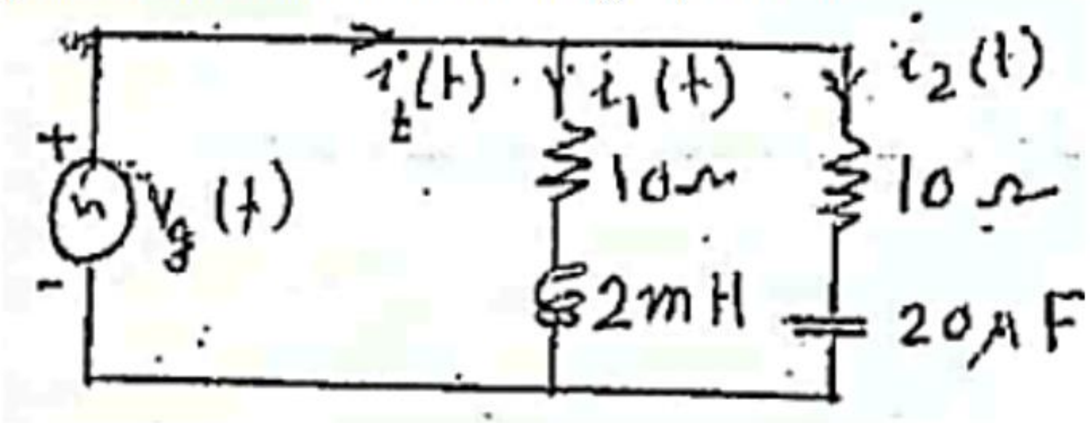
- Comments: Netlists often include comments for readability, usually starting with a specific character like '*'.

- Advanced Features:
  - More complex circuits may include subcircuits, models, and specialized components, which add to the complexity of the netlist.

- Netlists are crucial for electronic design automation (EDA) as they provide a standard way for describing circuits to be simulated or manufactured. Understanding how to read and write a netlist is a fundamental skill for anyone working in electronic circuit design.

Problem [2]:

In the circuit shown below, if $v_g(t) = 100\sin(5000t)\,V$. Find $i_1(t), i_2(t) and\ i_t(t)$ and sketch their waveforms on the same graph. Calculate the average power.

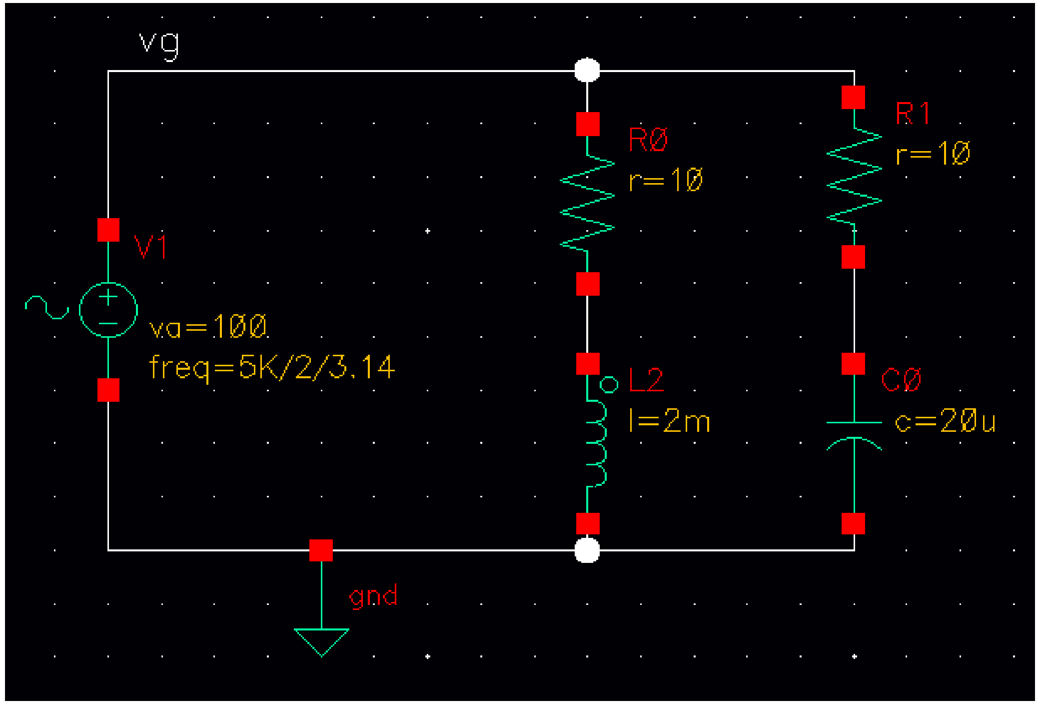


<u>Answers:</u> $i_1(t) = 5\sqrt{2}\sin(5000t - 45^o)\,A,\qquad i_2(t) = 5\sqrt{2}\sin(5000t + 45^o)\,A$

$i_t(t) = 10\sin(5000t)\,A,\qquad P_{avg} = 500W$

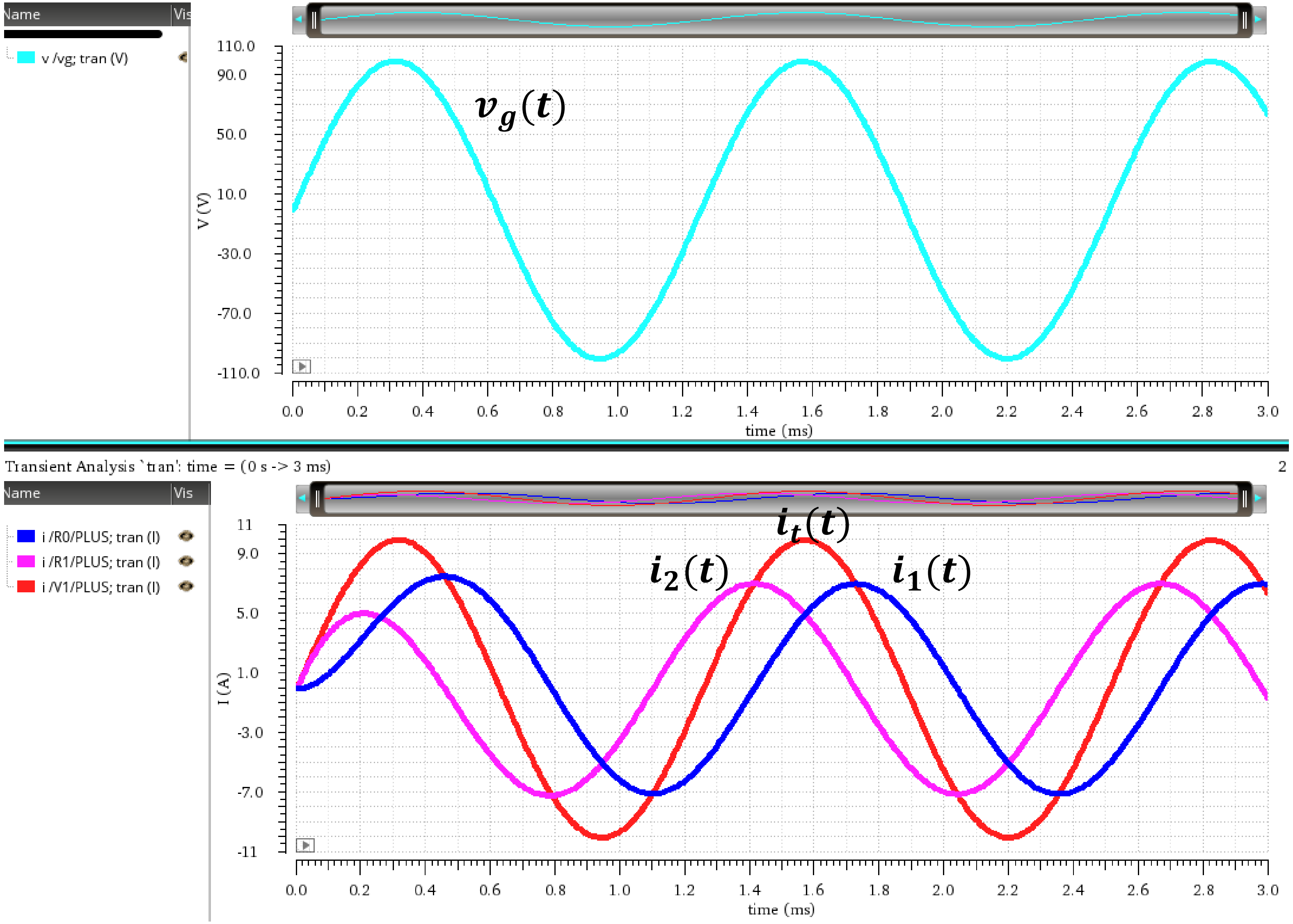
Demo 1: Sheet 7 – Problem 2

- Using Cadence Virtuoso
- Schematic and Netlist

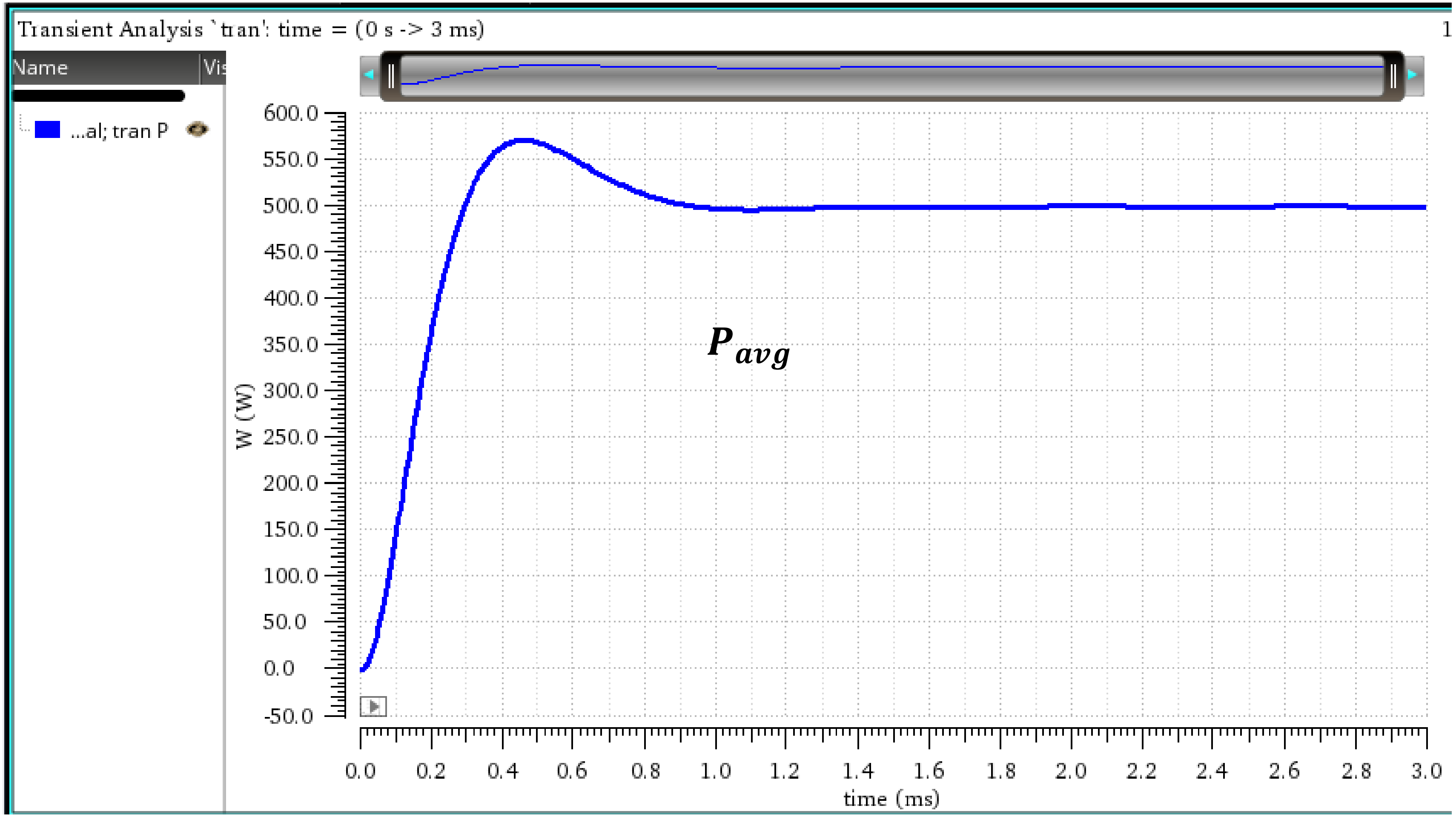



```
// Library name: Test_ob
// Cell name: demo3
// View name: schematic
V1 (vg 0) vsource type=sine ampl=100 sinephase=0 freq=5K/2/3.14
C0 (net5 0) capacitor c=20u
L2 (net6 0) inductor l=2m
R1 (vg net5) resistor r=10
R0 (vg net6) resistor r=10
simulatorOptions options reltol=1e-3 vabstol=1e-6 iabstol=1e-12 temp=27 \
    tnom=27 scalem=1.0 scale=1.0 gmin=1e-12 rforce=1 maxnotes=5 maxwarns=5 \
    digits=5 cols=80 pivrel=1e-3 sensfile="../psf/sens.output" \
    checklimitdest=psf
tran tran stop=3m errpreset=conservative write="spectre.ic" \
    writefinal="spectre.fc" annotate=status maxiters=5
finalTimeOP info what=oppoint where=rawfile
modelParameter info what=models where=rawfile
element info what=inst where=rawfile
outputParameter info what=output where=rawfile
designParamVals info what=parameters where=rawfile
primitives info what=primitives where=rawfile
subckts info what=subckts where=rawfile
save R0:1 R1:1 V1:p
saveOptions options save=allpub pwr=all currents=all
```

Demo 1: Sheet 7 – Problem 2



Demo 1: Sheet 7 – Problem 2



Monte Carlo Simulation

- Definition:
  - Monte Carlo simulation is a computational technique that uses random sampling and statistical modeling to estimate mathematical functions and mimic the behavior of complex systems.
- Application in Circuit Analysis:
  - In the context of electrical engineering, Monte Carlo simulations are used to analyze the impact of component variations and uncertainties in circuit designs.
- How Monte Carlo Simulations Work

1. Random Sampling:
   - The simulation randomly selects values for each component within predefined tolerance ranges. These values might represent resistances, capacitances, inductances, and other parameters that can vary in real-world conditions.
2. Multiple Iterations:
   - The circuit is simulated multiple times (hundreds or thousands of iterations), each time with a different set of randomly selected component values.
3. Statistical Analysis:
   - After all iterations are completed, the results are analyzed statistically to understand the probability of various outcomes, like the likelihood of a circuit operating within its required specifications.

Why Use Monte Carlo Simulations?

- Handling Uncertainties:
  - They help in understanding how variations and uncertainties in component values affect the overall performance of a circuit.
- Design Robustness:
  - Engineers use these simulations to ensure that a circuit design is robust and reliable, even when subjected to component tolerances and manufacturing variations.
- Risk Assessment:
  - Monte Carlo simulations can identify the risk of circuit failure, allowing designers to make informed decisions to mitigate these risks.
- Optimization:
  - They are useful in optimizing circuit designs for critical parameters while considering real-world variability.

Applications of MC Simulations

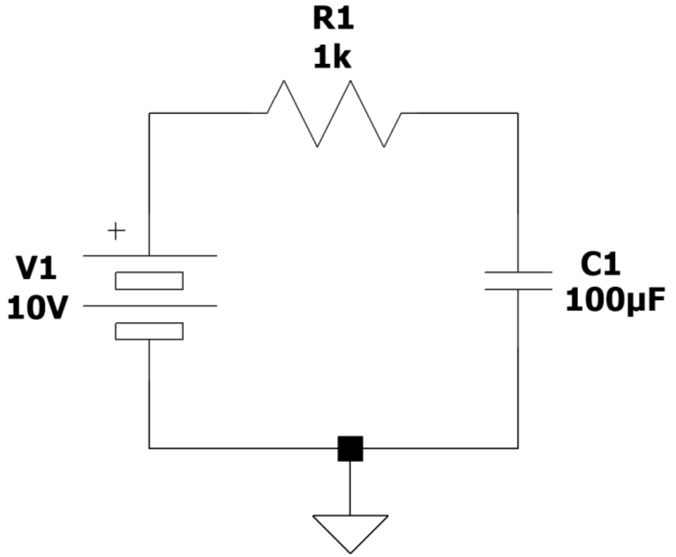
- Tolerance Analysis:
  - Evaluating how much variation in component values a circuit can tolerate before failing to meet its specifications.
- Yield Analysis:
  - Estimating the percentage of circuits that will function correctly out of a batch manufactured under certain tolerance conditions.
- Performance Variability:
  - Assessing how performance metrics like frequency response, gain, and signal-to-noise ratio might vary due to component tolerances.
- Devices Mismatch:
  - Assessing the impact of the mismatch between identical devices on the overall circuit's performance.

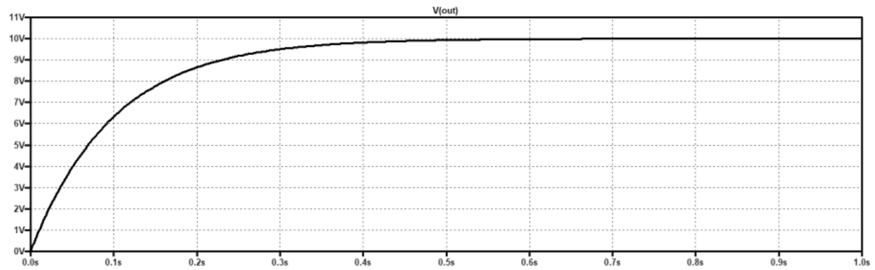
Implementing Monte Carlo Simulations

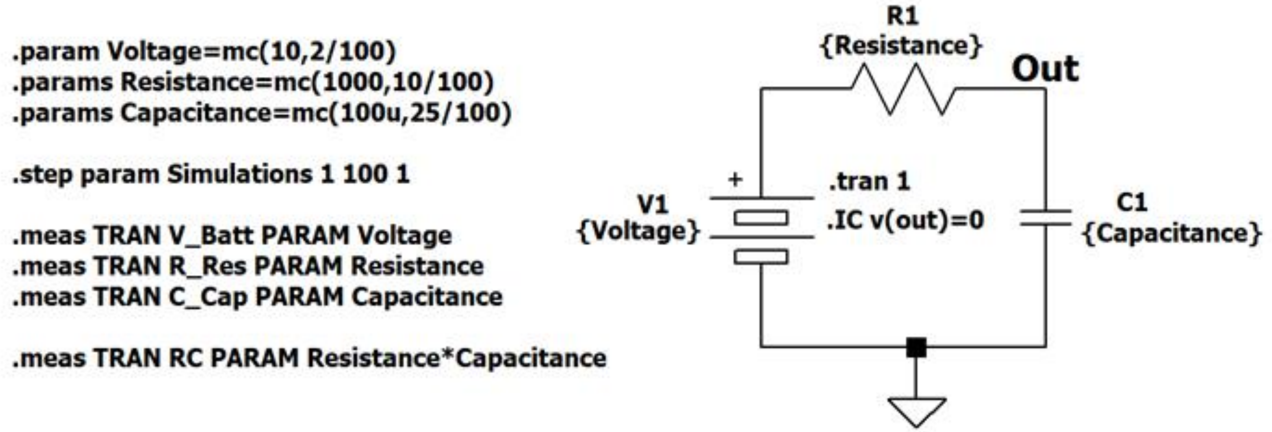
- Software Tools:
  - Many circuit simulation programs like SPICE offer built-in Monte Carlo analysis features.
- Custom Simulations:
  - For more specific needs, custom Monte Carlo simulation scripts can be written using programming languages like Python, MATLAB, or others.

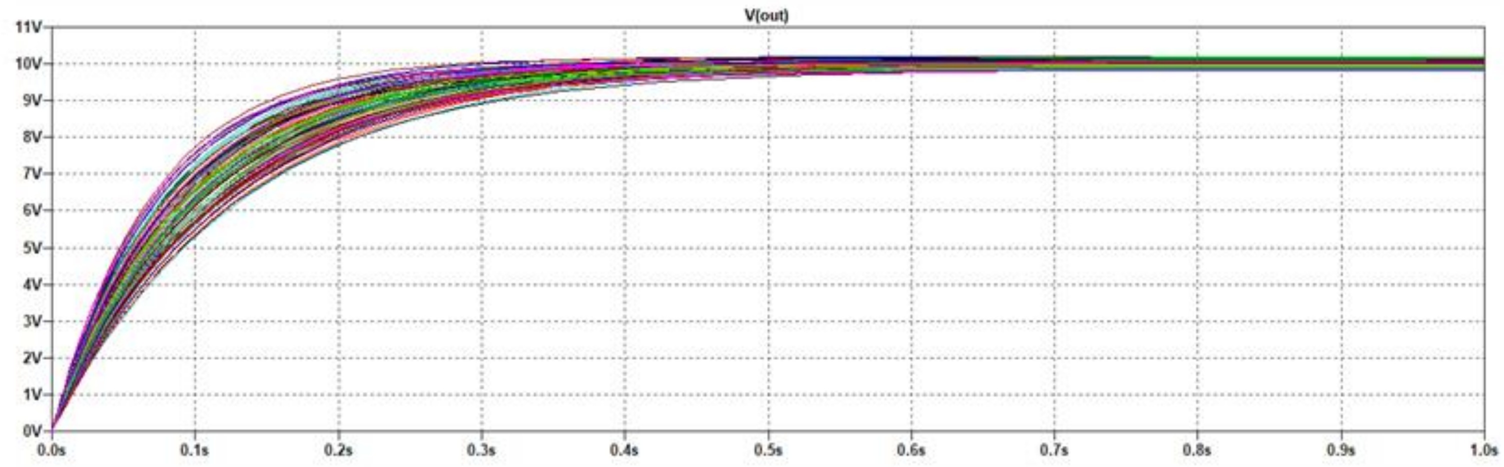
- In conclusion, Monte Carlo simulations play a vital role in modern circuit design, providing valuable insights into the performance and reliability of circuits in the face of real-world uncertainties. By leveraging statistical techniques, they allow engineers to design more robust and efficient circuits.

Example of MC Simulation









Source: https://www.powerelectronicsnews.com/power-supply-design-notes-tutorial-monte-carlo-analysis-with-sic/

Example of MC Simulation

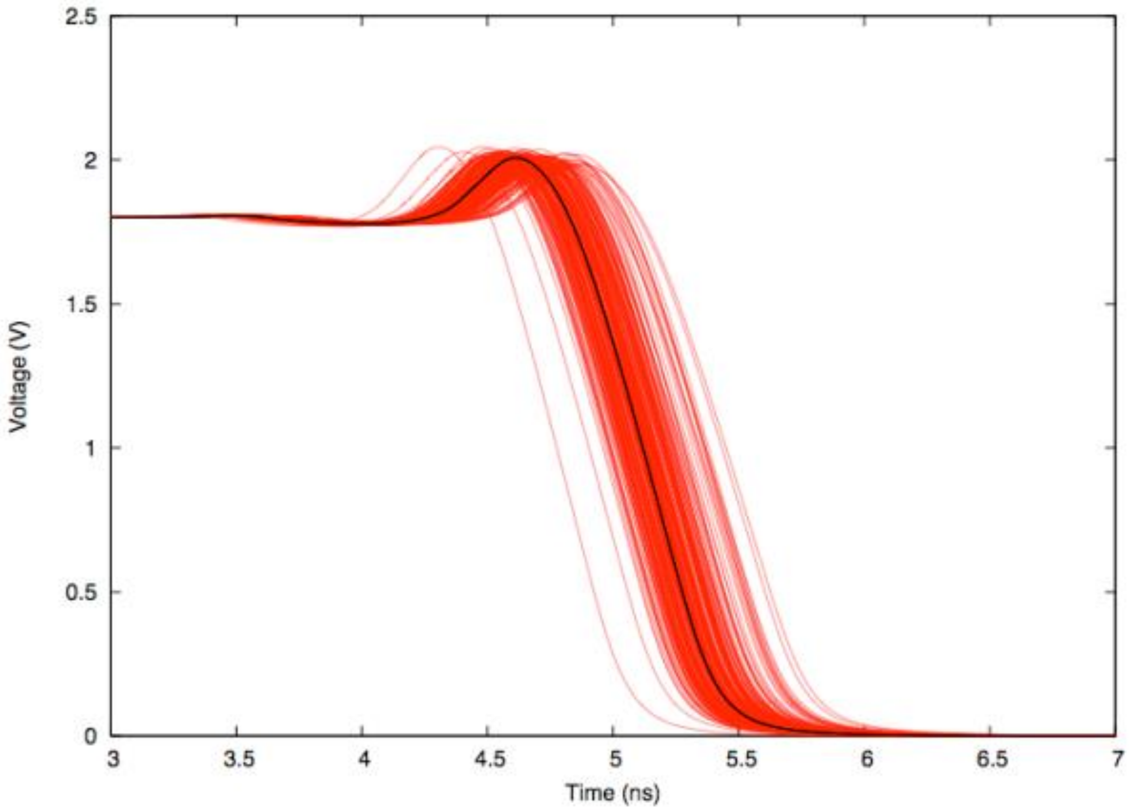


Figure 10. Graph of the output voltage versus time of the last inverters in the 8-stage inverter strings

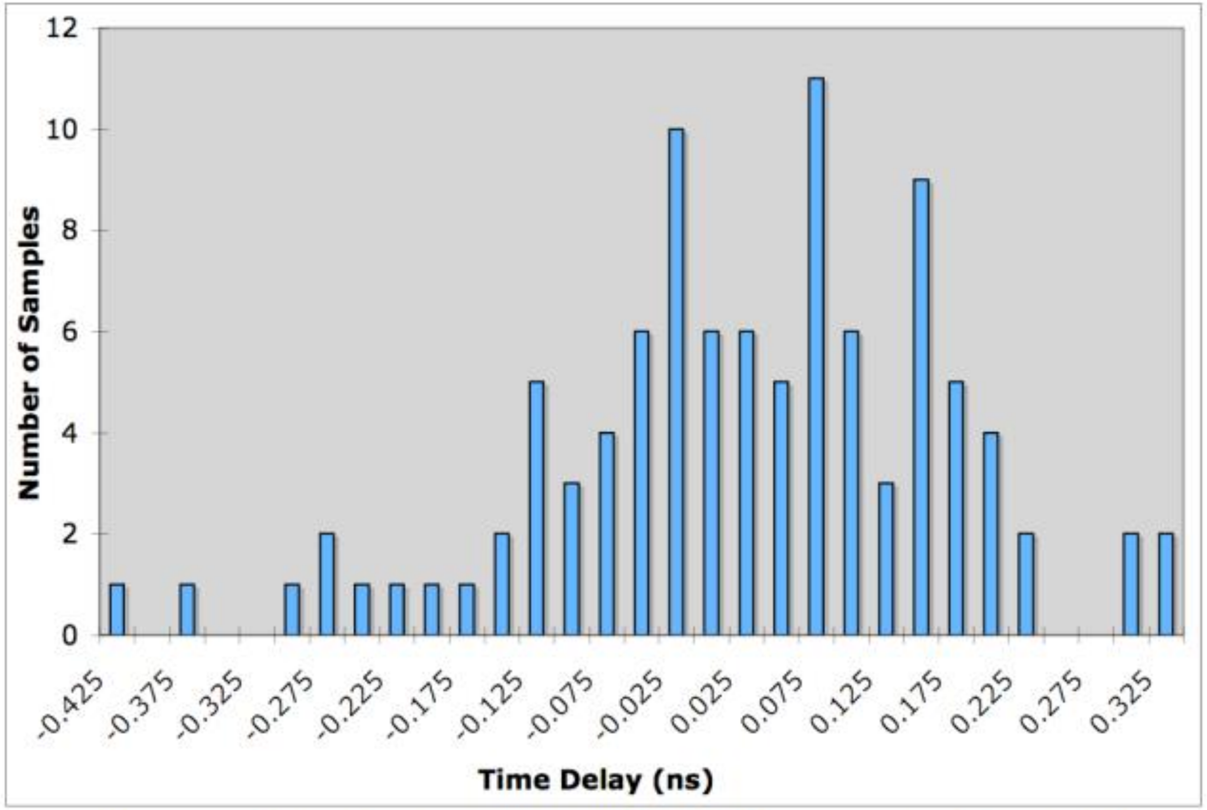


Figure 11. Histogram of the difference in time delay of the last inverters in the 8-stage inverter strings

Source: http://www.cisl.columbia.edu/kinget_group/student_projects/montecarlotools/MonteCarloDeviceMismatch.pdf

AI and Machine Learning in Circuit Simulation

- Automated Design Optimization:
  - AI algorithms can automate the process of optimizing circuit parameters, quickly identifying the most efficient designs that meet specified criteria.
- Predictive Modeling:
  - Machine learning models can predict circuit behavior under various conditions, reducing the need for exhaustive simulations.
- Error Detection and Correction:
  - AI can assist in identifying errors or inefficiencies in circuit designs, suggesting corrections or improvements.
- Data-Driven Simulations:
  - ML can analyze vast amounts of simulation data to uncover patterns and insights that might be missed by traditional analysis.

Enhanced Simulation Capabilities

- Faster Simulations:
  - Advances in computational power and algorithms are making it possible to simulate complex circuits more quickly and accurately.
- Cloud-Based Simulation:
  - Cloud computing offers scalable resources for high-performance simulations, making advanced simulation capabilities more accessible.
- Quantum Computing:
  - The potential integration of quantum computing could revolutionize circuit simulation, offering unprecedented simulation speeds and accuracy.

The Future of Circuit Simulators: Trends and Innovations

Improved User Interfaces and Accessibility

- Intuitive Design Tools:
  - Future simulators will likely feature more user-friendly interfaces, making circuit design and simulation accessible to a broader range of users.
- Virtual and Augmented Reality:
  - These technologies could be used to create immersive simulation environments, offering an intuitive understanding of circuit behavior.

The Future of Circuit Simulators: Trends and Innovations

Integration with Other Technologies

- Internet of Things (IoT) and 5G:
  - Simulators will need to adapt to support the design and testing of circuits in IoT devices and 5G technology.
- Customizable and Modular Simulation Tools:
  - The trend towards customizable simulation platforms allows engineers to tailor tools to their specific needs.

Educational and Collaborative Tools

- Enhanced Learning Tools:
  - Future simulators could incorporate AI-driven tutors or guides to enhance learning and understanding of circuit concepts.
- Collaborative Platforms:
  - Simulators may evolve to offer more collaborative features, enabling teams to work together seamlessly on circuit designs.

Challenges and Considerations

- Balancing Accuracy and Speed:
  - One ongoing challenge is to balance the need for fast simulations with the requirement for accuracy, especially in complex circuits.
- Data Privacy and Security:
  - As simulators move to the cloud and incorporate AI, ensuring the privacy and security of data becomes crucial.

- In conclusion, the future of circuit simulators is poised to be influenced significantly by advancements in AI, machine learning, and other emerging technologies. These advancements promise to enhance the speed, accuracy, and usability of simulators, profoundly impacting circuit design and analysis. The integration of these technologies will likely make simulators more powerful and insightful tools, essential in the increasingly complex world of electronic design.

Conclusions

- Circuit simulators are sophisticated tools that replicate the behavior of electronic devices by utilizing complex mathematical models and algorithms.
- They provide an essential virtual platform for the design, analysis, and optimization of electronic circuits in a safe, cost-effective, and efficient manner.
- Numerous circuit simulators are available, and the optimal approach is to synergize them to reap the utmost advantages.
- Circuit designers do not operate in isolation; instead, they work in concert with software/CAD engineers, modeling engineers, systems engineers, and others.
- Netlists provide a standard way for describing circuits to be simulated or manufactured. Understanding how to read and write a netlist is a fundamental skill for anyone working in electronic circuit design.