

Using Spice Models in OrCAD PSpice from vendor models

# **SPICE models and PSpice**

Using SPICE models and PSpice is also covered in, at least, two other resources, Appendix C of the PSpice User's Guide, pspug.pdf in the doc\pspug of the Cadence, OrCAD product installation, and sloa070, an Application Note at <u>www.ti.com</u> that discusses using Texas Instruments SPICE models with PSpice.

### **Fundamentals**

To simulate a circuit from Capture using PSpice, two library elements are required. The graphical representation of the device, from a Capture OLB file that defines the pins for connecting and contains associated properties to invoke the model when the simulator runs, and the model text itself, from a LIB file containing one, or more, simulation models. The SPICE model element in this is the model text; this text needs to be associated with a graphical symbol to define the connections and netlist the model correctly for the simulator to use.

## **Getting Started**

First, locate and download the required model from the vendor site. The exact details vary a bit from vendor site to vendor site but, whatever the exact process, the result will usually be a text file to download, or a page of model text to copy and paste to a text editor. Save the model text and name the file to represent the component that is modelled and change the file extension to .LIB - this will be more convenient for use with the PSpice Model Editor. Some vendors may provide an archive of all, or many, or their models in one large text file, the Model Editor will be able to handle this as well, check that the archive extracts correctly and change the name of the resulting text file to reflect the vendor, or model family, as required, and change the file extension of the extracted text file to .LIB

The model text will usually contain a number of comment lines, they begin with \*, that describe the model and maybe something about the usage and what behaviour is modelled for more complex devices. The actual model implementation text will start at a line that begins either .MODEL or .SUBCKT The components implemented with the .MODEL type contain a list of parameters for the intrinsic models within PSpice, the components implemented with the .SUBCKT type contain a "netlist" of intrinsic devices that implement the component model, in many cases there may also be some .MODEL entries that are specific to the .SUBCKT model.

### **Running the Model Editor**

Start the Model Editor from Windows icon>Cadence PCB Utilities 17.4-2019>Model Editor 17.4. Then choose either Design Entry HDL or Capture as the default design tool. For this example, we will use Capture. Now select File>Open and open the LIB file. For the purposes of having "something" to refer to as a LIB file, this note uses a file that contains four components, a transistor model, a transistor sub-circuit, a power MOSFET sub-circuit and an Op-Amp sub-circuit, this file is called "SampleSpice.lib" See the Sequence in the following screenshots:

## Using Spice Models in OrCAD PSpice from vendor models

🔁 PS	pice Model Edi	tor														- 0	×
File	Edit View	Model	Plot Tools	Window	Help											cāde	nce®
D	New		Ctrl+N	D R	$\oplus$	Θľ	50	4	XIY	7 4		일입					
$\Box$	Open		Ctrl+O				~				~ ~	HUIU	*				
	Close																
B	Save		Ctrl+S														
	Save As																
	Print		Ctrl+P														
Q	Print Preview																
	Page Setup																
	Export to Capt	ture Part Li	ibrary														
	Model Import	Wizard [Ci	apture]														
đ	Encrypt Library	/															
-	1 SampleSpice	lib															
	2 D:\Working\	\wc.lib															
	3 D:\Working\	\PRESSU	IRE.lib														
	4 D:\Working\	\Buffer_	sub.LIB														
	5 D:\Working\	\MY_DIC	DE.lib														
	6 D:\Working\	\ZENER.I	lib														
	Exit																
-				1													
Open	an existing libr	ary													(	CAP NUM	SCRL

## File>Open and select the file.

→ · · · · · · · · · · · · · · · · · · ·	> Model Editor 🗸 🗸	ර් Sear	ch Model Editor		9
Organise 🔻 New folder			=== -		?
20_048459c_arch ^ Name	Date mod	ified	Туре	Size	
allegro	12/06/200	6 14:04	LIB File		
MasterDocs_174					
New-DemoP					
land OneDrive					
💻 This PC					
🗊 3D Objects					
E. Desktop					
🚔 Documents					
🕹 Downloads					
b Music					
E Pictures					
📑 Videos					
Line Windows (C:)					
DATA (D:) V <					
File name: SampleSpice.lib		~ Mo	del Libraries (*.lib)		~
			Open	Cancel	-

Using Spice Models in OrCAD PSpice from vendor models File Edit View Model Plot Tools Window Help cādence® Models List Model Name Type Modifi QBC546A BJT PMST6429 SUBCKT 2N7002 SUBCKT OPA277 SUBCKT < CAP NUM SCRL

With the LIB file open, the model names and types are listed.

🛐 P	Spice Model Editor						 					- 0	×
File	Edit View N	/lodel Pl	lot Tools	Window	Help							cāde	n ce®
10	New		Ctrl+N	D R	$\oplus$	Θ.13	Ċ,	XIY		일교			
10	Open		Ctrl+O						 	HUTU -			
	Close												
18	Save		Ctrl+S										
1	Save As												
1	Print		CtrI+P										
a	Print Preview												
	Page Setup												
	Export to Capture	e Part Libra	ary										
	Model Import Wiz	zard [Capt	ure]										
D	Encrypt Library												
	1 SampleSpice.lib												
	2 D:\Working\\v	wc.lib											
	3 D:\Working\\F	PRESSURE	lib										
	4 D:\Working\\B	Buffer_sub	o.LIB										
	5 D:\Working\\M	MY_DIODE	lib										
	6 D:\Working\\Z	ZENER.lib											
	Exit												
<			>										
Save1	the active library											CAP NUM	SCRL!

Save the LIB file from the Model Editor to add the "PSpice indexing marks" to the LIB file.

#### Aside

All of the models supplied with the product installation and models available from vendors are text, or encrypted text. Searching large amounts of text to find models within the configured simulation libraries is not very efficient so PSpice uses index, IND, files and "indexing marks" to make searching for models significantly more efficient. The libraries supplied with the product are indexed during installation, any additional libraries are indexed when

the library is first used with a simulation, the index and LIB file timestamps are also compared at the start of every simulation and, if required, the indexes are rebuilt.

## Creating the Graphical Symbols for use in a Schematic

There are two options for generating graphical symbols for use in a schematic, in both cases, models of the .MODEL type will be associated with the appropriate intrinsic PSpice device graphical symbol. In the "default" method, which will be described first, the .SUBCKT type will have a rectangular shape with pins created as the graphics.

Use File>Export to Capture Part Library:

Ŕ	🕯 PS	pice Mo	del Edi	tor															-		×
ł	File	Edit	View	Model	Plot	Tools	Window	Help											(	ādenc	:e®
1	$\square$	New			C	Ctrl+N	n r	(+)	Θ	17		Ċ	XI	y 4		신신					
i N	$\square$	Open			c	Ctrl+O		~	~	-0(	~		iiiii I		S( - 1	E ⊂ AUTO	*				
5		Close																			
Ľ		Save				Ctrl+S															
		Save As	i																		
1		Print			(	Ctrl+P															
1	G	Print Pr	review																		
		Page Se	etup																		
		Export	to Capt	ure Part L	.ibrary																
		Model	Import	Wizard [C	apture]																
	ħ	Encount	Libran	,																	
		encippi																			
		1 Samp	leSpice	.lib																	
		2 D:\W	orking\	\wc.lib																	
		3 D:\W	orking\	\PRESSU	JRE.IID																
		4 D:\\W	orking\	\Butter_	SUD.LIB	)															
		6 D:\W	orking\		JUE.IID																
		0.0.00	orking																		
		Exit																			
4	:				>																
C	reate	e Captur	e or De	sign Entry	y HDL Pa	arts for t	the Indicate	d Mode	l Libraŋ	/									CA	NUM SCR	al la

The library selected for input will automatically be the library that is currently open:

Create Parts for Library	×
Enter Input Model Library: D:\Working\Model Editor\SampleSpice.lib Browse	
Enter Output Part Library:	
D:\Working\Model Editor\SampleSpice.olb Browse	
OK Cancel Help	

Select OK to complete the library creation and the resulting status will be displayed:

D:\Working\Model Editor\SampleSpice.err	×
STATUS: PSpice Schematics to Capture translator (17.4.0.d001) STATUS: STATUS: Translator started at Friday, January 31, 2020 12:07:55 STATUS: C:VCadence/SPB_17.4\tools\bin\sch2cap.f ''D:\Working\Model Editor\SampleSpice.lib'' -o ''D:\W INF0(0RSCH-1029): Using existing library 'C:\Cadence\SPB_17.4\tools\Capture\Library\PSpice\modeled.etc INF0(0RSCH-1031): Created new library 'D:\Working\Model Editor\SampleSpice.olb'. STATUS: Translator stopped at Friday, January 31, 2020 12:07:56 STATUS: 0 Error messages, 0 Warning messages	
	×
ОК	

That completes the "default" method, the resulting graphical symbols will be rectangles with pins for the .SUBCKT model type.

The second method uses the Model Import Wizard, in this the .SUBCKT model types can be associated with existing graphical symbols with a matching number of pins, either from the existing graphical libraries, or a user library generated for the purpose that holds "generic" symbols for common models that are likely to be imported – this just results in a much shorter list of symbols to search. In this case the user library is called "zzGenerics.olb". Start the Model Import Wizard:



© 2020 Parallel Systems Limited

Current library is again selected:

Model Import Wizard :	Specify Library	×
	Model Import Wizard automatically associates symbols for all the PSpice models it recognizes. It facilitates the user to : - associate symbols for the PSpice models that could not be recognized automatically. - update existing symbols for the PSpice models.	
	Enter Input Model Library : D:\Working\Model Editor\SampleSpice.lib Browse.	
	Enter Destination Symbol Library :           D:\Working\Model Editor\SampleSpice.olb         Browse.	
	< Back Next > Cancel H	lelp

Next will list the parts in the LIB file, the intrinsic part will have an associated symbol by default:

Model Import Wizard : Associate/Replace S	ymbol		×
Destination Symbol Libr You can do either of the (1) associate symbol for	rary : D:\Working\Model Editor e following : models without symbol, or (2) m ol	rr\SampleSpice.olb replace existing symbol for models. pol Symbol : QBC546A	
Model Name	Symbol Name		
Although and QBC546A	QBC546A	-	
PMST6429			
2N7002			
PA2/7		-	
Contraction of the second s		<b>↓</b>	
Rep	lace Symbol		
View Model	< Back	Finish Cancel He	lp.

The Symbol can be replaced if desired. The next part is selected:

Using Spice Models in OrCAD PSpice from vendor models	
Model Import Wizard : Associate/Replace Symbol         Image: Symbol Wizard : Associate/Replace Symbol         Destination Symbol Library : D:\Working\Model Editor\SampleSpice.olb         You can do either of the following :         (1) associate symbol for models without symbol, or (2) replace existing symbol for models.         Image: Models with symbol         Image: Models with symbol	×
Model Name       Symbol Name         QBC546A       QBC546A         PMST6429       2N7002         OPA277       0         Image: Comparison of the symbol Name       Image: Comparison of the symbol Name         Model Name       Symbol Name         QBC546A       QBC546A         PMST6429       Image: Comparison of the symbol Name         OPA277       Image: Comparison of the symbol Name         Mathematical Symbol Name       Image: Comparison of the symbol Name         Mathematical Symbol Name       Image: Comparison of the symbol Name         Mathematical Symbol Name       Image: Comparison of the symbol Name         Mathematical Symbol Name       Image: Comparison of the symbol Name	
View Model < Back Finish Cancel Help	
This is not an intrinsic part so a graphical symbol is required, use the Associate Symbol:	
Model Import Wizard : Select Matching	×
Select library to pick matching symbols : D:\Working\Model Editor\ZZGENERICS.OLB	

Contraction of the second	PMOSFET3	
Sector Sector	NPN	
	PNP	
CARDING PARTY IN		
and the second s		·
BACC/		
AV NYA		
AVANA.		

Show All

The zzGenerics library has been chosen as the graphical symbol source in this case, the model has 3 pins so all the symbols in the graphical library are listed as candidates, and the NPN symbol is the required one:

Model Import Wiza	rd : Select Matching	×
r <u>110</u>	Select library to pick matching symbols :	
	D:\Working\Model Editor\ZZGENERICS.OLB	✓ …
A Reference of the second seco	Model : PMST6429	Part : NPN
and a second sec	Show All	
	Matching Symbols	
Statement and	PMOSFET3	
And Annual Control of the second seco	NMOSFET3	
	NPN	
	PNP	
SSC1		
	V	ļ
View Model	< Back N	ext > Cancel Help
lext to get to the Pin	Association stage:	
Ū	-	
Model Import Wiz	ard : Define Pin Mapping	×
710	For each model terminal, associate a symbol pin. The optic	onal model terminals may be left
the second s		

	Show Invisible Symbol	Power Pins		
	Model Terminal	Symbol Pin	^	
A REAL PROPERTY AND A REAL	1	~		
	2	<none></none>		
	2			 
		e		
Contraction of Contraction				
O. Official and				
			v .	
		1		

The model pins are 1, 2, 3 for an NPN transistor the Pin Order c, b, e. The pins are selected from the drop-down list:

Using Spice N	Models in OrCAD	PSpice from vendo	or models		
	Model Import Wiza	rd : Define Pin Mapping			×
		For each model terminal, as unassociated. Use "View M	ssociate a symbol pin. The optio Model Text'' button to view the r	nal model terminals may be left nodel definition.	
		Model : PMST6429		Part : NPN	
		Show Invisible Symbol	Power Pins		
		Model Terminal	Symbol Pin ^		
		2	b		
		3	e ~		
	ASS 0				
			¥		
	View Model		< Back Save	Symbol Cancel Help	
Save Symbol	will save the Syn	nbol and Pin Associ	iation and name the	created symbol after the m	nodel name.
	Model Import Wizard	: Associate/Replace Symb	ool		×
	71	Destination Symbol Library :	D:\Working\Model Editor\San	npleSpice.olb	

	Destination Symbol Library You can do either of the fo (1) associate symbol for mo	: D:\Working\Model Edito llowing : dels without symbol, or (2)	r\Sam replac	npleSpice.olb ee existing symbol for models.	
	Models with symbol	Models without symb	ol	Symbol : PMST6429	
AND DESCRIPTION OF A DE	Model Name	Symbol Name	^		
A least lease least leas	QBC546A	QBC546A			
Left Land Autor 1 rank 6 Transaction and Control of Con	PMST6429	PMST6429			
A state of some sources on the source of the	2N7002				
And December 2011	OPA277				
And a second sec					
A CONTRACTOR OF A CONTRACT OF					
and the restance of the second					
ACCOUNT OF A DESCRIPTION	J		Υ.		
	Replac	e Symbol			
	1				
View Model		< Back		Finish Cancel Help	

Repeat the process for the other models to complete the association, any .SUBCKT types not mapped will have default, rectangular graphics with pins created.

	Destination Symbol Library You can do either of the fol 1) associate symbol for mo	: D:\Working\Model Editor\ lowing : dels without symbol, or (2) rej	Sample	Spice.olb
	You can do either of the fol 1) associate symbol for mo	lowing : dels without symbol, or (2) rej	place e	
And Annual Control of				existing symbol for models.
A DE LE CONTRACTOR DE L	Models with symbol	Models without symbol	5	Symbol : OPA277
AND A DAY	Model Name	Symbol Name	~	
1 2 Annual Sector II And Annual Sector and Annual Physics Sector and Annual Sector and Annual Sector Annual A Annual Annual Annu	QBC546A	QBC546A		
The second secon	PMST6429	PMST6429		
1 August 1 A	2N7002	2N7002		
No In Mala Addia In Contra Con	OPA277	OPA277		
an para di Ala San di A		•		
				001
A DECEMBER OF THE OWNER.				
and the state of the state				k
			~	
	Penlac	e Symbol		
		Model Name QBC546A PMST6429 2N7002 DPA277	Model Name QBC546A QBC546A PMST6429 2N7002 DPA277 OPA277 OPA277	Model Name     Symbol Name       CBC546A     CBC546A       PMST6429     PMST6429       2N7002     2N7002       OPA277     OPA277

Model Import Wizard : D:\Working\Model Editor\SampleSpice.symwiz.log X
INFO(ORSCH-1132): Log File Error File
STATUS: PSpice Model Import Wizard for "Capture" (17.4.0.d001) STATUS: INFO: LIB driven flow INFO: Input File: D:\Working\Model Editor\SampleSpice.olb STATUS: STATUS: Identifying matching symbols automaticallystarted at Friday, January 31, 2020 12:25:08 STATUS: INFO: Symbol "QBC546A" created for model "QBC546A". STATUS: STATUS: STATUS: Completed identifying matching symbols automaticallyat Friday, January 31, 2020 12:25 STATUS: INFO: Symbol "PMST6429" created for model "PMST6429". INFO: Symbol "PMST6429" created for model "PMST6429". INFO: Symbol "OPA277" created for model "OPA277". STATUS: 0 Error messages, 0 Warning messages
ОК

Again the library creation status is reported and the OLB file created.

The Model Editor can be closed at this point.

### Using the created parts in Simulations

Now that the graphical symbol, OLB, files and simulation model, LIB, files have been created, the libraries can be added to OrCAD Capture, a schematic created and the simulation run using the new models. Start OrCAD Capture, and a new project, File>New>Project. Make sure you check "enable PSpice Simulation"

### **Default symbols:**

To test this, a simple design with two Sallen-Key filters and the default rectangular sub-circuit symbol will be used. Whilst probably not aesthetically pleasing, the default rectangular symbol is perfectly adequate for "quick test" simulation purposes.

sing Spice Models in OrCAD PSpice from vendor models				
New Proje	ct	×		
Name	ModelTest			
Location	D:\Working\Model Editor			
	Enable PSpice Simulation			
	OK Cancel Help			
ne and dire	ctory, base this project on an "empty" project, or cr	reat		

Create PSpice Project		×
Create based upon an existing project		OK
empty.opj	Ŧ	Browse
Create a blank project		Cancel
		Help

Open PAGE1 of the project and draw the schematic.

Analog or Mixed A/D	
🛅 File 🏣 Hierarchy	I
🖃 🗂 Design Resources	I
i⊒  modeltest.dsn	I
SCHEMATIC1	I
PAGE1	I
🗄 🗝 Design Cache	I
Ē Library	I
Layout	I
Outputs	

Use Place>Part and the rectangular Add Library icon button to add the created OLB file.

Place Part	<b>▼</b> ×	
Part	<u></u>	£
Part List: 101/OPAMP 10104/OPAMP	7	
10104/TO/OPAMP 10106/OPAMP 107/OPAMP 11004/OPAMP 11401/OPAMP		~ 바 .
Libraries:		
ANALOG BREAKOUT Design Cache MISC	Add Library (Alt+A	)
OPAMP	-	•

Using Spi	ce Models in OrCAD PS	pice from vendor model		
This will s	elect the created OLB f	ile in the library list.		
	Place Part	<b>▼</b> ×		
	Part			
	2N7002			
	Part List:	Y		
	2N7002			
	OPA277 PMST6429			
	QBC546A			
			Place PCB SI Analysis PSpice Accessories Options W	/indow H
	Libraries:		Part P DPA277	ē.
	MISC		PSpice Component   PSpice Ground	
	OPAMP		Search Providers Capacitor	
	SAMPLESPICE		Database Part Z Diode	
	SOURCE	<b>v</b>	J <sup>o</sup> Wire W Inductor	
	Packag	ing	Auto Wire Resistor	
	Parts pe	r Pkg: 1	DIgital	•
	U? 💾 Part:	-	Junction J Discrete	•
		omogeneous	A Bus Entry E Passives	<b>&gt;</b>
	2N7002	onogonoda	Net Allas N Source	+
	⊕		Ver Power E Search	
			Power P	

Select the OPA227, place two of them and complete the circuit. Ensure you place parts from the PSpice enabled libraries.





Use PSpice>Edit Simulation Profile, or create a New Simulation Profile.

Using Spice Models in OrCAD PSpice from vendor models				
	PSp	oice Accessories C	Options	Window
	~	New Simulation Pro	file	
		Edit Simulation Prof	file	
	€	Run		F11
		View Simulation Res	sults	F12
		View Output File		
		Create Netlist		
		View Netlist		
		Advanced Analysis		•
		Markers		Þ
		Bias Points		•

On the Analysis tab, for this case, set the profile for an AC Sweep from 10Hz to 100kHz.

Simulation Settings - bias		
General Analysis Configuration Files Options Data Collection Probe Window	Analysis Type: AC Sweep/Noise Options: General Settings Monte Carlo/Worst Case Parametric Sweep Temperature (Sweep) Save Bias Point Load Bias Point	AC Sweep Type Linear Start Frequency: 10 Logarithmic End Frequency: 100k Decade Points/Decade: 11 Noise Analysis Enabled Output Voltage: I/V Source: Interval: Output File Options Include detailed bias point information for nonlinear controlled sources and semiconductors (.OP)
		OK Cancel Apply Reset Help

Also, while still editing the profile, go to the Configuration Files, select the Library entry on the left and use the upper Browse button to browse for the LIB file that defines the simulation models.

Simulation Settings - bias		
General Analysis Configuration Files	Category: Stimulus Library Include	Filename:       D:\Working\Model Editor\SampleSpice.lib       Browse       Configured Files
Options Data Collection Probe Window	Update Index	Add as Global Add to Design Add to Profile Edit Change
		Library Path "C:\Cadence\SPB_17.4\tools\PSpice\Library" Browse
		OK Cancel Apply Reset Help

Use the Add to Design button to add the simulation library.

nulation Settings - bias						
General	Category:	Filename:				
Analysis	Stimulus	D:\Working\Model Editor\SampleSpice.lib	Browse			
Configuration Files	Library	Configured Files	+			
Options Data Collection Probe Window	moude	D:\Working\Model Editor\SampleSpice.lib	Add as Global			
	Update Index	om.lib*	Add to Design			
			Add to Profile			
			Edit			
			Change			
		Library Path				
		"C:\Cadence\SPB_17.4\tools\PSpice\Library"	Browse			

OK the Simulation Settings to close the simulation Profile.

From the PSpice menu use Run to run the simulation. Once the simulation has completed the Probe window will open with the simulation results and the traces for the active probes displayed.

🚟 so	HEMATIC1-bias - PSpice A/D - [bias.dat	(active)]						- 🗆 ×		
File	Edit View Simulation Trace P	it View Simulation Trace Plot Tools Window Help CādenC								
l P		) R S C A		CHEMATIC1-bias			$\Theta \in \Theta$	QQN		
	bias.dat (active) ×									
*	20									
Ð,										
a										
, a										
~~ <u>*</u>										
Ľ∕∕2	0						<u>D</u>			
Ø							*			
4										
		£								
	-20							4		
	-40						i			
	10Hz □ DB(V(U2:OUT)) ◇	100 DB(V(U1:-))	Hz	1.0	KHZ	10	KHZ	100KHz		
Frequency										
Output Window										
	Circuit read in and checked, no errors Calculating bias point							<b></b>		
	Bias point calculated									
	AC (and Noise) Analysis AC Analysis finished									
	Total job time (using Solver 1) = INFO(ORPROBE-3190): Simulation com	.45 nplete								
								•		
	Simulation Status Output Window	Command Window	aias dat (active)			Freg = 100	0F±03 100%			
	and an	preer nes (ser nemerre l'(blas (t	inasiaat (active)			ricq = 100.	100%			

# Model Import Wizard Symbols

The circuit is redrawn with the Symbols created by the Model Import Wizard process and the simulation is run again. The modified circuit is shown below:

Using Spice Models in OrCAD PSpice from vendor models



When the simulation is run again, the same results are displayed, as before:



The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134 Allegro®, Cadence®, Cadence logo™, Concept®, NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®

from claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.

Other Trademarks All other trademarks are the exclusive property of their prospective owners.

NOTICE OF DISCLAIMER: Parallel Systems is providing this design, code, or information "as is." By providing the design, code, or information as one possible implementation of this feature, application, or standard, Parallel Systems makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Parallel Systems expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free