### **PSPICE for TI** A short overview

2/22/2022 Art Kay





## Thanks for support on this

- JC Zhu
- Sean Cashin
- Carolina Gomez
- Cynthia Sosa
- Keith Nicholas
- **Collin Wells**
- István Eperjesi (Design Soft)
- Dr Michael Koltai (Design Soft)
- And many others...
- Thanks!



# Table of contents (click link to jump to section)

Getting and Installing PSPICE for TI

New Project

Adding components

Adding simulation profile & Running a Simulation

Adding plots and curves

Post processing

Using Cursors

DC Sweep

AC Sweep and Noise analysis

Parameter Stepping

Monte Carlo Analysis

Using multiple schematics and simulation profiles

Getting professional PSPICE

Uploading a Model to the web and PSPICE TI library





# Getting and Installing PSPICE for TI

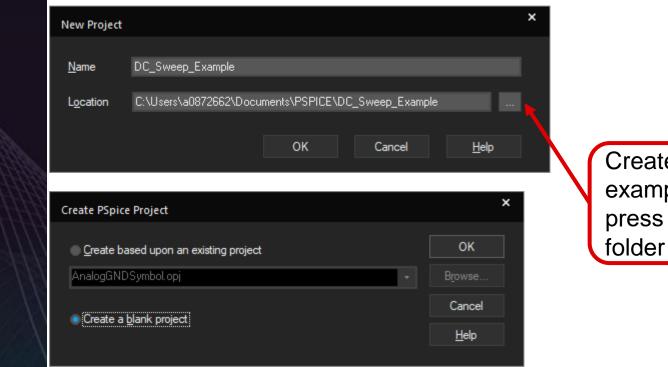
- Visit <u>https://www.ti.com/tool/PSPICE-FOR-TI</u> to make the request.
- Make sure your myTI information is complete, all required fields (marked with "\*") are filled in.
- An email will be sent to the registered email address containing a download link and an access key.
- Click the download link to get the installer. Run the installer, enter the registered email and the access key when prompted.



# New project

File	Design	Edit View	Tools	Place	Options	Window	Help	
	New				Project	t 📃	12	
	Open				Design	<b>-</b>	S 120	
	Close		Ctrl+F4		Text File			
8	Save		Ctrl+S		New Ctr	+ N	Open Ctrl + O	
	Save As							
	Save Project Archive Proj			2				New Project
	Print Previe			2				<u>N</u> ame
	Print Print Setup.		Ctrl+P	2	Ċ	uick	Start	Location
	Print Area			0			torial Videos	
	1 C:\Users			2	Conc	Training		
	2 ads8860_s			2		a come the		Create PSpie
	3 C:\Users\. 4 C:\USERS\			2	-	UNDERST	ANDING	Ø
	5 c:\users\				THE	SCHEMA	ATICS:	<u>C</u> reate b
	6 C:\Users\.					- O	<b>_</b>	AnalogGN
	7 ads8860_s							Create a
	- 8 ads8860_s				100	C. C. CORROR		
	9 C:\Users\.	\ADS8860.	cir			THE GUIDE	EMA	
	Exit				0.54		2.1.1.1.1.1.1.1.1.1.1.1.1.1.1.1.1.1.1.1	

- Select File>New>Project 1.
- 2. Choose project name and location. You may want to create a unique folder for each project. The project contains a \*.opj, \*.png, \*.dsn, and a separate folder with the net lists, simulation profiles, and other support files. If you put everything in one folder you will "mix-up" files for all your projects"
- 3. Select "create a blank project"

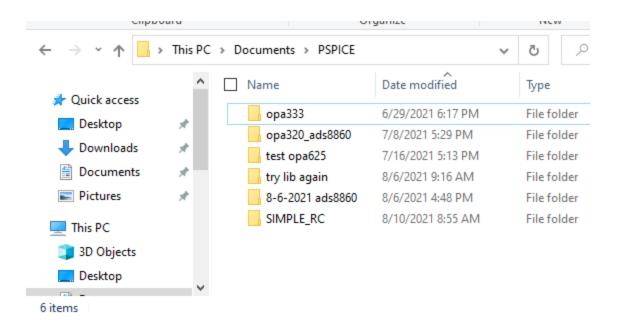


### **Back to Table of Contents**

Create a unique folder for each example. Press three dots and press "New Folder" to create a folder then select it.



### **Bad vs Good structure**



#### **Good Structure**

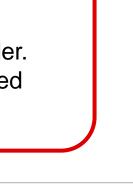
- Manually created new folder for each project.
- Each project folder contained in PSPICE main folder
- Files don't get mixed together

#### > This PC > Documents > pspice\_bad Name ~ Date modi 📌 Quick access AMP-PSpiceFiles 8/10/2021 Desktop R\_DIV-PSpiceFiles 8/10/2021 🖶 Downloads **RC-PSpiceFiles** 8/10/2021 Documents AMP 0.DBK 8/10/2021 Pictures R DIV 0.DBK 8/10/2021 RC 0.DBK 8/10/2021 💻 This PC AMP.DSN 8/10/2021 📜 3D Objects 🖉 R\_DIV.DSN 8/10/2021 Desktop 🖉 RC.DSN 8/10/2021 Documents AMP.opj 8/10/2021 R\_DIV.opj Downloads 8/10/2021 RC.opj 8/10/2021 Music AMP.png 8/10/2021 Pictures R\_DIV.png 8/10/2021 😽 Videos RC.png 8/10/2021 15 items

#### **Bad Structure**

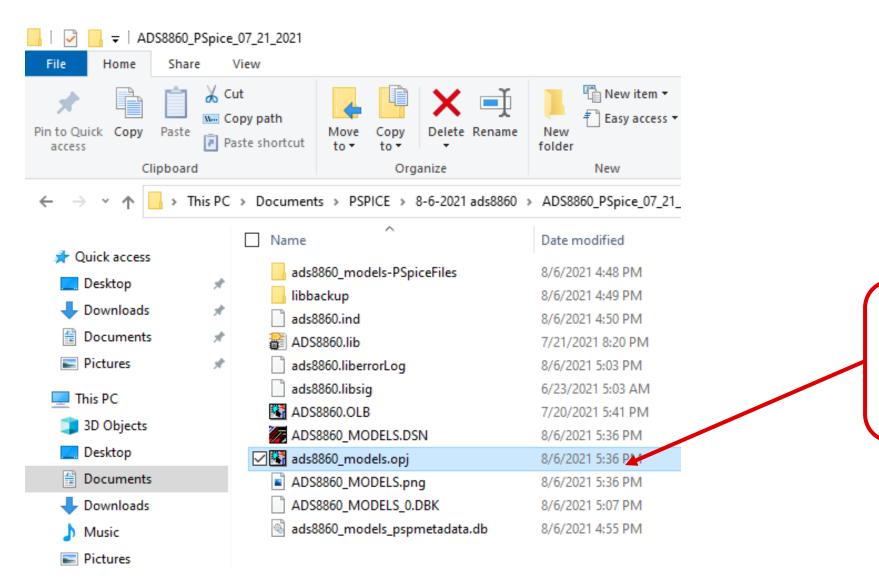
- All files mixed together
- All the PSPICE files in one folder.
- No project specific folder created automatically

	~	ē	Q	Sear
ified		Туре	^	
12:12 PM		File fo	older	
12:07 PM		File fo	older	
12:09 PM		File fo	older	
12:12 PM		DBK F	ile	
12:07 PM		DBK F	ile	
12:09 PM		DBK F	ile	
12:13 PM		DSN F	File	
12:09 PM		DSN F	File	
12:11 PM		DSN F	File	
12:13 PM		OPJ F	ile	
12:09 PM		OPJ F	ile	
12:13 PM		OPJ F	ile	
12:13 PM		PNG	File	
12:09 PM		PNG	File	
12:11 PM		PNG	File	





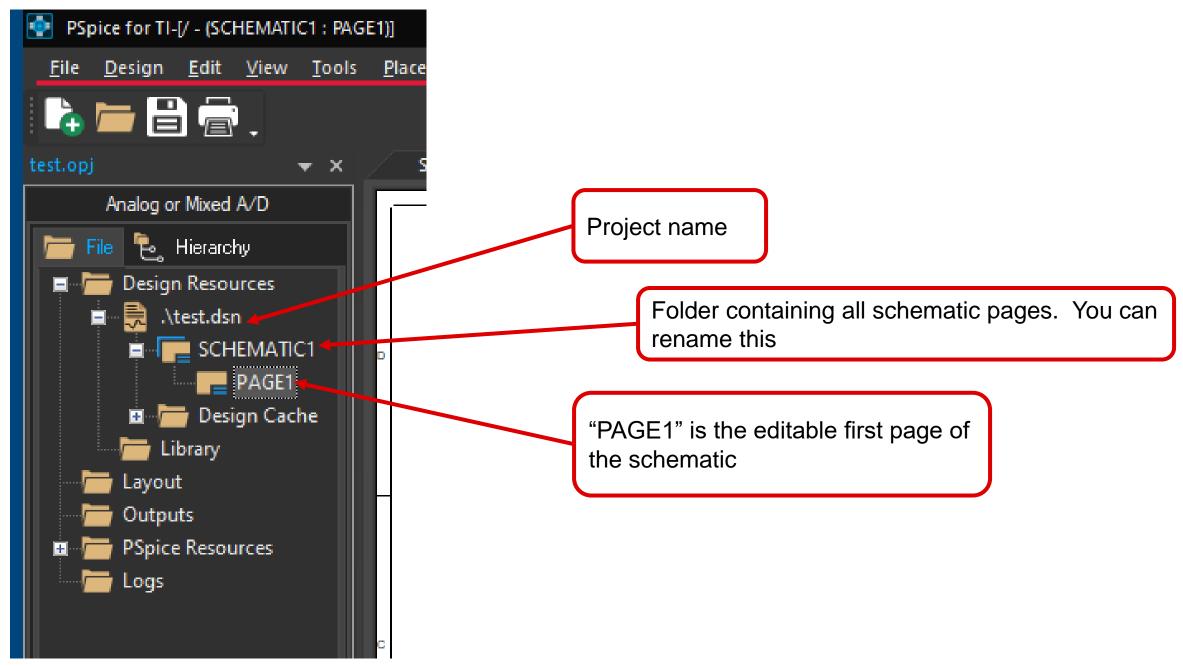
### Launch PSPICE from file explorer



If you already have a project and want to just launch form windows explorer, click on the OPJ file. This is the project file.

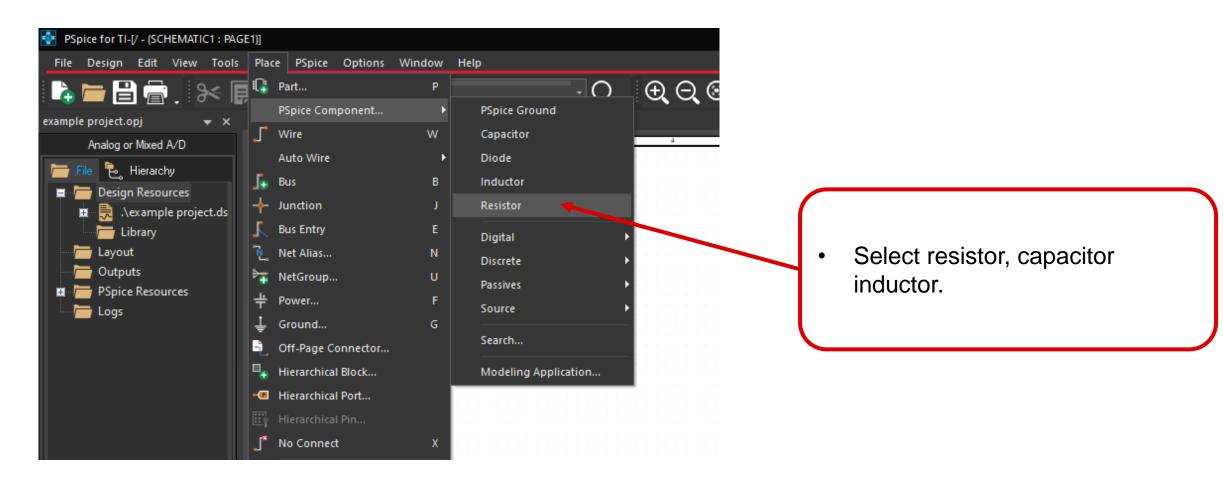


### **Find the Schematic**



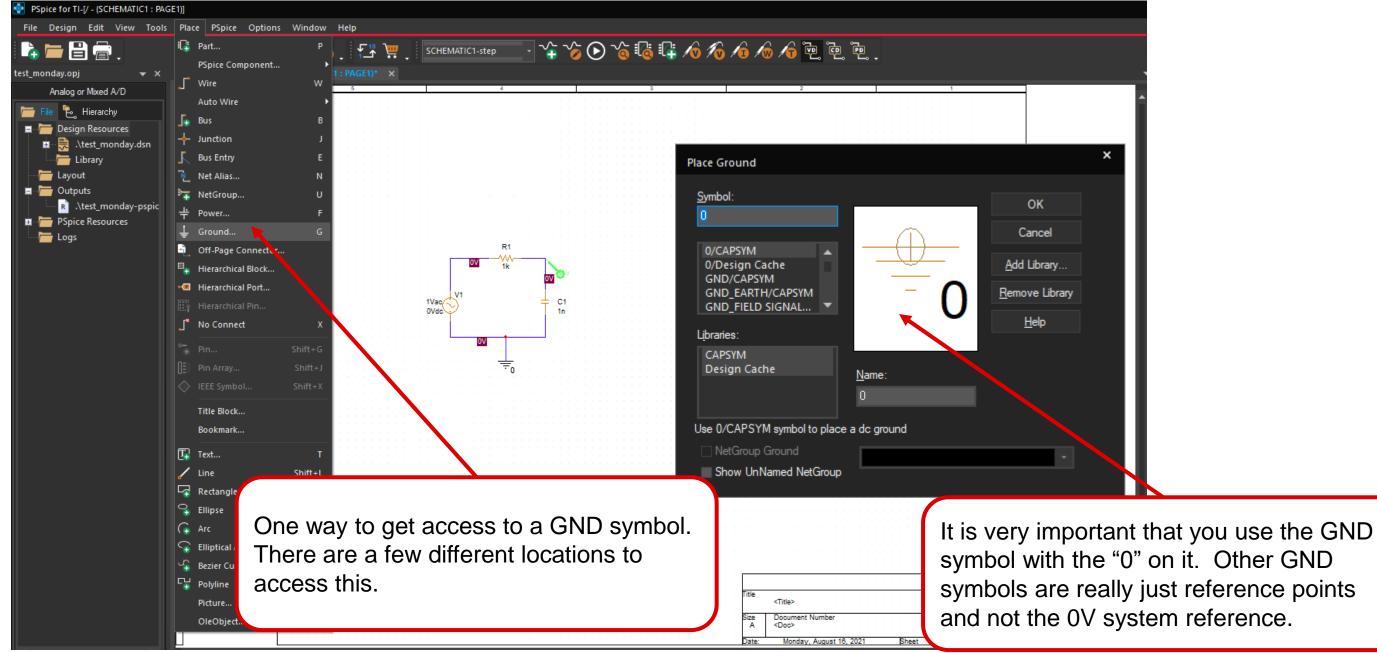


## **Adding passive components**



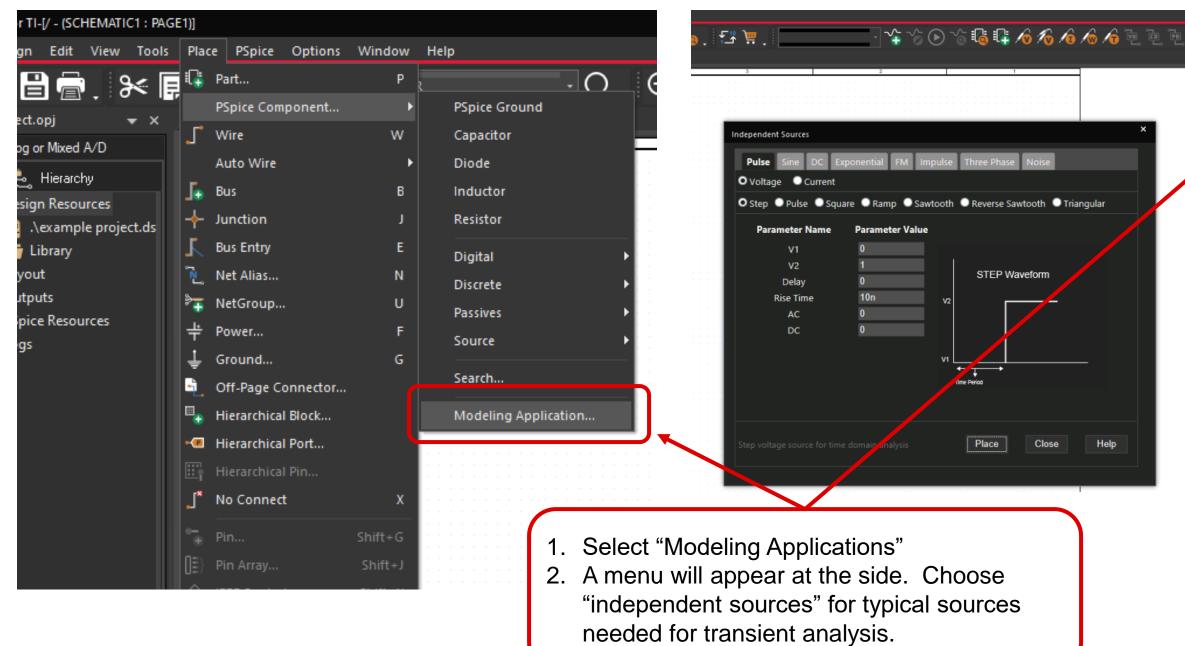


# **Adding GND symbol**





# Adding sources for transient analysis



	cadence®
<ul> <li>Modeling Application</li> </ul>	▼ ×
Power MOSFET	· · · · <b>&gt;</b>
Diodes	
Passives	г <sup>а</sup>
Sources	<u>_</u>
Independent Sources	24
PWL Sources	<u>_</u>
· · · · · · · · · · · · · · · · · · ·	└ ╠ └ <sup>®</sup> № º <sup>®</sup> <sup>™</sup> · · · · · · · · · · · · · · · · · · ·
	Ň.
	J.
	Ĩ
	<u>, </u>
	ᅷ
	Ţ
	, in the second s
	•
	12
	۰۰ 2
	<b>_</b>
	$\diamond$

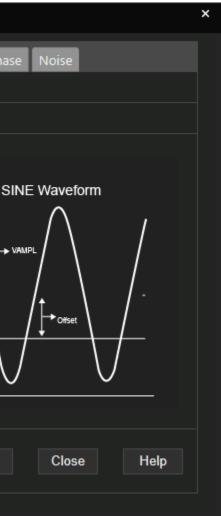


## **Adding sources continued**

Pulse       Sine       DC       Exponential       FM       Impulse       Three Phase       Noise         O Voltage       Ourrent         • Step       • Pulse       • Square       • Ramp       • Sawtooth       • Reverse Sawtooth       • Triangular         Parameter Name       Parameter Value       0       • Out ADE       • Mausfare
Parameter Name Parameter Value
V1 0
V2 5 Delay 0 Rise Time 10n Fall Time Period 10u AC 0 DC 0 V2 V1 SQUARE Waveform Time Period V2 V2 V2 V2 V2
Periodic square-wave voltage source for time domain analysis Place Close Help

Pulse Sine DC Ex	onential FM	Impulse	Three F
O Sine O Cosine O AC	Source		
Parameter Name	Parameter Va	alue	
OffSet	0		
VAMPL	154		$\sim$
Frequency	60		1
Delay	0		<i>!</i>   \
Phase	0		:  \
Damping Factor	0	5	<b>→</b> \
AC	0	Phase	i i
DC	0		

- Some common signal types
- Note that "pulse" includes square, ramp and other types





# **Find TI components**

#### PSpice for TI-[/ - (SCHEMATIC1 : PAGE1)] File Design Edit View Tools Place PSpice Options Window Help **≫ []** □ Part... $\oplus$ $\bigcirc$ $\odot$ - 0 PSpice Component... PSpice Ground xample project.op Wire w Capacitor Analog or Mixed A/D Auto Wire Diode 📂 File 🍡 Hierarchy 🔓 Bus В Inductor E Design Resources + Junction 1 Resistor 🗖 🗒 .\example project.ds Bus Entry Digital 🔚 Layout 🐧 Net Alias... N Discrete atputs 🔚 🔭 NetGroup... Passives 🚊 🛅 PSpice Resources 부 Power... E Logs Source Ground.. Search... Off-Page Connector... Hierarchical Block... Modeling Application... Hierarchical Port... No Connect

- 1. Select "Place>Pspice Component > Search"
- 2. A menu will appear at the side. Find "Texas instruments" and navigate to subcategory "Data converters"
- 3. Alternatively, search for a part in the search window.

					- 8	×					
					cādei	nce®					
	ice Part Search				▼ ×						
• <u> </u>	Hide Categories View					- -					
Ca	tegories Library										
	Favorites					J					
~	🔄 Texas Instruments					2					
	> 🖿 Amplifiers					5					
	> 🚞 Audio										
	🔪 🛅 Clocks & timing	1				ایا اللہ اللہ اللہ اللہ اللہ اللہ اللہ ا					
	🗸 🔤 Data converters										
	🗸 🦮 Analog-to-digital converters (ADCs)										
	<ul> <li>High-speed ADCs (&gt;10MSPS)</li> <li>Digital-to-analog converters (DACs)</li> </ul>										
	🔹 > 📄 Integrated s	erters		+							
	🔪 🚞 Interface					A					
	🔪 🚞 Isolation					÷					
	> 🖿 Loaic					Ţ					
	All Categories 🔹 🔻				Q	÷					
	PART NAME	DES	CRIPTION			- -					
7	ADC3444	Qua	d-channel 12-b	it 25-MSPS	anal	-1					
3	ADC34J45	Qua	d-Channel, 14-	Bit, 160-MS	SPS A	E.					
3	AFE5803	Fully	/ Integrated, 8-	Channel Ul	traso						
3	AFE5807	8-cł	annel analog fi	ont end wit	th pa	L.*					
=	AFE5808	8-cł	annel low nois	e analog fro	nt en	٠.					
=	AFE5808A	Fully	/-Integrated, 8-	Channel Ul	traso	<b>_</b>					
=	AFE5809	Fully	/-Integrated, 8-	Ch Ultrasou	und A	* 1 \$					
	DAC5311	8bit	, Single Channe	el, 80uA, 1.	8V-5						
						THEN					

### Back to Table of Contents

		▼ ×	
Categories View			
es Library			1.5
vorites			L
xas Instruments			₽.
Amplifiers			5
Audio			- <b>(</b> )
Clocks & timing			ب <b>و سي دي دي ب</b> → ↓ + - ↓
Data converters			
Interface			N
Isolation			
Logic			-
Microcontrollers (MCU	s) & processors		
Motor drivers			$\mathbf{\lambda}$
Power management			÷
RF & microwave			÷
tegories 🔻 OPA	1611	Q	÷
PART NAME	DESCRIPTION		•
OPA1611	1.1nV/OHz Noise, Low Power, Pr	ecisi	-@
			0-
			-h

>

>

>

>

>

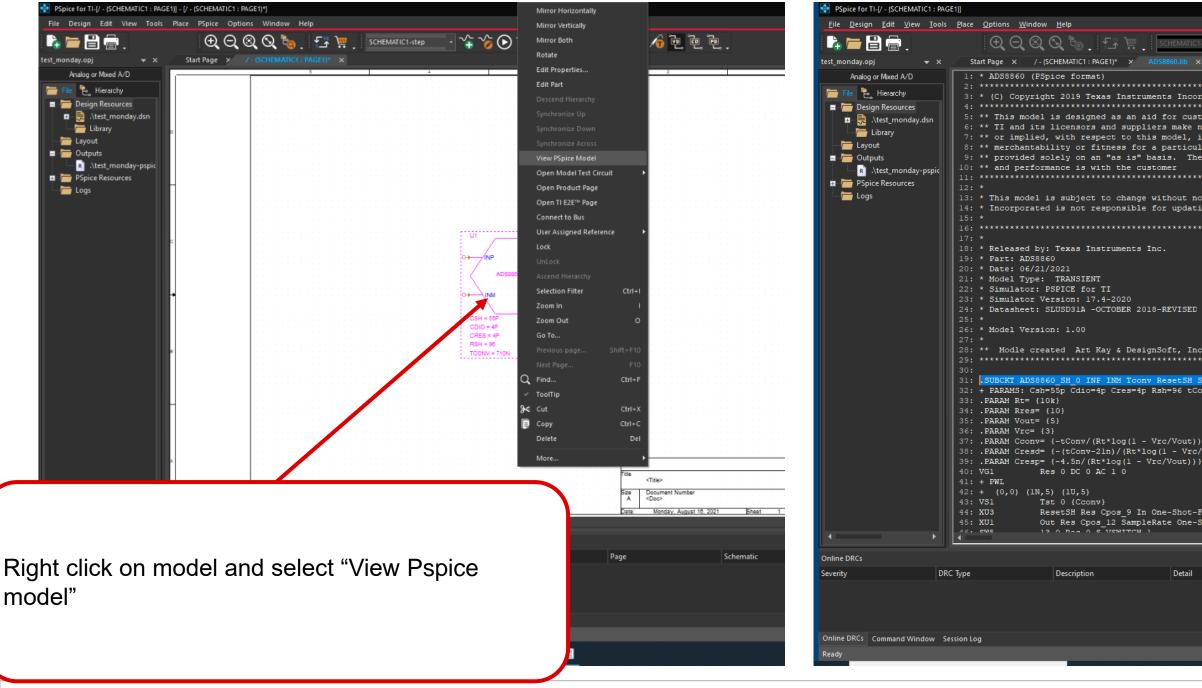
>

>

>



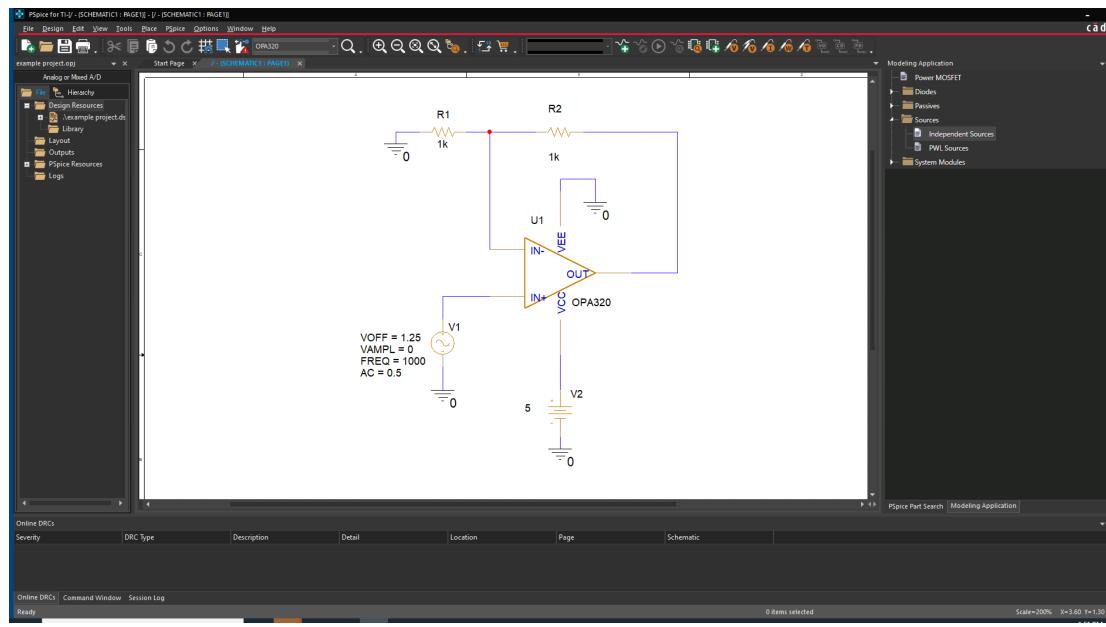
### **View PSPICE model**



-step 🔹 🌱 🎸	•	di 🕼 🔏	<i>f</i> o /o	A
porated. All right comers of Texas In to warranties, eit noluding the warr ar purpose. The entire risk as to thice. Texas Instru- ng this model.	ther expression of the second	****** ssed		
DECEMBER 2018	*****	****		
ampleRate Tacq V: onv=710n	amp			
} Vout))}				
<sup>r</sup> ast_0 Shot-Fast_0				
	Location		Page	



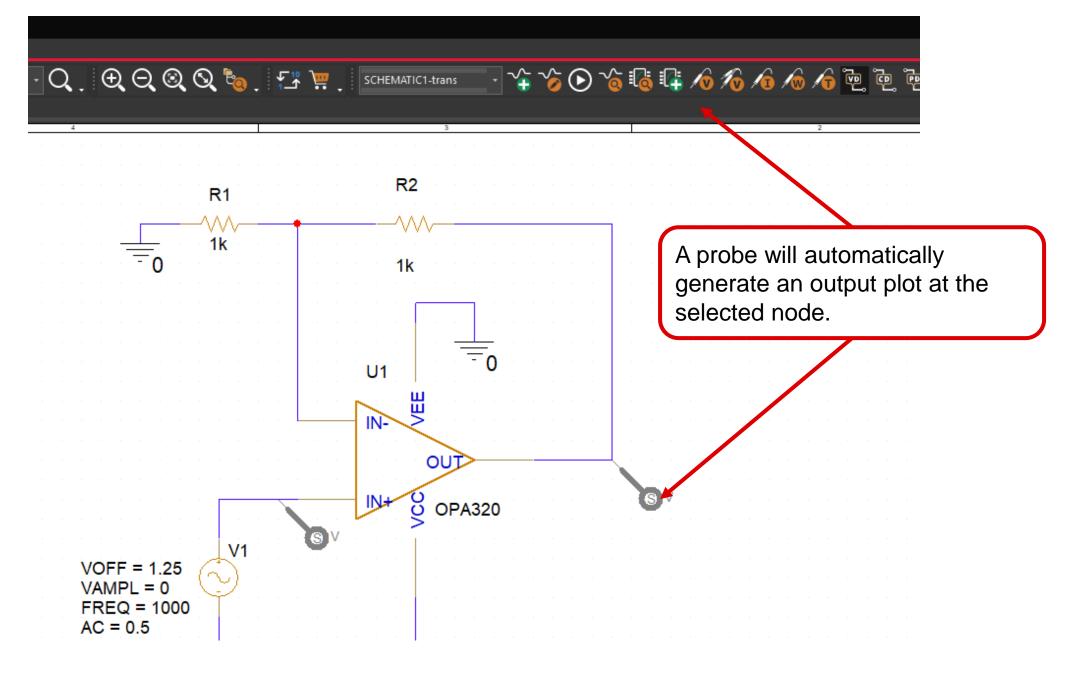
### Use shortcut "w" to wire







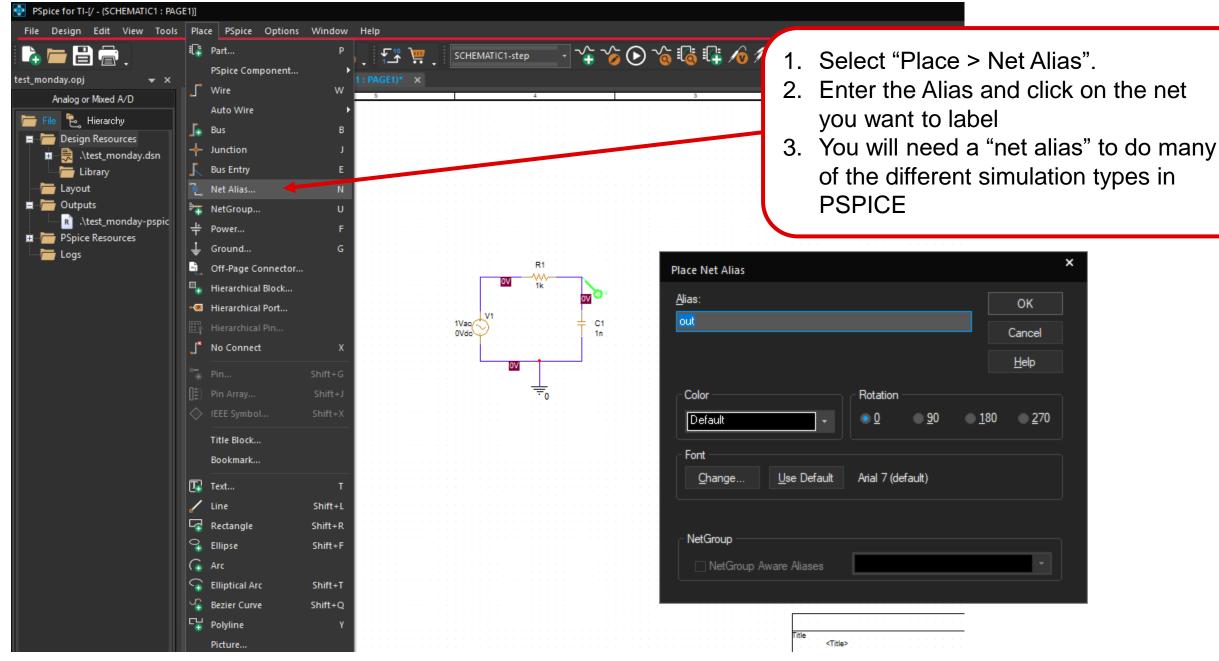
# Add probes (voltage markers) to input and output







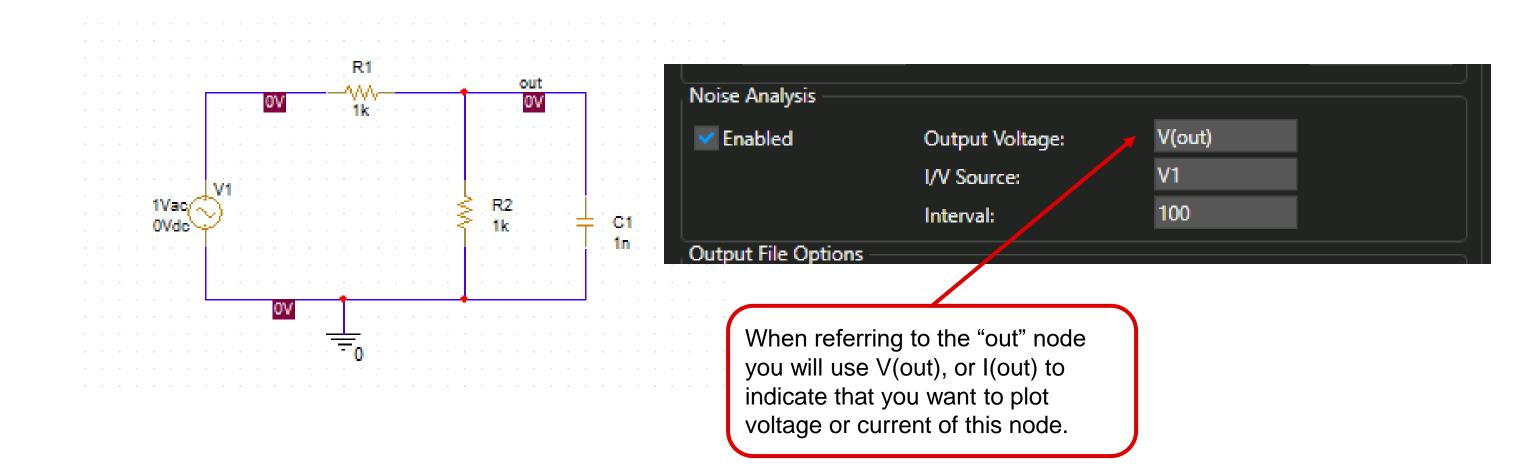
### Add net alias label







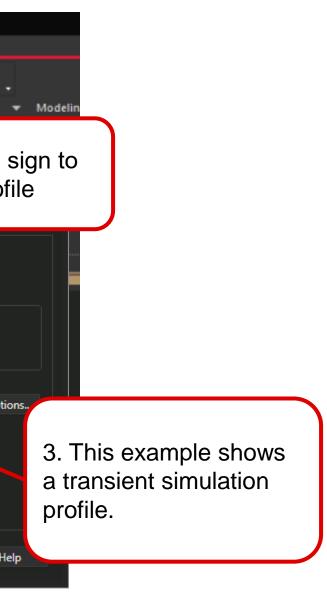
### The net alias is very important.





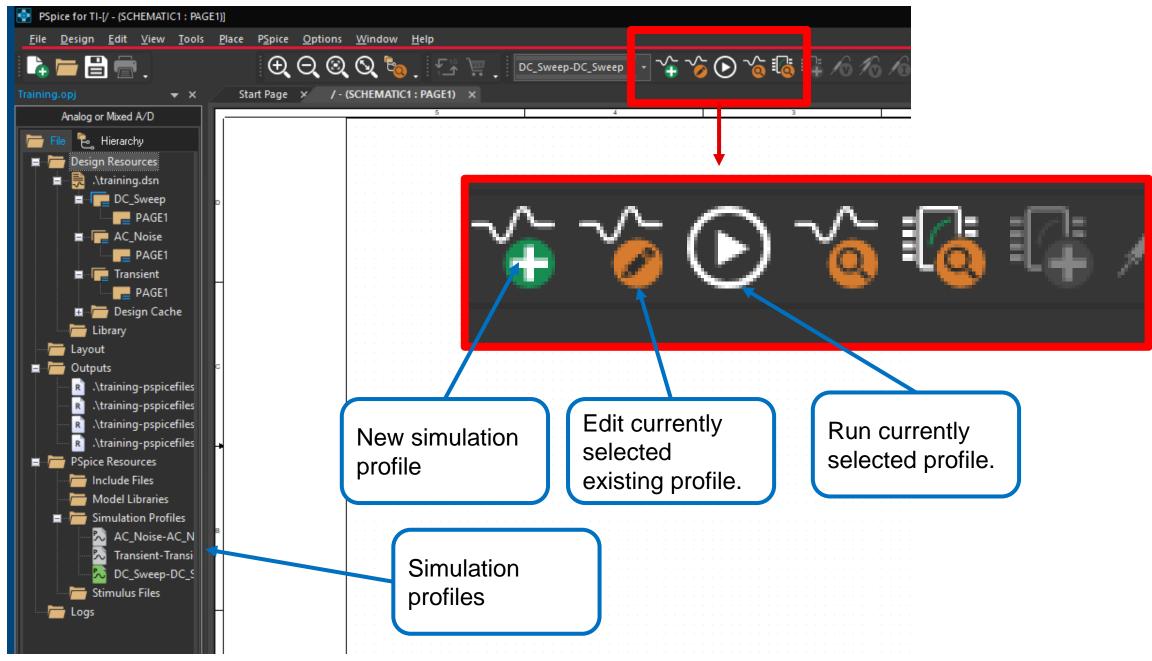
## Add "simulation profile" - transient

Help				
OPA320 - Q . 🕀 🖯	2 🔍 🔍 🍡 . 🖅 🛒 .	- 🏠	🏠 🕟 🖒 🍓 🕻	🛓 ⁄o ⁄o ⁄a \land 👘 裡 裡 .
C1 : PAGE1) ×				
A A A A A A A A A A A A A A A A A A A	Simulation Settings - trans	3 R2		1. Click on the green "+" get a new simulation pro
Name:       Create         trans       Cancel         Inherit From:       Cancel         none          Root Schematic:       SCHE MATIC1	General Analysis Configuration Files Options Data Collection Probe Window	Analysis Type: Time Domain (Transient) Options: General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep)	Run To Time : Start saving data after : Transient options: Maximum Ctep Size Skip initial transient bi	10m       seconds (TSTOP)         0       seconds         seconds       seconds         seconds       seconds
2. Choose a name for the simulation profile. You can		Save Bias Point Load Bias Point Save Check Point Restart Simulation	Run in resume mode	Output File Opti
save multiple simulation profiles if you want to quickly re-do your simulations.			ОК	Cancel Apply Reset H



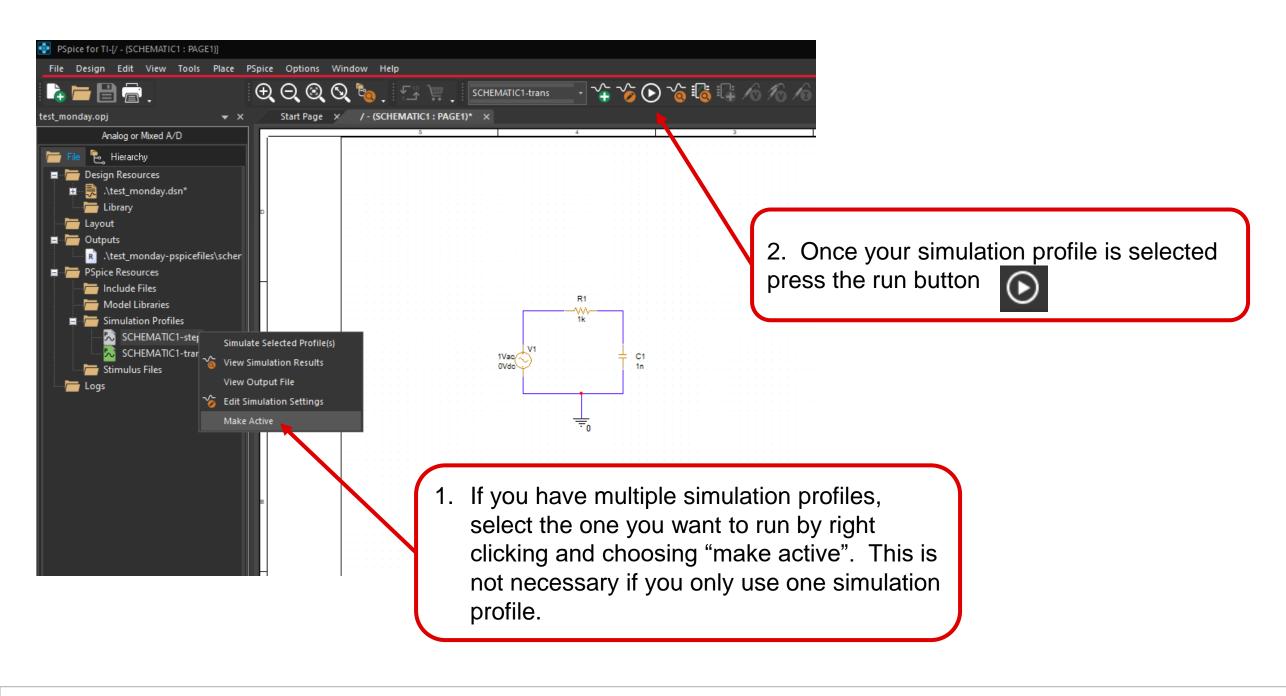


### **Simulation profiles**



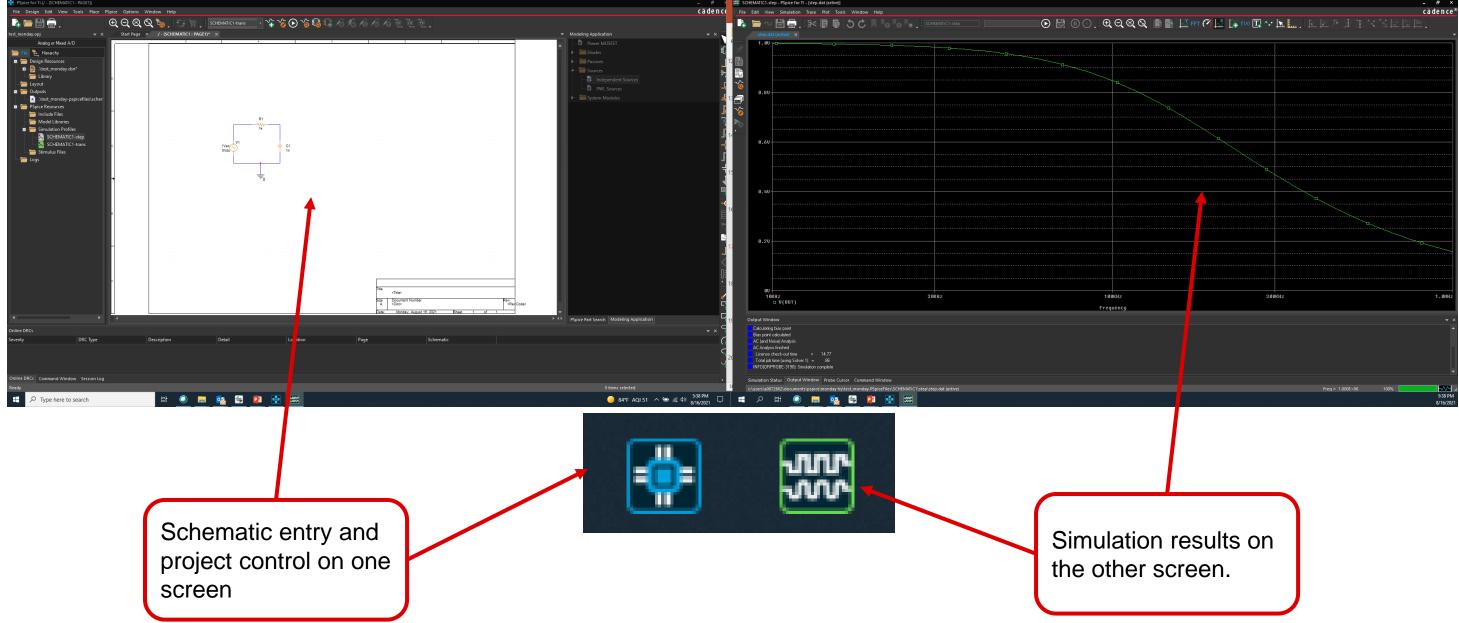


# **Running simulation**





### **Results**





### Adding plots and curves

SCHEMATIC1-step - PSpice for Ti - [step.dat (active)]	SCHEMATIC1-step - PSpice for TI - [step.dat (active)]	
File Edit View Simulation Trace Plot Tools Window Help	File Edit View Simulation Trace	3. Select the tra
step.dat (active) ×  1. Right click on the blank area to left of y-axis and select "add plot"  Add Trace  Add Trace  Add Plot  Add YAxis   Underster Pro  Delete YAxis  Ctrl = Shift = Y  Com In  Zoom Out  Zoom In  Zoom Out  Zoom Fit	step.dat (active) ×     blank plot area and select "Add trace".     Add Trace     Add Yatis     Delete Plot     Delete Plot     Delete Y Axis     SEL>>     Log X   Log Y   Fourier   Zoom Fit   Zoom Fit     Cursor On	Add Traces Simulation Output Variables * Frequency V(0) V(C1:1) V(C1:2) V(out) V(C1:2) V(out) V(C1:2) V(out) V(1:+) V(N1:-) V(1:
Cursor On       Paste         Mark Data Point       Add Text Label         Show All Traces       Hide All Traces	0.50     Mark Data Point       Add Text Label	V1(V1) V2(C1) V2(R1) V2(V1)
80   1 0KHz 3 0KHz □ V(0UT)	9U	Full List
Output Window         Calculating bias point         Bias point calculated         AC (and Noise) Analysis         AC (and Noise) Analysis         AC Analysis finished         License check-out time       = 14.77         Total job time (using Solver 1)       = .86         INFD(ORPROBE-3190): Simulation complete         Simulation Status       Output Window         Probe Cursor       Command Window         If       Image: Command Window	Output Window         Calculating bias point         Bias point calculated         AC (and Noise) Analysis         AC cand Noise) Analysis         AC Analysis finished         License check-out time       = 14.77         Total job time (using Solver 1)       = .86         INFD(DRPR0BE-3190): Simulation complete         Simulation Status       Output Window         Probe Cursor       Command Window         Image: Compared transmission of the cursor       Command Window	Trace Expression: V(out) 4. Press "ok" and added to the select

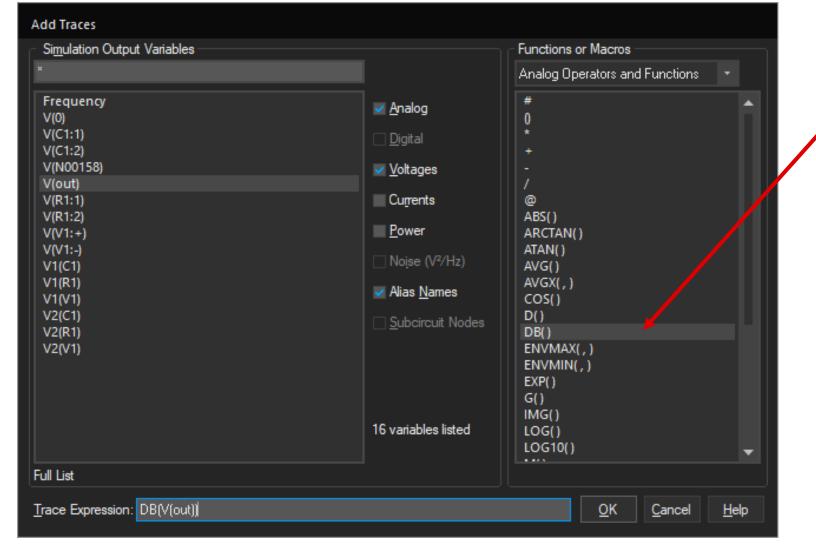
#### **Back to Table of Contents**

#### e trace you want to plot. In to uncheck some the current and power to easier to sort through.

	Functions or Macros
	Analog Operators and Functions 🔹
<mark>. ∠</mark> nalog	# 🔺
🗌 <u>D</u> igital	*
<mark>✓ <u>V</u>oltages</mark>	
Currents	/ @ ABS()
<u>P</u> ower	ARCTAN()
□ Noįse (V²/Hz)	ATAN() AVG()
🗹 Alias <u>N</u> ames	AVGX(,) COS()
□ <u>S</u> ubcircuit Nodes	D() DB()
16 variables listed	ENVMAX(,) ENVMIN(,) EXP() G() IMG() LOG() LOG()
	<u>Q</u> K <u>C</u> ancel <u>H</u> elp
 e trace w d plot.	rill be

TEXAS INSTRUMENTS

# Post processing



You may want to perform a mathematical operation on the waveform. You can select the math function here. DB() converts the voltage plot to decibels.

- 1. First press the math operation
- 2. Second select the signal
- In this example: DB(V(out)) will plot V(out) 3. in decibels.

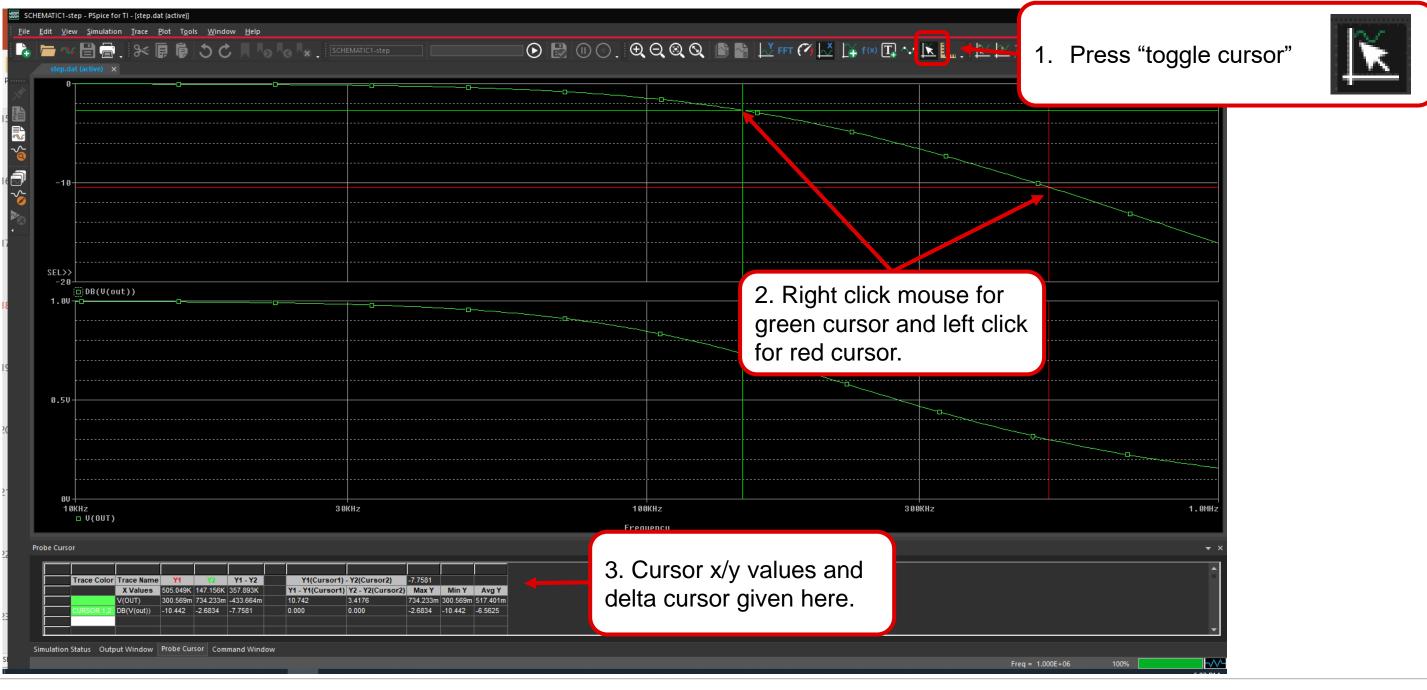


## **Post processing functions**

ABS(x) |x| **SGN(x)** +1 (if x > 0), 0 (if x = 0), -1 (if x < 0) **SQRT(x)** x<sup>1/2</sup> EXP(x) e<sup>x</sup> **LOG(x)** *In*(x) (log base *e*) **LOG10(x)** *log*(x) (log base 10) **M(x)** magnitude of x **P(x)**phase of x (result of degrees) **R(x)** real part of x **IMG(x)** imaginary part of x **G(x)** group delay of x (result in seconds) **PWR(x,y)** x<sup>y</sup> **SIN(x)** *sin*(x) (*x* in radians) COS(x) cos(x) (x in radians) **TAN(x)** tan(x) (x in radians) **ATAN(x)**  $tan^{-1}(x)$  (result in radians) **ARCTAN(x)** tan<sup>-1</sup>(x) (result in radians) d(x) derivative of x with respect to the X axis variable. **s(x)** integral of x over the range of the X axis variable. AVG(x) running average of x over the range of the X axis variable. AVGX(x,d) running average of X (from x-d to x) over the range of the X axis variable. **RMS(x)** running RMS average of x over the range of the X axis variable. **DB(x)** magnitude in decibels of x. **MIN(x)** minimum of the real part of x. **MAX(x)** maximum of the real part of x.



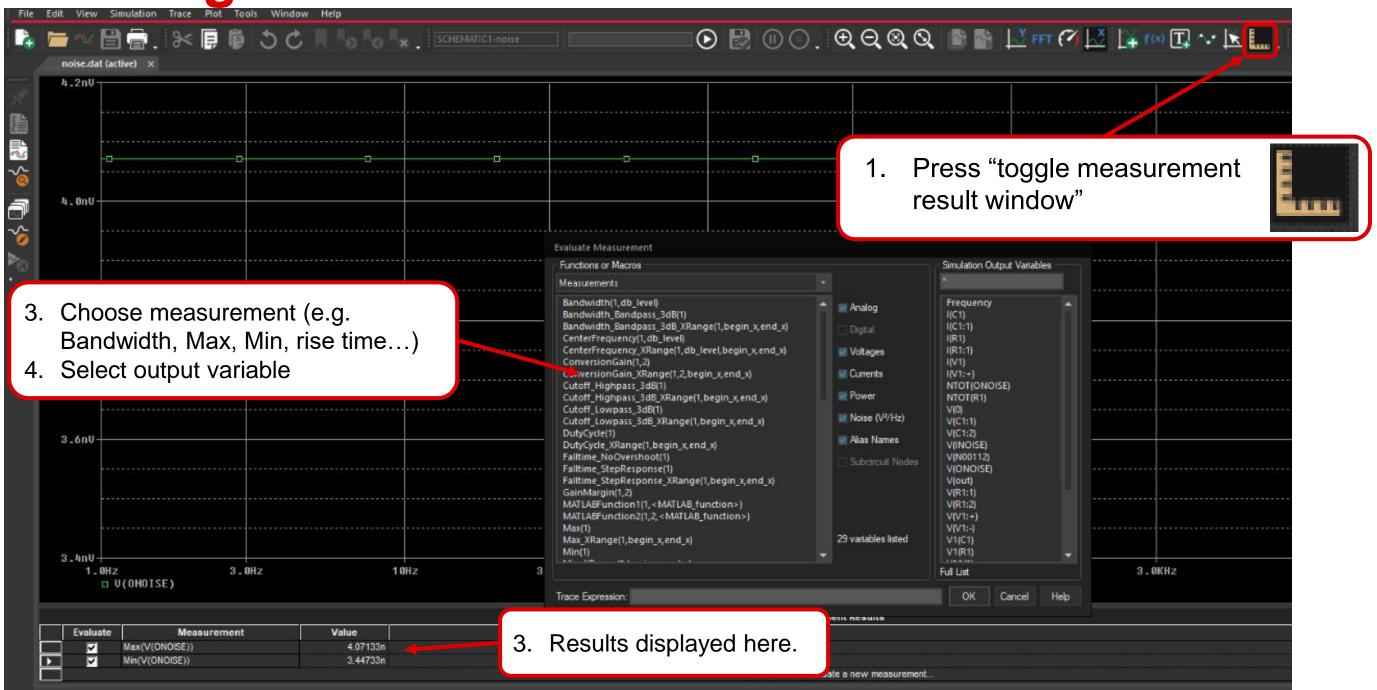
### **Using cursors**



#### **Back to Table of Contents**

#### **TEXAS INSTRUMENTS**

### **Adding Measurements**

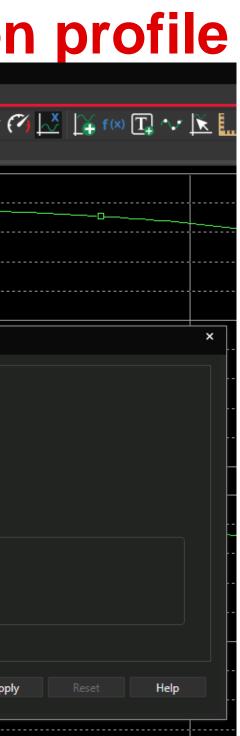


	-		-	-	-	-	-	-	-	-	-	-		-	-	-		-
	-	4		-				-		-	1				-		-	-
																		-
																		- 1
	-		-	-	-	-	-		-	-		-	-	-	-	-	-	
-	-			-		*		1		-							-	÷



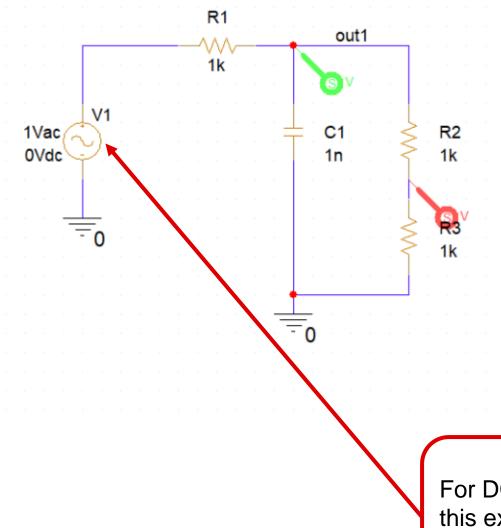
## **Remember plot window setup in simulation profile**

SCHEMATIC1-AC_Noise - PSpice for TI - [AC_Noise.dat (active)]			
File Edit View Simulation Trace Plot Tools Window Help			
	MATIC1-AC_Noise	▶ ₽ 00. €, Q, Q, 🖻 🕯	
2.8nU 2.8nU 2.8nU			
This is very helpful to retain all the different plots, and calculations in a simulation profile. 1. Click on edit simulation profile button. 2. Select probe window 3. Choose "last plot"	Simulation Settings - AC_Noise General Analysis Configuration Files Options Data Collection Probe Window	<ul> <li>Display Probe window when profile is opened</li> <li>Display Probe window:         <ul> <li>during Simulation.</li> <li>after simulation has been completed.</li> </ul> </li> <li>Show         <ul> <li>All Markers on open schematics.</li> <li>Last Plot</li> <li>Nothing.</li> </ul> </li> </ul>	
		OK Cancel	Ар





### **DC** sweep

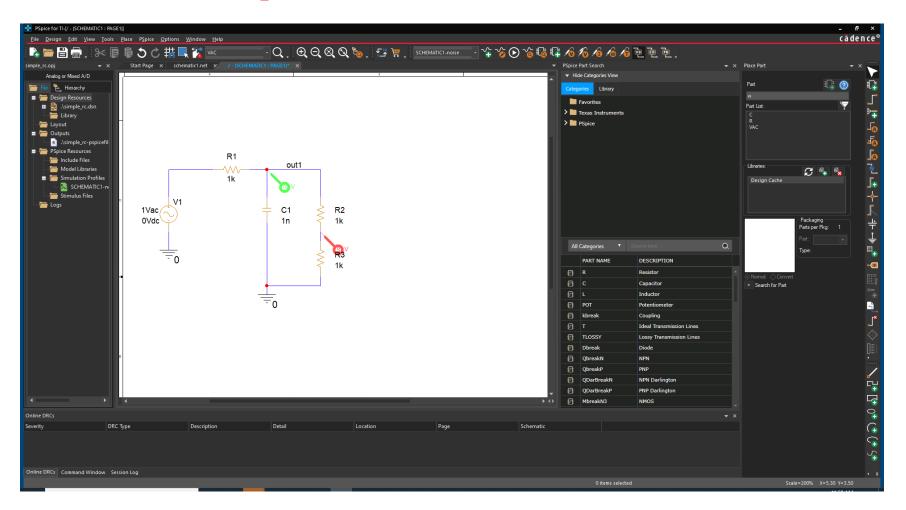


General	Analysis Type:	Sweep Variable				
Analysis Configuration Files Options Data Collection Probe Window	DC Sweep   Options:  Primary Sweep  Secondary Sweep  Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point Load Bias Point	<ul> <li>Voltage source</li> <li>Current source</li> <li>Global parameter</li> <li>Model parameter</li> <li>Temperature</li> <li>Sweep Type</li> <li>Linear</li> <li>Logarithmic</li> <li>Decod</li> <li>Value List</li> </ul>	End V	V1	0 5 .1	
	/ the voltage source yo / the sweep type and i	ou want to swee	Cancel Apply	Reset	Help	

For DC this exa



### **DC Sweep**



This shows the example results for DC sweep where two probes measure the outputs.

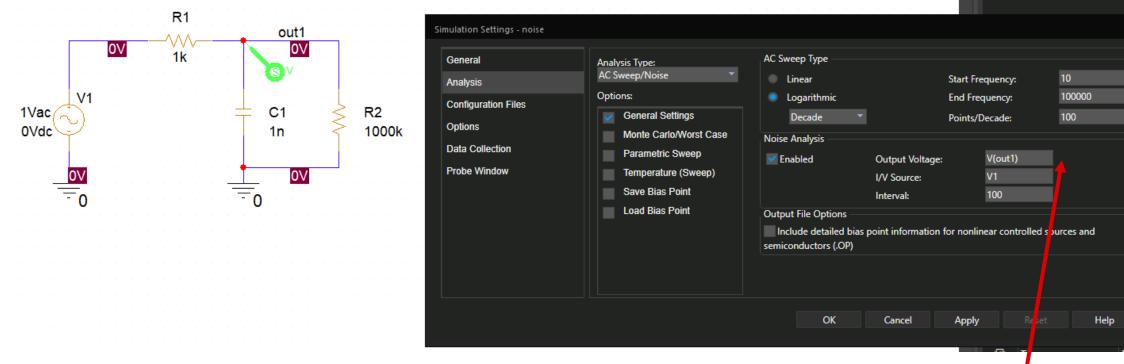
#### SCHEMATIC1-noise - PSpice for TI - [noise.dat (active)] File Edit View Simulation Trace Plot Tools Window Help 82. - ≫ 🖪 h 4.00 × **|** ∼₀ 3.00 **≣** ∕∕ 2.0V 1.0V 2.0V ØŲ $\Box$ V(OUT1) $\diamond$ V(R2:1) Output Window

	Reading and che	cking circuit		
	Circuit read in and	l checked, no	errors	
	DC Analysis			
	DC Analysis finish	ed		
	License check-c	out time		12.2
	Total job time (u:	sing Solver 1)		.8
	INFOIORPROBE	3190): Simula	tion.co	mol
•				
Siı	mulation Status	Output Win	ndow	Co





## **AC Sweep and Noise analysis**



Where *Output voltage: V(net\_name)* is used to plot voltage noise for *net\_name*. In this case the net name is *out1* V/I Source: source\_name – In this example the source name is V1





### **Noise analysis**

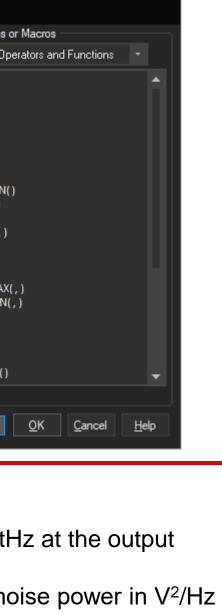
Si <u>m</u> ulation Output Variables		- Functions or Macros	
		Analog Operators and Functions	
Frequency NTOT(ONOISE)	<u> </u>	COS() D()	•
NTOT (R1) NTOT (R2)	<u>D</u> igital	DB() ENVMAX(,)	
	<u>V</u> oltages	ENVMIN(,) EXP()	
	Currents	G()	
	Power	IMG() LOG()	
	<mark>, ✓</mark> No <u>i</u> se (V²/Hz)	LOG10() M()	
	. ✓ Alias <u>N</u> ames	MAX() MIN()	
	Subcircuit Nodes	P()	
		PWR(,) R()	
		RMS() S()	
		SGN()	
	4 variables listed	SIN() SQRT()	
		TAN()	•
Full List			
Trace Expression: SQRT(S(NTOT(ONOISE)))		<u>O</u> K <u>C</u> ancel	<u>H</u> elp
Integrated Total DM	C naise		
Integrated Total RM			
<ul> <li>"S" is the integral i</li> </ul>	function		
	re root function		
• SQRT is the squa			
<ul> <li>SQRT is the squa</li> <li>NTOT(ONOISE) =</li> </ul>		$V^2/H_7$	
<ul> <li>SQRT is the squa</li> <li>NTOT(ONOISE) =</li> </ul>		V²/Hz	
•	Noise power in	V²/Hz	
•		V²/Hz	

#### **Back to Table of Contents**

Simulation Output Variables		Functions
×		Analog C
Frequency V(INOISE) V(ONOISE) V(out) V(V1:+) V(V1:-)	<ul> <li>✓ <u>A</u>nalog</li> <li>□ <u>D</u>igital</li> <li>✓ <u>V</u>oltages</li> <li>□ Currents</li> <li>□ <u>P</u>ower</li> <li>□ Noise (V<sup>2</sup>/Hz)</li> <li>□ Alias <u>N</u>ames</li> <li>□ <u>S</u>ubcircuit Nodes</li> </ul>	# 0 * - ABS() ARCTAN ATAN() AVG() AVGX(, COS() D() DB() ENVMAI ENVMIN EXP() G() IMG()
	6 variables listed	LOG() LOG10(
Full List		

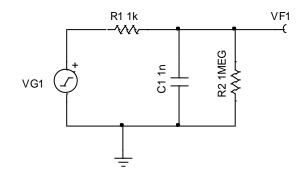
#### **Voltage Noise Spectral Density**

- V(ONOISE) is the noise density in V/rtHz at the output
- V(INOISE) is the input noise density
- Note: the "Noise" check box will give noise power in V<sup>2</sup>/Hz

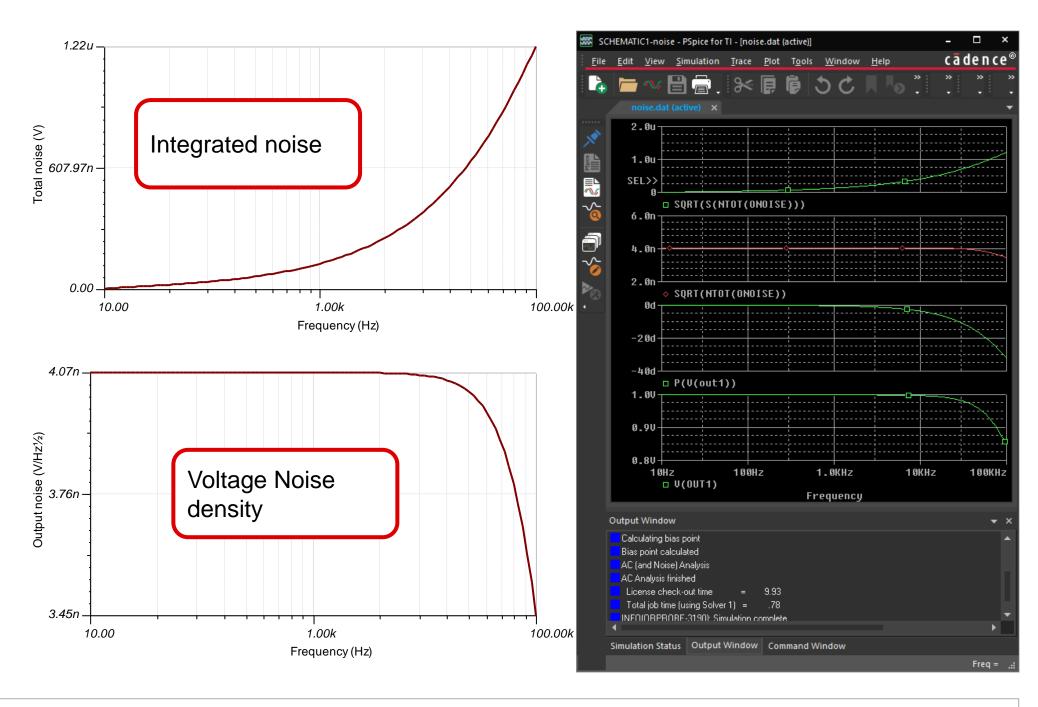




## **Noise analysis**

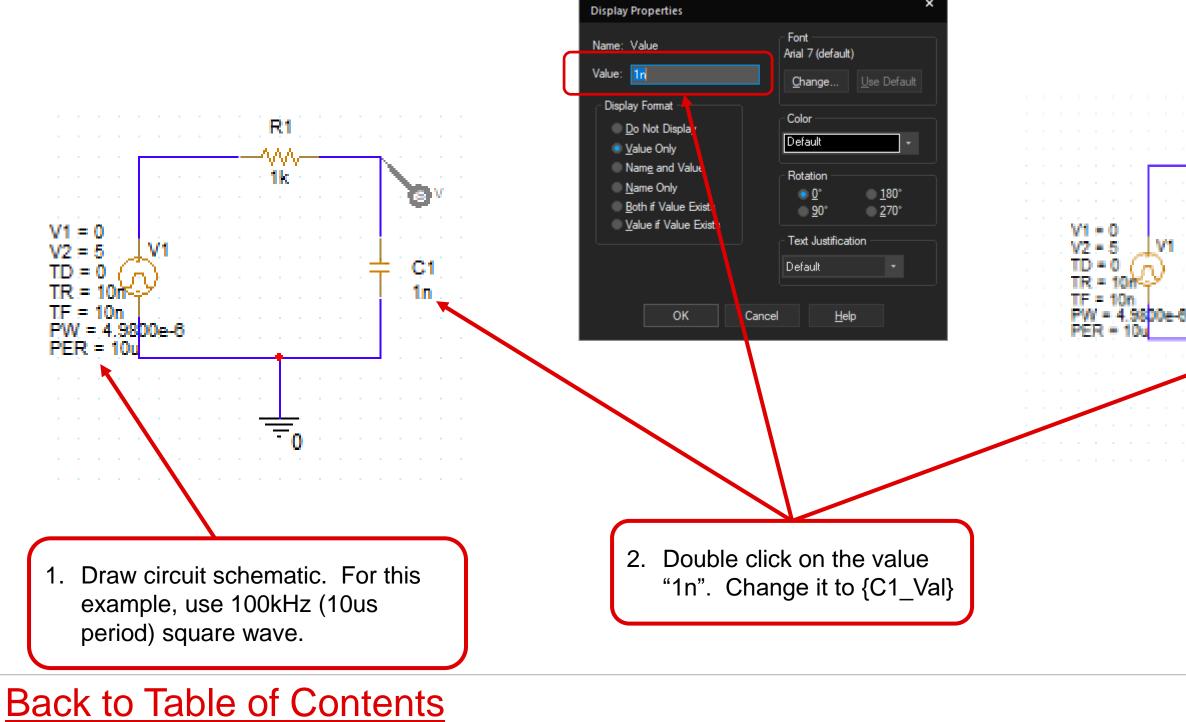


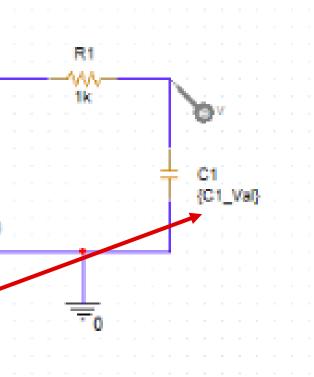
This slide shows the two traces for noise density and integrated noise match TINA. By default PSPICE plots noise power which isn't very useful





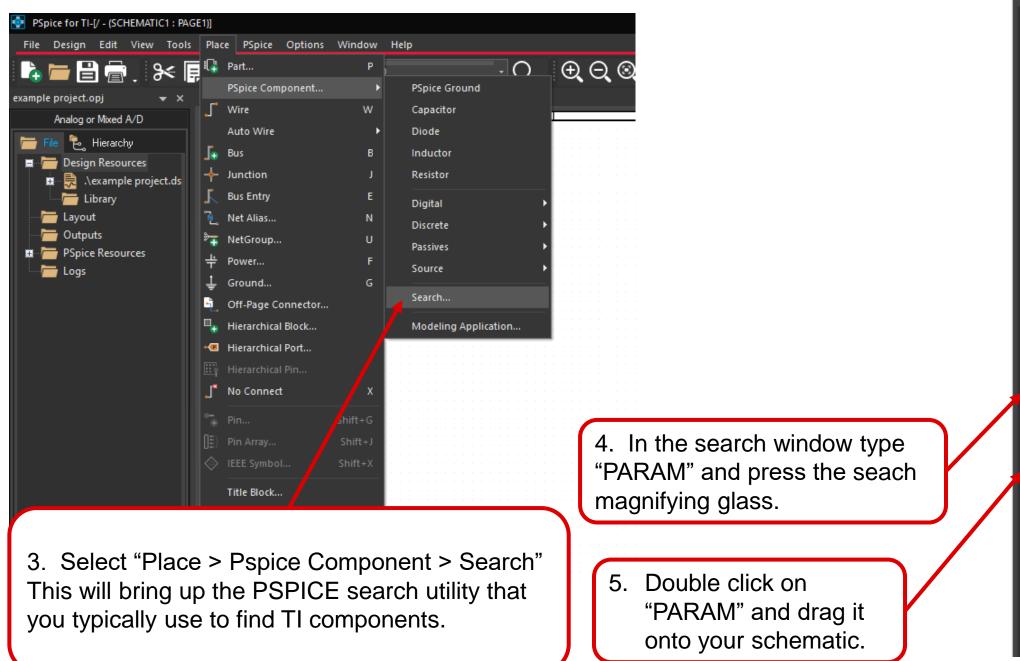
# Parameter Stepping: Edit schematic value







### Parameter Stepping: Find PARAM



### **Back to Table of Contents**

	<b>-</b> ×		
		÷٦	
		ا^L ا	
		≥_	
		@	
,		20	
8		Jø.	111
igital co	onverters (ADCs)	ז <del>אי ו</del> פ ייפ יפ יבי י <del>י</del>	
	onverters (DACs)		
	(<=10MSPS) 5		
special	function data converters		2
		$\mathbf{F}$	
transl	ation	ᅷ	
: u ansi	-	1	
PAR	AM Q	Į.	111
	DESCRIPTION		
	Delay Line: Parameterized delay		
	Used To Specify Pre-Defined Paramet	ΞŶ	
,	Louier Monaillerenten)	÷	
P	N Channel Power MOSFET (paramete	-	
•	P Channel Power MOSFET (parameter	۲×	
	Pulldown Resistor: general parameter		
	Pullup Resistor: general parameterize	$\sim$	
	Used To Specify Pre-Defined Subpara	UE}	
		•	

Hide Categories View

Texas Instruments

> Amplifiers

Clocks & timina

V Data converter

> Analog-to-d

🗸 🔤 Digital-to-a

> Integrated s

> Interface

> Isolation

All Categories

I DELAY

1

0

Logic & voltage

PART NAME

OTTER\_DIOD

POWER\_NMOS\_

PULLDOWN

SUBPARAM

PULLUP

PARAM

Precision

Library

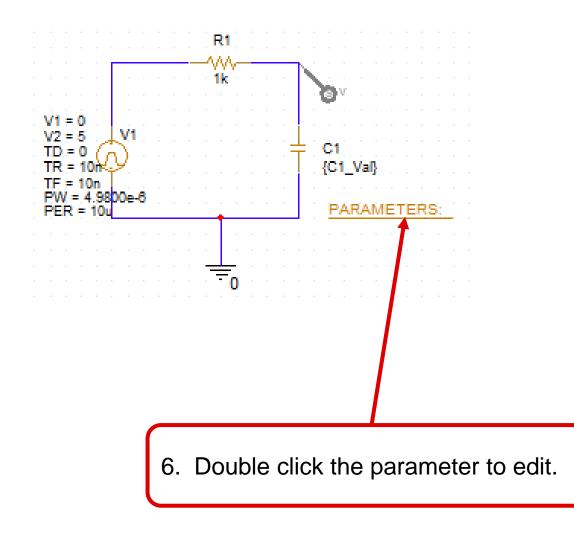
Categories

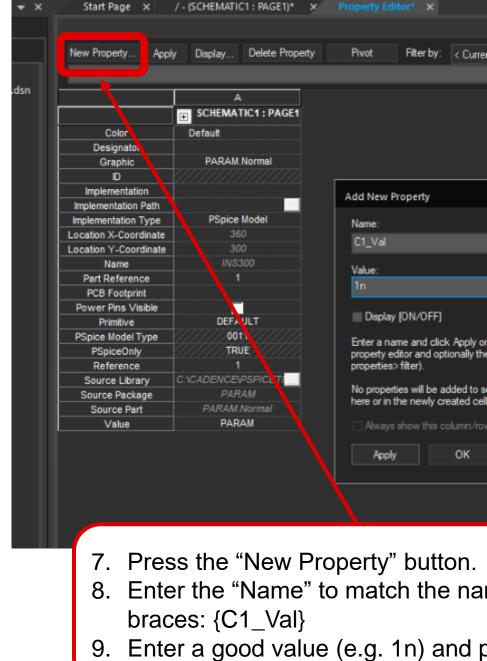
Favorites

> 🖿 Audio



### Parameter Stepping: Edit PARAM





### **Back to Table of Contents**

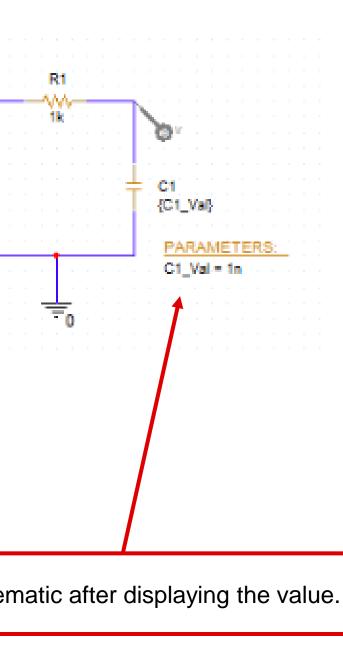
urrenit properties >		•	
	×		
ly or OK to add a column/row to the			
y the current filter (but not the <current< td=""><td></td><td></td><td></td></current<>			
to selected objects until you enter a value			
I cells in the property editor spreadsheet.			
1/row in this filter			
Cancel Help			
ame in the			
press ok.			
<table-of-contents> Texas Instr</table-of-contents>	RUM	<b>IEN</b>	TS

36

## **Parameter Stepping: Display value**

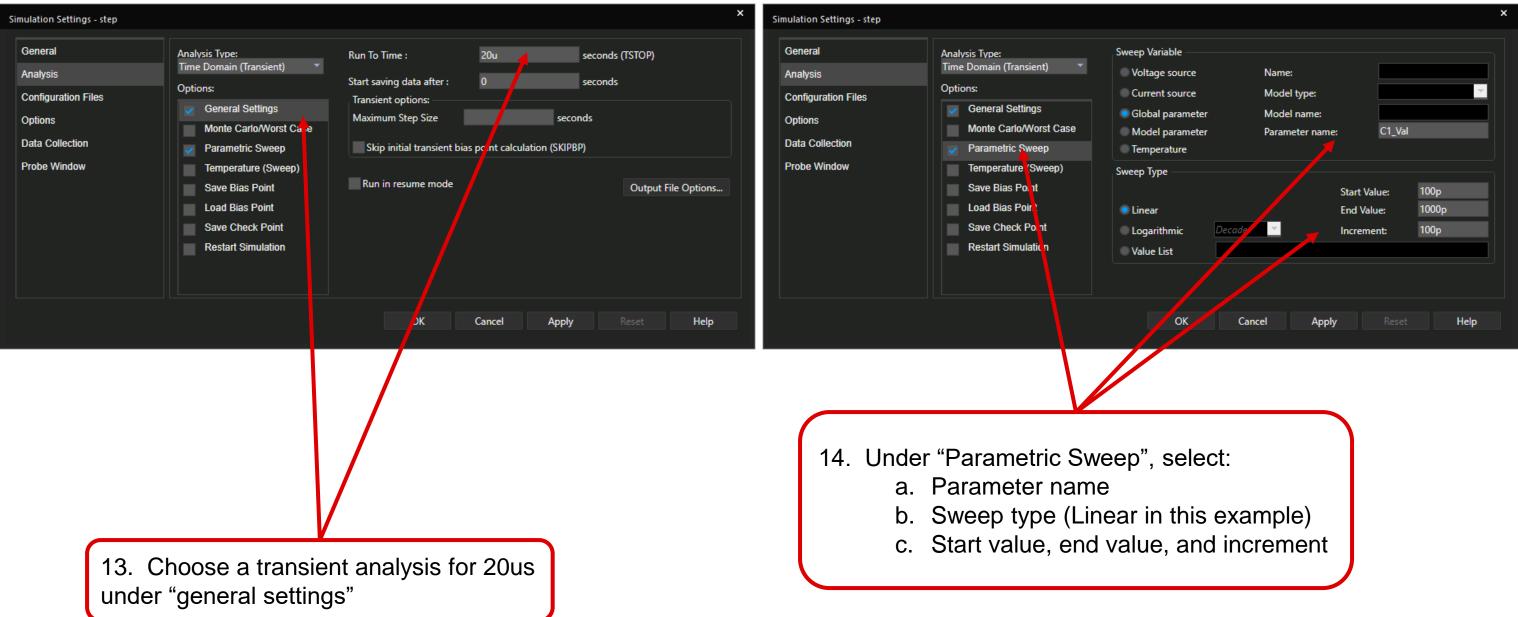
Color       Default       Pivot         signator       ParAl Morr         Display Properties         mentation       Display         Display Properties         Name:       C1_Val         Arrange       Genge         Name:       C00         X-Coordinate       60         X-Coordinate       60         Y-Coordinate       700         Name       N 300         Reference       Yalue         Piss Visable       DEF. ULT         Model Type       Yalue         PARAM Normal       PARA M         Yalue		A	
1_Val       In         Color       Default         isgnator       Prot         isgnator       PARA Norm         Edit       Delete Property         Delete Property       Display         intation Type       PSpice Model         X-Coordinate       60         Y-Coordinate       60         Y-Stable       In         Pins Visible       In         Initation Type       0 (11         Scoondy       11         Value       PARAM         Yalue       PARAM         Yalue       PARAM         Yalue       PARAM         Yalue       PARAM         Yalue       PARAM         Yalue<		SCHEMATIC1 : PA	\GE1
signator raphic PARA W Norr Edit D Detete Property Display Detete Property Display Properties Name: C1_Val Value: In Display Format Color Display Format Defoult Display Format Defoult Defoult Rotation Pins Visible Display Format Defoult Rotation Defoult Rotation Defoult Rotation Defoult Rotation Defoult Para M Name Only Both if Value Exit is Value of Value Exit is Value of Value Exit is Corr Defoult	C1_Val		
raphic Delete Property mentation mentation Type PSpic # Model X-Coordinate 00 Name INV 500 Name INV 500 INV 500	Color	Default	Pivot
D       Delete Property         mentation       Display         Arial 7       Display         Name: C1_Val       Arial 7         Walue:       In         V-Coordinate       00         Name:       Name:         Name:       Name: <td>esignator</td> <td></td> <td></td>	esignator		
D       Delete Property         mentation       Display         ntation Type       PSpk e Model         X-Coordinate       60         Y-Coordinate       60         Name       11         Seference       1         Piso Visable       1         Piso Visable       1         Immediation       01         Model Type       02         Value       Name         Name       10         Piso Visable       1         Immove DEF, ULT       Name         Model Type       02         Value       01         Name       101         Model Type       02         Value       01         Model Type       02         Value       PAR MM         Value       PAR MM         OK       Cancel         Help       0K         OK       Cancel         Help       0K         OK       Cancel         Help       0K         OK       Cancel         Help       0K	Graphic	PAR#M.Norm	Edit
mentation Path       Display         pentation Path       Display         nation Path       Display         Name       60         Y-Coordinate       60         Y-Coordinate       00         Name       1N 300         Reference       1         Footprint       Display Format         Pins Visible       1         Nodel Type       0 (1)         Model Type       0 (1)         Model Type       0 (1)         Model Type       0 (1)         Size Part       PARAM         PAR AM       Vormal         Value       PAR AM         Value       PAR	D		Delete Property
Initation Type PSpk # Model X-Coordinate 60 Y-Coordinate 00 Name 1/N 300 Reference 1 Footprint Format Color Pins Visible 1 Model Type /// OC 11 Model Type /// OC 11 Both if Value Exit is 0 Yalue of Value Exit is 0 Yalue for Value for Valu	ementation		
Initiation Type       PSpk e Modei         X-Coordinate       60         Y-Coordinate       00         Name       1N 300         Reference       1         Footprint       1         Pins Visible       1         Initive       DEF ULT         Model Type       0 (11)         Model Type       0 (11)         Reference       0 (11)         Reference       0 (11)         Both if Value Exits       0 (11)         Both if Value Exits       0 (12)         Ince Data       PAR AM         Ince Part       PARAM         Value       PAR AM            Value       PAR AM            Value       PAR AM            Value       PAR AM            Value       PAR AM            Value       PAR AM	entation Path		Display
Y-Coordinate       00         Name       IN 300         Reference       I         Footprint       Image: Stable         Pins Visible       I         Imitive       DEF, VULT         Model Type       0 (1//)         SideOnly       Both if Value Exit ts         Gething       Imitive         Package       PAA AM         Irree Part       PAR AM         Value       PAR AM         Value       PAR AM         Value       PAR AM         Irree Part       PAR AM         Value       PAR AM	entation Type		
Value       Value <td< td=""><td>n X-Coordinate</td><td></td><td></td></td<>	n X-Coordinate		
Reference Footprint Pins Visible Pins Visible Pins Visible Pins Visible Pins Visible DEF, VULT Model Type O(11 SoleCONIV ference ce Library CICADENCE PSPICET re Package PAR AM Vormal Value PAR AM Vormal Value CICADENCE PSPICET Rotation @ 0° @ 0° @ 270° Text Justification Default OK Cancel Help 11. Select "Name and Value" and	1 Y-Coordinate		
reference Footprint initive DEF VULT Model Type OC 11 Model Type C ICADENCE PSPICET tere Package PAA AM value PAR MM OK Cancel Help (Name and Value Exit is Value Exit is Value Exit is OK Cancel Help 11. Select "Name and Value" and	Name	IN 300	
Pins Visible       I         rimitive       DEF ULT         Model Type       0 (11)         piceOnly       TF UE         ference       Each if Value Exits         cc Library       C:CADENCE PSPICET         pe Package       PAR AM         value       PAR AM         value       PAR AM         of the Click and select "Display" to    11. Select "Name and Value" and		l	
rimitive DEF NULT Model Type 0(11 biceOnly TF JE ference ce Library C:CADENCE PSPICET PARAM Vormal Value PAR MM roce Part PARAM Vormal Value PAR MM OK Cancel Help 11. Select "Name and Value" and			
Infinitive Der volt Model Type 00 11 piceOnly ference ce Library CICADENCE PSPICET PAR AM Value PAR AM Value PAR AM OK Cancel Help OK Cancel Help 11. Select "Name and Value" and		DEE	
wided if type       vit if type         piceOnly       The use         ference       Text Justification         ce Library       CACADENCE PSPICET         urce Part       PAR AM         Value       PAR AM         Value       PAR AM         or       Cancel         Help         or       Cancel         Help         or       Cancel         Help         or       Cancel         Help         or       Cancel         Help       In the second se			
reference ce Library CVCADENCE PSPICET PARAM Normal Value PARAM Normal Value PARAM Value Concel Help 11. Select "Name and Value" and			
ce Library C: CADENCE PSPICET e Package PAR AM ince Part PARAM Normal Value PAR M Cite Cancel Help Cight click and select "Display" to 11. Select "Name and Value" and			
e Package PAF AM Ince Part PARAM Normal Value PAR M OK Cancel Help Oght click and select "Display" to 11. Select "Name and Value" and			7
And the part of th			
value РАРАМ OK Cancel Help aght click and select "Display" to 11. Select "Name and Value" and			
ight click and select "Display" to 11. Select "Name and Value" and	Value		
	iaht cli	ck and sele	ct "Displav" to
play the value on the schematic. "ok"	-		

### Back to Table of Contents



#### 🔱 Texas Instruments

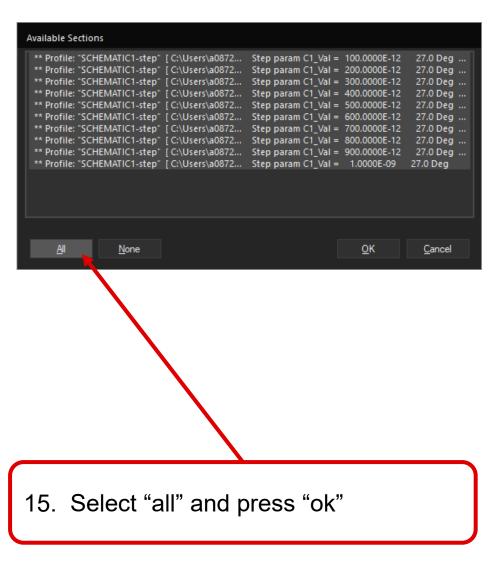
# Parameter Stepping: Set up analysis profile

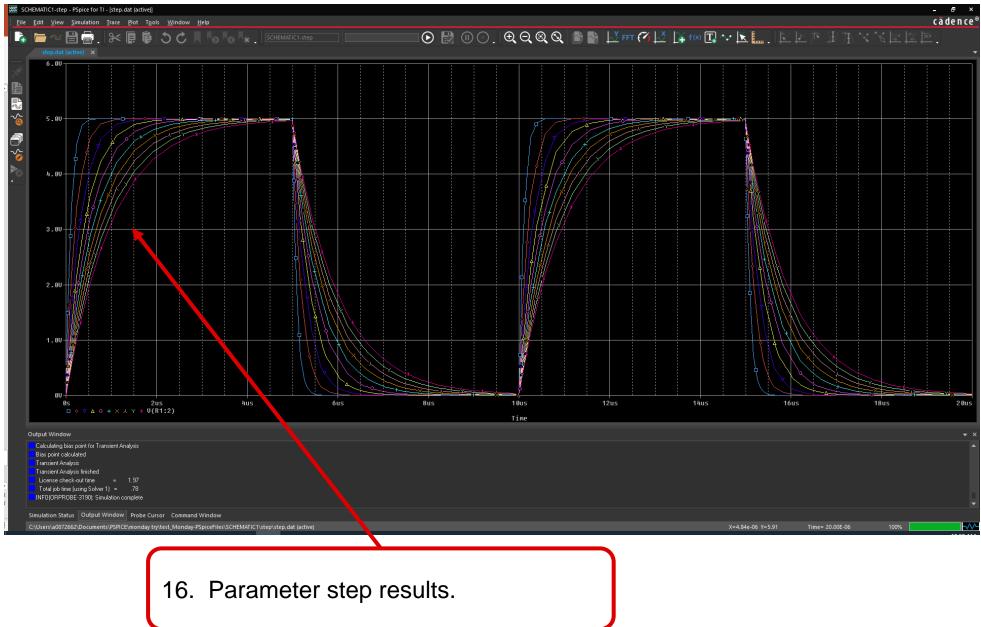






### **Parameter Stepping: Example Results**



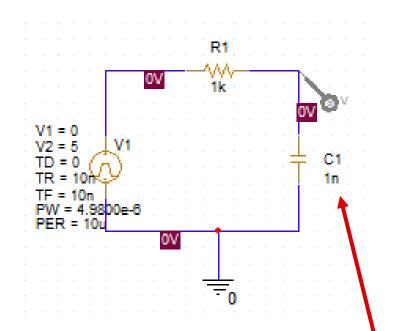


### **Back to Table of Contents**



39

E1)]														
B SI A <u>n</u> alysis	P <u>S</u> pi	ice	<u>A</u> ccessories	<u>R</u> eports	<u>O</u> ptions	<u>W</u> ine	dow	<u>H</u> elp						
)C 拱	<b>∻</b> ⊰⊚		v Simulation P			. (	Ð,	$\Theta_{\mathbf{v}}$	ର୍ (	Q	Ē <u>0</u>		ריי 11	2
age /	lacksquare						_	5	_	_	-	_	_	
								· · · · · ·		· · · · · · · · · · · · · · · · · · ·			· · · ·	
										· · ·				
		A <u>d</u> v	anced Analysi		•									
			ate Netlist											
		Viev	w Netlist			Q٠,	<u>O</u> pti							
								nte Carl						
						∦ ,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,,	Smo <u>P</u> ara	<u>k</u> e metric I	Plet					



1. Note: the full version of Cadence allows assigning tolerances to all components using "assign tolerance" option. If you did a complex design and wanted to compare 0.1% tolerance to 1% tolerance results this would be a useful option. This is not available on PSPICE for TI.

2. Choose tolerance for each component by double clicking on the "tolerance" parameter. In this example the resistor is set to 1% and capacitor is set to 5%.

PSpice for TI - [Property Editor\*]

1 📄 🖶 📻 .

Analog or Mixed A/D

🗖 😹 .\test\_monday.dsn

R .\test\_monday-pspic

💳 File 🍡 Hierarchy

E Layout

🚊 🔚 Outputs

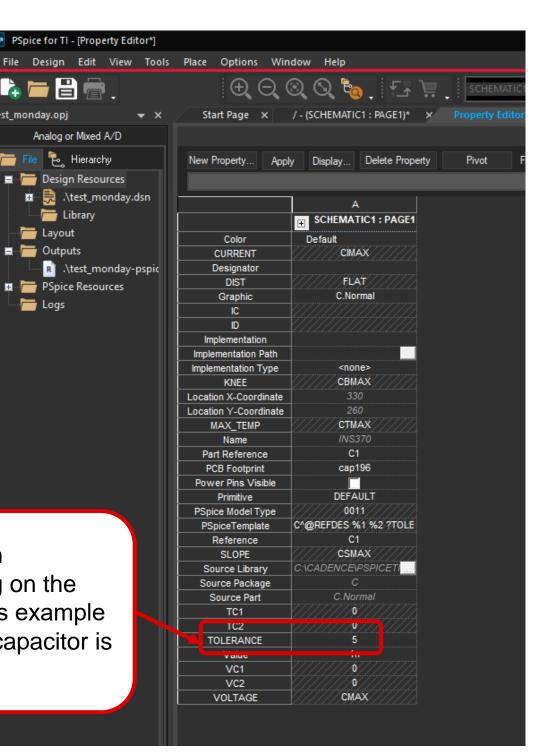
🔚 Logs

E- Cesign Resources

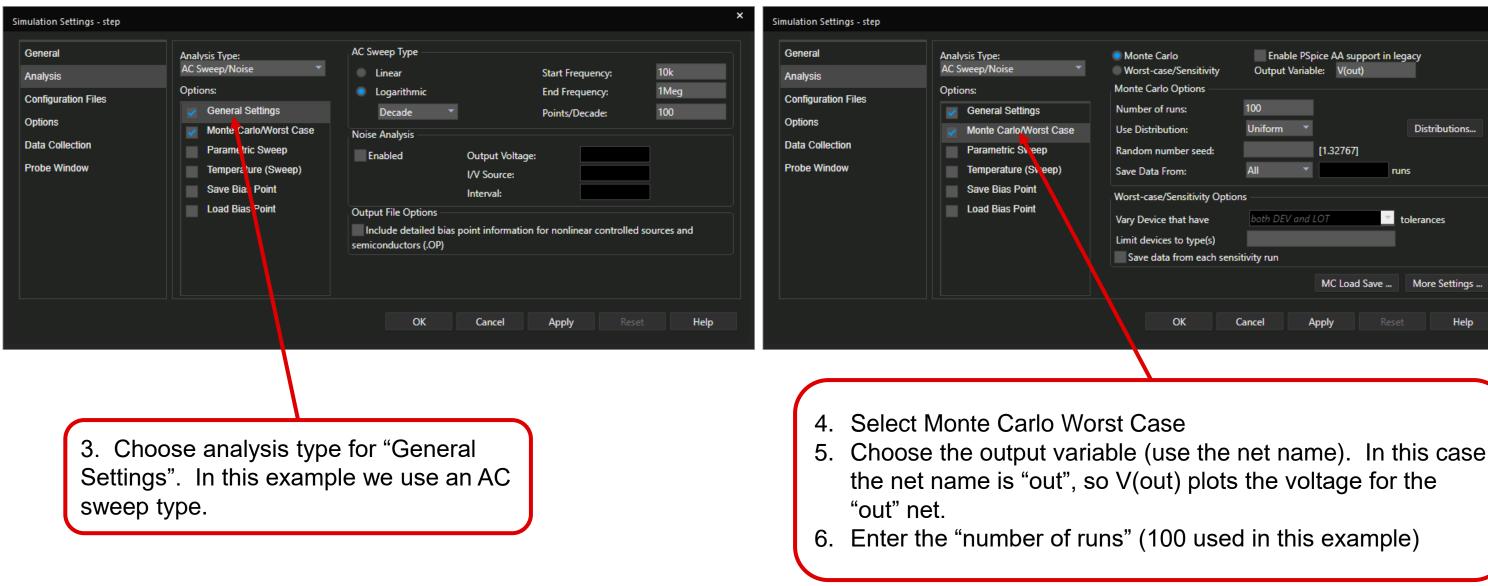
🚊 🔚 PSpice Resources

L à

test\_monday.opj





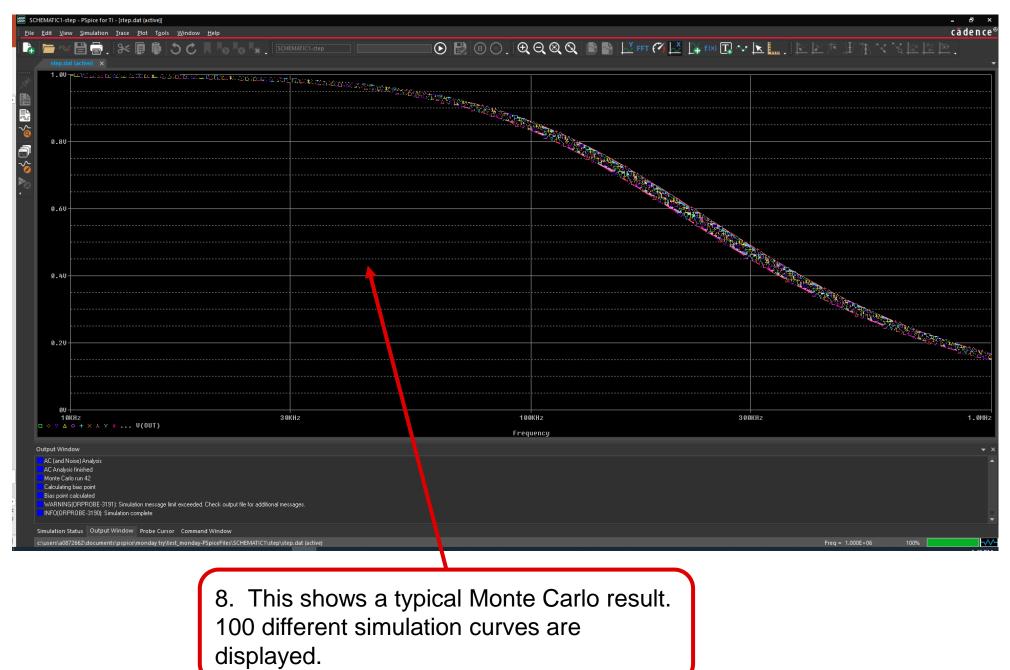


#### **Back to Table of Contents**

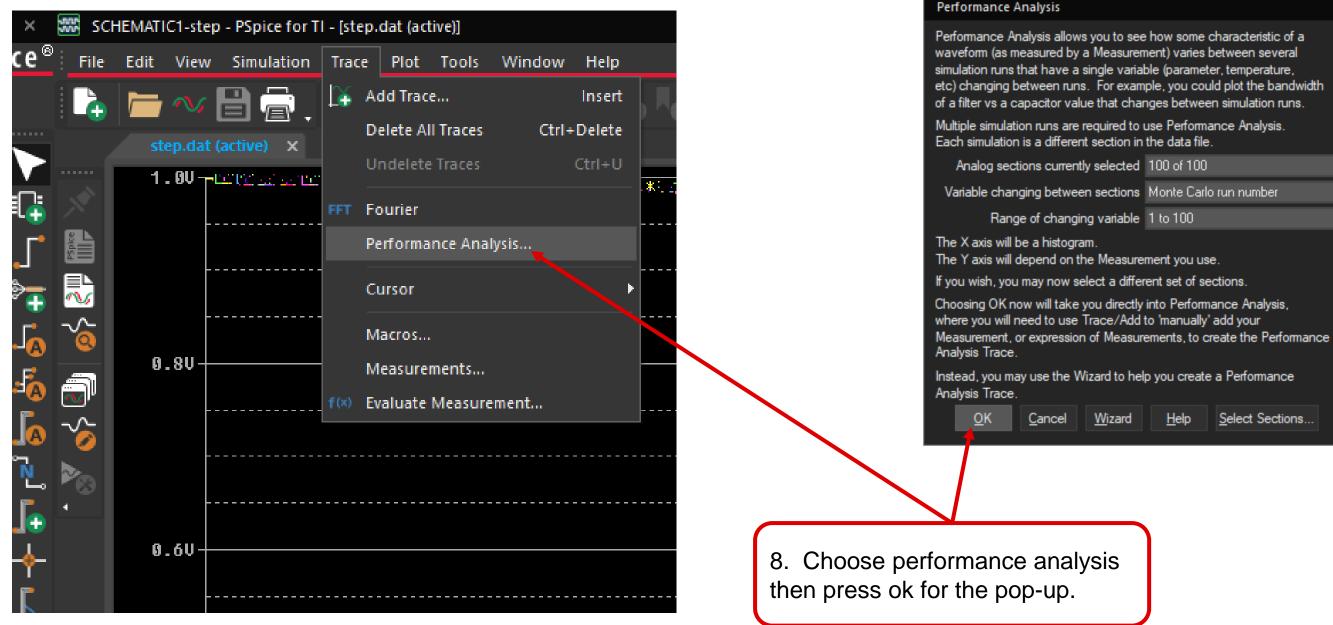
									×
itivit	y			ice AA su Ie: V(o		in lega	асу		
ns —	- 2	00 Jniform	•				Distr	ibutions	
eed:	F	AII	•	[1.32767	7]	runs	5		
vity (	Options								
ave pe(s)	l	both DEV	′ and L	.OT		▼ to	leranc	es	
each	sensitiv	ity run							
				MC Loa	ad Save	:	More	Settings	
	Can	cel	A	pply		Reset		Help	

#### **TEXAS INSTRUMENTS**

Available Sections					
** Profile: "SCHEMATIC1-step" ** Profile: "SCHEMATIC1-step"	[ c:\users\a0872 [ c:\users\a0872	Monte Carlo Pass Monte Carlo Pass	88 89 90 91 92 93 94 95 96 97 98 99 99	27.0 Deg 27.0 Deg	
7. Run the si simulation is o of available re	complet	e you w	/ill ge	t a list	







#### **Back to Table of Contents**

Select Sections..



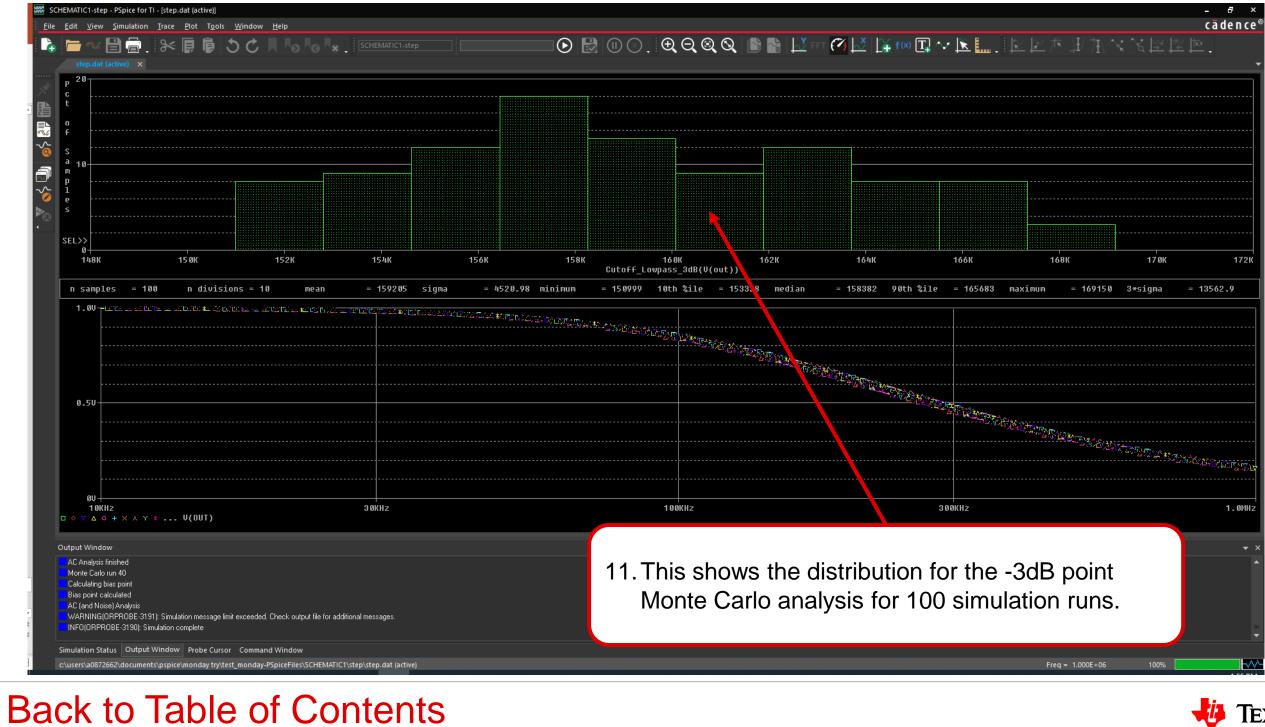
🐺 SC	HEMATIC1-st	ep - PSpice for T	I - [step.d	at (active	)]				
File	Edit Vie	w Simulation	Trace	Plot To	ools Wind	low He	lp		
		82.	≫	i i	) S (	と同		🗴 🚬 Існ	EMATIC1-step
	step.da	at (active) ×						 	
	P								
	c t				Add Trace				
	o				Add Plot				
₫	f								
$\sim$	s				Delete Plo	ot			
a	a m				Delete Y A		Ctrl+SI		
iiii	р 1				Log X				
	e s				Log Y				
<b>*</b> ©									
	SEL>>				Zoom In				
					Zoom Ou	t			
					Zoom Are	а			
					Zoom Fit				
	1.0V	- 11716- ac ac 101	القد عد عقا	ecrec	Cursor Or	n			
					Mark Data				
					Add Text I	abel			
					Show All	Traces			
				• • • • •	Hide All T	races			
	0.5V								

#### Add Traces Simulation Output Variables Functions or Macros Measurements Frequency Bandwidth(1,db\_level Analog V(0) Bandwidth\_Bandpass V(C1:1) Bandwidth Bandpas Digital V(C1:2) CenterFrequency(1,dl CenterFrequency XRa Voltages V(INUU158 V(out) ConversionGain(1,2) Currents ConversionGain XRa V(R1:2) Cutoff\_Highpass\_3d Power V(V1:+) V(V1:-) Cutoff\_Lowpass\_3dB Noise (V<sup>2</sup>/Hz) V1(C1) Cutoff Lowpass 3dE V1(R1) DutyCycle(1) Alias Names V1(V1) DutyCycle\_XRange(1, V2(C1) Falltime NoOvershop Subcircuit Nodes Falltime\_StepRespon V2(R1) V2(V1) Falltime\_StepRespor GainMargin(1,2) MATLABFunction1(1, MATLABFunction2(1.2 Max(1) 16 variables listed Max\_XRange(1, begin Min(1) Full List Trace Expression: Cutoff\_Lowpass\_3dB(V(out)) <u>о</u>к

- 9. Press "add trace" on the new plot.
- Select the function you want to apply then the output variable. In this case the function is "Cutoff\_lowpass\_3dB" and the variable is V(out).

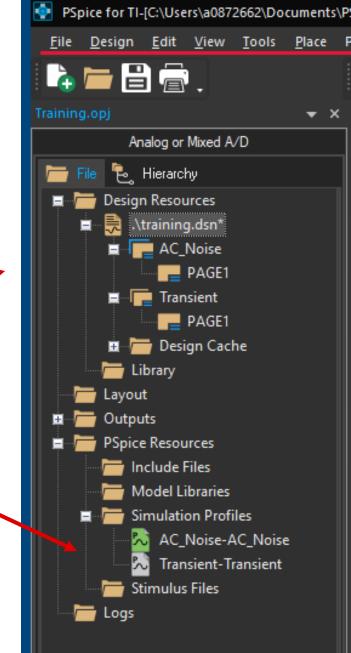
_3dB(1) _3dB_XRan _Ievel) nge(1,db_I	
ge(1,2,beg 1) /Pango(1	
) (Range(1	
egin_x,en :(1) e(1) e_XRange(	
MATLAB_f < MATLAB	
x,end_x)	Ŧ
<u>C</u> ancel <u>H</u> e	lp





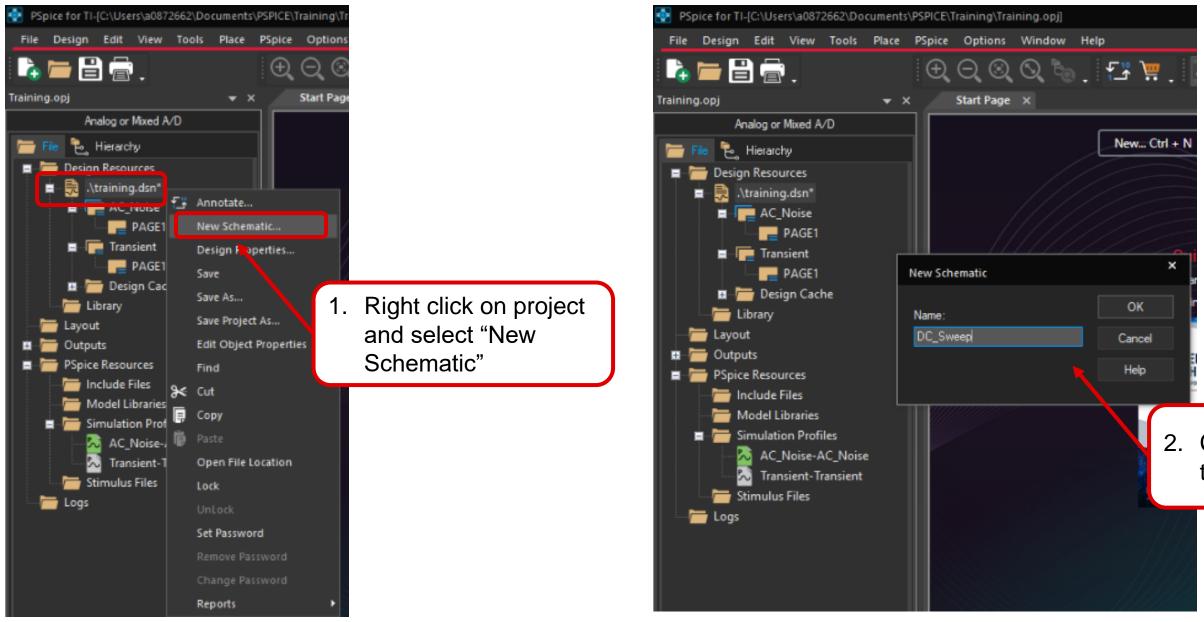
#### TEXAS INSTRUMENTS

This shows a project with two schematics added and two simulation profiles. This approach can be useful if you want to do variations on different types of simulations. For example, a transient simulation requires a square wave source, and an AC simulation requires an AC source. You can build two slightly different schematics and develop simulation profiles for each schematic.







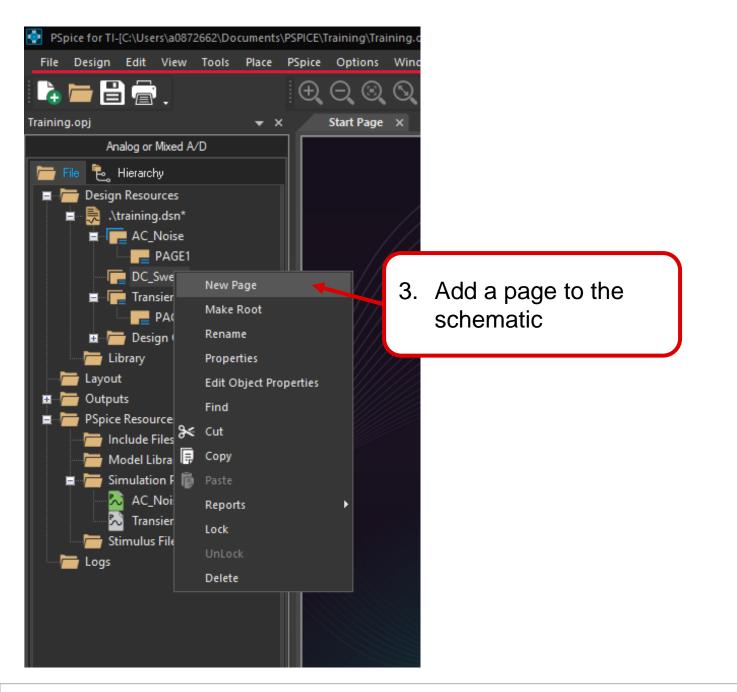


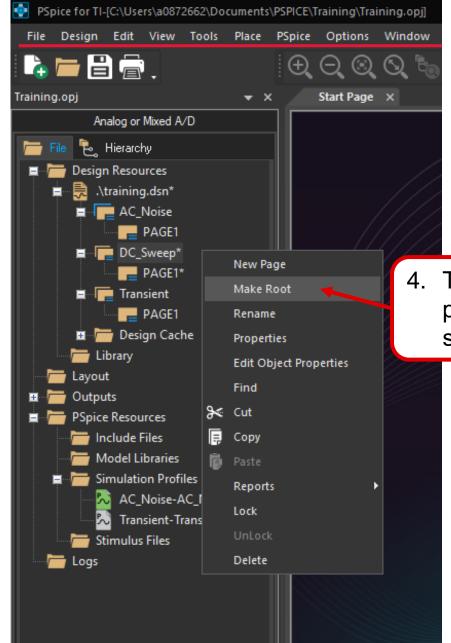
#### **Back to Table of Contents**



#### Choose a name for the schematic





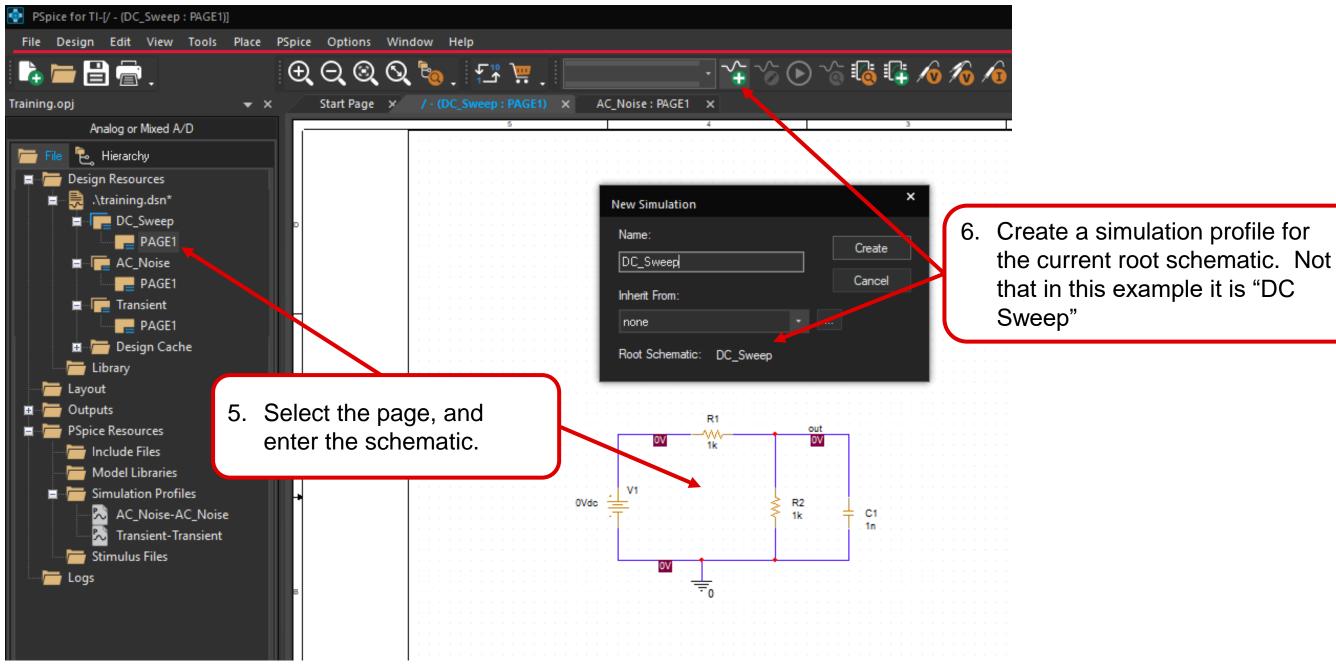


#### **Back to Table of Contents**



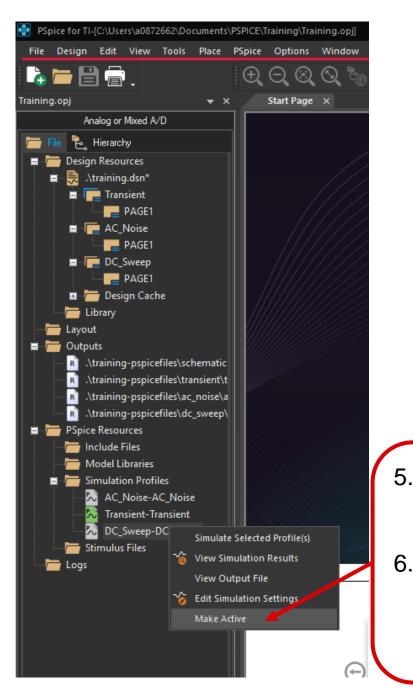
#### 4. To create a simulation profile for this schematic select "Make Root"

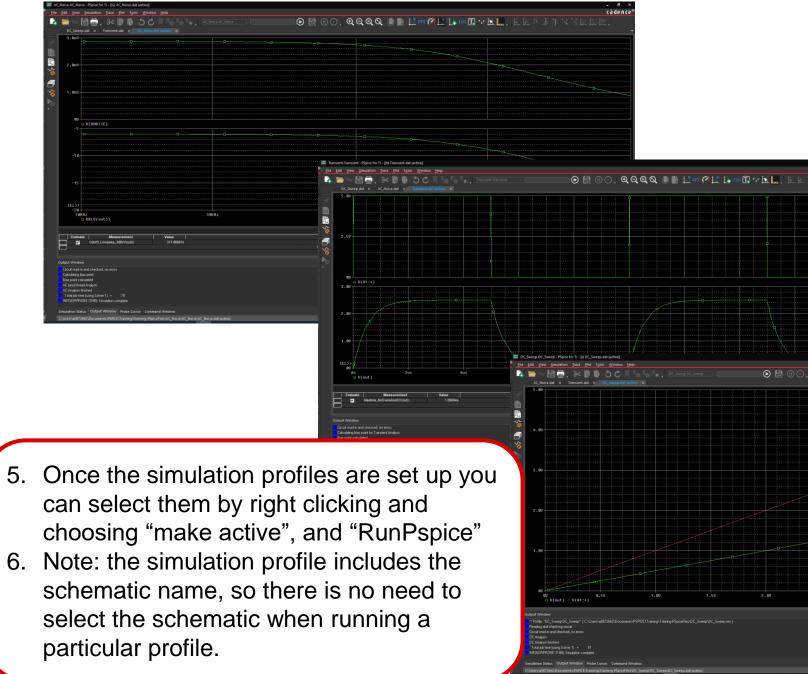










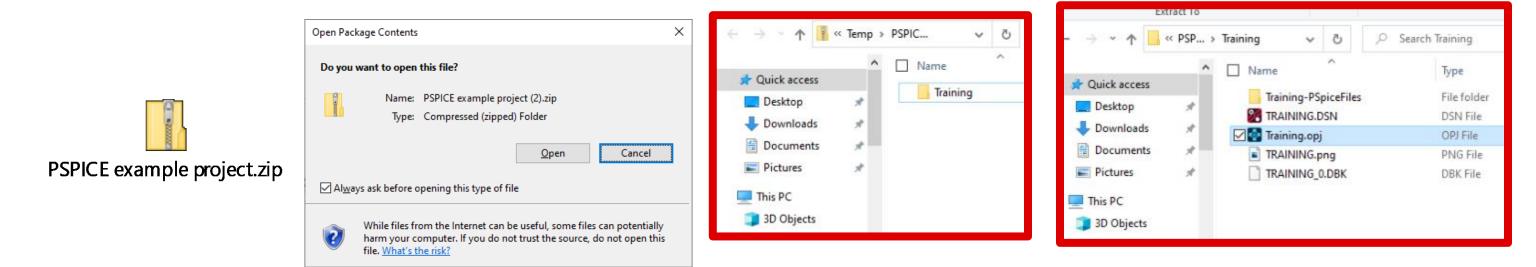




		-	8 ×
			dence
[			

			_				cadence
୦.ାଇ୍ର୍ଡ		7 m. 📢 🔽 🕇	🍝 f 👀 [] /	· k 🖬 . 🕸			
	: : : 1		: .	: : :			
	3.0	Ų	3.5⊎		່ອບ	4.50	5.00
U_U1							

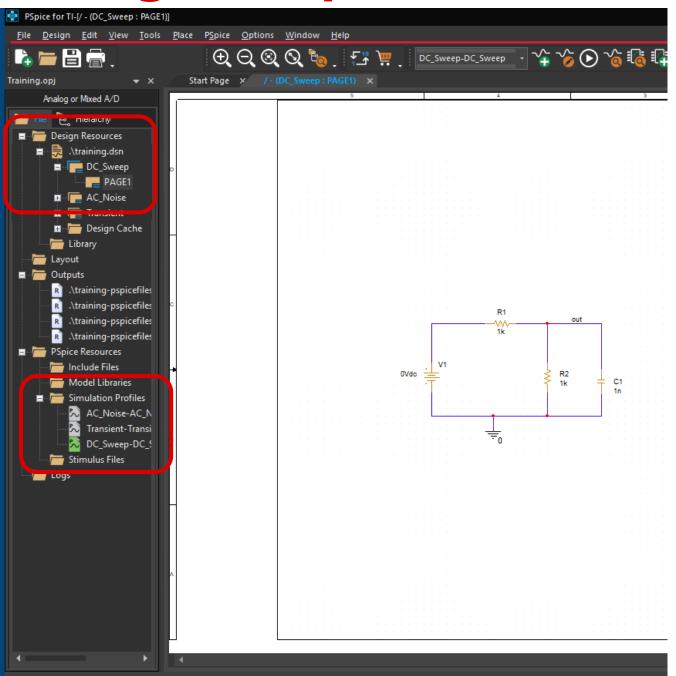




- Click on the imbedded file above.
- Open the file.
- 3. Drag the training folder to the location where you keep you PSPICE projects.
- 4. Click on "training.opj" to start the project







- 1. This is what the example project should look like once opened.
- 2. Select the different schematics under "training.dsn"
- 3. Choose the different simulation profiles and run them under "simulation profiles"







# **Getting professional PSPICE**

### Confluence instructions for installing full PSPICE from JC Zhu

#### Why you may want the full version

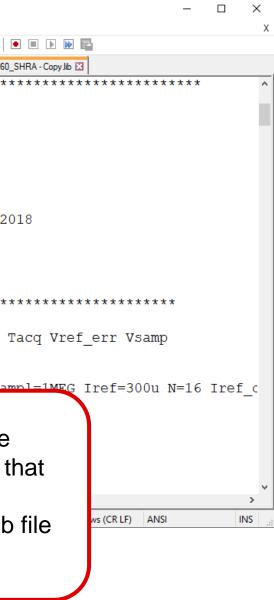
- You need to run Non-TI models. If this is the case, the PSPICE for TI is very limited (three probes)
- If you are developing your own models, or if you are working with a developer (e.g. design soft).
- If you want access to the PCB development ٠ tools. Since we use Altium this may not be too useful but I suppose if you had a customer design this may be helpful.

### Why you may not want the full version

- It is very large and takes a long time to install
- PSPICE for TI works well if you aren't using non-ti ٠ models. You can do all analysis types (e.g. montecarlo). The only limit happens when you us non-TI models. In this case you can only monitor three signals.

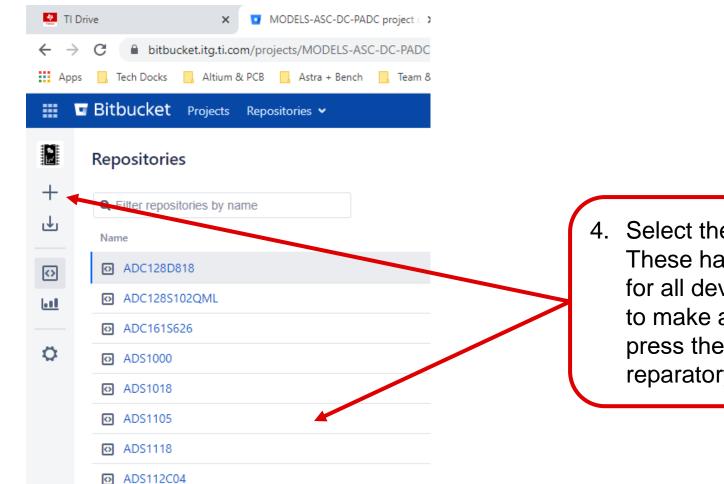


1 2 - ADS8860	-SHRA-PSPI Extract	– 🗆 ×	C:\Users\a0872662\Desktop\ADS8860_SHRA - Copy.lib - Notepad++
F Home Share		^ <b>e</b>	<u>File Edit Search View Encoding Language Settings Tools Macro Run Plugins Window ?</u>
Documents E Pictures		2	[]]]]]]]]]]]]]]]]]]]]]]]]]]]]]]]]]]]]
United States Docume			🔚 RegSpecTool.exe.config 🛛 🔚 ADS8860.lib 🛛 🔚 ADS8860.lib 🛛 🔚 ADS8860_SHRA.lib 🖾 🚍 ADS8860.lib 🖾
Pictures Videos	Extract		16 ************************************
	all		
Extract	То		18 * Released by: Texas Instruments Inc. 19 * Part: ADS8860
	DS8860-SHRA ∨ Ō	Search ADS8860-SHRA-PSPIC	20 * Date: 07/16/2021
			21 * Model Type: TRANSIENT
	▲ Name	Туре	22 * Simulator: PSPICE for TI
📌 Quick access			23 * Simulator Version: 17.4-2020
Desktop 🤉	ADS8860-SHRA.lib	LIB File	24 * Datasheet: SLUSD31A -OCTOBER 2018-REVISED DECEMBER 2
-	ADS8860-SHRA.OLB	OLB File	25 *
•			26 * Model Version: 1.10
Documents 🤉	*		27 * 28 ** Modle created Art Kay & DesignSoft, Inc.
E Pictures	e		29 ************************************
			30
💻 This PC			31 .SUBCKT ADS8860-SHRA INP INM TCONV ResetSH SampleRate
🧊 3D Objects			32 + Vsettling_err Vref_ideal Vref
Desktop			33 + PARAMS:
Documents			34 + Csh=55p Cdio=4p Cros=4p Psh=96 tConv=710p Wrof=5 fs
	1 "lib" filo librory fi	la containa not	35 .PARAM Rt= $\{1$
🕂 Downloads	1. ".lib" file – library fi	le contains net	36 .PARAM Rres= 37 .PARAM Krcr= 3. Note that the .lib file has the
👌 Music	list for your device.		38 PARAM Crcr=
Pictures	2. ".olb" file – contain		39 .PARAM Rd1= SUBCKT netlist. Also note
Videos		Strie	40 .PARAM Rd2= the name of the SUBCKT
	schematic symbol	and	
2 items	connections to ".lib		Normal text file matches the name of the .li
			and the .olb file.





### bitbucket repository for ADC model files **Confluance Bit Bucket Tutorial**

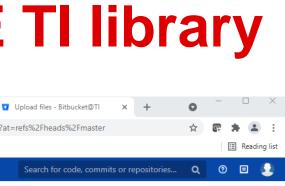


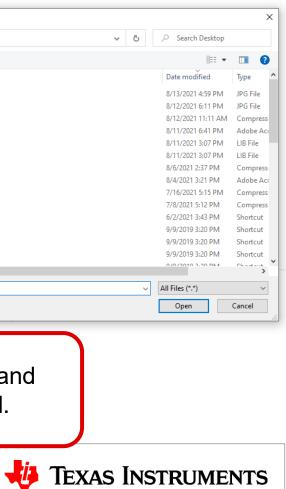
4. Select the appropriate repository. These have been pre-populated for all device names. If you need to make a custom model name press the "+" sign to add a new reparatory.





🙎 TI D	rive 🗙 🔽 Browse MOI	DELS-ASC-DC-PADC 🗙 🛛 🔽 MODELS	S-ASC-DC-PADC project C 🗙	Browse MODELS-ASC-DC-PADC × +		🙎 TI Drive	×   •	Browse MODELS-ASC-DC-PA	DC / 🗙 📔 MODELS-ASC-DC-PADC project 🔿 🗙
$\leftrightarrow \rightarrow$	C ibitbucket.itg.ti.com/projects/MOI	I			↓ R <b>*</b> *	$\leftrightarrow$ $\rightarrow$ (	C 🔒 bitbucket.itg.ti.com/	plugins/servlet/uploadFile/p	projects/MODELS-ASC-DC-PADC/repos/ads8860?
	Tech Docks Altium & PCB Astra	· · ·			Reading li	Apps	📙 Tech Docks 📙 Altium & PC	CB 🔜 Astra + Bench 🔜 T	eam & Support 🧧 TIPL 🧧 Models
	Bitbucket Projects Repositories 🗸		-	Search for code, commits or repositories	Q (? E 👤		Bitbucket Projects Re	epositories 🗸	
ų.	MODELS-ASC-DC-PADC / ADS8860 Source			+ New File @ Uplo	Watch	310° E	MODELS-ASC-DC-PADC / AD	958860	
រូះ ព្រ ល	Image: symmetry of the	0 releases	4 contributors	+ New File @ Uplo	ad Browse Filter	ິ່າ⊅ ©	Drop fi	les ho <del>re to add to t</del> Or choose your fil	
$\langle \rangle$	Source	Description			Last Modified	$\langle \rangle$		Open	
\$	ads8860_PSPICE_6_22_2022.zip	MDL-7024			23 Jun 2021	•	Uploaded files  Commit directly to the	← → × ↑ > This P Organize ▼ New folder	°C → Desktop →
รร	ads8860_PSPICE_8-10-2021.zip	MDL-7345			5 days ago	<u></u>	C 🏦 Branch and start a pull	^ [	Name
<u></u>	ads8860_PSPICE_8-6-2021.zip	Uploaded files			06 Aug 2021	ĥ	Commit	📌 Quick access	can not place PSPICE model.jpg
ň	C Readme.md	MDL-7345			5 days ago	ໃນ		➡ Downloads 🖈 🛱 Documents 🖈	<ul> <li>hot fix.jpg</li> <li>ADS8860-SHRA-PSPICE-8-11-2021.zip</li> <li>BOOSTXL-ADS1219.pdf</li> </ul>
îs •	Readme.md REPOSITORY EXAMPLES: https://bitbut Source Development-Pspice LM4030_NA2P5_PSPICE_TRANS.zip For individual GPN repository, the publish For 5. Press upl		A /	Gimulator" folder within the GPN folder	✔ Edit README	0		<ul> <li>Pictures</li> <li>This PC</li> <li>3D Objects</li> <li>Desktop</li> <li>Downloads</li> <li>Music</li> <li>Pictures</li> <li>Videos</li> <li>Windows (C;)</li> <li>File name</li> </ul>	ADS8860_SHRA - Copy.lib ADS8860_SHRA.lib I latest_pspice_adc_model-8-6-2021.zip 53622_ADS8555EVM.pdf opa625vsOPA333.zip try lib again.zip Camtasia - Shortcut aam det designs TPL
	file reposi	tory.							oose your files" a ZIP file to upload





New TI	Drive X Browse MODELS-ASC-DC-PADC X MODELS-ASC-DC-PADC project X Browse MODELS-ASC-DC	-PADC × + O	- 🗆 X	🛃 TI Dr	ive x Browse MODELS-ASC-DC-PADC x MODELS-ASC-DC-PADC project x
$\leftrightarrow$	C 🔒 bitbucket.itg.ti.com/projects/MODELS-ASC-DC-PADC/repos/ads8860/browse	\$	🖻 🖈 😩 E	$\leftarrow \rightarrow$	C bitbucket.itg.ti.com/projects/MODELS-ASC-DC-PADC/repos/ads8860/browse/ads8860_PSPIC
Арр	is 🧧 Tech Docks 🧧 Altium & PCB 📙 Astra + Bench 📙 Team & Support 📃 TIPL 📃 Models		📰 Reading list	Apps	Tech Ducks 📑 Aldum & PCD 📑 Astra + Dench 📑 Team & Support 📑 TiPE 📑 Models
	■ Bitbucket Projects Repositories ► Search for code, com	mits or repositories Q	0 🗉 😫		Bitbucket Projects Repositories 🗸
ц,	MODELS-ASC-DC-PADC / ADS8860 Source		() Watch	ţ,	MODELS-ASC-DC-PADC / ADS8860 Source
វ្រ	\$9 master ~ ··· ADS8860 / +	- New File 🥔 Upload	Browse Filter	ť	Image: Symmaster Y         Image: Mage:
ຳ ©	Image: 19 commits     Image: 19 commits     Image: 10 commit of the second seco			î۴ ص	Source view Diff to previous History • Edit •
Ŵ	Source Description		Last Modified		
<>	Legacy-Publish			$\langle \rangle$	
¢	ads8860_PSPICE_6_22_2022.zip MDL-7024		23 Jun 2021	¢	
ų	ads8860_PSPICE_8-10-2021.zip MDL-7345		5 days ago	29	
611	ads8860_PSPICE_8-6-2021.zip Uploaded files		06 Aug 2021	<u>L.01</u>	
ň	Readme.md MDL-7345		5 days ago	ĥ	Download to vew this f
î۶	C Readme.md	1	• Edit README	î۵	This file can't be viewed in your l
٥	REPOSITORY EXAMPLES: https://bitbucketitg.ti.com/projects/MODELS			0	Downloa 1 file
	Image: State in the state	within the GPN folder.			
>>	<ul> <li>Development-NameOfSimulator" folder is to put development files/test benches, documentation, and other thinas.</li> <li>7. Once you have uploaded your files. Click on the appropriate reparatory</li> </ul>			- »	8. Copy the link for your (intake form).



### **Confluence Model Intake Form makes Jira ticket**

### Example of completed Jira Ticket

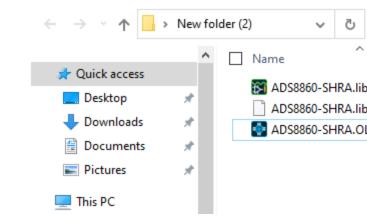
#### Model Request Intake Form

Created by JC Zhu, last modified on Jul 25, 2021

#### Choose from below options:

Select Engineering Need *	<ul> <li>Register Product Model Development         <ul> <li>New model</li> <li>Publish existing model</li> <li>Revise existing model (bug)</li> <li>Translate Model to another simulator (LTSpice is not supported)</li> </ul> </li> <li>Get General Help         <ul> <li>Request access to unpublished model (e.g. unencrypted, restricted)</li> <li>Generate training plan</li> <li>See tickets assigned to me (JIRA query shortcut)</li> <li>List all open tickets by SBE2 (JIRA query shortcut)</li> </ul> </li> </ul>
Modeling Needs *	<ul> <li>Develop a new model</li> <li>Publish / release a model that has been completed</li> <li>Revision to existing model due to bug OR change in model requirement / product specification</li> <li>Translation (simulation platforms with limited support)</li> <li>Translation (supported simulation platforms)</li> </ul>
Reason for Publishing *	<ul> <li>This is a new model (never published to ti.com)</li> <li>This is a revision to an already published model due to change in encryption status, CIP level, or access classification</li> </ul>
Select the Product Line *	SBE ASC SBE1 DC SBE2 DC-PADC Visit SC Marketing Automated Services to look up the product line (listed as DesignWIN Hierarchy) owning the underlying TI product by searching the GPN. Product line modeling appointees are listed on the modeling appointee page.

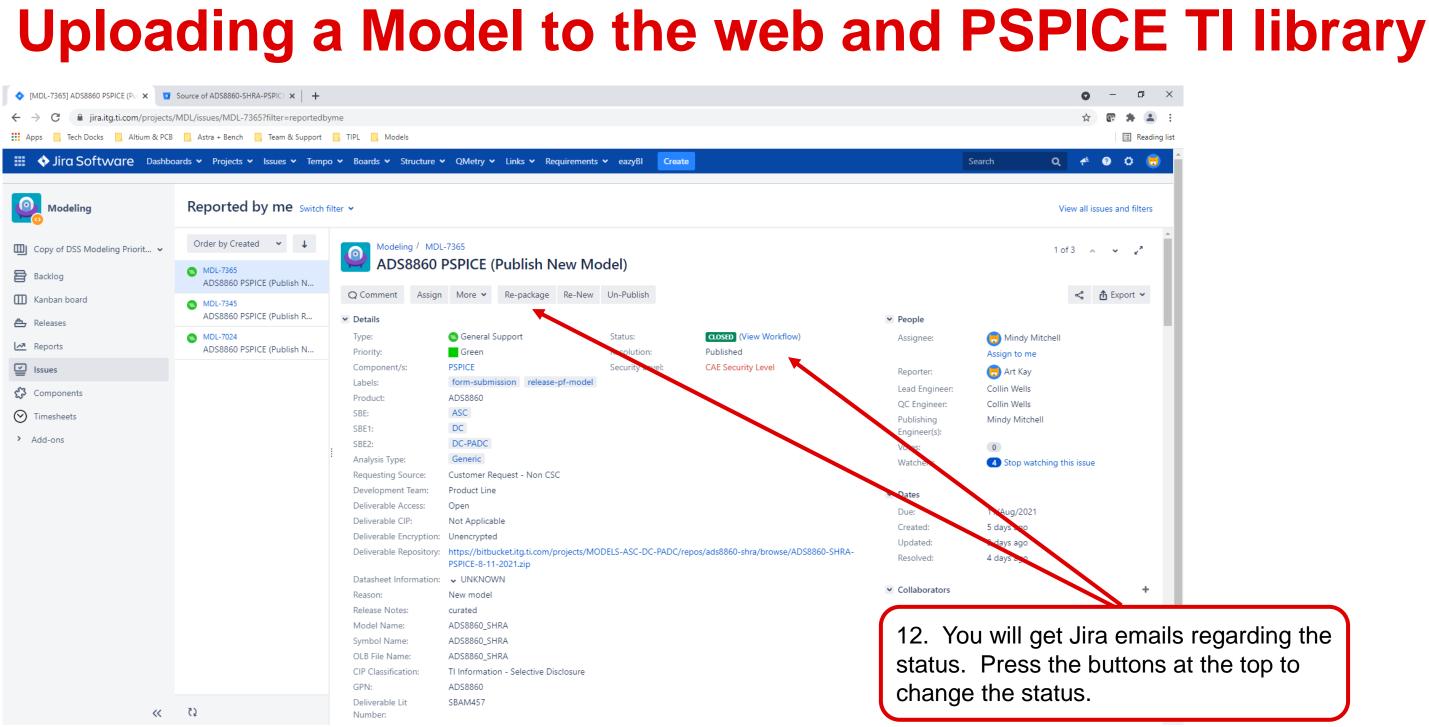
- 9. Click on the link to start your Jira ticket for model creation. 10. The form questions are mostly self explanatory. At one point in the form you will need to past in the bit bucket repository link from the previous slide.
- 11. A few hours after the files are uploaded, a signed copy of your files will automatically be uploaded back to the bit bucket repository. You can copy these to your computer and this will enable you to use PSPICE for TI without limitations. If the .lib is not signed, PSPICE for TI considers the model to be non-TI and limits you to three probes. See signed library below:





🔎 Search N	ew folder (2)
	Date modified
b	8/11/2021 3:07 PM
bsig	8/11/2021 8:54 PM
LB	8/11/2021 3:33 PM

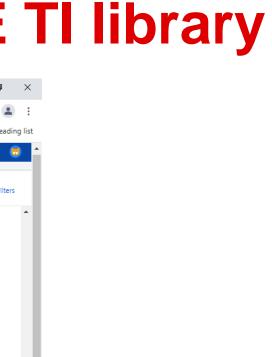






• [MDL-7365] ADS8860 PSPICE (Pu	× Source of ADS8860-SHRA-I	SPICE ×   +	<b>o</b> – Ø
← → C 🔒 jira.itg.ti.com	/projects/MDL/issues/MDL-7365?	ilter=reportedbyme	Q 🛧 💽 🚖
Apps 📙 Tech Docks 📙 Alti	ium & PCB 🔜 Astra + Bench 🔜	Team & Support 🧧, TIPL 🗧 Models	III Rea
📰 🚸 Jira Software Dashba	oards 🗙 Projects 🛩 Issues 👻 Tempo	▼ Boards ▼ Structure ▼ QMetry ♥ Links ♥ Requirements ♥ eazyBl Create	Search Q 📌 🖓 🗘
Modeling	Reported by me switch fi	ter Y	View all issues and filt
<ul> <li>Copy of DSS Modeling Priorit ~</li> <li>Backlog</li> <li>Kanban board</li> <li>Releases</li> </ul>	Order by Created ✓ ↓ MDL-7365 ADS8860 PSPICE (Publish N MDL-7345 ADS8860 PSPICE (Publish R	<ul> <li>Mindy Mitchell added a comment - 3 days ago</li> <li>Hi Art Kay and Collin Wells,</li> <li>Thanks for the explanation. I'll ask about adding this PSpice type to the guidelines for consistent naming. But for now</li> <li>The file has been loaded and should be active on the web within the next 24-48 hours.</li> </ul>	
Reports         Susce         Components         Timesheets         Add-ons	MDL-7024 ADS8860 PSPICE (Publish N	Trx, Mindy         Literature Number       Document Type       View Doc       Concise Description         SBAM457       PSpice Model       View       ADS8860 PSpice Model (SHRA)         • O DCIT Jenkins added a comment - 3 days ago         Publishing successful : https://bitbucket.itg.ti.com/projects/MODELS-ASC-DC-PADC/repos/ads8860-shra/browte/ADS8860-SHRA-PSPICE-8-11-2021.zip         • O DCIT Jenkins added a comment - 3 days ago         Pspice library curation successful: https://bitbucket.itg.ti.com/projects/MODELS-ASC-DC-PADC/repos/ads8860-shra/browse/ADS860-sHRA-PSPICE-8-11-2021.zip         • O Art Kay added a comment - 2 days ago         JC Tu         JC Tu         JC, Lee the new device in the model library, but I cannot place It into a schematic (ADS8860-SHRA). I can place a previously created model (ADS8860)         Image: Comment - 2 days ago         JC Tu         JC, Lee the new device in the model library, but I cannot place It into a schematic (ADS8860-SHRA). I can place a previously created model (ADS8860)         Image: Comment - 2 days ago         JC 2 hu         JC, Lee the new device in the model library, but I cannot place It into a schematic (ADS8860-SHRA). I can place a previously created model (ADS8860)         Image: Comment - 2 days ago         Image: Comment - 2 days ago         JC 2 hu         JC 2 hu         Image: Comment - 2 days ago	13. Use comments to se people involved in m "DCIT Jenkins" is an
~	23		JC Zhu's team is a g Also launch marketin also a good contact.

#### **Back to Table of Contents**



end messages to key odel release. Note that automated response. ood contact for issues. ng (e.g. Mindy Mitchel) is



# Thanks for your time!





🔽 Source of ADS9110-PSPICE-10-6 🗙 🛛 🚸 Infolink	🗙 🛛 💠 TI Web SSO Login	×	🗙 Model Request Intake Form - Mo 🗙	◆ [MDL-7365] ADS8860 PSPICE (P□ × +
← → C 🔒 confluence.itg.ti.com/pages/viewpage.action?spaceKey=MODEL&title=Model+Request+Intake+Form				
Apps 📙 Tech Docks 🔄 Altium & PCB 🔄 Astra + Bench 🤤 Team & Support 🔄 TIPL 🔄 Models				

Register Product Model Development

See tickets assigned to me (JIRA query shortcut) List all open tickets by SBE2 (JIRA query shortcut)

O Translation (supported simulation platforms)

• This is a new model (never published to ti.com)

• Publish / release a model that has been completed

O Translation (simulation platforms with limited support)

 $\mathbf{v}$ 

×

Product line modeling appointees are listed on the modeling appointee page.

!! Value MUST equal Pink Folder Name as found in Galileo!!

If you do not have access to Galileo, enter GPN

~  $\mathbf{v}$ 

•

Visit this page and enter the launch manager (for new model) or the web manager (for revising a model) here.

Translate Model to another simulator (LTSpice is not supported)

Request access to unpublished model (e.g. unencrypted, restricted)

O Revision to existing model due to bug OR change in model requirement / product specification

O This is a revision to an already published model due to change in encryption status, CIP level, or access

Visit SC Marketing Automated Services to look up the product line (listed as DesignWIN Hierarchy) owning the underlying TI

New model

Publish existing model Revise existing model (bug)

O Get General Help

Generate training plan

O Develop a new model

classification

SBE

ASC

SBE1

DC SBE2

DC-PADC

ADS9110

product by searching the GPN.

Mindy Mitchell (a0208213)

#### Choose from below options:

Select Engineering Need \*

Modeling Needs \*

**Reason for Publishing \*** 

Select the Product Line \*

Digital Marketing Contact \*

Product (Recorded in Galileo) \*

and a second second

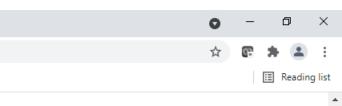
F	
"	
\$	
C	
۴.,	

5 2

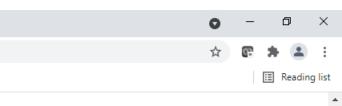
"
?
Ľ
Ŀ

Ö

>>

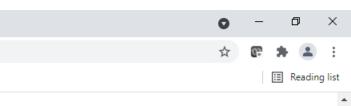


Source of	f ADS9110-PSPICE-10-6 🗙 🛛 🏘 Infolink	🗙 😽 TI Web SSO Login 🛛 🗙 🗙 Model Request Intake Form - Mo 🗙 💠 [MDL-7365] ADS8860 PSPICE (Po 🗙 🕇					
← → c	← → C  in confluence.itg.ti.com/pages/viewpage.action?spaceKey=MODEL&title=Model+Request+Intake+Form						
Apps	Tech Docks 🦷 Altium & PCB 📑 Astra + Be	nch 🧧 Team & Support 🔄 TIPL 🔄 Models					
5 ab		Visit this <b>page</b> and enter the launch manager (for new model) or the web manager (for revising a model) here.					
	Product (Recorded in Galileo) *	ADS9110					
Ē		!! Value MUST equal Pink Folder Name as found in Galileo!!					
>>	Literature Concise Description *	If you do not have access to Galileo, enter GPN					
Ş	Energiane concise bescription	ADS9110 PSPICE Model Enter the description of the model. This will be displayed on ti.com.					
	Product Folder(s) *	ADS9110					
		Enter the name of the product folder where models should be displayed. This may not be identical to the part number. If the					
ŧ:		model should show up in multiple product folders, list the names comma-separated.					
	BitBucket URL *	https://bitbucket.itg.ti.com/projects/MODELS-ASC-DC-PADC					
		Enter the URL to the file to be released (must be checked into the <b>modeling BitBucket</b> ). The repository and file names are case-sensitive so please copy and paste from your web browser.					
		Spaces in the folder hierarchy and file names are not allowed.					
		User "DC Jenkins Build" must have read/write permission to the repository.					
		The only file formats supported are: *.zip, *.tsc and *.sxcmp.					
		For more details, refer to <b>the BitBucket tutorial</b> .					
Ш	Requesting Source *	Active on ti.com - FAE, E2E, Legacy 💙					
	Simulator *	PSPICE: Cadence 🗸					
	OLB File Name *	ADS9110					
		Please enter the name of the PSpice OLB file (without the .olb extension). This is required for adding model into PSpice library for TI parts.					
	Symbol Name *	ADS9110					
		Please enter the name of the PSPICE symbol (must be in the OLB file above). This is required for adding model into PSpice library for TI parts.					
	Model Name *	ADS9110					
		Please enter the name of the PSPICE model (usually the top-level subckt). This is required for adding model into the PSpice library of TI parts.					
	Analysis Type *	O Average					
		Transient					
0		O Generic					
		Please select the analysis supported by the model (average, transient).					
»	OPJ (Project) File						



Ŧ

U	Source of	f ADS9110-PSPICE-10-6 🗙 🛛 💠 Infolink	🗙 🛛 👆 TI Web SSO Login 🛛 🗙	🗙 Model Request Intake Form - Mo 🗙	◆ [MDL-7365] ADS8860 PSPICE (P□ ×   +
←	$\rightarrow$ C	confluence.itg.ti.com/pages/viewpage.ac	tion?spaceKey=MODEL&title=Model+Request+Intake+F	Form	
	Apps 📙	, Tech Docks 📃 Altium & PCB 📃 Astra + Bench	n 🦲, Team & Support 📃, TIPL 📃 Models		
رج الح			for TI parts.		
		Symbol Name *	ADS9110		
F			Please enter the name of the PSPICE symbol (must be in the OLB f library for TI parts.	file above). This is required for adding mod	el into PSpice
"		Model Name *	ADS9110		
?			Please enter the name of the PSPICE model (usually the top-level s library of TI parts.	subckt). This is required for adding model i	nto the PSpice
Ľ		Analysis Type *	O Average		
ła			<ul> <li>Transient</li> <li>Generic</li> </ul>		
			Please select the analysis supported by the model (average, transi	ent).	
		OPJ (Project) File			
			If exists, enter the path to the project file (OPJ) inside the PSpice p "ADC3444_PSPICE_TRANS/ADC3444.opj".	roject hierarchy. E.g.	
		Model CIP *	<ul> <li>NDA Restriction</li> <li>Selective Disclosure</li> </ul>		
			Not Applicable		
		Model Access *	<ul> <li>Open</li> <li>Restricted</li> </ul>		
		Model Encryption *	<ul> <li>Encrypted</li> <li>Unencrypted</li> </ul>		
		Description *	PSPICE Model for ADS9110		
		Request on Behalf of	Arthur Kay		
			If you are requesting a service for someone else, type in his/her na you type.	ame and select a user from the drop down	that appears as
		Product Classification *			
		. Former ensammenten	Core 🗸		
0		Submit			



Ŧ