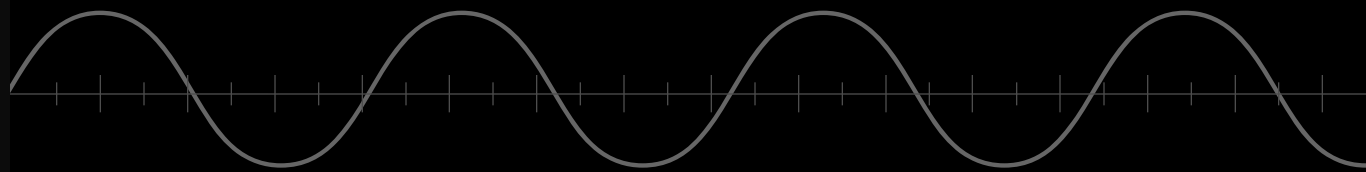


Importing Third-Party SPICE Models

Presented by Thomas Mosteller ADI FAE



Types of Third-Party SPICE models

There are two types of third-party SPICE models:

- ❖ **.MODEL:** For intrinsic SPICE devices like diodes and transistors.
 - ❖ The **.MODEL** statement gives the parameters for the specific component.
 - ❖ The behavior of the device is intrinsically understood by SPICE, only the parameters need to be given to finish specifying the component's electrical characteristics.
- ❖ **.SUBCKT:** Define the modeled component by a collection of circuitry of intrinsic SPICE devices.
 - ❖ Example: an opamp would be modeled as a subcircuit.

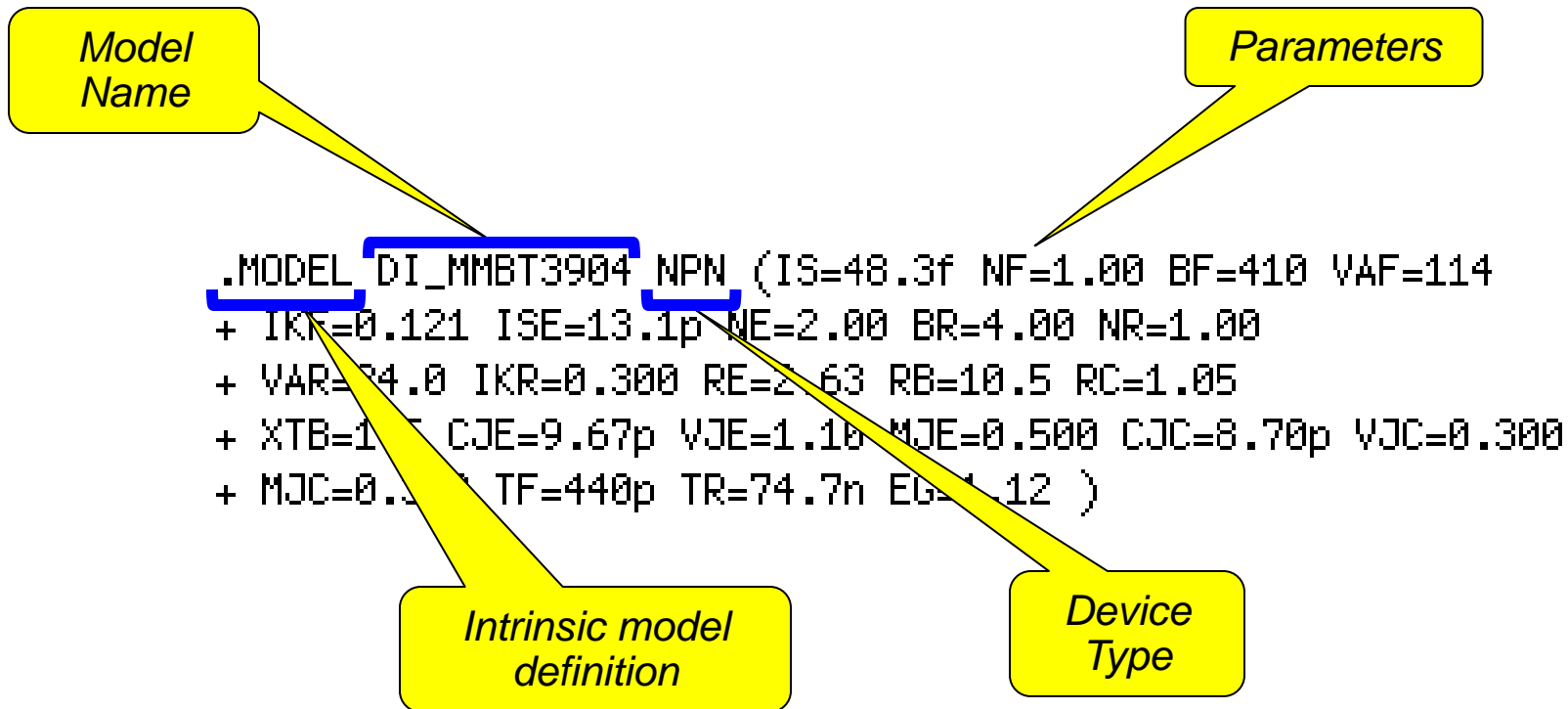
The method to import a model in LTspice depends on whether the model is given as a **.MODEL** statement or a **.SUBCKT**

Types of Third-Party SPICE models - Intrinsic

- ❖ The syntax of an intrinsic model is:

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

- ❖ Example of an intrinsic model:



Importing Third-Party Intrinsic Models

To import a third-party intrinsic spice model

- 1.) Download the spice model file from the manufacturer's website. Make sure to note the file location on the hard-drive.
- 2.) Add the following spice directive to the LTspice simulation file (Edit pull-down menu ---> SPICE Directive):

```
.include(1) [path (optional)] spicemodel_filename.abc
```

Important: The file name in the .include statement must match the spice model file name identically! The file name syntax can be anything, just make sure that all of the characters match.

Note: An absolute path name *<path>* can be omitted in the .include directive only if the file is located in:

- ❖ *<LTspiceIV>* \lib\sub
- ❖ The directory that contains the simulation file

(1) Where ever mentioned in this presentation, the .include could be substituted by .lib (refer to slide 7 for differences between the two spice directives).

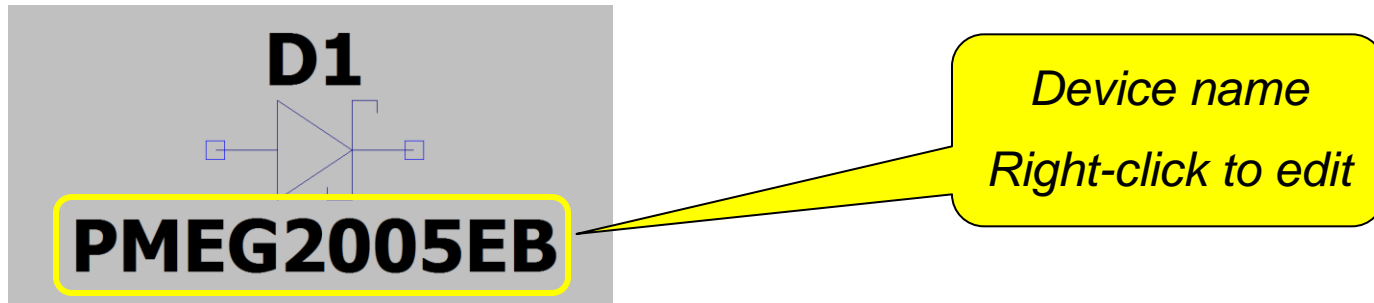
Importing Third-Party Intrinsic Models

To import a third-party intrinsic spice model (cont.)

3.) Open up the spice model file and note the device name

Important: The device name in the spice model file must match the device name in the LTspice schematic identically! The model name syntax can be anything, just make sure that all of the characters match !

4.) Modify the device name in the LTspice schematic to match the device name contained in the spice model file (Right-Click on the device name, and modify the text accordingly)



Third-Party Intrinsic Models – Model file

Spice Model Example #1:

Spice Model Example #2:

File Name = 1N5244B.mod

```

1N5244B Zener Diode
-----
.MODEL 1N5244B1
+ IS = 7.62E-10
+ RS = 0.3182
+ N = 1.69
+ XTI = 3.0
+ EG = 1.11
+ CJO = 4.582E-11
+ M = 0.3377
+ VJ = 2.983
+ FC = 0.5
+ ISR = 10E-21
  
```

Model name = 1N5244B1

File Name = Joe.txt

```

* 1N5244B Zener Diode
*
.MODEL Everest
+ IS = 7.62E-10
+ RS = 0.3182
+ N = 1.69
+ XTI = 3.0
+ EG = 1.11
+ CJO = 4.582E-11
+ M = 0.3377
+ VJ = 2.983
+ FC = 0.5
+ ISR = 10E-21
  
```

Model name = Everest

Summary: The file and model names are irrelevant. Just make sure the device name in the schematic and the file name within the .include statement match that of the spice model & model file.

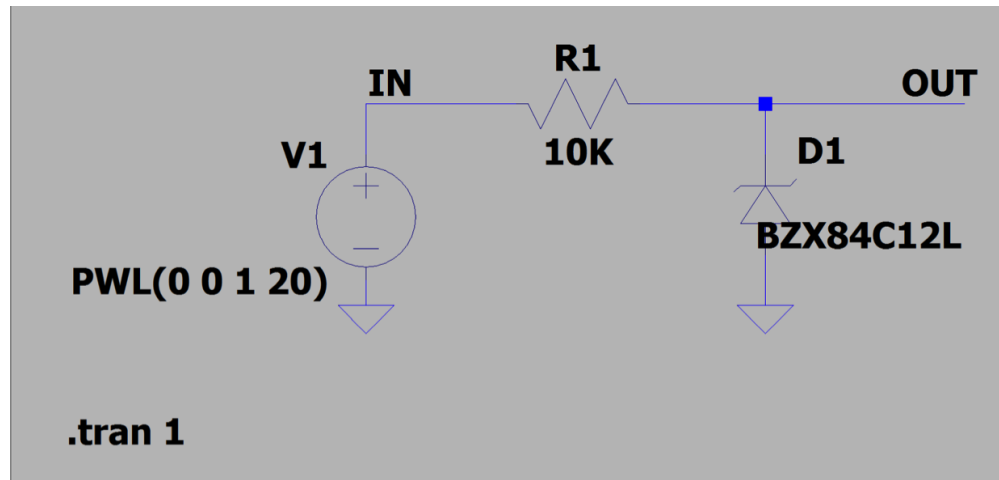
Including model libraries & Spice directives

- ❖ The `.include` / `.inc` and the `.lib` can both be used to include libraries. However, there's a difference between the two:
 - ❖ The `.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. In other words, `.LIB` is safer to use when it comes to importing models.
- ❖ The `.include spice` directive can be entirely omitted from the circuit schematic page if the schematic symbol was created so it automatically includes the library that contains the definition of the model.
 - ❖ Refer to the “*Creating Schematic Symbols*” module for more details.

Importing Third-Party Spice Models

Hands-on Exercise:

- ❖ Modify “ZenerImportExample.asc” so it uses the 1N5244B third-party SPICE model.
- ❖ The model’s file, [1N5244B.txt](#), is located in [<folder link>](#)



ZenerImportExample.asc

Tip: Do not forget to place the `.include` statement on the schematic after closing the text editor window

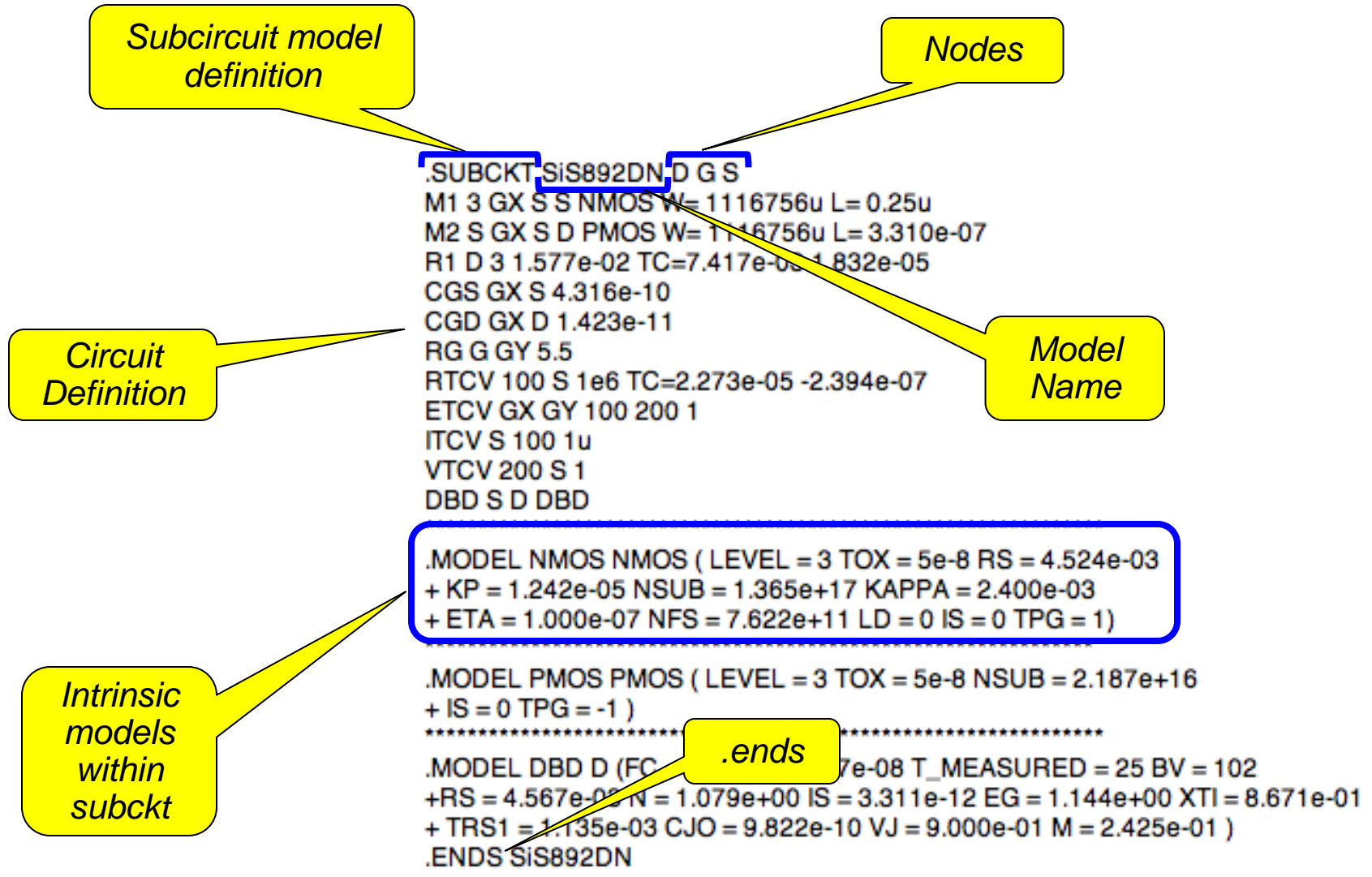
Types of Third-Party SPICE models - Subcircuits

- ❖ The syntax of a subcircuit model is:

```
.subckt <modelname> <nodelist>  
    <circuit made out of intrinsic devices>  
    <definition of intrinsic models>  
.ends
```

- ❖ The directive *.ends* marks the end of a subcircuit definition and must be present.
- ❖ An intrinsic model defined within a *.subckt* is accessible only within that *.subckt* model.
- ❖ Example (next slide) ..

Types of Third-Party SPICE models - Subcircuits



Importing Third-Party Subckt Models

To import a third-party Subckt spice model

- 1.) If the .subckt model doesn't have a schematic symbol that can be used, create a new one (refer to "Creating a schematic symbol" module). Otherwise, use one that's in the library.
- 2.) Verify the .subckt model is a good 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 and increments as such until the right most node corresponds to the last netlist order position.
- 3.) Repeat the steps to import a third-party intrinsic spice model

Importing Third-Party Subckt Models

To import a third-party Subckt spice model (cont.)

4.) Access the Component Attribute Editor window (Ctrl+Right-Click on the device) and change the device's prefix to "X"

CTRL+Right-Click

Change Prefix to "X"

Component Attribute Editor

Open Symbol: C:\Users\Raymond Vacha\Documents\Work\LTspice\lib\sy

This determines the type of a circuit element, e.g., R, C, or L

Attribute	Value	Vis.
Prefix	X	
InstName	Q2	X
SpiceModel		
Value	RJK0301DPB	
Value2		

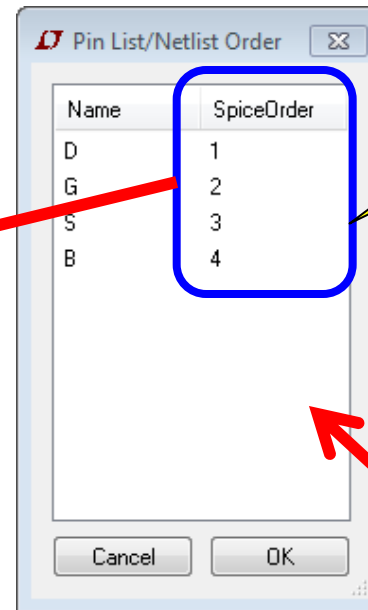
Cancel OK

Importing Third-Party Subckt Models

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

*FDC6000NZ at Temp. Electrical Model

*-----

.SUBCKT FDC6000NZ 20 10 30 50

*20=DRAIN 10=GATE 30=SOURCE 50=VTEMP

Rg 10 11x 1

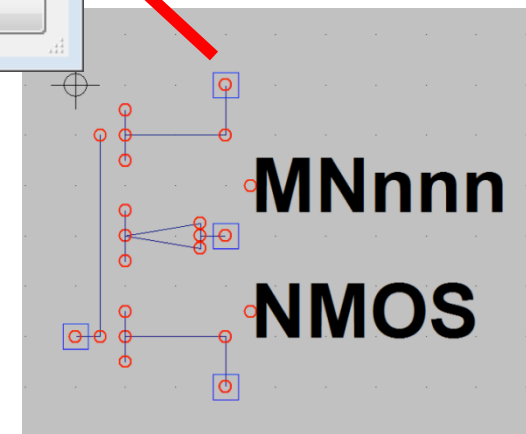
Rdu 12x 1 1u

M1 2 1 4x 4x DMOS L=1u W=1u

.MODEL DMOS NMOS(VT0=1.1 KP=3.79E+1

+THETA=0.133333 VMAX=7E5 LEVEL=3)

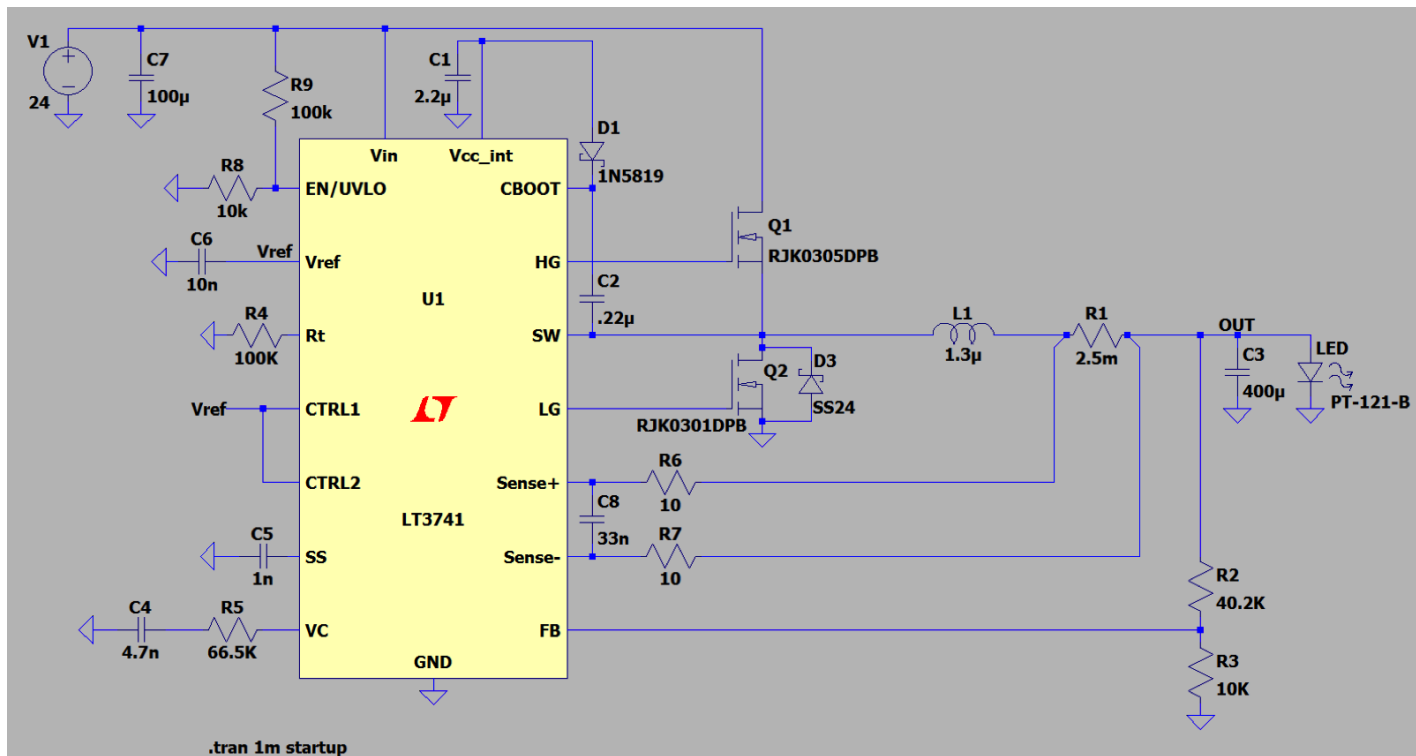
.subckt nodes



Importing Third-Party Subckt Models

Hands-on Exercise:

- ❖ Change the “LT3741.asc” circuit so it uses the PSMN2R2-30YLC FET subcircuit model instead of the RJK0301DPB.
- ❖ The model’s file, [PSMN2R2_30YLC.txt](#), is located in [<folder link>](#)



Making Circuit Files More Transportable

Previously shown methods of importing models rely on an external model/library file. Both the schematic and lib file must be sent when sharing schematics.

To avoid handling multiple files, the models can be attached to the LTspice schematic making it more transportable:

- ❖ Open .model or .subckt file and copy text to the clipboard
- ❖ Open new spice directive in LTspice
- ❖ Paste clipboard contents to directive text box
- ❖ Place on schematic
- ❖ Easier than using .include for simple models

Making Circuit Files More Transportable

Device model
on schematic

LTspice IV - [LTC3863ModelOnSchematic.asc]

File Edit Hierarchy View Simulate Tools Window Help

LTC3863 - 60V Low IQ Inverting DC/DC Controller
5V to 16V Input, -5V/1.7A Output, 350kHz Inverting Converter

```

.SUBCKT Si7129DN D G S
M1 3 GX S S PMOS W= 5400000u L= 0.25u
M2 3 GX S D NMOS W= 5400000u L= 4.065e-07
R1 D 3 6.553e-03 TC=3.931e-03, 3.644e-06
CGS GX S 1.077e-09
CGD GX D 1.014e-10
RG G GY 1.8
RTCV 100 S 1e6 TC=1.785e-04, -6.686e-06
ETCV GY GX 100 200 1
ITCV S 100 1u
VTCV 200 S 1
DBD D S DBD
*****
.MODEL PMOS PMOS ( LEVEL = 3 TOX = 5e-8
+ RS = 0 KP = 2.339e-06 NSUB = 5.199e+16
+ KAPPA = 1e-2 ETA = 1e-4 NFS = 8e11
+ LD = 0 IS = 0 TPG = -1)
*****
.MODEL NMOS NMOS ( LEVEL = 3 TOX = 5e-8
+ NSUB = 1.788e+16 IS = 0 TPG = -1 )
*****
.MODEL DBD D (
+ FC = 0.1 TT = 1.94e-08 T_MEASURED = 25 BV = 31
+ RS = 1.322e-03 N = 1.241e+00 IS = 2.173e-10
+ EG = 1.186e+00 XTI = 2.015e-01 TRS1 = 3.886e-03
+ CJO = 4.797e-10 VJ = 1.000e-01 M = 2.157e-01 )
.ENDS

```

.tran 1m startup

No .include !

LTC3863Transportable.asc

Application Circuit Examples

- ❖ FDC6301N - Fairchild's Power MOSFET
 - ❖ Import and use a FET model with a temp node



FDC6301NTestCircuit.asc