

ECGR 3131, Fall 2019 Project 2 – MOSFET Amplifier

Report Due by Tuesday Dec 15, 2020

1. Given and Required Specifications

The design of the amplifier shall be performed using the CD4007UBE Dual Complementary Pair Plus Inverter IC Package. This is an IC designed for use as a digital inverter, but due to hardware restraints, the N–MOS transistors inside the IC will be used for the amplifier. Specifications measured previously for the ECGR3155 labs can be taken with a grain of salt. New specifications for the CD4007UBE chip are given below. If you find the values differing from expected, please consult TA or Professor for additional instruction. For the PSpice simulation, a model for the IC is listed at the end of assignment. The model is courtesy of California State Polytechnic University. Please consult the data sheet for wiring specifications. Minimum and maximum operating conditions can also be found in the datasheet.

Given Specifications for the CD4007UBE:

- $V_T = 1.40 \text{ V}$
- $\mu_0 C_{ox}(W/L) = 600 \mu\text{A}/\text{V}^2$
- $|V_A| = 200 \text{ V}$

Specifications for Amplifier:

- $A_V = v_O/v_I \geq 10 \text{ V/V}$
- $R_L = 20 \text{ k}\Omega$
- $V_{DD} = 15 \text{ V}$ (rail voltage)
- Power Consumption $= V_{DD} * I_{DD} \leq 75 \text{ mW}$
- Swing $\geq 7 \text{ V}$ pk to pk
- $R_{\text{Input}} = \text{any value (must find)}$
- $R_{\text{Output}} = \text{any value (must find)}$
- Perform within 5% of calculated values.

Requirements for Project:

- One Partner Allowed
- Hand Analysis
 - DC Biasing
 - AC vs DC Load Line
 - AC Gain
 - R_{Input}
 - R_{Output}
- PSpice Simulation of Amplifier
 - DC Biasing
 - AC Gain
 - Transient Response
 - R_{Input}
 - R_{Output}

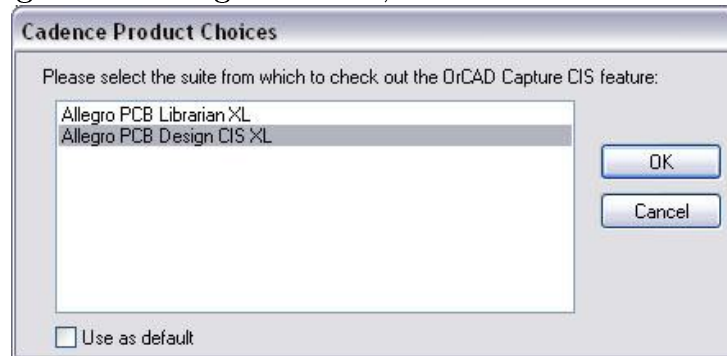
- Build and Test in Lab
 - DC Biasing
 - AC Gain
 - Transient Response
 - R_{Input}
 - R_{Output}
 - Must demonstrate to TA or Professor
- Write a Technical Report in the form of IEEE Journal Publication

2. Building the circuit on P-Spice

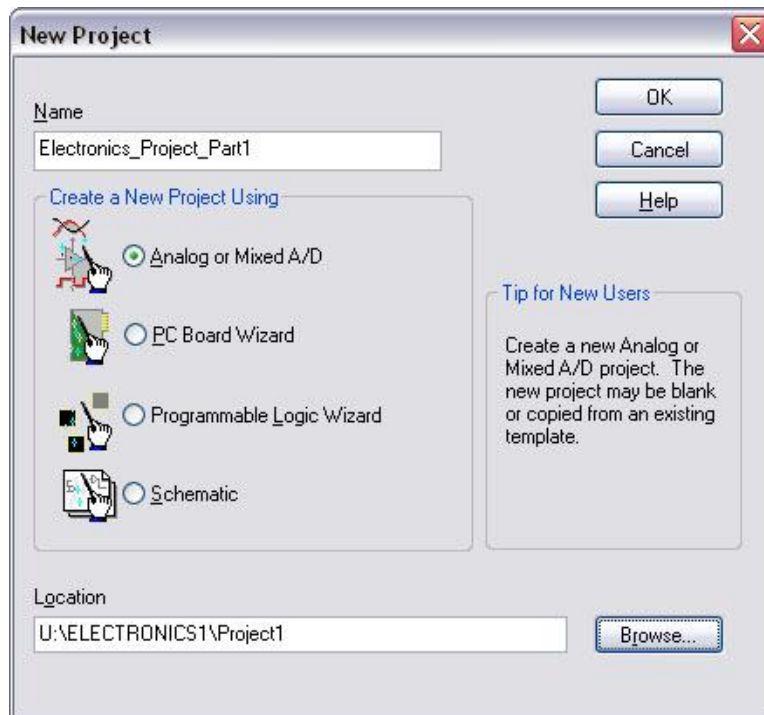
2.1 Getting started

To run Capture:

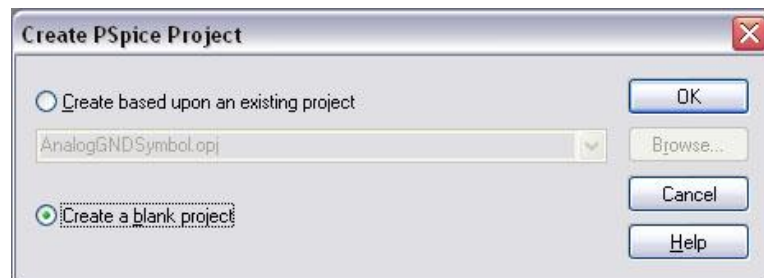
- Select Start → All Programs → Mosaic XP → Engineering → Electrical → Cadence SPB 16.01 (incl P-Spice) → Design Entry CIS.
- Select “Allegro PCB Design CIS XL”, then click “OK”



- File → New → Project.
- Choose “Analog or Mixed A/D”




- Type in the name of the new project and specify the location where it will be saved. Be sure to save it in a subdirectory (for instance, U:\ECGR3131\Project_2), because PSpice cannot re-open files that have been stored in the root directory and your project will be lost. Do not use spaces when creating a path to your project.
- Then you will be prompted to create a PSpice project. Choose “Create a Blank Project” and click ‘OK’.

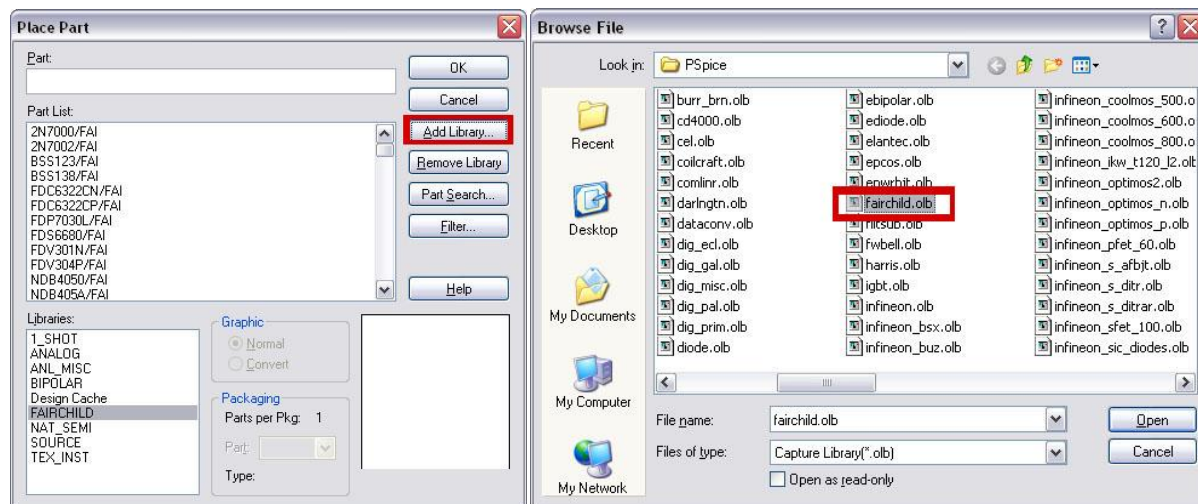


2.2 Drawing the circuit

The first step in drawing the circuit is acquiring the parts libraries. You do this by opening the Place Part browser and then choosing the Add Library option. You can invoke the Place Part browser in one of the following ways.

- Clicking on the Place Part icon () located on the right-hand toolbar.
- Pressing shift + p.
- Choosing *Place* > *Part* from the Capture CIS menu

To add the libraries click the Add Library button. This action opens a Browse File. This is the default directory for the PSpice libraries. The libraries you will need are analog.olb, source.olb, and fairchild.olb. All the three libraries can be found in the PSpice. To add a library, double click on it. The library will appear in the Libraries window in the Place Part browser. Add all three libraries and then click *OK* then The X, close button, to close the Place Part browser.



In addition to the standard libraries, you will need to add the custom CD4007 library provided by California State Polytechnic University. Go to Moodle and download the following files into your project directory. You will then need to add the library as before by first going to the project directory and then selecting the CD4007.OLB library.

Files: “cd4007.lib” and “CD4007.OLB”

Directory: “u:\Electronics1\Project2\

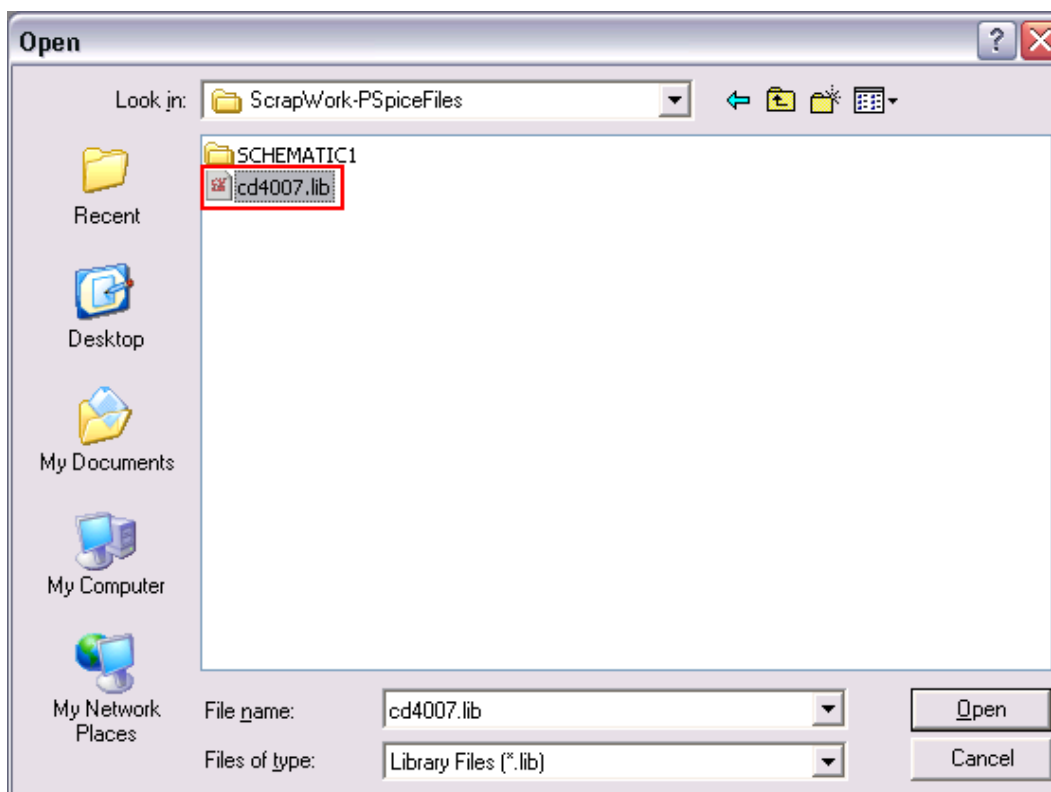
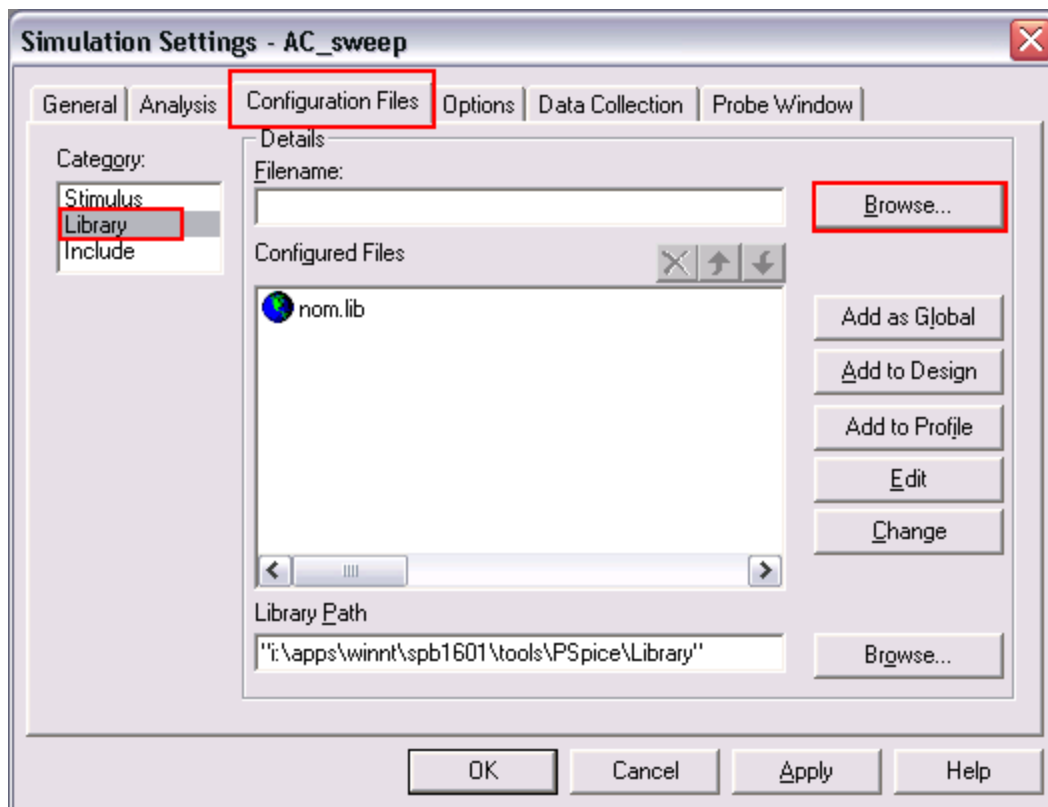
After adding all the necessary libraries you will be able to select and add parts to the schematic.

- a) Reopen the Place Part browser. To place a part, first highlight the library where the part resides, and then select the part in the Parts List and click *OK*. For instance to add a resistor, highlight ANALOG in the Libraries window then in the Part List window click on R for resistor and click *OK*. The part attaches itself to the cursor and is ready to be placed. The part can be placed by left clicking on the schematic, try it. To add a second resistor simply moves the cursor to another area of the schematic and left click again. The place part mode can be terminated by right clicking and choosing the End Mode option, then you are free to choose another part to place or move on to a new task. The Esc key can also be used to terminate modes.

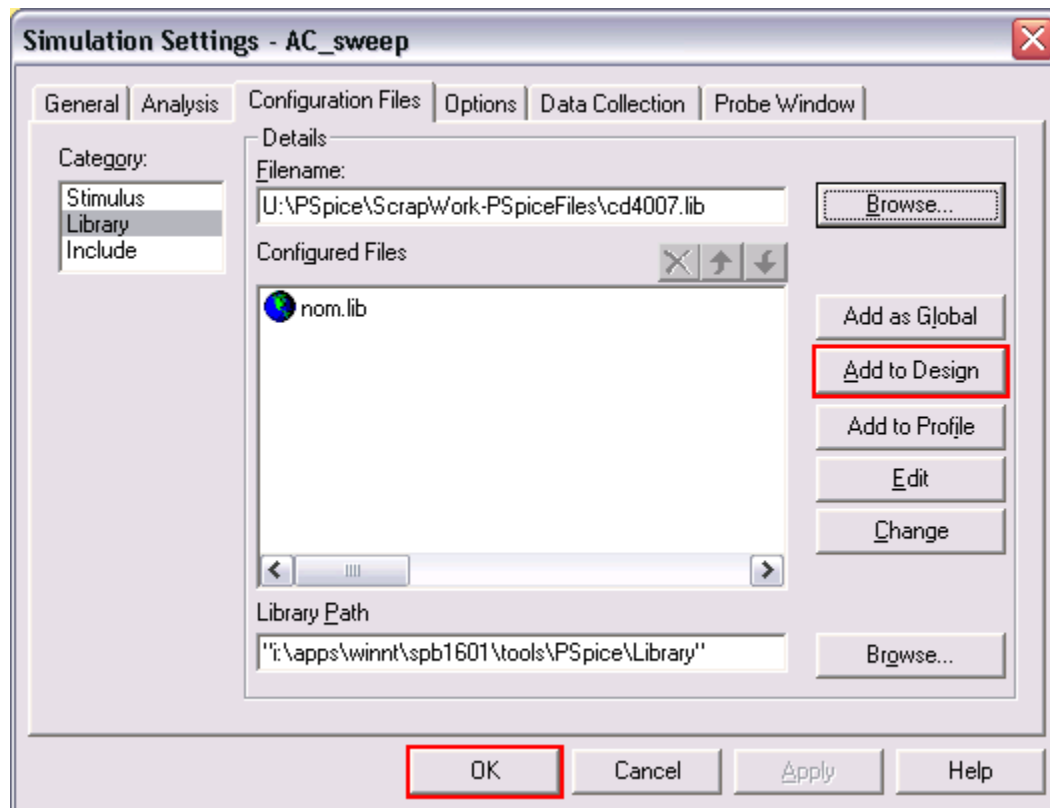
- b) Using the same procedures outlined above add the following parts to your schematic:
- Capacitors C and Resistors R(from ANALOG library).
 - cd4007_NMOS (from CD4007 library)
 - Power sources VSIN, VAC and VDC (from source library).
 - Add a Ground by clicking the Place Ground icon on the right hand toolbar or pressing shift + g and choose 0 from source library.
- c) Create a new simulation profile by selecting the menu from **PSpice** tab. Give some name for the simulation profile.
- d) The name and value of circuit elements can be modified by double clicking them. To change the name of R1, double click the name. To change to value of R1, double click on the current value and then input a new value. For the circuit above, use $R_L = 20\text{ k}\Omega$.

2.3 Configuring Include Files


Mosaic has Pspice configured to reference a vast library and model file collection in the I drive, where Pspice is installed. These references are included in the nom.lib file which you should **NEVER** edit. When you want to include a custom Pspice library with custom models, you have to include the reference to the model file as a separate include file for the project only. This can be done by going to the Simulation Settings and clicking on the Configuration Files Tab. Once here, select Library and click Browse to select the cd4007.lib file.



After selecting the cd4007.lib file, you need to add the file to the Design. Click the Add to Design button and click ok.



2.4 DC Analysis

From the PSpice tab, select edit simulation profile, and set the simulation profile to transient. Now run the circuit by clicking on the  icon. The simulation annotates the bias points on the circuit. Compare the simulated DC currents and voltages to the values obtained using your hand calculation. Note that the values may not update if you leave the Allegro Simulator open and proceed to run the simulation again. Make sure to close out the simulator between runs.

Include a schematic annotated with the simulated DC voltages and currents. Also include a table comparing the calculated DC voltage and currents to the simulated values and the ones obtained during the lab testing of the circuit. Show the percent error. Explain any differences in the values.

2.5 AC Analysis

Replace the VSIN with VAC if used and set the AC voltage to 1V. Select AC Sweep from the edit simulation tab, set start and end frequencies to 10 Hz and 10 MHz

respectively. The abbreviation MEG is used in PSpice for MHz. Make sure to add enough points per decade for a smooth plot. Delete both the markers. Select dB magnitude of voltage marker from PSpice → markers → advanced analysis. Run the circuit. Find the peak voltage and lower 3dB frequency point from the plot. There will not be an upper 3dB The AC analysis is also the point where the input and output impedance can be obtained. Using Ohm's Law, the input and output impedance can be found using the input and output voltage and current.

Include a plot of Gain vs. Frequency in Magnitude and dB in the report. Label and value the F_{-3dB} low and max gain. The values should include both the gain and the frequency. Be sure to include enough points per decade for a smooth plot.

Include a plot of Phase vs. Frequency in the report. Label and value the phase where the dominant f_{-3dB} point occurs. Be sure to include enough points per decade for a smooth plot.

Include a plot of R_{in} vs. Frequency to the report. Label and value the R_{in} . Explain why R_{in} increases as frequency decreases. Be sure to include enough points per decade for a smooth plot.

Include a plot of R_{out} vs. Frequency to the report. Label and value the R_{out} . Explain why R_{out} increases as frequency decreases. Be sure to include enough points per decade for a smooth plot.

Include a table comparing the calculated voltage gain, input impedance, and output impedance to the simulated values and the ones obtained during the lab testing of the circuit. Show the percent error. Explain any differences in the values.

2.6 Transient Analysis

Replace the VAC with VSIN if used. Edit the VSIN source to set the frequency to the peak gain frequency, VAMPL to 1 mV, and VOFF to 0V. From the PSpice tab, select edit simulation profile, set the simulation profile to transient and set the run time to span over two periods of the peak gain frequency. Set your step size small enough to get a smooth sine wave. Place voltage level markers at the input and the output terminals of the circuit. Now run the circuit by clicking on the icon. Calculate the gain from the simulation. Edit the VSIN source to the appropriate amplitude to check the swing of the amplifier.

Include the plot in the report of the gain at 1 mV. Also include a plot showing the swing of the amplifier.

3. IEEE Journal Publication Style

The IEEE Journal Publication style can be found on the IEEE website using the link below. The website includes resources for references, images, titling, and even templates that may be used to create the paper. The paper should include various sections such as:

- Introduction
- DC Analysis
- AC Analysis
- Results
- Areas for Future Research / Improvement
- Conclusion
- References

[IEEE Journal Publication Style Website](#)

4. Results to Include

The plots and tables previously asked for during the tutorial should be included in the results section. Discussion about what is happening with the plot or table should be included. The following about the frequency response should also be included in the results section for both simulated and measured.

Range of frequency of operation: _____ Hz to _____ Hz

3dB Bandwidth of the circuit: _____ Hz

Peak Gain Frequency: _____ Hz

5. Conclusion

A discussion of the circuit and how it performed should be included in the conclusion. The following questions should also be addressed:

Why is there a difference between the hand calculated gain, the simulated gain, and the measured gain?

At what value does the swing become unsymmetrical and why is that?

How did you approach solving this design problem (remember to write in passive voice)?

6. PSpice Model

As requested by several students, the following is an update showing what the model inside PSpice should be:

Filename: "cd4007.lib"

```
* PSpice Model Editor - Version 16.0.0
```

```
*$
```

```
* Model Created by California State Polytechnic University
```

```
* http://www.csupomona.edu/~bolson/web\_pages/how\_to\_spice.htm
```

```
.model nnMOS NMOS (LEVEL=2 VTo=1.4 Kp=.6m LAMBDA=0.005)
```

```
*$
```

```
.model ppMOS PMOS (LEVEL=2 VTo=-1.0 KP=.6m LAMBDA=0.01)
```

```
*$
```