# Piece-Wise Linear (PWL) Voltage Sources





Copyright © 2017 Analog Devices. All rights reserved.

## Creating a PWL Voltage Source

- Open up the simulation file titled "RCFilterTimeDomain.asc"
- Run the simulation and probe the IN and OUT nodes
- Right-Click on the voltage source and select the PWL function
- Configure the PWL source to manually recreate the pulse waveform as shown in the voltage source window on the right
- Rerun the simulation. Notice a single pulse is now present.

🛛 Linear Technol 🗖 🗖 🗙	Independent Voltage Source - V1	<u> </u>
K <u>F</u> ile <u>E</u> dit H <u>i</u> erarchy <u>V</u> iew <u>S</u> imulate Tools <u>W</u> indow <u>H</u> elp _	<ul> <li>Functions</li> <li>(none)</li> <li>PULSE(V1 V2 Tdelay Trise Tfall Ton Period Ncycles)</li> </ul>	DC value:
	<ul> <li>SINE(Voffset Vamp Freq Td Theta Phi Ncycles)</li> <li>EXP(V1 V2 Td1 Tau1 Td2 Tau2)</li> <li>SFFM(Voff Vamp Fcar MDI Fsig)</li> <li>PWL(t1 v1 t2 v2)</li> </ul>	Small signal AC analysis(.AC) AC Amplitude: AC Phase:
IN R1 OUT 10K C1 .1μ .1μ PULSE(0 5 1m 1u 1u 10m 20m 3) teon 60m	PWL FILE:       Browse         time1[s]:       0         value1[V]:       0         time2[s]:       1u         value2[V]:       5         time3[s]:       10m         value3[V]:       5         time4[s]:       10.001m	Make this information visible on schematic: ♥ Parasitic Properties Series Resistance[Ω]: Parallel Capacitance[F]: Make this information visible on schematic: ♥
	value4[V]: 0 Additional PWL Points Make this information visible on schematic: ✔	Cancel OK





RCFilterTimeDomain.asc

CS

#### **Repeating PWL Source**

- Open up the simulation file titled "RCFilterTimeDomainRepeat.asc"
- Run the simulation and probe the IN and OUT nodes
- Right-Click on the PWL text string and use the repeat command to create three cycles of the input square wave.

Enter new Value for V1						
Justification Left v Vertical Text	OK Cancel					
PwL repeat for 3 (0 0 1 u	5 10m 5 10.001m 0 20m 0) endrepeat					









CS

#### Importing Externally Generated PWL Sources

- To import a PWL waveform from a file, Right-Click on a voltage source, select "Advanced", and select "PWL File"
- The file format must contain pairs of numbers separated by white space (carriage return, spaces, tabs). The first number is time (in seconds) and the second number is voltage.
- Like the standard PWL, you can also repeat, as shown in these examples
- Example 1 (datapoints in a line):

**\***0 0 0.1 1 0.2 0.5 0.5 0 0.7 0.3 1 0

Example 2 (datapoints in columns):

**\***00

**\***0.1 1

**\***0.2 0.5

**\***0.5 0

**\***0.7 0.3

**\***10

PWLExamplesLine.asc

PWLExamplesColumn.asc



## Importing a PWL Waveform From an Oscilloscope

- PWL waveform data can be imported into LTspice from an oscilloscope waveform capture
- Data can be comma separated or space separated data pairs (Time Value). Ex. .csv file.
- Many Oscilloscopes will export in this format (ex. Save Format Spreadsheet option shown below)





## Importing a PWL Waveform From a File

Open up the simulation file titled "TekImport.asc" and follow the instructions in the simulation file.



CS

TekImport.asc

#### Importing a PWL Waveform From a File

One thing to notice is that data with a negative time is ignored. Your scope may consider the trigger time to be t=0 in the output file. If you want to use this data, edit the time column in the spreadsheet as needed.

NOW PART OF

	9-6	- <u>\</u>	-	-	C. Carro	-	TEK00000.C	SV - Mic	rosoft Excel			-	_	1.41	-		x
File	File Home Insert Page Layout Formulas Data Review View Developer 🛆 🕜 🗆 🛱 🔀											67 X					
Paste	Å ™ ₹	alibri B I <u>U</u> -	• 11 •	A A <u>A</u> .		≫- ≇ ≇ 8	🖥 Wrap Text 🖻 Merge & Cente	Gene	eral • % • .	.00,00 00,€	Conditiona Formatting	I Format ▼ Table	as Cell • Styles •	and Insert → The Insert → Delete → Format →	Σ · /	ort & Find & ilter * Select *	
Clipbo	bard 🖫	Font 🕞				Alignment 🕞 Number 🕞			á Styles			Cells Editing					
	A1	• (*	$f_x$	-0.024	9456												~
	А	В	С	D	E	F	G	Н	I		J	K	L	М	N	0	
239	-0.00115	-0.084															
240	-0.00105	-0.088															
241	-0.00095	-0.092															
242	-0.00085	-0.082															
243	-0.00075	-0.078															
244	-0.00065	-0.064															
245	-0.00055	-0.05															
246	-0.00045	-0.038															
247	-0.00035	-0.022															
248	-0.00025	-0.006															
249	-0.00015	0.006															
250	-4.56E-05	0.004															
251	5.44E-05	0															
252	0.000154	0.008															
253	0.000254	0.01															
254	0.000354	0.008															
255	0.000454	-0.006															
256	0.000554	0.002															
257	0.000654	-0.008															
258	0.000754	-0.006															
259	0.000854	-0.008															
260	0.000954	-0.002															
261	0.001054	-0.002															
262	0.001154	-0.004															-
H 4																	
Ready	y 🛅													□ □ 100%	Θ		+

## **Exporting Waveforms to ASCII**

- With a waveform window active, go to File -> Export.
- Select the node voltage or current to export and click OK.
- The file location is shown at the top of the dialog box.





