# Description

You will design and analyze via simulation and hardware implementation the circuit that you selected or was assigned to you for this project.

# Project components and grade contribution

The final grade for this project represents 100 points. **This accounts for 25% of the final grade** (note that the previous project 1 accounted for 10% of the final grade). The 100 points are distributed among the project 2 components as follows:

a) <u>**Project proposal report: 10%**</u> **(Due Feb. 16 2011)**
Write a report to describe the selected circuit. Describe briefly what the circuit will do. Use block diagrams or preliminary circuit schematic diagrams to help your description. Specify preliminary performance metrics for your circuit. Assemble an initial plan of action or roadmap with specific action items (AIs).

b) <u>**Midway milestone report: 30%**</u> **(Due Apr. 11 2011)**
This report should contain a detailed analysis of the proposed circuit. First, you should include a theoretical analysis of the main blocks. The analysis may include: derivation of expressions for input/output resistances/impedances, amplifications or gains, input/output loading, static and dynamic limitations of Op Amps, etc. Second, you should include detailed simulations with LTSpice. Include detailed schematic diagrams. Include also detailed discussions and interpretation of your design decisions and findings. The report should include a final section with references that you cite throughout your report. There is no page limit for this report. The report should be in conference paper format. The Latex and Word templates can be downloaded from the link provided in the Notes section below.

c) <u>**Final report: 30%**</u> **(Due May 6 2011)**
This report should continue to build on a refined version of the previous report. It should also contain a description of the experimental implementation and measurement as well as your interpretation of the results especially in comparison with the simulation data. There is no page limit for this report. The report should also be in conference paper format.

d) <u>**Hardware implementation and demonstration: 20%**</u> **(you may demonstrate the hardware any time)**
You should plan ahead of time for this part of the project. Once you finalized the circuit solution, you should search for component in the ECE component database (link provided in Notes). Missing components will have to be ordered through Jeff – so, plan ahead. It is encouraged that you implement also a PCB version of your final project. This should be done only after you verify your project on the protoboard. Links to online companies that do PCB manufacturing are provided in Notes section below. If you do only protoboard implementation, you may get only a maximum of half points for this project component.

e) <u>**Presentation in class: 10%**</u> **(Apr. 27 2011)**
You will create a PPT presentation and give a presentation in class.

# Notes

- IEEE Latex and Word templates for conference papers can be downloaded from here:
http://www.ieee.org/conferences_events/conferences/publishing/templates.html

- ECE parts inventory (created and maintained by Jeff):
http://www.ieee.ndsu.nodak.edu/parts_inventory.pdf

- Resources related to PCB manufacturing. This is not a complete list. There are many other tools and online websites. You should work with those you are already familiar with. Note that most of the PCB manufacturing companies offer significant discounts to students. Also, note that some integrated circuit companies provide free IC samples (for that you need to plan well ahead of time…).

Some CAD software for schematic/netlist entry and PCB layout and routing:
a) LTSpice can generate ASCII netlists for PCB layout. It can create a variety of netlist formats (see the LTSpice user manual). In particular, one example of such a format is ExpressPCB, which is free:
http://www.expresspcb.com/index.htm
b) Multisim, Ultiboard of NI (available in the computer cluster room):
http://www.ni.com/multisim
http://www.ni.com/ultiboard
c) OrCad (available in the computer cluster room):
http://www.cadence.com/products/orcad/pages/default.aspx

Once you have done your PCB layout, you can "ship" it (you actually send specific files) for manufacturing to online companies. A popular PCB manufacturer:
http://www.4pcb.com
More info on PCB tools:
http://www.olimex.com/pcb/dtools.html
http://www.freepcb.com
http://www.winqcad.com/index.html