Introduction To SPICE

© 2020 G. W. Roberts

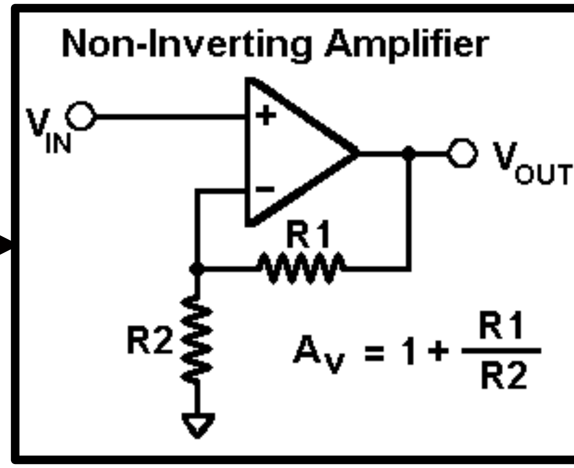
Outline

- Introduction
- Value of Prototyping
- Analog Simulation
- Where To Get Spice
- Summary

© 2020 G. W. Roberts

Open-Loop Design

Design Specs & Methods

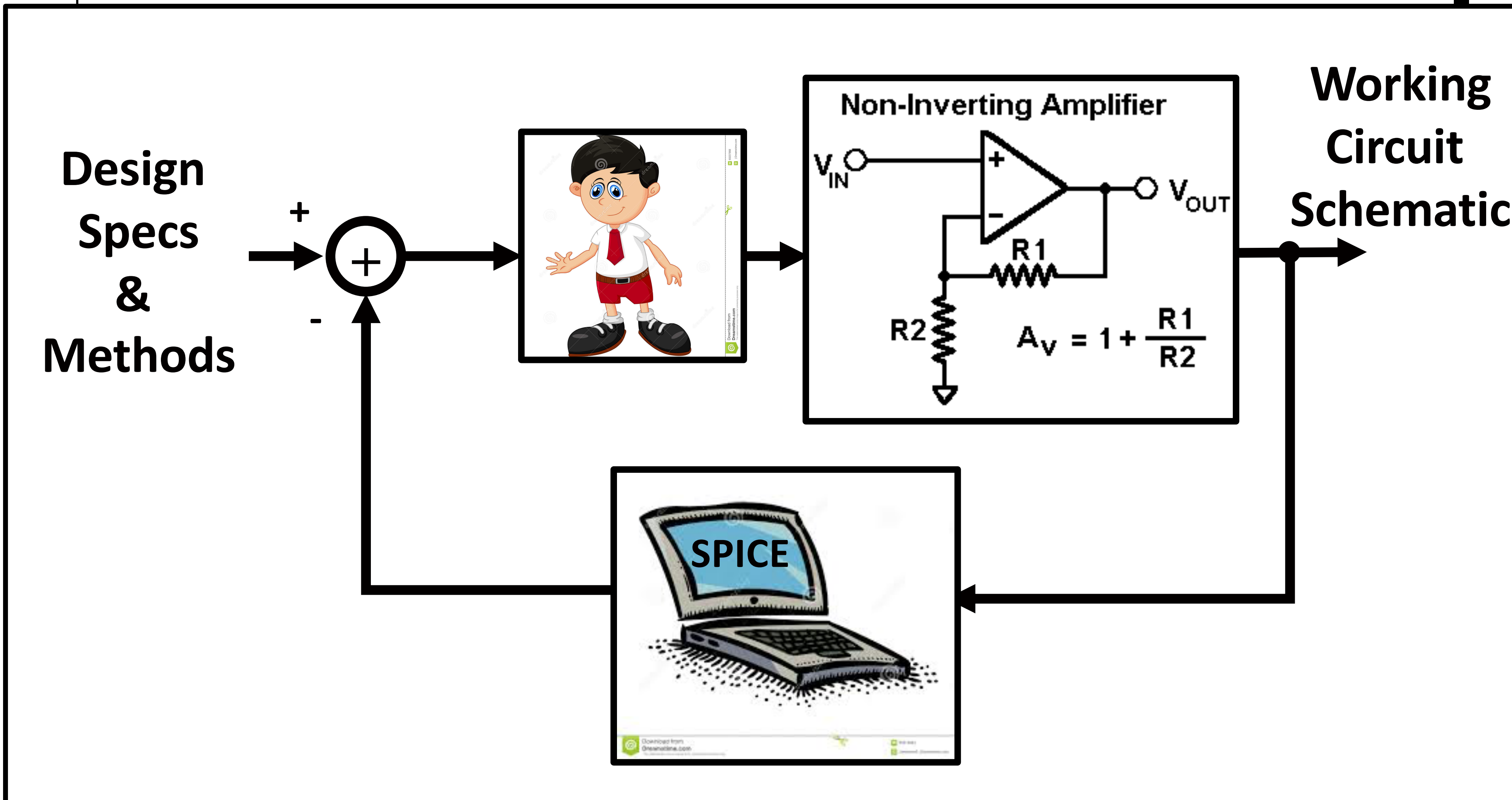


Circuit Schematic

- How does the student know the design methods lead to a workable solution?
- How does the student know the Professor knows what he talking about?

© 2020 G. W. Roberts

Spice Will Help Close The Loop

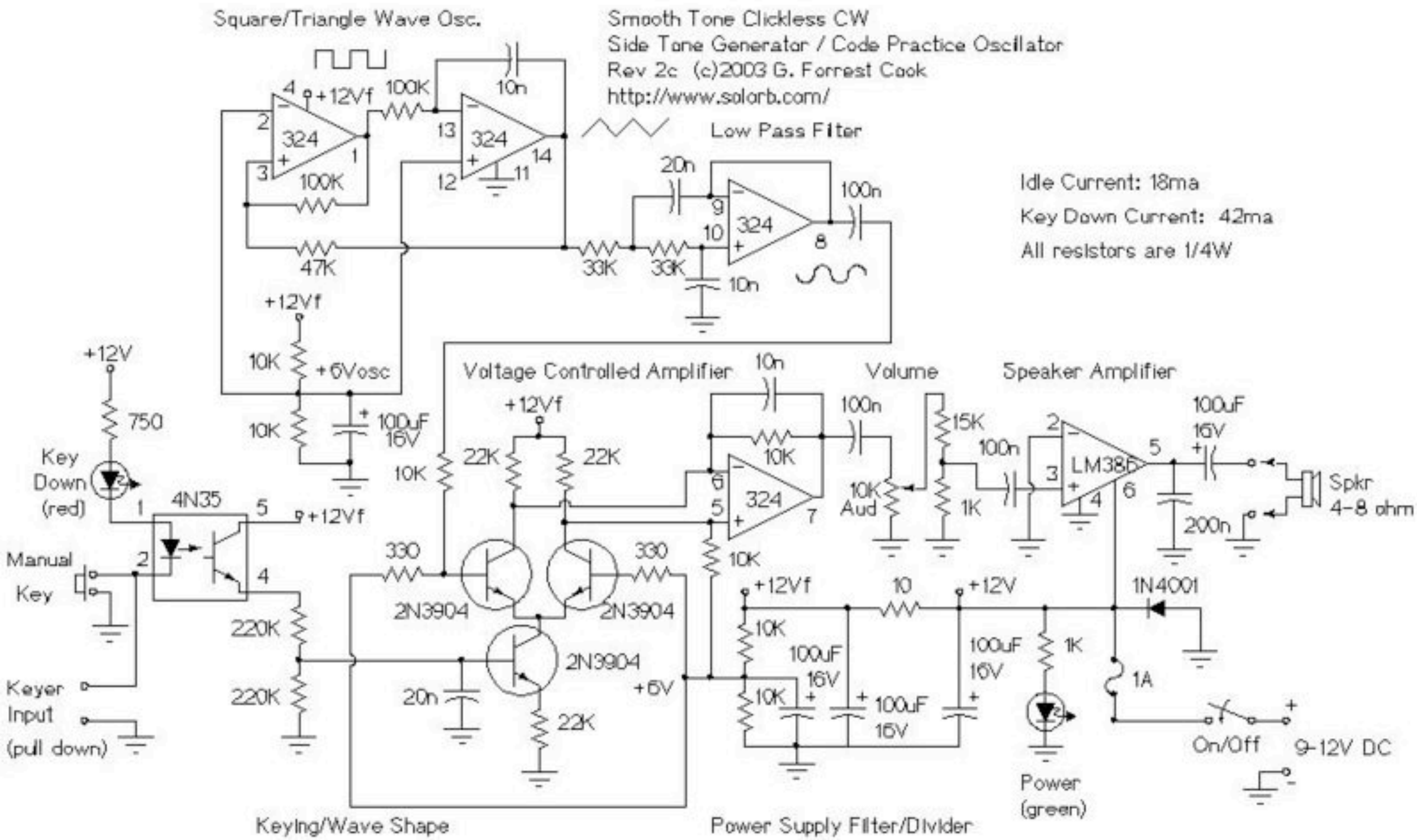


Outline

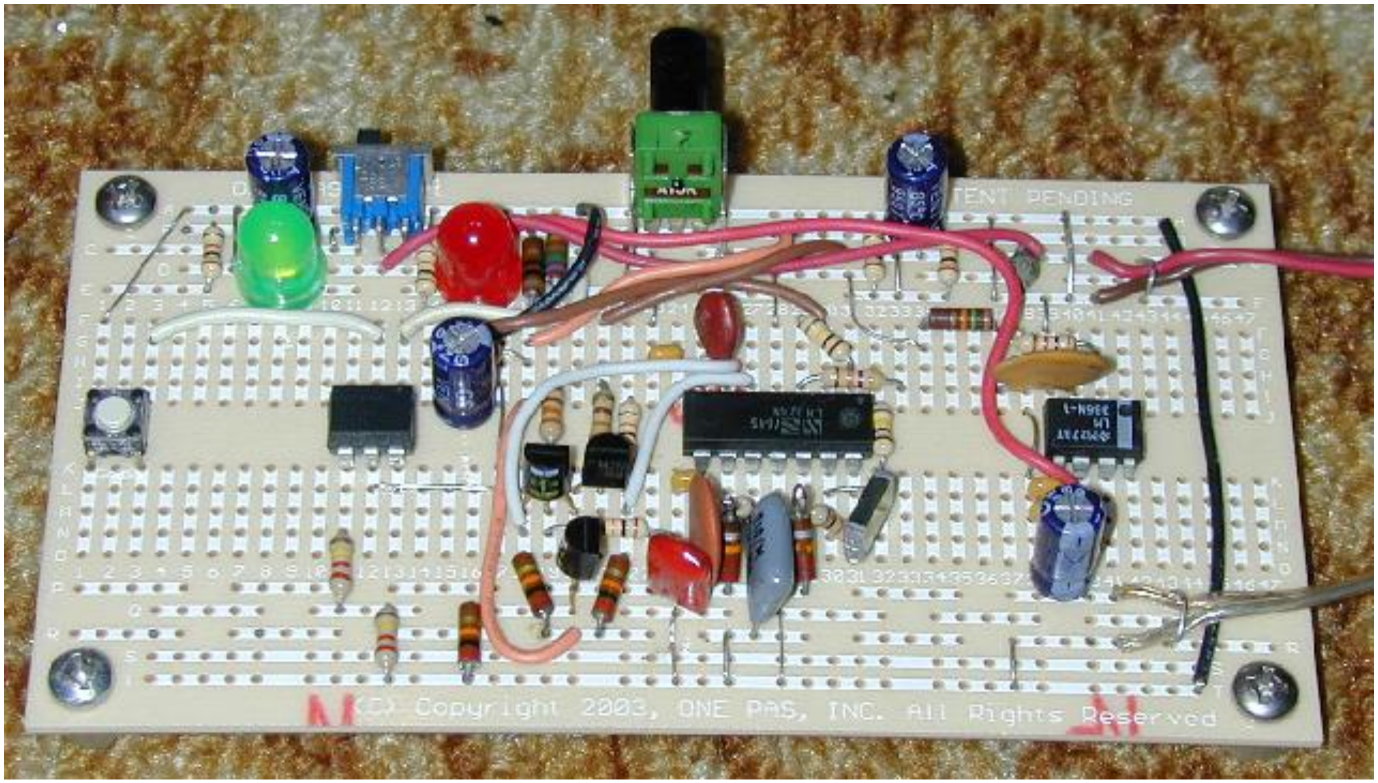
- Introduction
- Value of Prototyping
- Analog Simulation
- Where To Get Spice
- Summary

© 2020 G. W. Roberts

Sidetone Generator Circuit

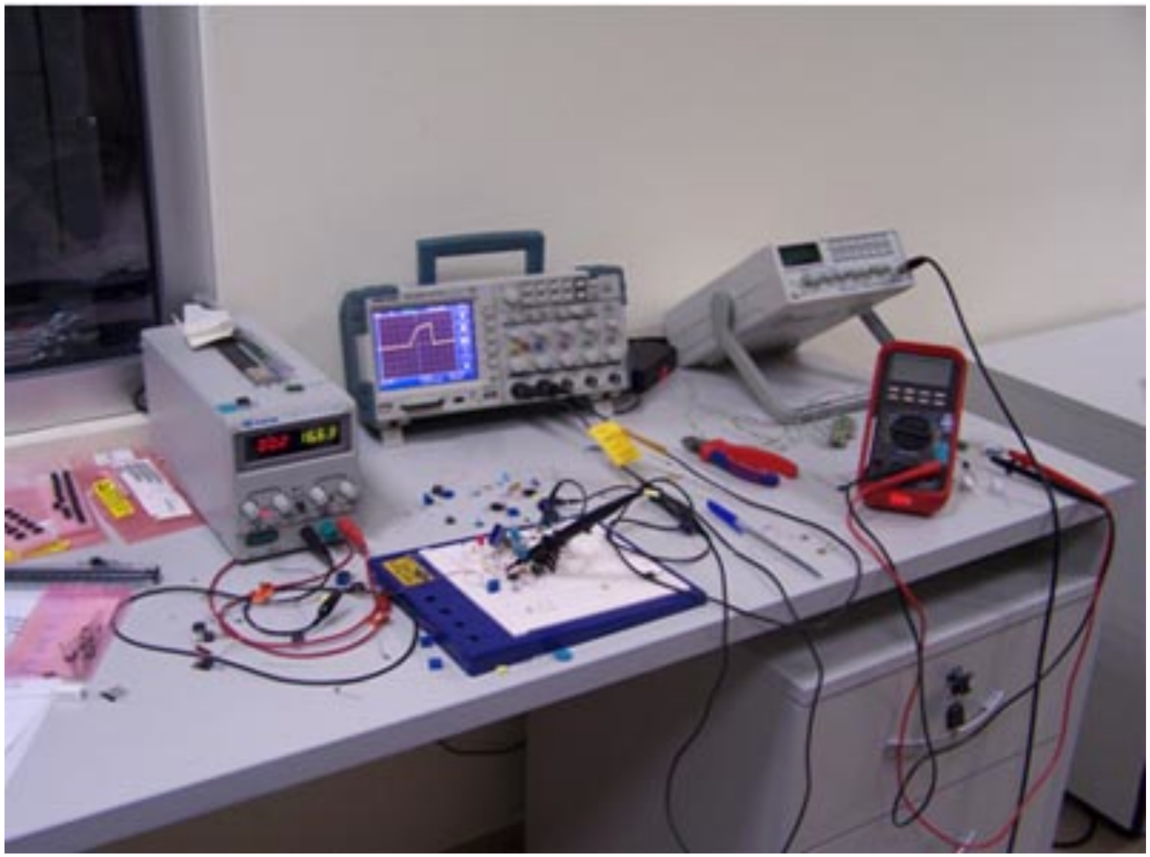


Prototyping Electronic Circuit



- To verify that your design works correctly, a prototype of the circuit is constructed.

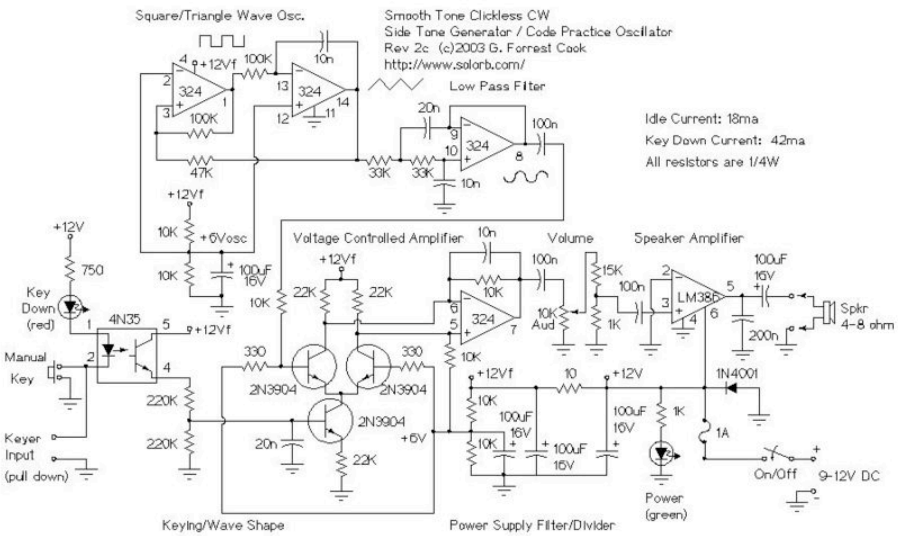
Laboratory Bench Setup

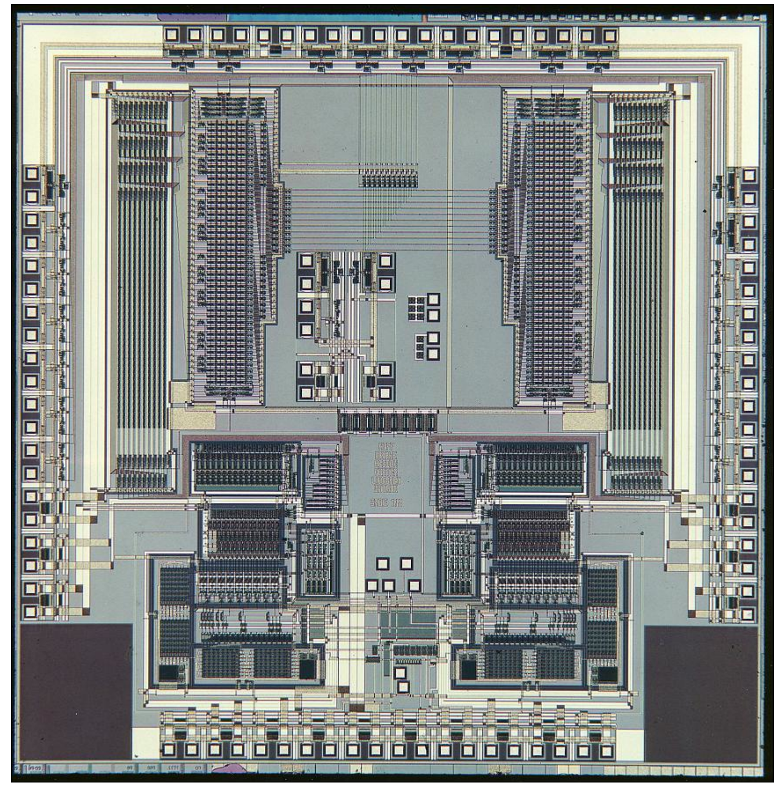


- Engineer will characterize design using a set of signal generators and measuring instruments.

© 2020 G. W. Roberts

Mass Production





- IC Implementation (1 cm x 1 cm dimensions) when volumes are high and small size is required.

Time And Money

- All this activity takes time.
- Engineering principle:
  - ***Every successful project takes at least two iterations to get it right.***
- Investors want ROI (Return On Investment) in near zero time.
  - Calls for innovative solutions.
    - » Electronic Engineers responded with the idea of a circuit simulator.
      - –improves design times and reduces engineering errors (get it right the first time).

© 2020 G. W. Roberts

Outline

- Introduction
- Value of Prototyping
- Analog Simulation
- Where To Get Spice
- Summary

© 2020 G. W. Roberts

Analog Circuit Simulation

- **SPICE** - Simulation Program with Integrated Circuit Emphasis
  - developed at University of California/Berkeley
  - coincided with emergence of “Silicon Valley” across the “bay”
  - widely adopted, *de facto* standard.
- Numerical approach to circuit simulation (as opposed to symbolic)
  - circuit nodes/connections define a matrix
  - external nodes provide boundary conditions.
- Circuit elements represented by device models
  - simple (eg., resistor) - complex (eg. MOSFET)

© 2020 G. W. Roberts

Types of Analysis

- Three basic analysis available:
  - **DC Analysis**
    - » calculates DC voltages and currents in a circuit (nonlinear)
    - » Also referred to as DC bias or operating point
  - **Transient Analysis**
    - » calculates the behavior of a circuit subject to time-varying input signals.
    - » most important analysis but very time consuming
  - **AC Analysis**
    - » Small-signal or linear analysis
    - » Take care when using this command, as amplitude of input signal has no significance.

© 2020 G. W. Roberts

Other Analysis Types

- Special case of the previous three:
  - **DC sweep, transfer function (TF) and temperature.**
    - » Repeated use of DC Analysis
  - **Fourier and Monte Carlo analysis.**
    - » Post-processing of results from Transient Analysis
  - **Noise and robustness analysis**
    - » Post-processing of results from AC Analysis

© 2020 G. W. Roberts

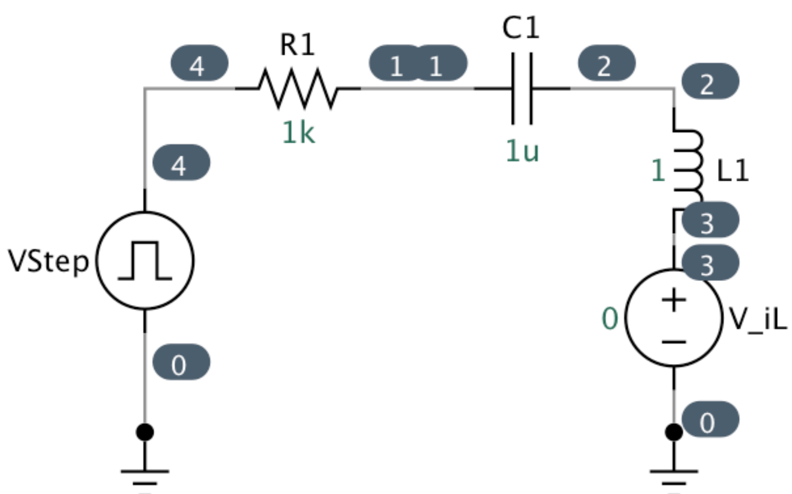
Transient Analysis

- Most important analysis, but most time consuming and riddled with convergence problems
  - failure to converge due to missing connection
  - time step too small due to rise and fall times
- Many of today's circuits are difficult to analyze with SPICE due to widely varying time constants (poles spread very far apart).
  - long simulation times required.
- Nonetheless, circuit simulation is here to stay; all engineering disciplines use some form of computer-based simulation.

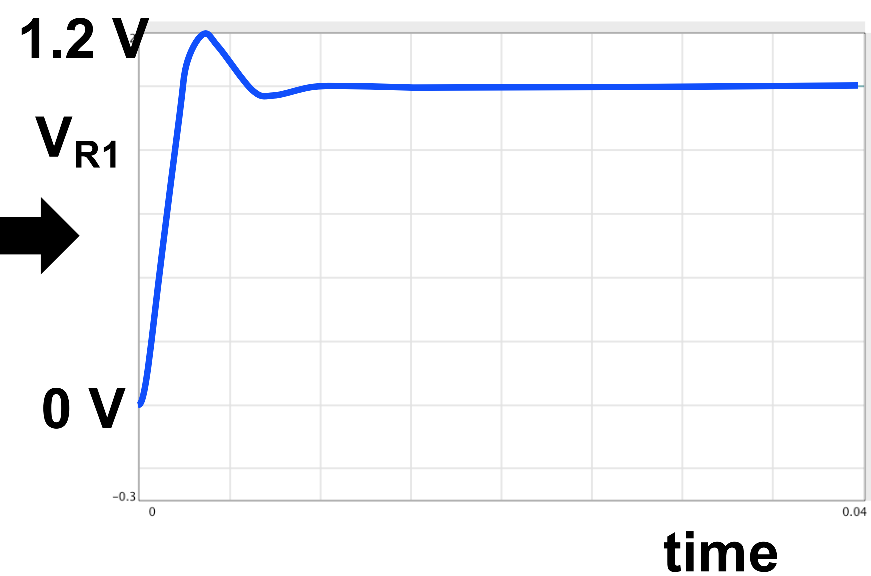
© 2020 G. W. Roberts

Input /Output To SPICE

SPICE Input File → **SPICE Simulation** → **SPICE Output File**



voltage across resistor R1



Spice Description Language:
Passive Elements

Passive Elements

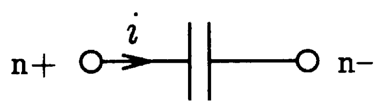
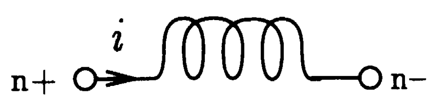
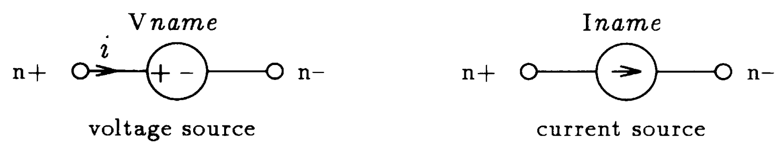
Element	Spice Description
	R*name* n+ n− *value*
	C*name* n+ n− *value* [IC=*initial_voltage_condition*]
	L*name* n+ n− *value* [IC=*initial_current_condition*]

Figure 1.4 Spice descriptors for passive elements. Fields surrounded by [ ] are optional.

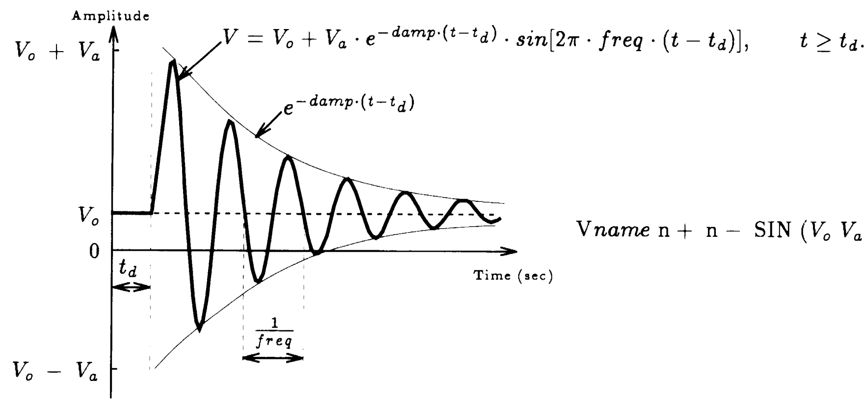
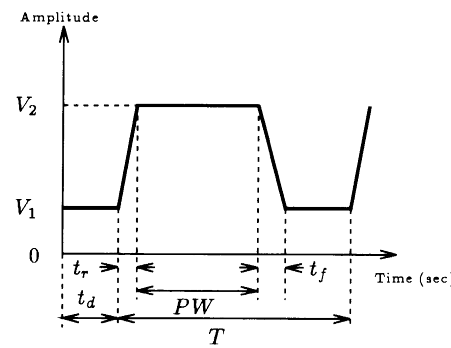
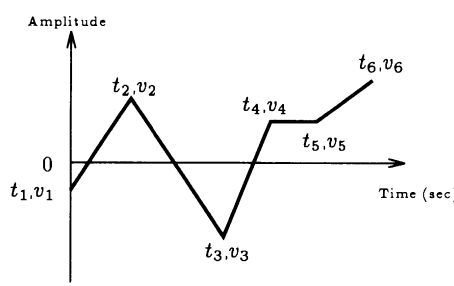
© 2020 G. W. Roberts

Spice Description Language: Independent Sources



Spice Description	Type Of Analysis
$\left\{ \begin{array}{c} \mathrm{V}name \\ \mathrm{I}name \end{array} \right\}$ n+ n− DC *value*	All Types
$\left\{ \begin{array}{c} \mathrm{V}name \\ \mathrm{I}name \end{array} \right\}$ n+ n− AC *magnitude phase_degrees*	AC Frequency Response
$\left\{ \begin{array}{c} \mathrm{V}name \\ \mathrm{I}name \end{array} \right\}$ n+ n− SIN (V_o V_a $freq$ t_d $damp$)	Transient
$\left\{ \begin{array}{c} \mathrm{V}name \\ \mathrm{I}name \end{array} \right\}$ n+ n− PULSE (V_1 V_2 t_d t_r t_f PW T)	Transient
$\left\{ \begin{array}{c} \mathrm{V}name \\ \mathrm{I}name \end{array} \right\}$ n+ n− PWL (t_1,v_1 t_2,v_2 ... t_n,v_n)	Transient

SDL: Time-Varying Signal Sources

Time-Varying Signal	Spice Description
	V*name* n + n − SIN (V_o V_a $freq$ t_d $damp$)
	V*name* n + n − PULSE (V_1 V_2 t_d t_r t_f PW T)
	V*name* n + n − PWL (t_1, v_1 t_2, v_2 ... t_n, v_n)

© 2020 G. W. Roberts

Spice Description Language:
Linear Dependent Sources

Linear Dependent Sources

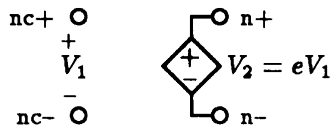
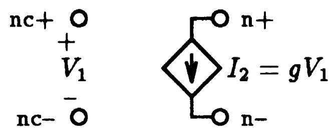
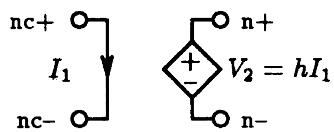
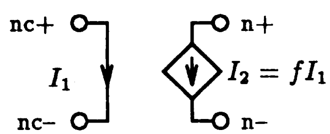
Element	Spice Description
 voltage-controlled voltage source	*E*name n+ n− nc+ nc− *e_value*
 voltage-controlled voltage source	*G*name n+ n− nc+ nc− *g_value*
 current-controlled voltage source	*H*name n+ n− *V*name *h_value* *V*name nc+ nc− 0
 current-controlled voltage source	*F*name n+ n− *V*name *f_value* *V*name nc+ nc− 0

Figure 1.7 Linear dependent sources. Notice that the CCVS and the CCCS are both specified using two Spice statements, unlike the other two dependent sources.

© 2020 G. W. Roberts

Spice Description Language:
Main Analysis Commands or Directives

Analysis Requests	Spice Command
Operating-point	.OP
DC sweep	.DC *source_name start_value stop_value step_value*
AC frequency response	.AC DEC *points_per_decade freq_start freq_stop* .AC OCT *points_per_octave freq_start freq_stop* .AC LIN *total_points freq_start freq_stop*
Transient response	.TRAN *time_step time_stop* [*no_print_time max_step_size*] [UIC] .IC V($node_1$) = *value* V($node_2$) = *value* . . . .

Spice Description Language:
PSpice Output Requests
(Not Used in LTSpice)

Output Requests	Spice Command
Print data points	.PRINT DC *output_variables* .PRINT AC *output_variables* .PRINT TRAN *output_variables*
Plot data points	.PLOT DC *output_variables* [(*lower_plot_limit, upper_plot_limit*)] .PLOT AC *output_variables* [(*lower_plot_limit, upper_plot_limit*)] .PLOT TRAN *output_variables* [(*lower_plot_limit, upper_plot_limit*)]

Notes:

1. Spice *output_variables* can be a voltage at any node V(*node*), the voltage difference between two nodes V($node_1$, $node_2$), or the current through a voltage source I(V*name*).
2. AC *output_variables* can also be

   Vr, Ir: real part

   Vi, Ii: imaginary part

   Vm, Im: magnitude

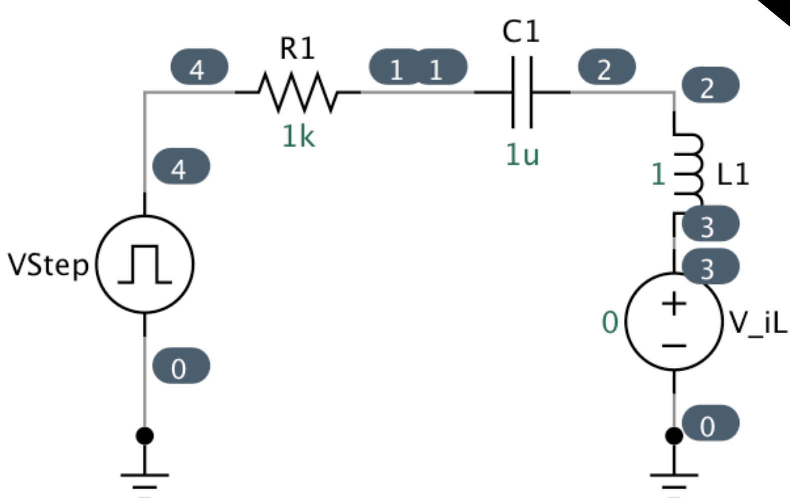
   Vp, Ip: phase

   Vdb, Idb: decibels
3. PSpice provides a greater flexibility for specifying *output_variables*.

© 2020 G. W. Roberts

Input /Output To SPICE

SPICE Input Netlist File → SPICE Simulation → SPICE Output File



SPICE Input Netlist File

Title

Circuit Descriptions

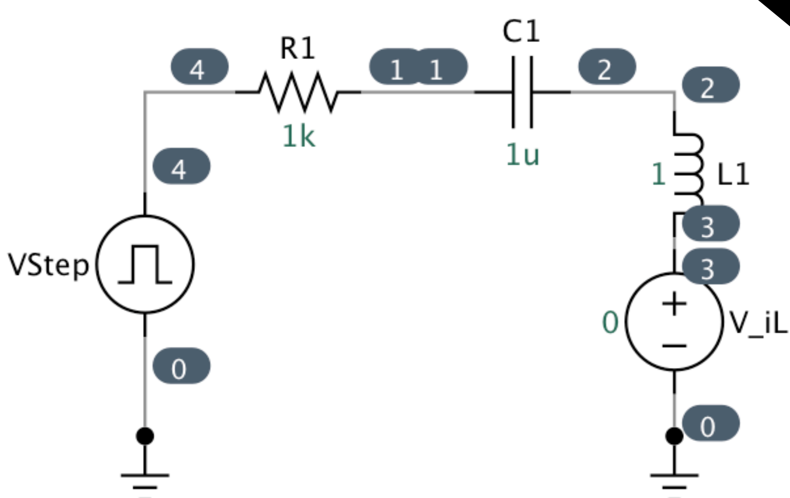
Device Models

Analysis Requests

© 2020 G. W. Roberts

Input /Output To SPICE

SPICE Input Netlist File → SPICE Simulation → SPICE Output File

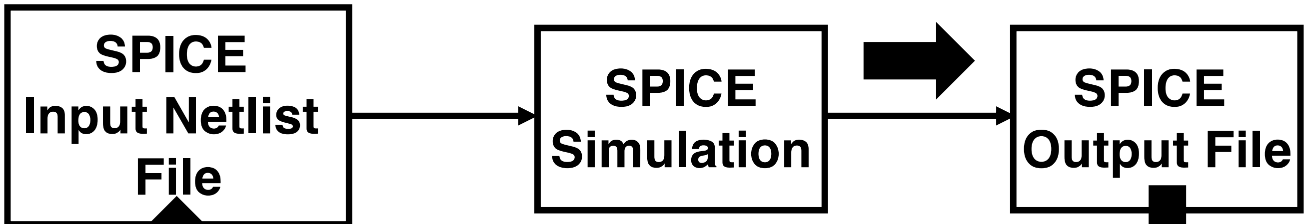


Spice Input Deck - Derived Directly From Mi-Sugar

```
* 2008-09-30 12:15:59 -0400
* Question P1.17, Roberts & Sedra
*
* Circuit Description
R1 4 1 1k
VStep 4 0 PULSE(0 1 0 10ns 10ns 40m 100ms)
C1 2 1 1u
L1 2 3 1
ViL 3 0 DC 0
*
* Circuit Analysis
.op
.tran 100u 40m 0 100u
.print tran v(1,2) v(2,3) v(3)
.print tran i(ViL)
.end
```

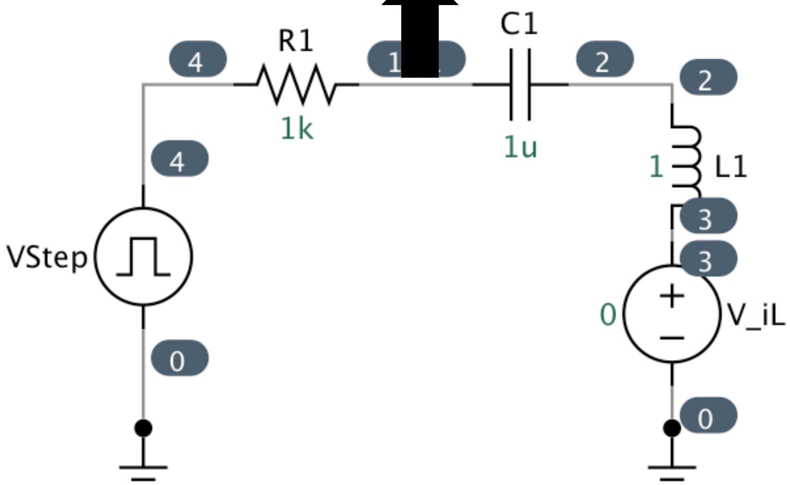
© 2020 G. W. Roberts

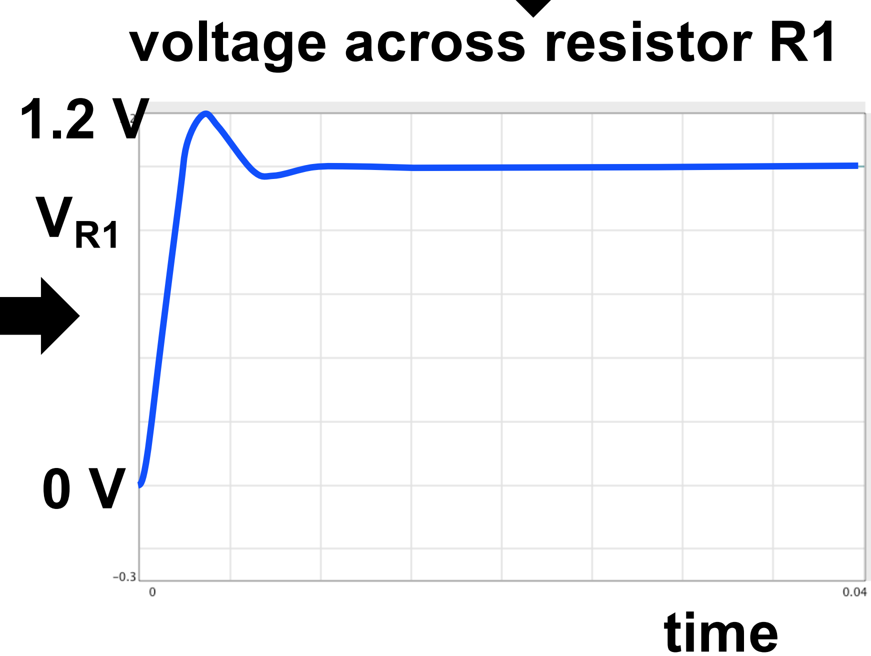
Input /Output To SPICE



Spice Input Deck - Derived Directly From Mi-Sugar

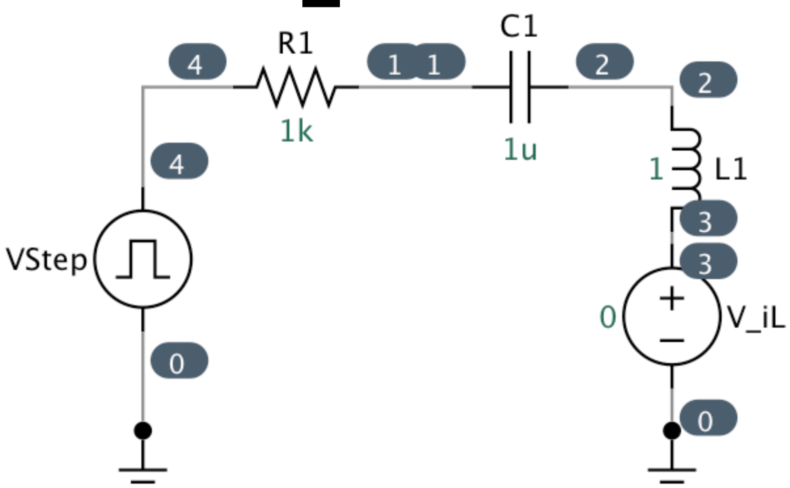
```
* 2008-09-30 12:15:59 -0400
* Question P1.17, Roberts & Sedra
*
* Circuit Description
R1 4 1 1k
VStep 4 0 PULSE(0 1 0 10ns 10ns 40m 100ms)
C1 2 1 1u
L1 2 3 1
ViL 3 0 DC 0
*
* Circuit Analysis
.op
.tran 100u 40m 0 100u
.print tran v(1,2) v(2,3) v(3)
.print tran i(ViL)
.end
```



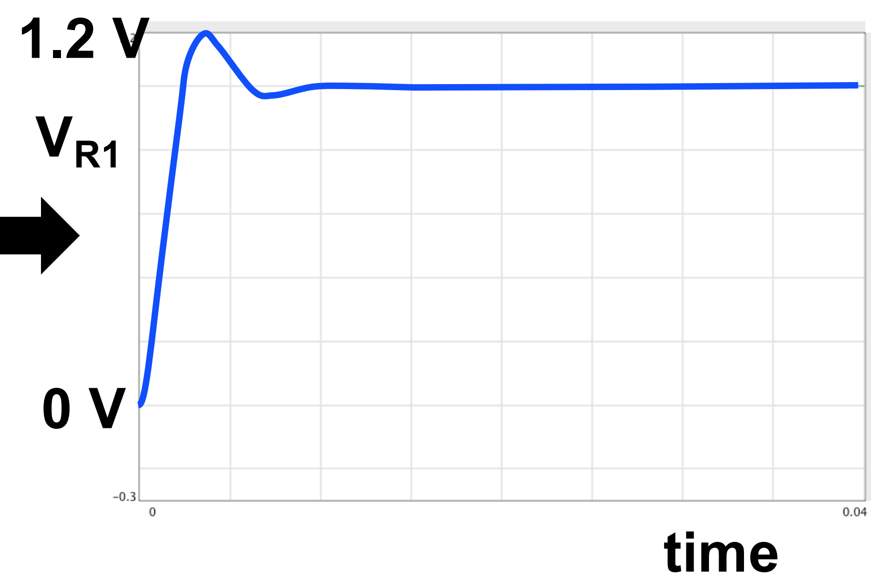


Input /Output To SPICE

Schematic Capture → SPICE Input Netlist File → SPICE Simulation → SPICE Output File



voltage across resistor R1



Outline

- Introduction
- Value of Prototyping
- Analog Simulation
- Where To Get Spice
- Summary

Where Do You Get SPICE

- The student will use SPICE extensively in this course.
- Free versions of SPICE can be downloaded directly off the internet.
  - Programs come in two flavors:
    - » Text Entry type
      - Spice3 (windows)
      - MACSpice (OS X)
    - » Schematic Circuit Entry
      - LTSpice
      - PSpice
      - TINA-TI

Many resources on internet but on your own

McGill

304-335

Course Web Site

Welcome | Contact | Research Activity | Courses | My Textbooks

ECSE 304-335 Microelectronics

Course Details:

- Course Description
- Quiz Dates

Course Support People:

- Teaching Assistants
- Lab Instructors

Instructor Material (login required, maybe twice):

- Lecture Notes
- Animations
- Online Spice Resources

Course Work (login required):

- Assignments
- Simulation Laboratory

Lecture Notes | Animations | Assignments | Simulation Lab Overview | Lab Description | Midterm Details | Final Details

On-Line Spice Resources

- My Textbooks on Spice
  - PSpice Reference
  - LTSpice Reference
- Software Downloads
  - Cadence PSpice
  - Analog Devices

ECSE 304-335

Summary

- An introduction to SPICE was given; the student will use SPICE throughout this course.

- The idea of negative feedback was described, and this will come to be the central idea on how to fix many of your circuit impairments.

© 2020 G. W. Roberts