

# **Application Note**

# SPICE Models: ROHM Voltage Detector ICs



BD48 G/FVE, BD49 G/FVE, BD52 G/FVE, BD53 G/FVE, BD45 G, BD46 G G, BU48 G/FVE/F, BU49 G/FVE/F, BU42 G/FVE/F, BU43 G/FVE/F, BD47 G

No.10006EAY01

## 1. INTRODUCTION

#### 1.1 SPICE

SPICE is a general-purpose circuit-simulation program for nonlinear DC, nonlinear transient, and linear AC analysis. It solves network equations for node voltages and was designed to solve both linear and nonlinear electrical circuits. SPICE models simulate a variety of circuits with high accuracy, from switching power supplies to RAM cells and sense amplifiers. A variety of circuit components are supported, including resistors, capacitors, inductors, mutual inductors, dependent/independent voltage/current sources, transmission lines, and common semiconductor devices such as diodes, bipolar junction transistors (BITs), junction field effect transistors (JFETs), metal-oxide-semiconductor field effect transistors (MOSFETs), and metal-semiconductor FETs (MESFETs).

#### 1.2 PSpice®

PSpice® is a SPICE analog circuit and digital logic simulation software that runs on personal computers.

#### 1.3 OrCAD® PSpice 9.1

OrCAD® PSpice simulates analog-only circuits. After preparing a design for simulation, OrCAD Capture generates a circuit file set, which contains the circuit netlist and analysis commands. The circuit file set is read by PSpice A/D for simulation. PSpice A/D formulates these into meaningful graphical plots, which can be marked for display directly from the schematic page.

OrCAD PSpice 9.1 (Student Version) is PSpice simulation software that is available for free but with certain limitations imposed on the libraries and functionality. A fully functional version can be purchased from Cadence Design Systems, Inc.

#### 1.4 SPICE Models: ROHM Voltage Detector ICs

ROHM SPICE model is made by typical data, and the manufacturing dispersions are not included.

Moreover, it does not guarantee all of the simulation result of execution by using this.

This model is being supplied as a aid to confirm the validity of a design approach and help to select surrounding component values.

While it reflects reasonably close to similarity to the actual device in terms of performance, it is not suggested as a replacement for breadboarding.

Simulation should be used as a forerunner or a supplement to a traditional lab testing.

Note:

OrCAD is a registered trademark of Cadence Design Systems, Inc. PSpice is a registered trademark of Cadence Design Systems, Inc.

## 2. SIMULATION CIRCUIT SETUP

## 2.1 Model File

A model defines the electrical behavior of a part. On a schematic page, this correspondence is defined by a part's implementation property, which is assigned the model name.

Depending on the device type that it describes, a model is defined as one of the following:

- · a model parameter set
- a subcircuit netlist

Both ways of defining a model are text-based, with specific rules of syntax.

## 2.2 Model Libraries

Device model and subcircuit definitions are organized into model libraries. Model libraries are text files that contain one or more model definitions. Typically, model library names have a .LIB extension.

Most model libraries contain models of similar type. For vendor-supplied models, libraries are also partitioned by manufacturer.

Two files are used, the LIB file and the OLB file. The LIB file holds the characteristics of the model while the OLB file contains the model symbols.

Although the LIB file alone is enough, circuits cannot be generated using OrCAD Capture. Instead, the netlist file must be used in PSpice A/D.

In contrast, having an OLB file of the model enables the user to make circuits in the OrCAD Capture, resulting in greater convenience and better circuit conceptualization.

The sample LIB and OLB model files used in the setup, simulation, and evaluation examples with OrCAD PSpice are BU4229.LIB and BU4229.OLB. These are PSpice models of a voltage detector IC manufactured by ROHM.

## 2.3 Simulation Setup

To start simulation setup, select the OrCAD Capture application.



## 2.3.1 Starting the Design Project

GrCAD Capture		
File View Edit Options V	Window Help	
New >	Project	
Save Ctrl+S Save As	Library VHDL File Text File	
Print Preview Print Ctrl+P Print Setup		
Import Design Export Design		
Exit	-	
Create new project.	Session Log	

Create a new project and enter the necessary parameters such as project name, project type, and project directory. In this example, the project name is BU4229\_SIM\_test and the type of project is Analog or Mixed A/D.

New Project		X
Name BU4229_SIM Create a New Project Using Analog or Mixed A/D Mailer PC Board Wizard Mailer Programmable Logic Wizard Mailer Schematic	<ul> <li>Tip for New</li> <li>Create a n</li> <li>Mixed A/D</li> <li>new projection</li> <li>or copied for template.</li> </ul>	OK Cancel Help v Users ew Analog or project. The t may be blank rom an existing
E:\BU4229_SIM	>	Browse

Next, create a blank project

Create PSpice Project	
C Create based upon an existing project	ОК
	Browse
Croste a blank project	Cancel
ce cleate a blank project	Help

## 2.3.2 Project Directory

Select the target directory for saving the project. In this example we will be saving our project on drive E:

Select Directory	
Directories:	ОК
d:\orcad_demo	Cancel
ORCAD_DEMO     Capture     PSpice	Help
	Create Dir
Drives:	
🖃 d: DATA 📃 💌	Network
🗖 d: DATA 📃 🔼	
e: Transfer	
E h: \\hillarv\com	
🖾 i: \\mahal\d\mtr_ds 🔽	

Create a project folder, such as BU4229\_SIM.

Select Directory	
Directories: e:\	ОК
e:\ BD6264 DS Crossover DS JOB InfanView KPU-UTILS	Help
Drives:	Click this button

#### BD48□G/FVE,BD49□G/FVE,BD52□G/FVE,BD53□G/FVE, BD45□□G,BD46□□G,BU48□G/FVE/F,BU49□G/FVE/F, BU42□G/FVE/F,BU43□G/FVE/F,BD47□G

Create Directory				
Current Directory: E:\	ОК			
Name: BU4229_SIM	Cancel			
	Help			

After creating the folder, select that folder then press OK

Select Directory	
Directories:	ОК
e:\bu4229_sim	Cancel
BU4229_SIM	Help
	Create Dir
Drives:	Network

## 2.3.3 The OrCAD Capture Environment

🖼 OrCAD Capture	
File Edit View Place Macro Accessories Options Window Help	
🗈 Design1 💶 🗖 🗙	K
PCB	₽
C SCHEMATIC1 : PAGE1)	1
	<u> </u>
▼ 	K
0 items selected Scale=100% X=5.30 Y=0.10	1. )

This is how OrCAD Capture will look like after successfully setting the project directory. The next step is to add parts to the circuit for simulation.

## 2.3.4 Adding Parts to the Schematic

0 🔝	rCAD	) Capt	ture										(		
File	Edit	View	Place Macro Accesso	ries Options	Window	Help	р								
1	3		Part	Shift+P				•	8		U? \$1	» 1		5	8
		- 24	wire Bus	Shift+W Shift+B											
	Desig	;n1	Junction	Shift+J											
		F	Bus Entry	Shift+E											-
C		a	Net Alias	Shift+N								1			1
Ę			Power Groupd	Shift+F Shift+C	_	+				3	_		اری		<u>N1</u>
	E	35,55	Off-Page Connector.	, Drinterta	100505	0.000	a sanaa	to to to to to	9/0/10/1	543535 - 545	and solution	10/20/202	otatat e	•	l
	e	33	Hierarchical Block		1.1.1										+
		202	Hierarchical Port		10000										1
			Hierarchical Pin	Shift±Y	1.1.1									-1	PHR
	D	23		Shirera	1203										GND
			Title Block Bookmark		1.1.1										::
	_	20	Bookingrkin	-	1.44										CIP
-			lext	Shift+1	10.000										<b>₩</b>
			Rectangle												≪c
			Ellipse												×
			Arc	chift and	1.1								•		
			Polyline	Shirt+Y									المشد	///	ž/
															<b></b>
Place	a part							0 ite	ems select	ted S	cale=100%	X=6.10	Y=0.00	Ξ,	1. )

Parts may be added as shown or by simply pressing Shift+P. Parts can also be added by clicking on the Place Part button on the right tool bar.

CrCAD Capture	
File Edit View Place Macro Accessories Options Window Help	
🖬 Design1 📃 🗆 🗙	💌 🖌
PCB	
SCHEMATIC1 : PAGE1)	
p	말
	на н
	2>>
	,
	la l
Place a part. 0 items	selected Scale=100% X=6.30 Y=1.10 // )

This is the Place Part window. Initially no libraries are included. Libraries need to be added for setup (i.e. the PSpice models for the BU4229).

#### BD48□□G/FVE,BD49□□G/FVE,BD52□□G/FVE,BD53□□G/FVE, BD45□□□G,BD46□□□G,BU48□□G/FVE/F,BU49□□G/FVE/F, BU42□□G/FVE/F,BU43□□G/FVE/F,BD47□□G



## 2.3.5 Adding Libraries

Place Part		×
Part:		
Part List:		Add Library Remove Library Part Search
Libraries: Design Cache	Graphic Convert Packaging Parts per Pkg: 1 Part: Type:	

To add Libraries, click on the Add Library button as shown above.

Select the OLB files required for simulation. Normally, sources.OLB and analog.OLB are necessary, which are a shared OLB file of OrCAD containing voltage/current sources and analog devices such as resistors and capacitors. Go to the Capture/Library/PSpice folder (default) in the OrCAD PSpice directory or to the relevant folder.

#### BD48□G/FVE,BD49□G/FVE,BD52□G/FVE,BD53□G/FVE, BD45□□G,BD46□□G,BU48□G/FVE/F,BU49□G/FVE/F, BU42□G/FVE/F,BU43□G/FVE/F,BD47□G

Browse File				? 🔀
Look in: 🔎	Library	•	🗢 🔁	
CAPSYM.C	ΡLΒ			
File name:				Open
Files of type:	Capture Library(*.olb)		•	Cancel
	Dpen as read-only			

From the Capture/Library/PSpice folder, select analog.OLB and source.OLB then press the Open button. (Press Ctrl and click on the files to select them both)

Browse File	? 🗙
Look in: 🔀	Pspice 💌 🗲 🔁 📸 📰 -
abm.olb analog.olb analog_p.( breakout.o eval.olb source.olb	sourcstm.olb special.olb olb olb
File name:	"source.olb" "analog.olb"
Files of type:	Capture Library(*.olb)
	C Open as read-only

This is what the Place Part window will look like after adding the libraries. You may see a list of parts from each library.

Place Part		
Part:		 ОК
Part List: C_elect C_var E EPOLY F FPOLY G GPOLY H		Cancel Add Library Remove Library Part Search Help
Libraries: ANALOG Design Cache SOURCE	Graphic Graphic Convert Packaging Parts per Pkg: 1 Part:	

Next, add the model library of the PSpice model for simulation. It is easier if the model files are located in the Project Folder. (E.g. BU4229.LIB and BU4229.OLB in E:¥BU4229\_SIM) Then add the BU4229.OLB library.



The parts list of the BU4229 library should now be displayed, along with (possibly) other associated parts.

Place Part			
Part: BU4229			ОК
Part List:			Lancel
BU4229			Add Library
COMP1 COMP2		- 1	Remove Library
		- 1	Part Search
Libraries: ANALOG BU4229 Design Cache SOURCE	Graphic Normal Convert Packaging Parts per Pkg: 1 Part: Type: Homogeneous		Help VOC VOUT SU 4225 GRO CT

Simulation setup can now begin.

## 2.3.6 Simulation Setup

Add parts in the OrCAD Capture Environment.

Place Part			×
Part: BU4229		-	ОК
Part List: BU4229 COMP1 COMP2			Cancel Add Library Remove Library Part Search
Libraries: ANALOG BU4229 Design Cache SOURCE	Graphic Convert Packaging Parts per Pkg: 1 Part: Type: Homogeneous		Help

#### BD48□G/FVE,BD49□G/FVE,BD52□G/FVE,BD53□G/FVE, BD45□□G,BD46□□G,BU48□G/FVE/F,BU49□G/FVE/F, BU42□G/FVE/F,BU43□G/FVE/F,BD47□G

	<b>:</b> (	)r(	CAI	D C	apt	ure	• - [	7 -	(SC	HE	MAT	FIC1	: 8	PAG	E1)																							×
Į	2	File	e 1	Edit	Vi	BW	Pla	се	Ma	cro	Ac	cess	ories	; C	ptior	ns	Win	dow	H	elp																	_ ť	s ×
	1		3		é	3	X	B			$\square$	0	1	BU4	1229	1							-	œ	G	<u>ا</u>	20		U?	111		8		X		۱Ņ	1 8	:
ŕ	_				_		-	-	_		_	-	+	_		_			_	_		<b>—</b>	_	_	-	_	_		_	-		3	-	_		_	白	
																	L i a																					<b>X</b>
															à	Г	01																					₽
															1	-	VE	D		٧	ΟL	JT	┢	0		-												긴
																		Т <mark>Р</mark>	й	γγ	a' l																	<u>N1</u>
															ŝ									2														l
															2		G١	١D			10	СΤ	┢	ა		-												+
																																						1
L	·																																					PHR
																																						GND
																																						Ē
																																						Takin I
																																						10011
																																						×
																																						$\geq$
с																																						٤
																																					-	
Ĺ	•																																			►		0
																							0 ite	ms s	sele	ted		Sc	ale=	200	%	X=3	3.70	۲ <u>–</u>	1.90	)	11	D

Next, add voltage sources and other components like resistors and capacitors. DC Voltage (VDC) and DC Current (IDC) sources can be found in the Sources library while passive devices like resistors (R) and capacitors (C) are located in the Analog library.

Place the parts. To rotate the device, right-click on the device and select Rotate.

	<b>S</b> (	)rC	AD C	aptur	e - [	/ - (S	CHE	мат	IC1 : I	PAGE1	)]											
Γ		File	Edit	View	Pla	ce M	lacro	Acc	essorie	s Optio	ons V	Vindow	Help	)							- ć	r ×
	ትን			A	X		R	0	el	B						- I @			el u	12 ti 😕 🖬 🖬 📰	5 N B	
ŀ	_	<u> </u>							•						-			1 1		3		
																					· · ·	x
																						Ð
	1																					ᅱ
	D .																	·	· ·		· ·	N1
											11	1						· .	P'	<b>5</b> - 1		-
																l á			. 19	•		느
									-		V	DD	· 1	/01	JT	0		3 🕞	-~\\		1 I I	+
												P	ามว่	ю <sup>.</sup>		1.1			⊡ 1I	Mirror Horizontally Mirror Vertically		1
													0422					. L		Rotate		PHR
										2	G	ND		. 0	т	3				Edit Properties		GND
																<b>.</b> .				Edit Part		ㅎ
																				Descend Hierarchy		
	1.																			Ascend Hierarchy		
																				Zoom In		ݮH
																				Zoom Out		≪C
																				Go To		×
	Ľ																			Cut		
																				Copy Delete		$\geq$
	1																		· · ·	Delete	1	$\geq$
																					· · · •	
																					▶	0
ĺ																1 item :	select	ed	Scale	=200% X=4.50 Y=1.2	) <i>[]</i> ,	$\overline{\mathbf{D}}$

Complete the necessary wiring as shown below. In this case the simulation will be run to determine the detection voltage. (Detection Voltage Parameter)

OrCAD Capture - [/ - (SCHEMATIC1 : PAGE1)]	
🛐 File Edit View Place Macro Accessories Options Window Help	_ 8 ×
	· · · · · · ≥ · <sub>R2</sub> · · · · · · · · · · · · · · · · · · ·
	S = 10 <sup>-1</sup>
U	
	онт 🗖 💷 и и и и и и и и и и и и и 🗌 📥
	e e la sue e e e e e e e e e e e e e e e e e e
V1 2	<u> </u>
OVdc++++++++++++++++++++++++++++++++++++	CT Pyr
	<u>e</u>
	<del>.</del> . <sup>(1</sup>
	n a a a a la lin a a a a a a a a a 🧰
	· · · · · · · · · · · · · · · · · · ·
	· · · · · · · · · · · · · · · · · · ·
	× · · · · · · · · · · · · · · · · · · ·
c · · · · · · · · · · · · · · · · · · ·	
	<u> </u>
Ready	0 items selected Scale=162% X=4.00 Y=1.10 // D

Place a ground by clicking on the GND button in the right tool box.



To set the values of the external components (i.e. resistor, capacitor, DC voltage source), click on the device then right-click and choose Edit Properties. For this example, R2 will be set to 470Kohms, C1 to 100nF, and V1 to 5V.

GrCAD Capture - [/ - (SCHEMATIC	1 : PAGE1)]		
📓 File Edit View Place Macro Access	sories Options Window Help		_ 8 ×
	GND		
		3	
			A
		<u></u> <u></u>	<b>-</b>
			· · · · · · · · · · · · · · · · · · ·
• • • • • • • • • • • • •		Mirror Horizontally	- · · · · · ·
	<u></u>	Mirror Vertically	
		T O Rotate	<mark>.</mark>
		Edit Properties	
	V1 2	Edit Part	
OVdc <sup>+</sup> =		T Descend Hierarchy	PHR Q
			GND
Η		Zoom Out	
		Go To	· · · · · ·
		Cut	· · · · · ·
		Сору	<u>ieu</u>
		Delete	<del></del>
			🗵
	<u> </u>		🔍 🥄
c · · · · · · · · · · · · · · ·	<del>.</del>		
			· · · · · ·
			<u> </u>
	1	item selected Scale=162% X=	=4.20 Y=0.90 // ◯

The Property Editor window will appear. Change the value of the part by typing the value with the proper prefix unit (i.e. k for kilo and p for pico).

🖫 OrCAD Capture - [Property Editor]					
🛐 File Edit View Place Macro Accessorie	s Options Windo	w Help			_ 8 ×
	GND			U? ‡1	
New Column Apply Display Delete	Property Filter by:	Capture			▼ Help
Value Refe	rence Designator	PCB Footprint	Power Pins Visible	Primitive	Source 🔺
1 E SCHEMATIC1 : PAGE1 : R2 1k	R2	RC05		DEFAULT	D\ORCAD_DEMO\CAPTURI
Parts (Schematic Nets (Pins ( Ti	tle Blocks 🖌 Globa	als 🖌 P d 🔳			•



Return to the schematic page by clicking on Window tab in the main pull-down menu.

GrCAD Capture - [Property Editor]				
🛐 File Edit View Place Macro Accessories Options	Window Help			- 8 ×
🗅 🖻 🖬 🍓 👗 🛍 💼 🛄 🖾 🖾 GND	New Window Cascade		U? ‡1	
New Column Apply Display Delete Property Fi	Tile Horizontally Tile Vertically			✓ Help
Value Reference Desi	Arrange Icons	s Visible	Primitive	Source 📥
1 E SCHEMATIC1: PAGE1: V1 VDC V1	1 Session Log		DEFAULT	D\ORCAD_DEMO\CAPTUR
	2 Design1			
	3 / - (SCHEMATIC1 : PAGE1)			
	4 Property Editor	]		
				-
▲ Parts ( Schematic Nets ( Pins ( Title Blocks )	Globals 🕻 Pd 🔺			•

The values can also be changed by double-clicking on the value displayed.

CrCAD Capture - [/ - (SCHEMATIC1 : PAGE1)]		
📔 File Edit View Place Macro Accessories Options Window Help	-	đΧ
12 🗃 🖶 🎒 🛍 🗈 🕮 GND		۴.
· · · · · · · · · · · · · · · · · · ·	3	
	· · · · · · · · · · · · · · · · · · ·	1 <b>~</b>
		~~
D	<b>≤</b> . R2	그
· · · · · · · · · · · · · · · · · · ·		<u>N1</u>
		ll
		+
BU	J4229	1
	· · · <sub>CT</sub> 3 · · · · · · · · · · · · · · · · · ·	PHR
		GND
	n na hara a bar i i i i i i i i i i i i i i i i i i i	흠
	100	<u> </u>
		<u></u> €®H
· · · · · · · · · · · · · · · · · · ·		≪c
		×
[]		$\geq$
°		$\geq$
	· · · · · · · · · · · · · · · · · · ·	
	<b>&gt;</b>	0
	0 items selected Scale=162% X=4.50 Y=2.20	(D)

Enter the value in the popup box, then press OK.

Display Properties	
Name: Value	Font Arial 7 (default)
Value: 100p	Change Use Default
Display Format Do Not Display Value Only Name and Value Name Only Both if Value Exists	Color           Default           Rotation           • 0°         180°           • 90°         270°
ОК	Cancel Help

## 2.3.7 Creating a Simulation Profile

To create a simulation profile, click on the PSpice tab in the main menu and select New Simulation Profile.

OrCAD Capture - [/ - (SCHEMATIC1 : PAGE1)]	
File Edit View Place Macro PSpice Accessories Options Window Help	- 8 ×
12 😂 🖬 🕘 👗 🖻 🔂 New Simulation Profile 💽 🖳 🔍 🔍 🔍 💷 🖽	N R
Edit Sindedon Prone	
Create Netlist View Netlist	
Place Optimizer Parameters U1 470K	· <u>N1</u>
Markers 1 VDD VOUT 0 Bias Points BU4229	· +
° V1 2 GND CT 3	
$\begin{bmatrix} 1 & 1 & 1 & 1 & 1 & 1 & 1 \\ 1 & 1 & 1 &$	
100p	
Create a new simulation profile. 1 item selected 5cale=200% X=5.60 Y=-0.40	



Enter a name for the simulation profile. In this example the name is BU4229\_Vdet.

New Simulation	
Name:	Create
BU4229_Vdet	
	Cancel
Inherit From:	
none	
Root Schematic: SCHEMATIC1	

In the Simulation Settings window, select DC Sweep under Analysis type in order to evaluate Vdet.

Simulation Settings - BU4229	_Vdet 🛛 🔀
General       Analysis       Include Files         Analysis type:	Libraries Stimulus Options Data Collection Probe Window Run to time: 1000ns seconds (TSTOP) Start saving data after: 0 seconds Transient options Maximum step size: seconds Skip the initial transient bias point calculation (SKIPBP) Output File Options
	OK Cancel Apply Help



Enter the required settings. In this case the DC voltage source V1 will be swept linearly from 5V to 0V with a stepping voltage of 0.1mV.

Simulation Settings - BU4	229_Vdet 🛛 🔀
General Analysis Include F	iles Libraries Stimulus Options Data Collection Probe Window
Analysis type: DC Sweep  Options:  Primary Sweep  Secondary Sweep  Monte Carlo/Worst Case  Parametric Sweep	Sweep variable         Voltage source       Name:         Current source       Model type:         Global parameter       Model name:         Model parameter       Model name:         Temperature       Parameter name:
☐ Temperature (Sweep) ☐ Save Bias Point ☐ Load Bias Point	Sweep type     Start value:     5       Linear     End value:     0       Logarithmic     Decade     Increment:     0.1m
	C Value list
	OK Cancel Apply Help

Simulation Settings - BU4229_Vdet	
General Analysis   Include File: Libraries Stimulus   Op	tions Data Collection Probe Window
Filename:	
	Browse
Library files	
nom.lib*	Add as Global
	Add to Design
	Edit
	Change
Library Path	
"d:\DrCAD_Demo\PSpice\UserLib";"d:\DrCAD_Demo\C	Browse
ОК С	ancel Apply Help



Locate the model file BU4229.LIB

Open		? 🗙
Look in: 🗀	) BU4229_SIM 📃 🖛 🗈 💣	
BU4229		
	-	
File name:		Open
Files of tupe:	Liberry Eiler (* Bb)	Cancel
Files of type:		

Add the library to the design then press OK.

Simulation Settings - BU4229_V det
General       Analysis       Include Files       Libraries       Stimulus       Options       Data Collection       Probe Window         Filename:
Add as Global Add to Design Edit Change
Library Path "d:\OrCAD_Demo\PSpice\UserLib";"d:\OrCAD_Demo\C Browse
OK Cancel Apply Help

Simulation is now ready to begin.

## 2.3.8 Running the Simulation

To run the simulation, go to PSpice tab in the main menu and select Run.

CrCAD Capture - [/ - (SCHEMATIC1 : PAGE1)]	
🛐 File Edit View Place Macro PSpice Accessories Options Window He	de _ □ ×
Image: Constraint of the state of the st	
Image: Second state     Image: Second state       Image: Second state     View Simulation Results       View Output File	3
Create Netlist View Netlist	R1 2
Place Optimizer Parameters Run Optimizer	U1 470K - T
Markers 1 Bias Points	
5 <u>+</u> V1 <u>2</u>	
· · · · · · · · · · · · · · · · · · ·	
	100p
	×
│	
Run PSpice simulation for active profile.	0 items selected Scale=200% X=3.20 Y=0.00 // 🛄

OrCAD PSpice A/D will automatically load and run the simulation.

eschematic1-BU4229_Vdet - OrCAD PSpice A/D Demo		
Eile View Simulation Tools Window Help 🚟		
🖹 🔹 😂 🔛 🎒 🐘 🛍 🕰 💭 😂 🛛 SCHEMATICI-BU4229_Vdet 💿 🕨		
▲④���     駈秋目		
		1
X		
Similation running No recognized product configuration selected.	,,	
Reading and checking circuit V_V1 = 2.661	End = 0	Progress of simulation
DC Analysis		
Analysis (Watch ) Devices /		
For Help, press F1	V_V1 = 2.661	46%

## 3. EVALUATION OF RESULTS

A completed simulation will look like this. Note the 100% shown on the progress bar.

Ele Ed Yeek Smiddon Jrace Bot Topols Window Help 語  Ele Ed Yeek Smiddon Jrace Bot Topols Window Help 語  Ele Ed Yeek Smiddon Jrace Bot Topols Window Help 語  Ele Ed Yeek Smiddon Jrace Bot Topols Window Help 語  Ele Ed Yeek Smiddon Jrace Bot Topols Window Help 語  Ele Ed Yeek Smiddon Jrace Bot Topols Window Help Ele  Ele Ed Yeek Smiddon Jrace Bot Help Ele  Ele Ed Yeek Smiddon Jrace Bot Help Help Help Help Help Help Help Help	🐺 SCHEMATIC1-BU4229_Vdet - OrCAD PSpice A/D Demo - [bu4229_sim-	-SCHEMATIC1-BU4229_Vdet (active)]	_ 7 🛛
● ・ ・ ・ ・ ・ ・ ・ ・ ・ ・ ・ ・ ・ ・ ・ ・ ・ ・ ・	>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>>		_ & ×
▲ ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ● ●	🖹 🔻 🚅 🎽 🛃 🎒 🕺 🖻 💼 🗠 🗠 🛛 SCHEMATICI-BU4229_Vdet	t II	
Image: Solid Matter Solid	� � � �   Ⅲ 蚯 ♥ 目    ☆ ₩ ▽ ♂    杰 苯 メ ኳ オ	<b>养蒜晶蛋苷</b> 型	
Image: Construct configuration selected       Image: Construct configuration selected         Image: Construct configuration selected       Image: Construct configur			
Image: Solid State of Solid State o			
Image: Subtraction selected       Image: Subtraction selected         Image: Subtraction s			
Image: State and checked, no errors       DC Analysis finished         DC Analysis finished       State 5			
Image: State and checked, no errors       Image: State and checked, no errors       Image: State and checked, no errors         Image: State and checked and state an			
80       8.50       1.60       1.50       2.60       2.50       3.60       3.50       4.60       4.50       5.60         0       9.50       1.60       1.50       2.60       2.50       3.60       3.50       4.60       4.50       5.60         0       0.01       0.01       0.01       0.01       0.01       0.01       0.01         Image: SchEMATICI-8U4229_Vdet" [E:\BU4229_SIM\bu4229_sim       Image: SchEMATICI-8U4229_Vdet" [E:\BU429_SIM\bu4229_sim       Image: SchEMATICI-8U429_Vdet" [E:\BU429_SIM_Vdet 229_sim       Image: SchEMATICI-8U429_Vdet" [E:\BU429_SIM_Vdet 229_sim       Image: SchEMATICI-8U429_Vdet" [E:\BU429_SIM_Vdet 229_sim       Image: SchEMATICI-8U429_Vdet" [E:\BU429_SIM_Vdet 29_sim       Image: SchEMAT			
BU       9,5U       1.8U       1.5U       2.8U       2.5U       3.8U       3.5U       4.8U       4.5U       5.8U         U			
8U       8.5U       1.6U       1.5U       2.6U       2.5U       3.6U       3.5U       4.6U       4.5U       5.6U         U			
80       0.50       1.90       1.50       2.90       2.50       3.90       3.50       4.90       4.50       5.90         U       0       0.50       1.90       1.50       2.90       2.50       3.90       3.50       4.90       4.50       5.90         U       0			
BU       9.5U       1.8U       1.5U       2.8U       2.5U       3.8U       3.5U       4.8U       4.5U       5.8U         U       U       U       U       U       U       U       U       U         Image: Solid Antificit Buld229_sim       Image: Solid Antificit Buld229_SIM\buld229_sim       Image: Solid Antificit Buld229_Vdet" [E\buld229_SIM\buld229_sim       Image: Solid Antificit Buld29_Vdet" [E\buld229_SIM\buld229_sim       Image: Solid Antificit Buld29_Vdet" [E\buld229_SIM\buld229_sim       Image: Solid Antificit Buld29_Vdet" [E\buld229_Vdet" [E\buld229_Vdet" [E\buld229_SIM\buld229_sim       Image: Solid Antificit Buld29_Vdet" [E\buld229_Vdet" [E\buld229_V			
6U       6.5U       1.6U       1.5U       2.6U       2.5U       3.6U       3.5U       4.6U       4.5U       5.6U         U U1         Example to a consider the second to a construction of the second to consecond to a consecond to a construction of t			
80       8,50       1,90       1,50       2,90       2,50       3,90       3,50       4,90       4,50       5,60         U_U1       U_U1 <t< td=""><td></td><td></td><td></td></t<>			
U U1	0V 0.5V 1.0V 1.5V 2	2.00 2.50 3.00 3.50 4.00 4	+.50 5.00
bu4229_sim  No recognized product configuration selected.  Profile: "SCHEMATIC1-8U4229_Vdet" [E:\BU4229_SIM\bu4229_sim Reading and checking circuit Circuit read in and checked, no errors DC Analysis iniched Circuit framework in the destination of the desti		V_V1	
No recognized product configuration selected.         ** Profile: "SCHEMATICI-BU4229_Vdet" [E:\BU4229_SIM\bu4229_sim Reading and checking circuit Circuit read in and checked, no errors DC Analysis         DC Analysis finished Circuit read in and checked.	💆 bu4229_sim		
Reading and checking circuit Circuit read in and checked, no errors DC Analysis finished CC Analysis finished	No recognized product configuration selected.		
DC Analysis DC Analysis finished Conclusional Conclusion Conclusio	Reading and checking circuit Circuit reading and checked no errors		
Condition second to	DC Analysis DC Analysis DC Analysis finished	tart = 5 V_V1 = 100.0E-06 End = 0	
Simulation complete	Simulation complete	B Andria (Wester) During (	
	Enr Help, press E1	(Analysis / watch / Devices /     (V, V1 = 100.0E-06	

## 3.1 Adding a Trace or Output

SCHEMATIC1-BU4229_Vdet - OrCAD PS	Spice A/D Demo - [b	u4229_sim-SCH	EMATIC1-B (	
Bile Edit View Simulation Irace Plot Too	ols <u>W</u> indow <u>H</u> elp 🚟		[	- 8 ×
🖹 • 🕞 😂 🖬 🚭 🕺 🗄 💼 🛃 🖌		1-BU4229_Vdet	• п	
	移いず【太口	化卡林林司	4 4 4 2 ×	)
Add Tr	race			
DU 1.0V	2.00	3.00	4.OV	5.0V
	V_V1			
 bu4229_sim				]
No recognized product configuration sele	×			
Reading and checking circuit				
Circuit read in and checked, no errors DC Analysis	Start = 5	V_V1 = 100	D.OE-06	Enc
DC Analysis finished Simulation complete	<			>
		Watch \ Devices /		
Add trace[s] to the selected plot	V_V1 = 100.0E-06	100%		

Select the pin, node, or branch for monitoring/measurement. In this case the VOUT pin is selected [V(U1:VOUT)].

#### BD48□G/FVE,BD49□G/FVE,BD52□G/FVE,BD53□G/FVE, BD45□□G,BD46□□G,BU48□G/FVE/F,BU49□G/FVE/F, BU42□G/FVE/F,BU43□G/FVE/F,BD47□G

Simulation Output Variables		Functions or Macros	
×		Analog Operators and Fun	ctions 💌
IS(X_U1.Mdrv) IS(X_U1.Mout) IS(X_U1.X1.M6) IS(X_U1.X1.M7) IS(X_U1.X52.M4) V(0) V(C1:1) V(C1:2) V(GND) V(N00014) V(N00017) V(N00055) V(R1:2) V(B1:2) V(U1:CT) V(U1:GND) V(U1:GND) V(U1:Y) V(V1:+) V(V1:-)	<ul> <li>Analog</li> <li>Digital</li> <li>Voltages</li> <li>Currents</li> <li>Noise (V<sup>2</sup>/Hz)</li> <li>Alias Names</li> <li>Subcircuit Nodes</li> </ul>	# () * + - / @ ABS() ARCTAN() ATAN() AVG() AVG() AVG() AVG() D() DB() ENVMAX(,) ENVMIN(,) EXP() G() MG()	
V(X_U1.10) V(X_U1.11) V(X_U1.14) V(X_U1.15)	171 variables listed	LOG() LOG10() M() MAX()	~

The OrCAD PSpice A/D should look like this.





## 3.2 Alternate Display

Enlarge the graph to full screen by clicking on the Alternate Display button.



It should look like this.



## 3.3 Measurement

To begin measurement, first enable the Cursor by clicking on the Toggle cursor button (shown below).



The cursor will appear, with the coordinates shown in a small window. Navigate to the desired point. In this example we will measure the threshold voltage.



Zooming is possible by selecting the area and pressing Ctrl+A. For this measurement the cursor is placed midway on the VOUT drop (VDD sweep from 5V to 0V).



The threshold voltage can be found by looking at the coordinates of the cursor. In this example the threshold voltage is at 2.9001V of VDD.

## 3.4 Annotate Measurement Values



To add an annotation of the measurement, click on the Mark Label button shown above. Ensure that the cursor is at the desired measurement point before making the annotation. Otherwise, the annotation will be incorrect. The annotation will display the coordinates of the cursor.



To fit the graph for viewing, go to View tab then select Fit under Zoom.

😾 SCHEMATI	C1-BU4229_Vdet - OrCAD P	Spice A/D Demo - [bu	14229_sim-SCHEMATIC1-B 🔳 🗖 🖡	×
🛃 Eile Edit	View Simulation Trace Plot To	ols <u>W</u> indow <u>H</u> elp 🔜	_ B >	×
- 	<u>Z</u> oom		Ctrl+F	
	編 Circuit File		Ctrl+I	
	🛃 Output File		· · · · · · · · · · · · · · · · · · ·	
@ <b></b>	👼 Simulation <u>R</u> esults	Previous		1
2	Simulation <u>M</u> essages			•
1 1	🧾 Simulation Queue	<u>R</u> edraw Bap - New Cep	Ctrl+L	
	✓ Output Window			
QE	Simulation Status Window	2.90010	2.90020 2.90030	
	Toolbars	U U1		
	✓ <u>S</u> tatus Bar			
🛃 bu4	🖌 Workbook Mode			-
VI Name	🛃 Alternate Display Shift+I	F12		_
** Profile:	🖉 Always <u>O</u> n Top			
Reading a Circuit rea	and checking circuit ad in and checked, no errors			
DC Analy DC Analy	sis	Start = 5	V_V1 = 100.0E-06 En	C
Simulation	n complete	<		j
		Analysis (W	/atch \ Devices /	
Zoom to show all	traces and labels	V_V1 = 100.0E-06		1



A fit view of the graph is shown below.



From the simulation results, VDET of the BU4229 is verified to be 2.9V with VOUT=470k $\Omega$  pulled up to VDD, CT=100pF, with a VDD sweep from 5V to 0V.

	Notes
N Co	o copying or reproduction of this document, in part or in whole, is permitted without the onsent of ROHM Co.,Ltd.
Т	he content specified herein is subject to change for improvement without notice.
TI "F W	he content specified herein is for the purpose of introducing ROHM's products (hereinafter Products"). If you wish to use any such Product, please be sure to refer to the specifications rhich can be obtained from ROHM upon request.
E: ill b	xamples of application circuits, circuit constants and any other information contained hereir ustrate the standard usage and operations of the Products. The peripheral conditions must e taken into account when designing circuits for mass production.
G H in	reat care was taken in ensuring the accuracy of the information specified in this document lowever, should you incur any damage arising from any inaccuracy or misprint of such formation, ROHM shall bear no responsibility for such damage.
TI ex in of us	he technical information specified herein is intended only to show the typical functions of and xamples of application circuits for the Products. ROHM does not grant you, explicitly on nplicitly, any license to use or exercise intellectual property or other rights held by ROHM and ther parties. ROHM shall bear no responsibility whatsoever for any dispute arising from the se of such technical information.
Ti ec ni	he Products specified in this document are intended to be used with general-use electronic quipment or devices (such as audio visual equipment, office-automation equipment, commu- ication devices, electronic appliances and amusement devices).
Т	he Products specified in this document are not designed to be radiation tolerant.
W P	/hile ROHM always makes efforts to enhance the quality and reliability of its Products, a roduct may fail or malfunction for a variety of reasons.
Pl ag fa sh so	lease be sure to implement in your equipment using the Products safety measures to guard gainst the possibility of physical injury, fire or any other damage caused in the event of the allure of any Product, such as derating, redundancy, fire control and fail-safe designs. ROHM hall bear no responsibility whatsoever for your use of any Product outside of the prescribed cope or not in accordance with the instruction manual.
TI sy in co of su	he Products are not designed or manufactured to be used with any equipment, device or ystem which requires an extremely high level of reliability the failure or malfunction of which hay result in a direct threat to human life or create a risk of human injury (such as a medica istrument, transportation equipment, aerospace machinery, nuclear-reactor controller, fuel- ontroller or other safety device). ROHM shall bear no responsibility in any way for use of any f the Products for the above special purposes. If a Product is intended to be used for any uch special purpose, please contact a ROHM sales representative before purchasing.
lf b ol	you intend to export or ship overseas any Product or technology specified herein that may e controlled under the Foreign Exchange and the Foreign Trade Law, you will be required to btain a license or permit under the Law.



Thank you for your accessing to ROHM product informations. More detail product informations and catalogs are available, please contact us.

## ROHM Customer Support System

http://www.rohm.com/contact/