

Example of PSpice with CIS

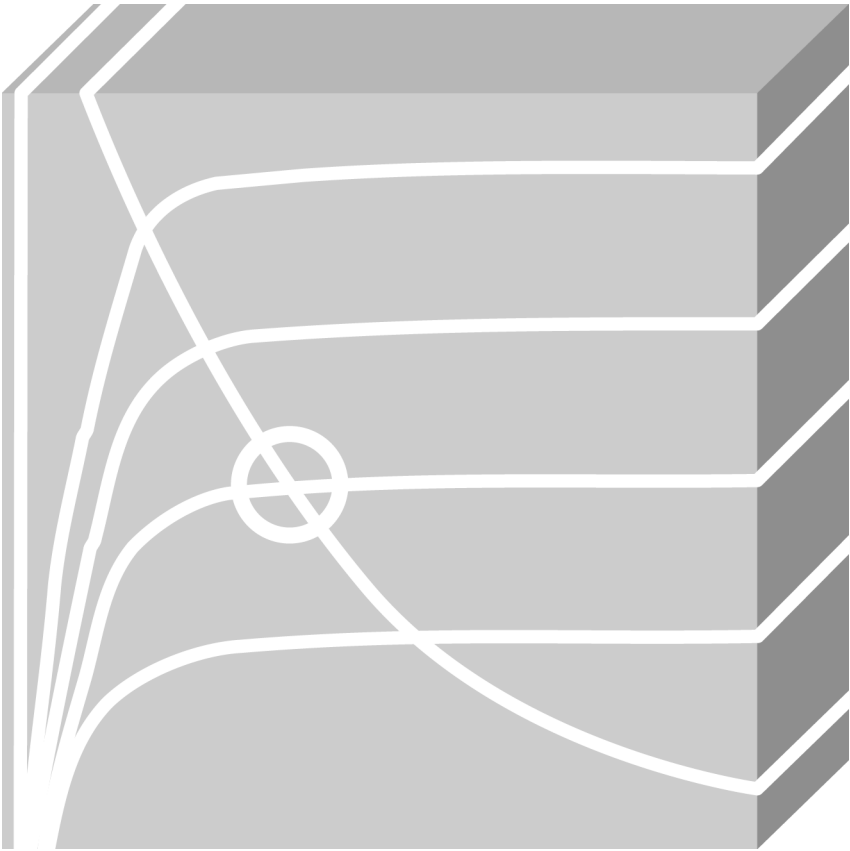


Table of Contents

- 1 Introduction 3**
- 2 Which Properties are Mandatory for Simulation?..... 3**
- 3 Definition of Components in the Database 4**
 - 3.1 Ideal R, L and C..... 4
 - 3.2 C and L with Parasitics..... 5
 - 3.3 Association Schematic Part with PSpice Model..... 9
 - 3.4 Download PSpice Models 9
 - 3.5 Homogeneous Parts 9
 - 3.6 Heterogeneous Parts 9
 - 3.7 Schematic Parts With More Pins Than Pins Has the PSpice Model 9
- 4 How to Simulate a Project Designed for Schematic and for Simulation?..... 10**
- 5 Feedback 12**

1 Introduction

The goal of this Application Note is to show a general example that mostly covers the different scenarios connecting schematic parts and PSpice models. In this way, it is also shown, that it is possible to maintain a database where there are properties for schematic design, PCB layout and for simulation. It makes the work much easier because everything is integrated in the same flow and an only component can be used for simulation, schematic design and layout.

This Application Note explains:

- How to define PSpice properties. It will be discussing if it is better to define such properties in the database or in the schematic part.
- Necessary properties to be defined for simulation.

It is **important** to mention that this Application Note is focused only in PSpice A/D simulations like transient analysis, Monte Carlo, sensitivity, temperature, etc. It means, that PSpice Advanced Analysis is not taken into account here, as many other sets of properties should be considered.

Finally, also take into account, that this Application Note covers a general procedure. For that, use this document as initial condition. Each database and DBC file have its own features. It means, that in many cases, it will be necessary to follow one strategy or another one depending on the features of such files. In this case, just contact FlowCAD to analyze this deeper.

2 Which Properties are Mandatory for Simulation?

There are three mandatory PSpice properties that must be considered and defined in a **schematic part** for simulation:

- **Implementation:** This is usually the same as the part name. This should correspond to the MODEL declaration in the PSpice model file that represents this part.
- **Implementation Type:** For symbols that will be simulated, the value will be PSpice Model.
- **PSpice Template:** A long string of variables and pin name notations that link the pins on the schematic symbol with the PSpice model and is used for the PSpice netlist generation. Symbols in the shipped Capture/Libraries/PSpice directory already have the PSpice Template.

These properties can be added as properties on the schematic symbol or in the CIS database. If the property exists in both the symbol and the database, the database property will overwrite the symbol property if placed through CIS.

However, the recommendation is to define them in the schematic part, because with the procedure that we will see for the association of PSpice models, such properties will be added automatically. Other properties like value, tolerance, etc. should be defined in the database.

Note

Ideal resistors, capacitors and inductors just need PSpice Template, because they are not developed using model library (.lib). Real inductors and capacitors with parasitics use PSpice Model (.lib), and they will have to be associated.

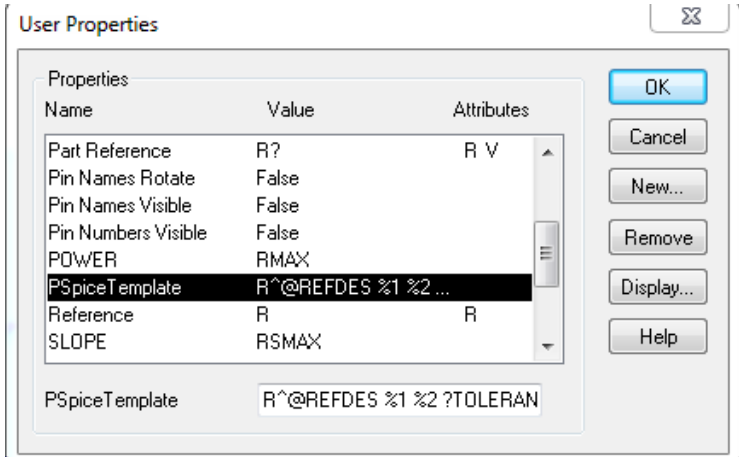
3 Definition of Components in the Database

In this chapter are analyzed different scenarios that a user could face and how it could be solved or done.

3.1 Ideal R, L and C

Ideal resistors, inductors and capacitors only use PSpiceTemplate property for simulation. This property should be defined in the schematic part. The procedure for that could be:

- Use directly the ideal R, L and C schematic parts from the analog.olb library, which is available in the default Cadence library.
- If you want to use your own schematic part, then open the ideal R, L or C in Capture, copy the value of PSpiceTemplate and paste it in your schematic part. For that you will have to check that the pin names are the same ones than in the schematic part as it is explained below.



The value for such property depending on R, L or C is next one:

Resistor

```
R^@REFDES %1 %2 ?TOLERANCE|R^@REFDES| @VALUE TC=@TC1,@TC2
?TOLERANCE|\n.model R^@REFDES RES R=1 DEV=@TOLERANCE% TC1=@TC1
TC2=@TC2|
```

As you can see, the resistor PSpiceTemplate value is referencing to different other property values. These values could be defined in the database or in the schematic part:

- Value: base resistance
- Tolerance: device tolerance
- TC1: linear temperature coefficient
- TC2: quadratic temperature coefficient

Please consider that the pin names of your schematic part must be the same than the pin names in the PSpiceTemplate. In this case, the pin names are 1 and 2. So if you have a

schematic part with the pin names A and B, you should change the PSpiceTemplate accordingly.

Inductor

```
L^@REFDES %1 %2 ?TOLERANCE|L^@REFDES| @VALUE ?IC/IC=@IC/  
?TOLERANCE|\n.model I^@REFDES IND L=1 DEV=@TOLERANCE% TC1=@TC1  
TC2=@TC2 IL1=@IL1 IL2=@IL2|
```

As you can see, the inductor PSpiceTemplate value is referencing to different other property values:

- Value: inductance
- Tolerance: device tolerance
- IC: initial current through the inductor during bias point calculation
- TC1: linear temperature coefficient
- TC2: quadratic temperature coefficient
- IL1: linear current coefficient
- IL2: quadratic current coefficient

Capacitor

```
C^@REFDES %1 %2 ?TOLERANCE|C^@REFDES| @VALUE ?IC/IC=@IC/ TC=@TC1,@TC2  
?TOLERANCE|\n.model C^@REFDES CAP C=1 DEV=@TOLERANCE% TC1=@TC1  
TC2=@TC2 VC1=@VC1 VC2=@VC2|
```

As you can see, the capacitor PSpiceTemplate value is referencing to different other property values:

- Value: capacitance
- Tolerance: device tolerance
- IC: initial voltage across the capacitor during bias point calculation
- TC1: linear temperature coefficient
- TC2: quadratic temperature coefficient
- VC1: linear voltage coefficient
- VC2: quadratic voltage coefficient

The way you could define these components in the database is quite simple. One proposal is to define the PSpiceTemplate in the schematic part and the other properties in the database. Then you will be able to simulate DC Analysis, AC, Transient, Monte Carlo, Worst Case, etc.

You find an example with European schematic parts in the folder
../FlowCAD_AN_PSpice_CIS/Ideal_RLC/European_RLC.OLB

3.2 C and L with Parasitics

In some precise circuits it may be important to define the passive components with their real parasitic parameters, to make the simulation more accurate. In such case, C and L are defined

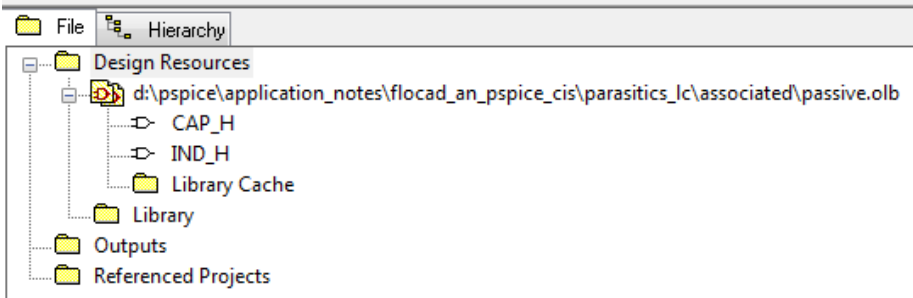
using a PSpice Model. It means, that you could have a subcircuit as a text file like the one shown below (where ESL, CAP, ESR and LEAK are the parasitic parameters):

```
.SUBCKT REALCAP 1 2 PARAMS: ESL=1P CAP=1N ESR=1M LEAK=1G
R_R1      N00127 N00131 {ESR} TC=0,0
L_L1      N00131 2 {ESL}
C_C1      1 N00127 {CAP} TC=0,0
R_R2      1 2 {LEAK} TC=0,0
.ENDS
```

Consider that you have your schematic part for C or L without the PSpice Properties defined. It means, that such C and L are not properly defined for simulation. The procedure to associate a PSpice Model to such schematic parts is:

Steps to Associate PSpice Model

- Open your library where C and L are considered:



- Make double click on IND_H and then click on **Options > Part Properties**. You will see that this schematic part is not prepared for simulation, because **Implementation Type**, **Implementation** and **PSpiceTemplate** properties are blank.

Package Properties

Part Numbering:

Package Type:

PCB Footprint:

Part Reference Prefix:

Section Count:

Part Aliases:

Part Properties

DIST:

Tolerance:

Name:

suffix:

Implementation Path:

Implementation:

Implementation Type:

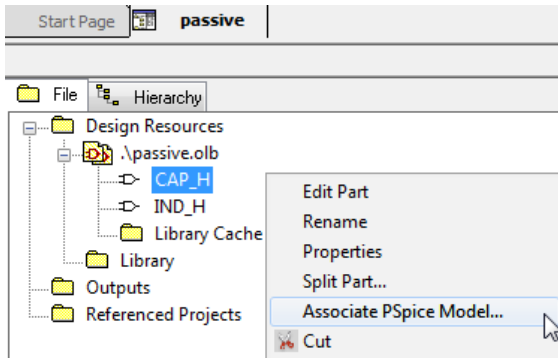
Value:

Pin Name Visible:

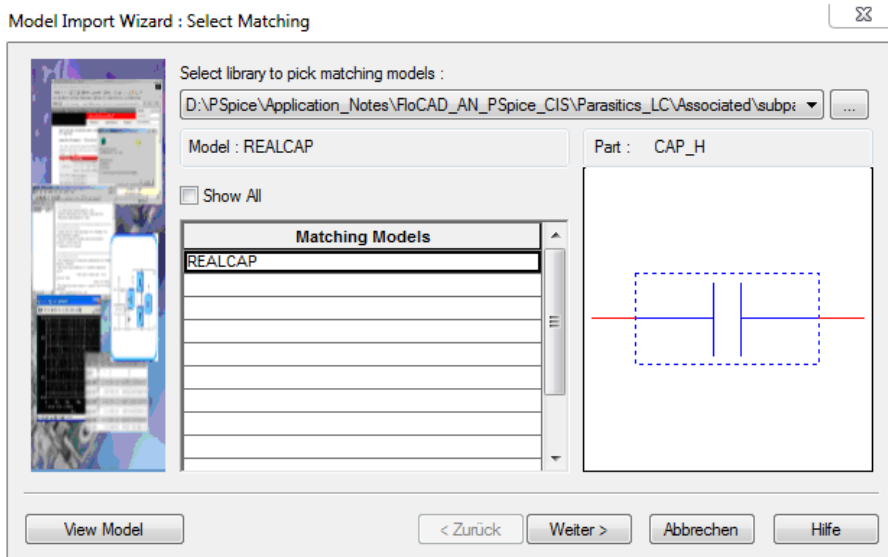
Pin Number Visible:

Pin Name Rotate:

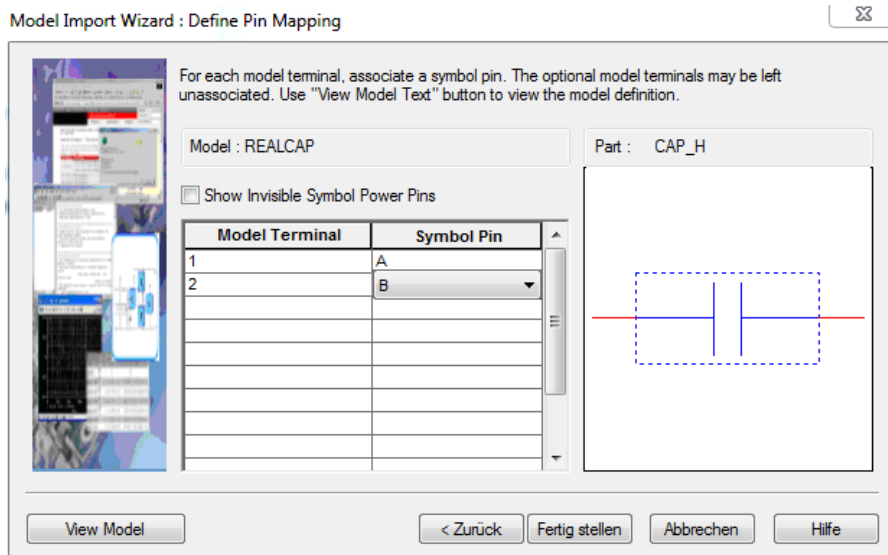
- Close this window, select the inductor and then click right mouse button. Click on **Associate PSpice Model**.



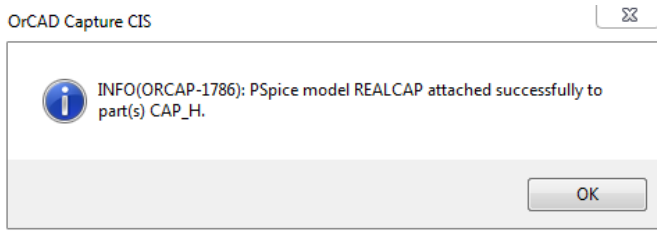
- Select the library where it is located the PSpice Model that you want to associate with this schematic part. In this case subcircuit.lib and the PSpice Model names realcap.



- Click on **Next**.
- Associate Model Terminal with Schematic Part pin:

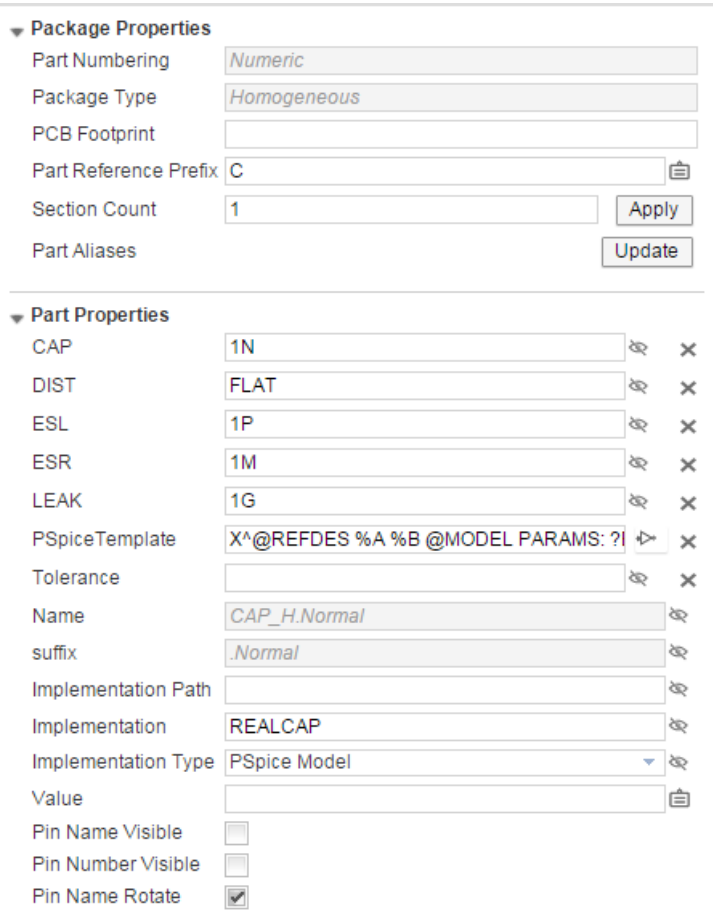


- Click **OK**.



Your schematic part is now available for simulation.

- Make double click on the capacitor and observe its properties. PSpiceTemplate, Implementation, Implementation Type and the parasitics parameters were automatically defined:



Now you can already define this new schematic part in the database. In it you should define new columns, where the parasitics with particular values are considered. When you place such component, values of database are transferred to the design. If it is helpful, open the sqlite database located under `../FlowCAD_AN_PSpice_CIS/site_172_internal/site_172/library/de_cis_172/ FC_lib_172.db` in order to see how it was done.

On the other side, you find an example of C and L without PSpice properties and PSpice Model under the folder `../FlowCAD_AN_PSpice_CIS /Parasitics_LC/Original/Parasitics.OLB`. You could associate them and define them in your database.

3.3 Association Schematic Part with PSpice Model

It was already described in the previous point **Steps to associate PSpice model**. In the moment you have a schematic part without a PSpice Model (for example a transistor, an opamp, an IC, etc.), all you have to do is to associate PSpice Model to this part, so that the pin mapping and the PSpice properties are defined automatically.

In the database you do not have to add any new column with properties, unless these have to be used as parameters by the PSpice Model (as it was shown in the previous point).

3.4 Download PSpice Models

The steps you have to follow are the same ones than in the previous point (associate PSpice Model).

3.5 Homogeneous Parts

PSpice supports this option. Suppose you have an IC package, where there are four identical opamps. All you have to do to use homogeneous parts with CIS and PSpice is to associate the schematic part with the PSpice Model (as it was shown previously). Then automatically each opamp of the package will have available the PSpice Model and it will be possible to run a simulation.

You can find an example in directory `../FlowCAD_AN_PSpice_CIS/Homogeneous/Original`.

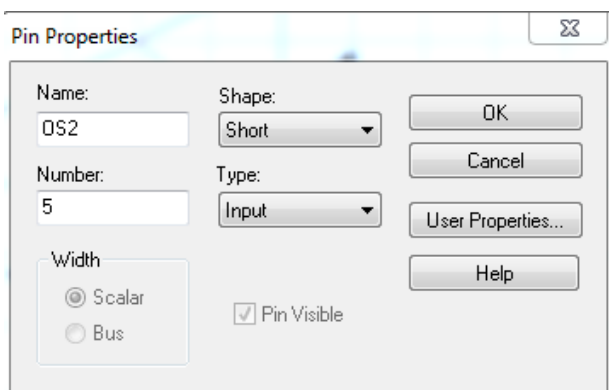
3.6 Heterogeneous Parts

It is **not** supported for simulation (although there is a workaround). Your feedback is important in order to find a solution. It means, if you use this flow, just let FlowCAD know how you do it and how you work in order to analyze the situation and find the best solution.

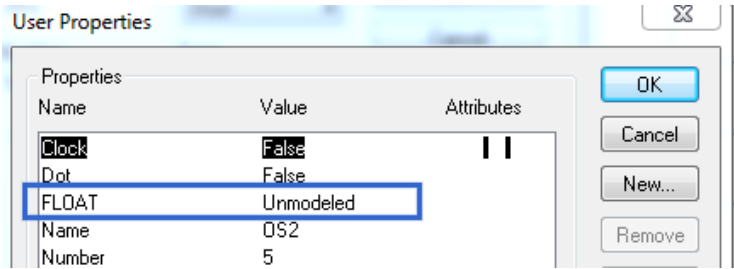
3.7 Schematic Parts With More Pins Than Pins Has the PSpice Model

Consider you have a schematic part which has more physical pins than pins are needed for simulation. In order to allow simulation, you have to define such pins as Unmodeled.

In order to understand this better, consider as example the operational amplifier ua741. This package has 8 pins, but for simulation just 5 pins are needed (in+, in-, V+, V- and OUT). In the schematic part click on each pin that is not used for simulation.



Then click on user properties and create a new property with the name FLOAT and the value Unmodeled:



Doing this, it will be possible to simulate schematic parts which have more pins than pins are used for simulation.

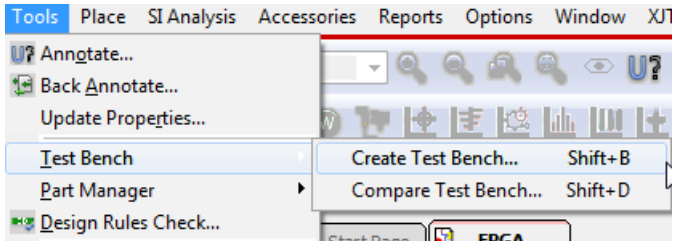
Find this example under `../FlowCAD_AN_PSpice_CIS/More_Pins/Example.olb`.

4 How to Simulate a Project Designed for Schematic and for Simulation?

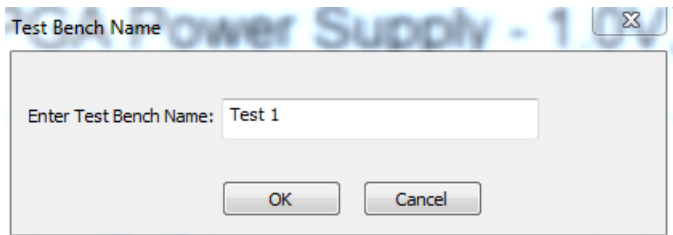
In this type of projects there are blocks that are not needed for simulation. For example, you could have DDR2, DDR3, FPGA, Power Supply, etc and only the power supply wants to be simulated. The **question** would be: Do I have to create a new project where the Power Supply is designed to be simulated? The **answer** is no, it is not necessary. If you are using the schematic – simulation – layout flow with CIS, the components that you placed for designing the power supply block will be already suited for simulation. All you will have to do will be to create a Test Bench for simulation. For that, the steps you have to follow are next ones.

Steps to Create a Test bench

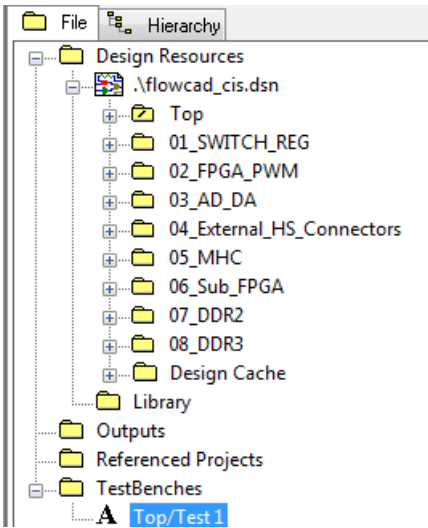
1. Open your project and the project manager click on **Tools > Test Bench > Create Test Bench**



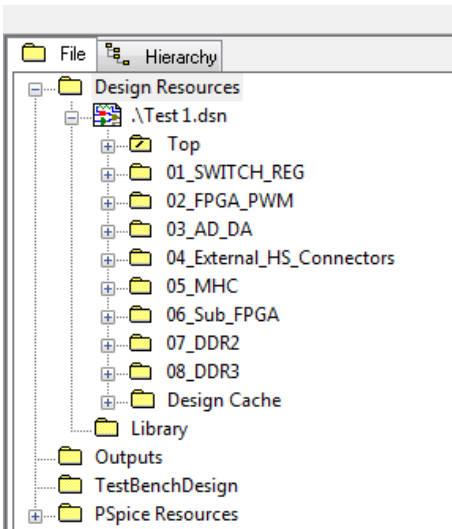
2. Automatically a window will pop up. Write the name for the Test Bench:



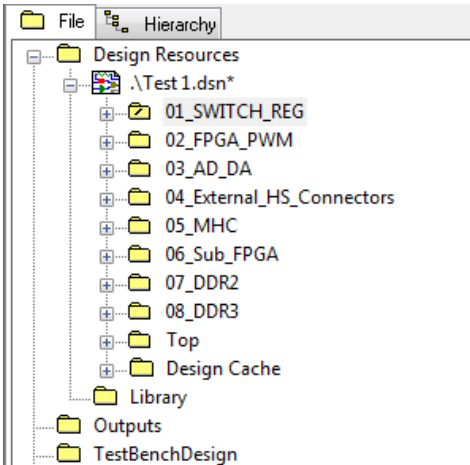
3. The Test Bench will appear in the project manager tree:



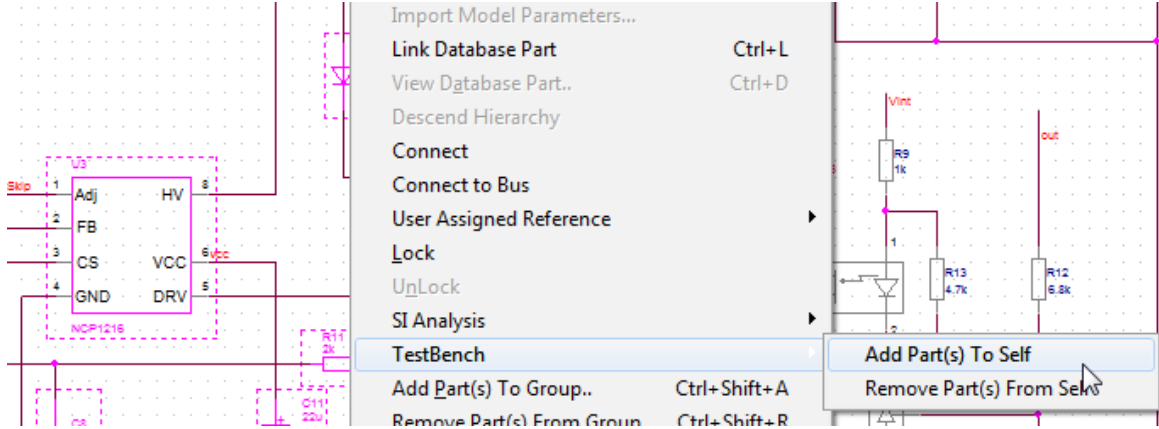
4. Make double click on it. Automatically a new design with the name test.dsn will pop up. This design is identical to the original one.



5. Make root the schematic you want to simulate. In this case 01_SWITCH_REG.



- 6. Open the circuit of this schematic and you will see that the whole components are grey. It means, that they actually are **invisible** for the simulation. So, if you want to consider them in the simulation you have to select each one you need and with RMB click on Add Parts to self.



- 7. After this step, these components will be colored again. It means that they are no longer invisible and they will be used for simulation.
- 8. Now you can place sources or whatever you need for simulation.
- 9. Create simulation profile, simulate and analyze the results.
- 10. Follow the same steps if you now want to simulate another block (schematic) of your design.

5 Feedback

As it was said in the introduction of this Application Note, you can find in this document general possibilities to configure schematic design, simulation and PCB layout using CIS in the same flow. If you consider that there is something you need and it is not considered here, or you have difficulties to do something, just let us know. Your feedback is important to improve this flow.