

INTRODUCTION TO HOW TO USE VISHAY MODELS WITH LTSPICE

Downloading LT SPICE from ADI website:

1. Go to ADI LT SPICE Website,
<https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>
2. Follow Instructions

Download LTspice

Download our LTspice simulation software for the following operating systems:

[Download for Windows 7, 8 and 10](#) Updated on Nov 26 2018

[Download for Mac OS X 10.7+](#) Updated on Nov 29 2018

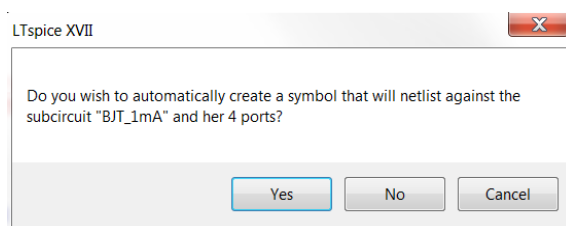
[Download for Windows XP](#) (End of Support)

Generating an LTSPICE symbol and “plugging it in” to a Vishay netlist

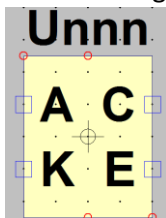
1. Got to netlist and highlight the .SUBCKT line.

```
* ==== BJT_1mA ====  
* A = diode anode  
* K = diode cathode  
* C = BJT collector  
* E = BJT emitter  
*$  
.SUBCKT BJT_1mA A K C E PARAMS: REL, CTR=1
```

2. Right click on Highlighted SUBCKT statement.
3. Go to the “Create symbol” command.
4. LTSPICE will ask you if you ‘really’ want to generate a symbol – Say “YES”



5. LTSPICE will generate a box with all the pins defined in the Netlist file



- a. If you are OK with working with a box, you are good to go.
 - b. If working with a “box” offends your aesthetic sensitivities this box can be edited to generate a more meaningful and aesthetically pleasing symbol
6. Too easy hey? Kind of the way software should be!