

## Foreword

This electronics lab manual contains experiments designed to accomplish three goals: to give the student hands-on experience with electronic circuits, with the PSpice circuit simulator, and with modern data collection and documentation techniques. Each experiment will contribute to one or more of these goals.

The undergraduate electronics lab has traditionally focused on the instruments; namely, the oscilloscope, function generator, multimeter and power supplies. However, the look and feel of basic electronics instrumentation has changed as microcontroller-based scopes and instruments took over the market beginning in the mid 1980s. New features which are rapidly becoming standard include digital memories for instrument settings and measured data, smart interactions among the controls, semi- or fully-automated measurement sequences and networking of test instruments in computer-controlled test and measurement clusters. Documentation of measured results is now done directly through a printer interface in some cases, or in others by passing the data to a word processor or spreadsheet running on a computer dedicated to the test and measurement station. The undergraduate electronics laboratory has been upgraded to reflect these ongoing changes in the way electronic test and measurement is done, and especially the way the results are reported.

Electronic design has traditionally been a two-sided iteration between hand-worked analysis and laboratory verification and testing. Recent improvements in circuit simulation packages have made this process a three-way iteration. Beginning with a hand calculation of a proposed circuit design, the process now typically goes to iterative simulation and design correction. The laboratory testing and verification is then done on a circuit which is fully believed to work to specification (based on the simulator). Many of the experiments in this lab manual are intended to give the student working experience with a commercial version of SPICE (Microsim's PSpice), and to encourage a side-by-side comparison of the simulator's predicted results with the actual lab circuit performance.

The central feature of the lab stations is a personal computer (PC). Each station is equipped with its own PC, which is interfaced to the scope, the function generator and the digital multimeter over an IEEE-488 bus. This PC is to be used in two ways: for documenting experimental results, and for automated data gathering. Waveforms appearing on the scope are easily transferred to the computer in formats suitable for a word processor or a spreadsheet. Because the PCs are on the campus network, this data can also be passed to any student account. The PCs are equipped with a programming language (HP-VEE) for setting up automated experiments. As expected, this language includes iconic statements which control program flow and data manipulation, but also objects which control the scope, function generator and multimeter over the interface bus. Students in this lab are not expected to program in VEE, but eventually there will be several automated experiments designed to introduce the concept of automated test and measurement which has such great practical importance in industry.

Roger King  
Prof. EECS  
Electronics Lab Coordinator  
January 1998

Revision 2 of this lab manual adds mention of the recently acquired GPIB-interfaced power supplies, and corrects an error in Experiment 6, Fig. 2. A new experiment (Experiment 9) on quantization effects in the A/D and D/A conversion process is introduced. Students who will be taking the lab associated with Electronics II (Electronics Laboratory, EECS 3440) may want to keep this manual for reference in that course.

Roger King  
Prof. EECS  
March 1999

## Table of Contents

<b>I. Lab Reports</b> .....	1
<b>II. Equipment Description</b> .....	3
<i>Supplies and Equipment</i> .....	3
<i>Digital Storage Oscilloscope</i> .....	4
<i>Function Generator</i> .....	6
<i>Digital Multimeter</i> .....	8
<i>Laboratory Software</i> .....	10
<i>Safety, Grounding Arrangements and Signals</i> .....	13
<b>III. Using SPICE Simulation</b> .....	17
<b>IV. Experiments</b> .....	23
<i>Experiment 1 Using the Digital Storage Oscilloscope</i> .....	25
<i>Experiment 2 OP AMPs - A Basic Analog Building Block</i> .....	29
<i>Experiment 3 Diodes</i> .....	33
<i>Experiment 4 The BJT in the Common-Base Configuration</i> .....	37
<i>Experiment 5 The BJT in the Common-Emitter and Common-Collector</i> <i>Configurations</i> .....	41
<i>Experiment 6 SPICE Modeling of the JFET and MOSFET</i> .....	45
<i>Experiment 7 Characterizing Digital Logic Gates</i> .....	49
<i>Experiment 8 Free-Running (Astable) Multivibrator</i> .....	53
<i>Experiment 9 Analog/Digital Conversion</i> .....	55
<i>Experiment 11 DC Analysis of a Diode Circuit Using SPICE</i> .....	59
<i>Experiment 12 Analysis of a Diode Clipper Using SPICE</i> .....	61
<i>Experiment 13 Simulation of a Bridge Rectifier Using SPICE</i> .....	63
<i>Experiment 14 Diode Reverse Recovery as Observed in SPICE</i> .....	65
<b>V. Appendix - Data Sheets and SPICE Models</b> .....	67

