



# Support for PSpice® Models in Altium Designer

## Summary

Application Note  
AP0136 (v1.3) June 20, 2006

This application note provides information on the level of support available for using PSpice® models when performing circuit simulations in Altium Designer.

The PSpice® simulation model format is the format of choice for many device manufacturers. Altium Designer's Mixed-Signal Circuit Simulator has strong support for PSpice models. The following sections provide summary information on:

- Changes made to existing Spice syntax to facilitate using expressions and global parameters to represent values in a PSpice-modeled circuit.
- Changes made to existing Spice3f5 devices in order to make them PSpice compatible.

## Spice Syntax Changes

To facilitate compatibility with PSpice, changes have been made to the existing Spice syntax. These changes include support for additional PSpice-based functions and operators, as well as the addition of global parameters.

### Function Support

The following additional functions are supported:

<b>ARCTAN(x)</b>	- returns the inverse tangent of x
<b>ATAN2(y, x)</b>	- returns the inverse tangent of y/x
<b>IF(t, x, y)</b>	- If t is TRUE then x, ELSE y
<b>LIMIT(x, min, max)</b>	- while $\min < x < \max$ , x is returned If $x < \min$ , min is returned If $x > \max$ , max is returned
<b>LOG10(x)</b>	- returns the decimal logarithm of x
<b>MAX(x, y)</b>	- returns the maximum of x and y
<b>MIN(x, y)</b>	- returns the minimum of x and y
<b>PWR(x, y)</b>	- returns x to the power of y
<b>PWRS(x, y)</b>	- returns signed x to the power of y:

## Support for PSpice Models in Altium Designer

- If  $x > 0$ , the result is positive  
If  $x < 0$ , the result is negative.
- SCHEDULE( $x_1, y_1, \dots, x_n, y_n$ )** - allows you to control the value of  $y$  based on time  $x$ . An entry for time = 0s must be entered.  
From time =  $x_1$  to  $x_2$ , returns  $y_1$   
From time =  $x_2$  to  $x_3$ , returns  $y_2$ , and so on.
- SGN( $x$ )** - returns the sign of  $x$  (a.k.a. the signum function).  
If  $x < 0$ , returns -1  
If  $x = 0$ , returns 0  
If  $x > 0$ , returns 1
- STP( $x$ )** - unit step function.  
If  $x > 0$ , returns 1  
If  $x < 0$ , returns 0
- TABLE( $x, x_1, y_1, \dots, x_n, y_n$ )** - allows you to construct a look-up table, returning the  $y$  value corresponding to  $x$  when all  $x_n, y_n$  points are plotted and connected by straight lines.  
If  $x >$  than the largest  $x$  value in the table, then the  $y$  value associated to that  $x$  value will be returned.  
If  $x <$  than the smallest  $x$  value in the table, then the  $y$  value associated to that  $x$  value will be returned.

## Additional Operator Support

The following additional operators are supported:

- **\*\*** (exponentiation)
- **==** (equality test)
- **!=** (non-equality test)
- **&** (Boolean AND)
- **|** (Boolean OR)

## In-Line Comment Support

The PSpice in-line comment character is supported. This character – a semicolon (;) – is treated as the end of a line in the circuit description. Any text after this character (on the same line) is treated purely as a comment and therefore ignored by the Simulator, which moves on to the next line in the circuit description.

The following example shows a single in-line comment, where comment text is added to one line in the circuit description:

```
R2 2 4 6 ; R2 is a feedback resistor
```

If you want to add comment text over multiple lines (creating trailing in-line comments), simply use the semicolon inline comment character to mark the beginning of each subsequent comment line:

```
R2 2 4 6 ; R2 is a  
      ; feedback resistor
```

An in-line comment can be used to replace a standard comment line, which must start with the \* character in the first column of the line. This can improve the readability of your circuit description.

## **.PARAM Support**

The PSpice .PARAM statement is supported. This statement defines the value of a parameter, allowing you to use a parameter name in place of numeric values for a circuit description. Parameters can be constants, expressions or a combination of the two. A single parameter statement can include reference to one or more additional parameter statements.

In addition, the following three internal variables (predefined parameters) are available for use in expressions:

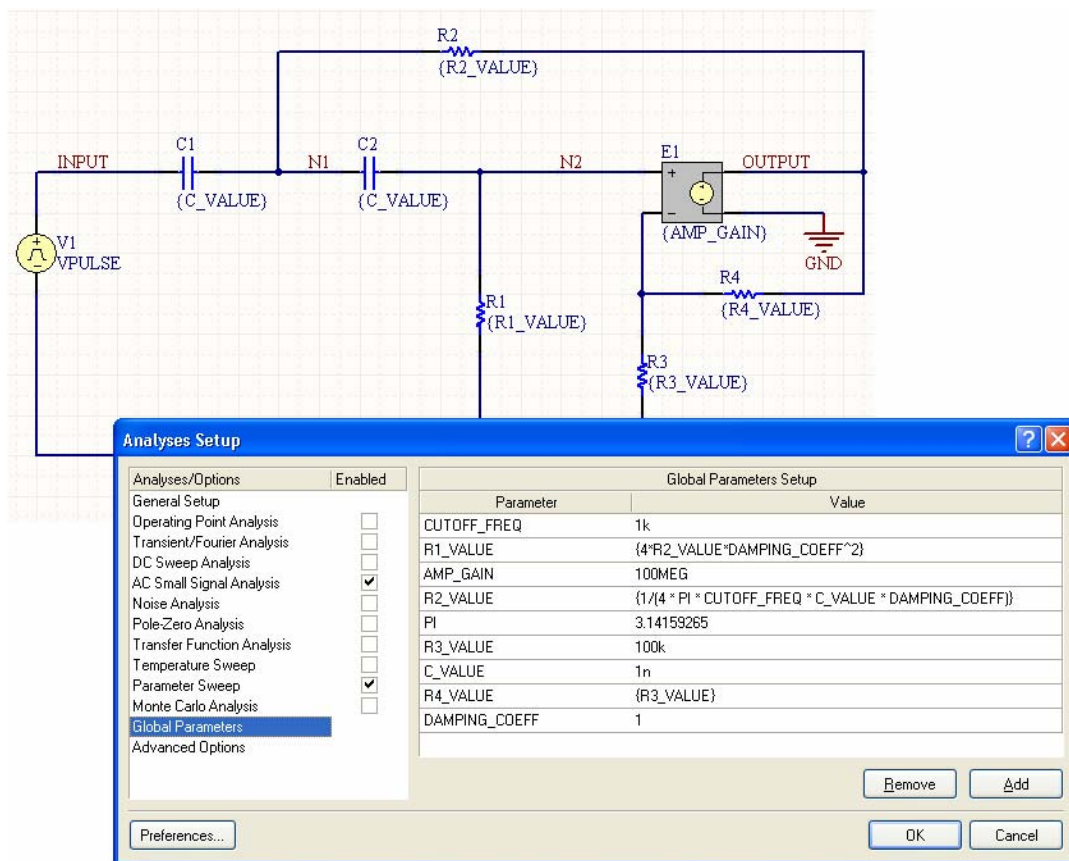
- GMIN** - shunt conductance for semiconductor p-n junctions.
- TEMP** - temperature.
- VT** - thermal voltage.


## **Global Parameters**

Altium Designer's Circuit Simulator supports the use of global parameters and equations. Use a global parameter in an equation and then use that equation in a component value on your schematic. Alternatively, define the equation as a global parameter and then reference the global parameter from a component value.

Simply include the expression or parameter name within curly braces {} – when the Simulator detects this it will attempt to evaluate it, checking the Global Parameters page of the Simulator's *Analyses Setup* dialog for the definition of any part of the expression that cannot be immediately resolved.


## Support for PSpice Models in Altium Designer




 For an example of using global parameters and equations in a simulation, refer to the example project Global Params.PrjPCB, which can be found in the \Program Files\Altium Designer 6\Examples\Circuit Simulation\PSpice Examples\Global Parameters folder of the installation.

## Spice Model Changes

To make the existing Spice3f5 device models compatible with PSpice, changes have been made to the general form for a device and/or additional parameter support has been added for use in a linked model file. The following sections summarize the changes that have been made.

 For additional detailed information on simulation models as well as simulation analyses, refer to the [Simulation Models and Analyses Reference](#). This reference should be used in conjunction with this application note in order to gather a complete listing of the supported parameters for each device model.

 For more detailed information concerning PSpice, consult the PSpice User's Guide and PSpice Reference respectively.

When browsing the subsequent sections, the following parameters – common to most devices in PSpice – are not supported:

T\_ABS  
T\_MEASURED  
T\_REL\_GLOBAL  
T\_REL\_LOCAL.

## Capacitor

The Netlist Template Format for this device remains unchanged:

```
@DESIGNATOR %1 %2 &VALUE
&MODEL ?LENGTH|L=@LENGTH| ?WIDTH|W=@WIDTH| ?"INITIAL VOLTAGE"|IC=@"INITIAL
VOLTAGE" |
```

The following additional model parameters are supported and can be entered into a linked model file (\*.mdl) for the device:

- C** - capacitance multiplier. (Default = 1).
- TC1** - linear temperature coefficient (in  $^{\circ}\text{C}^{-1}$ ). (Default = 0).
- TC2** - quadratic temperature coefficient (in  $^{\circ}\text{C}^{-2}$ ). (Default = 0).
- VC1** - linear voltage coefficient (in  $\text{Volt}^{-1}$ ). (Default = 0).
- VC2** - quadratic voltage coefficient (in  $\text{Volt}^{-2}$ ). (Default = 0).


Where a parameter has an indicated default, that default will be used if no value is specifically entered.

The format for the PSpice model file is:

```
.MODEL ModelName CAP(Model Parameters),
```

where

- *ModelName* is the name of the model, the link to which is specified on the **General** tab of the *Sim Model* dialog. This name is used in the netlist (&MODEL) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.

 For an example of using a PSpice-compatible capacitor model in a simulation, refer to the example project *Capacitor.PrjPCB*, which can be found in the \Program Files\Altium Designer 6\Examples\Circuit Simulation\PSpice Examples\Capacitor folder of the installation.

### Diode

The Netlist Template Format for this device remains unchanged:

```
@DESIGNATOR %1 %2 @MODEL &"AREA FACTOR" &"STARTING CONDITION" ?"INITIAL  
VOLTAGE" | IC=@"INITIAL VOLTAGE" | ?TEMPERATURE | TEMP=@TEMPERATURE |
```

The following additional model parameters are supported and can be entered into a linked model file (\*.mdl) for the device:

- IBVL** - low-level reverse breakdown knee current (in Amps). (Default = 0).
- IKF** - high-injection knee current (in Amps). (Default = infinite).
- ISR** - recombination current parameter (in Amps). (Default = 0).
- NBV** - reverse breakdown ideality factor. (Default = 1).
- NBVL** - low-level reverse breakdown ideality factor. (Default = 1).
- NR** - emission coefficient for isr. (Default = 2).
- TBV1** - bv temperature coefficient – linear (in °C<sup>-1</sup>). (Default = 0).
- TBV2** - bv temperature coefficient – quadratic (in °C<sup>-2</sup>). (Default = 0).
- TIKF** - ikf temperature coefficient – linear (in °C<sup>-1</sup>). (Default = 0).
- TRS1** - rs temperature coefficient – linear (in °C<sup>-1</sup>). (Default = 0).
- TRS2** - rs temperature coefficient – quadratic (in °C<sup>-2</sup>). (Default = 0).

Where a parameter has an indicated default, that default will be used if no value is specifically entered.

The format for the PSpice model file is:

```
.MODEL ModelName D(Model Parameters),
```

where

- *ModelName* is the name of the model, the link to which is specified on the **General** tab of the *Sim Model* dialog. This name is used in the netlist (@MODEL) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.



For an example of using a PSpice-compatible diode model in a simulation, refer to the example project *Diode.PrjPCB*, which can be found in the `\Program Files\Altium Designer 6\Examples\Circuit Simulation\PSpice Examples\Diode` folder of the installation.

## Inductor

The existing model has been changed to support the general PSpice model form:

```
L<name> <(+) node> <(-) node> [model name] <value> [IC = <initial value>]
```

A PSpice model of this type should be linked to a schematic component using a model file. Simply specify the model in a model file (\*.mdl) then, in the *Sim Model* dialog, set the **Model Kind** to General and the **Model Sub-Kind** to Generic Editor. The Netlist Template Format should then be entered as follows:

```
@DESIGNATOR %1 %2 @VALUE @MODEL ?"INITIAL CURRENT"|IC=@"INITIAL CURRENT"|
```

The value for the INITIAL CURRENT parameter is entered on the **Parameters** tab of the *Sim Model* dialog.

The netlist format for a PSpice Inductor model is specified using the Generic Editor due to the fact that the existing Spice3f5 Inductor model does not support use of a linked model file.

For the circuit to be parsed correctly, ensure that the **Spice Prefix** field is set to L.

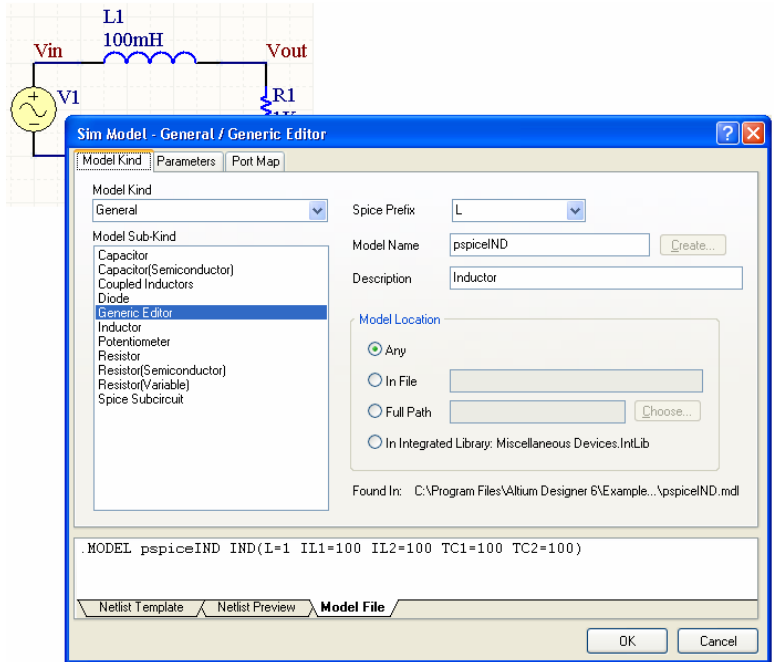
In the **Model Name** field, enter the name specified for the model in the model file. Use the options in the Model Location region of the dialog to point to the required file. Click on the **Model File** tab to view the content of the model file.

The following additional model parameters are supported and can be entered into a linked model file (\*.mdl) for the device:

- L** - inductance multiplier. (Default = 1).
- IL1** - linear current coefficient (in Amp<sup>-1</sup>). (Default = 0).
- IL2** - quadratic current coefficient (in Amp<sup>-2</sup>). (Default = 0).
- TC1** - linear temperature coefficient (in °C<sup>-1</sup>). (Default = 0).
- TC2** - quadratic temperature coefficient (in °C<sup>-2</sup>). (Default = 0).

Where a parameter has an indicated default, that default will be used if no value is specifically entered.

The format for the PSpice model file is:



## Support for PSpice Models in Altium Designer

```
.MODEL ModelName IND(Model Parameters),
```

where

- *ModelName* is the name of the model, the link to which is specified on the **General** tab of the *Sim Model* dialog. This name is used in the netlist (@MODEL) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.



For an example of using a PSpice-compatible inductor model in a simulation, refer to the example project *Inductor.PrjPCB*, which can be found in the `\Program Files\Altium Designer 6\Examples\Circuit Simulation\PSpice Examples\Inductor` folder of the installation.

## Current-Controlled Switch

The Netlist Template Format for this device remains unchanged:

```
V@DESIGNATOR %1 %2 0V
@DESIGNATOR %3 %4 V@DESIGNATOR @MODEL &"INITIAL CONDITION"
```

The following additional model parameters are supported and can be entered into a linked model file (\*.mdl) for the device:

- IOFF** - control current for OFF state (in Amps). (Default = 0).
- ION** - control current for ON state (in Amps). (Default = 1E-3).

Where a parameter has an indicated default, that default will be used if no value is specifically entered.

The format for the PSpice model file is:

```
.MODEL ModelName ISWITCH(Model Parameters),
```

where

- *ModelName* is the name of the model, the link to which is specified on the **General** tab of the *Sim Model* dialog. This name is used in the netlist (@MODEL) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.

## Voltage-Controlled Switch

The Netlist Template Format for this device remains unchanged:

```
@DESIGNATOR %3 %4 %1 %2 @MODEL &"INITIAL CONDITION"
```

The following additional model parameters are supported and can be entered into a linked model file (\*.mdl) for the device:

- VOFF** - control voltage for OFF state (in Volts). (Default = 0).
- VON** - control voltage for ON state (in Volts). (Default = 1).

Where a parameter has an indicated default, that default will be used if no value is specifically entered.

The format for the PSpice model file is:

```
.MODEL ModelName VSWITCH(Model Parameters),
```

where



- *ModelName* is the name of the model, the link to which is specified on the **General** tab of the *Sim Model* dialog. This name is used in the netlist (@MODEL) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.



For an example of using a PSpice-compatible voltage-controlled switch model in a simulation, refer to the example project PSpice\_Switch.PrjPCB, which can be found in the \Program Files\Altium Designer 6\Examples\Circuit Simulation\PSpice Examples\PSpice\_switch folder of the installation.

## JFET

The Netlist Template Format for this device remains unchanged:

```
@DESIGNATOR %1 %2 %3 @MODEL &"AREA FACTOR" &"STARTING CONDITION" ?"INITIAL
D-S VOLTAGE" |IC=@"INITIAL D-S VOLTAGE", @"INITIAL G-S
VOLTAGE" | ?TEMPERATURE |TEMP=@TEMPERATURE |
```

The following additional model parameters are supported and can be entered into a linked model file (\*.mdl) for the device:

<b>ALPHA</b>	-	ionization coefficient (in Volt <sup>-1</sup> ). (Default = 0).
<b>BETATCE</b>	-	BETA exponential temperature coefficient (in Amp/Volt <sup>2</sup> ). (Default = 1E-4).
<b>ISR</b>	-	gate p-n recombination current parameter (in Amps). (Default = 0).
<b>M</b>	-	gate p-n grading coefficient. (Default = 0.5).
<b>N</b>	-	gate p-n emission coefficient. (Default = 1).
<b>NR</b>	-	emission coefficient for isr. (Default = 2).
<b>VK</b>	-	ionization knee voltage (in Volts). (Default = 0).
<b>VTOTC</b>	-	VTO temperature coefficient (in Volt/°C). (Default = 0).
<b>XTI</b>	-	IS temperature coefficient. (Default = 3).

Where a parameter has an indicated default, that default will be used if no value is specifically entered.

The format for the PSpice model file is:

```
.MODEL ModelName NJF ( Model Parameters ) – N-channel JFET
.MODEL ModelName PJF ( Model Parameters ) – P-channel JFET
```

where

- *ModelName* is the name of the model, the link to which is specified on the **General** tab of the *Sim Model* dialog. This name is used in the netlist (@MODEL) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.



For an example of using a PSpice-compatible diode model in a simulation, refer to the example project JFET.PrjPCB, which can be found in the \Program Files\Altium Designer 6\Examples\Circuit Simulation\PSpice Examples\Jfet folder of the installation.

## Resistor

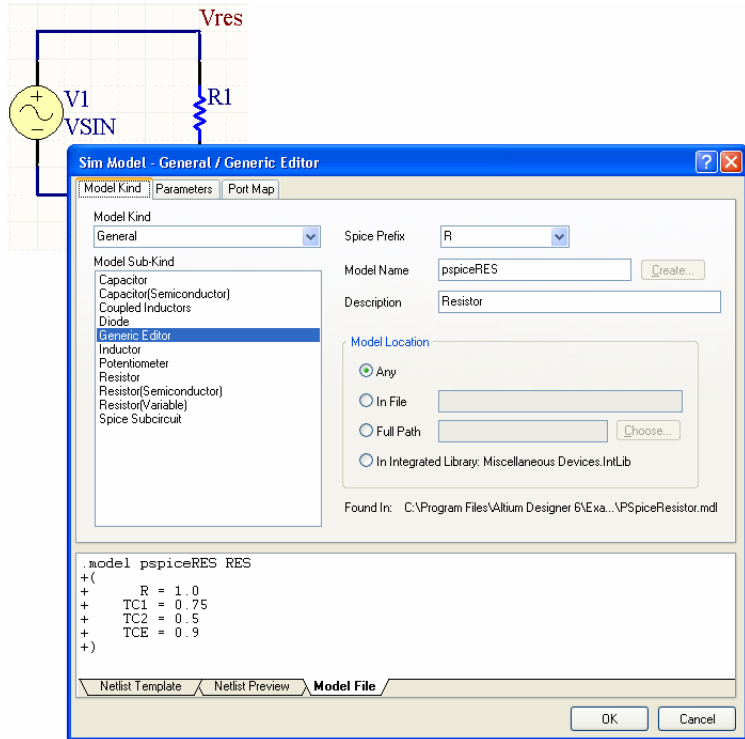
The existing model has been changed to support the general PSpice model form:

R<name> <(+) node> <(-) node> [model name] <value> [TC = <TC1> [,<TC2>]]

A PSpice model of this type should be linked to a schematic component using a model file. Simply specify the model in a model file (\*.mdl) then, in the *Sim Model* dialog, set the **Model Kind** to **General** and the **Model Sub-Kind** to **Generic Editor**. The Netlist Template Format should then be entered as follows:

```
@DESIGNATOR %1 %2 &MODEL
&VALUE ?TC1/TC=@TC1?TC2|,
@TC2| /
```

Although you could use the existing Spice3f5 Resistor (Semiconductor) model – as this model type allows use of a linked model file – specification of the netlist format for a PSpice Resistor model using the Generic Editor allows you to make use of the additional PSpice parameters ([TC = <TC1> [,<TC2>]]).



For the circuit to be parsed correctly, ensure that the **Spice Prefix** field is set to **R**.

In the **Model Name** field, enter the name specified for the model in the model file. Use the options in the Model Location region of the dialog to point to the required file. Click on the **Model File** tab to view the content of the model file.

The following additional model parameters are supported and can be entered into a linked model file (\*.mdl) for the device:

- R** - resistance multiplier. (Default = 1).
- TC1** - linear temperature coefficient (in °C<sup>-1</sup>). (Default = 0).
- TC2** - quadratic temperature coefficient (in °C<sup>-2</sup>). (Default = 0).
- TCE** - exponential temperature coefficient (in %/°C). (Default = 0).

Values for TC1 and TC2 can be entered on the **Parameters** tab of the dialog. Where a parameter has an indicated default, that default will be used if no value is specifically entered – either on the **Parameters** tab or in the linked model file.

The format for the PSpice model file is:

.MODEL *ModelName* RES(*Model Parameters*),

where

- *ModelName* is the name of the model, the link to which is specified on the **General** tab of the *Sim Model* dialog. This name is used in the netlist (&MODEL) to reference the required model in the linked model file.
- *Model Parameters* are a list of supported parameters for the model, entered with values as required.



For an example of using a PSpice-compatible capacitor model in a simulation, refer to the example project *Resistor.PrjPCB*, which can be found in the \Program Files\Altium Designer 6\Examples\Circuit Simulation\PSpice Examples\Resistor folder of the installation.

## Voltage-Controlled Voltage/Current Source

The following general PSpice model forms are supported:

- E<name> <(+) node> <(-) node> VALUE = { <expression> }
- E<name> <(+) node> <(-) node> TABLE { <expression> } = < <input value>, <output value> >
- E<name> <(+) node> <(-) node> POLY(<value>) < <(+) controlling node> <(-) controlling node> > < <polynomial coefficient value> >

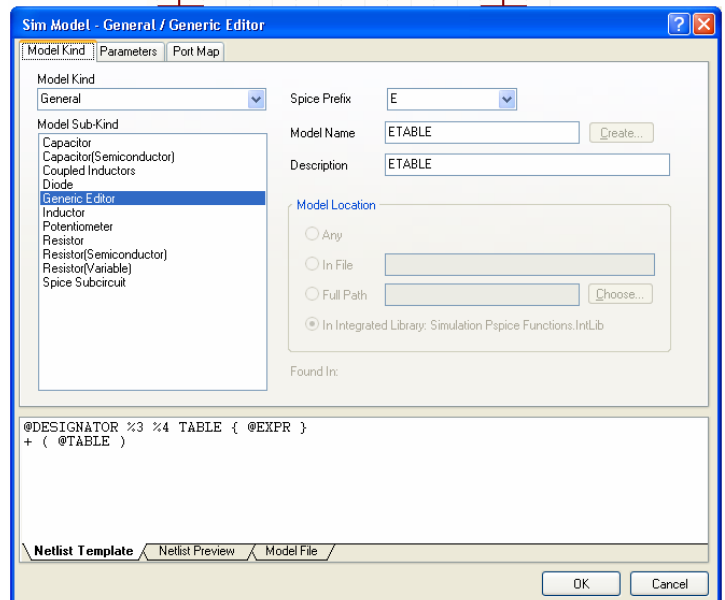
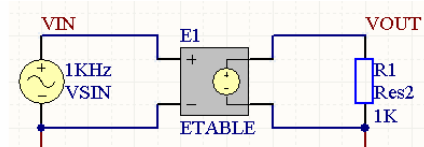
**Note:** For linear Voltage-Controlled Current Sources, the formats are the same as those above, but substituting G for E.

These devices do not support linked model files. The netlist format for a PSpice model in one of the above forms should be specified using the Generic Editor. In the *Sim Model* dialog, set the **Model Kind** to General and the **Model Sub-Kind** to Generic Editor.

For the circuit to be parsed correctly, ensure that the **Spice Prefix** field is set to the entry applicable to the device being used:

- VCVS – E
- VCCS – G.

The following are examples of generic netlist template formats that could be used for these model types.



### VALUE model

```
@DESIGNATOR %1 %2 VALUE = {@EXPR}
```

The value for the EXPR parameter is entered on the **Parameters** tab of the *Sim Model* dialog.

### TABLE model

```
@DESIGNATOR %1 %2 TABLE {@EXPR} = @ROW1 ?ROW2|@ROW2| ?ROW3|@ROW3|
```

Values for the EXPR and ROW parameters are entered on the **Parameters** tab of the *Sim Model* dialog. Any number of ROW parameters can be defined, in the format (<input value>, <output value>).

The netlist format could be entered using the following alternative entry:

```
@DESIGNATOR %3 %4 TABLE { @EXPR } ( @TABLE )
```

Values for the EXPR and TABLE parameters are again entered on the **Parameters** tab of the *Sim Model* dialog. The value for the TABLE parameter is specified in the form:

```
(<input1>, <output1>)(<input2>, <output2>)...(<inputn>, <outputn>)
```

### POLY model

```
@DESIGNATOR %3 %4 POLY (@dimension) (%1, %2) @coeffs
```

The values for the dimension and coeffs parameters are entered on the **Parameters** tab of the *Sim Model* dialog.



For an example of using a PSpice-compatible expression-based voltage-controlled voltage source in a simulation, refer to the example project EVALUE.PrjPCB, which can be found in the \Program Files\Altium Designer 6\Examples\Circuit Simulation\PSpice Examples\VCVS - Value folder of the installation.



For an example of using a PSpice-compatible lookup table-based voltage-controlled voltage source in a simulation, refer to the example project TABLE.PrjPCB, which can be found in the \Program Files\Altium Designer 6\Examples\Circuit Simulation\PSpice Examples\VCVS - Table folder of the installation.



For an example of using a PSpice-compatible polynomial-based voltage-controlled voltage source in a simulation, refer to the example project EPOLY.PrjPCB, which can be found in the \Program Files\Altium Designer 6\Examples\Circuit Simulation\PSpice Examples\VCvs - poly folder of the installation.

## Current-Controlled Voltage/Current Source

The following general PSpice model form is supported:

- H<name> <(+) node> <(-) node> POLY(<value>) <controlling V device name> < <polynomial coefficient value> >

**Note:** For a linear Current-Controlled Current Source, the format is the same as that above, but substituting F for H.

These devices do not support linked model files. The netlist format for a PSpice model in the above form should be specified using the Generic Editor. In the *Sim Model* dialog, set the **Model Kind** to General and the **Model Sub-Kind** to Generic Editor.

For the circuit to be parsed correctly, ensure that the **Spice Prefix** field is set to the entry applicable to the device being used:

- CCVS – H
- CCCS – F.

The following example generic netlist template format could be used for these model types:

```
@DESIGNATOR %1 %2 POLY (@dimension) @ControlSource @coeffs
```

The values for the `dimension`, `ControlSource` and `coeffs` parameters are entered on the **Parameters** tab of the *Sim Model* dialog.

## Bipolar Junction Transistor (BJT)

Many of the parameters that can be included in a linked model file for this type of device are common to both Spice3f5 and PSpice. Those that are supported can be found in the `SPICE3f5\models\Transistors\Bipolar Junction Transistor (BJT)` section of the [Simulation Models and Analyses Reference](#). The following PSpice-based parameters are *not supported*:

<b>CN</b>	-	quasi-saturation temperature coefficient for hole mobility
<b>D</b>	-	quasi-saturation temperature coefficient for scattering-limited hole carrier velocity
<b>GAMMA</b>	-	epitaxial region doping factor
<b>ISS</b>	-	substrate p-n saturation current
<b>NK</b>	-	high-current roll-off coefficient
<b>NS</b>	-	substrate p-n emission coefficient
<b>QCO</b>	-	epitaxial region charge factor
<b>QUASIMOD</b>	-	quasi-saturation model flag for temperature dependence
<b>RCO</b>	-	epitaxial region resistance
<b>TRB1</b>	-	RB temperature coefficient (linear)
<b>TRB2</b>	-	RB temperature coefficient (quadratic)
<b>TRC1</b>	-	RC temperature coefficient (linear)
<b>TRC2</b>	-	RC temperature coefficient (quadratic)
<b>TRE1</b>	-	RE temperature coefficient (linear)
<b>TRE2</b>	-	RE temperature coefficient (quadratic)
<b>TRM1</b>	-	RBM temperature coefficient (linear)
<b>TRM2</b>	-	RBM temperature coefficient (quadratic)
<b>VG</b>	-	quasi-saturation extrapolated bandgap voltage at 0° K
<b>VO</b>	-	carrier mobility knee voltage

## Support for PSpice Models in Altium Designer

- XCJC2** - fraction of CJC connected internally to Rb
- XCJS** - fraction of CJS connected internally to Rc

## Metal Oxide Semiconductor Field-Effect Transistor (MOSFET)

Of the existing MOSFET device models, the following is not supported with respect to PSpice compatibility:

- BSIM3 model version 2.0

For the other supported MOSFET device models, many of the parameters that can be included in a linked model file are common to both Spice3f5 and PSpice. Those that are supported can be found in the `SPICE3f5\models\Transistors\Metal Oxide Semiconductor Field-Effect Transistor (MOSFET)` section of the [Simulation Models and Analyses Reference](#). The following PSpice-based parameters are *not supported* for this device type:

- GDSNOI** - channel shot noise coefficient (use with NLEV=3)
- JSSW** - bulk p-n saturation sidewall current/length
- L** - channel length
- N** - bulk p-n emission coefficient
- NLEV** - noise equation selector
- PBSW** - bulk p-n sidewall potential
- RB** - bulk ohmic resistance
- RDS** - drain-source shunt resistance
- RG** - gate ohmic resistance
- TT** - bulk p-n transit time
- W** - channel width

## Revision History

---

Date	Version No.	Revision
19-Dec-2005	1.0	Initial release
18-Jan-2006	1.1	Modified netlist template formats for Inductor and VCVS/VCCS/CCVS/CCCS devices (POLY model).
19-Apr-2006	1.2	Removed EKV model entry from list of non-supported models in the MOSFET section.
20-Jun-2006	1.3	Updated for Altium Designer 6.3. Added section on in-line comment support.

Software, hardware, documentation and related materials:

Copyright © 2006 Altium Limited.

All rights reserved. You are permitted to print this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document is made. Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium Limited. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties, including fines and/or imprisonment. Altium, Altium Designer, Board Insight, CAMtastic, CircuitStudio, Design Explorer, DXP, LiveDesign, NanoBoard, NanoTalk, Nexar, nVisage, P-CAD, Protel, SimCode, Situs, TASKING, and Topological Autorouting and their respective logos are trademarks or registered trademarks of Altium Limited or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.