Third-Party Models, Creating and Editing Symbols, and Managing Libraries in LTspice

CHARLY EL-KHOURY

IN PARTNERSHIP WITH ARROW

LTspice
Free – Fast – Unlimited

Agenda

► Importing Third Party Models

► Creating Schematic Symbols

► Managing Libraries

► Additional resources and tools

► Homework assignment

**ANALOG
DEVICES**
AHEAD OF WHAT'S POSSIBLE™

Importing Third Party Models

LTspice
Free – Fast – Unlimited

Two Types of Third-Party SPICE Models

- ► .MODEL
  - Intrinsic SPICE devices like diodes and transistors
    - Behavior of the device is intrinsically understood by SPICE
  - Statement provide the parameters to specify the component's electrical characteristics
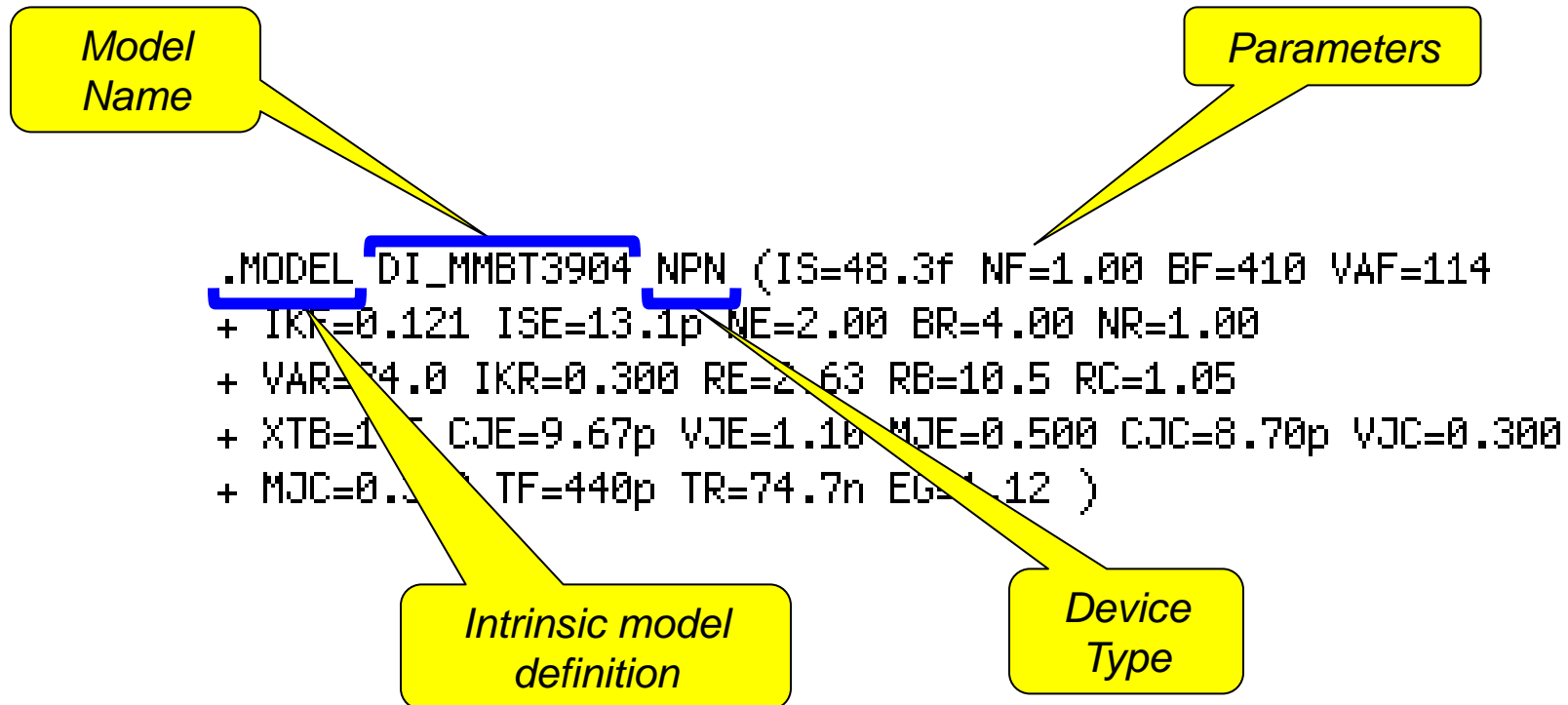
- ► .SUBCKT
  - Define the modeled component by a collection of circuitry (of intrinsic SPICE devices) like an op amp

> The method to import a model in LTspice depends on whether the model is given as a .MODEL or a .SUBCKT

ANALOG
DEVICES

AHEAD OF WHAT'S POSSIBLE™

Intrinsic SPICE Models Syntax

.model <*modelname*> <*DeviceType*> *(*<*parameter list*>*)*

Model Name

Parameters

```
.MODEL DI_MMBT3904 NPN (IS=48.3f NF=1.00 BF=410 VAF=114
+ IKF=0.121 ISE=13.1p NE=2.00 BR=4.00 NR=1.00
+ VAR=34.0 IKR=0.300 RE=2.63 RB=10.5 RC=1.05
+ XTB=1.5 CJE=9.67p VJE=1.10 MJE=0.500 CJC=8.70p VJC=0.300
+ MJC=0.... TF=440p TR=74.7n EG=1.12 )
```

Intrinsic model definition

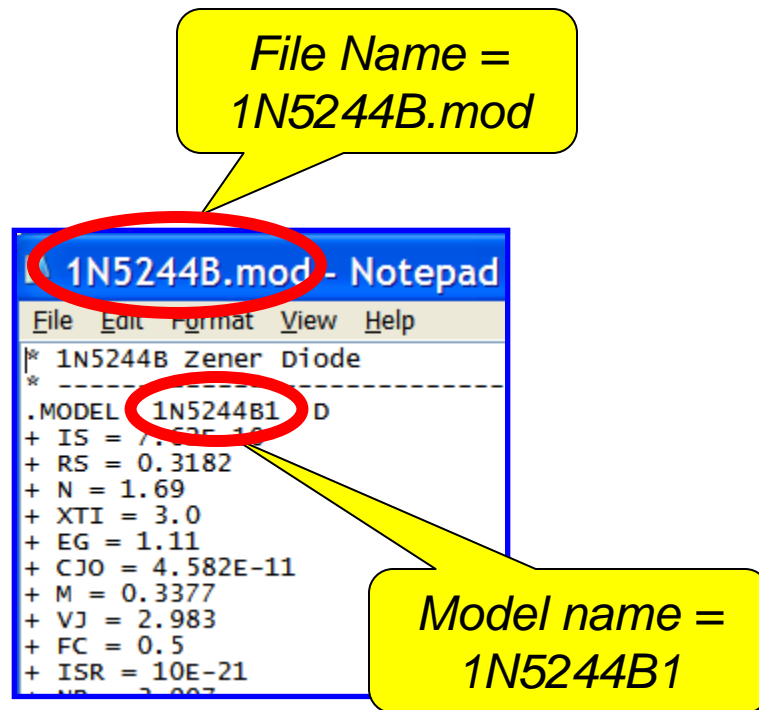**Device Type**

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE™

Importing Intrinsic SPICE Models

► Download the model file from the manufacturer's website to your development directory

► Add .include [path(optional)] spicemodel_filename.abc directive to the schematic
  ▪ Path can be omitted if the file is in the same directory as schematic file (recommended)

► Open model file in LTspice and note the device name and device type

► Add the device type symbol and edit the component attribute (Ctrl + right click) to match the device name contained in the SPICE model file
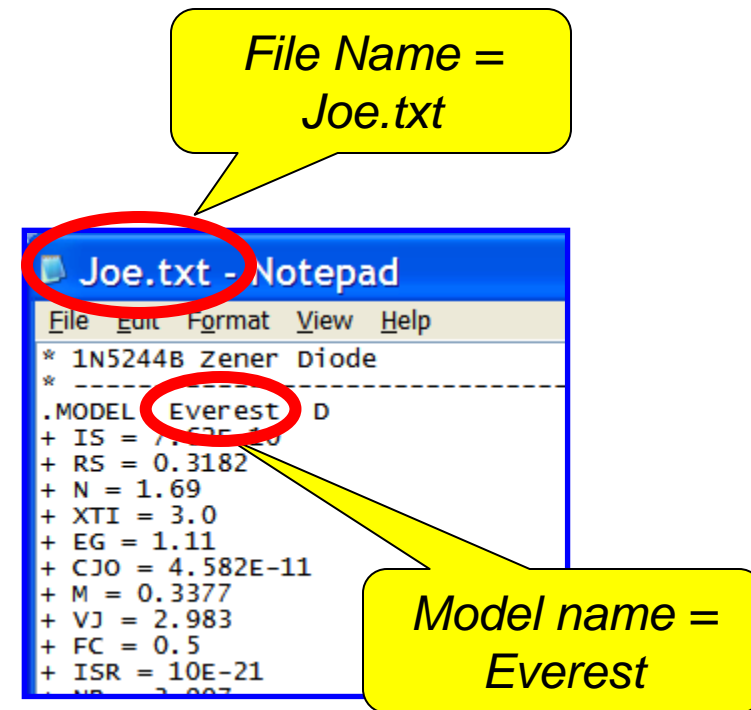

Device name Right-click to edit

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE™
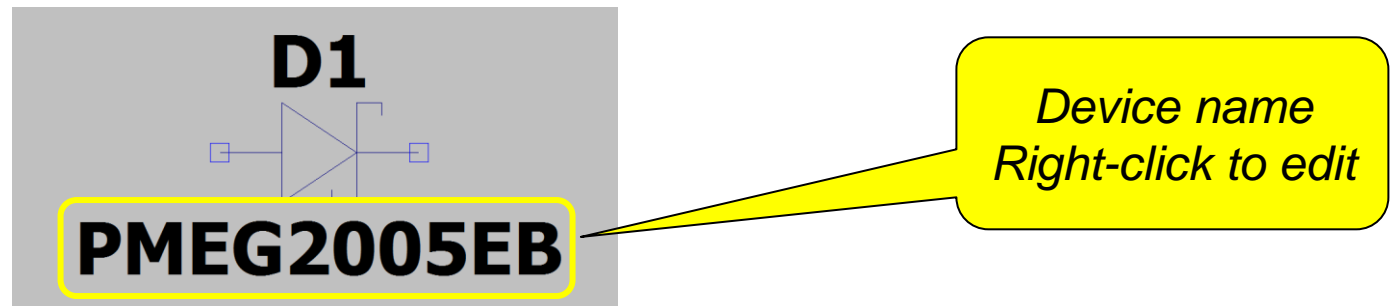
Third-Party Intrinsic Models – Model File

Spice Model Example #1:

Spice Model Example #2:

File Name = 1N5244B.mod

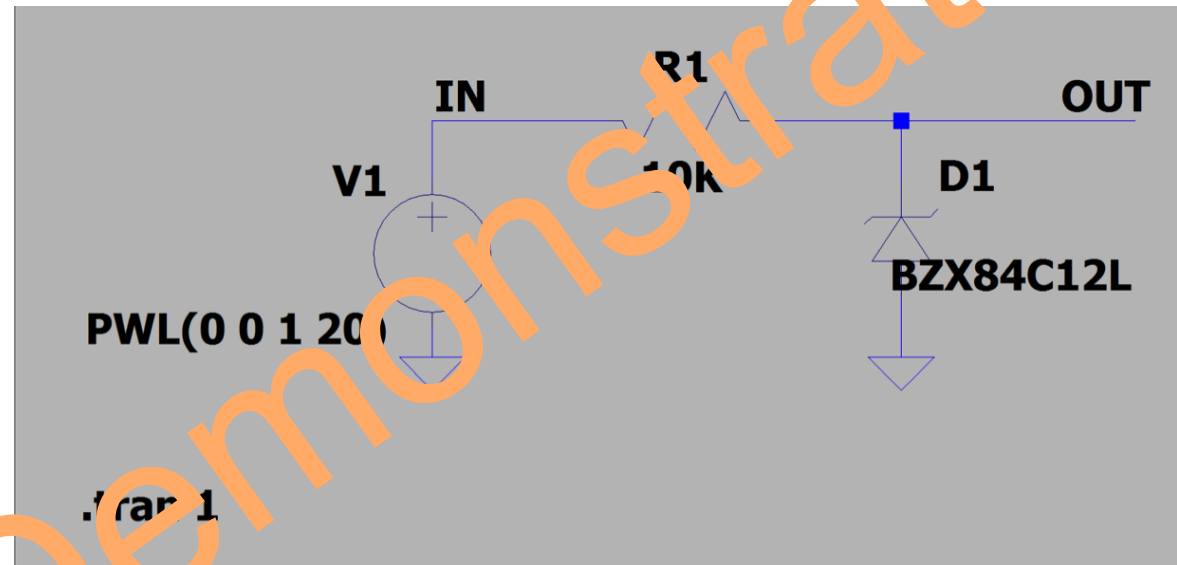Model name = 1N5244B1

File Name = Joe.txt

Model name = Everest

The file name in the .include statement must match the model file name

The symbol's device name in the schematic must match the model's name

**ANALOG
DEVICES**
AHEAD OF WHAT'S POSSIBLE™

Importing Third-Party Spice Models

► Modify "ZenerImportExample.asc" so it uses the 1N5244B third-party SPICE model.

ZenerImportExample.asc

Do not forget to place the .include statement on the schematic

ANALOG
DEVICES
AHEAD OF WHAT'S POSSIBLE™

Subcircuits Syntax

.subckt `<modelname> <nodelist>`

`<circuit of intrinsic devices>`

`<definition of intrinsic models>`

.ends

► Intrinsic models defined within a .subckt are only accessible within that .subckt model

Subcircuit model definition

Nodes

Model Name

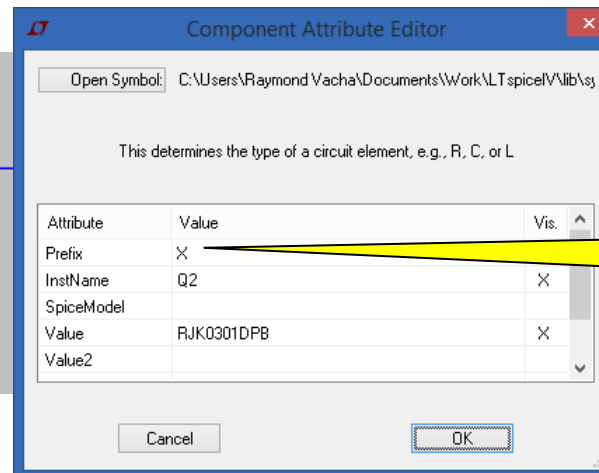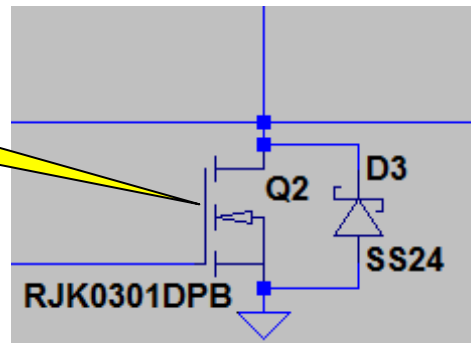**Circuit Definition**

Intrinsic models within subckt

.ends

```
.SUBCKT SiS892DN D G S
M1 3 GX S S NMOS W= 1116756u L= 3.05u
M2 S GX S D PMOS W= 1116756u L=3.310e-07
R1 D 3 1.577e-02 TC=7.417e-03 1.832e-05
CGS GX S 4.316e-10
CGD GX D 1.423e-11
RG G GY 5.5
RTCV 100 S 1e6 TC=2.273e-05 -2.394e-07
ETCV GX GY 100 200 1
ITCV S 100 1u
VTCV 200 S 1
DBD S D DBD

.MODEL NMOS NMOS ( LEVEL = 3 TOX = 5e-8 RS = 4.524e-03
+ KP = 1.242e-05 NSUB = 1.365e+17 KAPPA = 2.400e-03
+ ETA = 1.000e-07 NFS = 7.622e+11 LD = 0 IS = 0 TPG = 1)

.MODEL PMOS PMOS ( LEVEL = 3 TOX = 5e-8 NSUB = 2.187e+16
+ IS = 0 TPG = -1 )
**************************************************************
.MODEL DBD D (FC = 0.1 TT = 2.427e-08 T_MEASURED = 25 BV = 102
+RS = 4.567e-03 N = 1.079e+00 IS = 3.311e-12 EG = 1.144e+00 XTI = 8.671e-01
+ TRS1 = 1.135e-03 CJO = 9.822e-10 VJ = 9.000e-01 M = 2.425e-01 )
.ENDS SiS892DN
```

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE™

Importing Subckt SPICE Models

- ► If the .subckt model does not have a schematic symbol or one is not available in LTspice, create a new one (covered next)

- ► Verify the .subckt model is a match for the associated symbol:
  - .subckt model has the same number of nodes as there are pins on the schematic symbol
  - The symbol's netlist order matches the sequence of the model's nodes
    - Left most node in the model definition is associated to the 1st position of the netlist order
    - Increments as such until the right most node corresponds to the last netlist order position

- ► Repeat the steps used to import a third-party intrinsic spice model

- ► In the symbol component attribute editor (Ctrl-Right-Click on the symbol), change the device's prefix to "X"
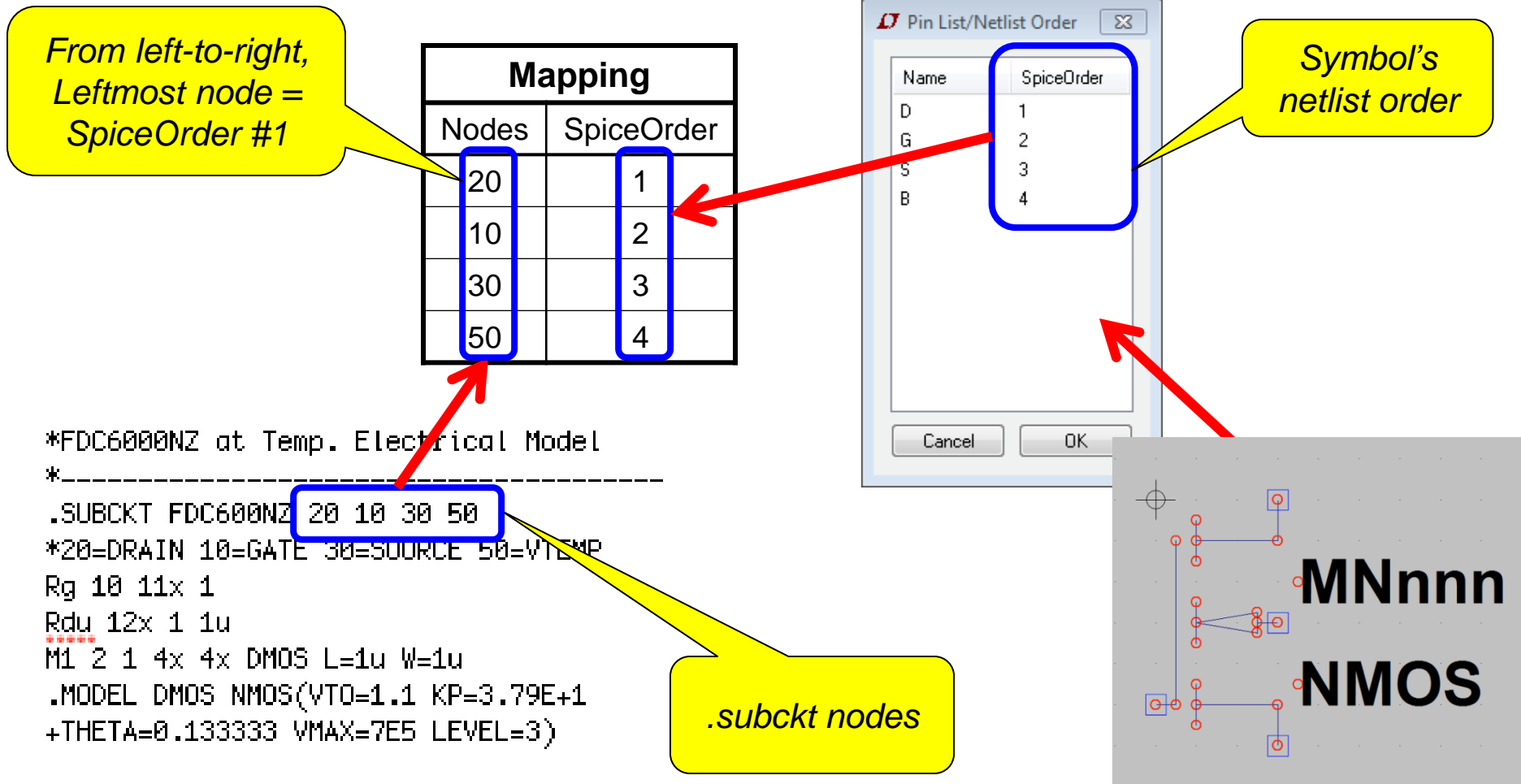
CTRL+Right-Click

Change Prefix to "X"

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE™

Symbol's netlist order and model's node sequence

From left-to-right, Leftmost node = SpiceOrder #1

Mapping

Nodes	SpiceOrder
20	1
10	2
30	3
50	4

Symbol's netlist order

Pin List/Netlist Order

Name	SpiceOrder
D	1
G	2
S	3
B	4

Cancel   OK

```
*FDC6000NZ at Temp. Electrical Model
*---------------------------------------
.SUBCKT FDC600NZ 20 10 30 50
*20=DRAIN 10=GATE 30=SOURCE 50=VTEMP
Rg 10 11x 1
Rdu 12x 1 1u
M1 2 1 4x 4x DMOS L=1u W=1u
.MODEL DMOS NMOS(VTO=1.1 KP=3.79E+1
+THETA=0.133333 VMAX=7E5 LEVEL=3)
```

.subckt nodes

MNnnn

NMOS

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE™

Importing Third-Party Subckt Models

► Change the "LT3741.asc" circuit so it uses the PSMN2R2-30YLC FET subcircuit model instead of the RJK0301DPB
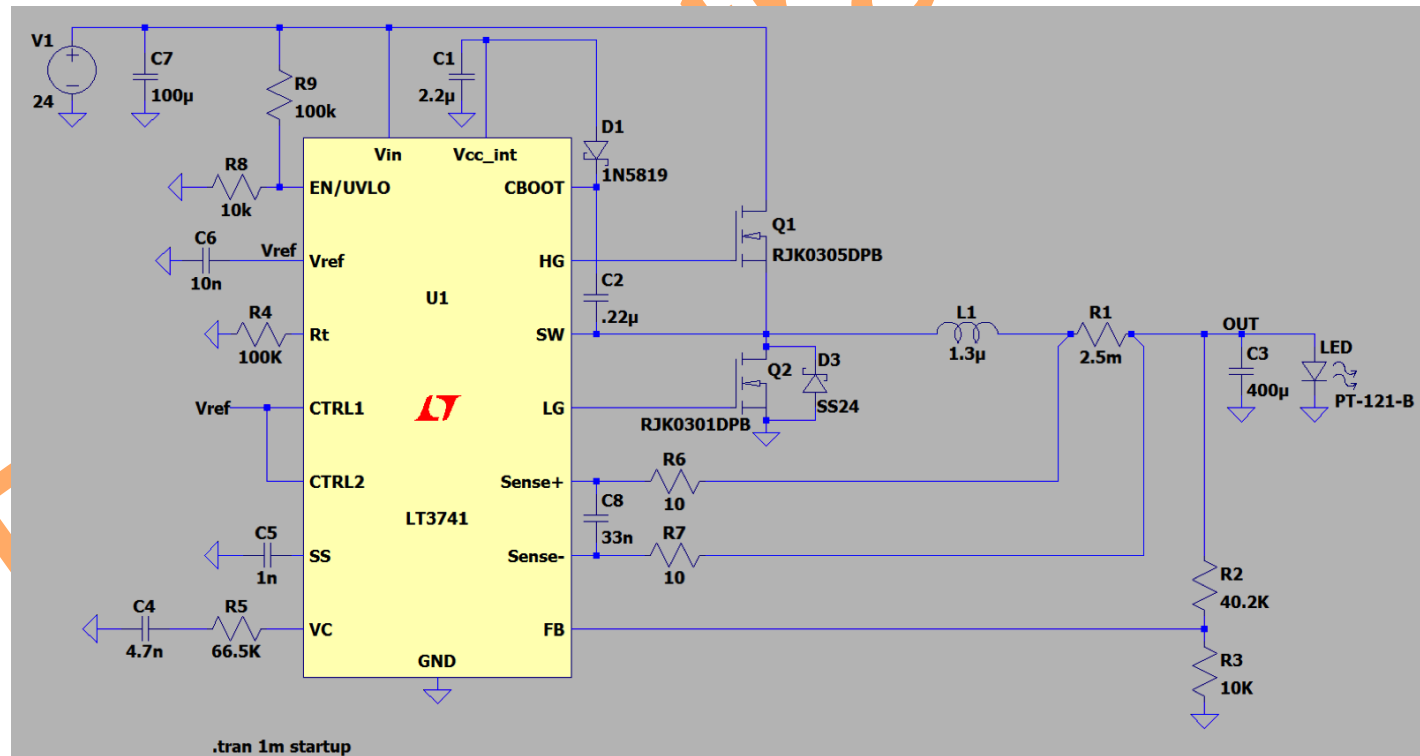
Including Model Libraries .INC and .LIB Statements

- .LIB directive will only add models (.model and .subckt) from the specified file
  - Other directives within the file, for example a .IC statement, would be ignored
- The .INC directive will include everything in the specified file into your project, regardless if it is a model, directive or anything else

.LIB is safer to use when it comes to importing models

ANALOG
DEVICES

AHEAD OF WHAT'S POSSIBLE™

Creating Schematic Symbols

LTspice
Free – Fast – Unlimited

A Symbol Can Be Automatically Generated In Two Situations

► Subcircuit models
  ▪ When editing an ASCII netlist that contains subcircuit definitions, you place the cursor on the line containing the name of the subcircuit, right click, and execute context menu item "Create Symbol"
  ▪ Saved under ..\Lib\Sym\AutoGenerated

► Hierarchical schematic
  ▪ When editing a schematic, you can execute menu item Hierarchy➔Open this Sheet's Symbol
  ▪ When no symbol is found, LTspice will ask if you would like one automatically generated
  ▪ Saved under the "working" folder

**ANALOG
DEVICES**
AHEAD OF WHAT'S POSSIBLE™

Automatic Symbol Creation - Subcircuits

1. Open SPICE .subckt file with LTspice

2. Right-click on .subckt line

3. Create symbol

4. Subcircuit symbol automatically saved

Saved under ..\Lib\Sym\AutoGenerated

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE™

Automatic Symbol Creation - Subcircuits

► The symbol can be placed on a schematic via the "Select Component Symbol" window.

Located under ..\Lib\Sym\AutoGenerated

1. Select Component Window

2. AutoGenerated folder

3. subckt symbol
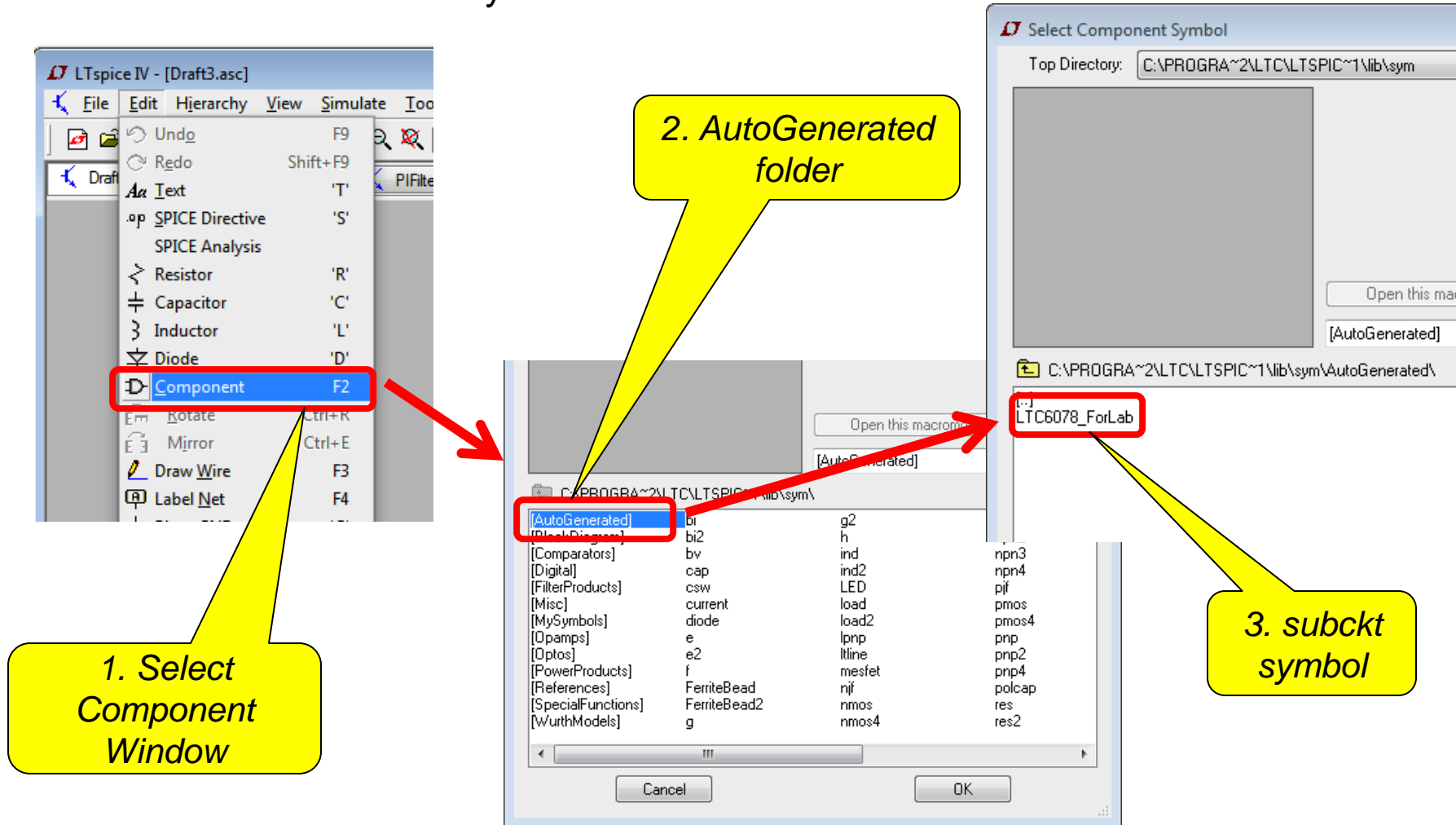
Automatic Symbol Creation - Subcircuits

► Create the symbol for the "LTC6078ForLab" model using the automatic symbol creation feature.

► The model file, "LTC6078ModelForLab.txt"

► Once generated, use the "LTC6078TestCircuit.asc" circuit to verify the newly created model.
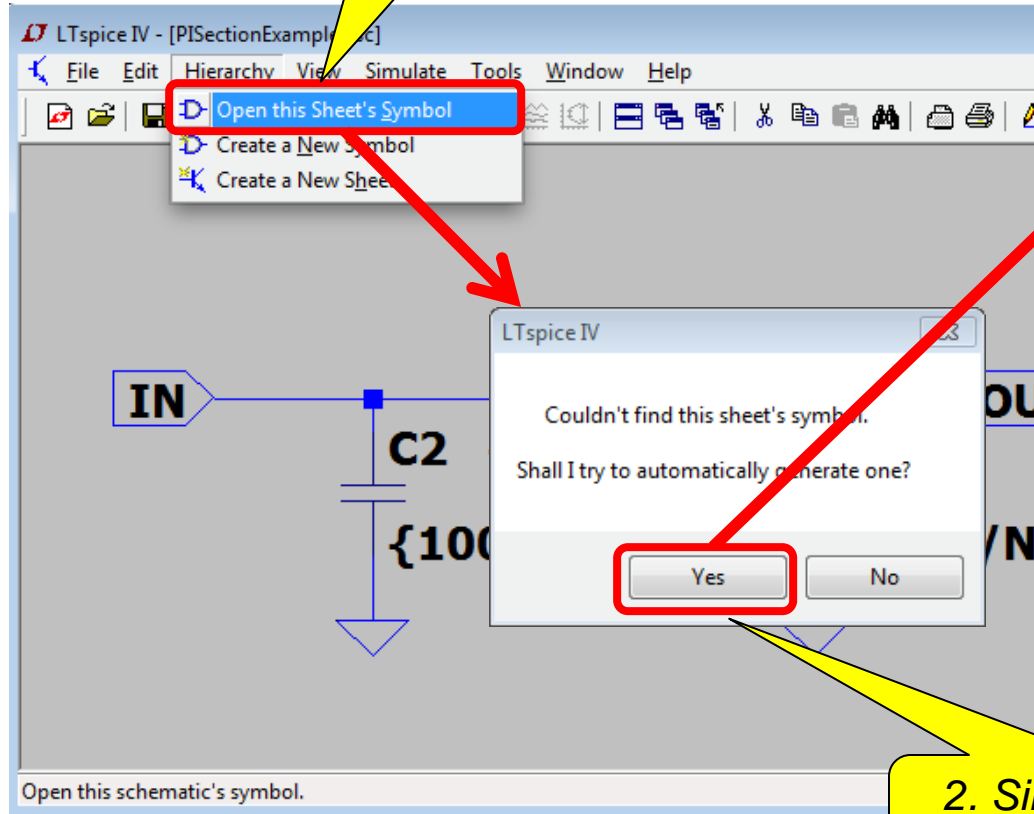
LTC6078ModelForLab.txt   LTC6078TestCircuit.asc
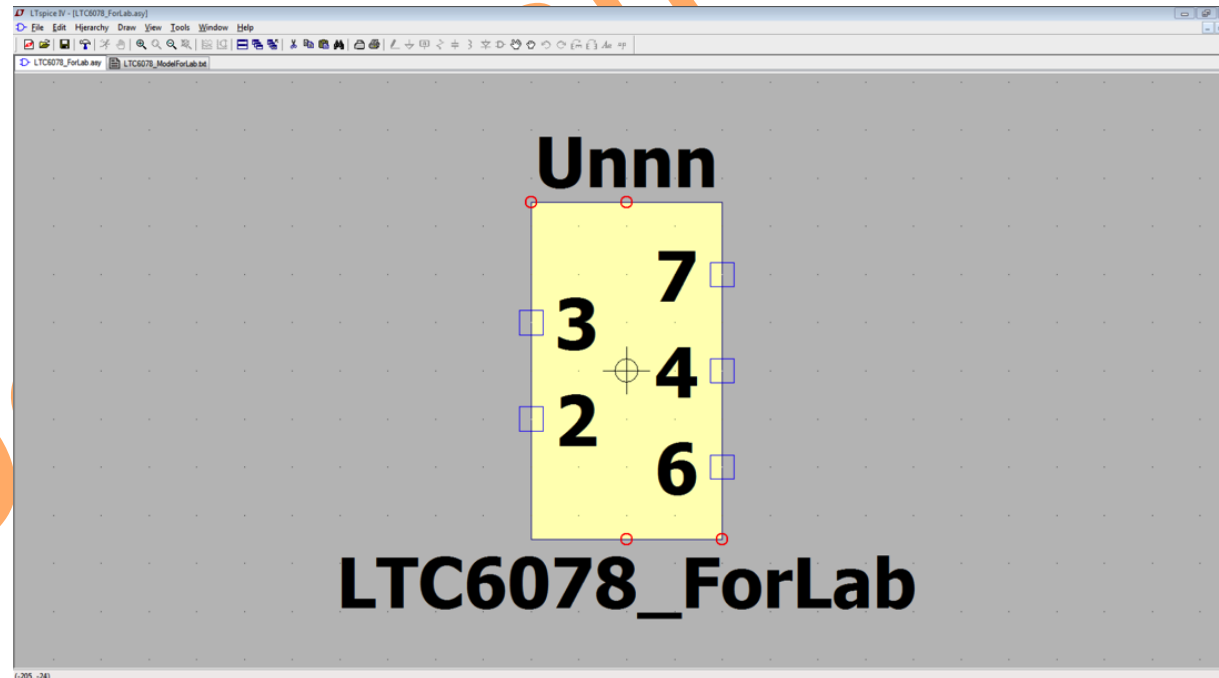
Hierarchical Schematics

► Encapsulate larger circuits can be drafted while retaining the clarity of the smaller schematics

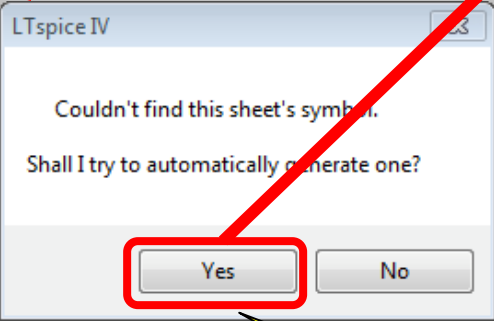► Abstraction allows schematics to be handled in so that is can be used across several schematics
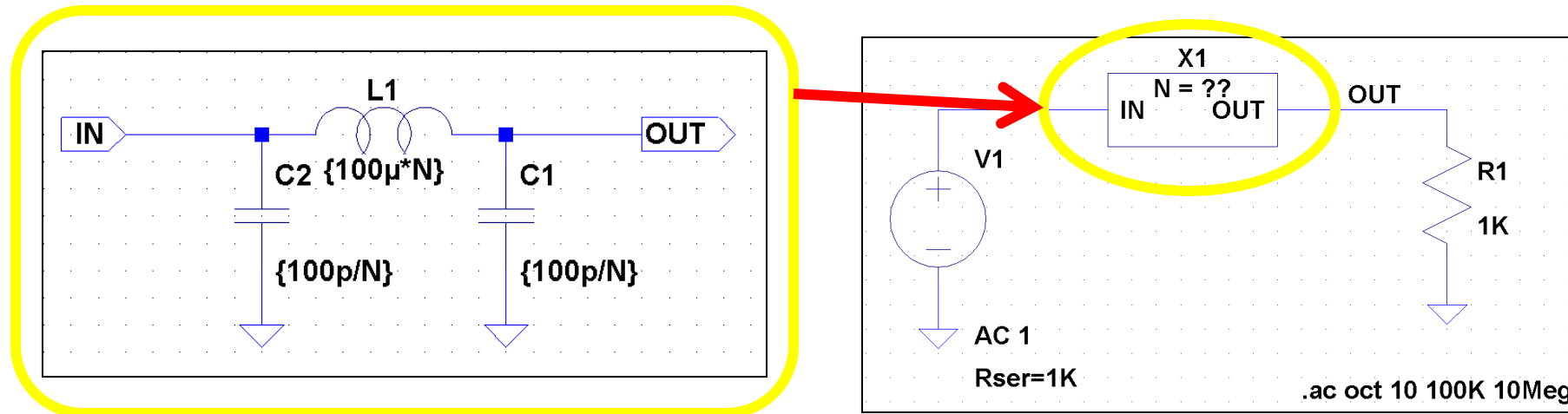
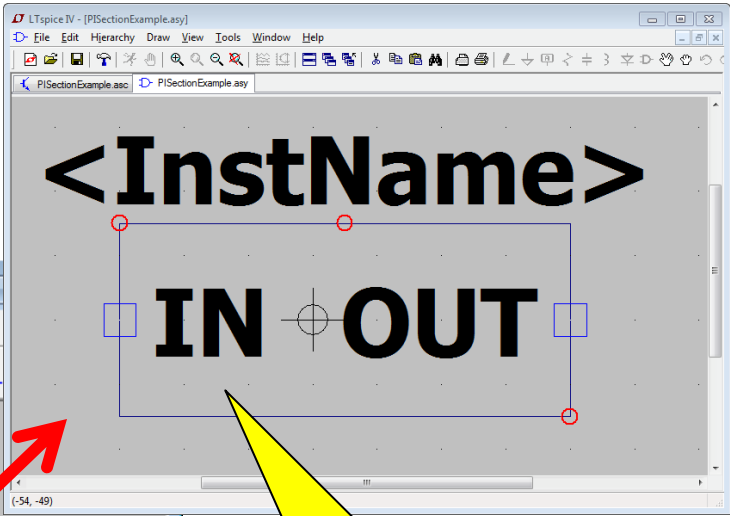# Automatic Symbol Creation - Hierarchical

1. Hierarchy ➔ Open this Sheet's Symbol

2. Since the symbol doesn't exist…

3. Hierarchical symbol automatically saved to the working folder

Automatic Symbol Creation – Hierarchical symbols

➤ To place a hierarchical block on a schematic:

2. *Change Top Directory*

1. *Edit ➔ Component*

3. *Select and place*

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE™
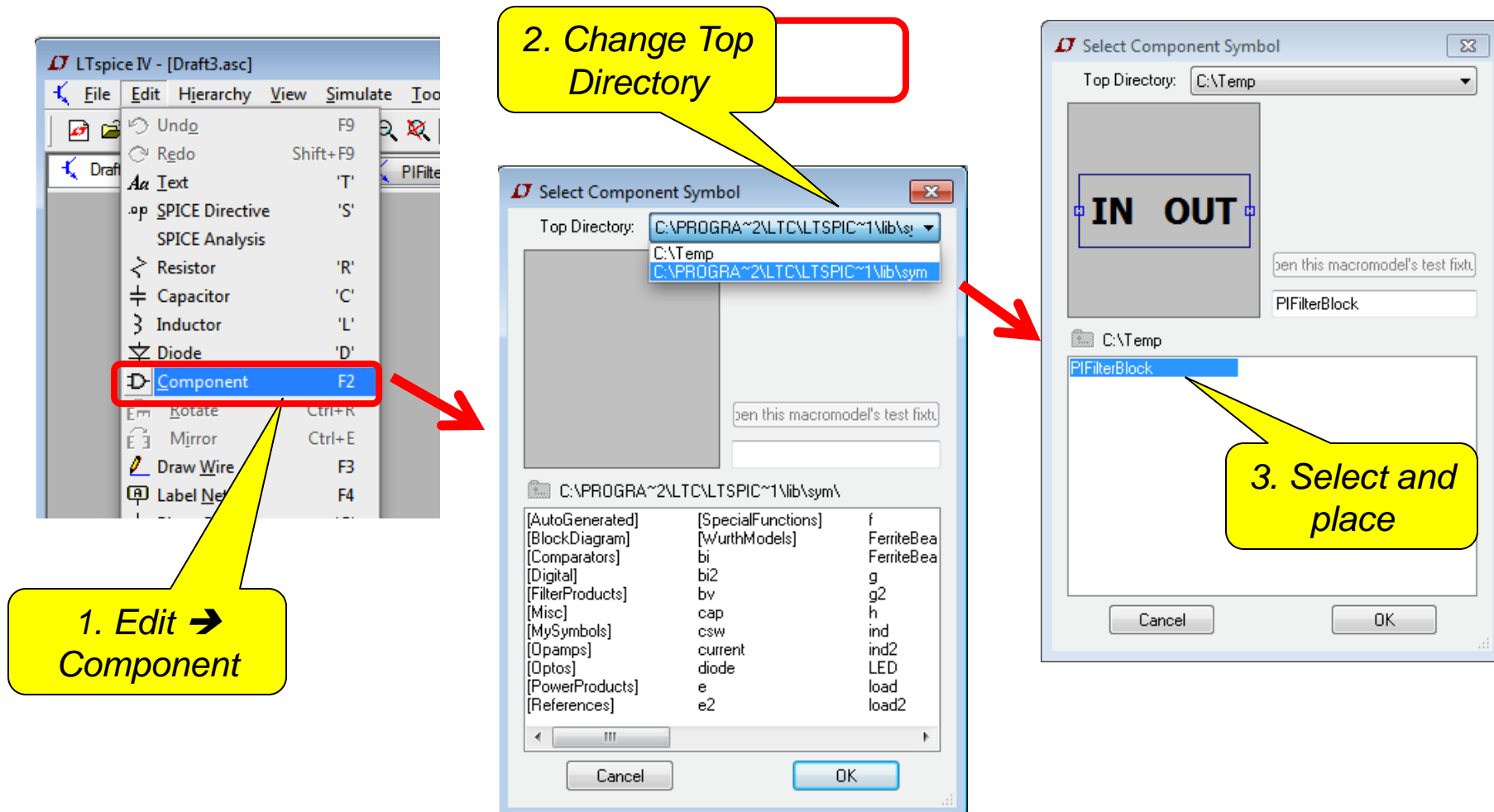
► Using the automatic symbol creation feature, create a hierarchical symbol for the circuit "PIFilterBlock.asc".

► Once the symbol is created, verify its functionality by using the test jig "PIFilterTestCircuit.asc".
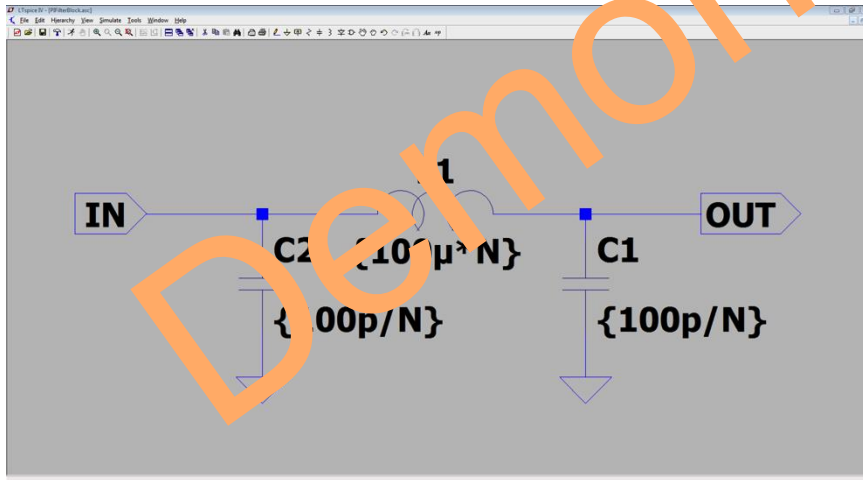
PIFilterBlock.asc
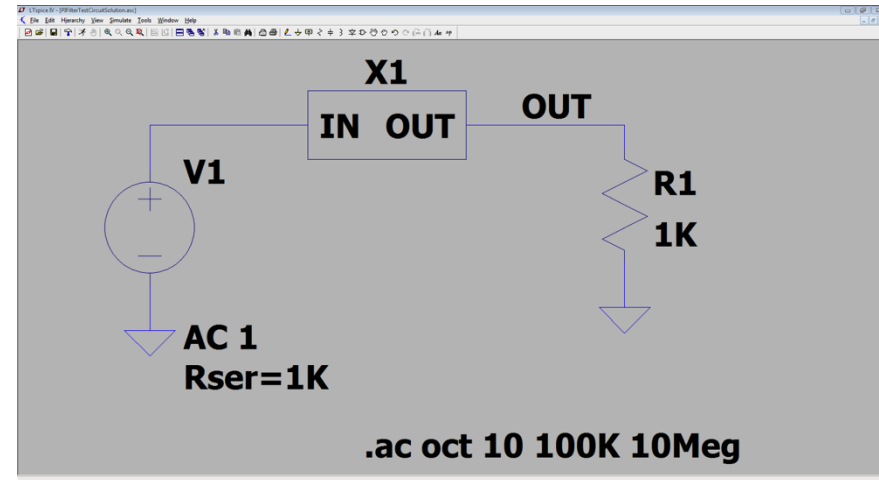
PIFilterTestCircuit

Creating/Editing schematic symbol

► Editing the symbols can significantly improve its ease-of-use

  ▪ The automatically generated symbols might not be effective and easy to wire-up

► To launch the symbol editor:

  ▪ Ctrl+RightClick on the symbol then "Open Symbol"

  ▪ File pull-down menu ➔ New symbol,

  ▪ Open up a known symbol (.asy). File menu ➔ Open

ANALOG
DEVICES
AHEAD OF WHAT'S POSSIBLE™

Symbol Editing – Drafting and Adding Pins

- ► Drafting [Draw menu]:
  - ▪ Use Draw to sketch the symbol

- ► Add Pin/Port [Edit menu]:
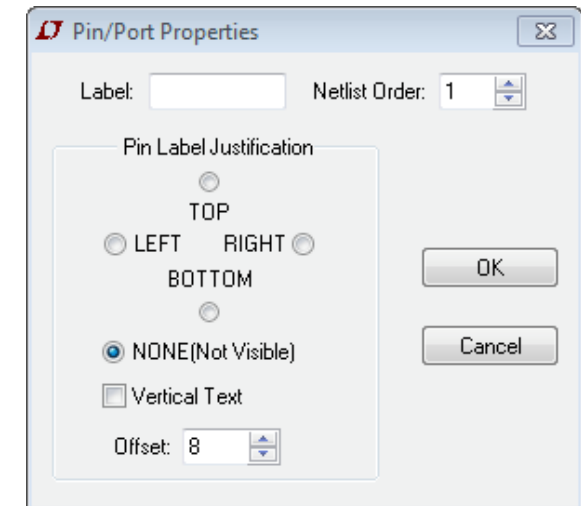 - ▪ The "Pin Label Position" determines how the pin label is presented (text justification).
  - ▪ Label: If the symbol represents …
    - ▪ a SPICE primitive element or a subcircuit from a library, then the pin label has no direct electrical impact on the circuit.
    - ▪ the lower-level schematic of a hierarchical schematic, then the pin name is significant as the name of a net in the lower level schematic.
  - ▪ The "Netlist Order" determines the order this pin is netlisted for SPICE

Pin/Port Properties dialog: Label, Netlist Order: 1. Pin Label Justification — TOP, LEFT, RIGHT, BOTTOM, NONE(Not Visible), Vertical Text, Offset: 8. OK / Cancel.

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE™

Symbol Editing – Pins and Netlist order

► "Netlist Order" determines the order this pin is netlisted for SPICE

- It links a pin to a node of a SPICE model
- The left most node in the model definition is associated to the 1st position of the netlist order and increments as such until the right most node corresponds to the last netlist order position.

Netlist order

Netlist Order: 1 2 3 4 5

.SUBCKT LTC6078_ForLab 3 2 7 4 6
CCMRR 126 109 0.3p
CG1 117 0 1E-6
CG3 118 0 2.43E-5
CG5 122 0 1E-7
CTN2 4 3 9nF

Setting Netlist Order 3 to pin VCC links VCC to Node7 of the SPICE model

25
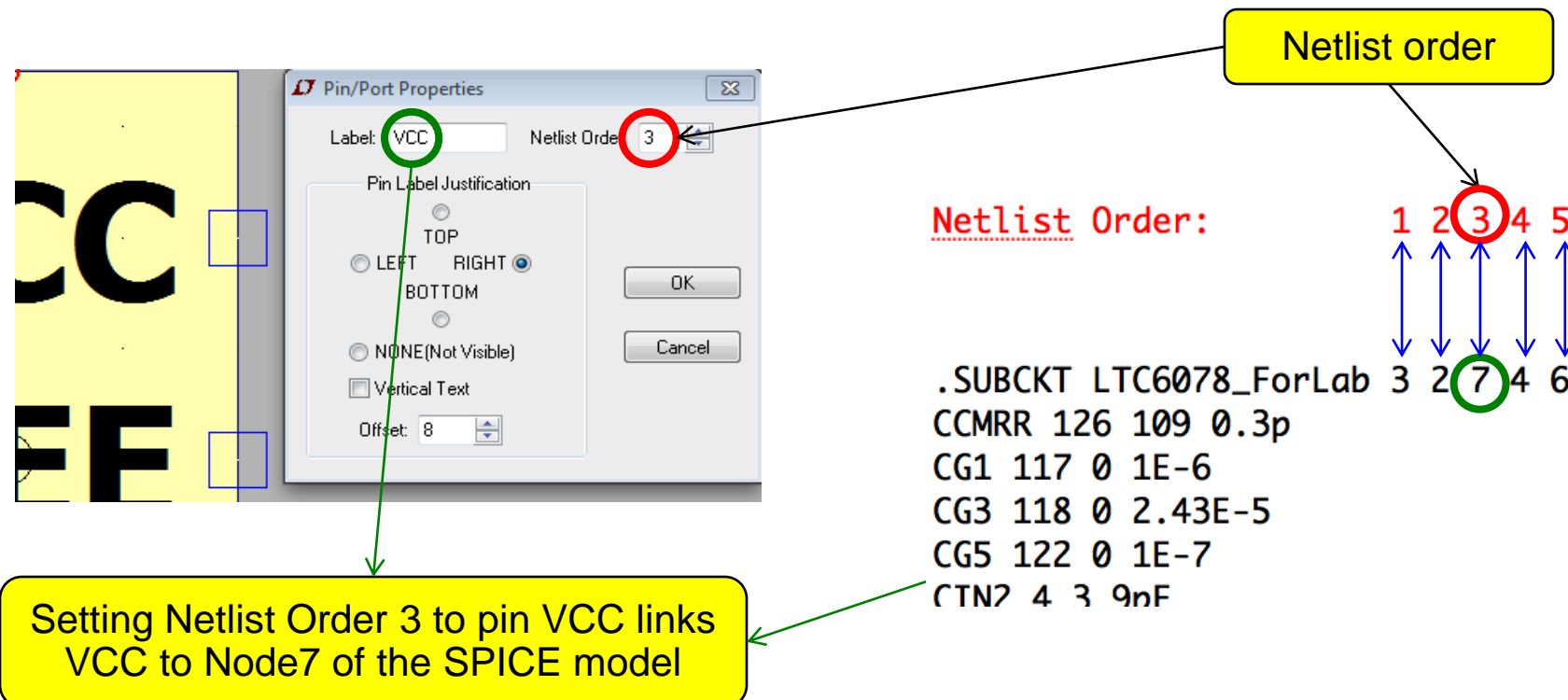
Managing Libraries

LTspice
Free – Fast – Unlimited

LTspice Standard Library Files

- RCL Libraries (easy to edit and expand)
  - standard.res
  - standard.cap
  - standard.ind
  - standard.bead

- Intrinsic Device Libraries (more complicated to edit and expand)
  - standard.dio
  - standard.bjt
  - standard.mos
  - standard.jft

File path for standard library files:
C:\Program Files\LTC\LTspiceXVII\lib\cmp

Custom entries into the standard library files will not be removed by a Sync Release.
See caveat on next page.

ANALOG
DEVICES
AHEAD OF WHAT'S POSSIBLE™

# Managing Standard Library Files

► Open the library file you want to edit using LTspice

► Custom entries into the standard library files will not be removed by a Sync Release but…

► A fresh re-installation of LTspice will delete the entire content of the standard library files

  ▪ Not recommended unless you archive your installation

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE™

Managing Resistor, Inductor, and Capacitor Database Files

To add an entry to an RLC library file:

- Scroll to the end of the library file
- Double-click right below the last row of data (empty row)
- Fill-in the fields in the "Database record" pop-up window
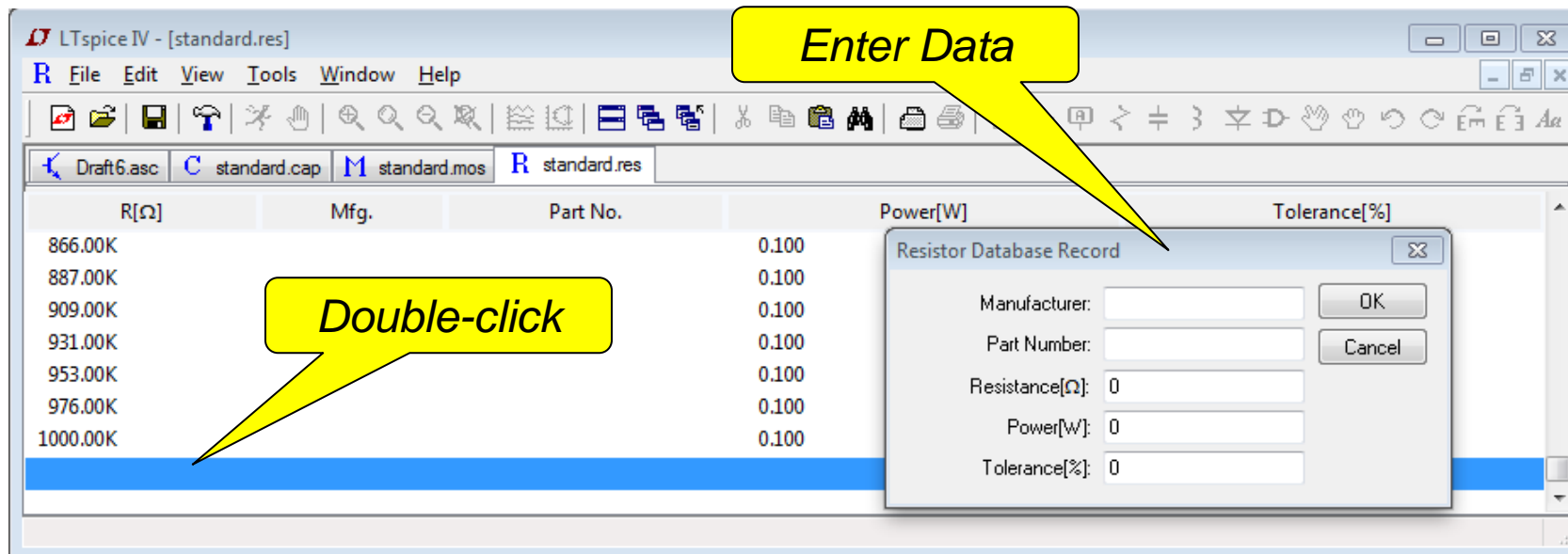
Managing RLC Database Files

► To add multiple entries using a spreadsheet:

Enter data in spreadsheet.
Column sequence is important

Copy and paste data to standard library file

This technique works well with data from parametric tables or datasheet tables.

Managing RLC Libraries

1.) Add a new tantalum capacitor which has the following characteristics to the standard.cap library:

- Manufacturer: Dummy Inc.
- Part number: CNEW
- Capacitance: 133uF
- Voltage: 1500V
- RMS Current: 3A
- ESR: 2mOhm
- Parasitic inductance: 0.1nH

2.) Place CNEW on a LTspice schematic and verify that its parameters match the ones listed above.

Capacitor - C1

Manufacturer: Dummy Inc
Part Number: CNEW
Type: Tantalum

Select Capacitor

Capacitor Properties

Capacitance[F]:	133μ
Voltage Rating[V]:	1.5K
RMS Current Rating[A]:	3
Equiv. Series Resistance[Ω]:	0.002
Equiv. Series Inductance[H]:	100p
Equiv. Parallel Resistance[Ω]:	
Equiv. Parallel Capacitance[F]:	
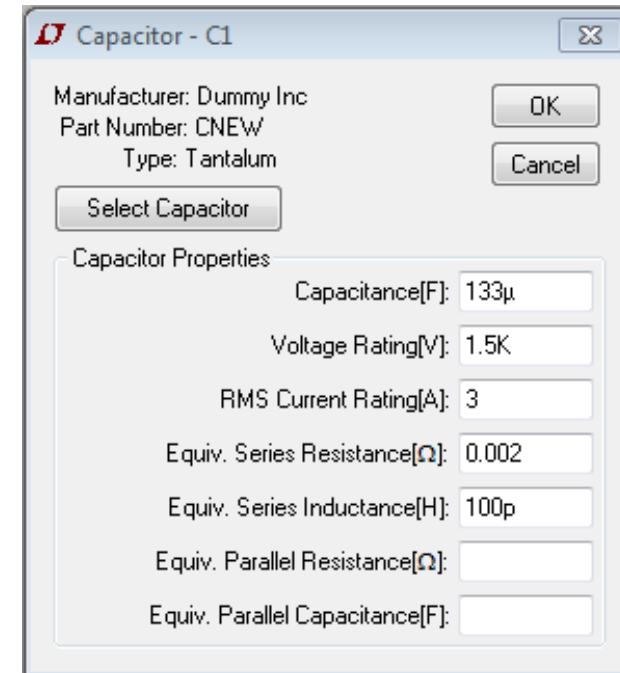
Adding a Device To An Intrinsic Device Library File (Eg Diode, Transistor, FET)

1.) Download or create a .model statement

.SUBCKT is not a valid model to add to an LTspice Intrinsic device library

If the .model statement is not available for that .SUBCKT model, then it must be created.

2.) Copy the .model statement

The syntax must be:

.model <modelname> <type> (<parameter list>)

where <type> can be:

- D: Diode
- NPN, PNP: Bipolar Transistors
- NJF, PJF: N and P-Channel JFET transistors
- VDMOS: Vertical Double Diffused Power MOSFET
- Other MOS types are available but not as widely used in the standard.mos library (refer to LTspiceHelp for additional details)

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE™

Managing Intrinsic Device Libraries

To add a device to an Intrinsic Device library file (cont.):

In LTspice open up the library file, scroll to the end of the library file, and paste the copied .model statement into an empty row.

(Optional) Annotate the .model statement with part ratings & manufacturer specifics. This information is displayed in the schematic capture GUI to assist in selecting a device but does not directly impact the electrical behavior in simulation.

- Available part ratings / mfg information to add are:
  - Diode: Vpk= [V], Iave= [A], mfg= [name], type= [name]
  - MOS: Vds= [V], Ron= [ohm], Qg= [C], mfg = [name]
  - Bipolar: Vceo= [V], Icrating= [A], mfg= [name]
  - JFET: mfg= [name]
- Annotated model example:
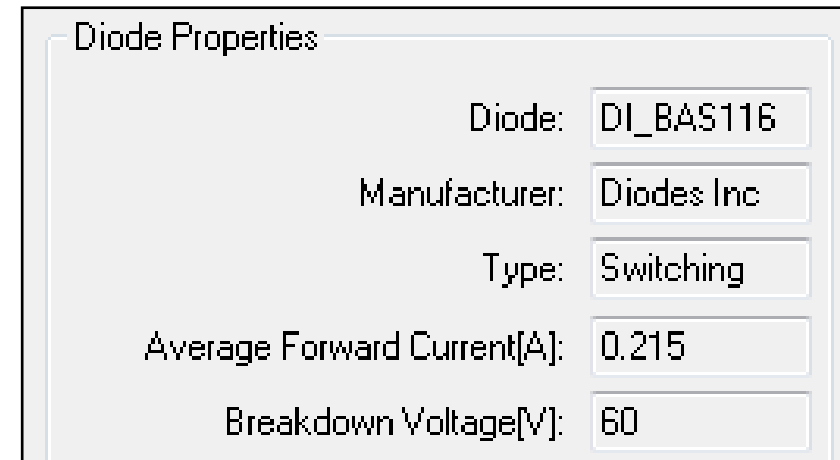
.model BZX84C12L D(Is=.6n Rs=.5 Cjo=150p nbv=5 bv=12 Ibv=1m Vpk=12 mfg=OnSemi type=Zener)

<modelname> <type>   *<electrical parameters>*   *<GUI information>*

33

Add Diodes Inc's BAS116 diode model to the standard.dio library

1) The model can be copied from the file DI_BAS116.txt

2) Add the manufacturer information and part ratings that will be displayed in the schematic capture GUI

3) Open "DiodeLibraryExample.asc" and follow the steps listed to verify the newly added model

4) Verify that the "Diode Properties" window looks like this:

Diode Properties	
Diode:	DI_BAS116
Manufacturer:	Diodes Inc
Type:	Switching
Average Forward Current[A]:	0.215
Breakdown Voltage[V]:	60

Quick Review of Additional Resources and Support

LTspice

Free – Fast – Unlimited

LTspice Help File (F1) and Independent LTwiki.org

LTspiceHelp

Hide   Back   Print   Options

Contents | Index | Search

Type in the keyword to find:

mod

List Topics

Select Topic to display:

.LIB – Include a library
startup

Display

.LIB -- Include a Library

Syntax: .lib <filename>

This directive includes the model and subcircuit definitions of the named file as if that file had been typed into the netlist instead of the .lib command. Circuit elements at global scope are ignored.

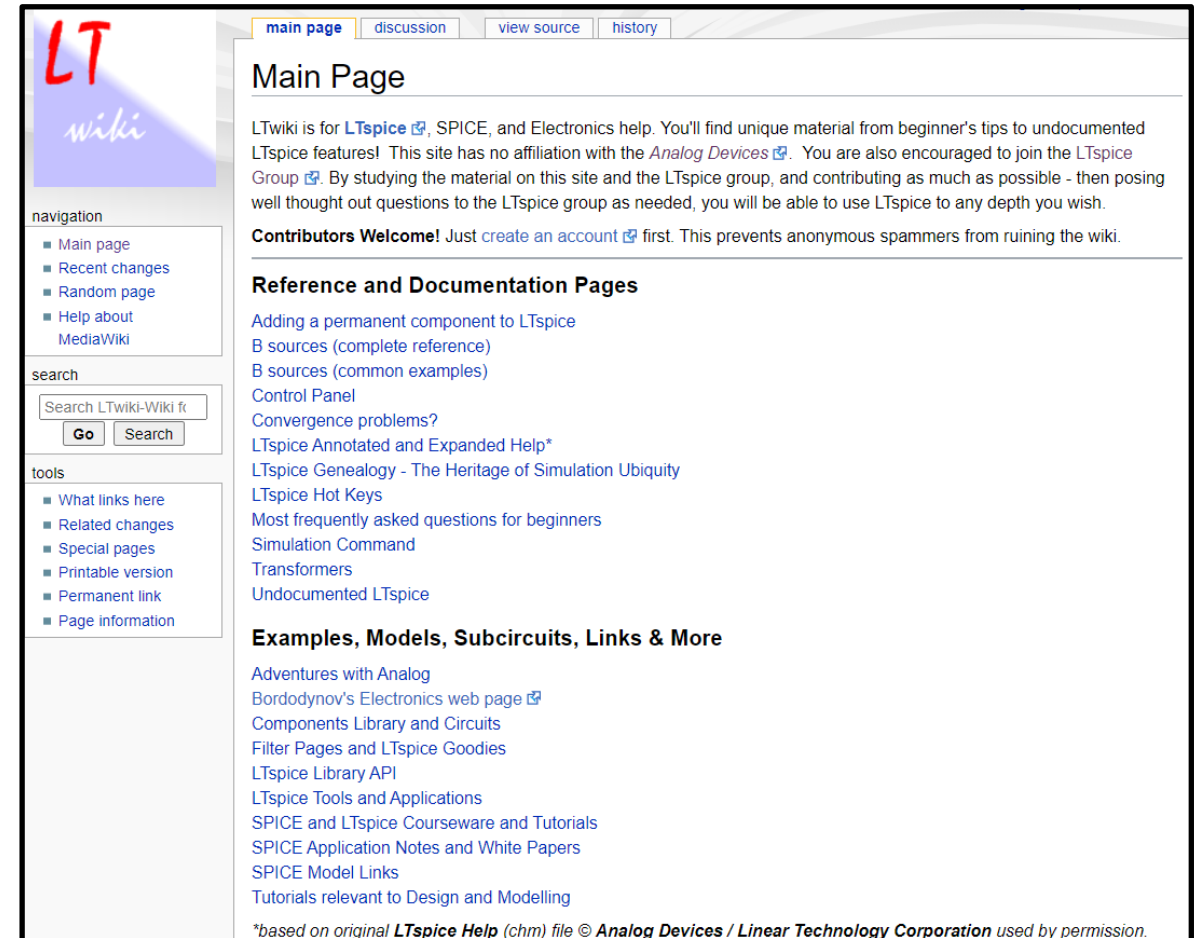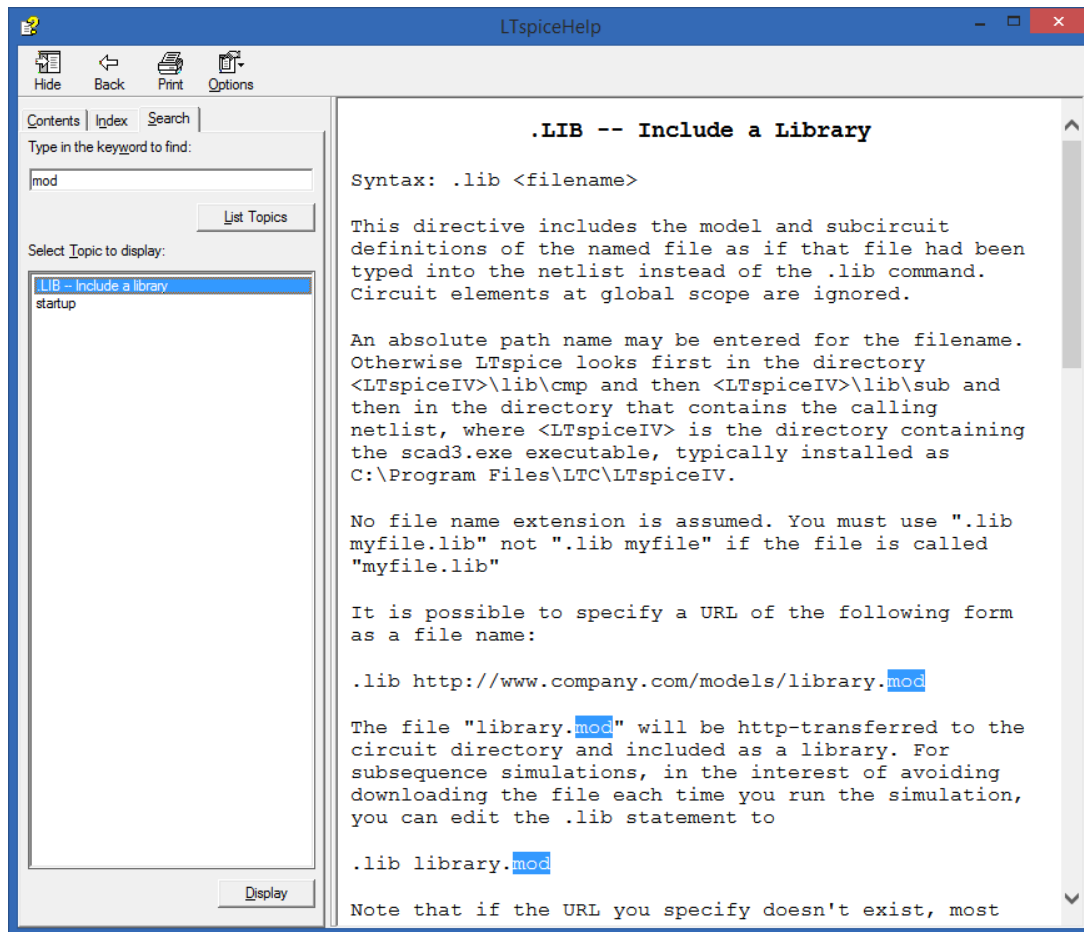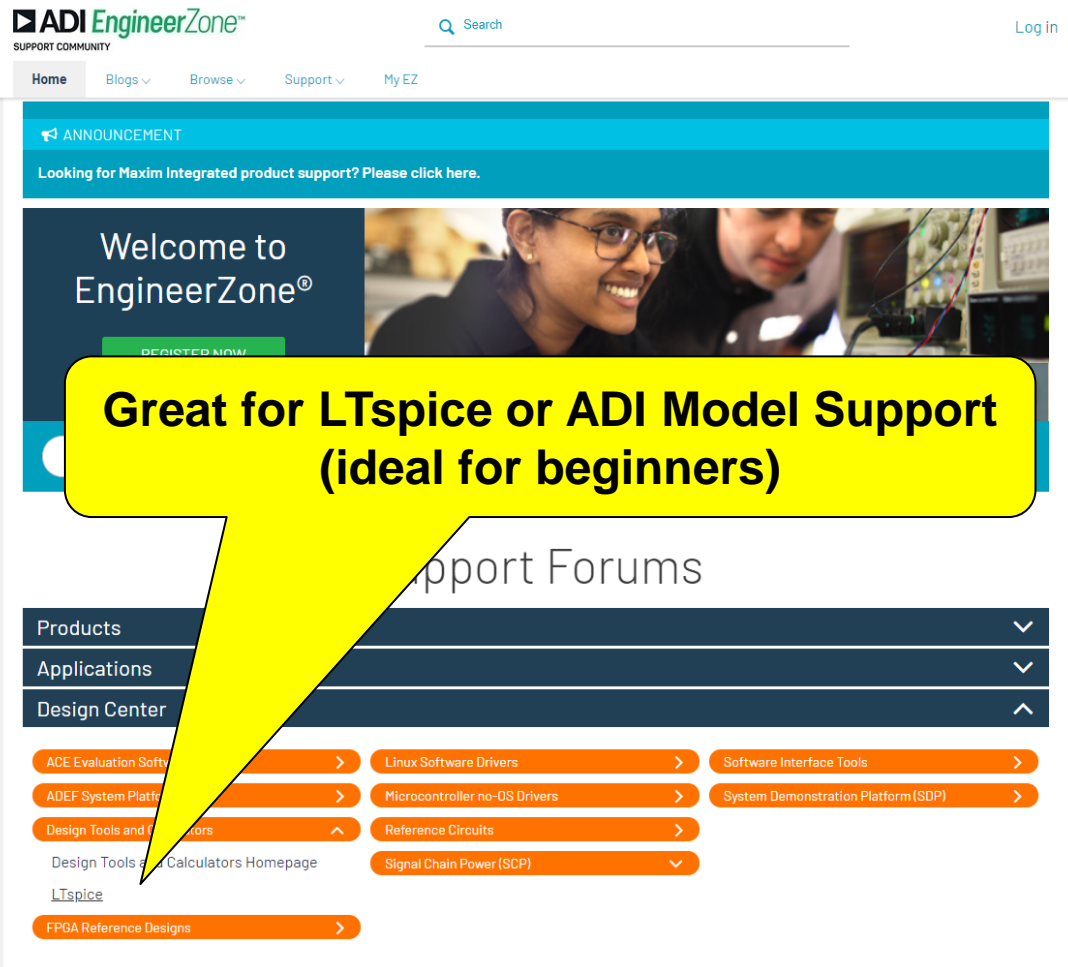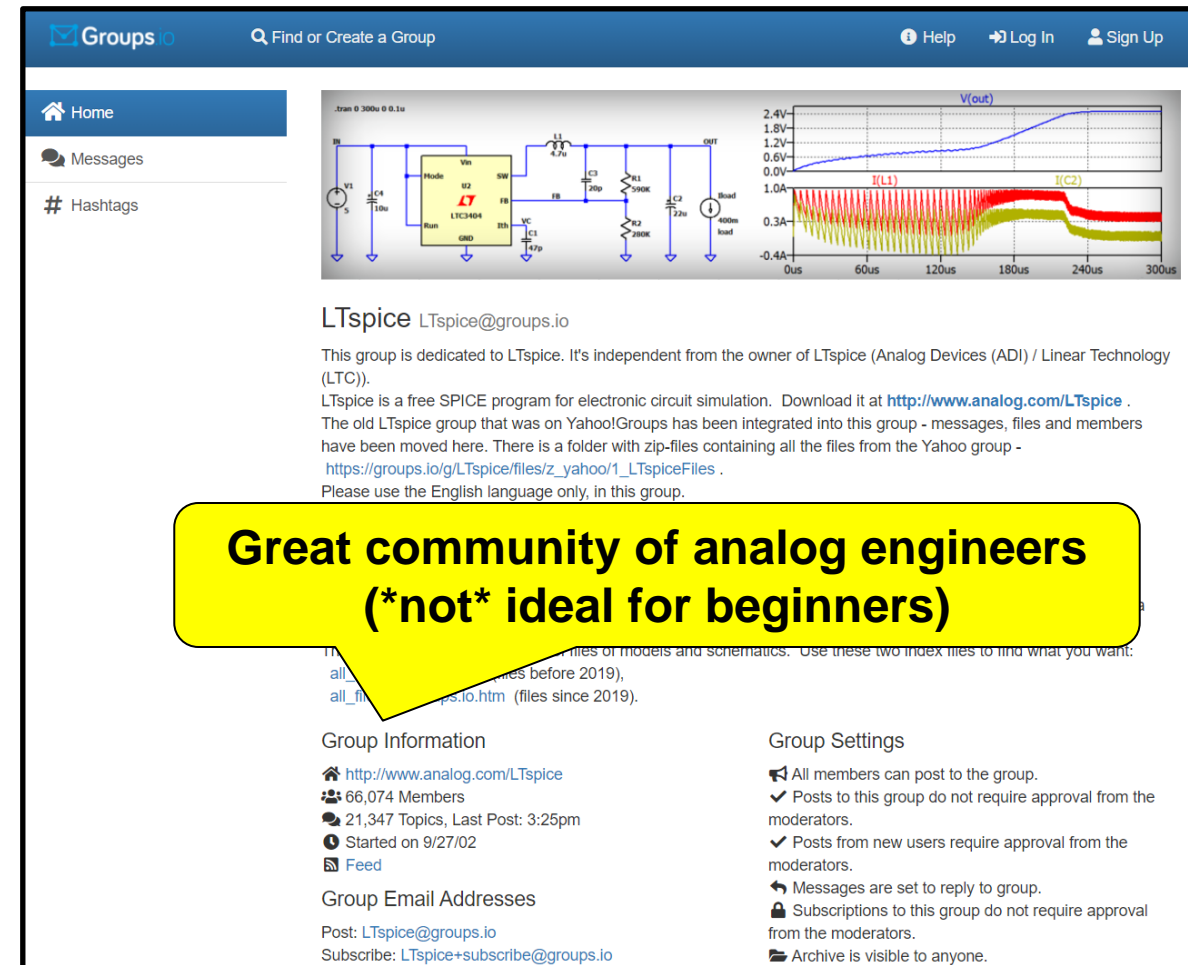An absolute path name may be entered for the filename. Otherwise LTspice looks first in the directory <LTspiceIV>\lib\cmp and then <LTspiceIV>\lib\sub and then in the directory that contains the calling netlist, where <LTspiceIV> is the directory containing the scad3.exe executable, typically installed as C:\Program Files\LTC\LTspiceIV.

No file name extension is assumed. You must use ".lib myfile.lib" not ".lib myfile" if the file is called "myfile.lib"

It is possible to specify a URL of the following form as a file name:

.lib http://www.company.com/models/library.mod

The file "library.mod" will be http-transferred to the circuit directory and included as a library. For subsequence simulations, in the interest of avoiding downloading the file each time you run the simulation, you can edit the .lib statement to

.lib library.mod

Note that if the URL you specify doesn't exist, most

---

main page | discussion | view source | history

Main Page

LTwiki is for **LTspice**, SPICE, and Electronics help. You'll find unique material from beginner's tips to undocumented LTspice features! This site has no affiliation with the *Analog Devices*. You are also encouraged to join the LTspice Group. By studying the material on this site and the LTspice group, and contributing as much as possible - then posing well thought out questions to the LTspice group as needed, you will be able to use LTspice to any depth you wish.

Contributors Welcome! Just create an account first. This prevents anonymous spammers from ruining the wiki.

Reference and Documentation Pages

Adding a permanent component to LTspice
B sources (complete reference)
B sources (common examples)
Control Panel
Convergence problems?
LTspice Annotated and Expanded Help*
LTspice Genealogy - The Heritage of Simulation Ubiquity
LTspice Hot Keys
Most frequently asked questions for beginners
Simulation Command
Transformers
Undocumented LTspice

Examples, Models, Subcircuits, Links & More

Adventures with Analog
Bordodynov's Electronics web page
Components Library and Circuits
Filter Pages and LTspice Goodies
LTspice Library API
LTspice Tools and Applications
SPICE and LTspice Courseware and Tutorials
SPICE Application Notes and White Papers
SPICE Model Links
Tutorials relevant to Design and Modelling

*based on original **LTspice Help** (chm) file © **Analog Devices / Linear Technology Corporation** used by permission.

navigation
- Main page
- Recent changes
- Random page
- Help about MediaWiki

search

Search LTwiki-Wiki fo

Go   Search

tools
- What links here
- Related changes
- Special pages
- Printable version
- Permanent link
- Page information

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE™

EngineerZone LTspice Forum and Independent User Group

https://ez.analog.com/

groups.io/g/LTspice

Great for LTspice or ADI Model Support (ideal for beginners)

Great community of analog engineers (*not* ideal for beginners)

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE™

www.analog.com/LTspice

Documentation

Additional support for LTspice can be found within our documentation, including keyboard shortcuts and a visual guide.

LTspice Information Flyer & Shortcuts (PDF)
Mac OS X Shortcuts (PDF)
Get Up and Running with LTspice

LTspice Technical Articles & Videos

Our extensive collection of technical resources tackles a wide range of LTspice topics, like keyboard shortcuts, evaluating electrical quantities, and parametric plots.

View our Technical Articles

LTspice Getting Started Tutorial

LTspice Essentials Tutorial

LTspice AC & Noise Analysis Tutorial

LTspice HotKeys

	Schematic	Symbol	Waveform	Netlist
Modes	ESC – Exit Mode	ESC – Exit Mode		
	F3 – Draw Wire			
	F5 – Delete	F5 – Delete	F5 – Delete	
	F6 – Duplicate	F6 – Duplicate		
	F7 – Move	F7 – Move		
	F8 – Drag	F8 – Drag		
	F9 – Undo	F9 – Undo	F9 – Undo	F9 – Undo
	Shift+F9 – Redo	Shift+F9 – Redo	Shift+F9 – Redo	Shift+F9 – Redo
View	Ctrl+Z – Zoom Area	Ctrl+Z – Zoom Area	Ctrl+Z – Zoom Area	
	Ctrl+B – Zoom Back	Ctrl+B – Zoom Back	Ctrl+B – Zoom Back	
	Space – Zoom Fit		Ctrl+E – Zoom Extents	
	Ctrl+G – Toggle Grid		Ctrl+G – Toggle Grid	Ctrl+G – Goto Line #
	U – Mark Uncon. Pins	Ctrl+W – Attribute Window	'0' – Clear	
	A – Mark Text Anchors	Ctrl+A – Attribute Editor	Ctrl+A – Add Trace	
	Atl+Click – Power		Ctrl+Y – Vertical Autorange	Ctrl+R – Run Simulation
	Ctrl+Click – Attr. Edit		Ctrl+Click - Average	
	Ctrl+H – Halt Simulation		Ctrl+H – Halt Simulation	Ctrl+H – Halt Simulation
Place	R – Resistor	R – Rectangle		
	C – Capacitor	C – Circle		
	L – Inductor	L – Line		
	D – Diode	A – Arc		
	G – GND			
	S – Spice Directive			
	T – Text	T – Text		
	F2 – Component			
	F4 – Label Net			
	Ctrl+E – Mirror	Ctrl+E – Mirror		
	Ctrl+R – Rotate	Ctrl+R – Rotate		

Command Line Switches

Flag	Short Description
-ascii	Use ASCII .raw files. (Degrades performance!)
-b	Run in batch mode.
-big or -max	Start as a maximized window
-encrypt	Encrypt a model library
-FastAccess	Convert a binary .raw file to Fast Access Format
-netlist	Convert a schematic to a netlist
-nowine	Prevent use of WINE(Linux) workarounds
-PCBnetlist	Convert a schematic to a PCB netlist
-registry	Store user preferences in the registry
-Run	Start simulating the schematic on open
-SOI	Allow MOSFET's to have up to 7 nodes in subcircuit
-uninstall	Executes one step of the uninstallation process
-wine	Force use of WINE(Linux) workarounds

Simulator Directives – Dot Commands

Command	Short Description
.AC	Perform a Small Signal AC Analysis
.BACKANNO	Annotate Subcircuit Pin Names on Port Currents
.DC	Perform a DC Source Sweep Analysis
.END	End of Netlist
.ENDS	End of Subcircuit Definition
.FOUR	Compute a Fourier Component
.FUNC	User Defined Functions
.FERRET	Download a File Given the URL
.GLOBAL	Declare Global Nodes
.IC	Set Initial Conditions
.INCLUDE	Include another File
.LIB	Include a Library
.LOADBIAS	Load a Previously Solved DC Solution
.MEASURE	Evaluate User-Defined Electrical Quantities
.MODEL	Define a SPICE Model
.NET	Compute Network Parameters in a .AC Analysis
.NODESET	Supply Hints for Initial DC Solution
.NOISE	Perform a Noise Analysis
.OP	Find the DC Operating Point
.OPTIONS	Set Simulator Options
.PARAM	User-Defined Parameters
.SAVE	Limit the Quantity of Saved Data
.SAVEBIAS	Save Operating Point to Disk
.STEP	Parameter Sweeps
.SUBCKT	Define a Subcircuit
.TEMP	Temperature Sweeps
.TF	Find the DC Small-Signal Transfer Function
.TRAN	Do a Nonlinear Transient Analysis
.WAVE	Write Selected Nodes to a .WAV file

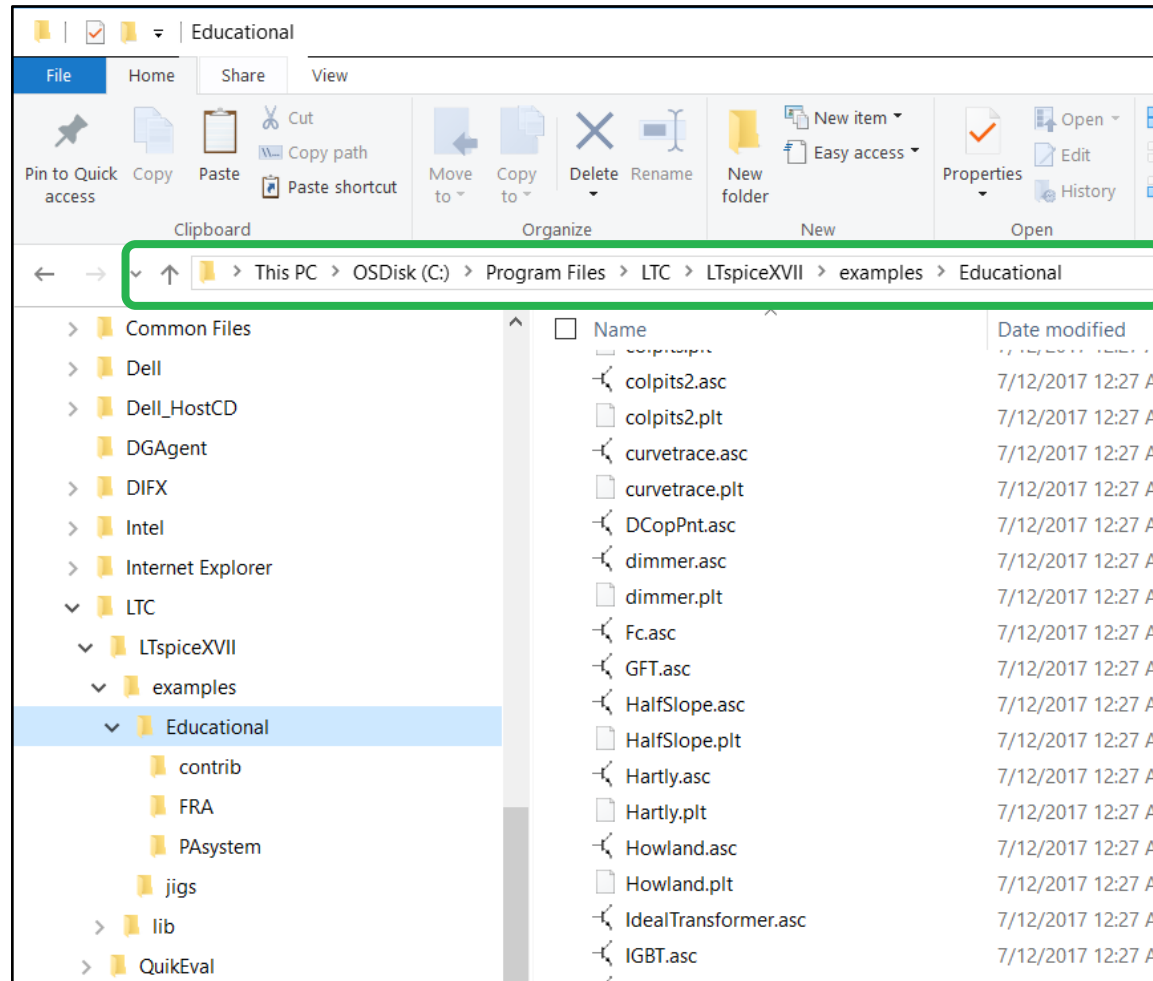| Suffix		Suffix		Constants	
		f	1e-15	E	2.718281828459045235|4
T	1e12	p	1e-12	Pi	3.141592653589793238|46
G	1e9	n	1e-9	K	1.3806503e-23
Meg	1e6	u	1e-6	Q	1.602176462e-19
K	1e3	M	1e-3	TRUE	1
		Mil	25.4e-6	FALSE	0

©2018 Analog Devices, Inc. All rights reserved. Trademark and registered trademark are the property of their respective owners. Ahead of What's Possible is a trademark of Analog Devices. (LTspice-0/18E)

analog.com

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE

LTspice

ANALOG DEVICES
AHEAD OF WHAT'S POSSIBLE™

LTspice Examples and Demo Circuits

www.analog.com/LTspice

LTspice Technical Support on Software Bugs and ADI Models

► LTspice@analog.com

- Please provide the following details and attach your LTspice schematic file (.asc), project file (.plt), and any non-standard model and symbol files (.asy, .lib, .txt, etc.)
  - Full Name:
  - Organization:
  - Phone Number:
  - Application:
  - Primary ADI Parts:
  - Detailed Description:

If you have an issue with a third-party model,
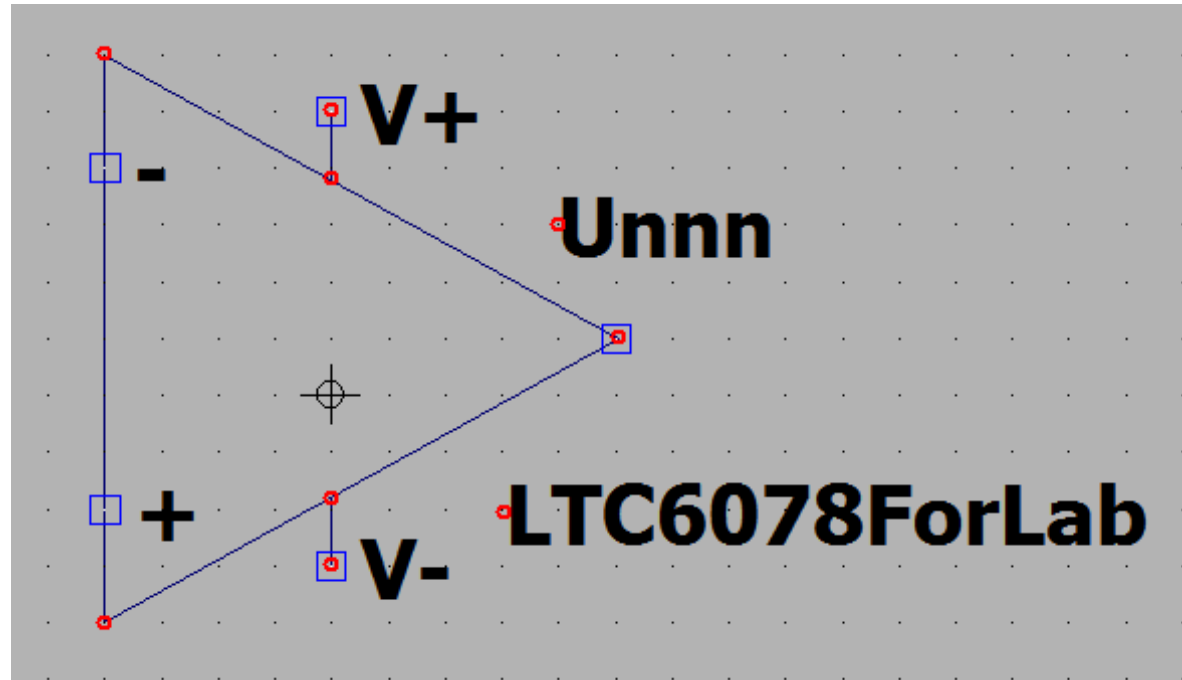please contact that vendor for support!

ANALOG
DEVICES
AHEAD OF WHAT'S POSSIBLE™

Homework

LTspice

Free – Fast – Unlimited

Homework

- Modify the previously generated LTC6078ForLab symbol so it looks like an op amp

- Once completed, reuse the "LTC6078TestCircuit.asc" circuit to verify the symbol is still functional

LTC6078ForLabSolution.asy   LTC6078TestCircuit.asc   LTC6078TestCirc