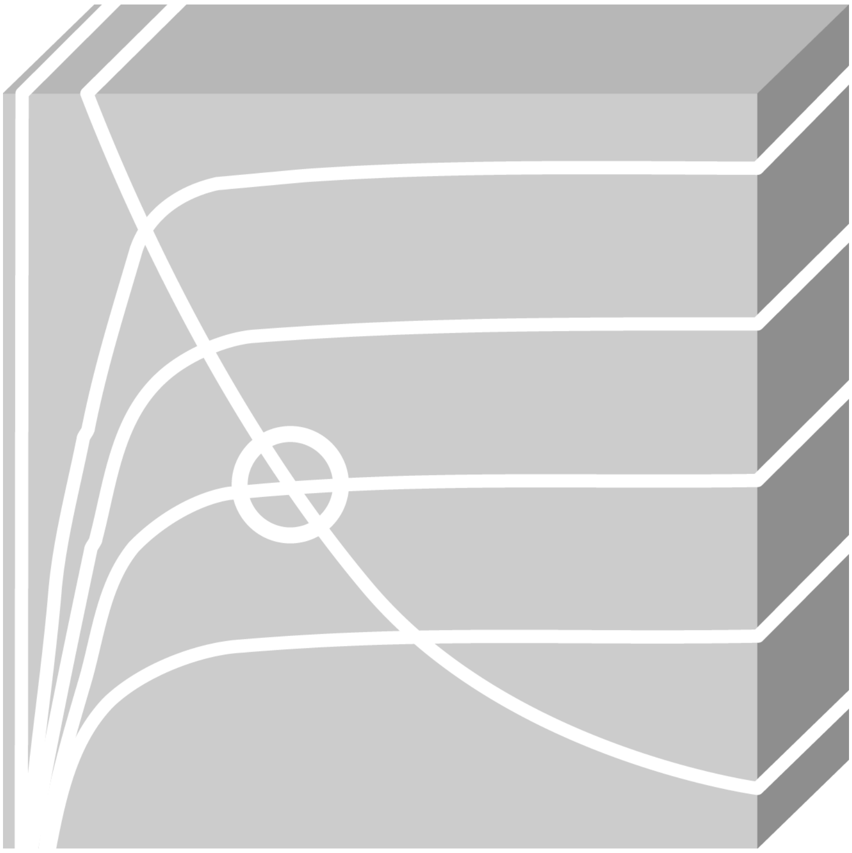
FlowCAD

PSpice AA Enhancements



PSpice Advanced Analysis

Application Note | V1

Table of Contents

1 Introduction

This Application Note covers:

- Introduction to the **Assign Tolerances** GUI to add tolerances to models, subcircuits, voltage / current sources and global variables.
- The steps to be followed to enable existing PSpice users to run PSpice Advanced Analysis modules on existing designs without any update of parts / models.

These enhancements allow that all existing PSpice users can run Advanced Analysis on existing designs, where there are components from Cadence library, models downloaded from website or models described by themselves.

Key functional enhancements:

- Ability to assign tolerances on device/model parameters
- Ability to assign tolerances on global variables
- Ability to assign tolerances on Voltage and Current sources
- Ability to assign tolerances on subcircuit parameters
- Models downloaded from web can be readily used in Advanced Analysis flow
- Enhancements in PSpiceAA GUI.

These enhancements are available up PSpice 17.2 Hotfix 11.

Note

Attached with this Application Note, there is a design called mycuk_Solution with the whole configuration. The goal is that you design this circuit by yourself and try to get the same results.

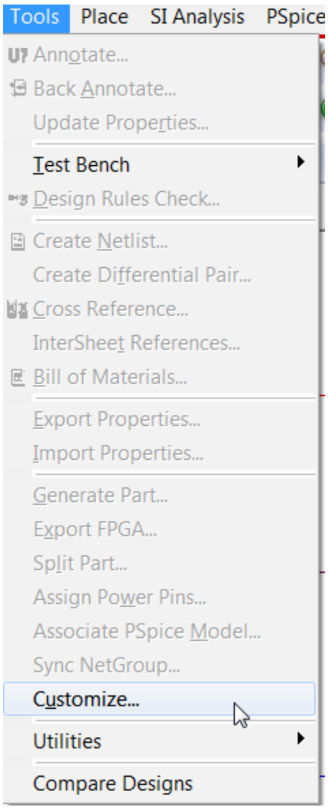
To be considered

This document is thought to introduce the PSpice AA enhancements. It is not training or workshop. If you need more technical information about PSpice A/D and / or PSpice AA, please contact with FlowCAD.

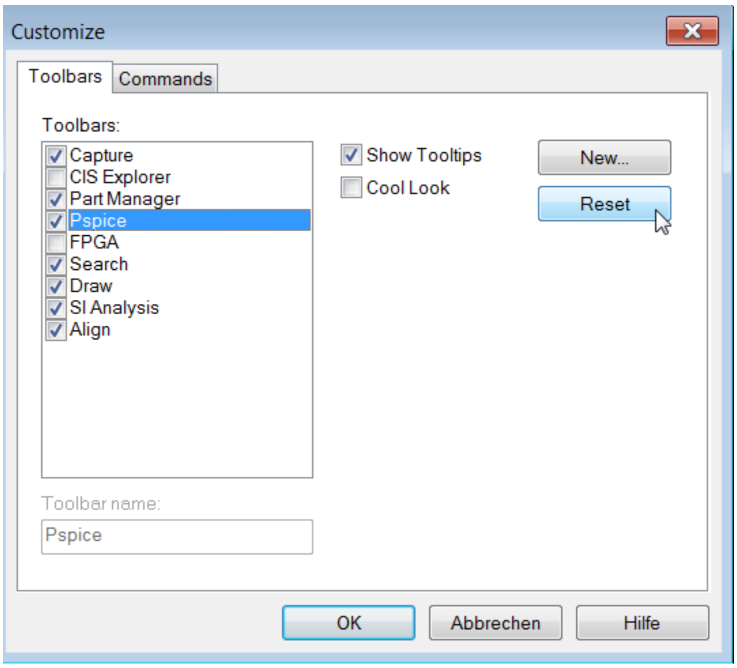
License for OrCAD PSpice Designer Plus or Allegro AMS Simulator is required.

2 How Can I Activate the New Enhancements?

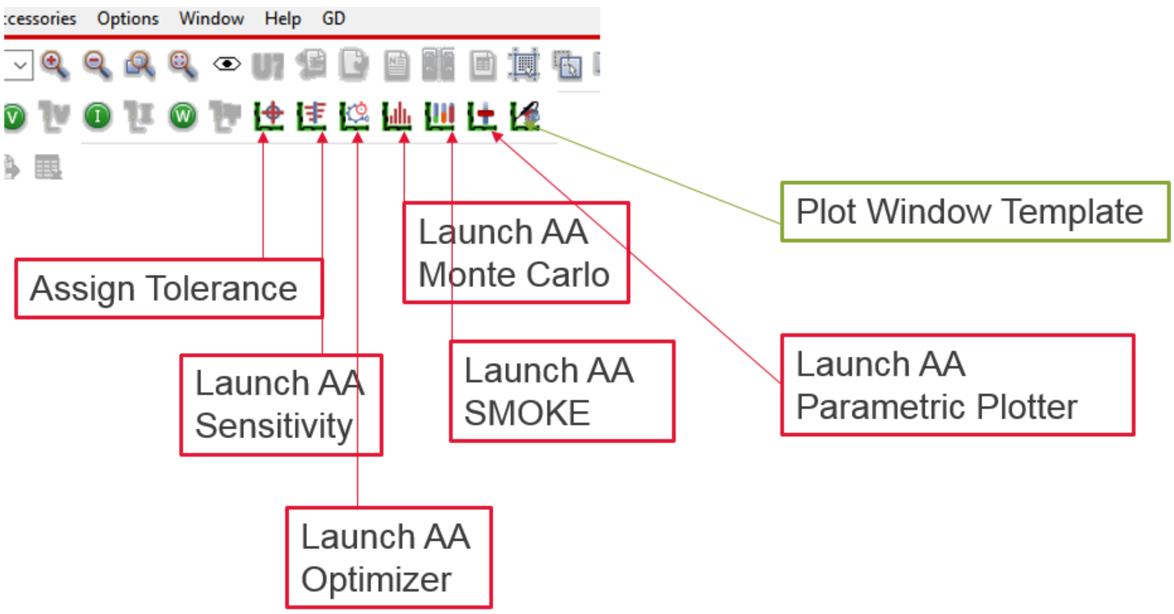
1. Open OrCAD Capture
2. Click on **Tools > Customize…**



3. Select **PSpice** and click on **Reset:**



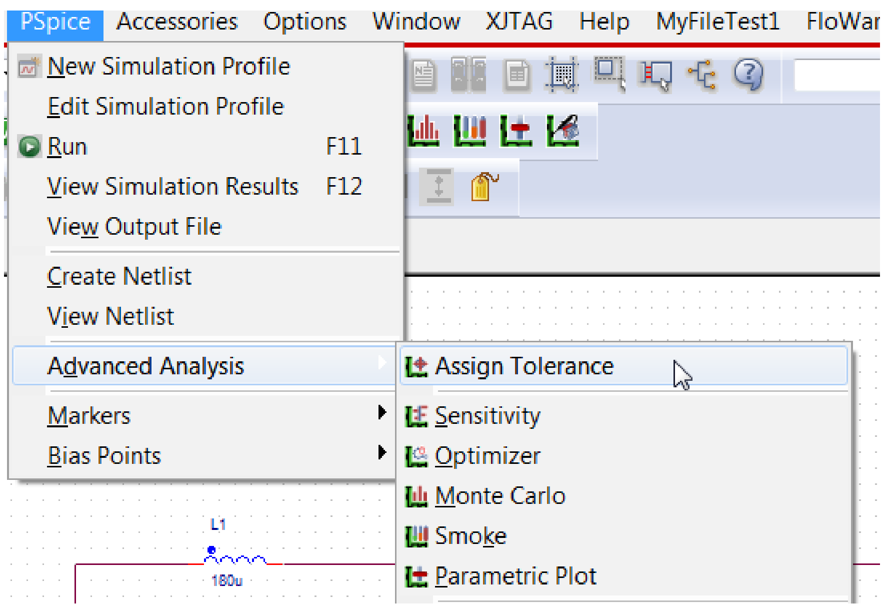
4. You will be able to see this new toolbar.



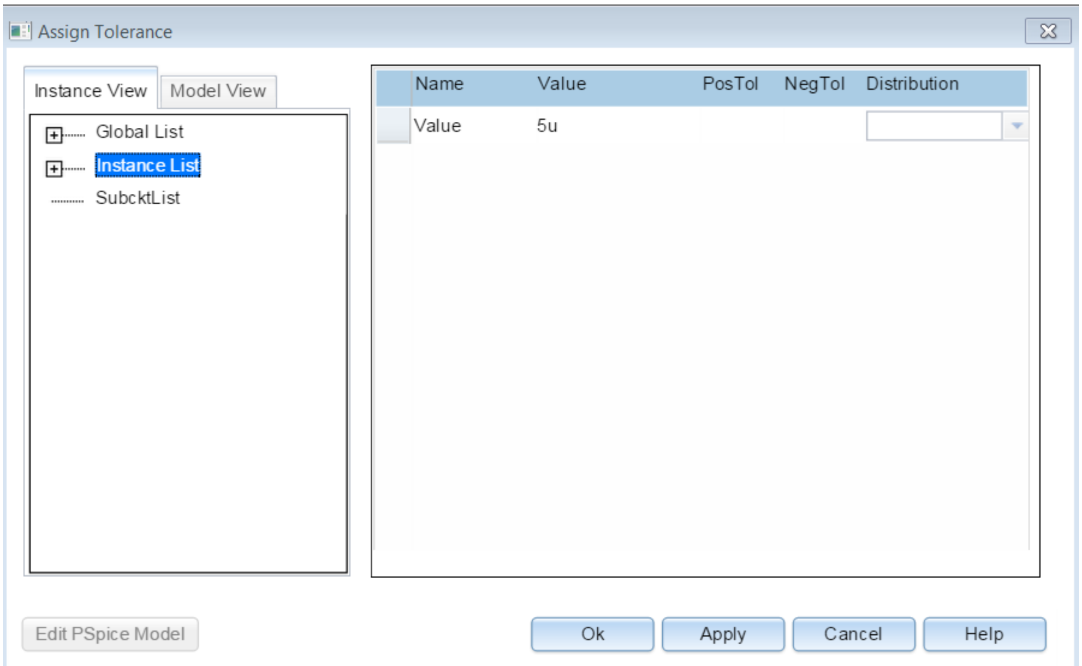
3 How Can I Add Tolerances to Components?

If you want to run a Monte Carlo or Sensitivity Analysis for your design in PSpice AA, you have to add tolerances at least to one component or variable, which is part of your circuit. To do that, follow the next steps:

1. Click on **Assign Tolerance.**



Next window will pop up:



In this GUI you can distinguish two areas.

– In the left, you can find the components you have used in your design, differentiating them between Instance View and Model View.
  - Instance View: It refers to each of the components placed on the design, distinguishing them by their part reference. For example, R1, R2, C2, D2, U1, Z2, etc.
  - Model View: it refers to the different PSpice Models you have used in your design. For example, IRF530, 1n4148, etc.

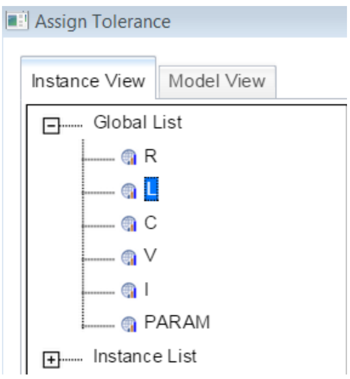
– In the right you can see the properties that define a component with their corresponding values, tolerances and distributions.

2. You can add tolerances (with flat or Gaussian distribution) to instances as Global, Instance and Subcircuit List:

   – Global List:

   You can add tolerances to affect globally to all the instances that refer to such reference designator. In this case, Resistors, Inductors, Capacitors, Voltage and Current Sources.

   For global PARAM’s it is the same issue. All the components, which refer to one of the global parameters defined in the component PARAM will use the same tolerance value.



NOTE

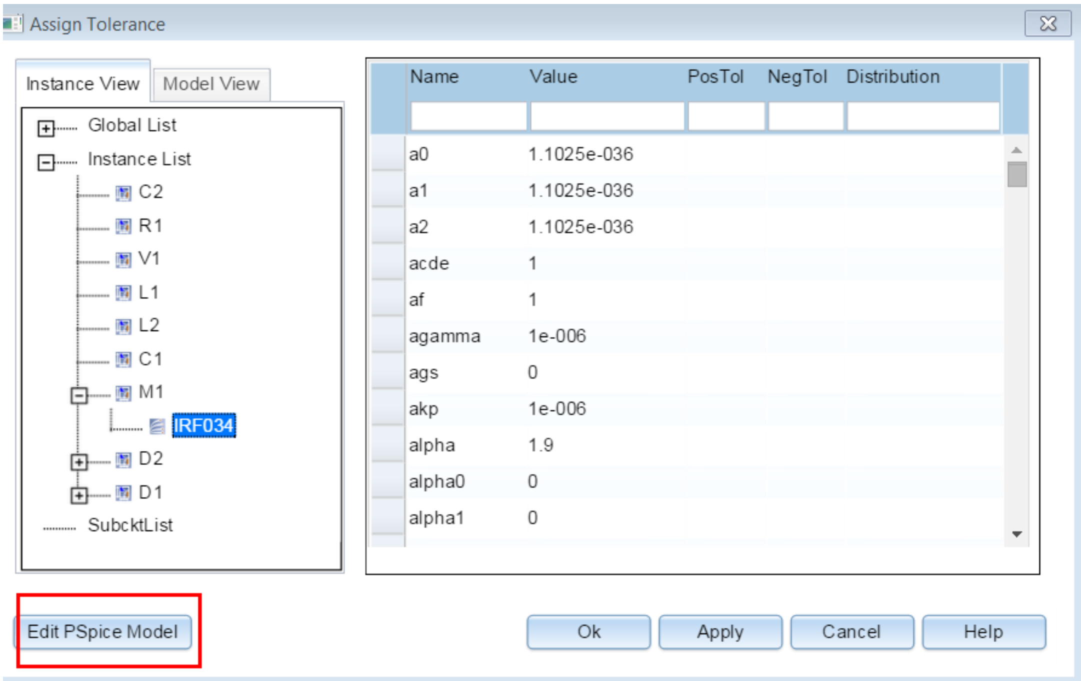
The tolerances defined globally are saved in the PSpice.ini.

```
GLOBAL_TOLERANCE_R=
GLOBAL_TOLERANCE_L=
GLOBAL_TOLERANCE_C=
GLOBAL_TOLERANCE_V=
GLOBAL_TOLERANCE_I=
ASSIGNTOLUI_INSTANCEMODE=
```

– Instance List:

In this list you can distinguish the tolerances that affect to each component placed in your design. It is organized using the reference designator. Resistor, capacitor, inductor, current and voltage source tolerances can be assigned in this GUI independently from each other. For example, you can assign a particular tolerance to R1 and another one to R2.

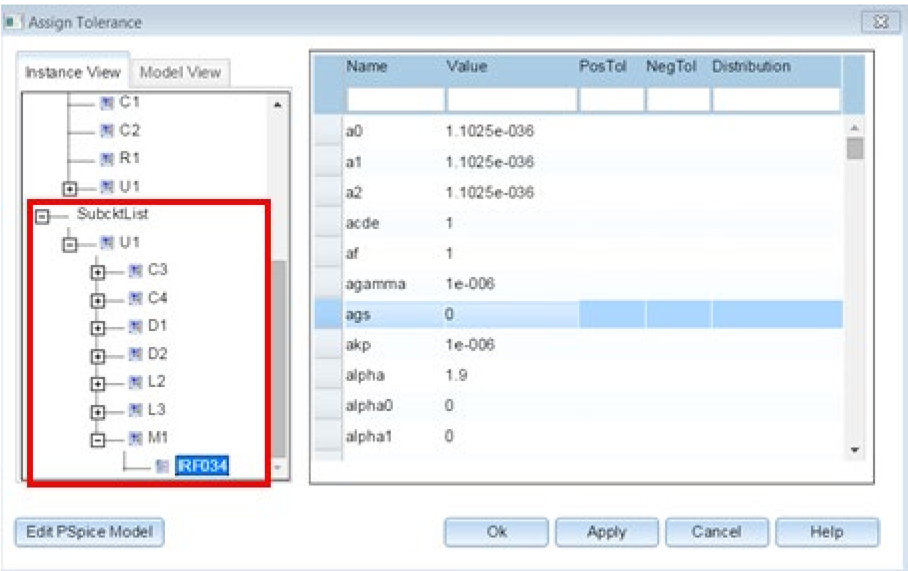
However, semiconductors or components which are described as PSpice Model in a PSpice Library, will be only readable in order to avoid that the original data is changed. Of course, librarians will be able to modify or define new tolerances if desired clicking on **Edit PSpice Model** directly from this GUI and opening the model description, but not directly from this GUI.



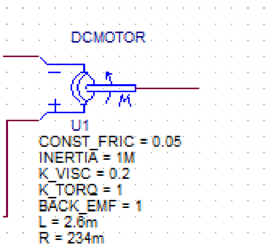
– SubcktList:

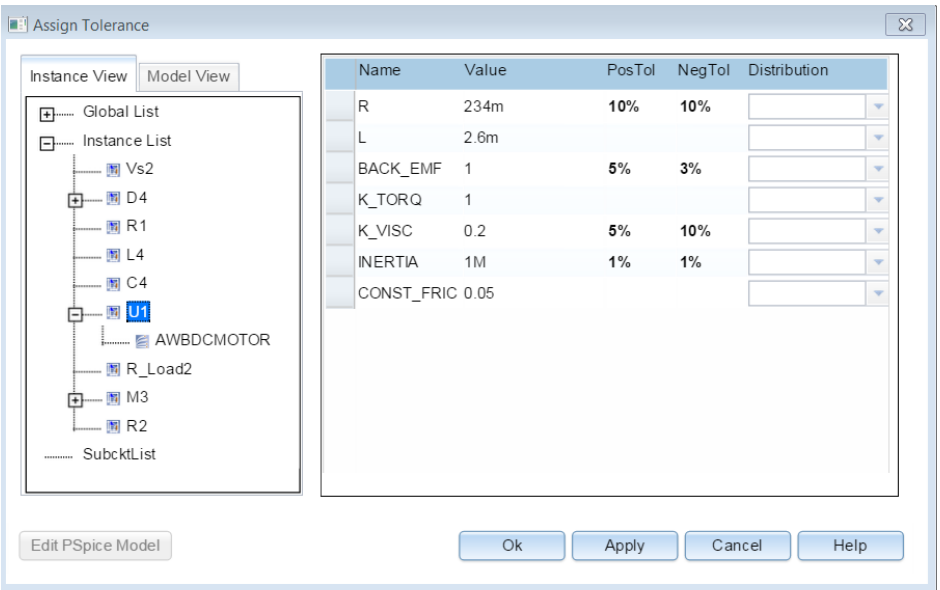
If you are using subcircuits in your designs, you have to take into account that there are two types of subcircuit parameters:

- Parameters related to devices embedded inside the subcircuit and not exposed to outer world. Model developer does not want this to change by design engineers. These parameters cannot be changed from this GUI. One need to follow edit model editor flow for this. The subcircuit will be showed in Instance List and in the Subcircuit List (here only if .model statements are used).



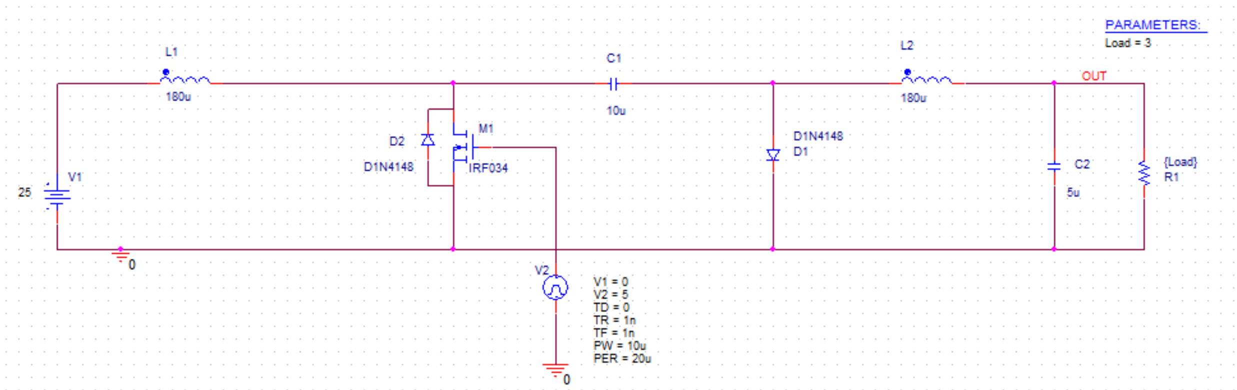
- Parameters exposed by model developer for designer to change. These are parameters exposed by PARAMS: statement in subcircuit definition. These parameters can be changed by designer from this GUI.





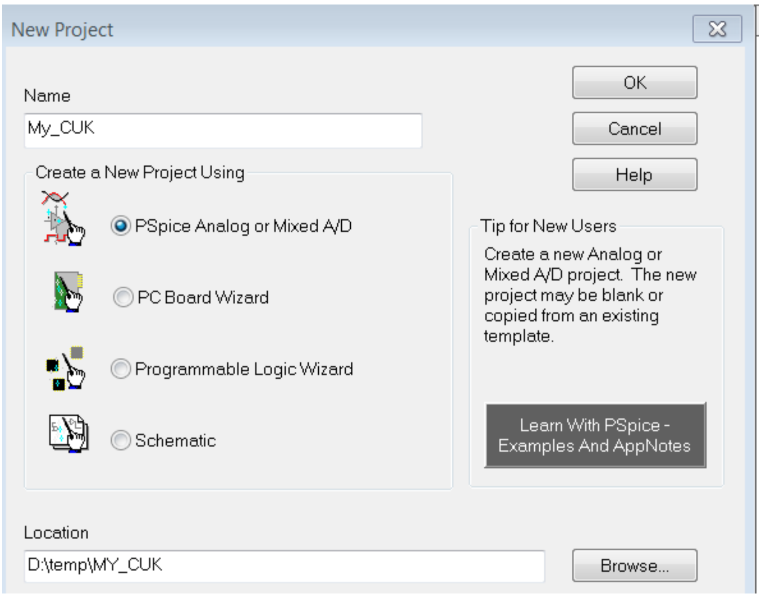
4 Simulation of a Monte Carlo and Sensitivity Analysis in PSpice AA

In this module it is showed an example to understand how flow works and how tolerances for discrete and for semiconductor components are defined. For that, consider the following circuit:

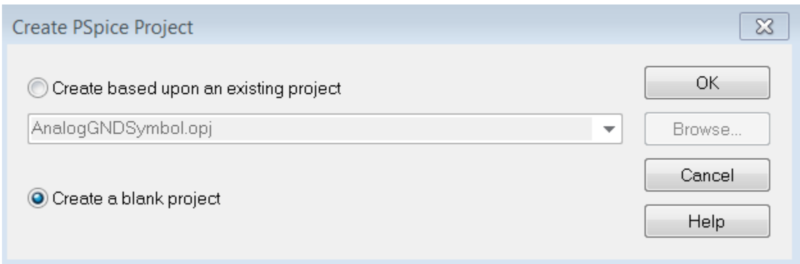


Create a new Project

1. Open OrCAD Capture
2. Click on **File > New > Project…**



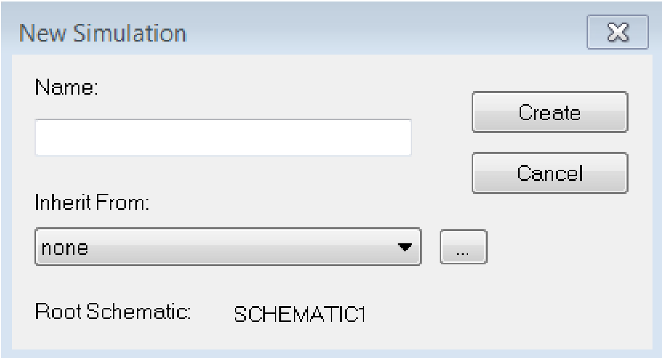
3. Select **Create a blank project** and click **OK**.



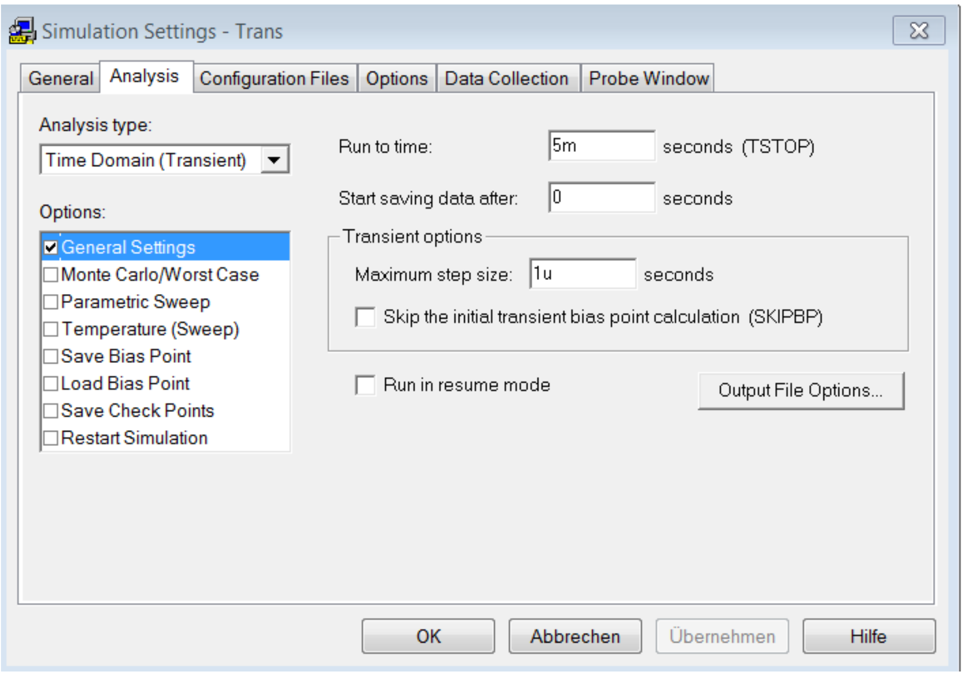
4. **Design** the circuit shown previously.
5. **Save**

Create Simulation Profile

1. Click on **PSpice > New Simulation Profile**



2. Name it **Trans** and click **Create**



3. Click **OK**
4. **Save**

Note

If you do not define a Simulation Profile previously, you cannot open the Assign Tolerances UI.

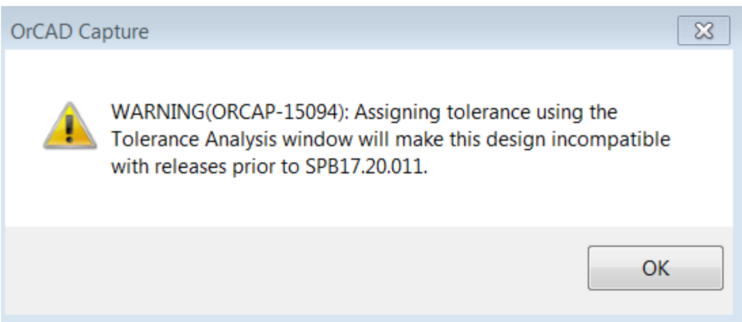
Set tolerances to the different components

Now you have to add the tolerances to the different components of the design. Consider that you are the librarian and the design engineer that is why, you will add the tolerances to the models D1, D2 and M1, because they have not been defined yet.

	Positive Tolerance	Negative Tolerance
L1	10%	5%
L2	15%	8%
C1	5%	3%
C2	5%	3%
V1	5%	5%
R1	3%	3%

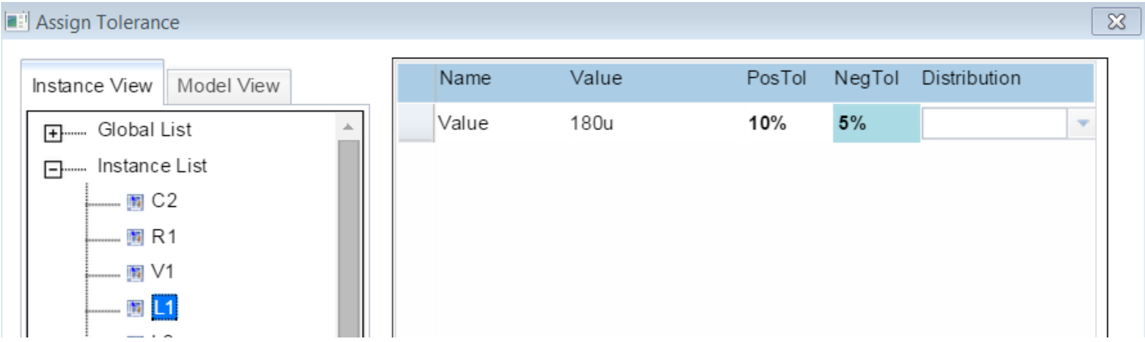
		Positive Tolerance	Negative Tolerance
D1	bv	5%	7%
	Cjo	7%	10%
	Is	3%	3%
D2	bv	5%	7%
	Cjo	7%	10%
	Is	3%	3%
M1	Cgdo	5%	10%
	Cgso	10%	10%

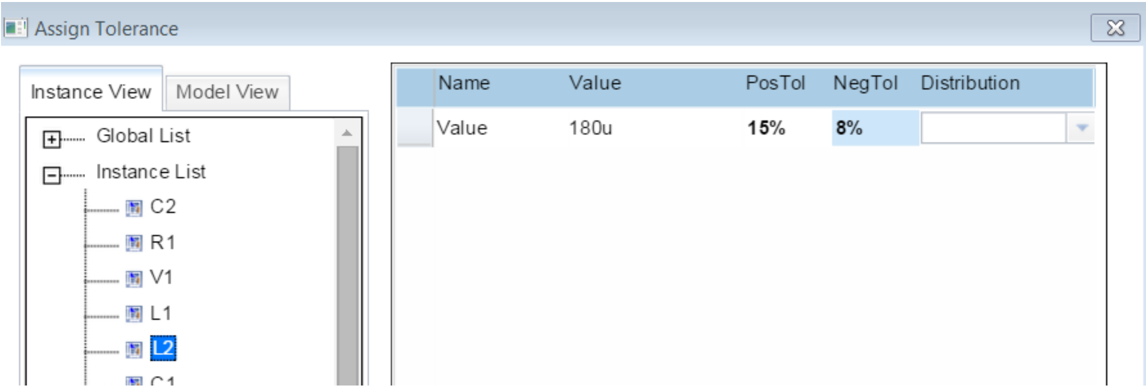
1. Click on **PSpice > Advanced Analysis > Assign tolerance.** A warning message will appear, informing you that the design will be incompatible if you add tolerances using this UI with releases prior to SPS17.20.011.
   Just click **OK**.



2. Define the tolerances for the discrete parts and the voltage source.

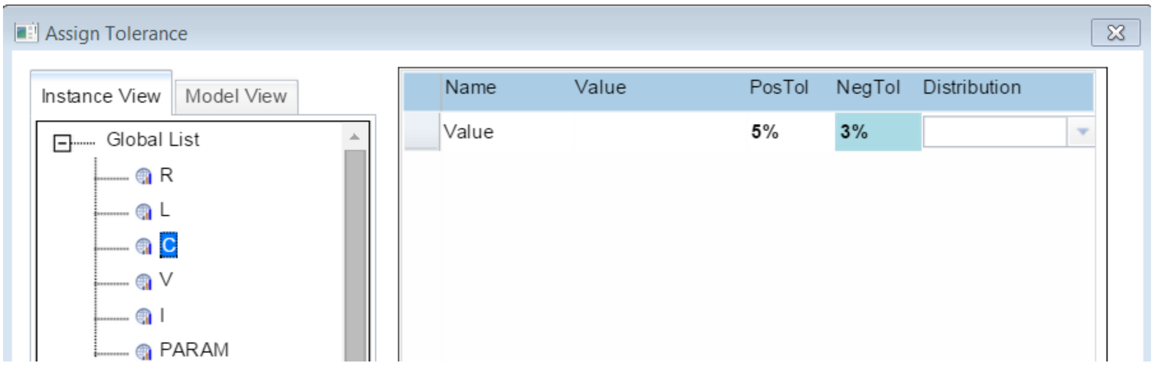
– As L1 and L2 have different tolerances, you have to define them using the Instance List:





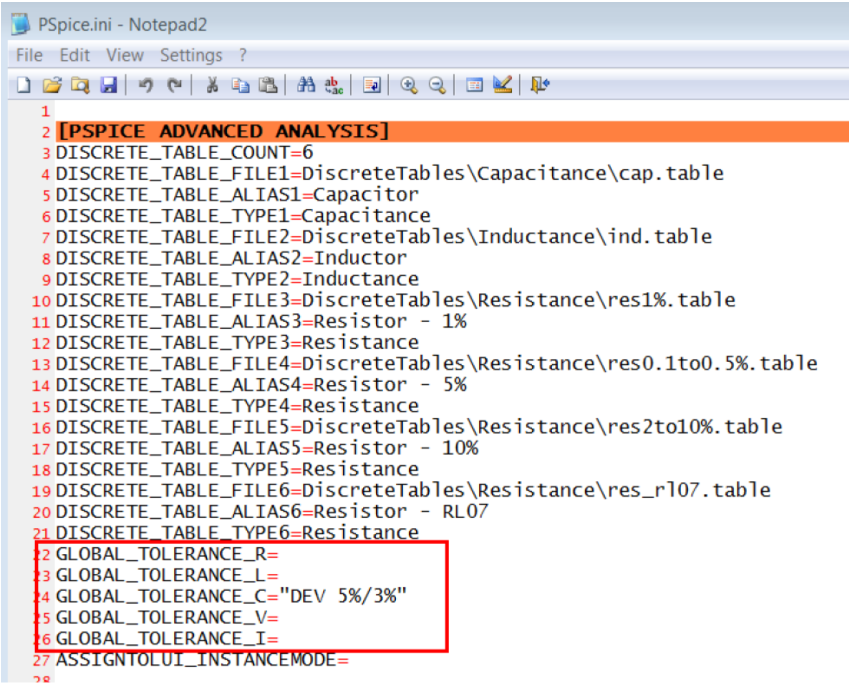
–

– As C1 and C2 have the same tolerances, we can define them in the Global List:



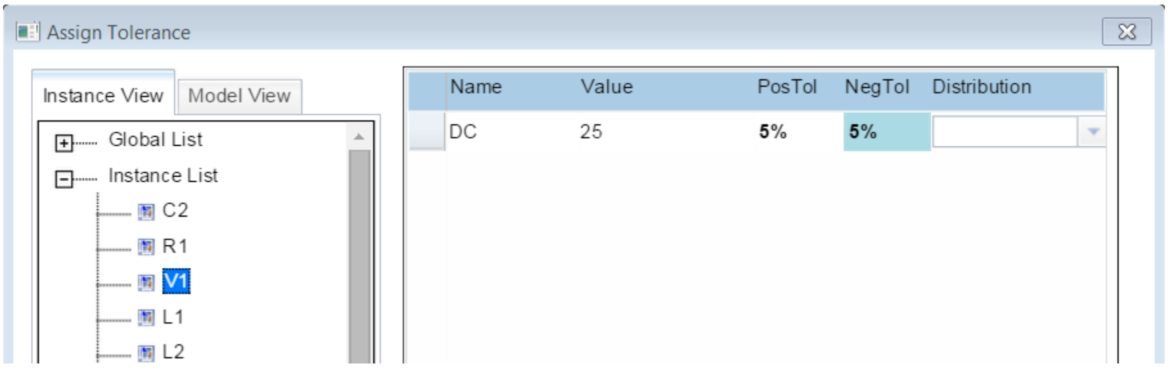
NOTE

The global tolerances defined in the Global List are saved in your PSpice.ini.

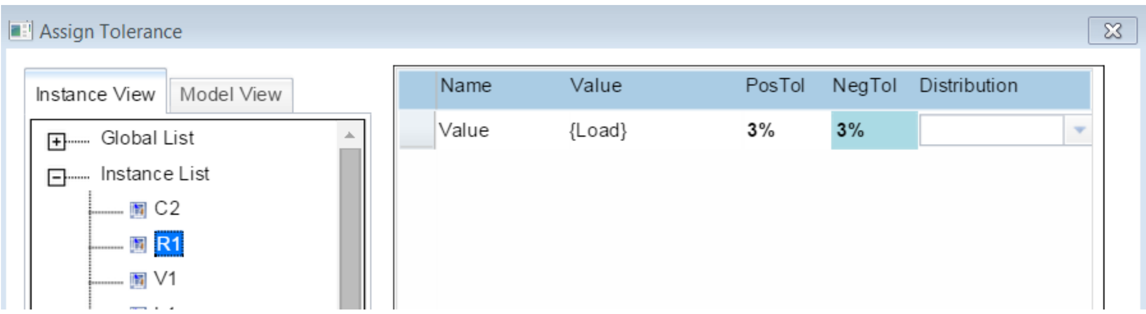



```
[PSPICE ADVANCED ANALYSIS]
DISCRETE_TABLE_COUNT=6
DISCRETE_TABLE_FILE1=DiscreteTables\Capacitance\cap.table
DISCRETE_TABLE_ALIAS1=Capacitor
DISCRETE_TABLE_TYPE1=Capacitance
DISCRETE_TABLE_FILE2=DiscreteTables\Inductance\ind.table
DISCRETE_TABLE_ALIAS2=Inductor
DISCRETE_TABLE_TYPE2=Inductance
DISCRETE_TABLE_FILE3=DiscreteTables\Resistance\res1%.table
DISCRETE_TABLE_ALIAS3=Resistor - 1%
DISCRETE_TABLE_TYPE3=Resistance
DISCRETE_TABLE_FILE4=DiscreteTables\Resistance\res0.1to0.5%.table
DISCRETE_TABLE_ALIAS4=Resistor - 5%
DISCRETE_TABLE_TYPE4=Resistance
DISCRETE_TABLE_FILE5=DiscreteTables\Resistance\res2to10%.table
DISCRETE_TABLE_ALIAS5=Resistor - 10%
DISCRETE_TABLE_TYPE5=Resistance
DISCRETE_TABLE_FILE6=DiscreteTables\Resistance\res_rl07.table
DISCRETE_TABLE_ALIAS6=Resistor - RL07
DISCRETE_TABLE_TYPE6=Resistance
GLOBAL_TOLERANCE_R=
GLOBAL_TOLERANCE_L=
GLOBAL_TOLERANCE_C="DEV 5%/3%"
GLOBAL_TOLERANCE_V=
GLOBAL_TOLERANCE_I=
ASSIGNTOLUI_INSTANCEMODE=
```

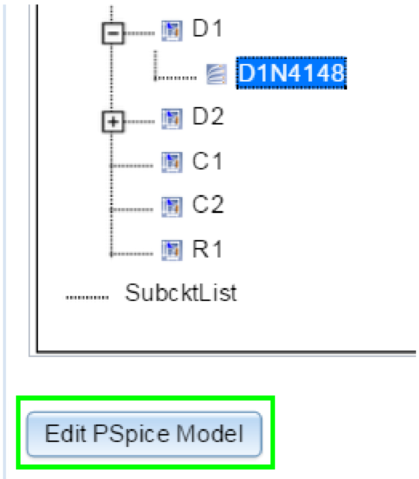
– For V1:



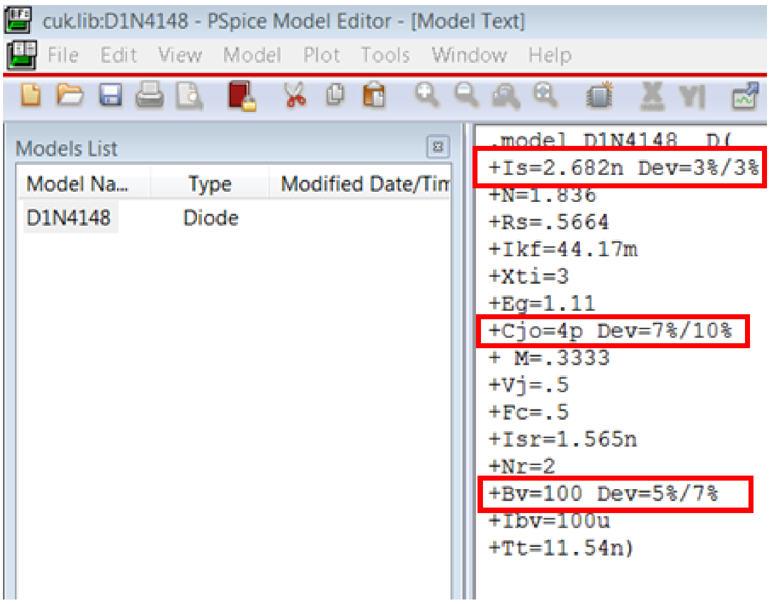
– For R1:



3. Set now the tolerances for the semiconductors, but take into account that such tolerances were not defined in their PSpice models, that is why, they have to be added before (as we are responsible in this case for the PSpice Library as well):
   – D1 and D2 are using the PSpice Model D1N4148. Highlight it and click on **Edit PSpice Model.**



   – Add the tolerances to the parameters indicated before in this way:

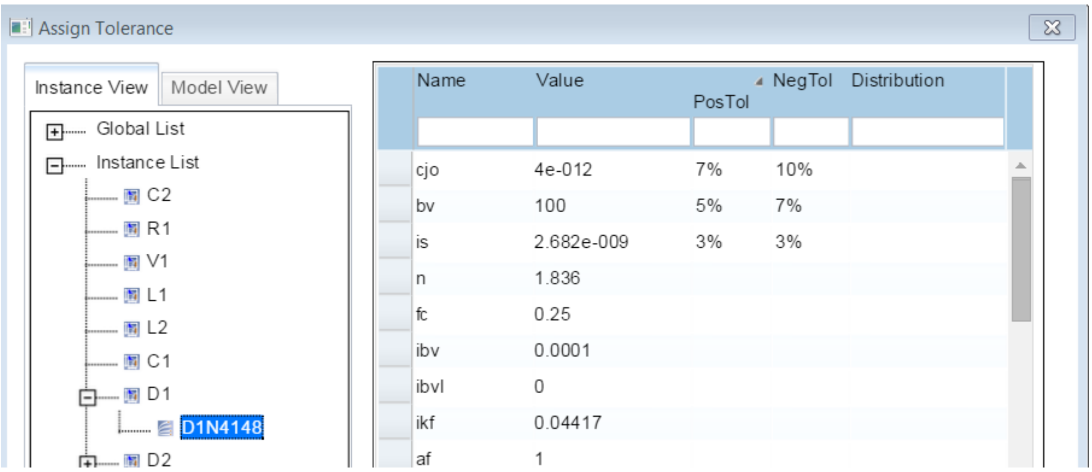


Now you can see such values on the **Assign Tolerance GUI,** but they are only readable, because as it was said, what you have done in first step of 3. is what the librarian typically does. It is not pretended to allow to change tolerances of PSpice models from this GUI in order to protect them.

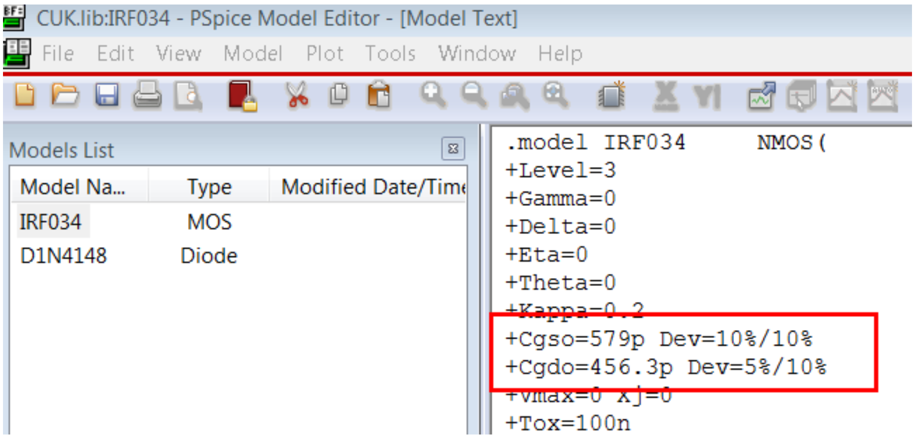
Note

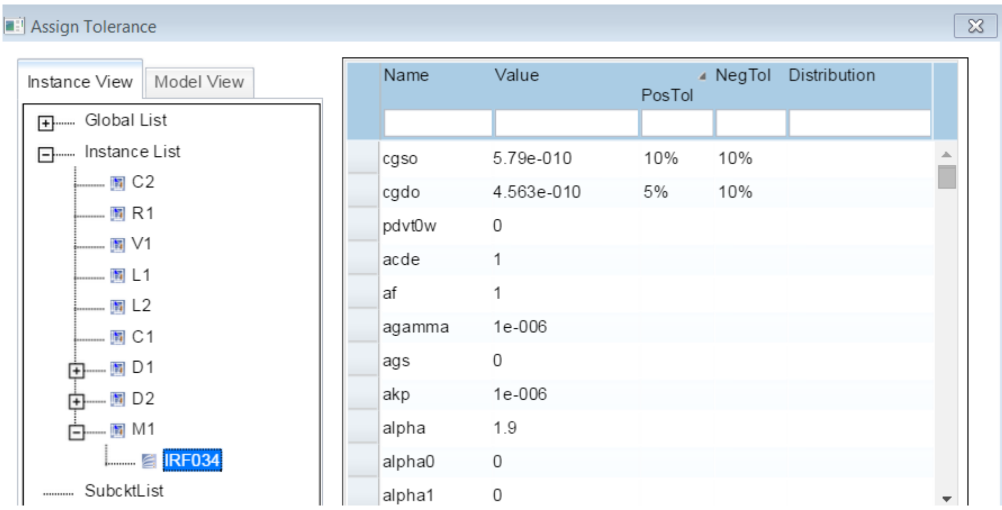
The first number in Dev is the positive tolerance and the second one is the negative tolerance. On the other hand, as we are changing components from the default Cadence

Libraries, a new library will be automatically generated and attached to our design (automatically added to your simulation profile).



– M1 is using the PSpice Model IRF034. Highlight it and click on **Edit PSpice Model.**
– Add the tolerances to the parameters indicated before in this way:

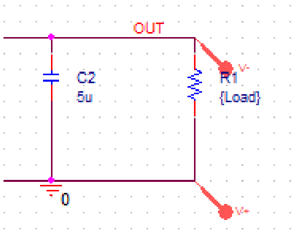


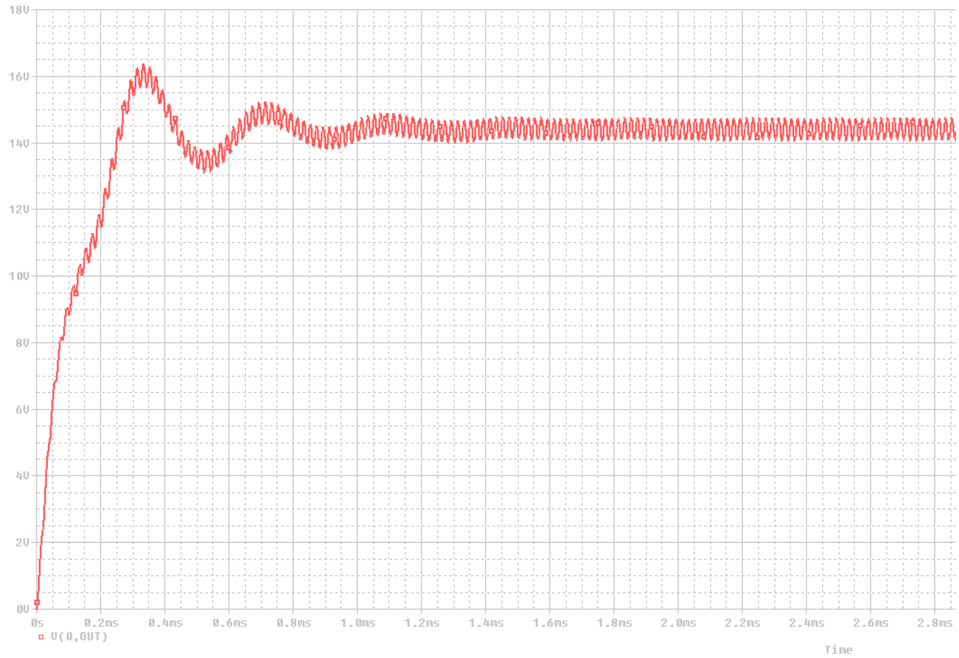


4. Click on **Apply** or **OK** to save your changings and close Assign Tolerance window.

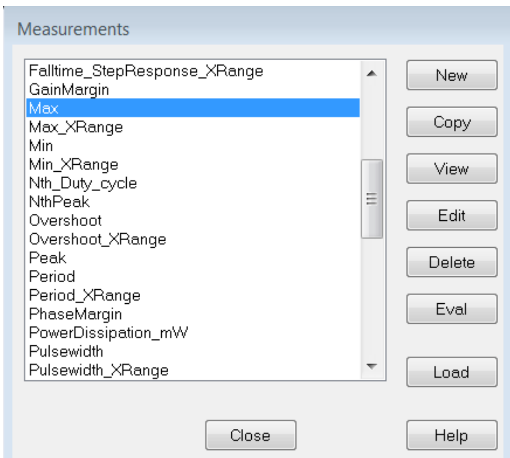
Simulate and define goals

1. Simulate the transient Analysis simulation profile you created previously.
2. Set the **Voltage Differental marker** on the output net to visualize the results in the Probe Window (take into account that a CUK converter inverts the signal):

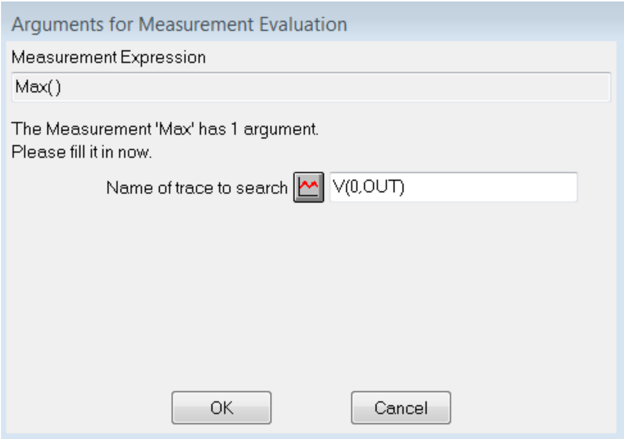




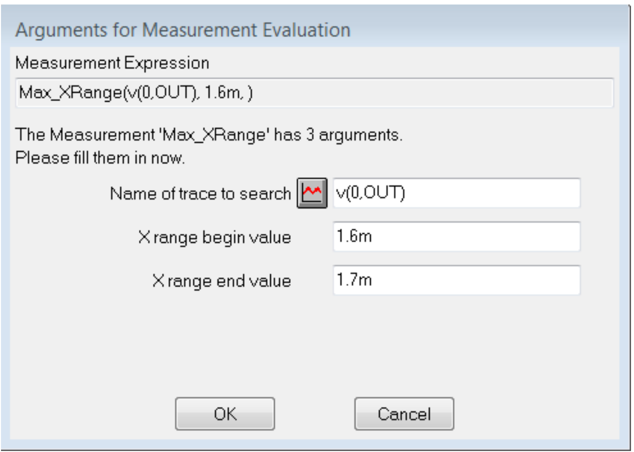
3. Click on **Trace > Measurements**
4. Select **Max** and click **Eval**:



5. Write **V(0,OUT)**



6. Click **OK** twice.
7. Select **Max_XRange** and click **Eval.**
8. Write these arguments:



9. Click **OK** twice.
10. Click on **View > Measurements Results.** You will see something like that:

Evaluate	Measurement	Value
☑	Max(V(0,OUT))	16.3591202446
☑	Max_XRange(v(0,OUT), 1.6m, 1.7m)	14.6955706308

11. Modify the second measurement like the image. Just click over the measurement expression for that:

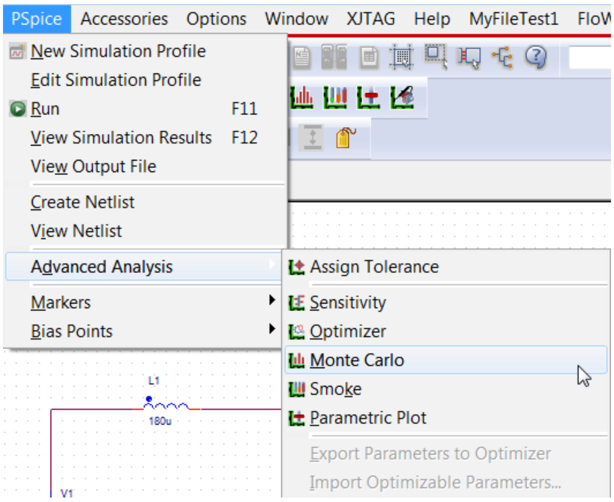
Evaluate	Measurement	Value
☑	Max(V(0,OUT))	16.3591202446
☑	Max_XRange(v(0,OUT), 1.6m, 1.7m) - Min_XRange(v(0,OUT), 1.6m, 1.7m)	620.1948351717m

Note

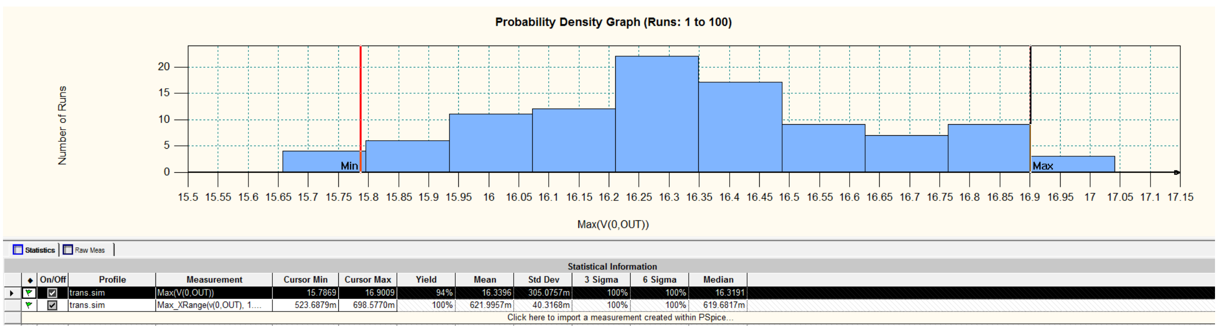
Remember that the measurements are the **goals** or requirements that must be achieved by your design. Each of these measurements will be used in PSpice Advanced Analysis to calculate the yield of your design or the sensitivity of the components in relation to the tolerances defined before.

Monte Carlo Simulation in Advanced Analysis

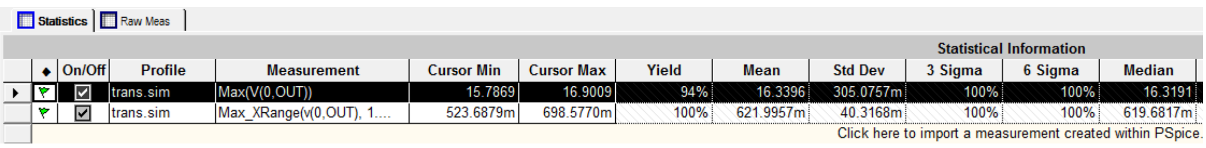
1. In OrCAD Capture, click on **PSpice > Advanced Analysis > Monte Carlo.**



2. Click on **Click here to import a measurement created within PSpice** to import the defined measurements in PSpice A/D:
3. Click **OK.**
4. Click on **Edit > Profile Settings** and select **Number of Runs** to 100.
5. Simulate.
6. Analyze the results:



It is possible to analyze in a glance the Yield achieved for each measurement and compare it with your requirements.



Statistics | Raw Meas

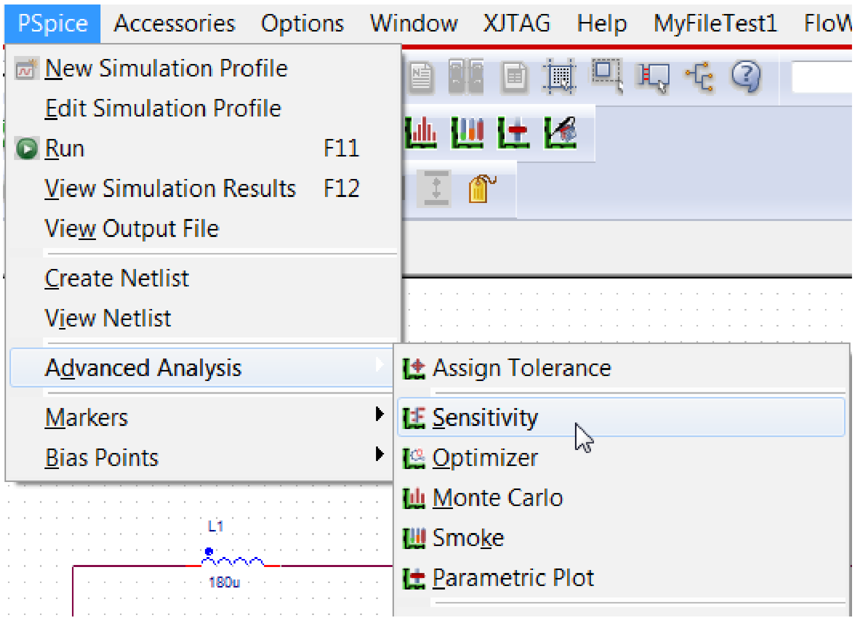
On/Off	Profile	Measurement	Cursor Min	Cursor Max	Yield	Mean	Std Dev	3 Sigma	6 Sigma	Median
☑	trans.sim	Max(V(0,OUT))	15.7869	16.9009	94%	16.3396	305.0757m	100%	100%	16.3191
☑	trans.sim	Max_XRange(v(0,OUT), 1....	523.6879m	698.5770m	100%	621.9957m	40.3168m	100%	100%	619.6817m

Statistical Information

Click here to import a measurement created within PSpice.

Sensitivity Simulation in Advanced Analysis

1. Come back to OrCAD Capture and click **PSpice > Avanced Analysis > Sensitivity.**



2. A window pops up with a list of all the component which have associated tolerances:

Component	Parameter	Original
V1	DC	25
L1	VALUE	180u
L2	VALUE	180u
C1	VALUE	10u
C2	VALUE	5u
R1	VALUE	3
D1N4148(model)	bv	100
D1N4148(model)	cjo	4e-012
D1N4148(model)	is	2.68200000000000e-009
IRF034(model)	cgdo	4.56300000000000e-010
IRF034(model)	cgso	5.79000000000000e-010

3. Click on **Click here to import a measurement created within PSpice a**nd import the measurement defined in PSpice A/D.
4. Run simulation.
5. Analyze the results for each measurement:

Sensitivity Component Filter = [*]

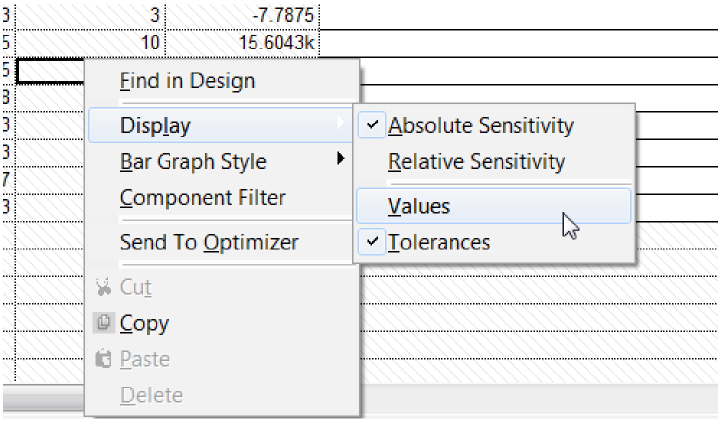
Component	Parameter	Original	Negtol(%)	Postol(%)	Rel Sensitivity	Linear
C1	VALUE	10u	3	5	3.4286m	3
C2	VALUE	5u	3	5	-1.8593m	1
D1N4148(model)	is	2.6820e-009	3	3	294.7465u	< MIN >
D1N4148(model)	cjo	4e-012	10	7	-27.9533u	< MIN >
D1N4148(model)	bv	100	7	5	15.9603u	< MIN >
IRF034(model)	cgso	5.7900e-010	10	10	652.1929u	< MIN >
IRF034(model)	cgdo	4.5630e-010	10	5	-6.3237m	5
L1	VALUE	180u	5	10	-1.5216m	1
L2	VALUE	180u	8	15	-7.0366m	6
R1	VALUE	3	3	3	112.5300m	99
V1	DC	25	5	5	72.9084m	64

	◆	On/Off	Profile	Measurement	Original	Min	Max
▶		☑	trans.sim	Max(V(0,OUT))	16.3592	15.4188	17.1224
		☑	trans.sim	Max_XRange(v(0,OUT), 1.6m, ...	620.4989m	492.4080m	733.1870m

6. Observe that the Negtol and the Postol for each parameter are shown well in % or well in variation:

	Component	Parameter	Original	Negtol(%)	Postol(%)
▶	C1	VALUE	10u	3	5
	C2	VALUE	5u	3	5
	D1N4148(model)	is	2.6820e-009	3	3
	D1N4148(model)	cjo	4e-012	10	7
	D1N4148(model)	bv	100	7	5
	IRF034(model)	cgso	5.7900e-010	10	10
	IRF034(model)	cgdo	4.5630e-010	10	5
	L1	VALUE	180u	5	10
	L2	VALUE	180u	8	15
	R1	VALUE	3	3	3
	V1	DC	25	5	5

Click **RMB > Display Values**



	Component	Parameter	Original	@Min	@Max	Rel Sensitivity
▶	C1	VALUE	10u	9.7000u	10.5000u	3.4286m
	C2	VALUE	5u	5.2500u	4.8500u	-1.8593m
	D1N4148(model)	is	2.6820e-009	2.6015n	2.7625n	294.7465u
	D1N4148(model)	cjo	4e-012	4.2800p	3.6000p	-27.9533u
	D1N4148(model)	bv	100	93	105	15.9603u
	IRF034(model)	cgso	5.7900e-010	521.1000p	636.9000p	652.1929u
	IRF034(model)	cgdo	4.5630e-010	479.1150p	410.6700p	-6.3237m
	L1	VALUE	180u	198u	171u	-1.5216m
	L2	VALUE	180u	207u	165.6000u	-7.0366m
	R1	VALUE	3	2.9100	3.0900	112.5300m
	V1	DC	25	23.7500	26.2500	72.9084m