NATIONAL RADIO ASTRONOMY OBSERVATORY
GREEN BANK, WEST VIRGINIA

ELECTRONICS DIVISION TECHNICAL NOTE NO. 132

Title: **SPICE: A Generalized Circuit Simulation Program Now Running on the Green Bank Lab MASSCOMP**

Author(s): Richard F. Bradley

Date: June 20, 1985

DISTRIBUTION:

GB	CV	TU	VLA
GB Library	CV Library	Library Downtown	VLA Library
W. Brundage	H. Hvatum	Library Mountain	P. Napier
R. Weimer	M. Balister	J. Payne	J. Campbell
D. Schiebel	S. Weinreb		
E. Childers	C. Burgess		
S. Srikanth	S-K. Pan		
C. Brockway	A. R. Kerr		
J. Coe	P. Siegel		
G. Behrens	M. Faber		
R. Mauzy			
R. Norrod			
R. Bradley			
R. Fisher			
F. Crews			
B. Peery			
R. Lacasse			

SPICE
A Generalized Circuit Simulation Program
Now Running on the Green Bank Lab MASSCOMP

Richard F. Bradley June 20, 1985

The circuit simulation computer program SPICE2 is currently running on the lab Masscomp. SPICE2 is a program that simulates the electrical performance of electronic circuits. The program will determine the quiescent operating point of the circuit, the time-domain response of the circuit, or the small-signal frequency-domain response of the circuit. The circuit is encoded by nodal connections on an element by element basis. The input syntax is a free-format style not requiring fixed columns.

The following analysis types are available.

DC ANALYSIS
- DC Operating Point
- Linearized Device Model Parameterization
- Small-Signal Transfer Function
- Small-Signal Sensitivities
- DC Transfer Curves

TRANSIENT ANALYSIS
- Time-Domain Response
- Fourier Analysis

AC ANALYSIS
- Small-Signal Frequency-Domain Response
- Noise Analysis
- Distortion Analysis

The following is a list of all circuit elements recognized by SPICE.

LINEAR ELEMENTS
- Resistor
- Capacitor
- Inductor
- Mutual Inductor
- Independent Voltage Source
- Independent Current Source
- Linear Voltage-Controlled Current Source

NONLINEAR ELEMENTS

- Nonlinear Voltage-Controlled Current Source
- Diode
- Bipolar Junction Transistor
- Junction Field-Effect Transistor
- Insulated-Gate Field-Effect Transistor

The program allows the user the model many types of transistors. The program supplies reasonable default values for circuit parameters that are not specified and performs a considerable amount of error-checking to insure that the circuit has been entered correctly. A beginning user need specify a minimal number of circuit parameters and the simulation controls to obtain reasonable simulation results.

Attached is a simple example of how a schematic diagram is entered into SPICE. Please contact me if you would like a SPICE demonstration analysis of YOUR circuit. Anyone interested in using SPICE should contact me for a users manual and helpful hints for getting started.

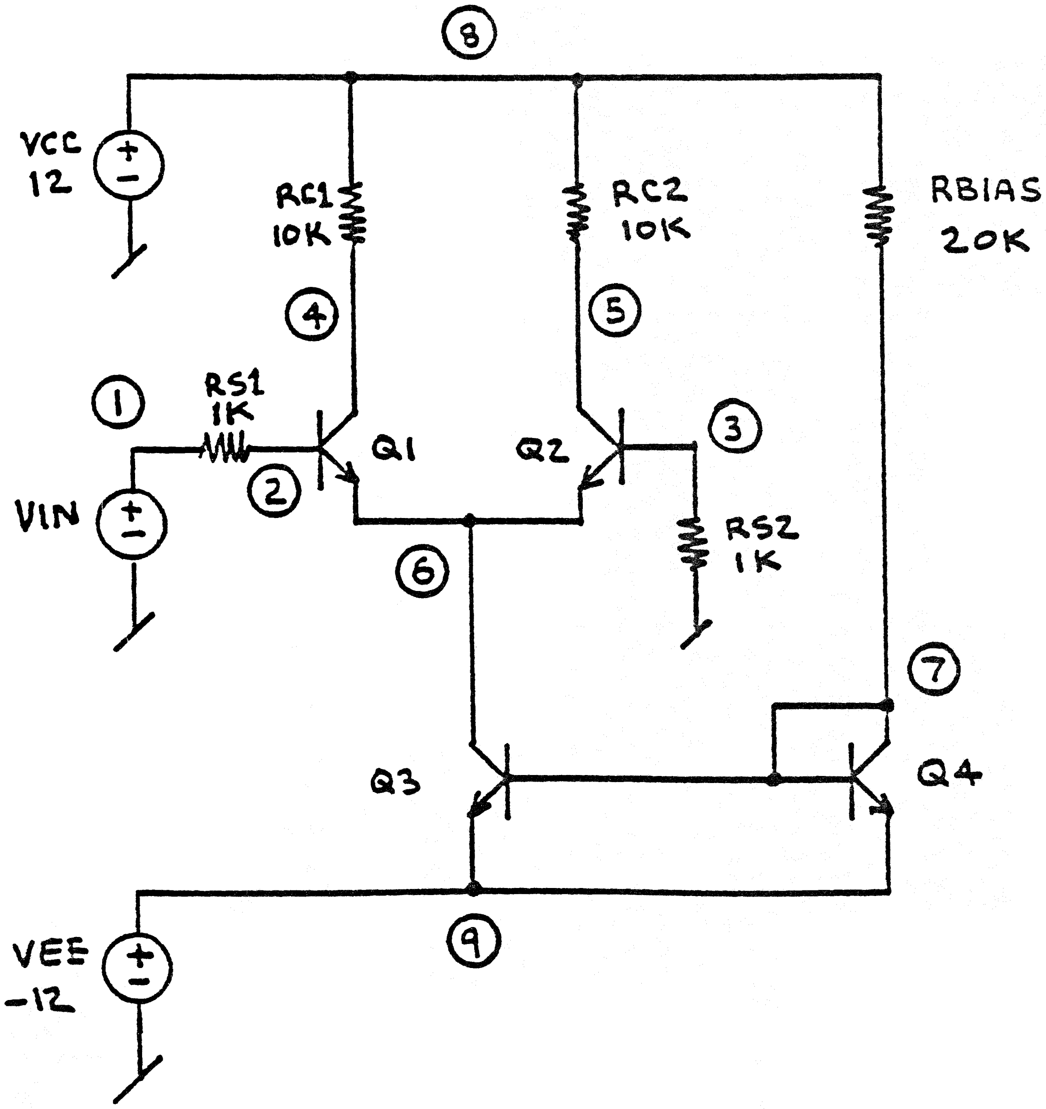

Fig. A1.1. Differential-Pair Circuit (DIFFPAIR).

```
DIFPAIR CKT - SIMPLE DIFFERENTIAL PAIR
.WIDTH IN=72
.OPT ACCT LIST NODE LVLCOD=2
.TF V(5) VIN
.DC VIN -0.25 0.25 0.005
.AC DEC 10 1 10GHZ
.TRAN 5NS 500NS
VIN 1 0 SIN(0 0.1 5MEG) AC 1
VCC 8 0 12
VEE 9 0 -12
Q1 4 2 6 QNL
Q2 5 3 6 QNL
RS1 1 2 1K
RS2 3 0 1K
RC1 4 8 10K
RC2 5 8 10K
Q3 6 7 9 QNL
Q4 7 7 9 QNL
RBIAS 7 8 20K
.MODEL QNL NPN(BF=80 RB=100 CCS=2PF TF=0.3NS TR=6NS CJE=3PF CJC=2PF
+   VA=50)
.PRINT DC V(4) V(5)
.PLOT DC V(5)
.PRINT AC VM(5) VP(5)
.PLOT AC VM(5) VP(5)
.PRINT TRAN V(4) V(5)
.PLOT TRAN V(5)
.END
```