Quick Start OrCAD PSpice

Version 17.2

Contents

Preliminary Notes

- This documentation applies to the first-time users of the PSpice simulation software, in particular the DEMO version. It should not be understood either as a training manual or as a complete operating manual.
- Basic knowledge in electronic circuitry is required.
- Due to the brevity and compactness of this documentation, it is not possible to address all existing functions and their acuteness. Please refer to **Help > PSpice Documentation.**
- After some preliminary information on the software, the instructions begin with the circuit diagram. This circuit will be showed using the American and the European symbols.
- Using a simple circuit diagram for a power supply, the most important functions and steps are presented and explained with PSpice, which allows the first-time user to gain a first impression with a minimum effort at the time of the initial training and, if necessary, to cope with the first tasks.
- To get started, just start the DEMO version and unzip the PSpice_Demo.zip file to a folder of your choice.

Introduction

System Requirements

Operating System	Microsoft Windows 7 Professional, Enterprise, Ultimate or Home Premium (64-bit) (All Service Packs); Windows 10 (64-bit); Windows 2008 Server R2; Windows 2012 Server (All Service Packs)
Minimum Hardware	Intel Pentium 4 or AMD Athlon XP 2000 with multi-core CPU 8 GB RAM Virtual memory at least twice physical memory 50 GB free disk space 1024 x 768 display resolution with true color (16 bit color) Broadband Internet connection for some services Ethernet card Three-button Microsoft-compatible mouse
Recommended Hardware	Intel Core 2 Duo 2.66 GHz or AMD Athlon 64 X2 5200+ 8 GB RAM 500 GB free disk space 1280 x 1024 display resolution with true color A dedicated graphics card Broadband Internet connection for some services

OrCAD Lite Limits

The free version is a fully functional design package including PSpice, which is limited only by the number of components, component pins, network nodes and integration of additional simulation models.

You can design, simulate and create small circuits in the schematic, create your layout, and create output for your production. This data can be stored.

Larger circuits and layouts can be viewed, but **NOT** stored.

In the current 17.2 version the following limits must be observed:

Capture

Maximal 60 parts and 75 nets, including the hierarchical blocks in the design, no part with more than 100 pins, maximal 1.000 parts in the CIS Database, FPGA flow is not available, you cannot validate Electrical Csets, Altium translator is not available.

PSpice

Maximal 75 nodes, 20 transistors, no sub-circuit limits but 65 digital primitive devices, and 10 transmission lines (ideal or non-ideal) with not more than four pairwise coupled lines, Model Editor limited to diodes, all libraries are included, you cannot use level 3 of Core model, MOSFET BSIM 3.2, or MOSFET BSIM 4, only simulation data created using the Lite version of the simulator, only power transformers in Magnetic Parts Editor, maximal nodes in digital circuit 250, non-ideal Tline is limited to 4, PSpice DMI models are not supported, IBIS import is not supported, PSpice SLPS flow is not supported.

Installation and Settings

- The demo version is included in the full version 17.2. If you have installed the 17.2 full version, a demo version will automatically be switched over when no license server has been installed.
- Inserting the DVD, the setup program will start automatically. If the Autorun function of your computer is deactivated, use Windows Explorer to open the folder of the DVD and start **setup.exe**.
- Start the product installation. Follow the instructions in the installation program. Do not install license manager!

An installation guide can also be found in this DVD or at http://www.FlowCAD.de/Application_Notes (Quick Start PSpice 17.2 Lite).

OrCAD allows to use a variety of personal settings, from product configuration to design templates to color selection of the design elements used.
These possibilities are not discussed here as they go beyond the purpose of this documentation. It is only mentioned that many of these settings are stored in the corresponding INI files.

Basic Handling in Capture / PSpice

Operating Concept

OrCAD Capture and OrCAD PSpice are basically menu driven.

It is highly recommended to use only underline as a special character in the path and name of the design.

All inputs or commands are made by one of the following options:

- Pull-down menus
- Icons
- Shortcuts
- Pop-up window
- Command window
- TLC functions

In Capture and PSpice context-sensitive menus are used. This means that depending on the selected elements, workspaces or commands, the resulting pop-up windows change, or the pull-down menu changes its appearance (Capture).

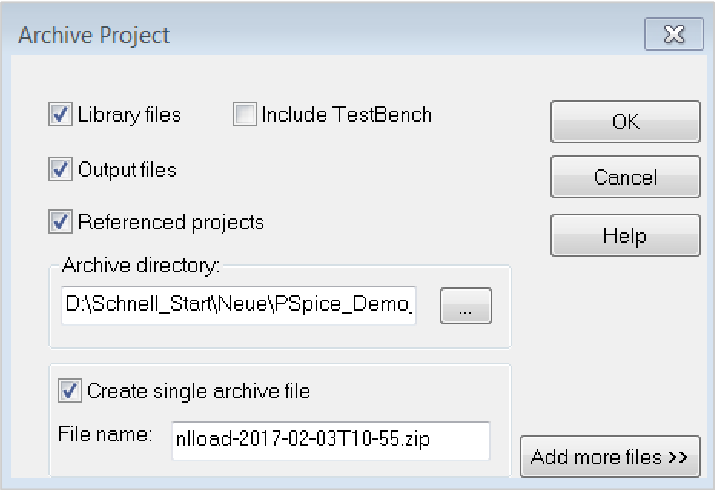
Main File Extensions in OrCAD (PSpice)

.OPJ	OrCAD Capture Project File
.DSN	OrCAD Capture Design File
.DBK	Design Backup
.OLB	Capture / PSpice Symbol Library
.UPD	Property Update File
.DRC	Design Rules Check
.BOM	Bill of Materials File
.EXP	Export Properties File
.LIB	PSpice Model Library File
.DAT	Probe Data File
.OUT	PSpice Output File
.SIM	Simulation Profile
.NET	PSpice Netlist
.PRB	Probe Configuration File
.STL	Stimulus Library File
.CIR	PSpice Circuit File

Transfer of Projects in OrCAD (PSpice)

- **Archive command:** Use this option to transfer projects. Select in OrCAD Capture **Design_Name.dsn > File > Archive Project…**

  It copies all necessary files into the new area.



- **Manual transfer:**
  - .opj
  - .dsn
  - .stl
  - .sim
  - .lib

All other files are automatically generated form the above files during the generation of the netlists or the simulations.

Unit of Measurement in PSpice

PSpice only calculates with pure numbers, not with units.

PSpice knows however these expressions.

1f	(femto-)	= 10exp-15
1p	(pico-)	= 10exp-12
1n	(nano-)	= 10exp-9
1u	(micro-)	= 10exp-6
1m	(milli-)	= 10exp-3
1K	(kilo-)	= 10exp3
1Meg	(mega-)	= 10exp6
1G	(giga-)	= 10exp9
1T	(tera-)	= 10exp12

Tip

- Units such Ohm, Volt, Farad, Henry, etc. are not considered. They are used only for the better readability of the relevant circuit.
- If an input is made after such a measure or expression, PSpice interprets this as a comment.

  1KOhm = 1K

- PSpice is not case-sensitive 1m = 1M „M“ or „m“ in PSpice mean „milli“ in contrast to the usual form „MEGA“ in Europe. For Mega the prefix „Meg“ must be used.

Proposed Circuit (I)

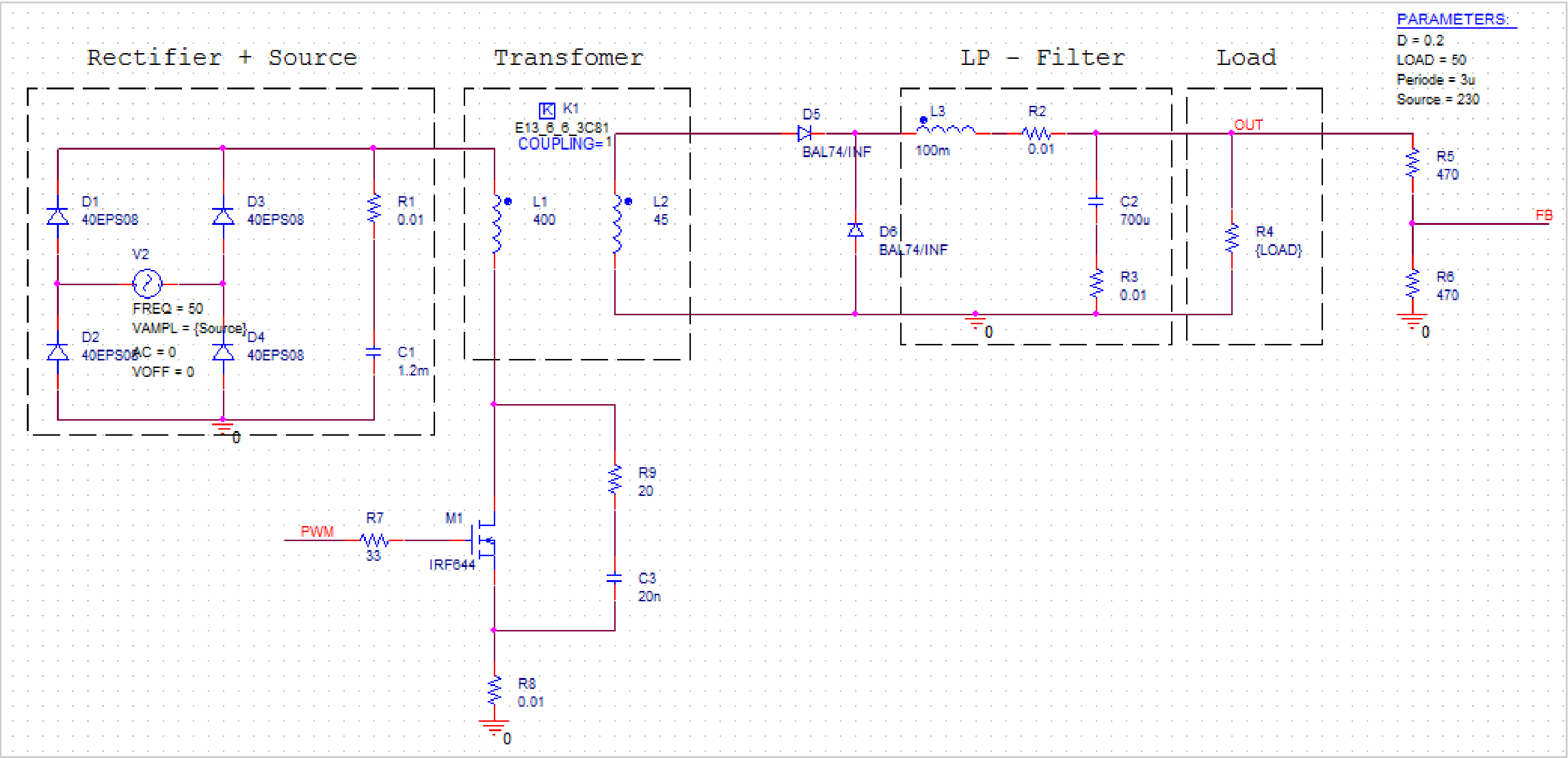
- The aim of this PSpice introduction is to simulate a complete circuit, particularly a Switched Mode Power Supply (SMPS), where a rectifier, a PWM Control and a transformer are used. The illustrated circuit includes three additional areas (source, filter and load) required to simulate the overall circuit.
- In the **solution** directories, there is an example for simulating the overall circuit shown. You will find the European solution. Default Cadence symbols are American ones, that is why, you will use an external European symbol library for R, L and C (Europe.olb).
- On the following pages, you will find a brief overview of the actions you can take to implement the circuit diagram below.

Tip

The correct definition of the mass potential must also be used. > 0

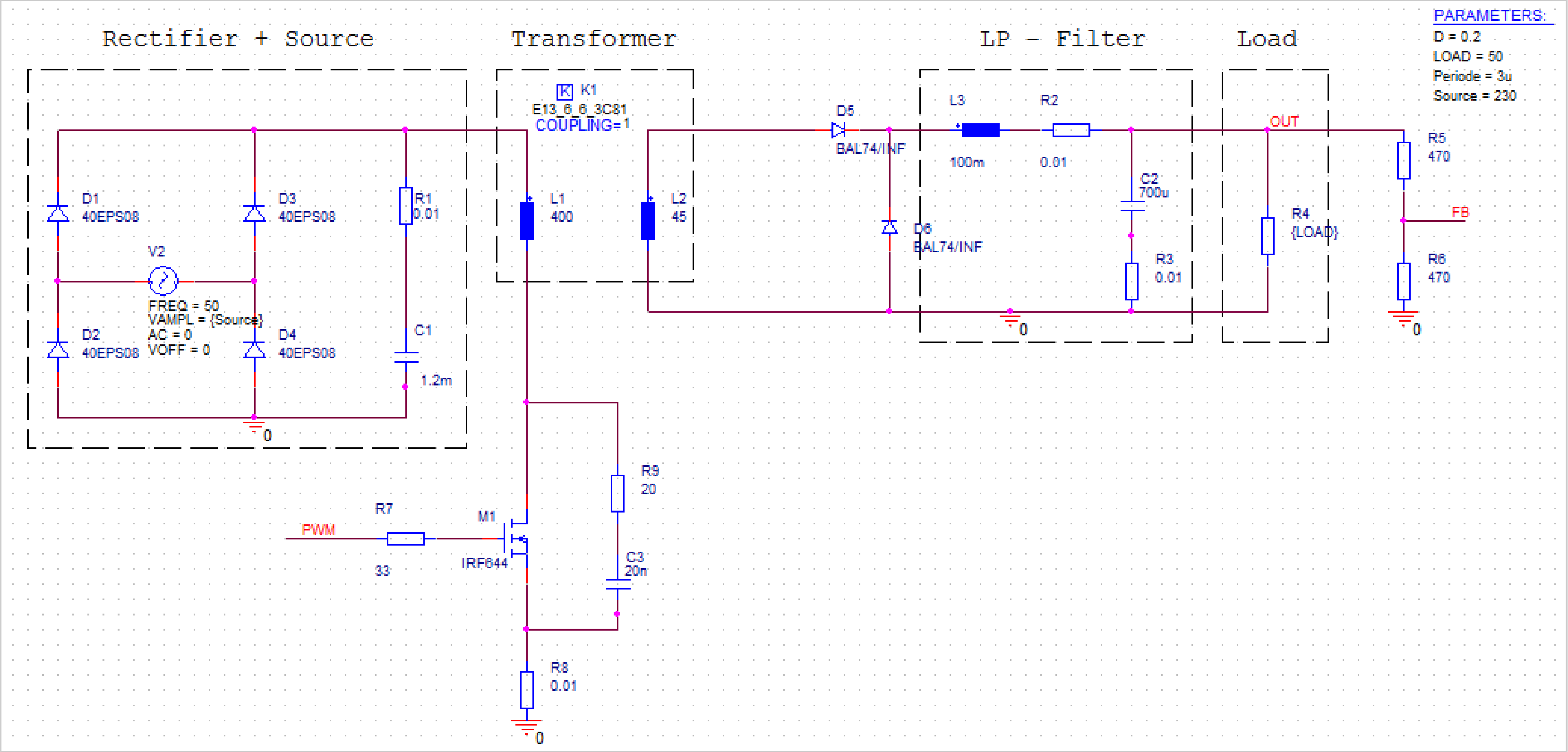
Proposed Circuit (II)

American Style



Proposed Circuit (III)

European Style



Run a PSpice Simulation

There are two ways to start a simulation in PSpice:

- Opening directly PSpice:
  - **Start > All Programms > Cadence Release 17.2-2016 > OrCAD Products > PSpice AD**
  - However, PSpice requires the netlist file <*.net> and the circuit file <*.cir> of the circuit to be simulated.
- Directly from OrCAD Capture
  - **Start > All Programms > Cadence Release 17.2-2016 > OrCAD Products > Capture**
  - This is the preferred way to start a simulation by first using Capture to design the circuit and creating the simulation profile.
  - PSpice simulation is started automatically from Capture using related Run-Icon. Netlist for PSpice is created automatically in the background.

FlowCAD

Capture Session Frame

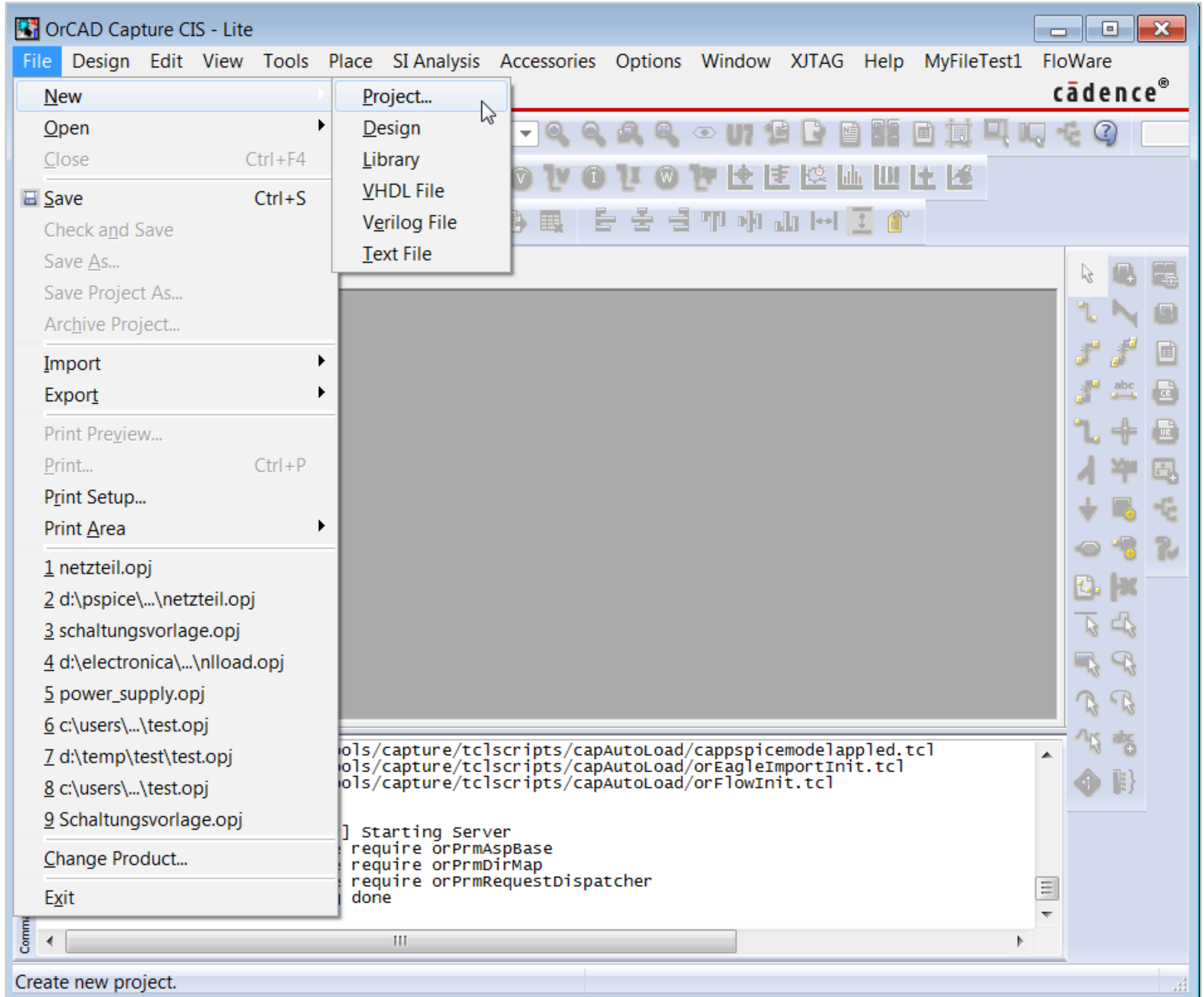
All subsequent processes are started from this session frame window.

File > New > Project ... creates a new project where the design is defined.

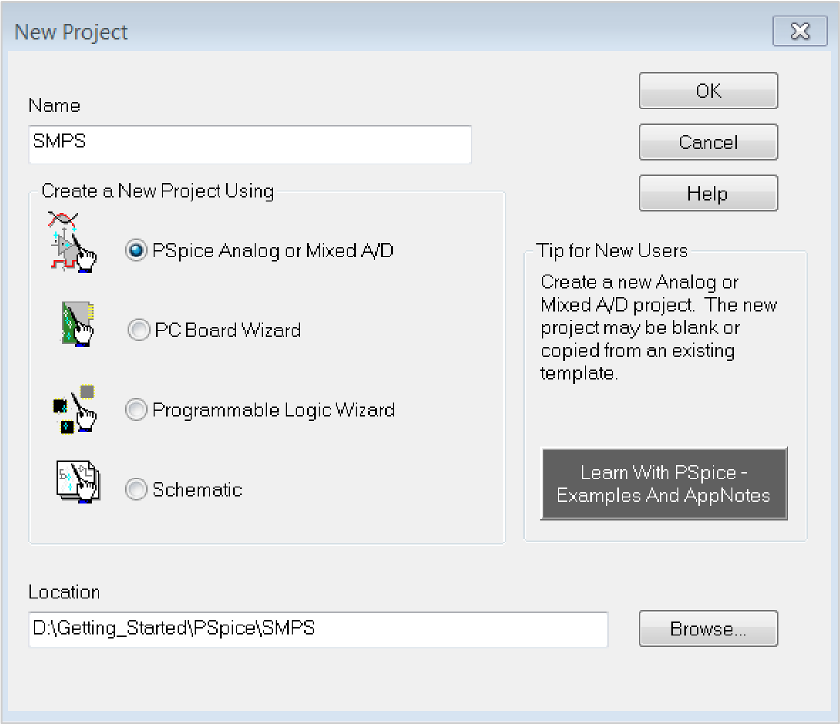
The following menu appears.

NOTE

If you click on **File > Open > Demo Designs**, you will be able to open examples with documentation.



New PSpice Project



In the upper field under Name enter the name of your project, e.g. SMPS.

As project type, select Analog or Mixed A/D as we want to perform a simulation with this project.

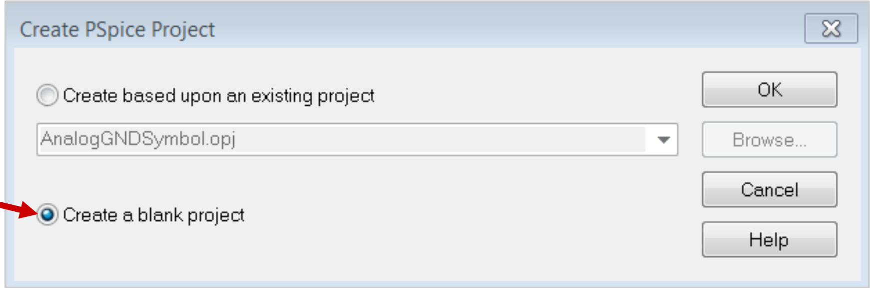
In the lower field under Location, select the folder in which your new project should be stored. It is highly recommended to use the project name again as a folder entry. As a result, all project data generated for this simulation project are stored in this folder, which considerably increases the overview.

Then we click OK.

NOTE

In the right side at the bottom you find an option called **Learn with PSpice**. If you click on it, you will open the Learning PSpice documentation.

Since we do not have a simulation project yet, we are generating a completely new project.



Project SMPS

Done!

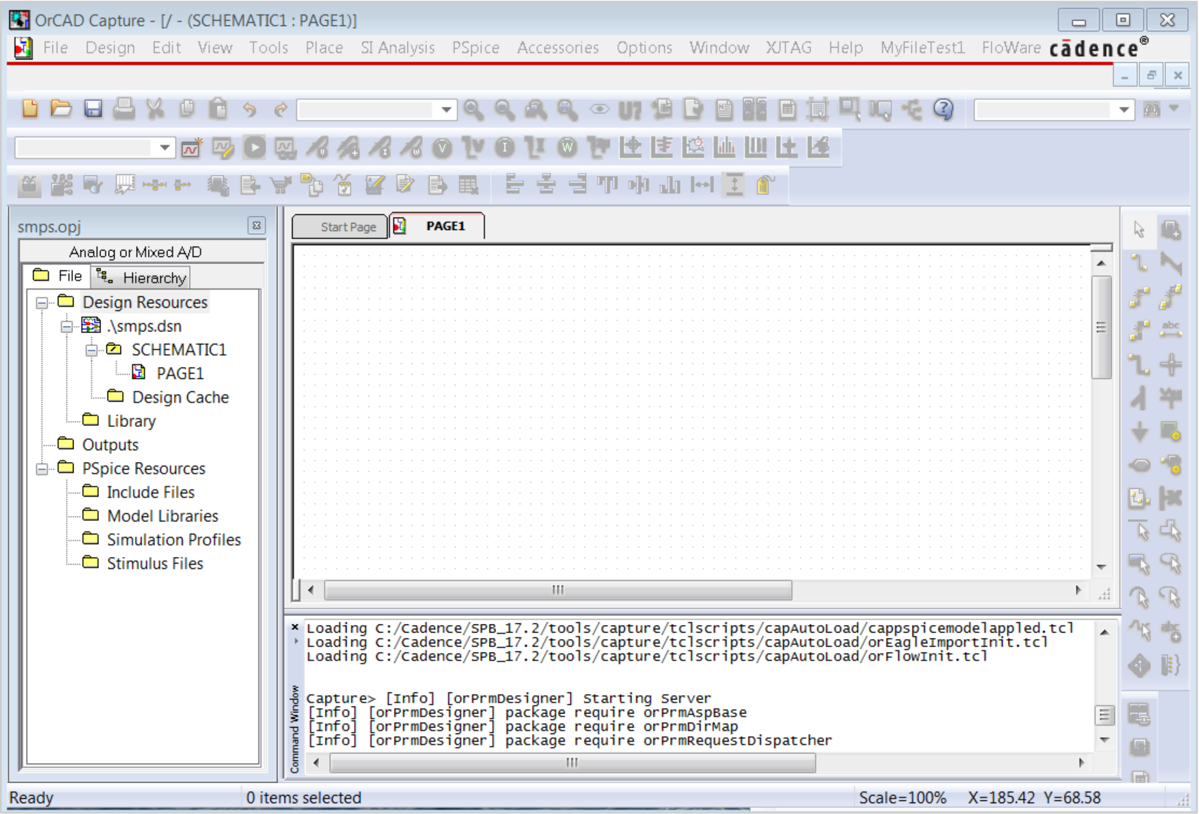
A new project called **SMPS** with a design of the same name **smps.dsn.**

At the same time, the first page of your design was opened with the name PAGE1. Similarly, the drawing frame and the head were automatically placed. Other view-presets are possible.

Please note the folder structure in the project manager on the left side of the image.

The folder structure is virtual. It only exists within the project manager. The PAGE1 located under Schematic1 is only found within the smps.dsn file.

The icons for component placement and wiring are already visible on the right-hand side.



Place Components

To place new components, you can use the following options:

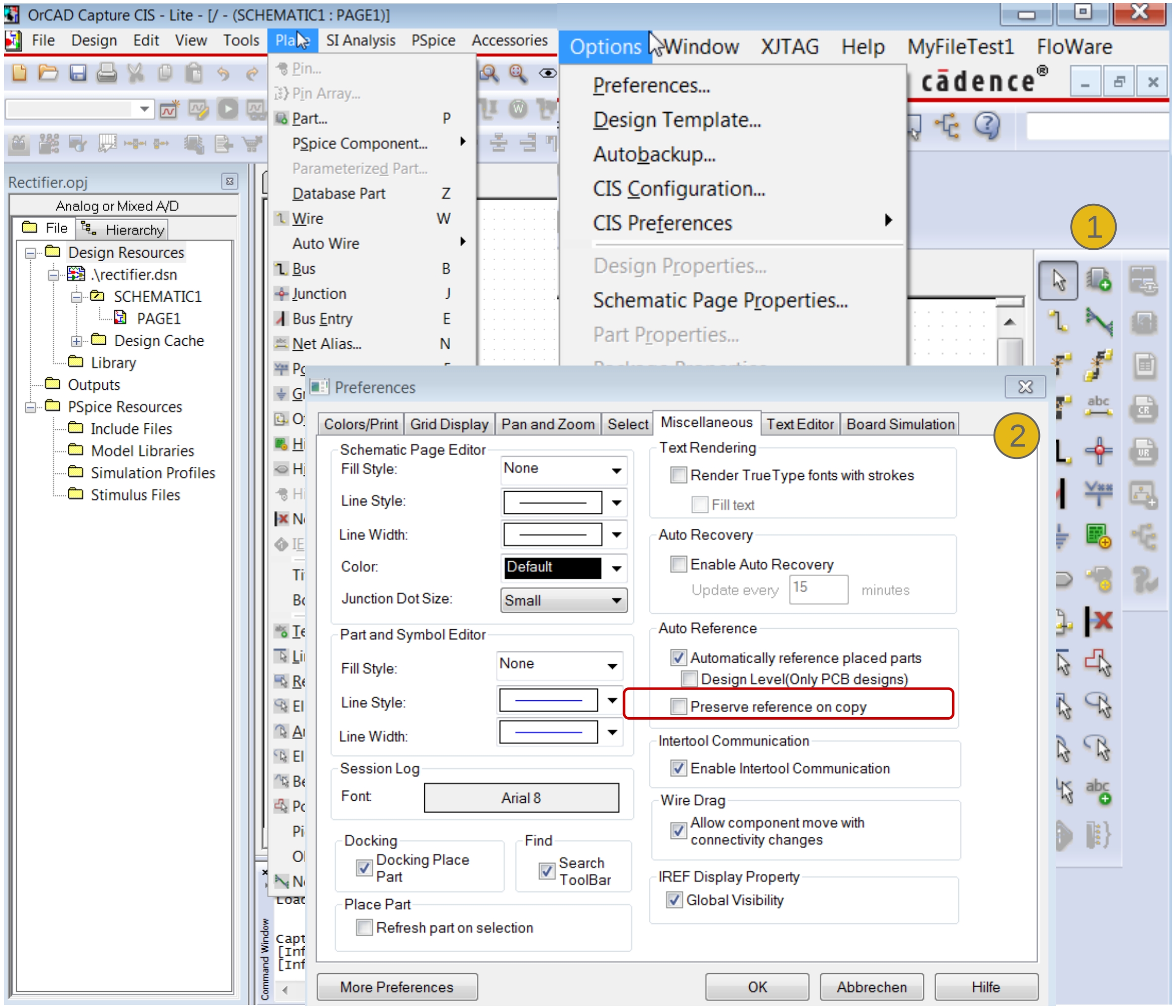
- **Place > Part** ... using the pull-down menu

or

- P (keyboard)

or

- Place icon  on right edge



Tip 1
For the Schematic icon bar to be active (right margin), one of the pages in the Schematic must be opened and activated.

Tip 2
Auto Reference allows you to automatically annotate component references during placement.

Library Assignment

After placement command, you will see the Place Part menu.

Under **Libraries**, you can select one or more libraries to search for your part (part).

The part description is entered under **Part**. This already acts as a filter, but **without** wildcards "*".

The search result and the corresponding library are output in the **Part List.**

Add Library allows adding libraries to the search path. The default directories with Cadence PSpice symbols are:

<installation>\tools\capture\library\pspice

<installation>\tools\capture\library\pspice\advanls

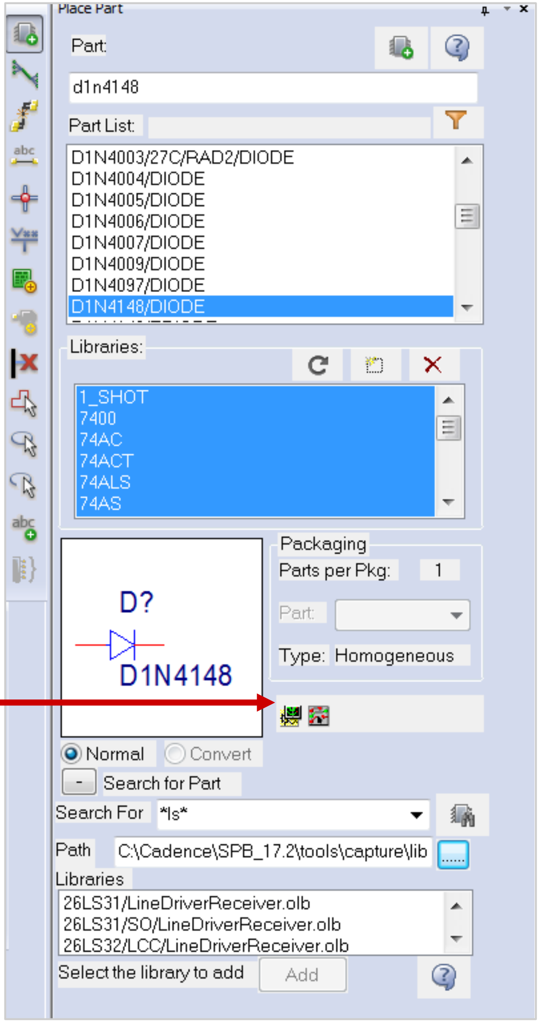
All libraries entered here are added into the pspice.ini file and are available also for all other PSpice projects.

Packaging is used to indicate whether a block consists of several gates (for example, a resistor network).

To the right of the preview window, the two icons indicate whether a PSpice model and / or a PCB footprint is assigned to the relevant component.

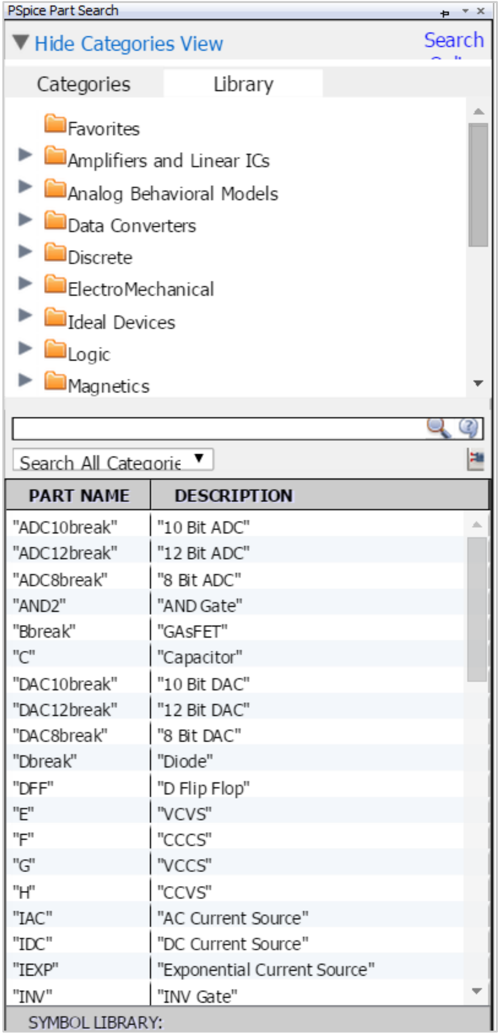
With a double-click in the Part List, you return to the Schematic and can place the component with LMB (Left Mouse Button).

Search for Part allows searching with wildcards.



PSpice Library Browser

- PSpice Library Browser is a container where it is possible to organize and classify parts into categories or libraries the components that can be used in PSpice simulations in order to make easier the search und use of these components in your designs.
- It allows to compare properties of an amount of components in a glance. Moreover, it is possible to have all the components stored in a centralized server allowing all local users to access to this data globally.
- The components included in this PSpice Library Browser at the beginning are components that Cadence delivers by default.
- **Place > PSpice Component > Search**



NOTE

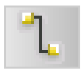
If you are looking for a particular model, but you do not know, where it is located, try looking in this browser.

Wire the Components

After placing, the wiring is done with either:

- **Place > Wire**
- **W (keyboard)**

or

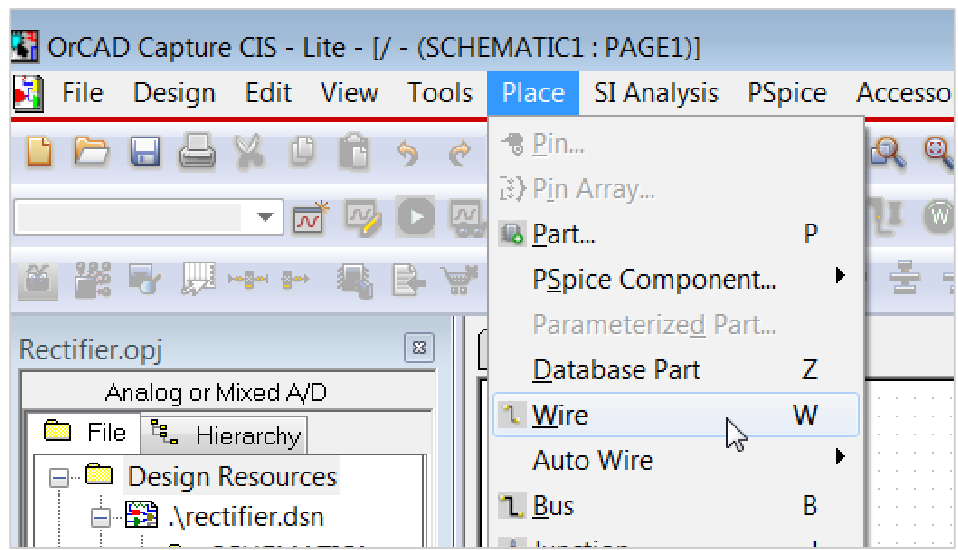
- **Place Wire Icon** 

Pure text notes can be placed with

- **Place > Text** ...
- **T (keyboard)**

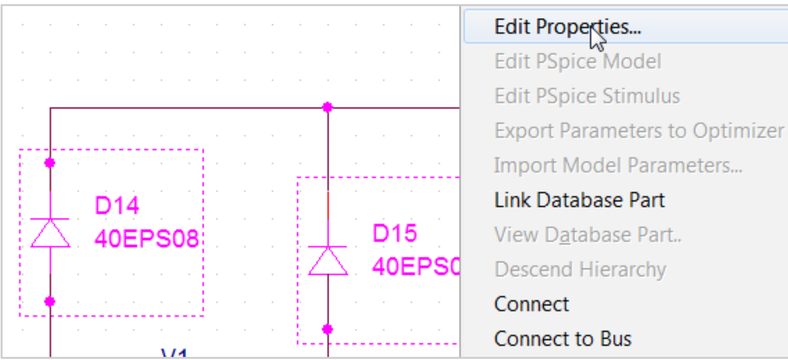
or

- **Place Text Icon** 



Edit Circuit Diagram (Properties)

Edit REFDES and Value at the same time



Tip

It is also possible to display multiple, or all symbols of a page, or even the entire schema, in the property editor.

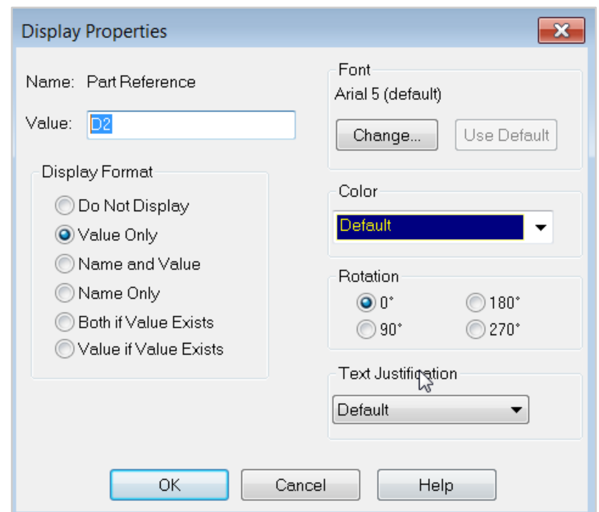
Use Ctrl + LMB-click or **Ctrl + A** to select the components and then **RMB > Properties** ...

In Project Manager, select Page or Design (.dsn), **Edit Object Properties** from the pull down menu.

Edit only REFDES or Value

The selective Display Properties window can be opened by **Select (LMB)** and **RMB > Edit Properties** or by **double-clicking LMB.**

Via **Display Properties**, various settings regarding the visibility in the circuit diagram are possible.



Edit Circuit Diagram (Net Alias)

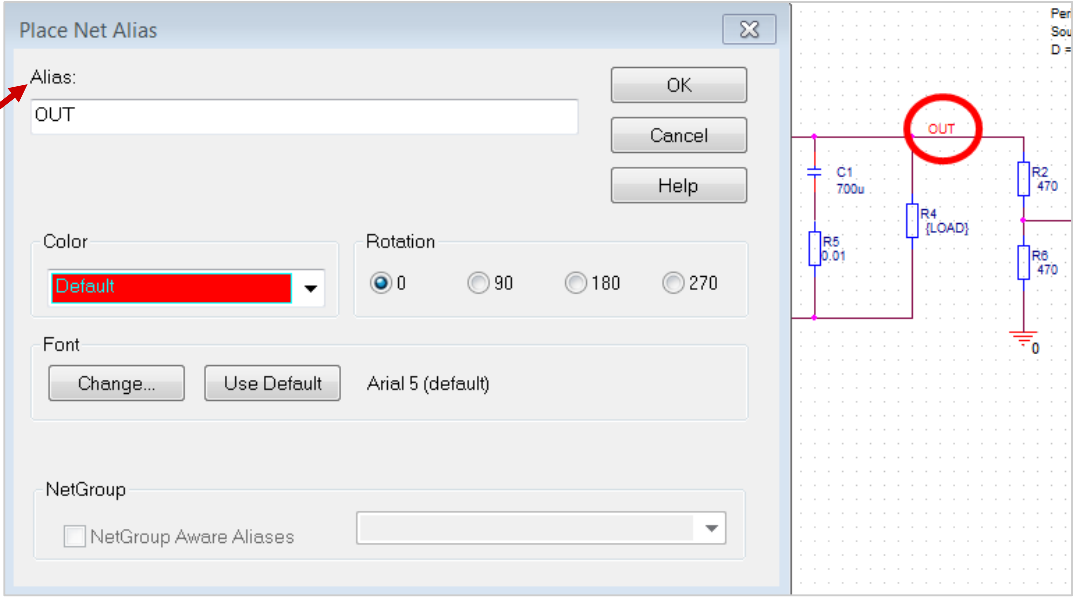
If component pins are to be interconnected on one side, this is done by means of the **Place > Wire** command by pulling wire connections.

Another possibility is to use NetAlias.

Place > Net Alias ...

Place Net Alias Icon

A net name is assigned to partial nets, and the connection of two components is realized. An example is the network called OUT.



Tip

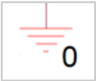
If networks are to be connected over several pages or even designs, then offpage connectors or port connectors must be used. For more information, see the documentation.

Tips for Simulation

To run a simulation of a circuit created with Capture, follow the next tips:

1. All component symbols used should be taken from the PSpice Library.
   E.g. **<Install_dir> \ tools \ capture \ library \ pspice** or
   <Install_dir> \ tools \ capture \ library \ pspice \ advanls
   Capture symbols contain no entry for the simulation models, which are mandatory for a PSpice simulation.

2. For the simulation, a reference potential is required, which is usually represented by a ground symbol and it is found clicking on  and selecting the symbol in the source library.

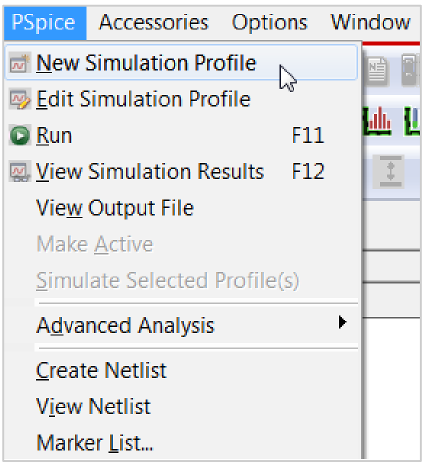
   This must necessarily have the value 0 as a potential name. 

3. Large circuits should first be divided into subcircuits if problems occur.

Types of Simulation

Types of Simulation in PSpice

With PSpice, 4 basic types of simulation are possible. The type of simulation and the parameters for the project in question are set and assigned via the simulation profiles. For all 4 types of simulation you will find examples on the following 4 pages. The Solution / Samples folder contains a completely predefined example.

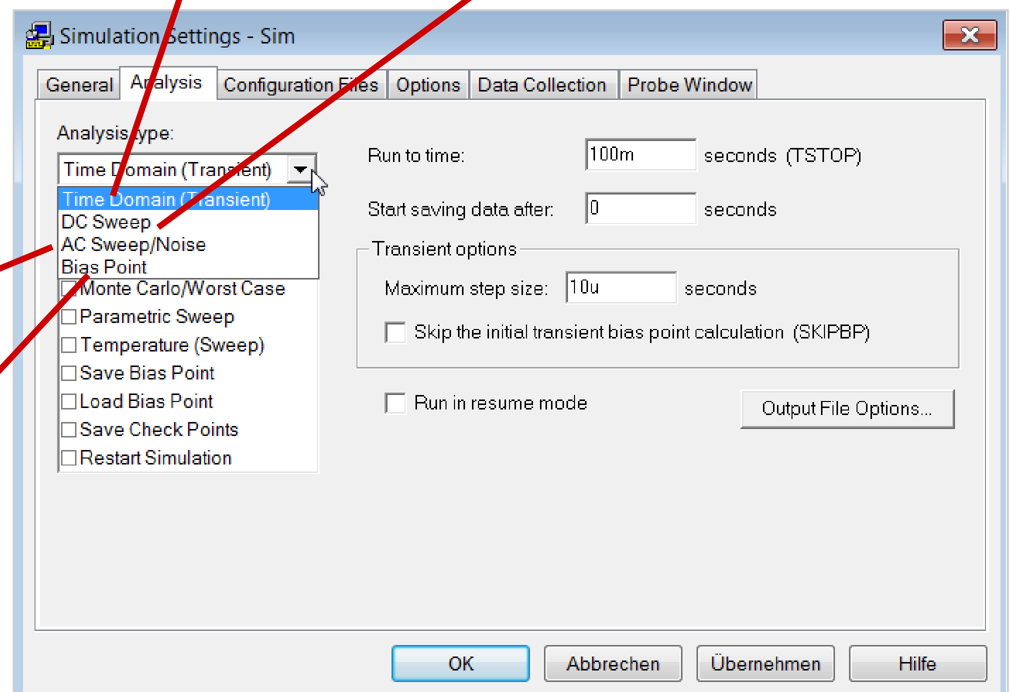


Time Domain (Transient)

Analysis of the time behavior of electrical processes

DC Sweep

DC analysis of all node potentials



AC Sweep

Frequency analysis of voltage and current (amplitude and phase)

Bias Point

DC Current analysis

Bias Point (DC Analysis)

The Bias Point simulation is the simplest simulation mode that can be performed with PSpice. Here, PSpice calculates all DC currents in the individual branches and all DC voltages of the individual nodes with respect to the mass potential which is defined by the placement of the ground symbol.

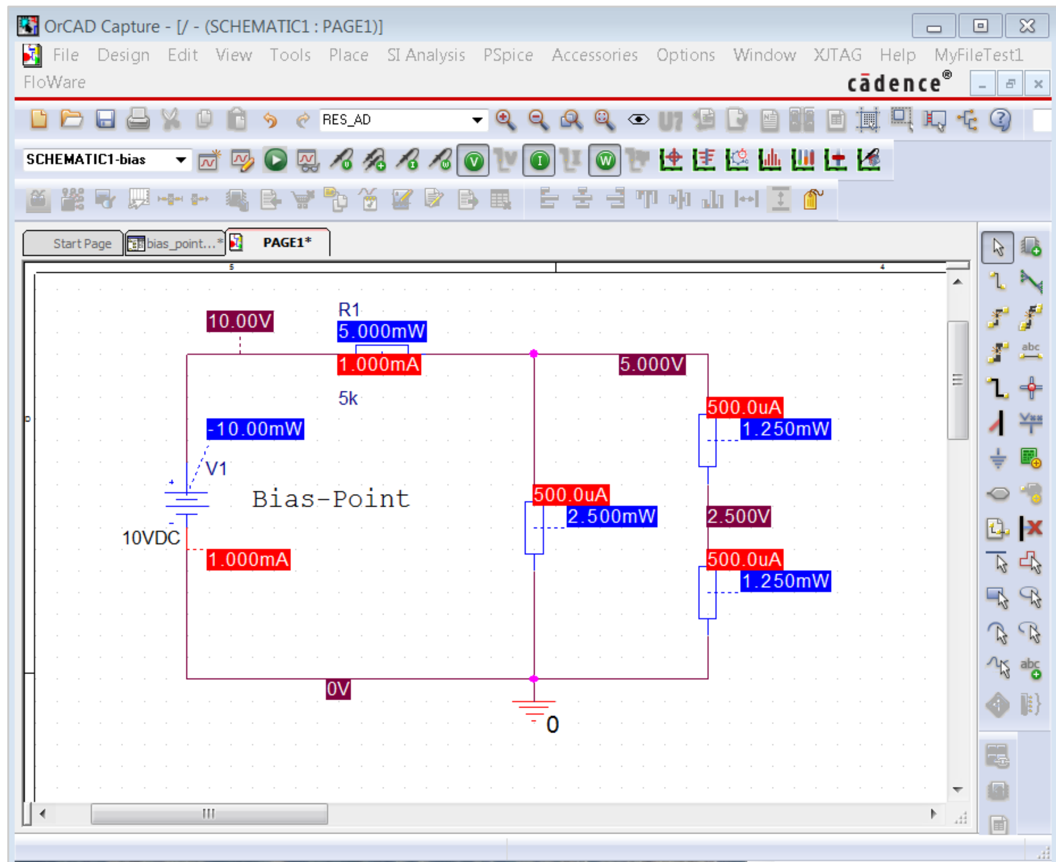
You can also find these results under the help of a calculator, but PSpice can make this much faster and clearer.

The U-I-W values can be displayed over **PSpice > Bias Points > Enable** and using the icons

However, the simulation must be run once before. The opening probe window can then be closed immediately.

Tip

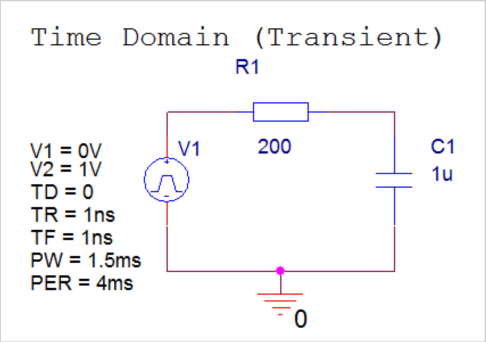
Example bias_point in the folder Solution \ Sample

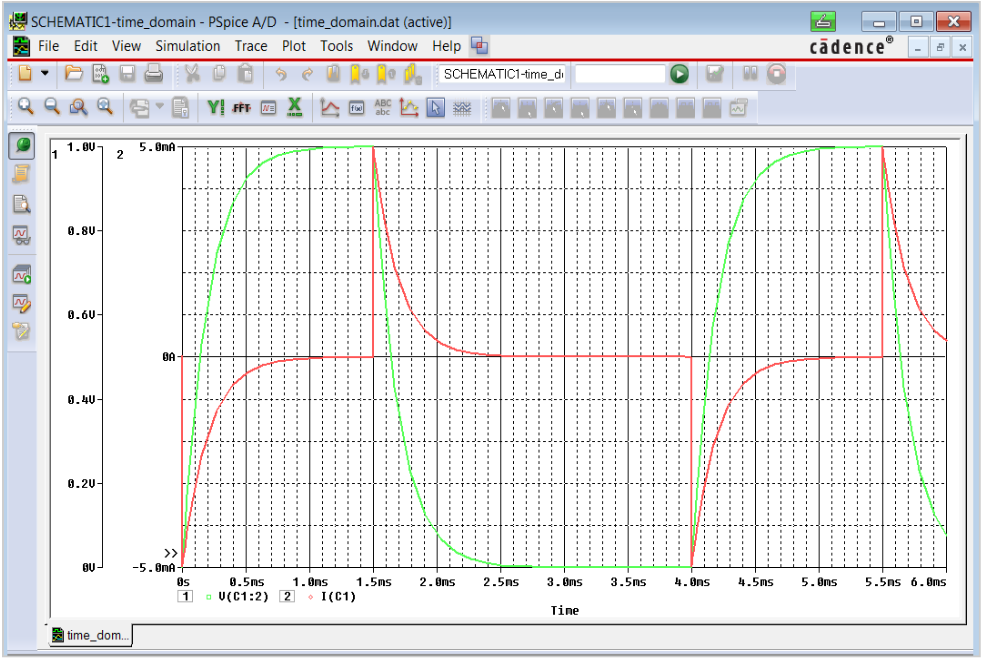


Time Domain (Transient Analysis)

In order to graphically depict the temporal sequence of electrical processes, an oscilloscope is necessary in practice. However, a corresponding circuit configuration must exist in the form of a specific hardware.

If now the time sequence is different, possibly even non-linear components are to be examined, this is actually no longer possible with conventional means (hardware).





Tip

Example time_domain in the folder Solution \ Sample

One of the simplest and most common variants of a time-dependent process is e.g. the charging and discharging operation of a capacitor.

C1 is periodically charged and discharged via a series resistor R1 and a pulse source.

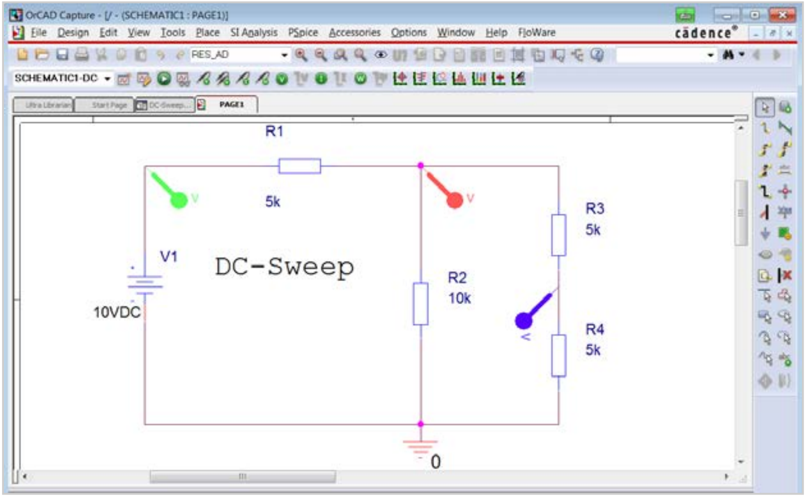
DC Sweep

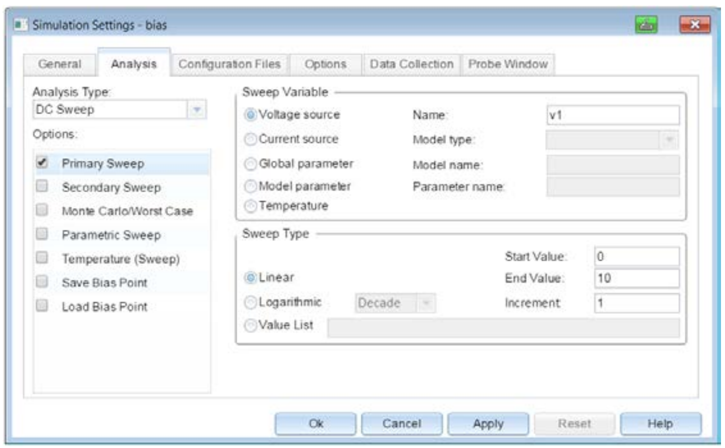
The DC sweep calculates the static voltages and currents of a circuit when source, model parameters, global parameters or temperature are varied within a certain range.

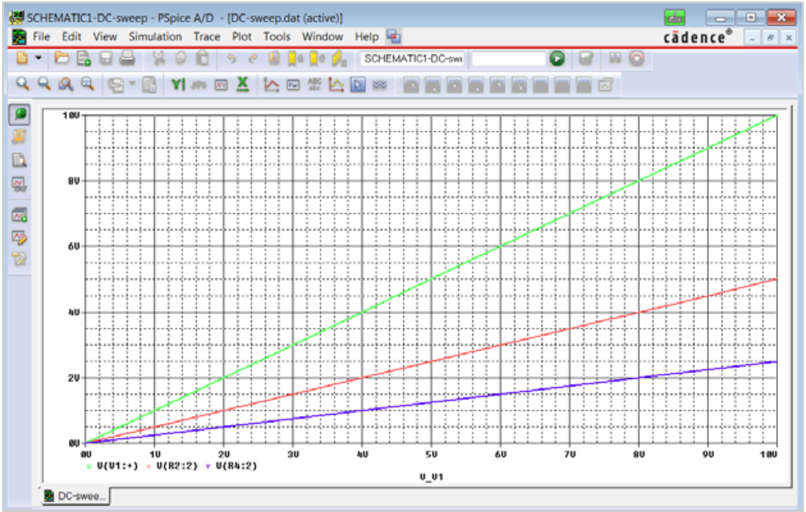
The result can be displayed in the output file or in the probe window.

The right upper window shows only the result of the last sweep (V1 = 10 V), which is only possible via a bias point simulation.

The DC sweep of the voltage source V1 is defined in the simulation profile and the sweep in the probe window is displayed in the right lower window.







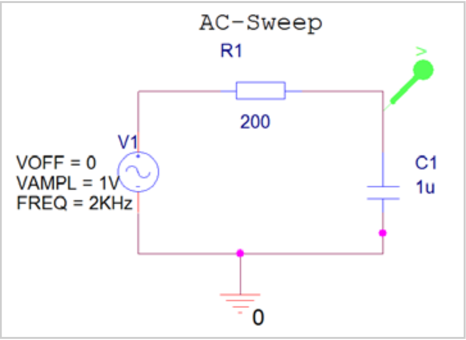
Tip

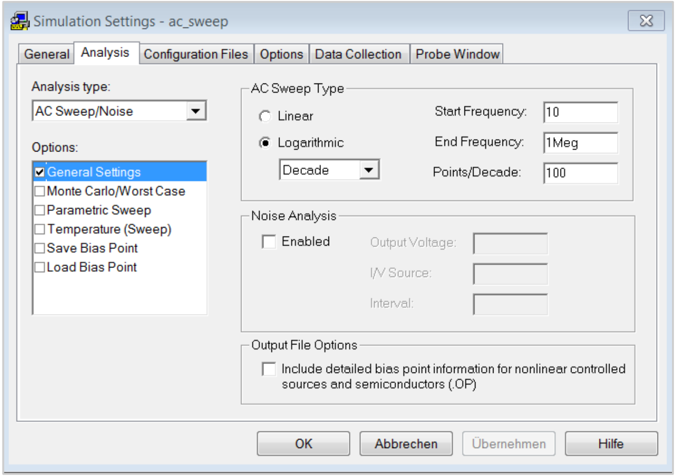
Example DC Sweep in the folder Solution \ Sample

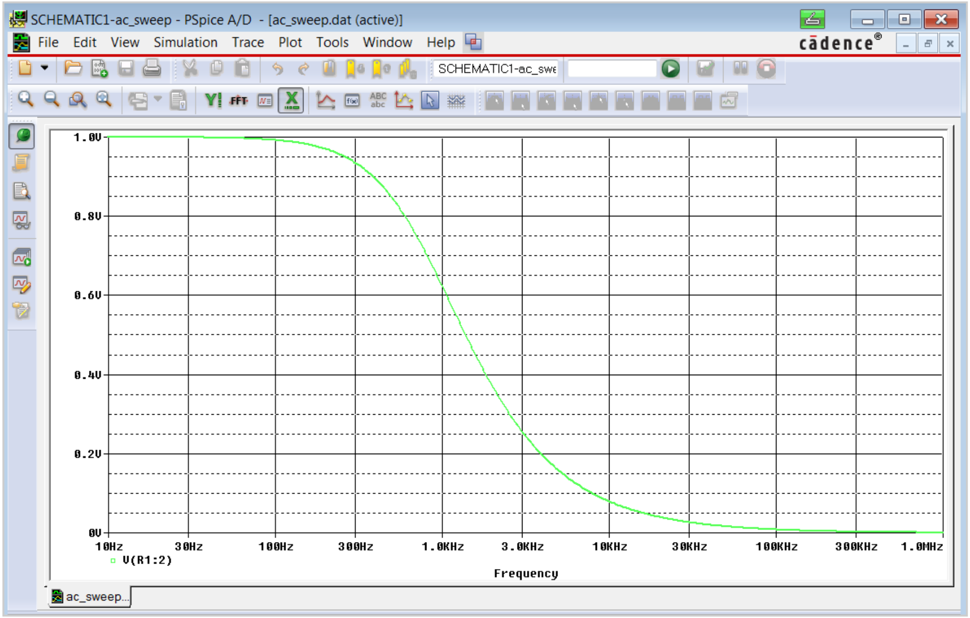
AC Sweep

The AC analysis calculates the small signal behavior of a linear or linearized AC circuit as a function of the frequency. These circuits may consist of linear components (RLC) or / and non-linear components (transistors, diodes, etc.).

The AC sweep calculates the behavior of the circuit over a predetermined frequency range by performing a series of individual AC analyzes at different frequencies.







Tip

Example AC Sweep in the Solution \ Sample folder

Simulation Options

With each simulation it is possible to include different options to run particular conditions:

- Temperature Sweep: Analyze how responds your design depending upon temperature.
- Monte Carlo: Simulate taking into account tolerances or discrete components, semiconductors, etc.
- Sensitivity Analysis: Find out, which components are sensitive or not very sensitive, reduce the price and increase the quality of your design.
- Worst Case Analysis: Analyze the functional limits of your project.
- Parametric Sweep: Vary different parameters and visualize their effects.
- Noise Analysis

Design Example:
Switched Mode Power Supply (SMPS)

Goals

- You will firstly design the SMPS circuit shown previously, where a 50 Hz and 230 V input signal will be converted into 5 V output. In this case a permanent 333 KHz voltage pulse with Duty Cycle 20 % will be used to control the MOSFET, because due to the Lite Version limitations, we cannot use the PWM IC.

- You will define different simulation profiles and analyze the results in the probe window using markers and adding measurements.

Tip 1

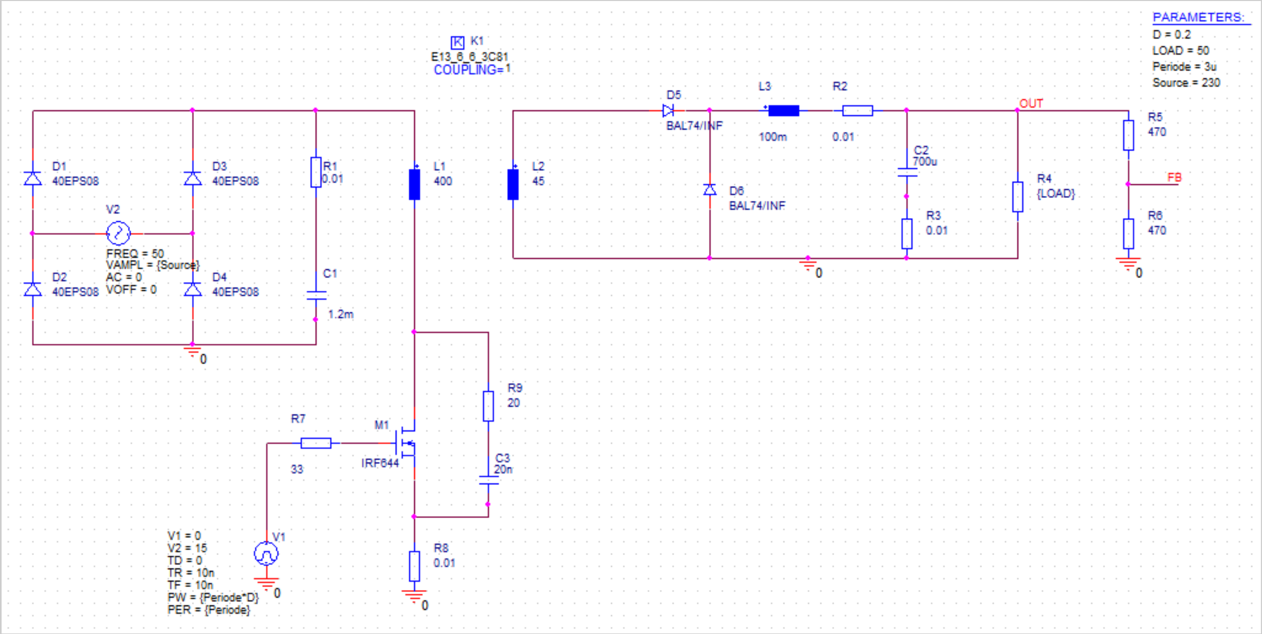
You will find the overall circuit in the **Solution folder.**

Tip 2

For a better understanding of the software, it is recommended to design the circuit yourself following the explained steps.

SMPS – Design (I)

- Please enter the following circuit under the project name SMPS.
- Ensure that the values of the components are set according to the specifications in this circuit to ensure the logical sequence of the subsequent steps of the simulation.
- You will find the procedure for generating a new project on pages 17 and 18.
- Libraries you need to add are: source.olb, irf.olb, Infineon.olb, special.olb, magnetic.olb, PWRMOS.olb
- You have also to add the library Europe.olb that you will find in the unzipped file.
- **Tip 1**
  If you wants to use American symbols, use the R, L and C from the analog.olb library.
- **Tip 2**
  V2 is VSIN and
  V1 is VPULSE.

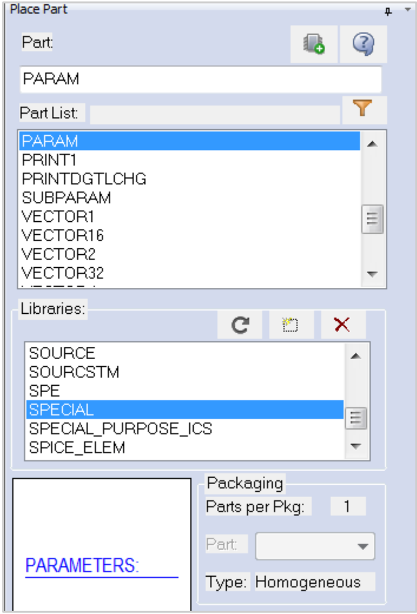


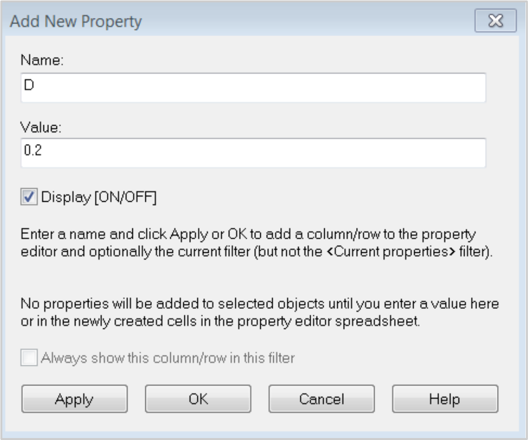
SMPS – Design (II)

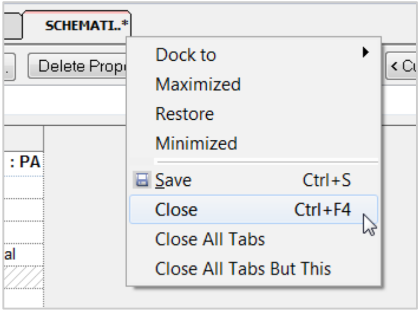
- How can you define the global parameters D, LOAD, Periode and Source (upper right)?
  1. Place the component **param** from the library **special.olb** and make double click on it.
  2. Click on New Property and complete window with the global parameters. Start with D and continue with the next ones.
  3. Close the opened tab (automatically you are seeing the circuit again) and save.

Tip

Global parameters are defined in the components value between curly brackets { }.

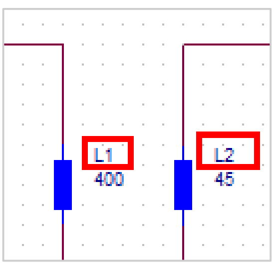






SMPS – Design (III)

- How can you define the transformer?
  1. Place the core E13_6_6_3C81 from the library **magnetic.olb** and make double click on it.
  2. Connect the inductors L1 and L2 between them using for that their reference designator.



	A
	SCHEMATIC1 : PA
Color	Default
COUPLING	1
Designator	
Graphic	E13_6_6_3C81.Normal
ID	
Implementation	E13_6_6_3C81
Implementation Path	
Implementation Type	PSpice Model
L1	L1
L2	L2
L3	
L4	

  3. Close the opened tab, come back to the circuit and save the project.

Tip

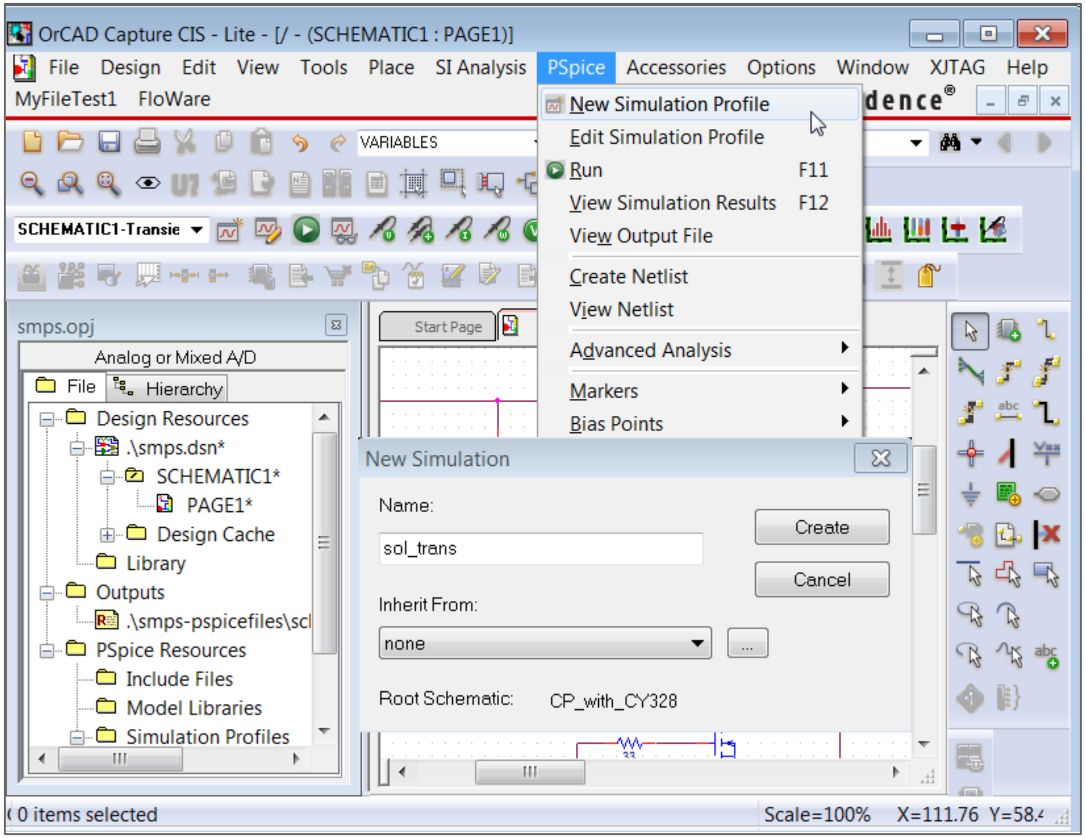
This is a nonlinear transformer based on the Jiles-Atherton model. The value of the inductors refers to the number of turns around it in the primary and secondary side.

SMPS – Simulation Profile (I)

- Click on **PSpice > New Simulation Profile** or click on .
- Write the name **Transient** and click on create.

Tip
It is possible to define in a project more than one simulation profile.

If you want to modify anything,
just click on **Edit > Simulation Profile**
or click on .



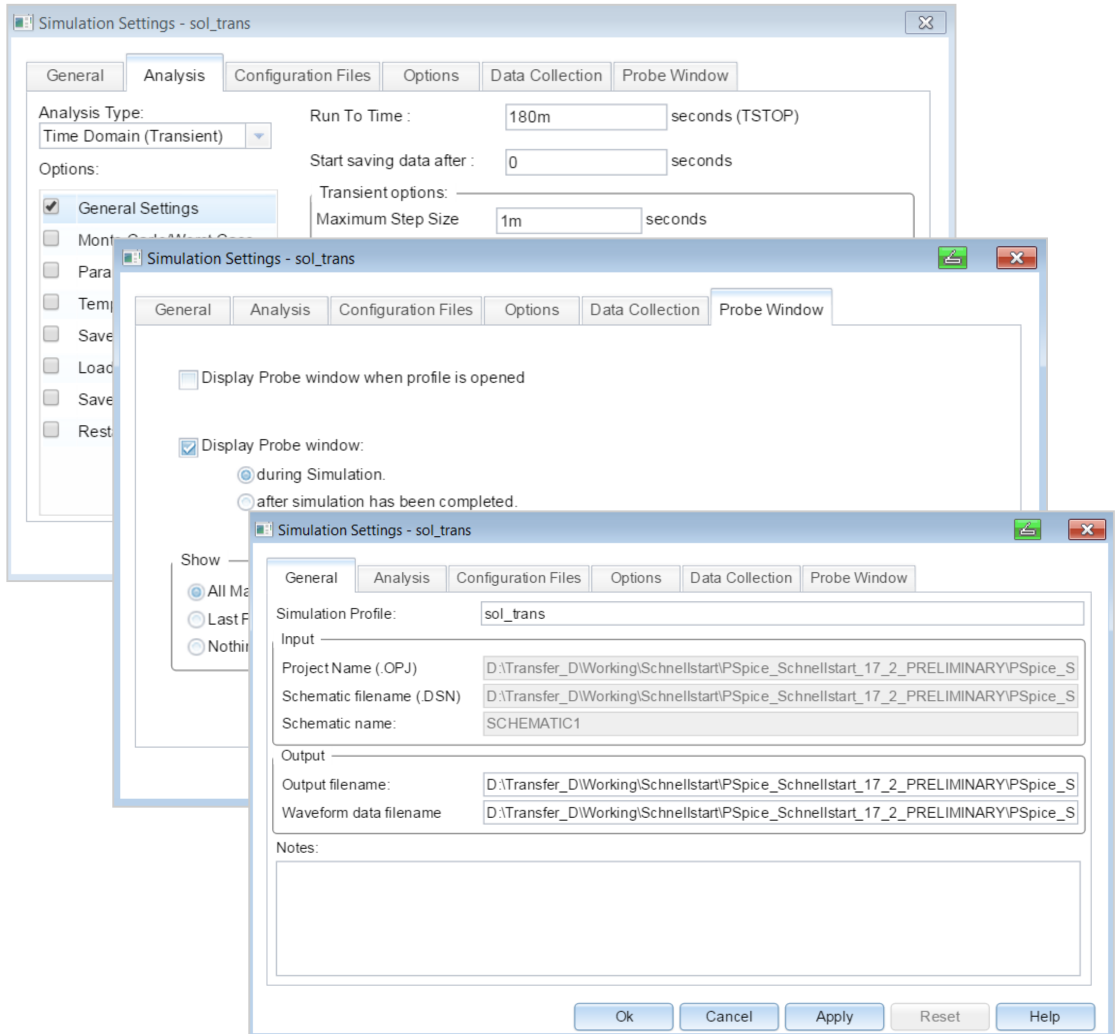
SMPS – Simulation Profile (II)

- Simulation Settings Window pops up
- Under Analysis type, you define the simulation type, the duration of the simulation, and the step size.
- Under Probe Window, you can define what is displayed in the probe window.

Tip

If no value is entered under Maximum Step Size, a default value (Final Time / 50) is used as start value, and the displayed curve may be inaccurate.

However, a too little step increases the simulation time significantly.



SMPS – Simulation

- After the definition of the Simulation Profile, you can simulate the circuit.
- Click **PSpice > Run** or **F11** or  .
- If you have designed the circuit as in the page 36, you will get opened the Probe Window, but no trace will be shown, although you selected in the Simulation Settings **All markers on open schematic**, as you did not place any marker.
- What can you do?
  - You can place markers afterwards  in the circuit, where you want to show the results without additional simulation.
  - You can click in the Probe Window on **Trace > Add Trace** and select the trace you want to view.

Tip 1

V marker on the net connection, **I marker** on the component pin, **W marker** on the component.

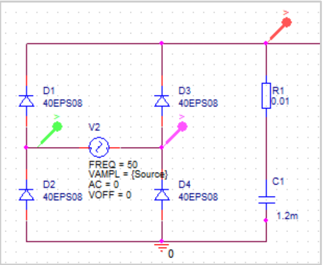
Tip 2

The simulation will take approximately 4 minutes, so you have time for a coffee.

SMPS – Results (I): Rectified Voltage

- Place a Voltage Marker on the left side of V2, other on the right side and the last one after the rectifier.

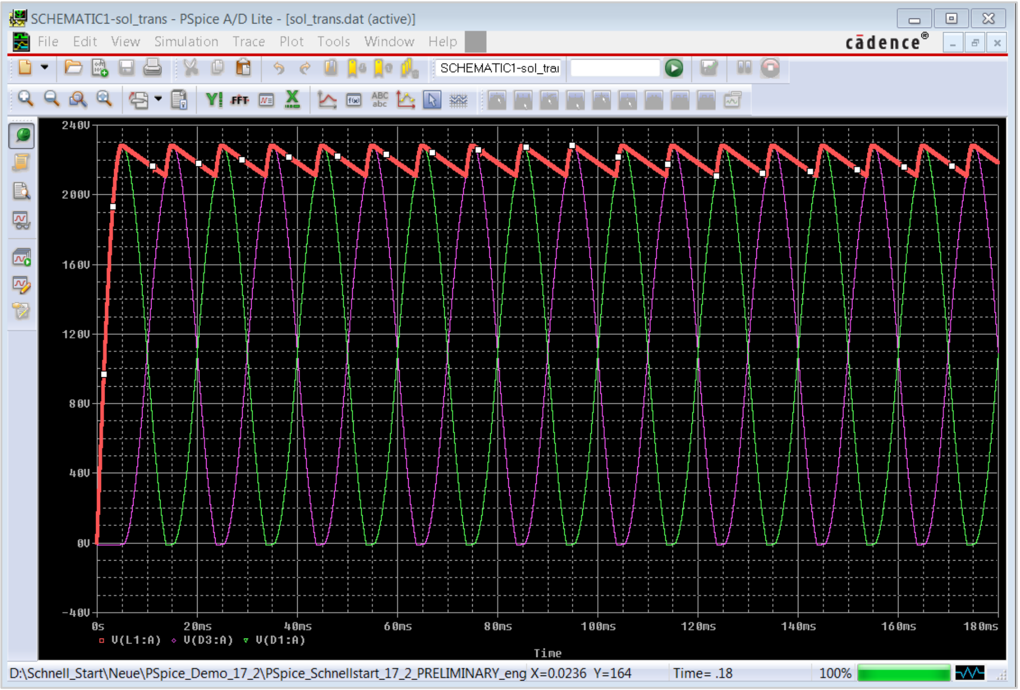
- What are we seeing?
  - Red: Rectified AC-DC voltage
  - Green: Positive input AC source
  - Violet: Negative input AC source



Evaluation:

The capacitor C1 (known as reservoir capacitor) (1.2 mF) has sufficient effect, that is why the rectified wave is so flat.

On the next page, the U-I behavior on the C1 is examined.



Tip
The color assignment takes place according to a predefined sequence in the pspice.ini file, and the corresponding input sequence.

SMPS – Results (II): U-I Behavior on C1

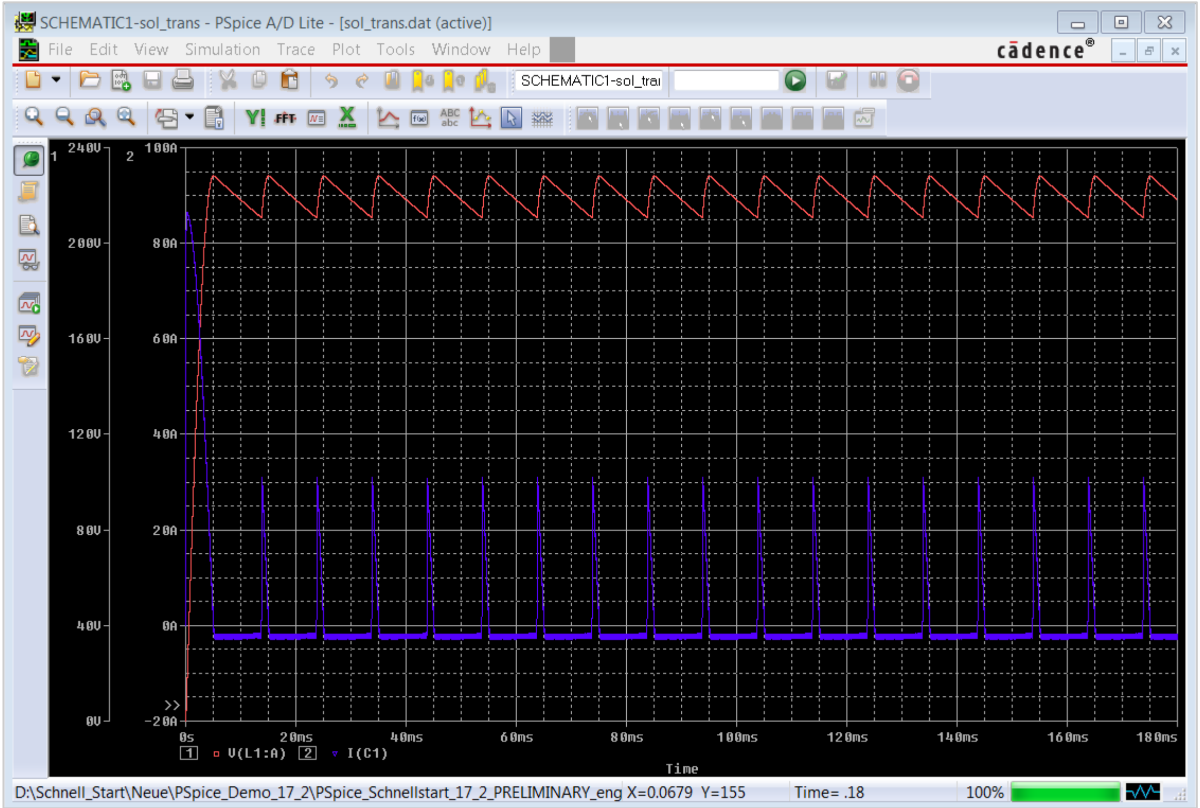
In this plot, the current-voltage behavior at capacitor C1 becomes clear. Consider that the value of this capacitor is important for the smoothness of the output voltage.

Tip 1

- Delete previous traces
- Select name and press key **Entf** or **Del**
- Add Trace V(L1:1)
- Add Y Axis (**Plot > Add Y Axis**)
- Add Trace I(C1)

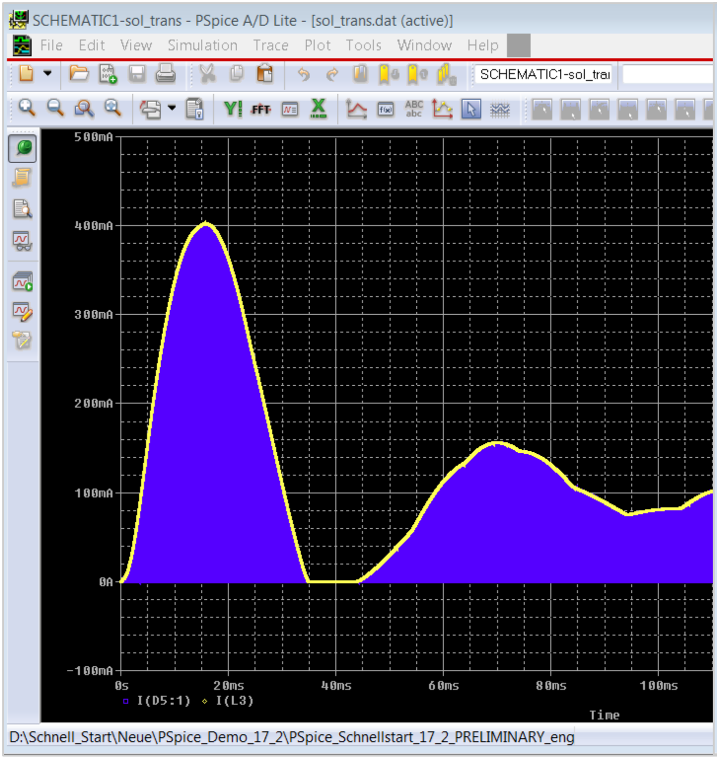
Tip 2

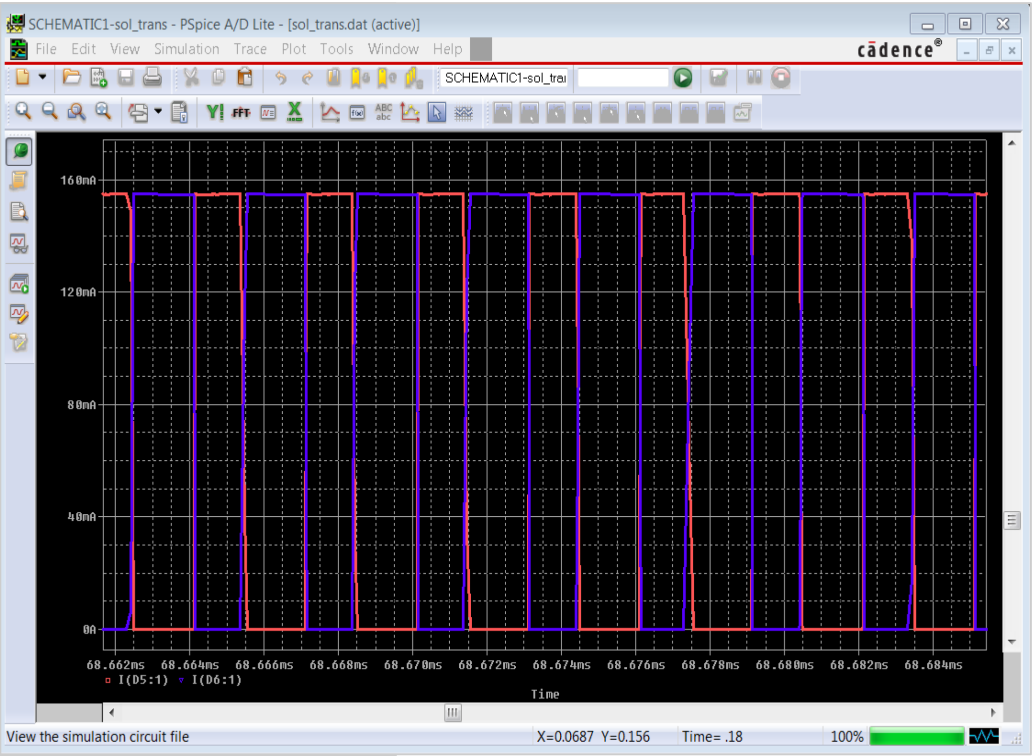
You can see a negative sign ahead of I(C1). It is so because the nomenclature is to take the current from pin 1 to pin 2, what means that we have placed the capacitor in the other direction. This is not any error!!



SMPS – Results (III): Generation of HF Signals

As we have designed a SMPS circuit using a voltage pulse of approximately 330 KHz, we have generated a high frequency square wave, converting then DC to AC again. Signals in the secondary side of the circuit are now isolated and converted to DC using the diodes D5 and D6 together with a LP Filter to eliminate the AC source. In this procedure, the diode D5 is ON when MOSFET is ON and D6 is ON, when the MOSFET is OFF. L3 is driving here (yellow) then the sum of both of them.





SMPS – Results (IV): Behavior of the LP Filter

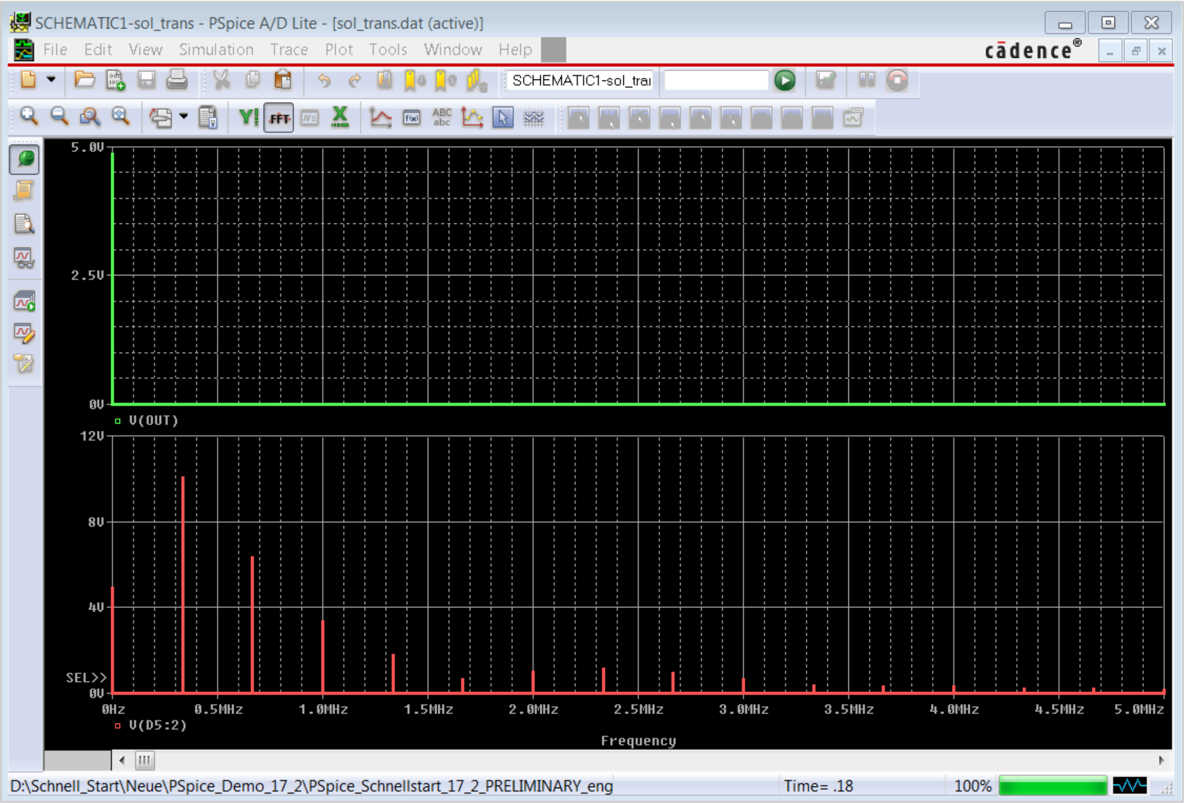
The low pass filter is designed to remove the high frequencies, eliminating then the AC component from the output, and getting the desired DC Output.

Tip 1

To add a new window, click on **Plot > Add plot to window.**

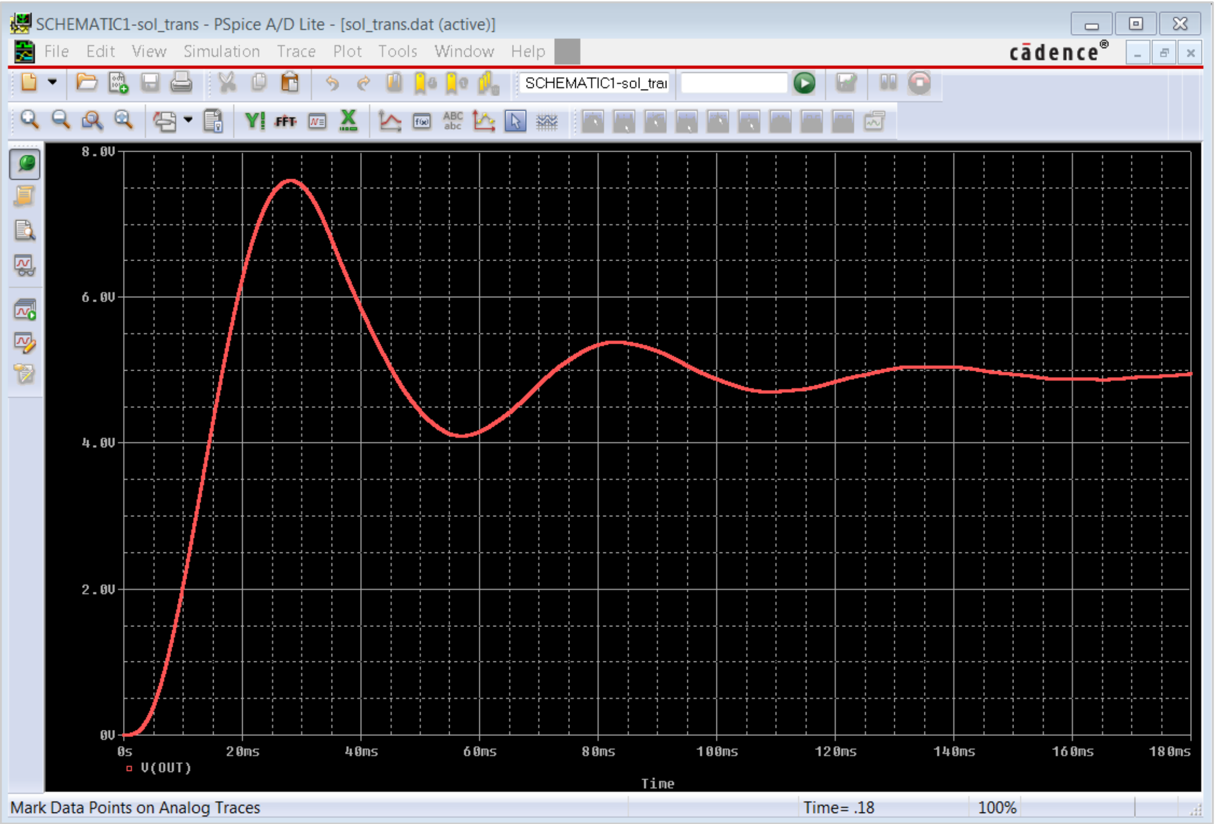
Tip 2

Once you have plotted the desired traces, click on FFT to show the traces in the frequency domain.



SMPS – Results (V): Output Voltage

The DC voltage rises oscillating until it is stabilized to a value of approx. 5 V (transient response) and is quite flat. This smoothness is achieved thanks to the capacitor C1 (among other reasons) and the low pass filter we have used.



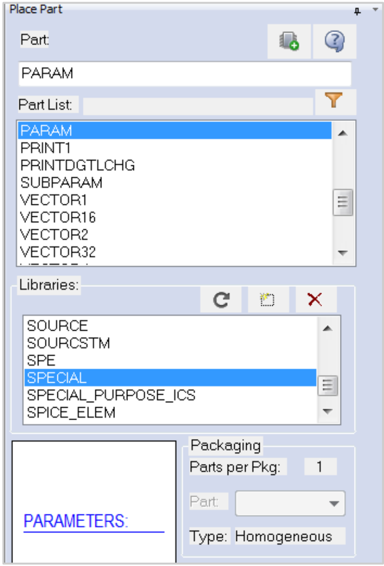
SMPS – Parametric Sweep (I)

The Parametric Sweep allows you to simulate a circuit with different values of a component and to display the results in a view with a single call.

How do you have to do it? Basically there are 4 steps.

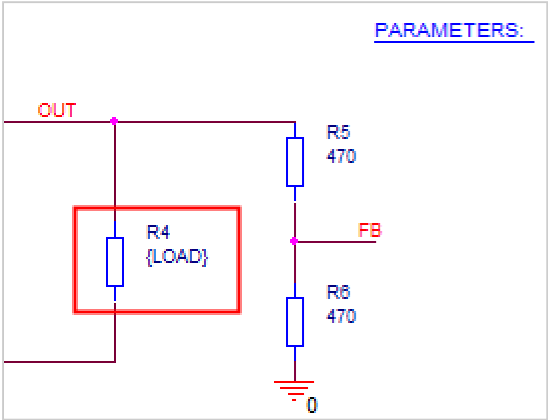
1. **Definition of a global variable**

   Place the part **Param** from the library special.olb somewhere on the Schematic, it serves only as a placeholder for several global variables, here the nominal value for R4 (50 Ohm).



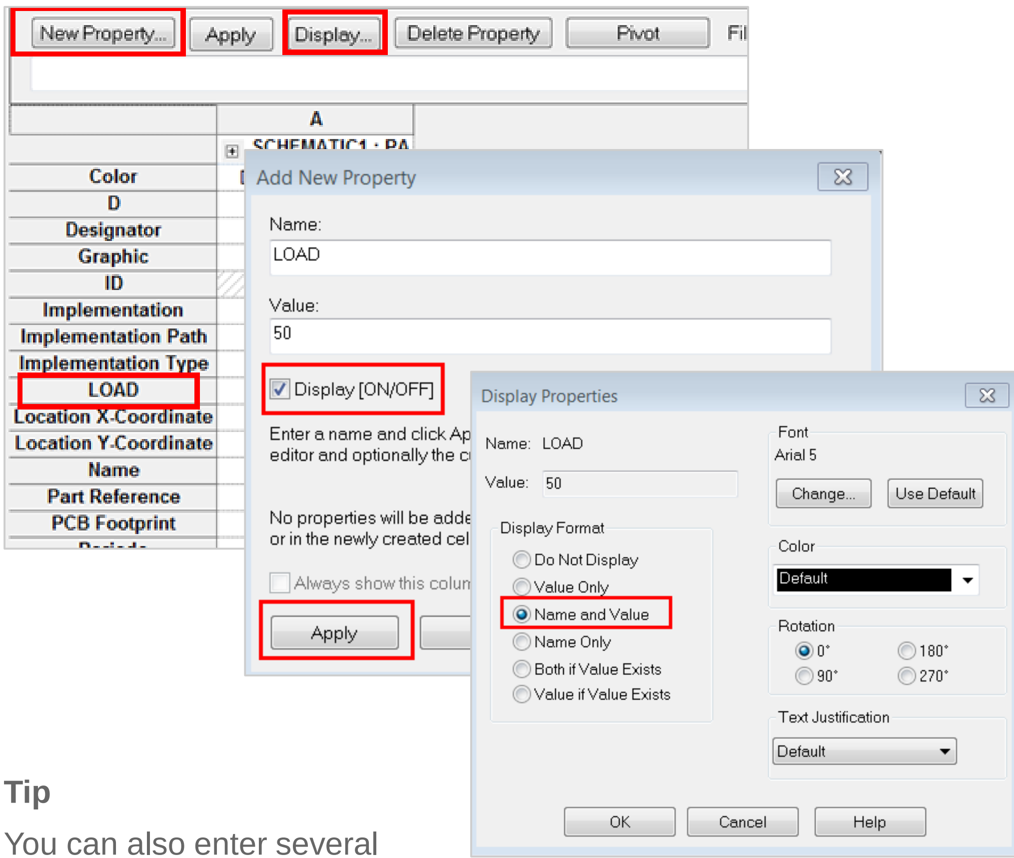
2. **Specify the component using this global variable**

   To do this, double-click on the value of the desired component and confirm with OK. Be sure to use the curly brackets {}.

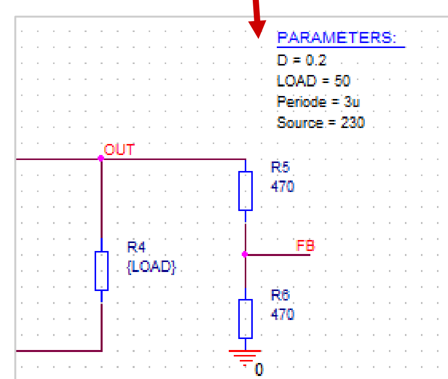


SMPS – Parametric Sweep (II)

3. **Assign and Display global variables in the Schematic**
   Double-click Parameters and perform the following steps:



- **New Property** in the Property Editor
- **LOAD**, **50** in Add New Row Window
- **Apply** in Add New Row Window
- Exit from Add New Property with **Cancel**
- **LOAD** is added as a new property
- **Select** LOAD and use **display**
- Display Properties on **Name and Value**
- Do the same for D, Periode and Source
- After confirming with **OK**, the entries are visible under Parameters.



Tip

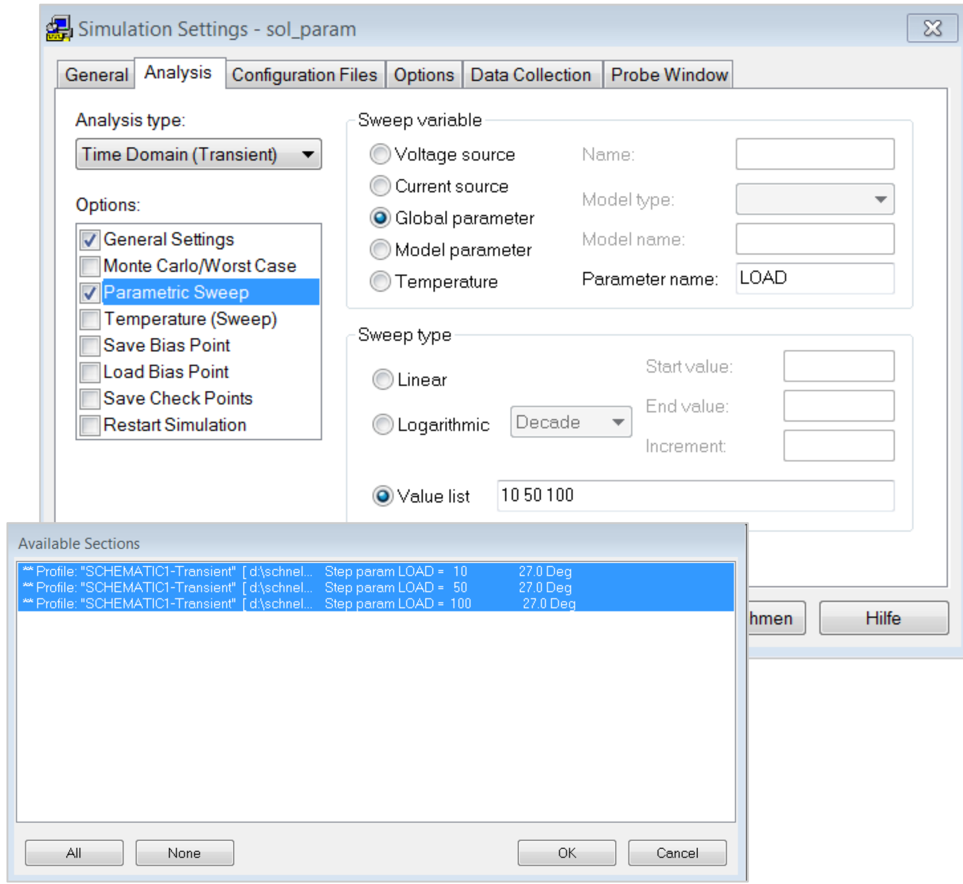
You can also enter several variables under Parameters.

SMPS – Parametric Sweep (III)

4. **Simulating Profiles with Parametric Sweep**

   Edit the existing simulation profile on **PSpice > Edit Simulation Profile** or as follows:

- Analysis Type on **Time Domain**
- Selection of **Parametric Sweep**
- Sweep variable on **global parameter**
- Parameter name on **LOAD**
- Sweep Type **Value List**
- Enter the values with spaces
- Accept the settings and OK
- Start the simulation with e.g. F11 or
- You now get a selection window, since a separate simulation graph was generated for each parameter. You have approximately 10 minutes for a coffee.



SMPS – Parametric Sweep Results (I)

According to the settings in the previous window (All), you will get the following display in the probe window for your parameter sweep.

Conclusion: A change in the load changes the transient response with a different output voltage.

Tip 1

Clicking **RMB > Trace Information** on a single graph displays the associated variable value.

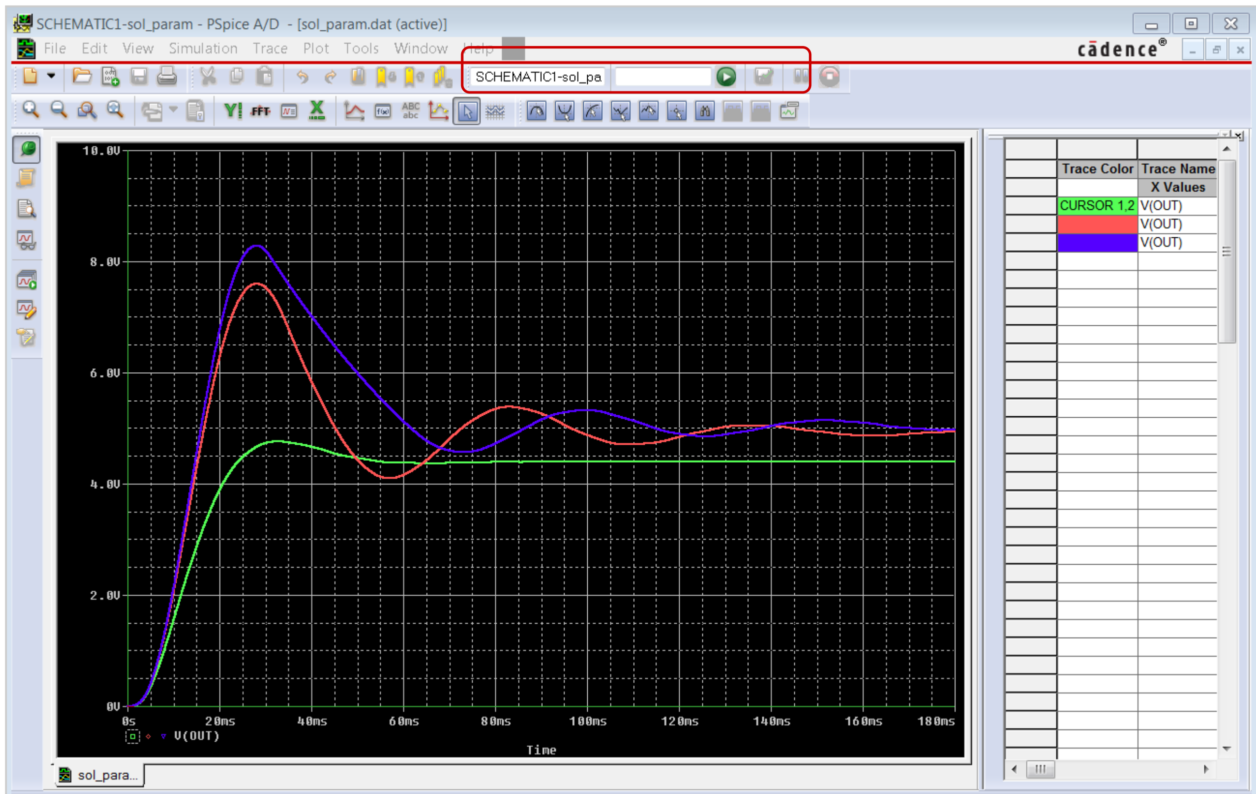
Via **Plot > Transient** you get the selection window again. When selecting a single simulation, only those curves are shown.

Tip 2

Via **Trace > Cursor > Display**

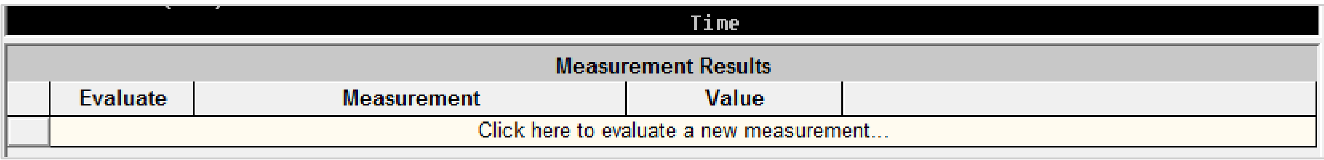
or  get that Probe cursor window.

Using different sub-menus icons, you can perform specific queries on the individual curves.

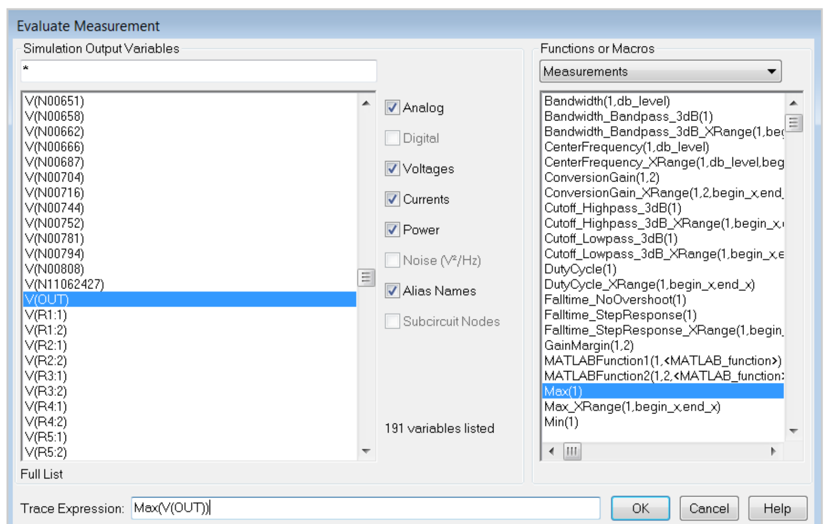


SMPS – Parametric Sweep Results (II)

- You can also take measurements to the plotted traces in order to evaluate the requirements of your design:
  - Click on **View > Measurements Results.** This window will pop up:



  - Click on **Click here to evaluate a new Measurement** and add max(v(out)):
    - At first select the Mesurement and then Output Variable Max( ) > Max(V(out))



SMPS – Parametric Sweep Results (III)

– Visualize the maximum value for each particular trace

Measurement Results					
	Evaluate	Measurement	1	2	3
▸	☑	Max(V(OUT))	4.76254	7.59829	8.28274
	Click here to evaluate a new measurement...				

- You can also use the Toggle Cursors to take measurements between traces in the X and Y axis.
- There are 2 Cursors available with 2 kinds of cursor probe windows:
  – Click on and one of these 2 windows will pop up.

New style

Trace Color	Trace Name	Y1	Y2	Y1 - Y2		Y1(Cursor1) - Y2(Cursor2)		-2.824276246		
	X Values	32.65308313m	27.34714754m	5.305935584m		Y1 - Y1(Cursor1)	Y2 - Y2(Cursor2)	Max Y	Min Y	Avg Y
CURSOR 1	V(OUT)	4.762595382	4.627701665	134.89371709m		0.00000000	-2.959169963	4.762595382	4.627701665	4.695148523
CURSOR 2	V(OUT)	7.228815258	7.586871628	-358.05636918m		2.466219877	0.00000000	7.586871628	7.228815258	7.407843443
	V(OUT)	7.881059180	8.270256740	-389.19756044m		3.118463798	683.38511272m	8.270256740	7.881059180	8.075657960

Old style

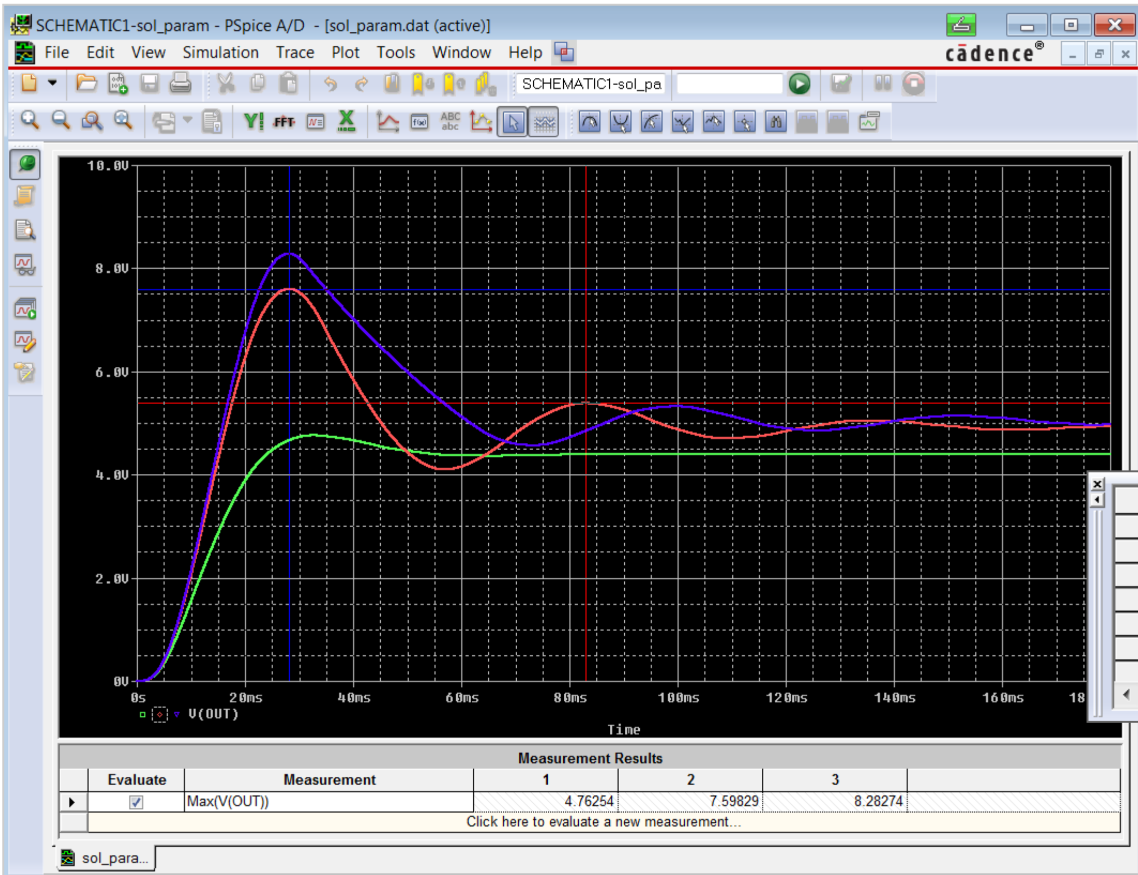
Probe Cursor

```
A1 =      0.000, -656.5E-27
A2 =      0.000, -656.5E-27
dif=      0.000,      0.000
```

– To change style use: **Tools > Options > Cursor Settings**

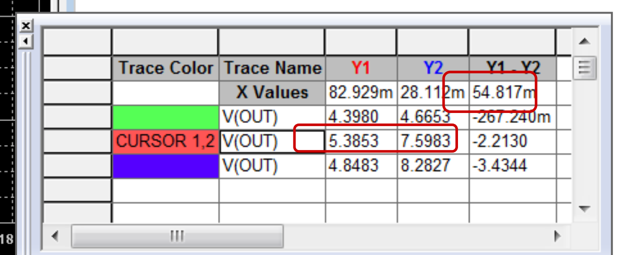
SMPS – Parametric Sweep Results (IV)

- Use cursor 1 with the LMB (Left Mouse Button) and cursor 2 with RMB to measure e.g. the time difference between maximum values.



Max Values are:
5.3853 V and 7.5983 V

Time between is:
54.817 ms



Final Remarks

- After performing all the examples included in this Quick Launch, you should be able to perform simple simulations with PSpice and its basic functions.
- This does not replace, of course, any PSpice training as it is given by FlowCAD or Cadence. It is only intended to provide the most important steps necessary to get an overview of the possibilities of PSpice.
- If the PSpice models can not be found in the standard library, you can always ask directly to the manufacturer. These models can also be included in the demo version, opening the Simulation Settings of your simulation profile and adding them from the Configuration Files tab. In addition, PSpice also has a powerful model editor, you can use to create your own models from datasheets.
- Do you want PSpice to simulate mechanical elements, e.g. of shock absorber of a vehicle or of an electric windscreen lifter, which is quite possible, you must use electrical replacement models for those mechanical elements, which of course are not a part of the usual libraries shipped by Cadence.
- In case you have digital and analog signals coexisting in the same circuit, take into account that PSpice supports analog and digital simulation at the same time in one simulation.

PSpice Extensions

- PSpice is not only thought as an analog / digital simulator, but as a tool to improve the performance, reliability, productivity, predictability, costs and quality of your circuits, using Sensitivity Analysis, Monte Carlo Analysis, Smoke Analysis and Optimizer from PSpice Advanced Analysis Option.
- PSpice also allows simulation of mixed signal systems thanks to the possibility to abstract the functionality of models like microcontrollers, microprocessors, etc. using C / C++, SystemC and Verilog-A languages. It allows to evaluate the whole system with its algorithms in only one simulator platform.
- Another possibility to extend PSpice's application area is to link PSpice to MATLAB. For this, Cadence offers an adjustable interface SLPS (Simulink-PSpice), which enables co-simulation with MATLAB and also the choice to send data to MATLAB for post-processing and the use of MATLAB Block Functions in the PSpice environment.

Kontakt zu FlowCAD / Contact us

Für weitere Fragen und Informationen stehen wir gerne zur Verfügung.
Please don‘t hesitate to contact us.

FlowCAD Deutschland
Mozartstr. 2
85622 Feldkirchen bei München
T +49 89 4563-7770
F +49 89 4563-7790
info@FlowCAD.de

FlowCAD Schweiz
Hintermättlistr. 1
5506 Mägenwil
T +41 56 485 91 91
F +41 56 485 91 95
info@FlowCAD.ch

FlowCAD Polen
ulica Sasiedzka 2A
80-298 Gdansk
T +48 58 342 75 94
F +48 58 342 70 60
info@FlowCAD.pl

Follow us

www.facebook.com/FlowCAD

Join our Facebook page where we focus on giving a glimpse into ongoing innovations. You will find selected news, events, success stories and insights.

www.twitter.com/FlowCAD

On FlowCAD's Twitter we provide press releases, news articles, films and images as well as reports from events.

www.youtube.com/FlowCAD

On our YouTube channel you will find 100+ video tutorials to learn more about electronic circuits. With the PSpice Lite version from our website everyone can easily simulate.
In our playlists we also offer product news and webinars.

Don't forget to subscribe, share and like!

FlowCAD