SPICE SIMULATION TUTORIAL

DESIGN ENTRY TOOL---

This tutorial will show you how to open, modify and simulate a project using the Cadence simulation tool. The tutorial is based on four parts. Part I shows the basics of opening, modify and simulate a project based on SiPM device model. Part 2 expands the design adding an amplifier. Part 3 further expands the design adding a comparator. Finally a parametric simulation will be carried out both for the amplifier and discriminator.

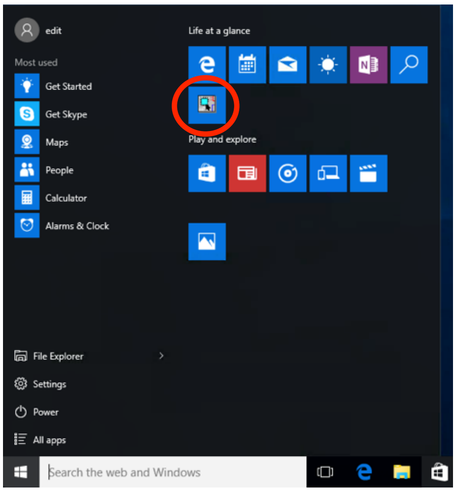

Figure 1: Cadence Design Entry Tool

Open the **Design Entry Tool**

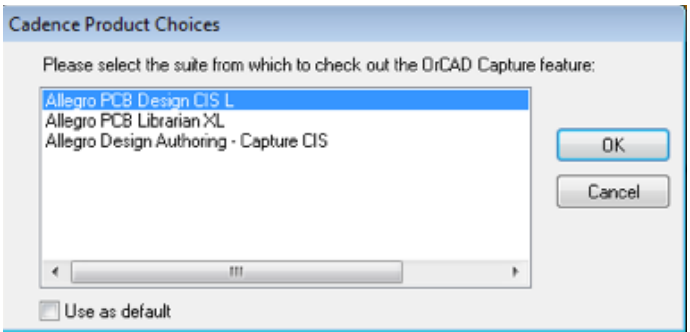

Figure 2: Licence Dialog box

Click **OK** to accept

SiPM MODEL RESPONSE SIMULATION ----------------------------

File → Open → Desktop → EDIT_SPICE → EDIT_I_L → SiPM_SIM → Open

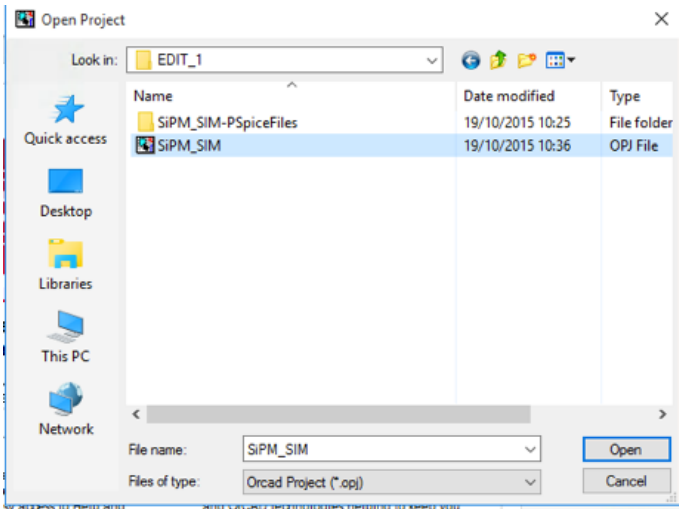

Figure 3: Open the project

Expand sipm_sim.dsn and Open SiPM_MODEL

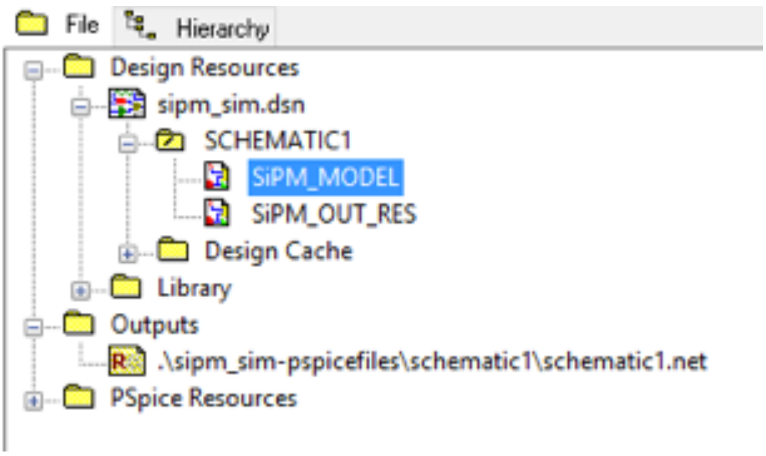

Figure 4: Project Hierarchy window

NOTE: we setup the generator to generate an output charge of 100 fC.

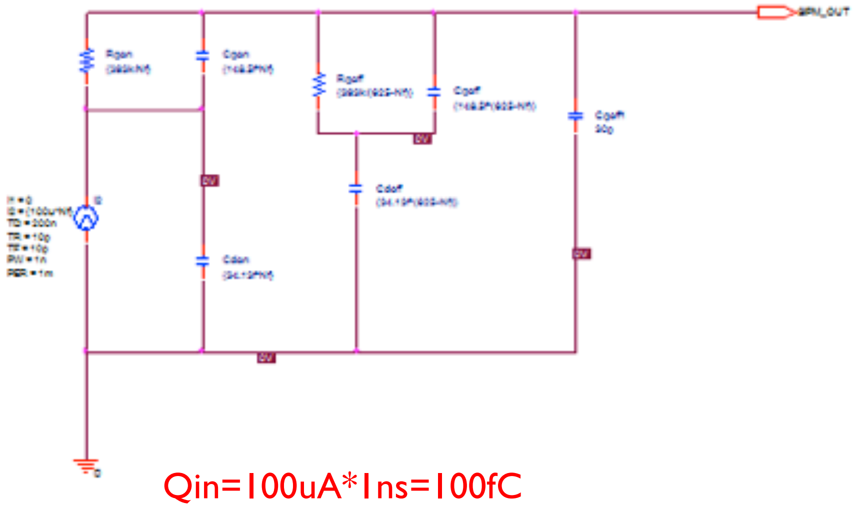

Figure 5: SiPM model

Open Edit Simulation Settings (if the window do not show up try move aside the main window or have a look on the task bar)

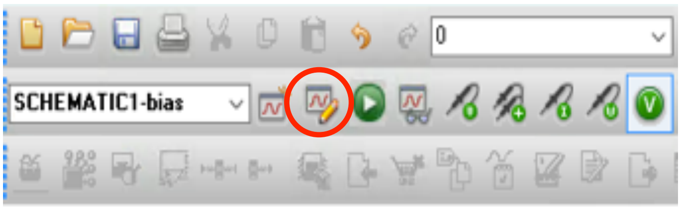

Figure 6: open Simulation Profile panel

Accept and close the window.

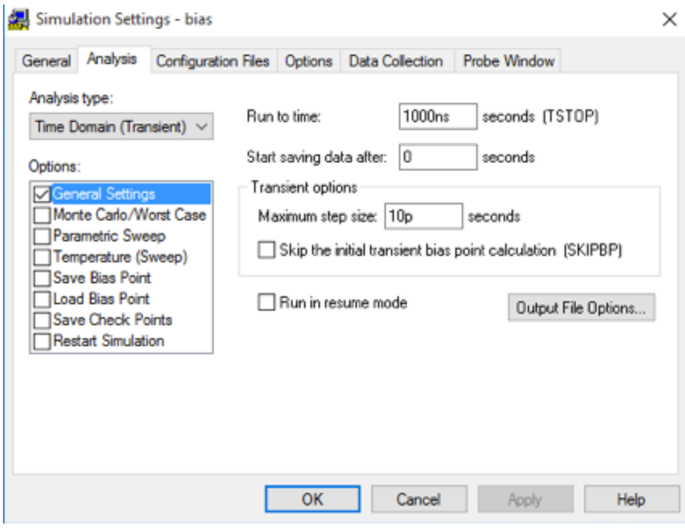

Figure 7: Edit Simulation Profile window

Launch the simulator

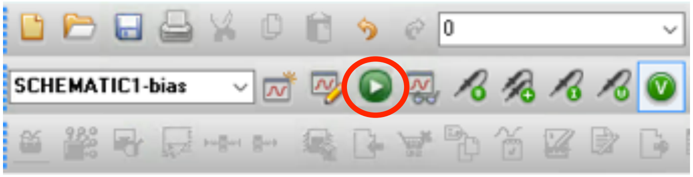

Figure 8: run simulation

The **Allegro AMS Simulator** window shows up reporting some errors

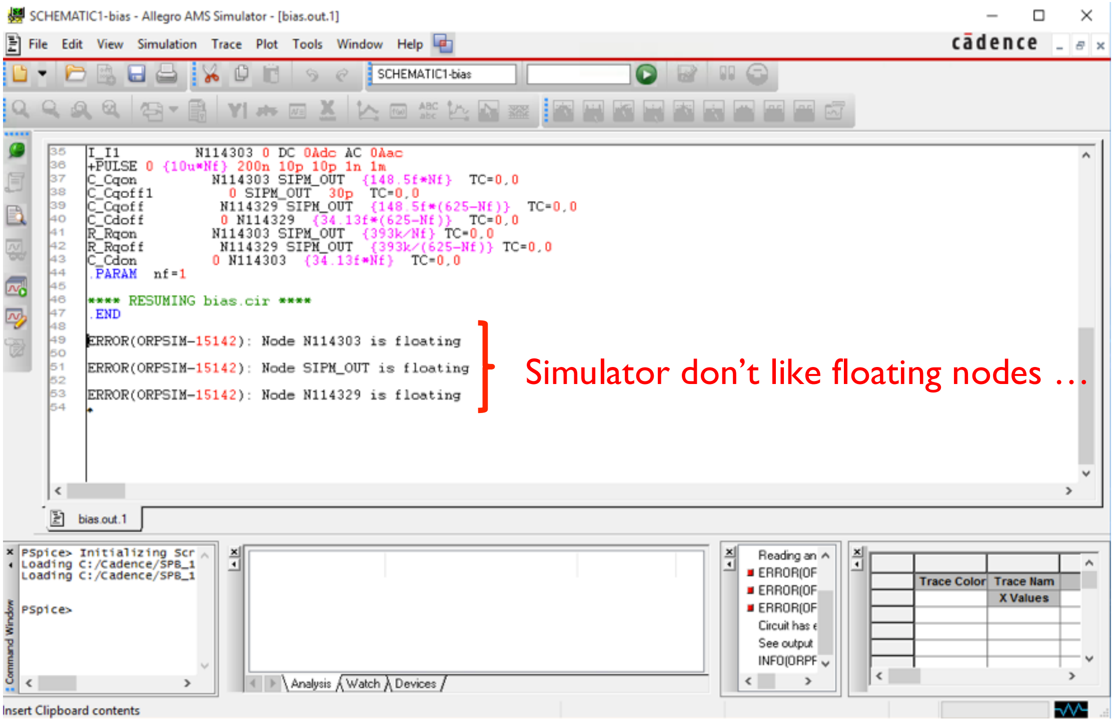

Figure 9: Allegro AMS Simulator window

Open the SiPM_OUT_RES, it's empty, so because SiPM_OUT port has no connections the simulator detects open nodes then generating error messages. In the following we'll fix them and re-launch the simulator.

Figure 10: Spice Tools window

Open the **Place Hierarchical Port** tool and add the PORTRIGHT-R port to the SiPM_MODEL sheet.

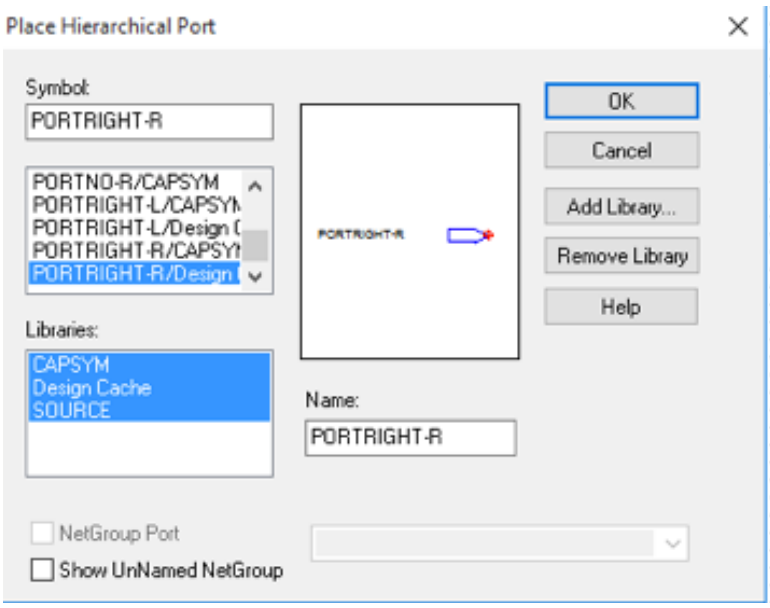

Figure 11: Place Hierarchical Port window

Change the name of the port to match the output port of SiPM_MODEL sheet (i.e. SiPM_OUT)

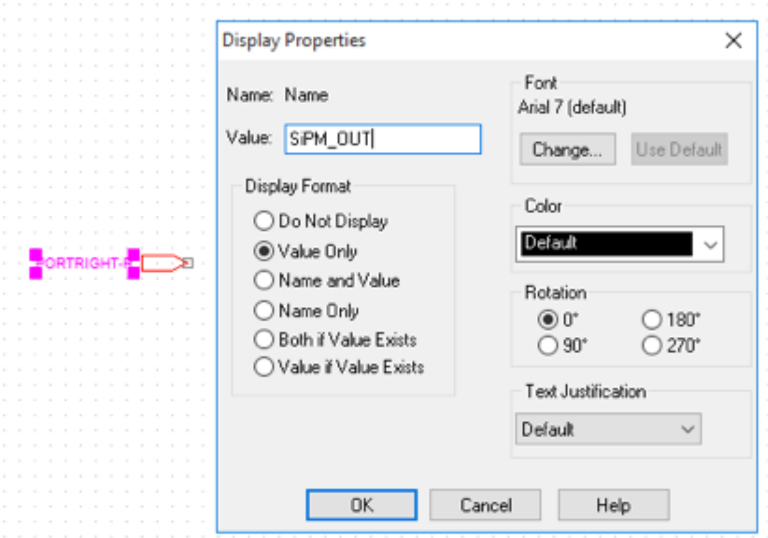

Figure 12: Output Port change value

From the **Design Cache Components** add a resistor to the sheet

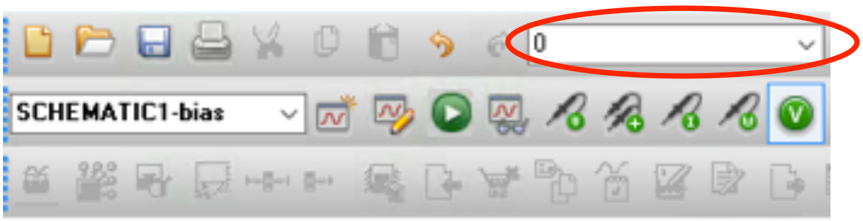

Figure 13: Design Component Cache (every component used in the design is available in the component cache)

Finally copy the GND symbol from previous sheet and connect the components using "Wire" tool

Figure 14: Wire tool

The final circuit is shown in Fig. 15 (change the resistor value to 50 ohm).

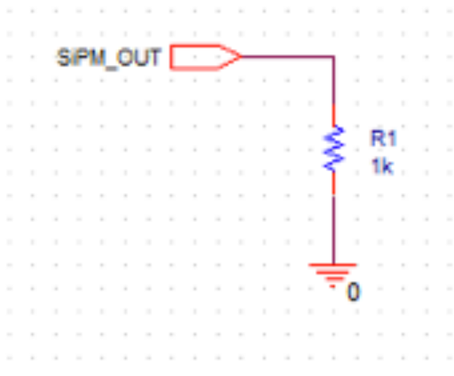

Figure 15: SiPM_OUT_RES sheet circuit

Re-launch the **Simulator**. Now simulation should run smoothly and **Allegro AMS Simulator** window should shows the SiPM output signal (a Voltage Probe is supposed to be connected to the wire after the input port in the SiPM_OUT_RES sheet or before the SiPM_OUT poer in the SiPM_MODEL sheet) as shown in Fig. 16. Open the Simulation Profile tool and adjust time parameters to center and expand the signal.

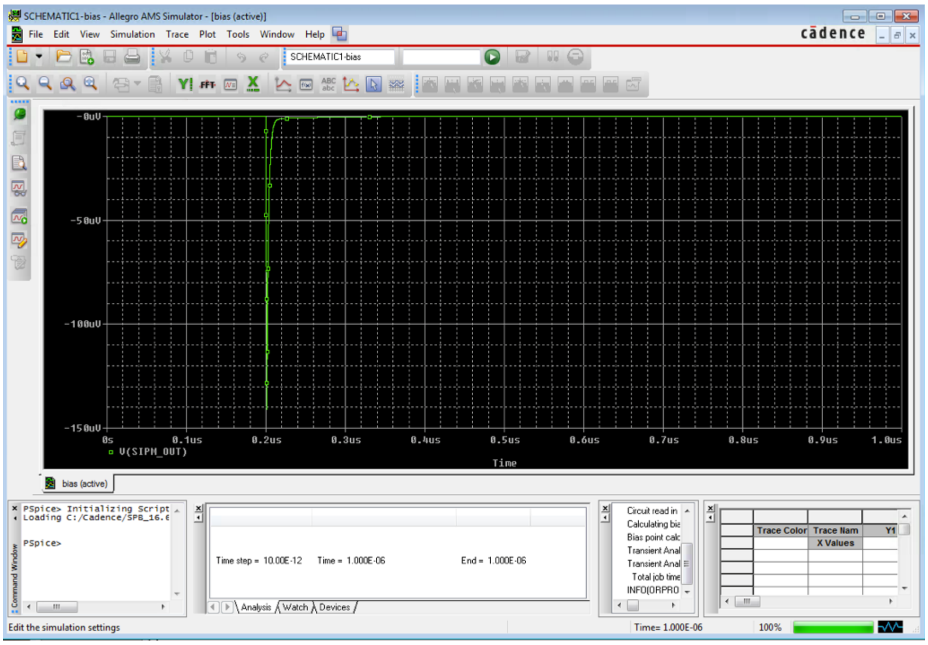

Figure 16: SiPM output signal

SiPM OUTPUT CHARGE MEASUREMENT -------------------------

Click on the **Component** section of the bar and select the FPOLY component from the **Analog Lbrary**. Build the circuit shown in Fig. 18 (capacitor can be selected from **Component Cache**)

Figure 17: Open the Component section

FPOLY → analog library

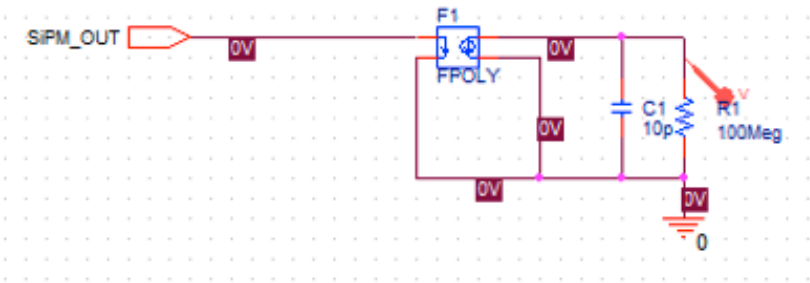

Figure 18: Circuit to measure the SiPM output charge

Run a simulation; now the **Allegro AMS Simulator** should shows the signal shown in Fig. 19.

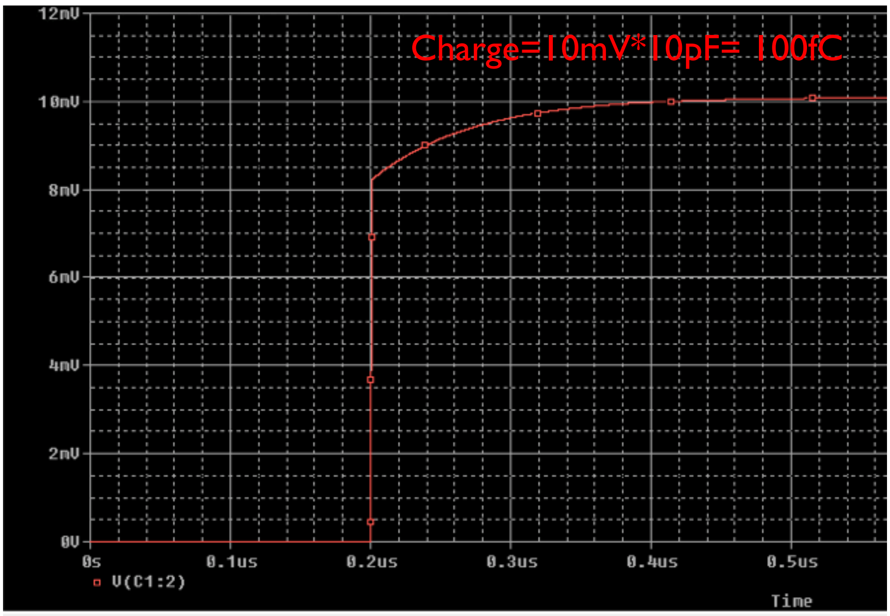

Figure 19: Output of the charge measurement circuit

AMPLIFIER RESPONSE--

Open the EDIT_2_L circuit:
File→Open→Projects→Desktop→EDIT_SPICE→EDIT_2_L→SiPM_SIM→Open

Complete the circuit according to Fig. 20.

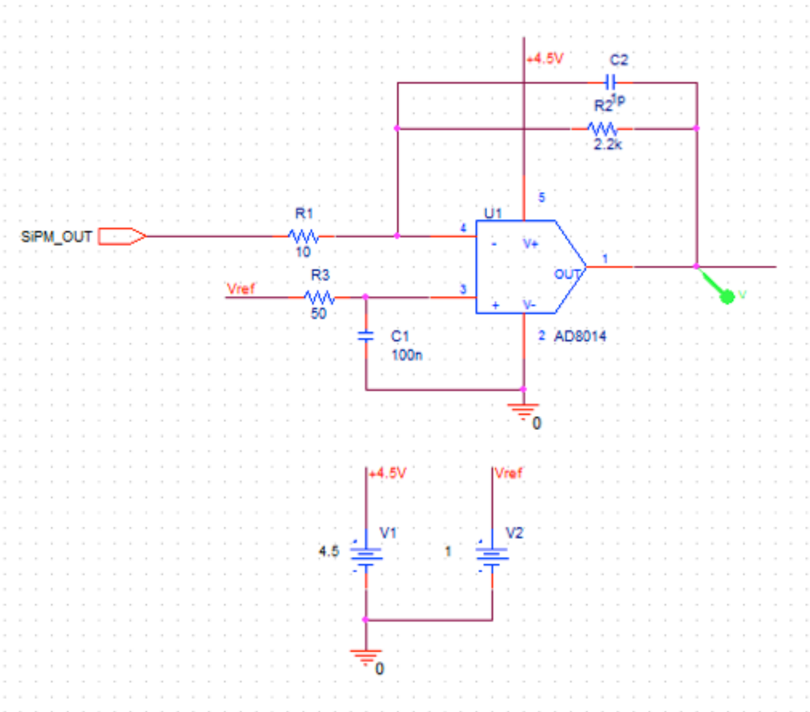

Figure 20: circuit for amplifier response simulation

Before simulating the circuit the AD8014 must be added to the library.

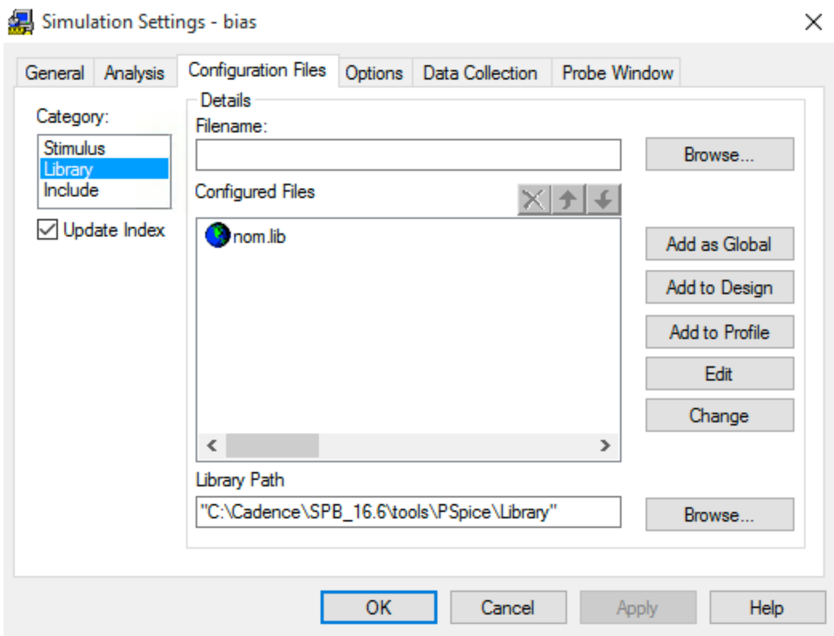

Figure 21: Simulation Setting - Library Configuration Files

Browse to EDIT_SPICE→PSPICE_LIB→OPAMP_GIULIO, select AD8014 and Open

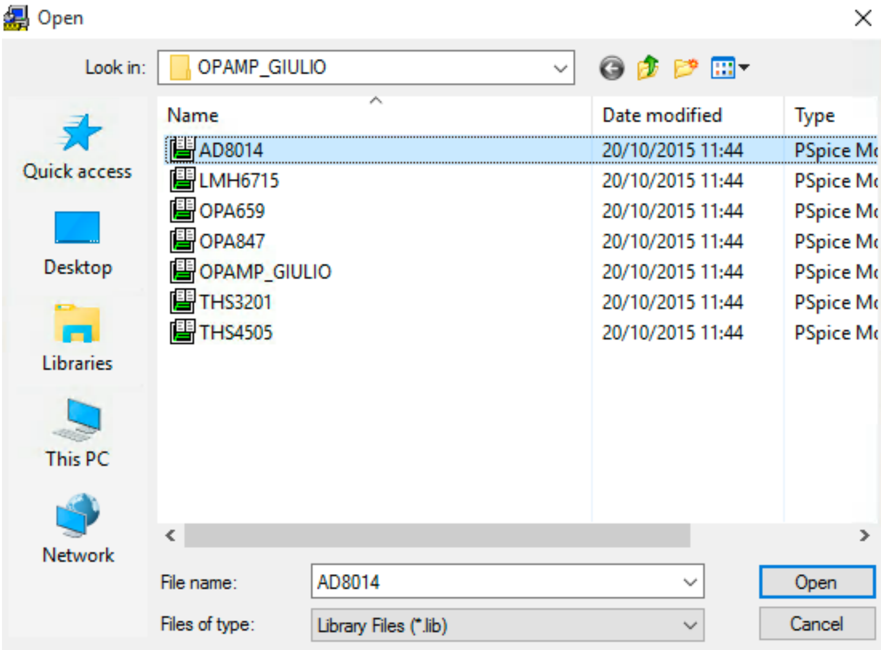

Figure 22: Open Library

Finally select **Add as a Global**, **Apply** and **OK**.

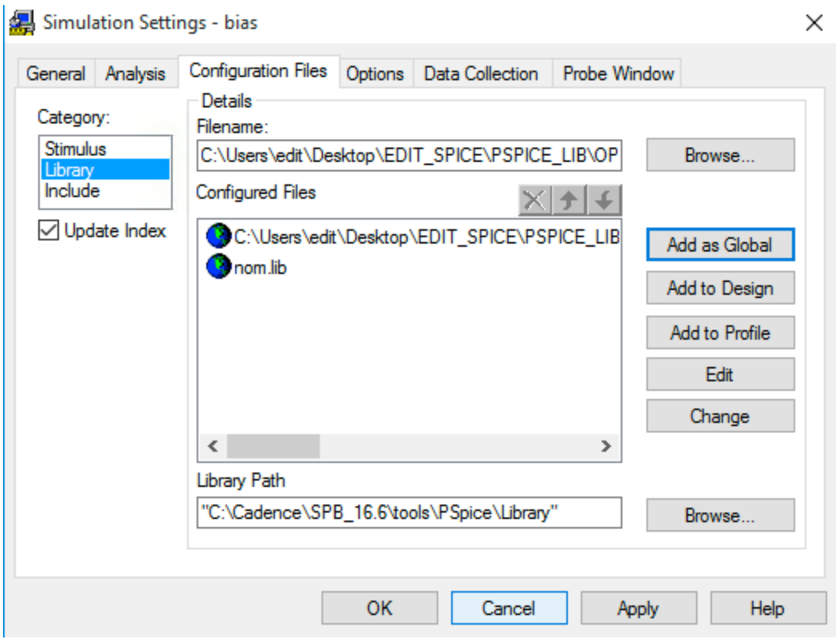

Figure 23: AD8014 component added for simulation

Simulate the circuit and calculate the gain V_{OUT}/Q_{IN}.

DISCRIMINATOR RESPONSE--

Open the EDIT_L3_P project:
File→Open→Projects→Desktop→EDIT→EDIT_3_L→SiPM_SIM→Open

Complete the DISCRIMINATOR sheet according to the circuit shown in Fig. 24 and simulate.

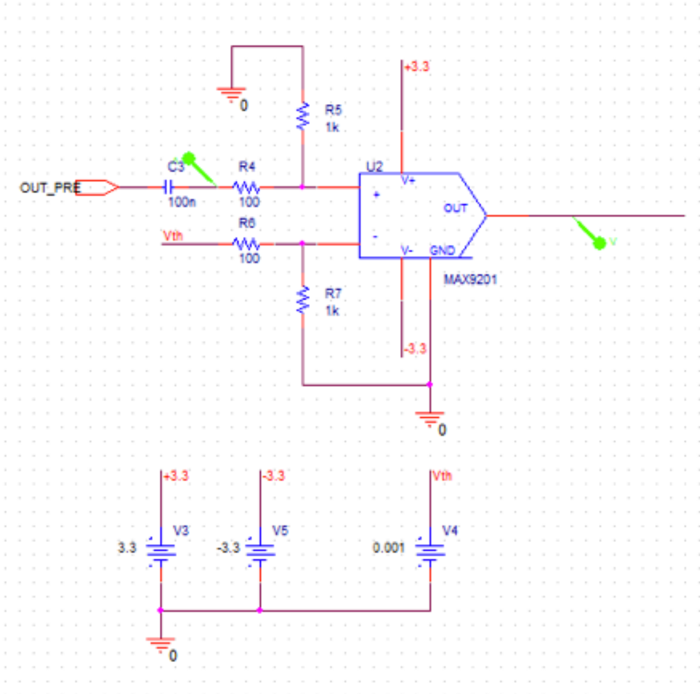

Figure 24: Comparator Circuit

N.B. before simulation the MAX9201 must be added to the Library File using the same procedure described to add the AD8014 amplifier.

As before use the Simulator Setting tool to expand and center the output signal as shown in Fig. 22.

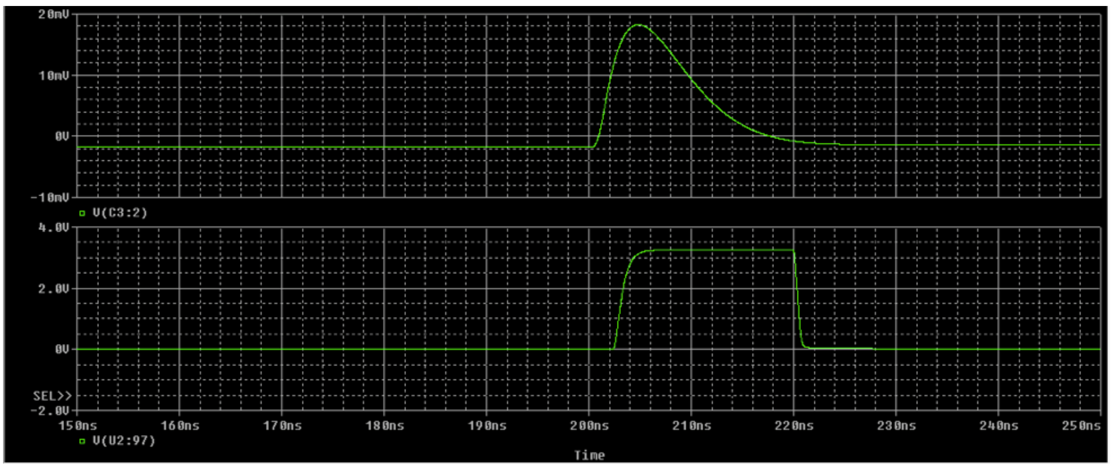

Figure 25: Amplifier and Discriminator outputs

PARAMETRIC SIMULATION ---

Open the EDIT_2_L_P project:

File →Open→Projects→Desktop→EDIT_SPICE→EDIT_2_L_P→SiPM_SIM→Open

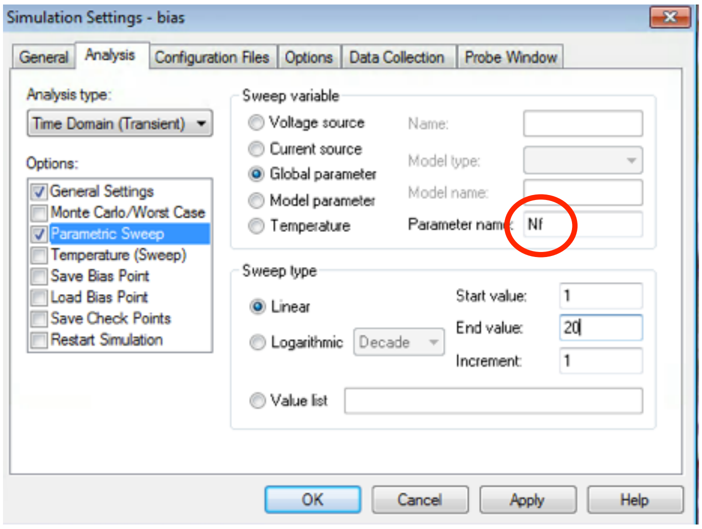

Figure 26: sweep parameter in Simulation Setting window

Click **Apply** and **OK** and run a simulation.

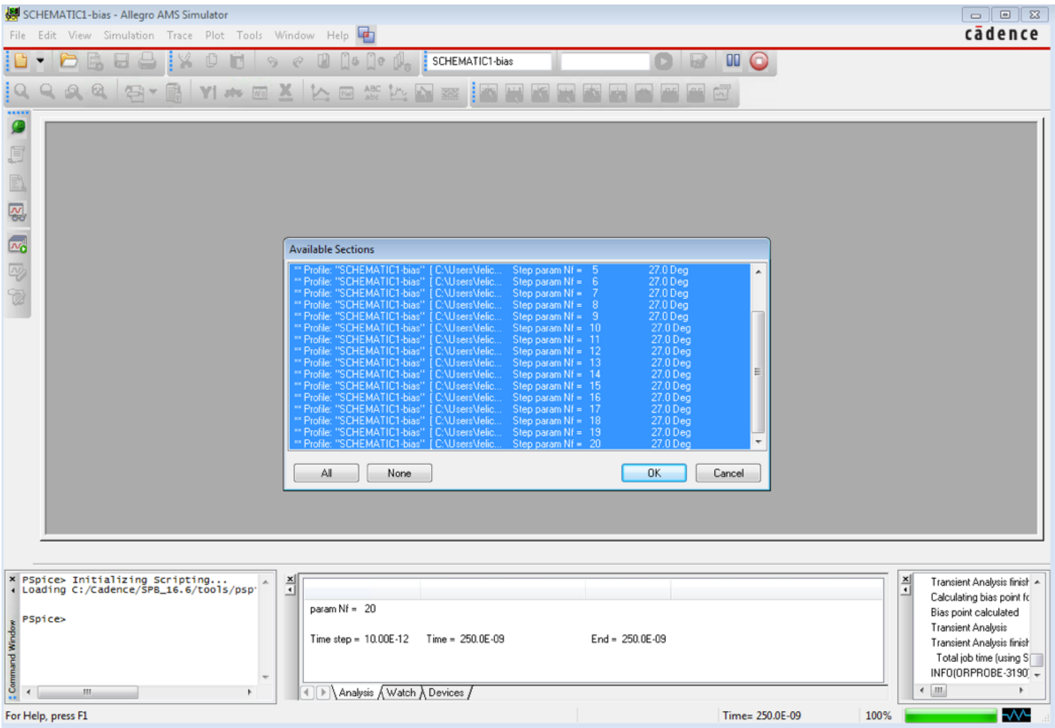

Figure 27: Allegro AMS Simulator output window (the number of files corresponds to the number of points in the Simulation Settings)

Click **OK** to accept. The **Allegro AMS Simulator** will show a family of signals corresponding to a SiPM output charge in the range 100 – 2000 fC (NB: the sheet must be open).

Finally open the EDIT_3_L_P project:
File→Open→Projects→Desktop→EDIT_SPICE→EDIT_3_L_P→SiPM_SIM→Open

Set a 30 mV threshold, run a parametric simulation and explain the result.