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**

Cadence Product Choices	
Please select the suite from which to check out the OrCAD Capture	e feature:
Allegro PCB Design CIS L Allegro PCB Librarian XL Allegro Design Authoring - Capture CIS	OK Cancel
•	
📃 Use as default	

Figure 2: Licence Dialog box

Click **OK** to accept

SiPM MODEL RESPONSE SIMULATION ------

 $\mathsf{File} \rightarrow \mathsf{Open} \rightarrow \mathsf{Desktop} \rightarrow \mathsf{EDIT}_\mathsf{SPICE} \rightarrow \mathsf{EDIT}_\mathsf{I}_\mathsf{L} \rightarrow \mathsf{SiPM}_\mathsf{SIM} \rightarrow \mathsf{Open}$

💽 Open Projec	t				×
Look in:	EDIT_1		~	G 🤌 📂 🛄 🗸	
Quick access Desktop Libraries This PC	Name SiPM_SIM-P SiPM_SIM	^ SpiceFiles		Date modified 19/10/2015 10:25 19/10/2015 10:36	Type File folder OPJ File
Network	< File name: Files of type:	SiPM_SIM Orcad Project (*.opj)		~	> Open Cancel

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

👹 SCHEMATIC1-bias - Allegro AMS Simulator - [bias.out.1]	- 0	×
🖹 File Edit View Simulation Trace Plot Tools Window Help 🖷	cādence	_ # ×
I L1 N114303 0 DC OAdc AC OAac FVDLSE 0 (10+*)6; 200n 10p 10p In 1m C_Cqon N114303 SIFM_OUT (148.5t*N1) TC=0.0 C_Cqoff 0 N114303 SIFM_OUT (148.5t*N1) TC=0.0 C_Cqoff N114329 SIFM_OUT (148.5t*(625-N1)) TC=0.0 C_Cqoff N114329 SIFM_OUT (1931x/N1; TC=0.0 C_Cqoff N114329 SIFM_OUT (3931x/N1; TC=0.0 FRAM nf=1 ERROR(ORPSIM-15142): Node N114303 is floating ERROR(ORPSIM-15142): Node SIFM_OUT is floating ERROR(ORPSIM-15142): Node N114329 is floating F (nodes	· · · · · · · · · · · · · · · · · · ·
bias out.1		
PSpice> Initializing Scr Loading C:/Cadence/SPB_1 PSpice> PSpice>	e Color Trace Nar X Values	^ ^

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.

Place Hierarchical Port	×	
Symbol: PORTRIGHT-R PORTRIGHT-L/CAPSYM PORTRIGHT-L/CAPSYM PORTRIGHT-R/CAPSYM PORTRIGHT-R/CAPSYM PORTRIGHT-R/Design Libraries: CAPSYM Design Cache SOURCE Name: PORTRIGHT-R		
NetGroup Port Show UnNamed NetGroup		

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)

Display Property	rtier V
Display Proper	nues ~
Name Name	Font
Name: Name	Arial 7 (default)
	Pilar ((dordan)
Value: SiPM	OUT THE REAL PLANE
	Uhange Use Default
Display Form	nat
	Color
O Do Not	Display
	Default
Value 0	Jnly
	wed Value
	Rotation
O Name D	
	• 0° 0 180°
Both if \	√alue Exists O 90° O 270°
OVeterx	Value Eviate
Value II	Value Exists
	Text Justification
	Default
	D'ordan -
	OK Cancel Help

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).



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.

SCHEMATICI-bias - Allegro AMS Simulator - [bi	ias (active)] Tools Window Help 🖛			cādence
	SCHEMATICI-bias		A	
-sout -				
-100.0				
-15800 85 0.105 • V(SIPH_OUT)	0.2us 0.3us	0.4us 0.5us Tine	0.6us 0.7us	0.8us 0.9us 1.0us
bias (active)				
X PSpice> Initializing Script A X Loading C:/Cadence/SPB_16.0 A 1 PSpice> PSpice> TT T T T T T T T T T T T T	ime step = 10.00E-12 Time = 1.000E-06	End = 1.000E-06	Circuit read in A Calculating bie Biss point calc Traniert Anal Traniert Anal Tratiet Anal Tratiet bit me NFO(ORPRO +	Trace Color Trace Ham Yi
Edit the simulation settings			Time= 1.000E-06	100%

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



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 \rightarrow Open \rightarrow Projects \rightarrow Desktop \rightarrow EDIT_SPICE \rightarrow EDIT_2_L \rightarrow SiPM_SIM \rightarrow 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.

General Analysis Category:	Configuration Files Options Data Collection Probe Window Details Elements
Stimulus Library Include	Configured Files
Update Index	nom.lib Add as Global Add to Design Add to Profile Edit Change
	Library Path "C:\Cadence\SPB_16.6\tools\PSpice\Library" Browse

Figure 21: Simulation Setting - Library Configuration Files

Browse to EDIT_SPICE \rightarrow PSPICE_LIB \rightarrow OPAMP_GIULIO, select AD8014 and Open

🛃 Open				×
Look in:	OPAMP_GIU	JLIO ~	G 🤌 📂 🛄 -	
-	Name	^	Date modified	Туре
	AD8014		20/10/2015 11:44	PSpice Mo
Quick access	HMH6715		20/10/2015 11:44	PSpice Mo
	PA659		20/10/2015 11:44	PSpice Mo
	PA847		20/10/2015 11:44	PSpice Mo
Desktop	OPAMP_GI	ULIO	20/10/2015 11:44	PSpice Mo
-	THS3201		20/10/2015 11:44	PSpice Mo
	HS4505		20/10/2015 11:44	PSpice Mo
Libraries				
This PC				
S				
Network	<			>
Retwork	File name:	AD8014	~	Open
	Files of type:	Library Files (*.lib)	\sim	Cancel

Figure 22: Open Library

Finally select Add as a Global, Apply and OK.

General Analysis	Configuration Files Options Data Collection Probe Window
Category:	Details Filename:
Stimulus Library	C:\Users\edit\Desktop\EDIT_SPICE\PSPICE_LIB\OP Browse
Include	Configured Files
Update Index	C:\Users\edit\Desktop\EDIT_SPICE\PSPICE_LIB
	Add to Design
	Add to Profile
	Edit
	Change
	< >>
	Library Path
	"C:\Cadence\SPB_16.6\tools\PSpice\Library" Browse

Figure 23: AD8014 component added for simulation

Simulate the circuit and calculate the gain $V_{\mbox{\scriptsize OUT}}/Q_{\mbox{\scriptsize IN}}$

DISCRIMINATOR RESPONSE-

Open the EDIT_L3_P project: File \rightarrow Open \rightarrow Projects \rightarrow Desktop \rightarrow EDIT \rightarrow EDIT_3_L \rightarrow SiPM_SIM \rightarrow Open

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



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.





PARAMETRIC SIMULATION -

Open the EDIT_2_L_P project:

 $File \rightarrow Open \rightarrow Projects \rightarrow Desktop \rightarrow EDIT_SPICE \rightarrow EDIT_2_L_P \rightarrow SiPM_SIM \rightarrow Open$

ieneral Analysis Configurat	on Files Options Data	Collection	Probe Window	
Inalysis type: Time Domain (Transient) Options: General Settings Monte Carlo /Worst Case	Sweep variable Voltage source Current source Global parameter Model parameter	Name: Model Model	type:	
Parametrio Sweep Temperature (Sweep) Save Bias Point Load Bias Point Save Check Points Restart Simulation	I emperature Sweep type Elinear Logarithmic De	cade v	Start value: End value: Increment:	1 20 1
	⊘ Value list			

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 \rightarrow Open \rightarrow Projects \rightarrow Desktop \rightarrow EDIT_SPICE \rightarrow EDIT_3_L_P \rightarrow SiPM_SIM \rightarrow Open

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