Bharat Muthuswamy, Andy Neureuther Fall 2005

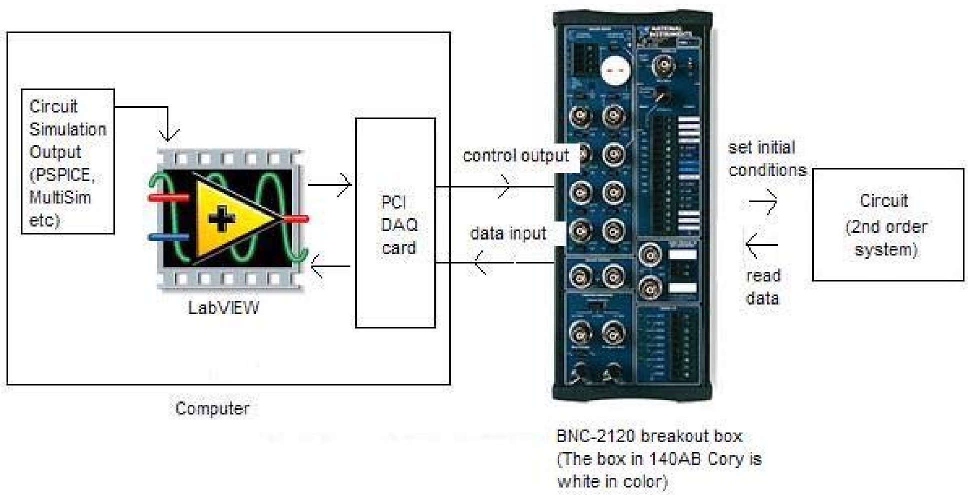
Project Overview

I. Objective

The student will develop a working knowledge of circuit simulation, computerized measurements and debugging of an electronic circuit of medium complexity.

II. Overview

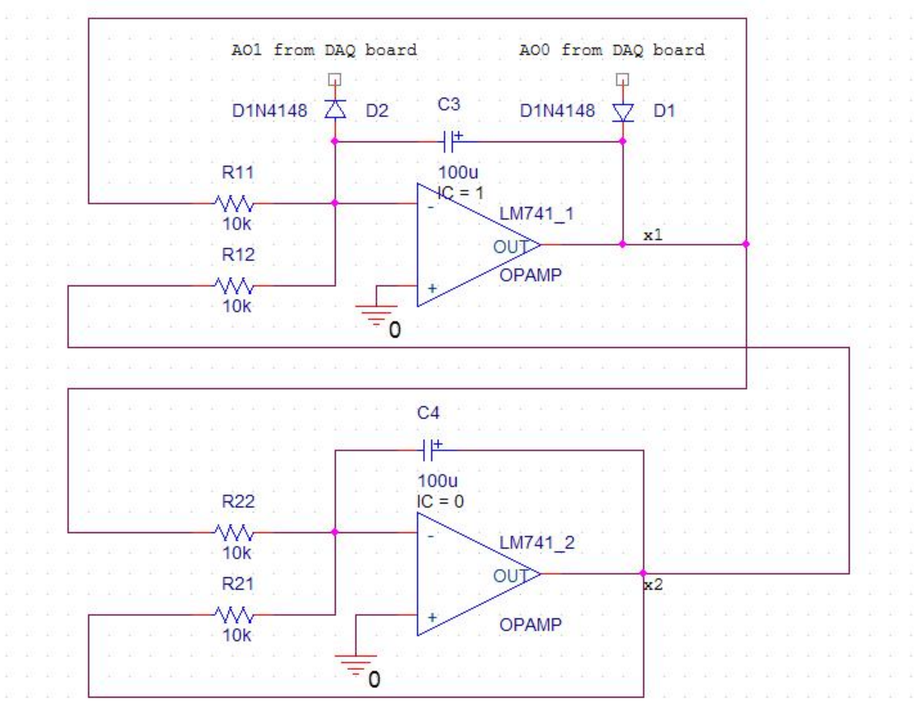
The emphasis is on developing these skills and demonstrating them in extending a basic integrated environment for automatically testing, an electronic Circuit Under Test. The project is based on PSPICE or MultiSim for circuit simulation, LabVIEW for automation, and the construction and debugging of a Circuit Under Test of medium complexity. These functional blocks are shown in Figure 1.



Block diagram of the basic EE40 Fall 2005 project. The idea is to compare simulation results to real-data using the LabVIEW instrumentation tool

II. Recommended Circuit Under Test

A good candidate for a Circuit Under Test on which to develop the integrated automatic testing is shown in Figure 2. The circuit consists of cross-coupled Op-Amps with capacitors in the feedback circuits. This circuit produces a pair of coupled rate equations with damping. These equations might correspond to the difference from equilibrium of concentrations in a chemical reaction or even step increases in populations of species in an environmental system. The coefficients of the rate equations can be set by adjusting the resistor values. By pre-charging the capacitors and switching them into the circuit various initial conditions can be applied.



For Fall 05 the circuit to initialize C3 has been simplified to set the voltage to a fixed value (see web site).

Figure 2. Cross-coupled Op-Amps. Note that each output feeds back current to the input of both op-amps and that the initial charge on either capacitor decays to zero.

You may wish to embellish this circuit at your own initiative to produce more interesting behavior or to model other systems of differential equations. As shown this circuit is very stable and well behaved. You may wish to add positive feedback to produce a more oscillator behavior or to add sources that act like forcing functions. Those who wish to be even more creative may propose your own candidate electronic circuit for the Circuit Under Test. In this case you should submit a one page proposal to your TA consisting of a circuit diagram and a description. If you are interested in the mathematics of this coupled 2nd-order set of equations as one of the TA's from the systems area.

III. Support Materials

Thee EE40 web site will contain helpful material including links to the following:

- EECS Server Remote Access Instructions
- PSPICE: Hambley Appendix D and CD-ROM
- LabView: Hambley CD-ROM (40 min) Tutorial

- LabView: Starting VI (with single output voltage/switch and single input)
- Hot-Spots: Running check list of problems to avoid on web site
- Word template for write up

IV. Time Line and Process

The duration of the project is three weeks and many will likely finish in two weeks. You and your partner will be given a circuit breadboard and a bag of parts including the Op-Amps and capacitors. You may assemble your circuit in advance and even test it in advance. For example, the UCSEE Laboratory on the 2nd floor of Cory Hall has equipment similar to the EE 40 Laboratory that could be used.

The suggested work plan is as follows.

Prior Project Lab #1: Read Appendix D of Hambley (especially Fig D.1, D.6 and D.11. Then run PSPICE either on your computer or on the EECS server. Make a simulation of your Circuit Under Test. Then select typical resistor and capacitor values. Save the output for a single probe to a .txt file of X,Y pairs. Complete the LabVIEW 40 min tutorial.

Project Lab #1: Build the Circuit Under Test and use the oscilloscope to view the signals versus time to be sure the circuit is in reasonable working order. Use the starting LABVIEW A/D converter as is to 1) check that the scope shows that initial condition is being applied correctly, 2) collect and display data on the output voltage versus time, and 3) make a plot from your .txt file appear on the screen with the measurement plot.

Prior to Project Lab #2: Sketch the basic extension to LABVIEW VI that you intend to build and debug to demonstrate your working knowledge of the basics of LabVIEW. It might include data collection, data display, comparison of measurement of simulation or control of initial circuit conditions. Sketch the extension of the Circuit Under Test that you intend to build and debug to demonstrate your working knowledge of hands-on electronics. This might include simple changes in R and C and the expected change in the circuit output, the addition of forcing functions, use of more complex feedback or application of a different type of an initial condition.

Project Lab #2: Build, test and debug the extensions to the Circuit Under Test and LabVIEW that demonstrate your working knowledge of simulation, automatic measurement and debugging electronic circuits. Continue on the build, test and debug of the extension that require significant exploration on your own and which qualify for bonus points.

Prior to Project Lab #3: Identify the simple changes of the LabVIEW VI and of your Circuit Under Test that your lab group has made or plans to make to demonstrate working knowledge of Lab VIEW and circuit debugging techniques. State what your lab group plans to complete on your own during the third lab session to earn bonus points. The areas are 1) applications for your Circuit Under Test that would interest other students, 2) extensions of LabVIEW (such as additional measurements, control of initial conditions, graphics, comparison with simulation, or 3) generalization of the circuit under test. Prepare a preliminary draft of your Project Report with simulation and measurement data already inserted. Use the Project Report template on the web site.

Project Lab #3: Complete debugging of all embellishments to the Circuit Under Test and LabVIEW. Demonstrate them to the TA. Make a screen capture of your final version of LabVIEW VI components and the data that they have taken and displayed. (Please use white background plots to save toner.)

V. Report

Your report should be submitted on-line to your TA for the project with 24 hours of the completion of your last lab. It may be as short as 2 pages plus attachments for the VI that you designed. Be sure to insert a screen capture of the SPICE schematic and of the measured data and SPICE simulation for your Circuit Under Test. The report should contain the following.

- Executive Summary stating your approach to building the integrated framework, your choice for the Circuit Under Test, if and where you added value to the experiment (for bonus points), and a highlight of a measured result. (2 sentences)
- Schematic diagram for your Circuit Under Test with actual component values.
- Explanation of how the circuit that you developed for applying the initial conditions works. (3 sentences)
- Screen shot of your VI showing data that has been captured from successful operation of your circuit and linking to the .txt simulation file.
- An analysis of discrepancies in measurement results between those of your integrated environment and those of analog instruments (scope) and your simulation. (3-5 sentences)
- Where you attempted to added value to the experiment describe the approach, the implementation and the outcome. (1-6 sentences). Include screen shots and schematics.
- Recommendations for further improving your experiment, this Laboratory Assignment and LABVIEW. (One sentence in each category)

VI. Grading

The Project Lab is worth 3 regular labs which is 60 points. Be sure to come prepared to each lab as the Prelab points will be in play as usual. The grading will be based on the correct operation and description of the following:

- Circuit Under Test (30%)
- Data collection (30%)
- Comparison with simulation (30%)
- Report (10%)

Completion of the above will be judged in part by your ability to conduct basic extensions such as changing R and C values and recording/discussing the new measurements and simulation results and changing the initial voltage level and polarity. These basic extensions are evidence that you have earned your Pilot's Licenses for LabVIEW and PSPICE.

You may earn extra bonus points for the Project through extensions. Using a baseball analogy, Straightforward Extensions are like singles and Major Extensions are like Home Runs.

- Straightforward extension of the circuit, LabVIEW measurements and SPICE studies will be worth 2 points in each up to a maximum of 6 points. These extensions need to be distinct from the basic extensions described above. Straight forward extensions are characterized by exploring new turf that requires exploring new features in Lab VIEW or new circuits. Straightforward extensions include the following.
    - Generalizing the LabVIEW VI for a second input data channel and displaying results.
    - Modifying the circuit with a third coupled Op-Amp.
    - Studying the forced response.
- Major extensions are of a caliber that **add significant value for the entire class** and are worth up to 6 points each up to a maximum of 12 bonus points. Major extensions might include the following.
    - Finding a scheme for initializing only the starting capacitor voltage (and allowing the Circuit Under Test to change it with time),
    - Finding (and documenting via PowerPoint for use in class) an application of coupled differential equations that fits the coupled differential equations,
    - Finding and testing a novel Circuit Under Test that is suitable for future class projects, or
    - Developing an interface between LabVIEW and SPICE that automatically determines the element parameters in SPICE that best match the experiment.