



Quickstart OrCAD PSpice

Version 22.1



Table of Contents

- [Introduction to OrCAD Capture Setting up a Simulation in PSpice](#)
- [Basic Types of Simulation in PSpice](#)
- [Design Example: Switched Mode Power Supply \(SMPS\)](#)
- [SMPS Parametric Sweep](#)
- [Conclusion](#)
- [Appendix](#)



Preliminary Notes

- This documentation applies to the first-time users of PSpice circuit simulation software. It should not be understood either as a training manual or as a complete operating manual.
- In order to run the software, you need a valid PSpice Designer License (PO1520), or you have received valid licenses from the [OrCAD student program](#). You can also [request a free trial](#) version from the OrCAD home page.
- Basic knowledge in electronic circuitry is helpful.
- Due to brevity and compactness of this documentation, it is not possible to address all existing functions and their details. Please refer to **Help > PSpice Documentation**.
- Based on a simple circuit diagram for a power supply, most important functions and steps are presented, which will allow first-time user to quickly manage his own tasks with minimum training efforts spent.
- After some brief information on user interface of the software and on different simulation types, hands-on instructions begin with entering circuit diagram from scratch on [page 30](#).
- To get started, just unzip **pspice-demo-22-1.zip** file to a folder of your choice and then start already installed OrCAD Capture software.
 - Make sure, that the **pathname** and the **name of the folder** does **not contain any special characters or white spaces!**
- Detailed training offers for the different tools can be found at <https://www.flowcad.com/en/training.htm>



Introduction to OrCAD Capture Setting up a Simulation in PSpice



Capture / PSpice User Interface

OrCAD Capture and OrCAD PSpice are two tightly connected, but separated tools where OrCAD Capture is used to design your circuit on a schematic including the preparation of all the inputs required by OrCAD PSpice.

OrCAD PSpice itself is the circuit simulator coming along with a comfortable user interface for displaying the simulation results.

OrCAD Capture and OrCAD PSpice are mostly menu driven.

All Capture and PSpice commands are launched by

- Pull-down menus
- Icon Toolbars
- Keyboard Shortcuts
- Pop-up windows
- Command windows
- Tcl functions

Basically all menus in Capture and PSpice are context-sensitive. This means that depending on which elements are selected, workspaces or commands, resulting pop-up windows or pull-down menus will change its appearance. It will show only relevant functions or objects you need to work on for just selected elements.

As there are restrictions which special characters in PSpice data, it is highly recommended to use only underline character (_) as a special character in path and name of the design!



Unit of Measurements, Value Multipliers in PSpice

PSpice only calculates with plain numbers, not with units.

PSpice recognizes these value multipliers:

1f	(femto-)	= 1 x 10 ⁻¹⁵
1p	(pico-)	= 1 x 10 ⁻¹²
1n	(nano-)	= 1 x 10 ⁻⁹
1u	(micro-)	= 1 x 10 ⁻⁶
1mil	(mil)	= 2.54 x 10 ⁻⁵
1m	(milli-)	= 1 x 10 ⁻³
1K	(kilo-)	= 1 x 10 ³
1Meg	(mega-)	= 1 x 10 ⁶
1G	(giga-)	= 1 x 10 ⁹
1T	(tera-)	= 1 x 10 ¹²

Tip

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

1KOhm = 1K

- Although **Capture** translates expressions like 2k2 correctly into 2.2k for the PSpice netlist, it is **not recommended** to use such expressions in your circuit schematics. You may fail with other tools and break with other conventions.

Attention

- PSpice is **not case-sensitive**.
1M = 1m
M or **m** in PSpice means both **milli**
- 1 x 10⁶ = 1Meg = 1MEG = 1meg



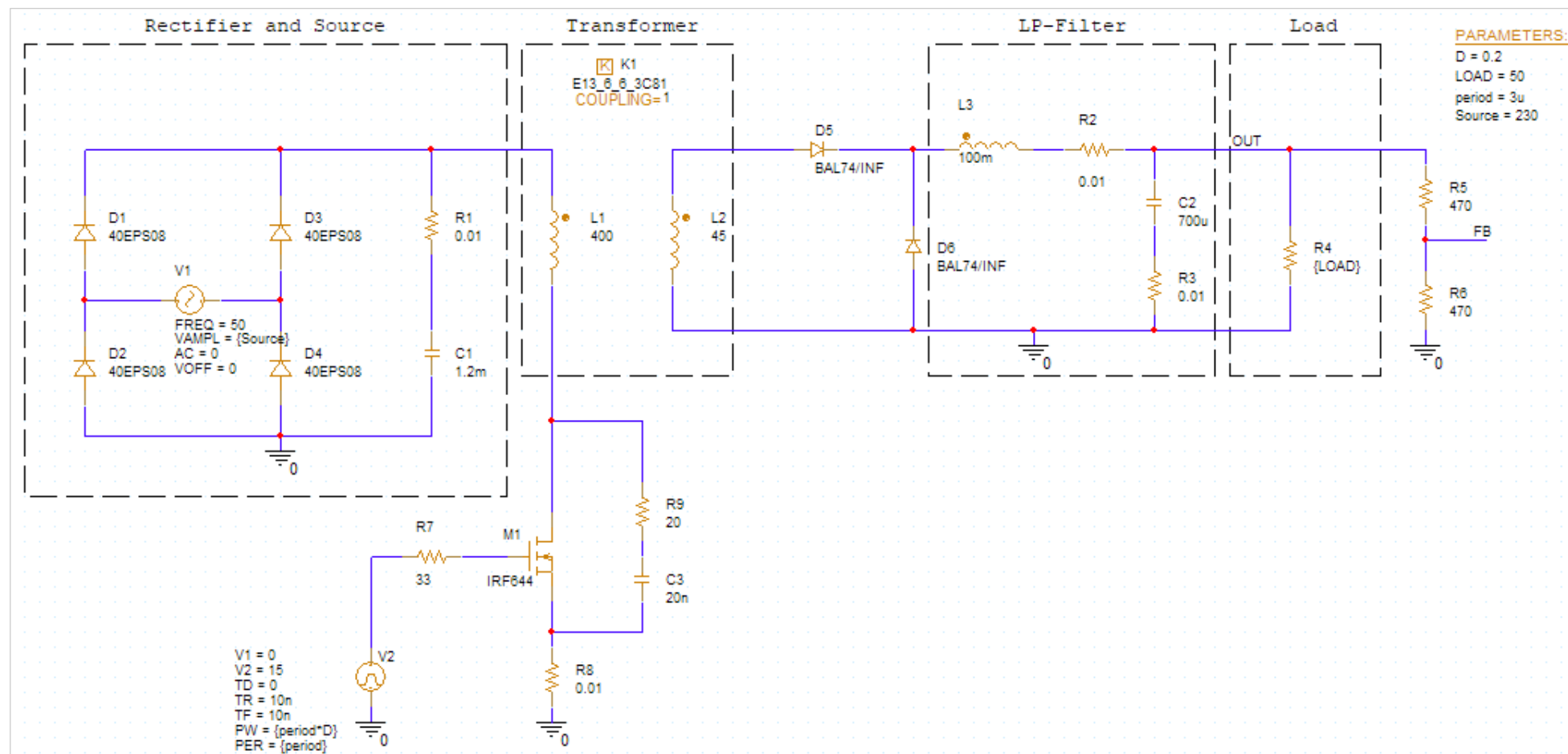
Proposed Circuit

- Aim of this PSpice introduction is to simulate a complete circuit, particularly a Switched Mode Power Supply (SMPS), which consists of a source, a rectifier, a PWM-Control, a transformer, a LP-Filter plus the load.*
- When you extract data in quickstart container, you'll find a **solution** directory, which contains an example of how to simulate the overall circuit. You will find here a circuit using European schematic symbols. 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 following pages, you will find a brief overview of actions you can take to implement circuit diagram below from scratch.

* plus mandatory simulation ground "0"



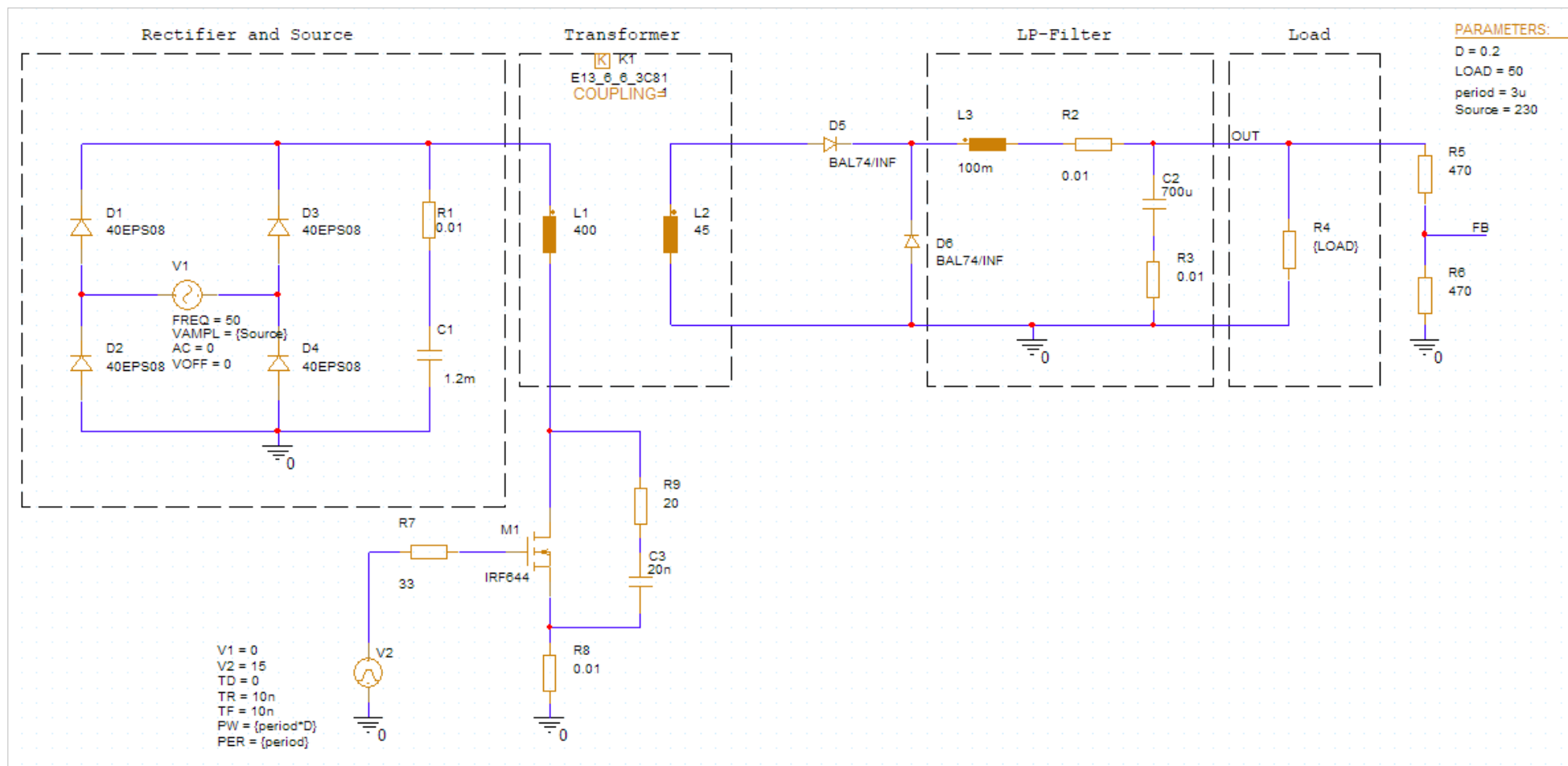
SMPS Circuit – American Style (Default)



American Style



SMPS Circuit – European Style (Using Europe.olb)



European Style



Run a PSpice Simulation


There are two ways to start a simulation in PSpice:

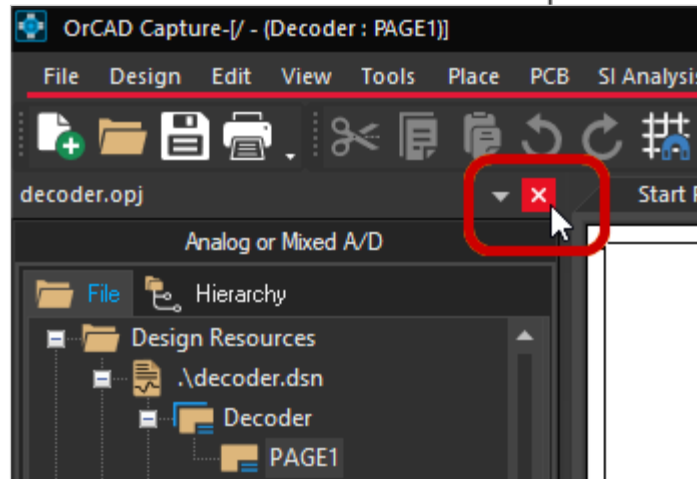
- Directly from OrCAD Capture (preferred way, used in this tutorial):
 - Launch Capture (full installation): **Start > Cadence PCB 2022 > Capture CIS 2022**
 - (trial installation): **Start > OrCAD Trial > Capture CIS 2022**
or double-click on the Capture icon which is on your desktop after the trial installation
 - First using Capture to design the circuit and creating the simulation profile, and then starting PSpice simulation from Capture environment via a run-button. Required netlisting and preparation of the simulation commands is done in the background.
- Or Opening directly PSpice:
 - **Start > Cadence PCB 2022 > PSpice AD 2022**
 - However, PSpice requires netlist file <*.net> and circuit file <*.cir> of circuit to be simulated. The user has to prepare both, the netlist and the command file. How you can do this is far beyond the intention of this tutorial.
- Lets get started with our tutorial:
Launch Capture CIS. **Start > Cadence PCB 2022 > Capture CIS 2022** or double-click






Starting Capture Session

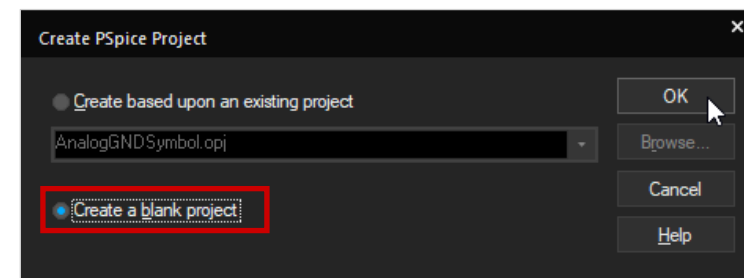
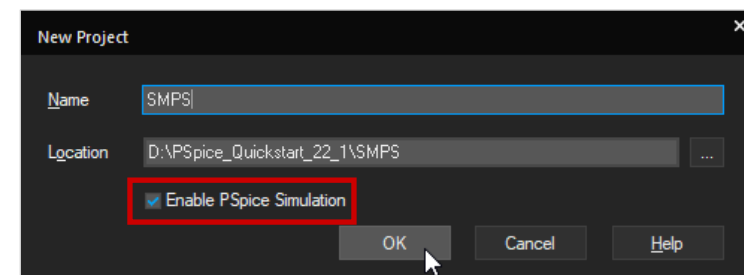
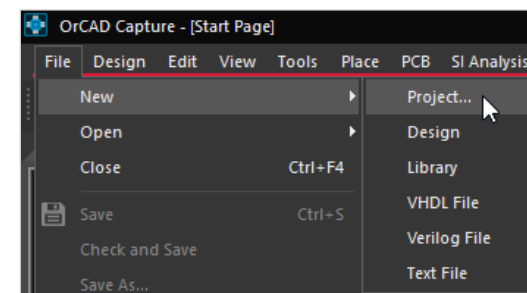
- When starting, Capture displays an informative Start Page (e.g. it shows the current tool version, support information, last projects you've been working on, ...).
- Try to open an existing demo design with **File > Open > Demo Designs**.
 - After inspecting a demo design, close it simply by clicking the  icon in the project manager.
- You can also open an already existing project with **File > Open > Project**.






Create a New PSpice Project

- Select **File > New > Project**. New Project window opens.
- In upper field under **Name**, please enter the name of your project, e.g. **SMPS**.
- In **Location** field, enter the folder in which your new project should be stored. You can browse to location by clicking the browse icon . It is highly recommended to use 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. Do **not** use any special character (e.g. umlauts) or white space characters for the location and for the name!
- Select **Enable PSpice Simulation**.
- Click **OK**.
- A new window pops up.
- Here select **Create a blank project**.
- Click **OK**.



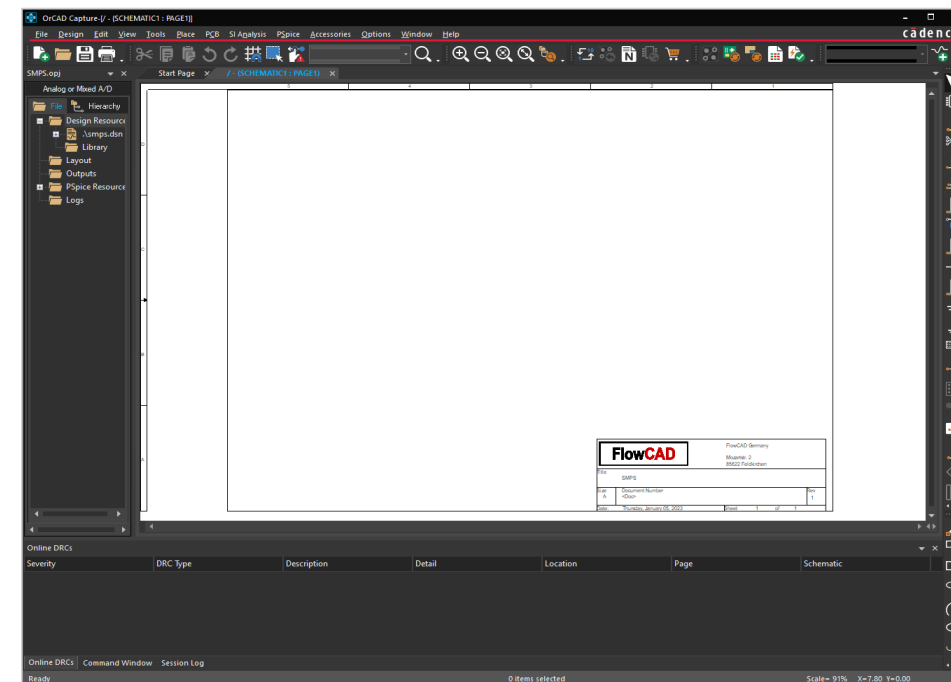
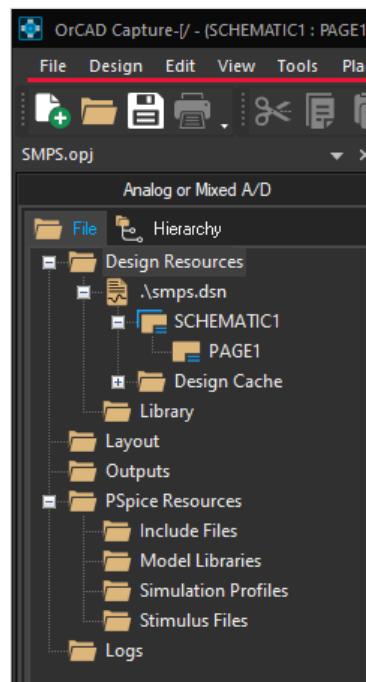


Project SMPS

- You have created a new project called **SMPS.opj** with a design of same name **smps.dsn**.
- At the same time, the first page of your design was opened with the name PAGE1. Similarly, a drawing frame and a Title Block* were automatically placed. Other view-presets are possible.
- Please note the folder structure in project manager on the left side. Expand the items by clicking on the  icon in front of the folder.
- The folder structure is virtual. It only exists within project manager. PAGE1 located under Schematic1 is only found within smps.dsn file.
- Icons for component placement and wiring are already visible on right-hand side.

On next slides, we will explain some necessary steps which you can practice later with your newly created project.

* Your Title Block looks different, as we have chosen a customized template for this slide deck.

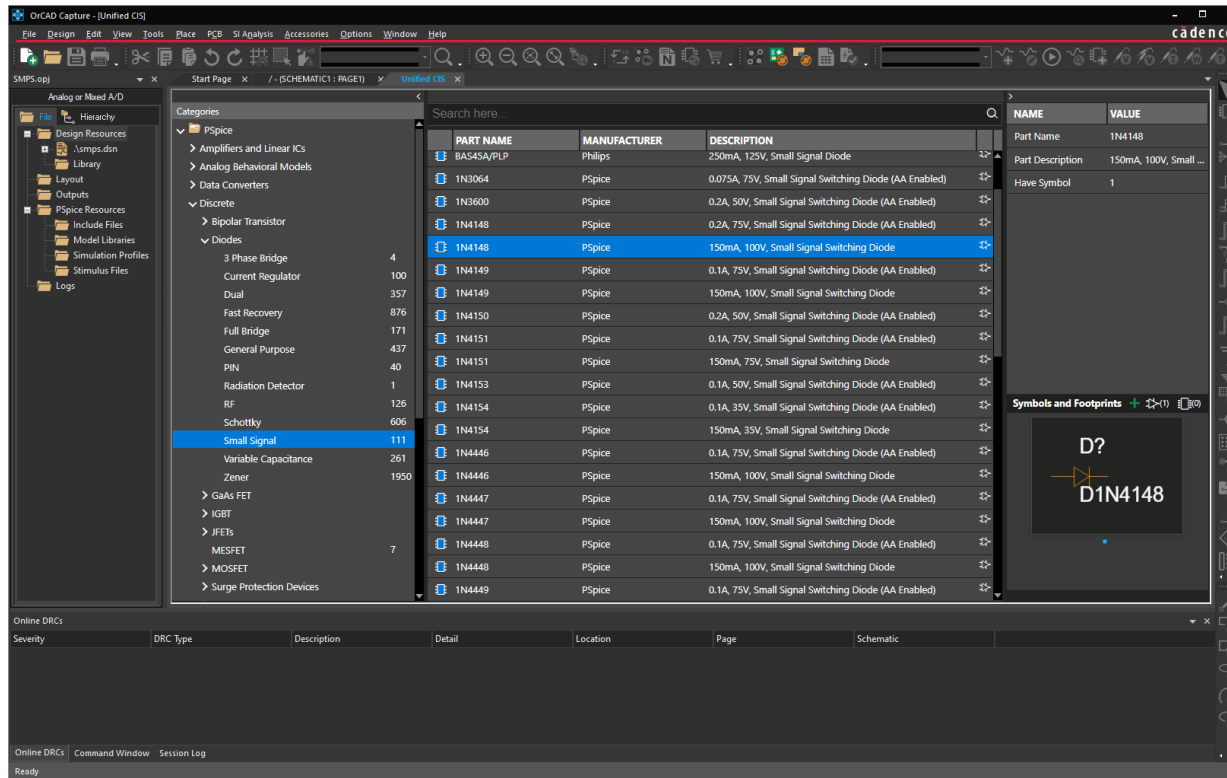




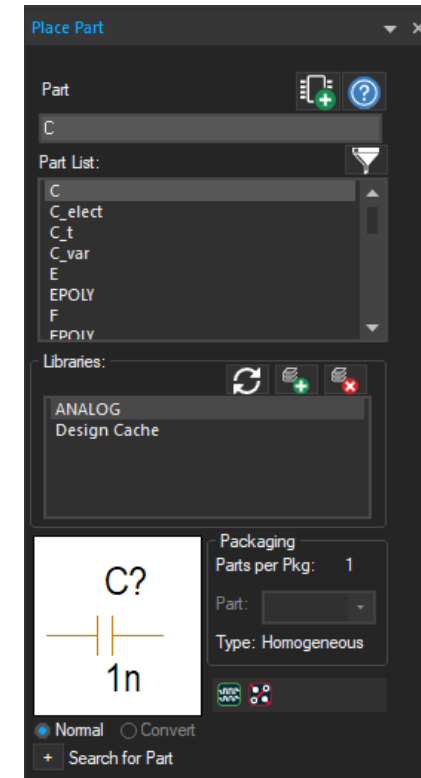
Placing PSpice Components

- There are several ways, how you can place PSpice components in your design.
- In this tutorial, you will learn how to make use of:

Unified CIS Browser



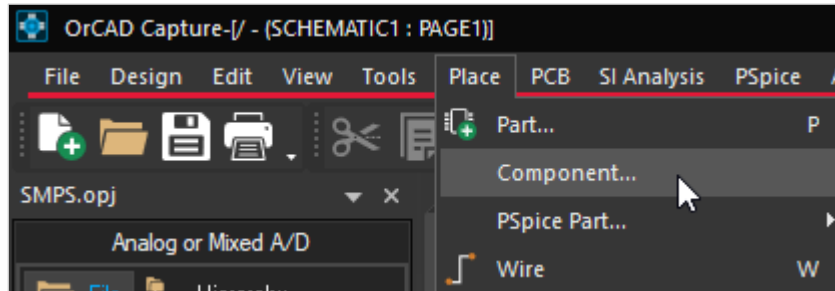
Place Part Command





Unified Component Information System (CIS) Browser

- The Unified CIS Browser is a component management system with pre-configured databases.
- Allows you to browse, view, search and place components.
- No need to care for location of supplied PSpice libraries.
- Open with **Place > Component...**
- You can either **Browse** through the PSpice components, walking through categories and sub-categories ...
- Or you can use the intuitive **Search** to find your components.
 - Preselecting a category or sub-category will narrow down your search results.
- Double-clicking on a selected component will place it in the current schematic.





Unified CIS Using Browse

- Select Category PSpice and Browse

Left pane: Categories browser

Middle pane: Part browser

Right pane: Properties browser

Categories

- PSpice
 - Amplifiers and Linear ICs
 - Analog Behavioral Models
 - Data Converters
 - Discrete
 - Bipolar Transistor
 - Diodes
 - 3 Phase Bridge 4
 - Current Regulator 1...
 - Dual 3...
 - Fast Recovery 8...
 - Full Bridge 1...
 - General Purpose 4...
 - PIN 40
 - Radiation Detector 1
 - RF 1...
 - Schottky 6...
 - Small Signal 1...**
 - Variable Capacitance 2...
 - Zener 1...
 - GaAs FET
 - IGBT

Search here...

PART NAME	MANUFACTURER	DESCRIPTION
1N4153	PSpice	0.1A, 50V, Small Signal Switching Diode (AA E...
1N4154	PSpice	0.1A, 35V, Small Signal Switching Diode (AA E...
1N4154	PSpice	150mA, 35V, Small Signal Switching Diode
1N4446	PSpice	0.1A, 75V, Small Signal Switching Diode (AA E...
1N4446	PSpice	150mA, 100V, Small Signal Switching Diode
1N4447	PSpice	0.1A, 75V, Small Signal Switching Diode (AA E...
1N4447	PSpice	150mA, 100V, Small Signal Switching Diode
1N4448	PSpice	0.1A, 75V, Small Signal Switching Diode (AA E...
1N4448	PSpice	150mA, 100V, Small Signal Switching Diode
1N4449	PSpice	0.1A, 75V, Small Signal Switching Diode (AA E...
1N4449	PSpice	150mA, 100V, Small Signal Switching Diode
1N4454	PSpice	0.2A, 75V, Small Signal Switching Diode (AA E...
1N4938	PSpice	0.15A, 200V, Small Signal Diode (AA Enabled)
1S2836	PSpice	0.1A, 50V, Small Signal Common Anode Dual ...
1S2837	PSpice	0.1A, 30V, Small Signal Common Cathode Du...
1S2838	PSpice	0.1A, 50V, Small Signal Common Cathode Du...
1S953	PSpice	2A, 35V, Small Signal Diode (AA Enabled)

NAME	VALUE
Part Name	1N4448
Part Description	150mA, 100V, Small ..
Have Symbol	1

Symbols and Footprints + (1) (0)

D?

D1N4448



Unified CIS Using Search

- Or Select **PSpice** (or any other subcategory) and enter a search string.

The screenshot displays the FlowCAD interface with a search for 'LM741' in the 'PSpice' category. The search results are shown in a table with columns for Part Name, Manufacturer, and Description. The selected part is 'LM741/NS' by National Semiconductor, described as a 'General-Purpose Operational Amplifier'. The right panel shows the part's details, including its name, description, and a schematic diagram of the LM741/NS operational amplifier.

PART NAME	MANUFACTURER	DESCRIPTION
LM741/NS	National Semiconductor	General-Purpose Operational Amplifier
LM741	PSpice	General-Purpose Operational Amplifier (AA Enabled)
LM741	PSpice	General Purpose Operational Amplifier
LM741A	PSpice	General-Purpose Operational Amplifier (AA Enabled)
LM741C	PSpice	General-Purpose Operational Amplifier (AA Enabled)
LM741E	PSpice	General-Purpose Operational Amplifier (AA Enabled)

NAME	VALUE
Part Name	LM741/NS
Part Description	General-Purpose Op...
Have Symbol	1

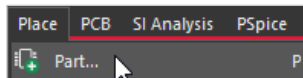
Symbols and Footprints + (1) (0)



Place Part and Automatic Reference Designator Assignment

The traditional way of placing parts is to use:

- **Place > Part...**



- **P** (Keyboard)

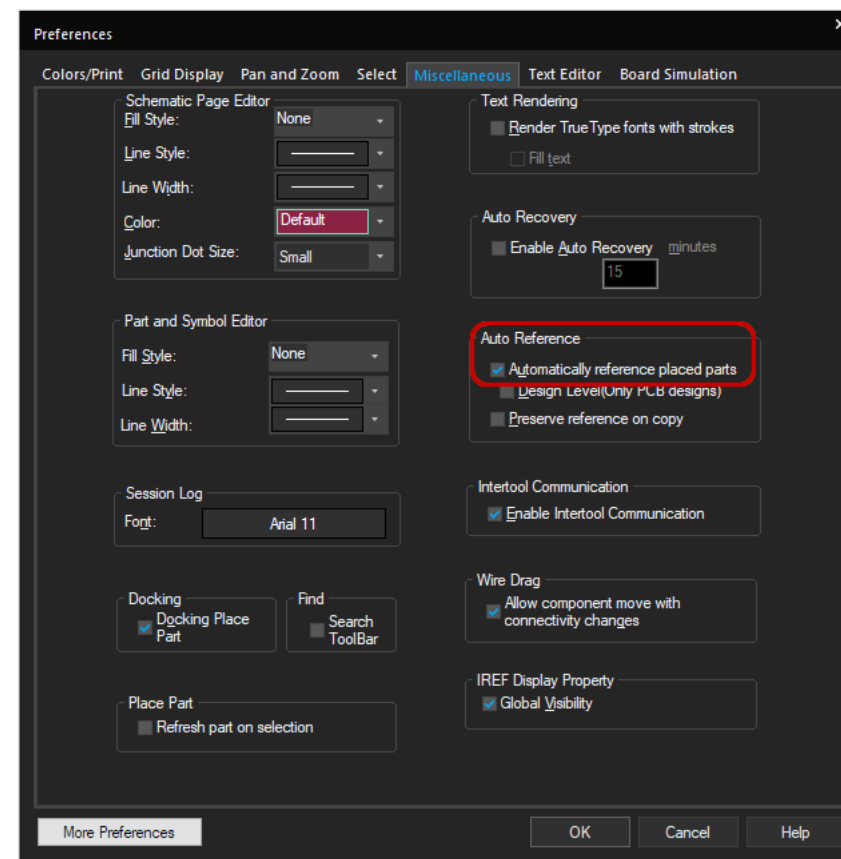
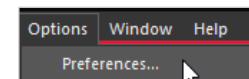
- Place Part icon



Note

In order to activate the Place Part Window, you must select a schematic page first.

In **Options > Preferences**, **Miscellaneous Tab** select **Automatically reference placed parts** which allows you to automatically annotate component references during placement.





Library Assignment, Selecting and Placing Parts

When you are in the **Place > Part** mode, the Place Part Window shows up on the right.



Under **Libraries**, you can select one or more libraries. All parts in these libraries will be shown in the **Part List** scrollbox. In the **Part** entry field, you enter the part which you are looking for. This entry field acts as a simple filter. No wildcards are allowed here. Clicking on a line in the scroll box will fill the part field with the selected content.

Use the **Add Library** icon  to add Part Libraries. The default directories with PSpice symbols are:

```
<installation_dir>\tools\capture\library\pspice  
<installation_dir>\tools\capture\library\pspice\advans
```

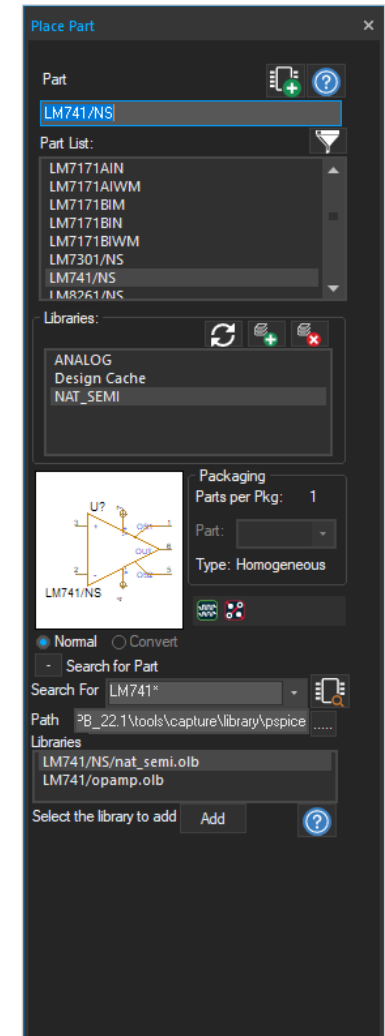
All the libraries shown in the Libraries window are stored in the user specific `capture.ini` file. For new PSpice projects, you don't have to add the libraries again.

Packaging shows the number of parts inside a package.

To the right of the Symbol Preview window the two icons indicate, whether a PSpice Model  and / or a PCB Symbol  is assigned to the selected component.

A double-click on a Part in the Part List, or Pressing Enter in the Part line will return to the schematic and lets you place the selected part by pressing the LMB (Left Mouse Button).


Search for Part allows searching with wildcards. Make sure, that the path is correctly entered. Pressing Add will add the library to the currently selected ones.

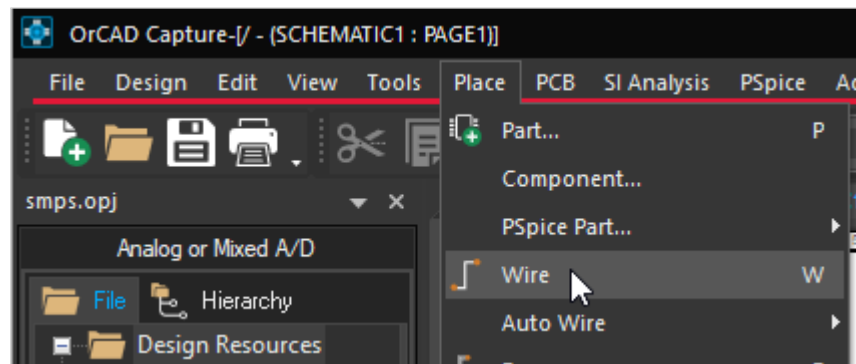





Wiring Components, Adding Text Notes or Other Objects

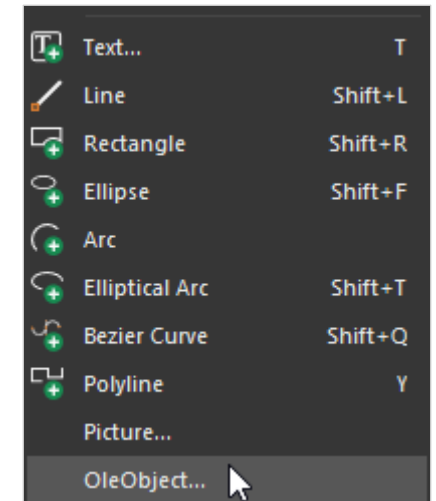
You need to connect the placed components. The simplest method of placing a connection is using a wire between two pins of components. Entering the wire mode by

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



You can also place graphical objects or pure text notes (e.g. to explain your circuit) on the schematic page:

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




Even pictures or OleObjects can be placed on a schematic page.



Assign Net Names (Net Alias)

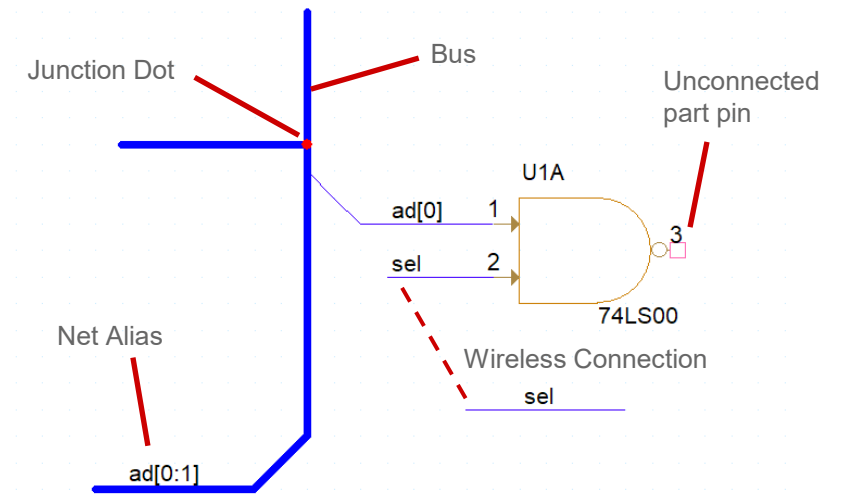
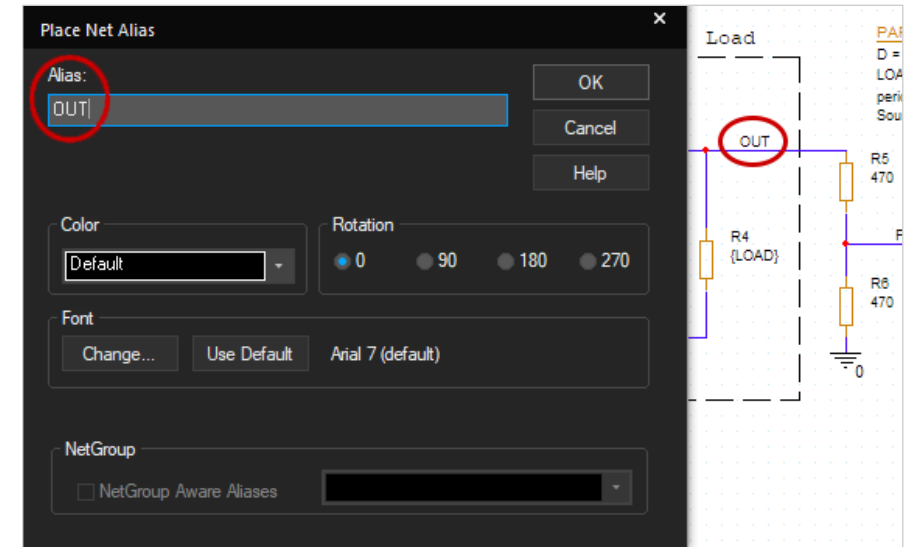
- Any new wire will get a system-generated net name like N0001234 by default.
- Those system-generated net names can be replaced by net aliases on a wire.
- Within a schematic page, any wires, off-page connectors or hierarchical ports with the same name are connected even just by their name.
 - You do not have to connect them by physical wires (wireless connection).

Use

- **Place > Net Alias**
- **N** (keyboard)
- **Net Alias Icon** 

Note

When you need to connect nets over several pages or even design hierarchies, you must make use of off-page connectors and hierarchical ports. Please check with the tool documentation for details.






Tips for a Successful Simulation

To run a simulation of a circuit created with Capture, follow these general recommendations:

1. All components must have PSpice models attached. When placing components using the Place Part method, make sure that you are using components sourced from the default PSpice Library at:

```
<install_directory>\tools\capture\library\pspice  or  
<install_directory>\tools\capture\library\pspice\advanls
```

The default path with Capture points to `<install_directory>\tools\capture\library` and will use libraries without a PSpice model attached!

2. PSpice needs a reference potential called “0” to run the simulation. Usually you place a global net symbol “0”  from the capsym library and connect it to the reference net(s).
Use e.g. **Place > PSpice Part... > PSpice Ground** to get such a symbol.
3. Start with smaller circuits, rather than large ones. Divide larger circuits into smaller subcircuits.
4. Think before blindly accepting a simulation result. Apply **Rule #9*** before you run a simulation.

* Source: [Signal Integrity Journal](#)



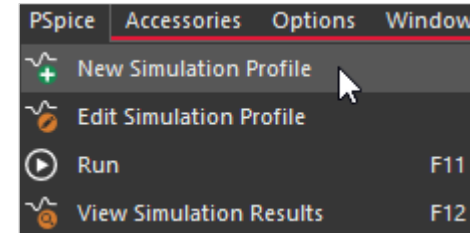
Basic Types of Simulation in PSpice



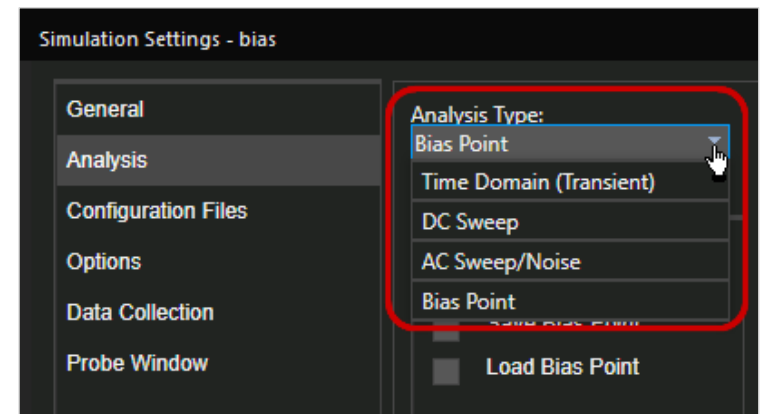
Types of Simulation in PSpice

First you must setup a **simulation profile** before you can run a PSpice simulation. Here the type of the simulation and all of its input parameters are set. PSpice supports 4 basic types of a circuit simulation:

- **Bias Point (or DC) Analysis**
 - Static analysis, analyzing the operating and equilibrium point
- **DC Sweep**
 - Multiple DC Analysis with one variable parameter
- **Transient (time domain) Analysis**
 - Analyze the response of the circuit as a function of time
- **AC Sweep**
 - Frequency-domain analysis resulting in amplitude and phase of currents and voltages



Create a new simulation profile




Selecting the simulation type



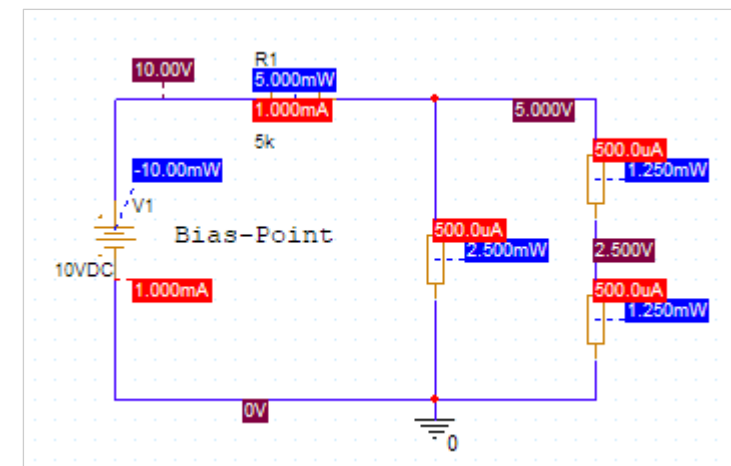
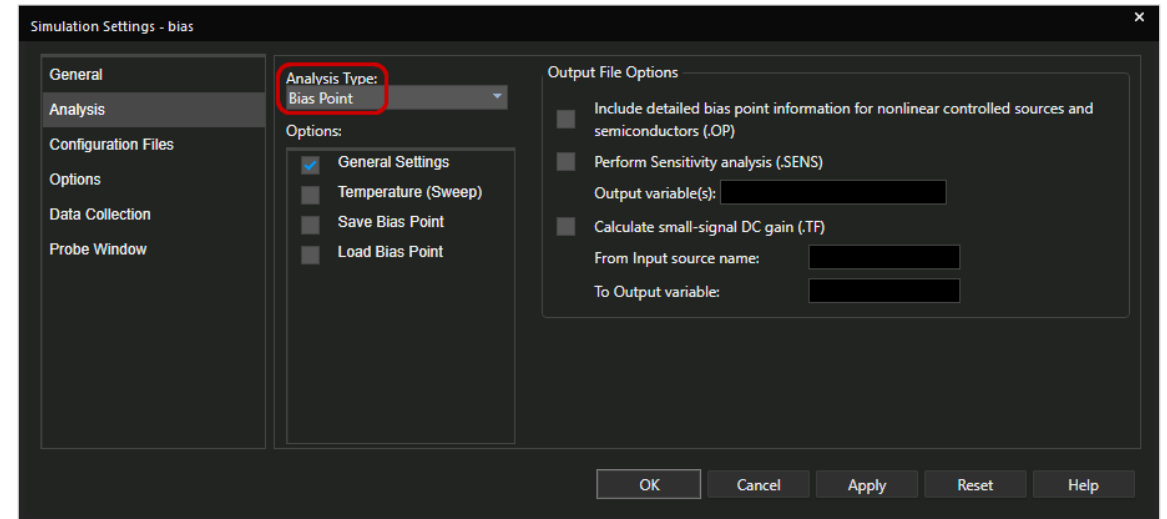
Bias Point, DC Analysis

The Bias Point analysis finds an operating point of the circuit which are assumed to be equilibrium points – solutions which do not vary over time (static).

- Static voltage, current or power values can be displayed with **PSpice > Bias Points > Enable** or using the icons in the toolbar: 
- Variants of the DC Analysis
 - DC Sensitivity Analysis
 - Transfer function (DC gain) analysis

Example

- See the example **bias_point** in folder **Solution\Samples\bias_point\bias_point.opj**





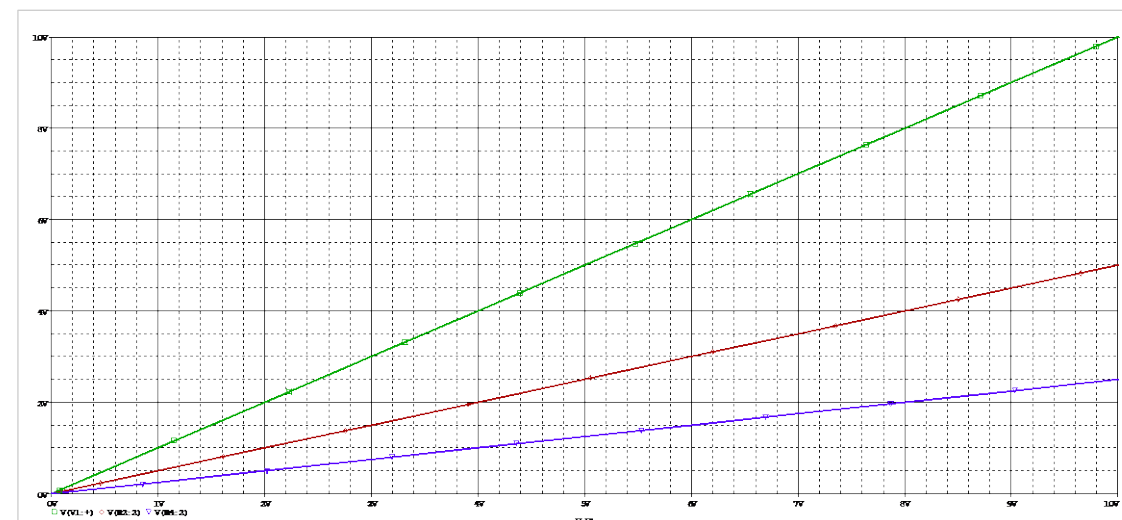
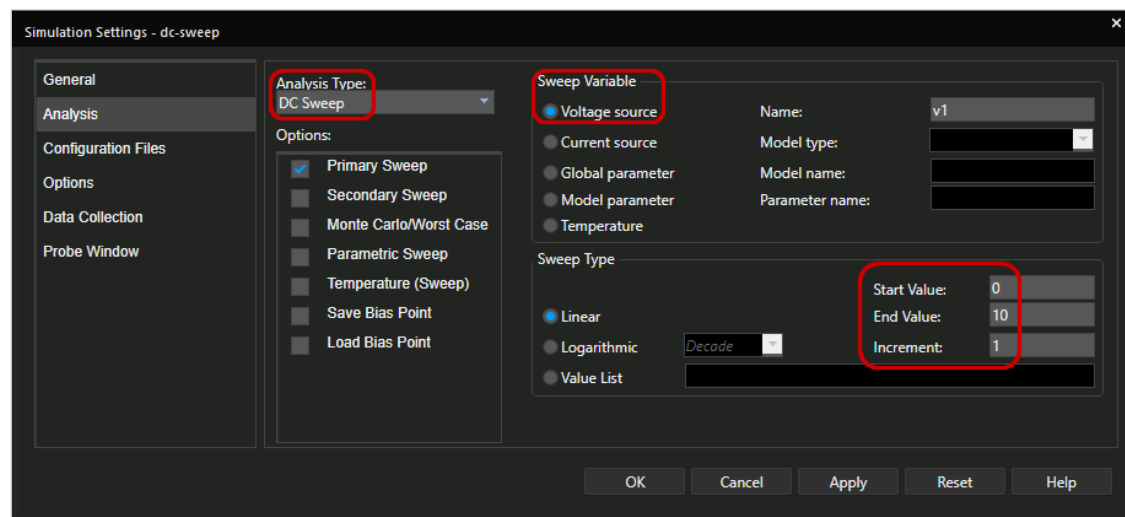
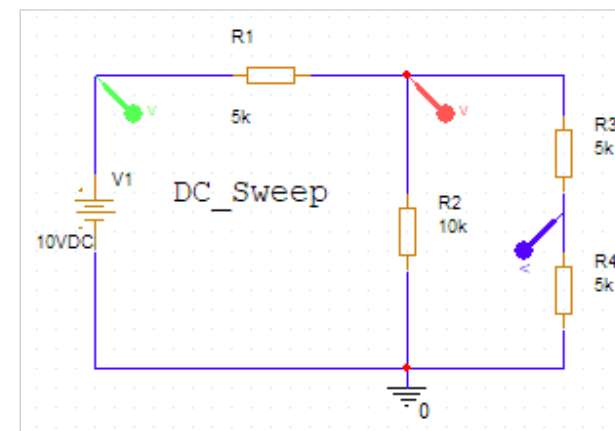
DC Sweep Analysis

The DC Sweep analysis allows you to vary (sweep) a source (voltage or current), a global parameter, a model parameter or the temperature through a range of values. For each sweep value, the bias point is calculated.

- The results are put into the simulation output file (.out) or can be graphically displayed in Probe.

Example

- See the project **DC_Sweep.opj** in the folder **Solution\Samples\DC_Sweep**





Time Domain (Transient) Analysis

The transient analysis computes the response of a circuit as a function of time. PSpice will display the results in probe.

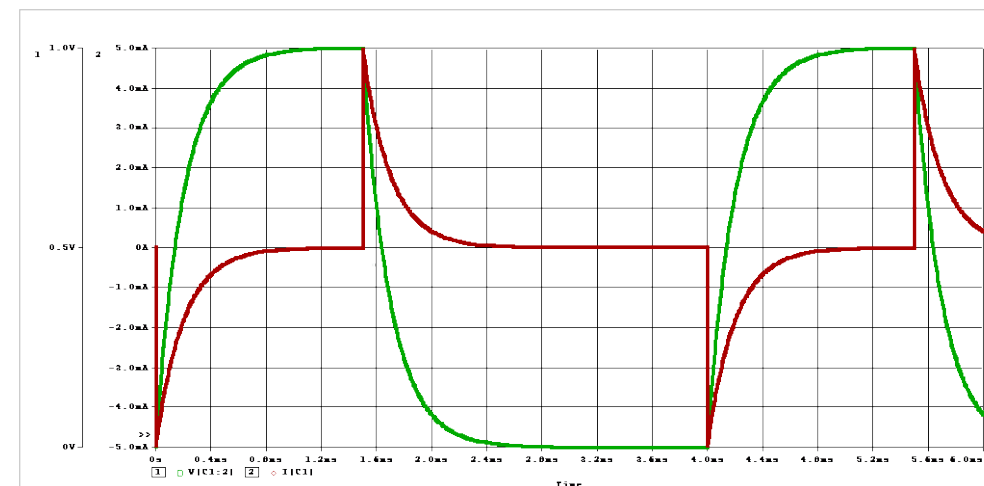
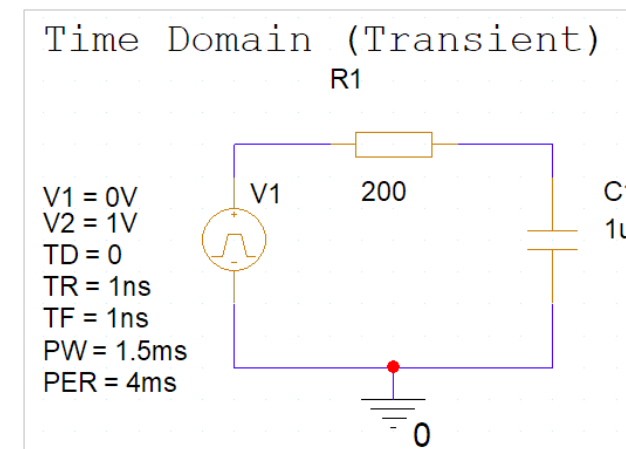
For the transient analysis, the simulator needs to solve nonlinear differential equations in the time domain. The results will be node voltages and currents over time. Visualizing the results in Probe is like working with an oscilloscope.

Example

- Project `time_domain.opl` in the folder **Solution\Samples\time_domain**
- The circuit will demonstrate how a capacitor is charged and discharged through a resistor by a periodic pulse source.

Tip

- You can change the color settings of the simulation output window in **Tools > Options**. Select the **Color Settings** tab.



Probe result: Charge Current (red), Capacitor Voltage (green)



AC Sweep Analysis

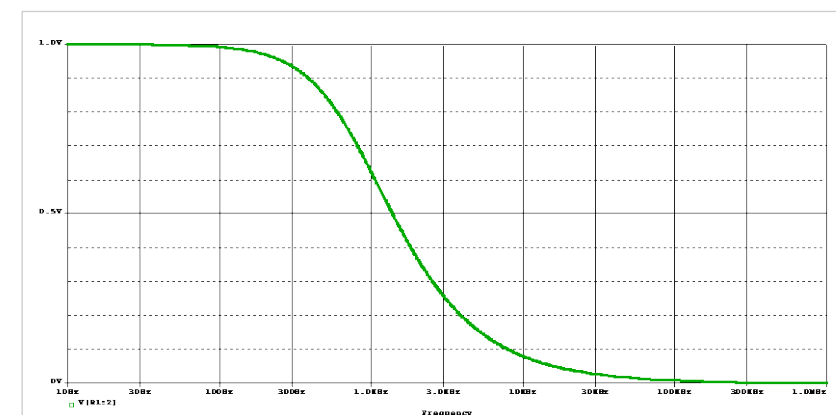
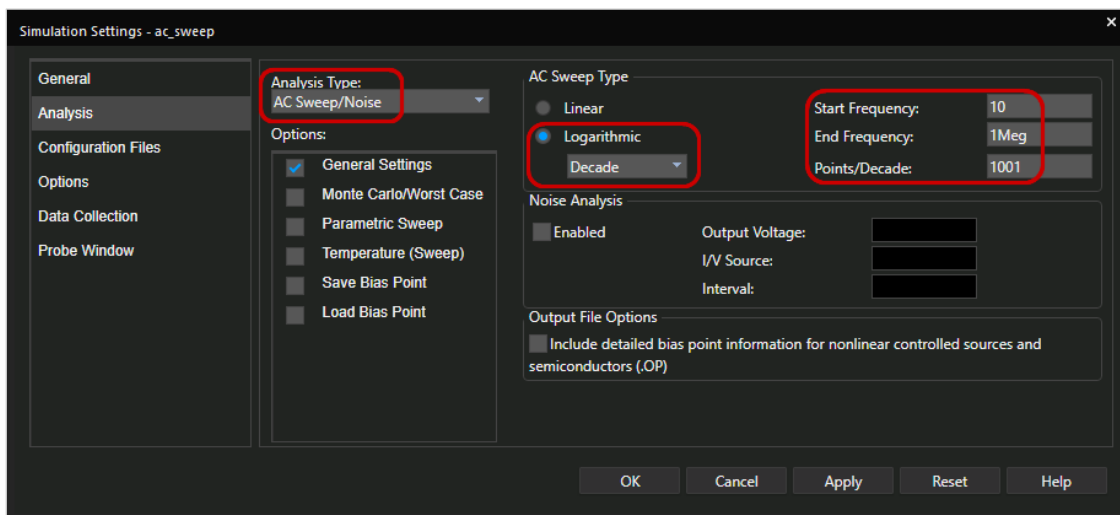
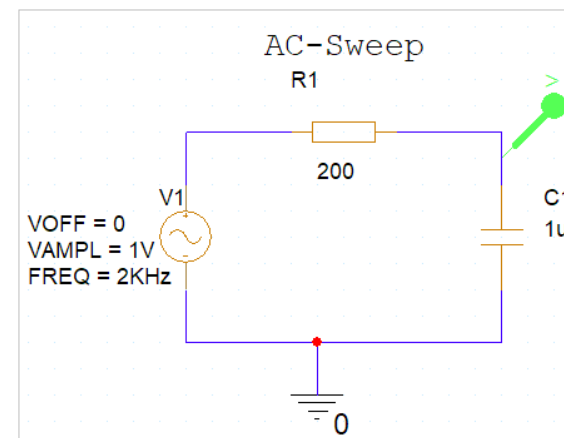
PSpice calculates the small-signal sinusoidal steady state response of the linearized circuit around the DC operating point. The AC Analysis is a frequency domain simulation.

The results are voltages and currents which can be represented with two numbers: the magnitude and the phase.

All sources in the circuit will operate with sinusoids at the same frequency as the input signal.

Example

- Project **AC_Sweep.opj** in the directory **Solution\Samples\AC_Sweep**



The magnitude of the capacitor voltage over the frequency



More Simulation Options

In PSpice you can run further simulations options, which are related to the basic simulation types presented so far:

- Fourier-Analysis as part of the transient analysis.
- Temperature sweep: See the effects of your circuit depending upon the temperature.
- Worst Case and Monte Carlo Analysis: Simulate the effects of tolerances and predict the yield.
- Sensitivity Analysis: Find out, which component values will have a large effect on the output value. Optimize cost and increase quality of your design.
- Parametric Sweep: Modify parameters of the circuit and visualize their effects.
- Noise Analysis: as part of the AC Sweep analysis.
- Digital Simulations: Analyze digital circuits in an event driven simulation.

In this quickstart we will make use of the Parametric Sweep and the Fourier-Analysis.



Design Example: Switched Mode Power Supply (SMPS)



Goals

- You will design the SMPS circuit shown earlier. In this design, a 230 V / 50 Hz source will be converted into a 5 V DC output.
- In this circuit, no PWM controller IC is used. Instead a constant 333 kHz voltage pulse with a 20 % duty cycle will be used to control the MOSFET.
- You will define different simulation profiles and analyze the results in the probe window using markers and additional measurements.

Note 1

The complete circuit can be found in the folder **Solution\Demo_Proposed_Circuit**.

Note 2

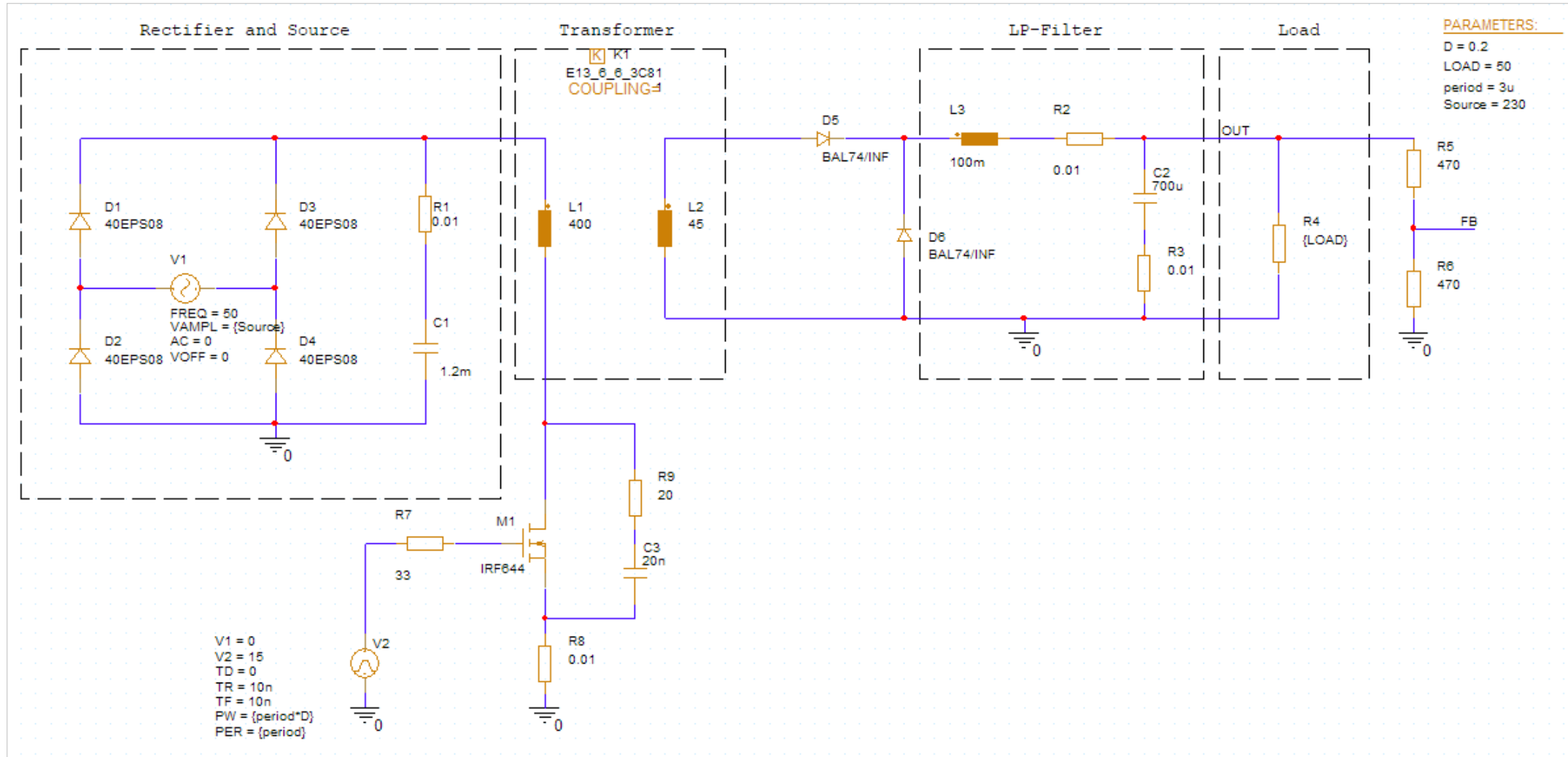
You will achieve the best learning and understanding how to use the software, when you follow the all the steps in this quickstart.



SMPS Design (I)

- You created the SMPS project already earlier in this guide. If not, then please find the necessary steps on the pages [13](#) and [14](#).
- If not already open, open the PAGE1 of the project SMPS in Capture.
- Recap the slides on how to find components, how to perform wiring and how to change values of various components.
- Ensure, that the component values are matching with the ones shown in the circuit diagram.
- There are extra slides, how to create the coupling device K1 and how to setup the parameters.
- When you will use the traditional way of placing components (Place Part), you will need components from the libraries: Infineon.olb, irf.olb, magnetic.olb, PWRMOS.olb, source.olb and special.olb.
- When you want to use the European Style components, you will need to configure the Europe.olb, shipped with this tutorial, and then place them using the **Place Part** command. You can use the analog.olb if you want to use American schematic symbols.
- There will be some additional hints, on how to design the circuit.

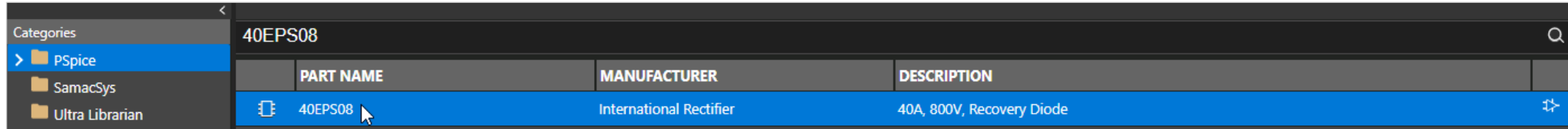
Schematic of the SMPS Design (European Style)



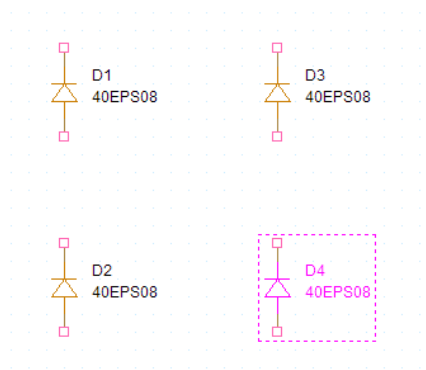


Design the Rectifier and Source

- Select **Place > Component...** and enter 40EPS08 in the Search field. Double-click on the component shown:



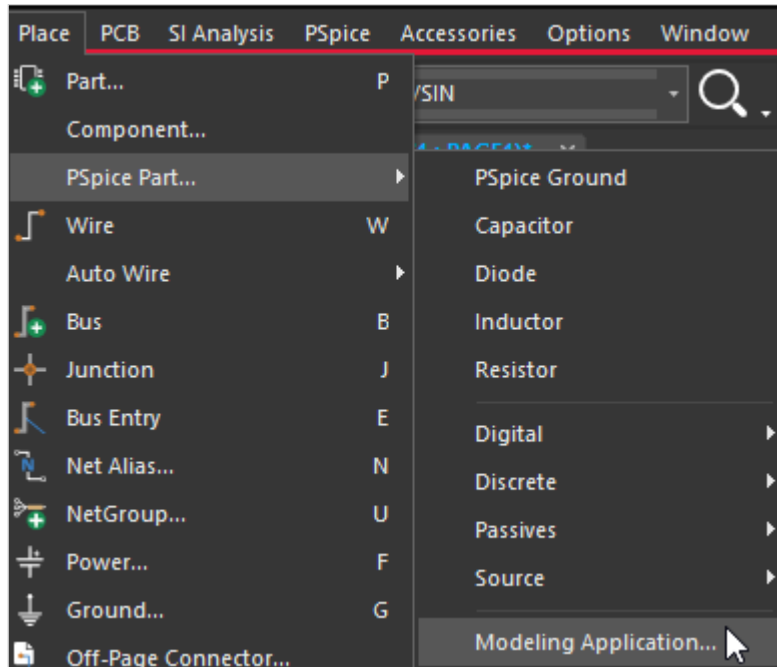
- Rotate the diode (R) and place 4 instances of it. Press **ESC** to leave the placement mode.



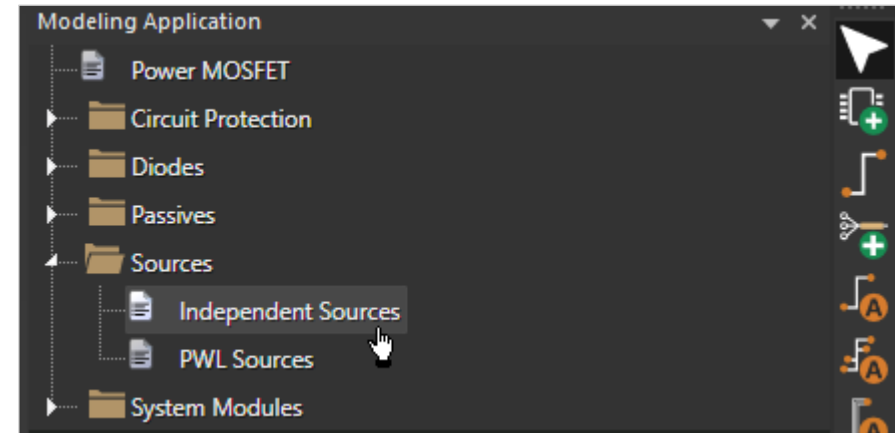


Use the Modeling Application to Place the Sine Source

- Use **Place > PSpice Part... > Modeling Application...** or click on this icon in the PSpice toolbar



- Select Independent Sources

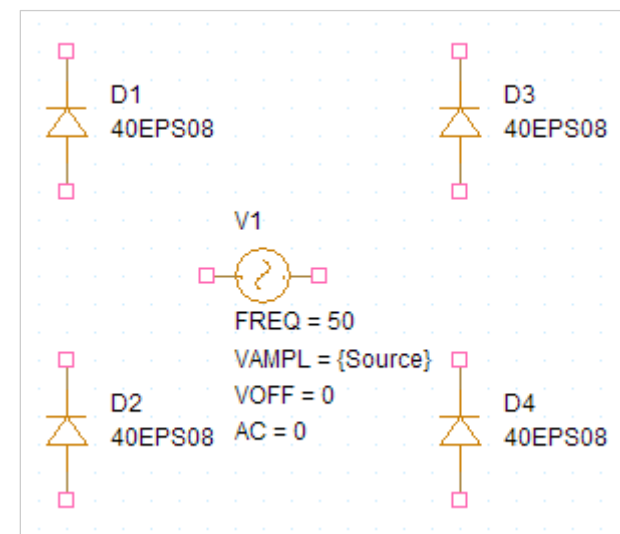
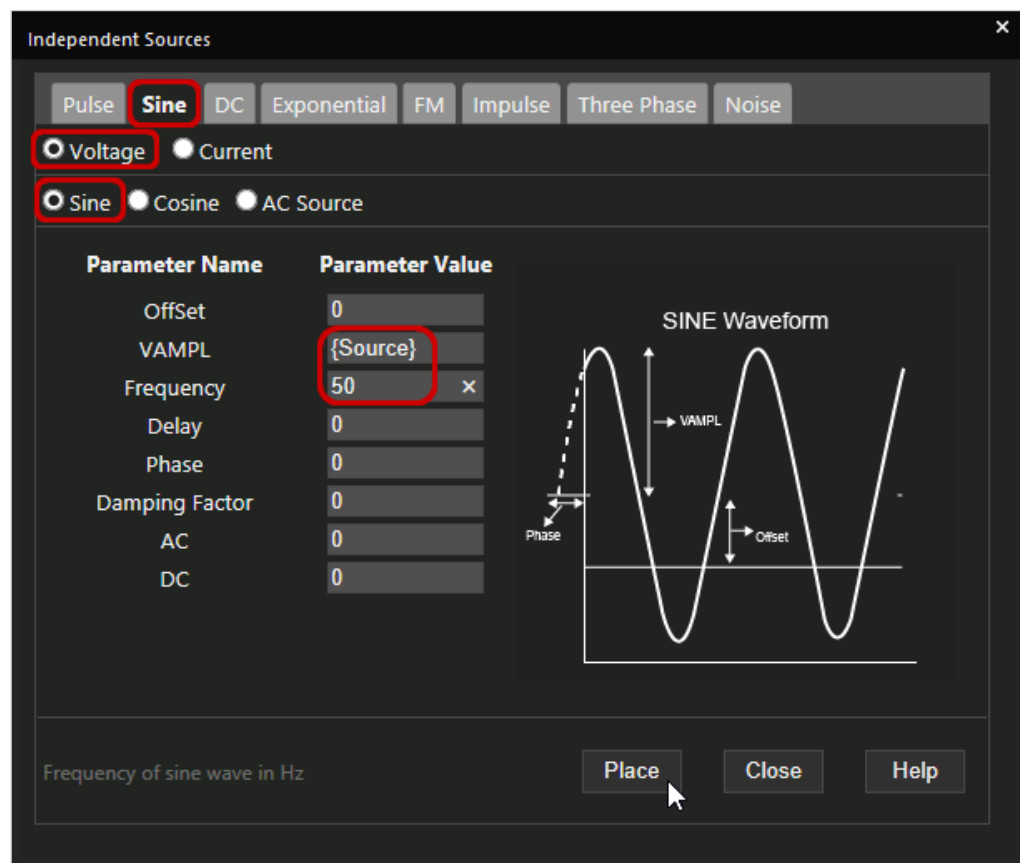




Define the Sine Source and Place it

- Select as shown and enter **{Source}** in the VAMPL parameter, and **50** as the Frequency. Then click **Place**.

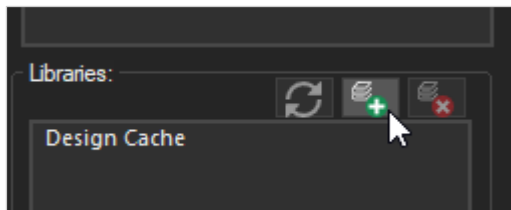
- Rotate the source once and place it.



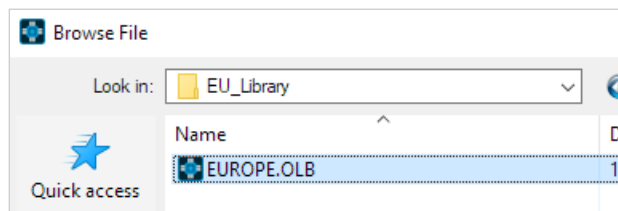


Place Parts From the Europe.olb

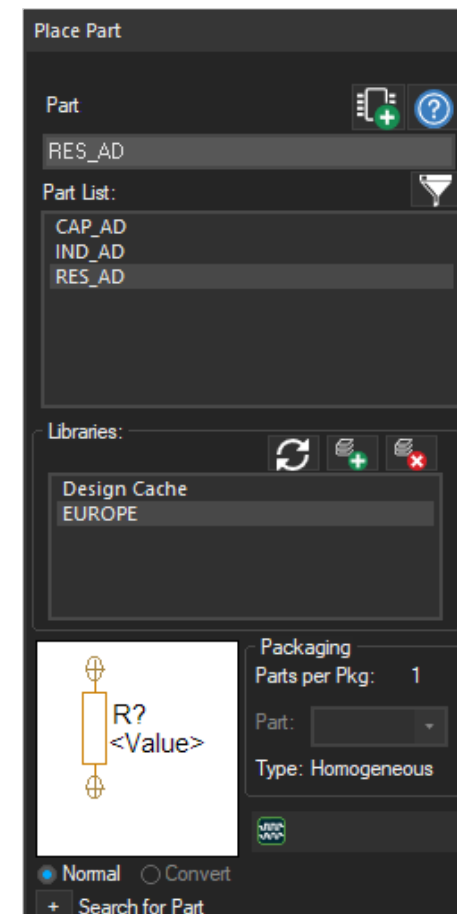
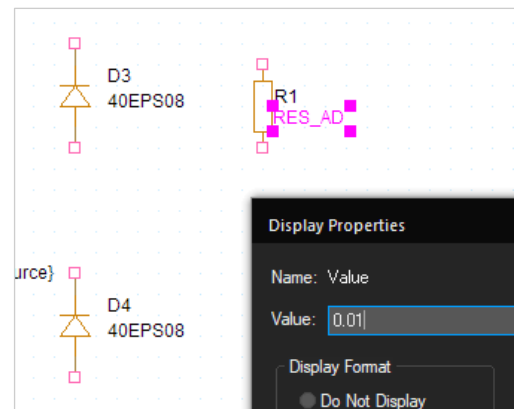
- We use the **Place Part** command for all parts, which are not shipped with PSpice.
- In the active Schematic press **P**.
- In the Library section of the Place Part Window, click the **Add Library** icon



and browse to the PSpice_Demo_22_1\Solution\EU_Library\Europe.olb on your disk and click **Open**.



- You will now find three components in the Part List from which you can choose from:
- Double-click **RES_AD** and place it on page1.
- Double-click the RES_AD text and change the value to 0.01.

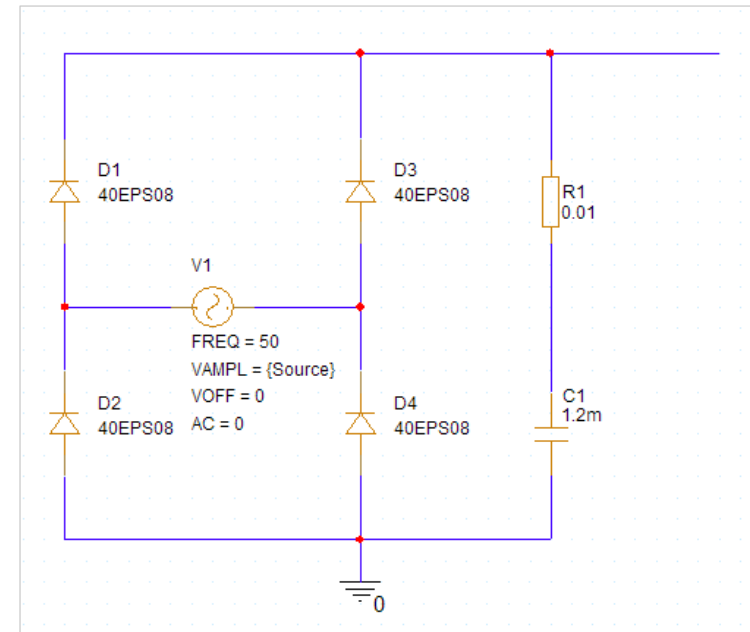
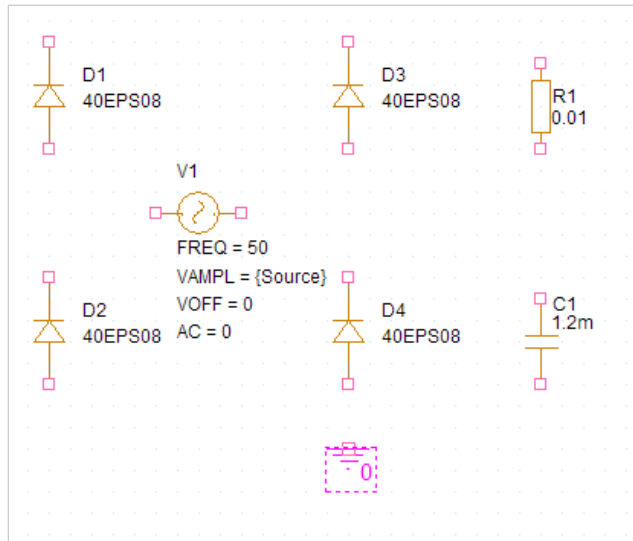




Finalize the Rectifier and Source Part of the Schematic

- Applying the same method, place the Capacitor C1 and change the value to 1.2 m.
- Select **Place > PSpice Part... > PSpice Ground** and place a "0" Symbol.

- Press **W** to enter the Wiring Mode.
- Draw all the required connections:





Finalize the Rest of the Schematic

- You should now be able to finalize the rest of the schematic by applying the methods shown before (excl. K1 and PARAMETER).
- Hint:** The Source driving the MOSFET is an independent Voltage Pulse.

- The Time Period Parameter does currently not accept an expression value. As a workaround, leave the fix value and replace the parameter on the schematic by double-clicking on the fixed value and entering the correct **{period}** expression for the value:

Independent Sources

Pulse Sine DC Exponential FM Impulse Three Phase Noise

Voltage Current

Step Pulse Square Ramp Sawtooth Reverse Sawtooth Triangular

Parameter Name	Parameter Value
V1	0
V2	15
Delay	0
Rise Time	10n
Fall Time	10n
Pulse Width	{period*D}
Time Period	10u
AC	0
DC	0

PULSE Waveform

V2

Time Period

Pulse Width

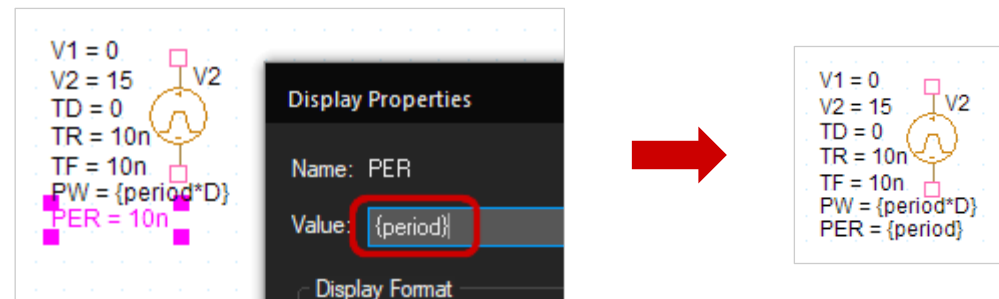
V1

Rise Time

Fall Time

Periodic rectangular pulse voltage source for time domain analysis

Place Close Help

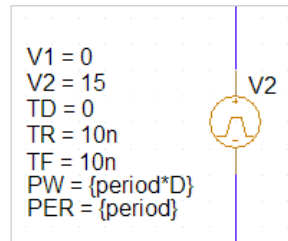
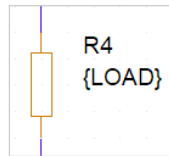
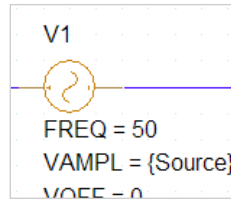


- The PARAMETER and the K1 symbols will be placed next.

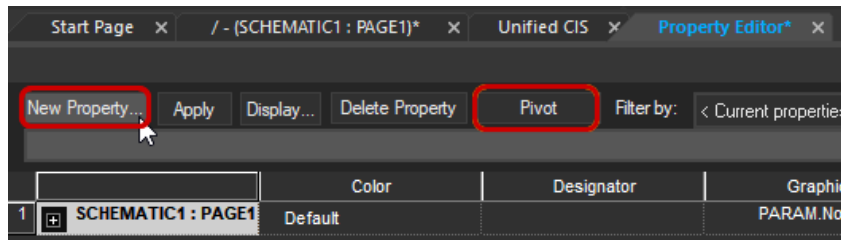
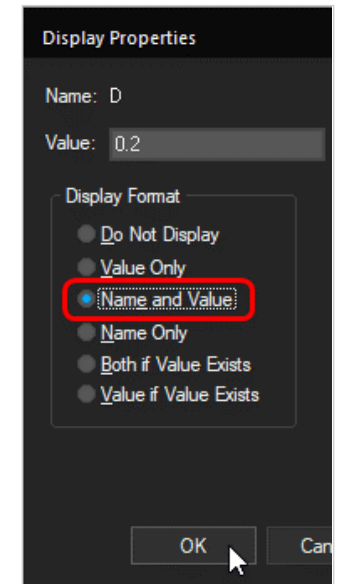
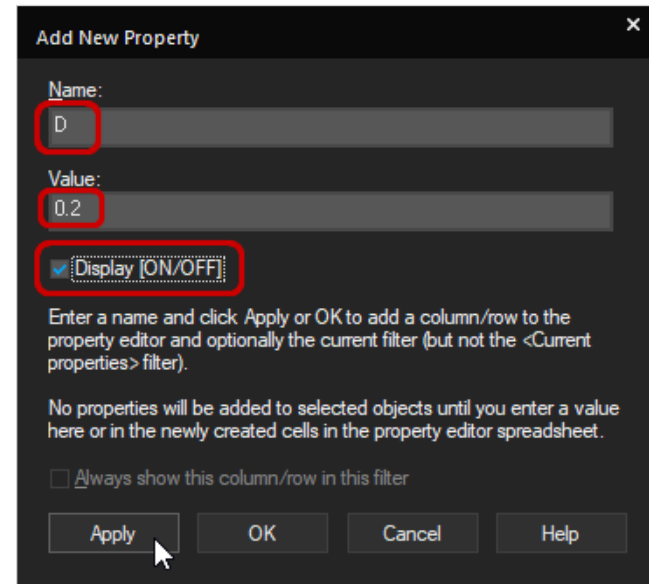


Add the Global Parameters

- Place the inductors with the correct orientation (see the dots).
- Adapt all values according to the design.
Note: Global Parameters values are defined between curly braces { }.
- Don't forget to label the nets OUT and FB.
- Put the grounds into the right locations.
- In Unified CIS search for the component **PARAM** and place it on a free space in your schematic.
- Double-click** on the PARAMETERS component. The Property Editor will open:

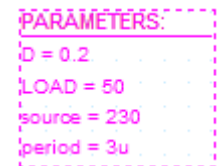
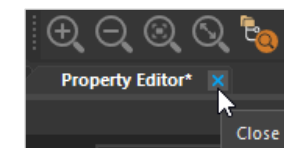


- Enter the Name **D** and the Value **0.2**, check the **Display** (ON / OFF) and hit **Apply**.



- For your curiosity: Check what **Pivot** does.
- Click on **New Property**.

- Select **Name and Value** and click **OK**.
- Repeat the steps for LOAD (50), period (3u), Source (230).
- Close** the Property Editor.



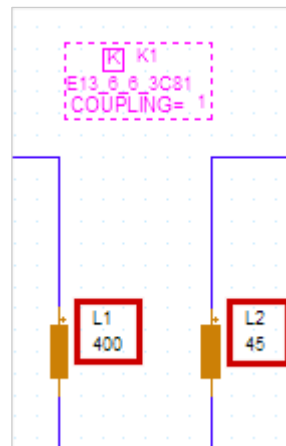


Build the Transformer

We build a nonlinear transformer based on the Jiles Atherton simulation model.

- In Unified CIS search for the core **E13_6_6_3C81**.
- **Double-click** on the only part returned from the search and **place it** on the schematic. A good place would be above the coils L1 and L2, but the location does not matter.
- **Double-click** on the placed part in the schematic (**Tip: Double-click on the K**).
- Tell the core model, that you will couple the inductors L1 and L2.

A	
SCHEMATIC1 : PAGE1	
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	



Note

The values of the inductors L1 and L2 refer to number of turns in conjunction with the simulation model used for the transformer.


- **Close** the Property Editor.




Build the Simulation Profile

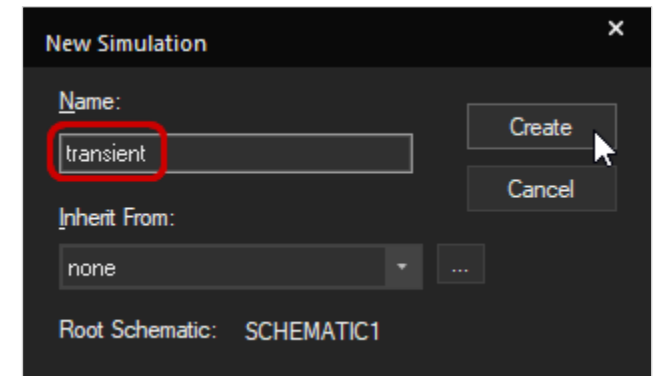
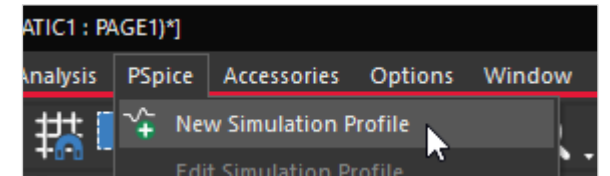
The circuit is complete now and should look like the reference.

Next step is to setup a simulation profile for a transient simulation.

- Click on **PSpice > New Simulation Profile** or click the icon 
- Name it **transient** and click **Create**.

Note

- You can create more than one simulation profile for the active schematic.
- Modifying an active profile is done with **PSpice > Edit > Simulation profile** or use the icon 





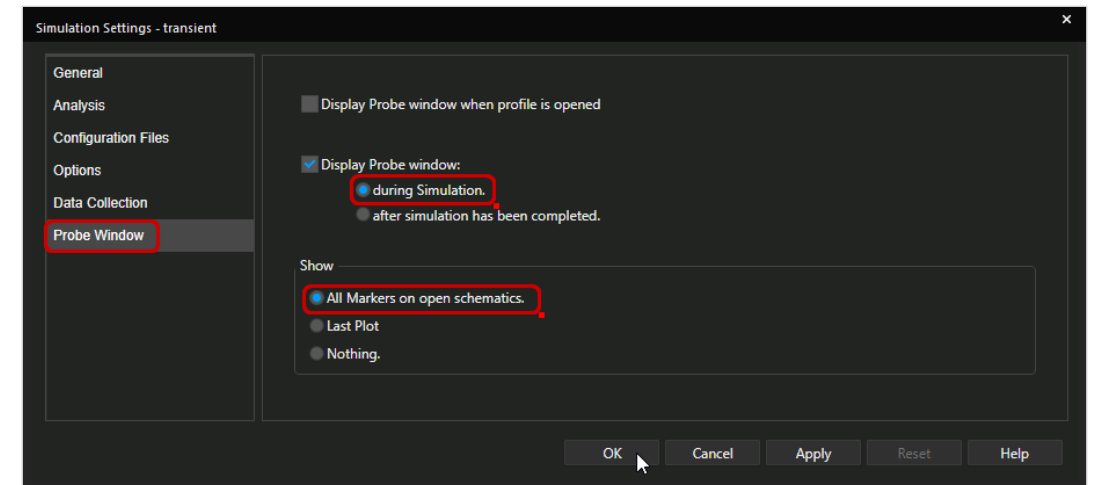
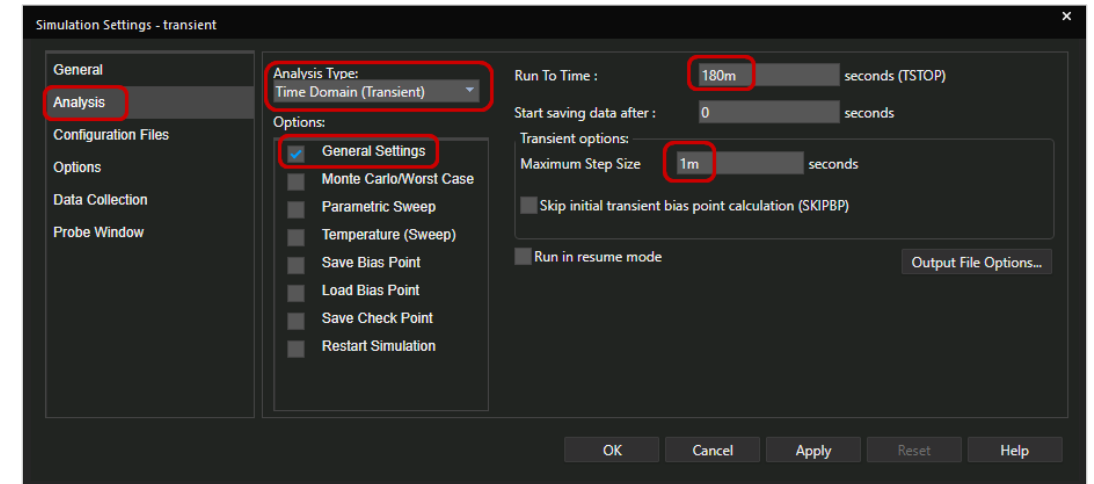
Add the Simulation Details

The Simulation Settings window pops up.

- Make sure to select the **Time Domain (Transient)** as Analysis. The General Settings Option should be pre-selected. You enter the Run to Time as **180 ms** and the Maximum Step Size to **1 ms**.
- On the left side, click on **Probe Window**. Select **during Simulation** and **All Markers on open schematics** as display options for Probe.
- Press **OK** to close the simulation profile.



Note

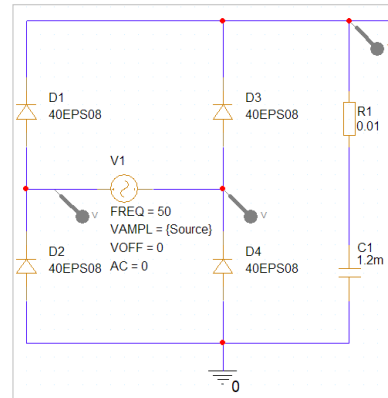
- When the Maximum Step size field is left blank, a default value of 2 % of the Run to Time is used. The resulting curve displayed in Probe may be not accurate enough.
- Entering a too small value will increase the simulation time and the data generated.
- A reasonable value is about 1/1000 of the Run to Time.





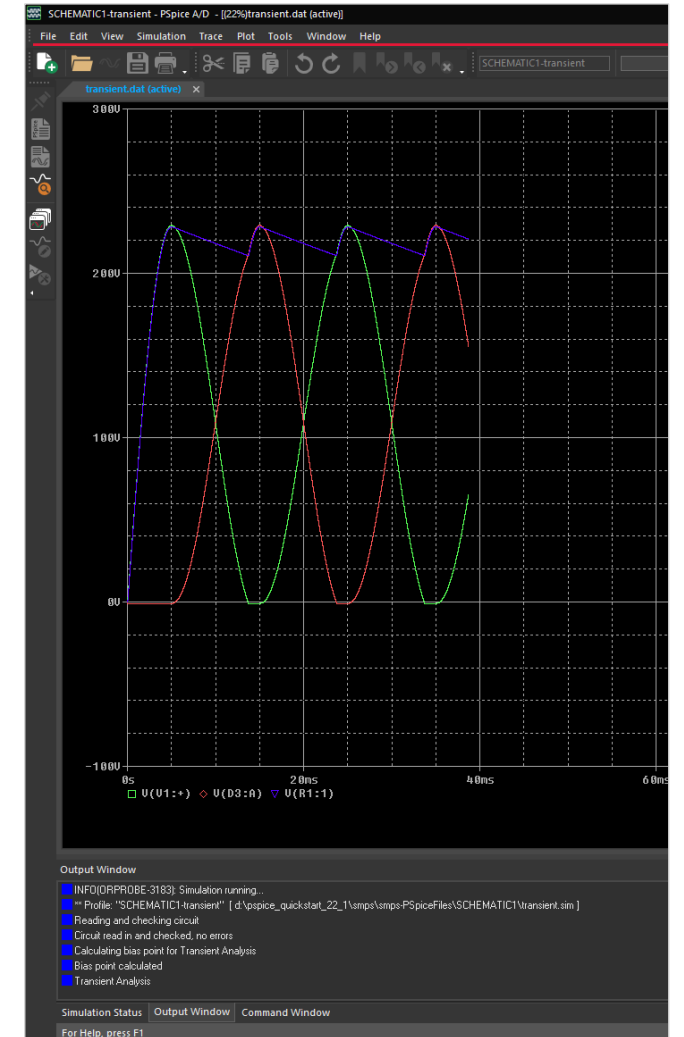
Run the Simulation

- Place voltage markers (**PSpice > Markers > Voltage Level**) or use the icon  as shown in the picture on the bottom (the colors occur, when the simulation starts).
- Press **F11**, or click  to start the simulation.
- When the circuit was entered correctly, you will see that the Probe window will open and the waveforms for the node voltages will show-up whilst the simulation is running.
- Ignore the large data file warning you may receive at the end of the simulation (which takes app. 2 min depending on the CPU type).



Note

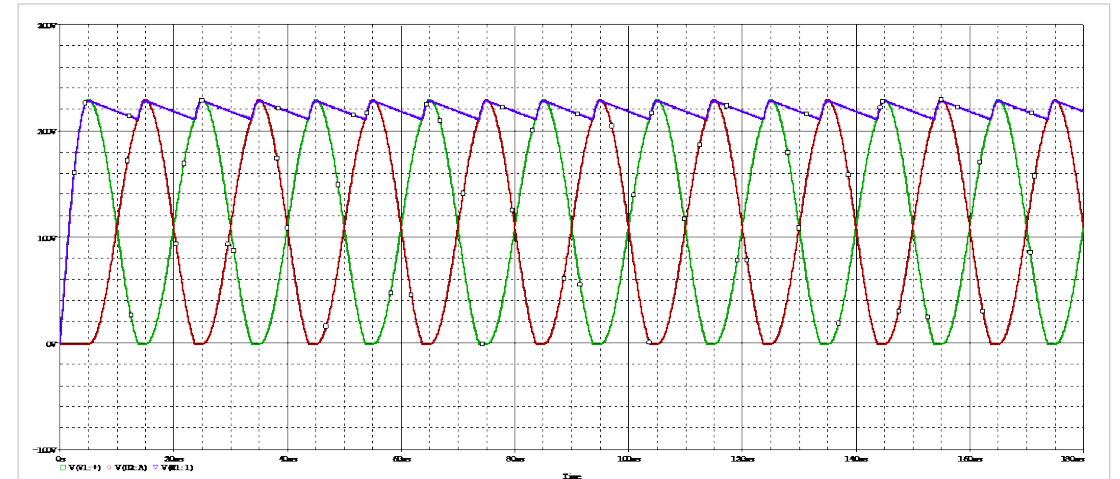
- Place **V markers** on the net, **I markers** on the component pin and **W markers** on the component.





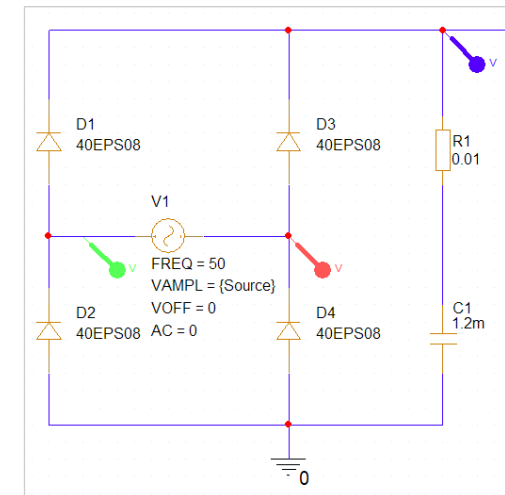
Simulation Results: Rectified Voltage

- You added already 3 voltage markers:
 - Blue: the rectified voltage
 - Red / Green: the voltage at the AC source
- Evaluation:
 - The reservoir capacitor C1 flattens the rectified wave. The large value of the capacitor has a sufficient effect, that the resulting waveform is already quite flat.



Note


- You can add or delete more voltage markers after the simulation and see the immediate effect on the probe window.
- In Probe you can also use **Trace > Add Trace** to display desired traces.
- The color assignment of the traces is done according to a predefined sequence in the pspice.ini file. You may see different colors at your end.





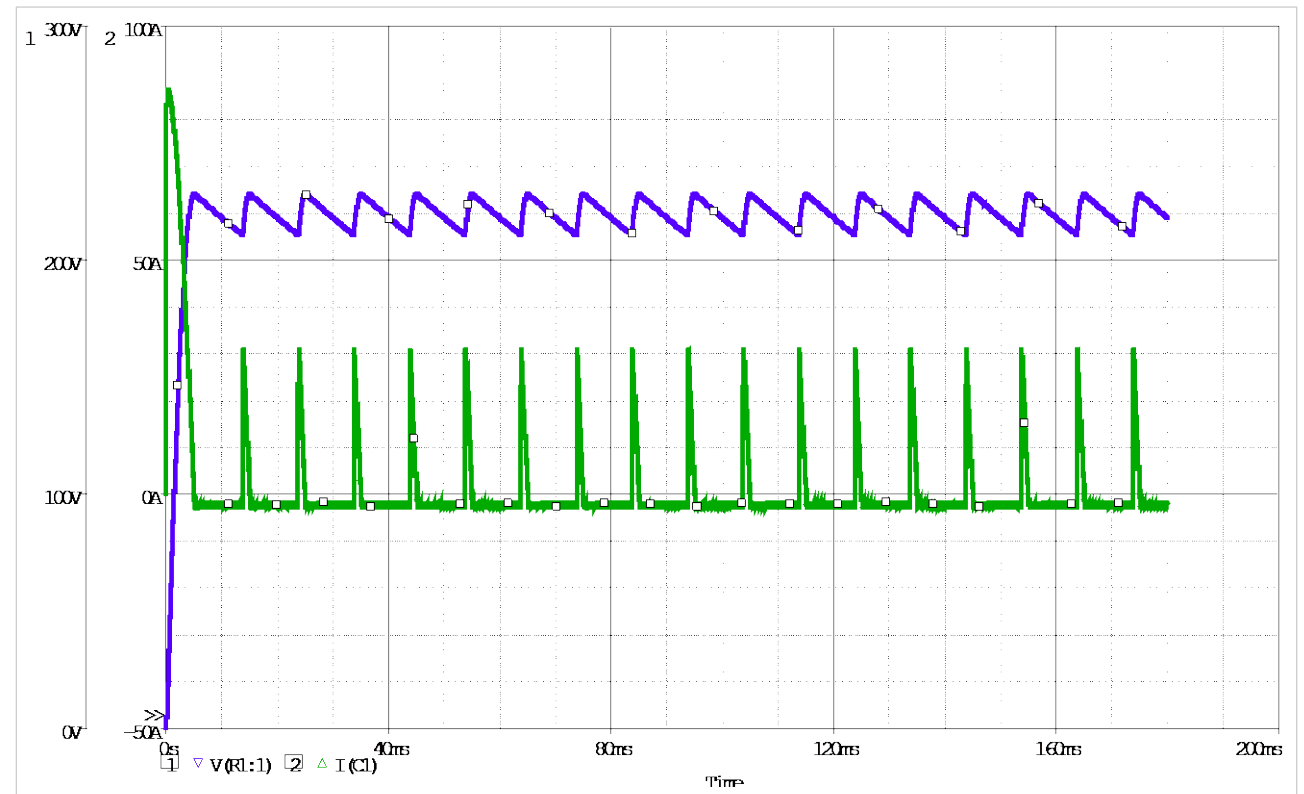
Current Voltage Behavior at C1

Looking at the U-I curves at C1, you see every 10 ms (100 Hz) charging current spikes and a high frequency discharging current. The relatively flat voltage depends on the high capacity value of the capacitor.

- Delete the previous traces of the AC source by **removing in our case the red and green voltage markers** in the schematic.
- In Probe, select **Plot > Add Y Axis**.
- **Trace > Add Trace**, or click  Then click on **I(C1)**.

Note

- If the current I(C1) is negative in your case then your Capacitor C1 is rotated by 180°. PSpice measures the positive current from pin1 to pin 2. When you rotate C1, the sign of the current is negated. This is not an error.

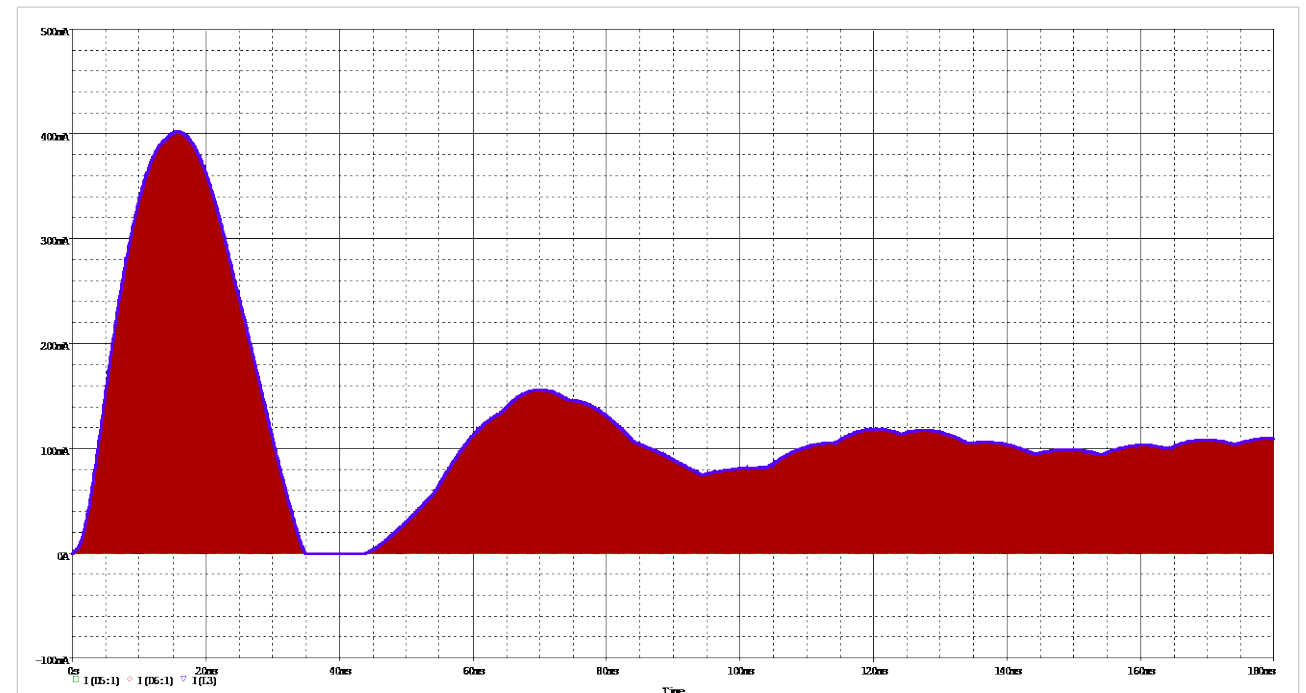
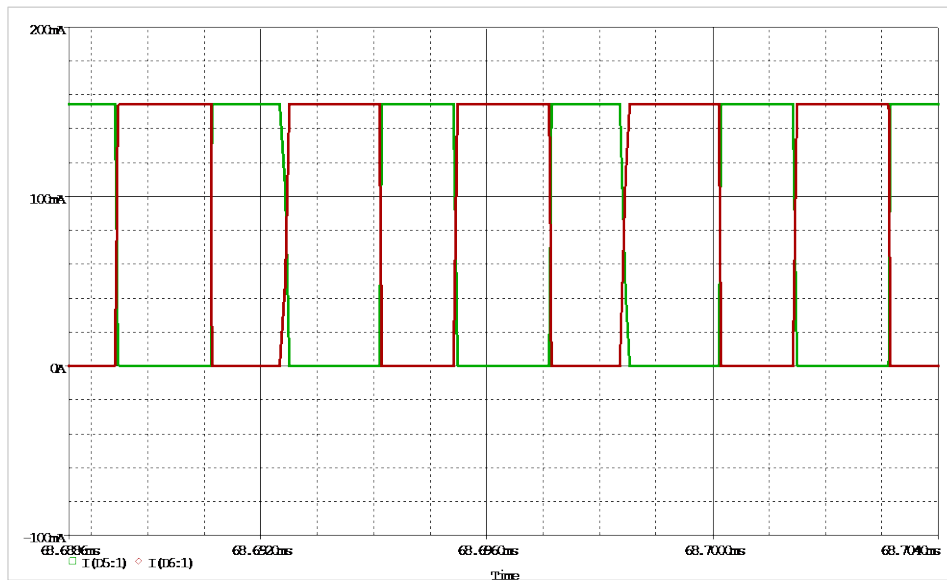




Generation of HF Signals

The MOSFET is turned on and off by the generated voltage pulse of approximately 330 kHz, causing a HF DC current through L1, inducing a HF AC voltage in L2 of the transformer.


The currents on the secondary side of the circuit are now isolated and converted to DC using the diodes D5 and D6 (green and red). Both currents are combined again entering L3 (blue curve).

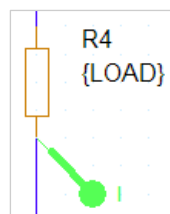
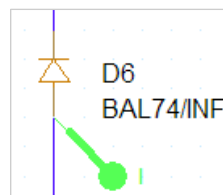




The Low Pass (LP) Filter

The current through the Diodes contain HF components. The LP filter should eliminate those components. To verify this, we will compare in two separate plot windows the current through, e.g. D6 and through the load resistor R4.

- Close Probe and delete all Markers in the schematic.
- Press F12, or Select **PSpice > View Simulation Results**  Probe will open again with no plot visible.
- Put an Current Marker at the Anode of D6.
- In Probe **Plot > Add plot to Window**.
- Add a Current Marker at Pin 1 of R4.



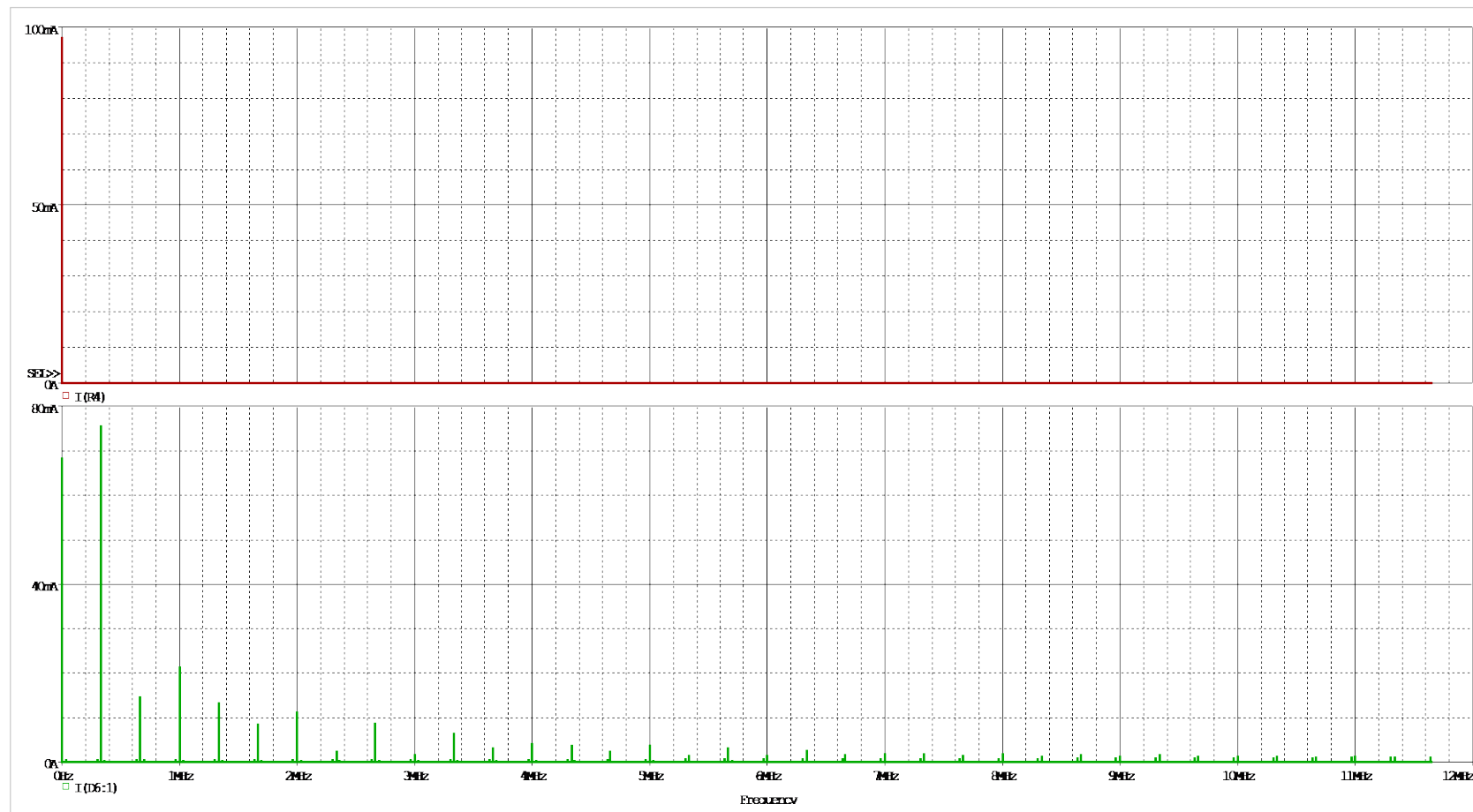


The Fast Fourier Transformation (FFT)

- Press the FFT button in the top tool bar:



- Only a DC component in the current through the Load is left (top plot).
- You will see many HF components in the Diode current (bottom plot).

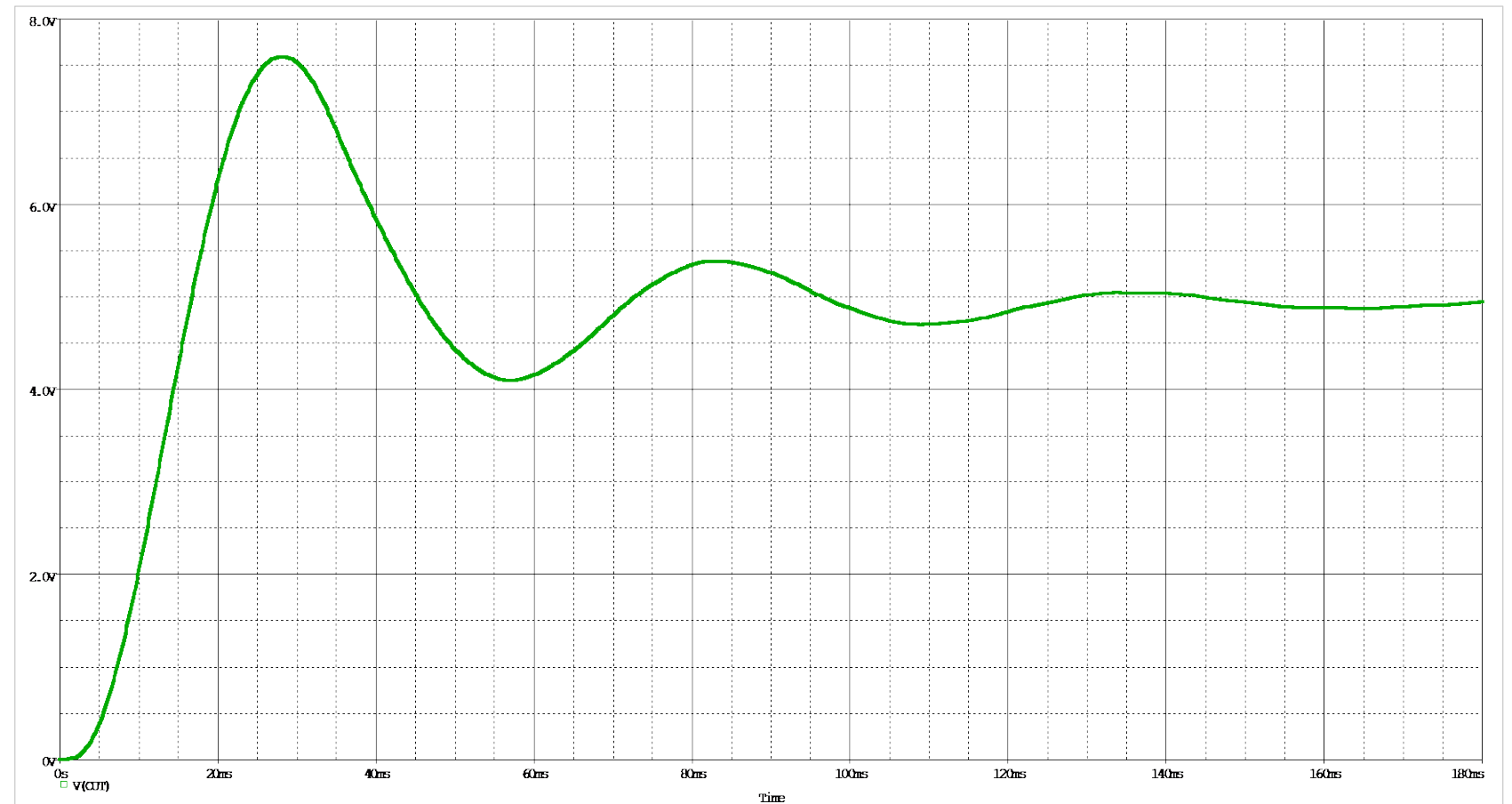




Output Voltage

- Delete all the current markers in the schematic.
- In Probe: **Plot > Delete Plot.**
- Place a **voltage marker** on the **OUT** wire.

The DC voltage rises oscillating until it is stabilized to a value of approx. 5 V (transient response). The smoothness is achieved by the capacitor C1 and the low pass filter.





SMPS Parametric Sweep



Parametric Sweep, Step 1 of 4

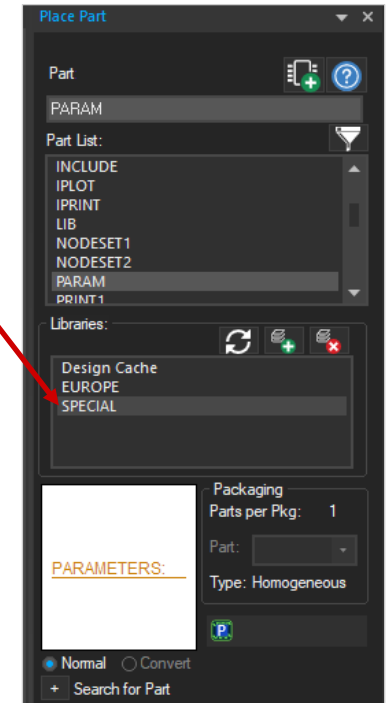
The Parametric Sweep allows to vary (sweep) a source (voltage or current), a global parameter, a model parameter or the temperature through a range of values. E.g., you can simulate a circuit with different component values and display the results with a single call.

You already prepared the design for a parametric sweep, but lets recall the 4 required steps to vary a global parameter (as one example):

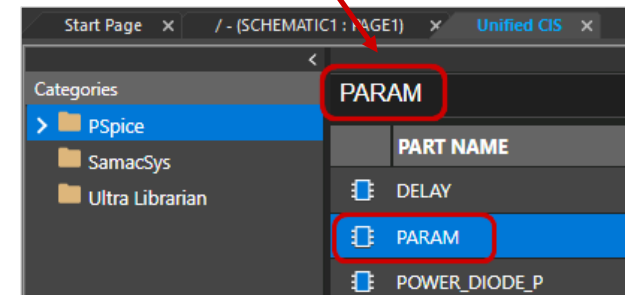
- **Step 1: Define the global parameter**
 - Place the **PARAM** symbol somewhere on the schematic.
 - It serves as a placeholder for the global parameters and defines default values.

Finding the **PARAM** Symbol, either use....

a) Place Part method
Special.olb must be configured



b) Search in Unified CIS



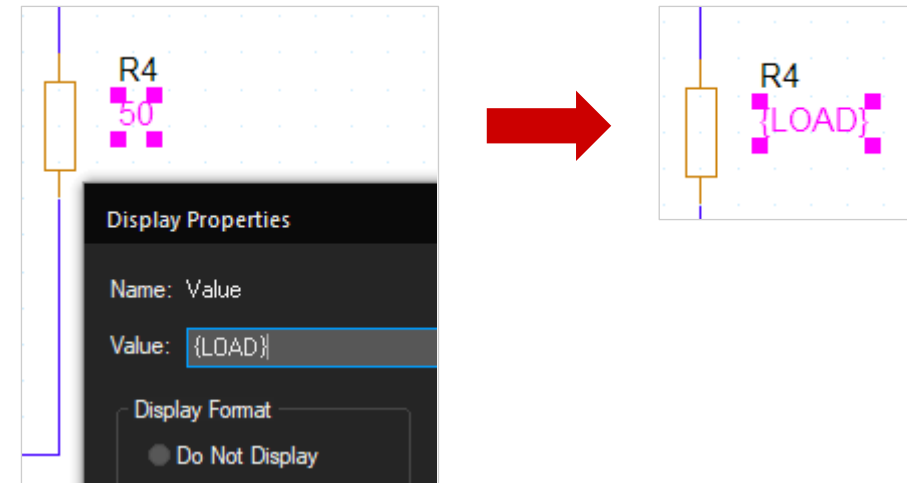


Parametric Sweep, Step 2 of 4

- **Step 2: Substitute the fixed device value with a parameter value**
 - **Double-click** on the value of the device.
 - Substitute the device value by the defined parameter name and **enclose it in curly braces { }** and click **OK**.

PARAMETERS:

D = 0.2
LOAD = 50
period = 3u
Source = 230





Parametric Sweep, Step 3 of 4



- **Step 3: Assign and display the global parameters in the schematic**
 - **Double-click** on the PARAMETERS schematic symbol to open the Property Editor.
 - **Click** New Property.
 - **Type in** the name and the default value of the new property.
 - Click **Apply** (for more parameters to edit), or **OK** for the last parameter.
 - On the Display Properties select **Name and Value**.
 - Confirm with **OK**.
 - Continue with all required parameters.
 - **Close** the Property Editor.

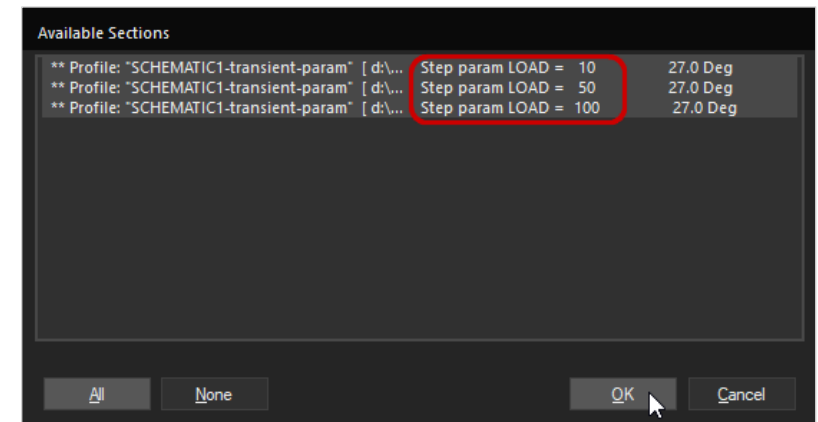
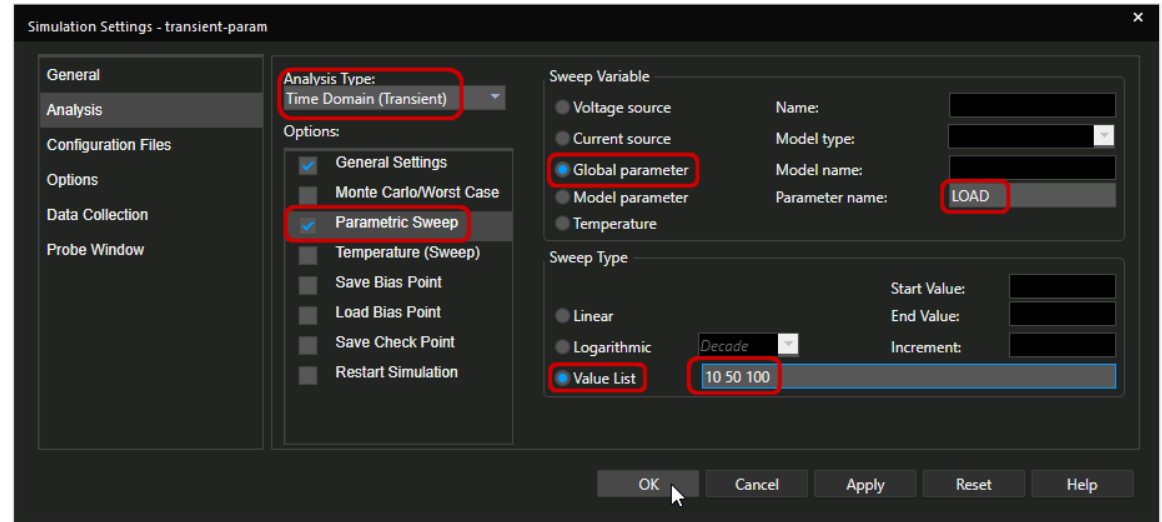
Now the parameters are defined and visible in the schematic.

PARAMETERS:
D = 0.2
LOAD = 50
period = 3u
Source = 230



Parametric Sweep, Step 4 of 4

- **Step 4: Set up a simulation profile with Parametric Sweep enabled**
 - Edit an existing simulation profile **PSpice > Edit Simulation Profile** or click the  icon.
 - Set the required simulation type: here **Time Domain**
 - **Optional:** In the **General Settings**, remove the value of 1m for the Maximum Step Size (to reduce the amount of data generated).
 - Select the **Parametric Sweep**.
 - Set the sweep Variable to **Global Parameter**.
 - Set the Parameter Name to **LOAD**.
 - Define the sweep values: here select **Value List**.
 - Enter the values **10, 50** and **100** separated by spaces.
 - Click **OK**.
 - Start the simulation with **F11** or 
 - Choose **ALL** and **OK** in the Available Sections window which occurs after the simulation.





Parametric Sweep Results (I)

You selected to show all sweep results, hence you will see 3 different traces in Probe.


Conclusion


A change in the load value will change the transient response of the output voltage.

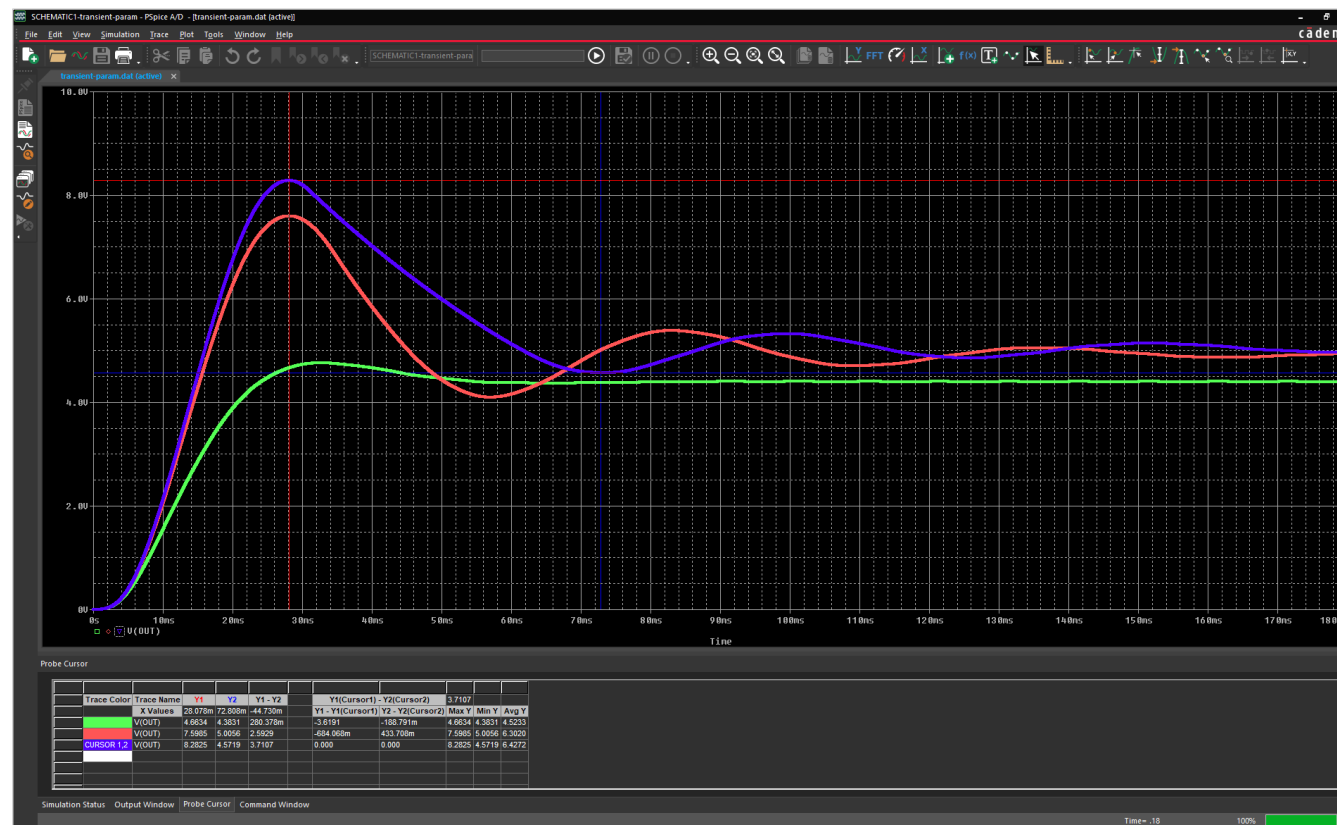
Notes

Click **RMB > Trace Information** on a single trace displays simulation details (like the step parameter).

Plot > Transient opens the section selection window again.

Trace > Cursor > Display or clicking  opens the Probe Cursor.

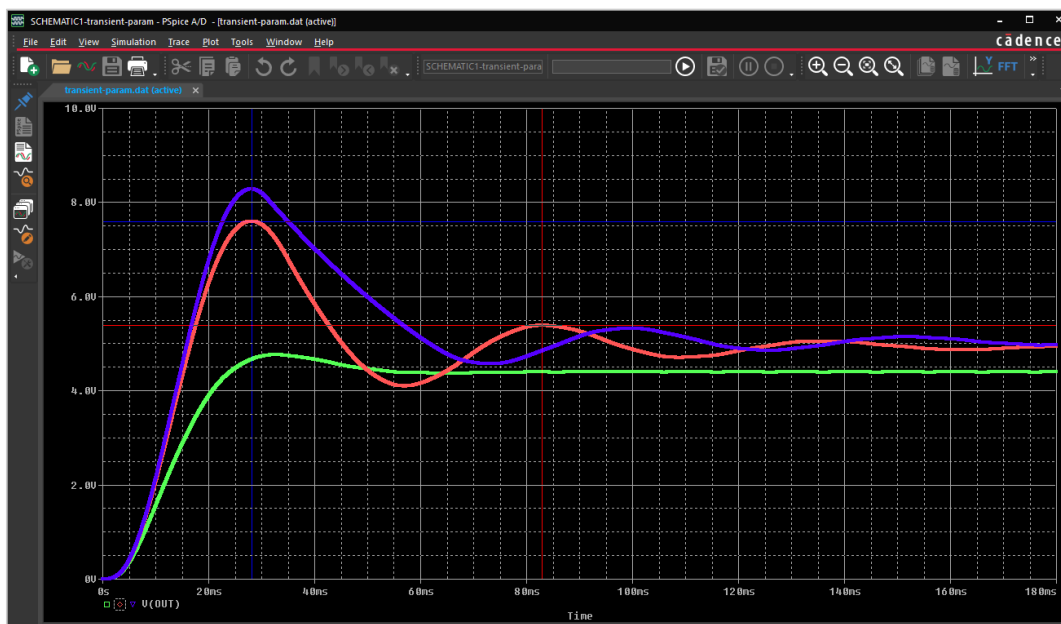
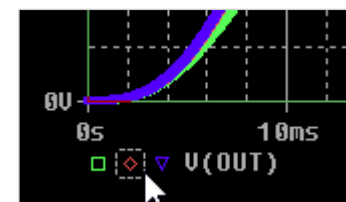
Use the icons  or the **Trace > Cursor** submenu to perform specific queries on individual curves.





Parametric Sweep Results (II)

- You can use 2 cursors at the same time. Placing them on different traces allows to make measurements on the same or between two traces and displaying various values on the x and y axis.
- Use the **cursor 1** with the LMB (Left Mouse Button) and **cursor 2** with the RMB to measure e.g. the time difference between two peaks of one of the traces.
- First select with LMB and RMB which trace you want to measure, e.g. the red one. You could select here also different traces with the LMB or RMB.



- You can change cursor styles with **Tools > Options > Cursor Settings**.


Max values: 7.5985 V and 5.386 V
 Time difference: 54.8 ns

Probe Cursor									
	Trace Color	Trace Name	Y1	Y2	Y1 - Y2	Y1(Cursor1) - Y2(Cursor2)			
		X Values	82.938m	28.129m	54.809m			-2.2125	
		V(OUT)	4.3980	4.6867	-268.673m	Y1 - Y1(Cursor1)	Y2 - Y2(Cursor2)	Max Y	Min Y
		CURSOR 1,2	5.3860	7.5985	-2.2125	0.000	0.000	7.5985	5.3860
		V(OUT)	4.8463	8.2824	-3.4361	-539.680m	683.893m	8.2824	4.8463



Parametric Sweep Results (Measurements)

You can also take measurements on plotted traces in order to evaluate characteristics of a trace:

- Click on **View > Measurement Results** or click . This will dock this window below the simulation results.

Measurement Results				
Evaluate	Measurement	1	2	3
<input type="checkbox"/>				
Click here to evaluate a new measurement...				

- Click on **Click here to evaluate a new measurement...** and add, e.g., `max(v(out))` as trace expression.

You can also build the Trace by clicking Max(1) in the left part, and then select V(out) in the right part of the window.

- Click **OK**. You will get the measurement result for each trace:

Measurement Results				
Evaluate	Measurement	1	2	3
<input checked="" type="checkbox"/>	Max(V(OUT))	4.76245	7.59850	8.28252
Click here to evaluate a new measurement...				

Evaluate Measurement

Functions or Macros

Measurements

- Bandwidth(1,db_level)
- Bandwidth_Bandpass_3dB(1)
- Bandwidth_Bandpass_3dB_XRange(1,begin_x,end_x)
- CenterFrequency(1,db_level)
- CenterFrequency_XRange(1,db_level,begin_x,end_x)
- ConversionGain(1,2)
- ConversionGain_XRange(1,2,begin_x,end_x)
- Cutoff_Highpass_3dB(1)
- Cutoff_Highpass_3dB_XRange(1,begin_x,end_x)
- Cutoff_Lowpass_3dB(1)
- Cutoff_Lowpass_3dB_XRange(1,begin_x,end_x)
- DutyCycle(1)
- DutyCycle_XRange(1,begin_x,end_x)
- Falltime_NoOvershoot(1)
- Falltime_StepResponse(1)
- Falltime_StepResponse_XRange(1,begin_x,end_x)
- GainMargin(1,2)
- MATLABFunction1(1,<MATLAB_function>)
- MATLABFunction2(1,2,<MATLAB_function>)
- Max(1)
- Max_XRange(1,begin_x,end_x)
- Min(1)

Analog

Digital

Voltages

Currents

Power

Noise (V²/Hz)

Alias Names

Subcircuit Nodes

191 variables listed

Simulation Output Variables

- V(N00502)
- V(N01200)
- V(N01213)
- V(N01242)
- V(N01255)
- V(N01272)
- V(N01289)
- V(N01296)
- V(N01300)
- V(N01374)
- V(OUT)
- V(R1:1)
- V(R1:2)
- V(R2:1)
- V(R2:2)
- V(R3:1)
- V(R3:2)
- V(R4:1)
- V(R4:2)
- V(R5:1)
- V(R5:2)
- V(R6:1)

Full List

Trace Expression:



Conclusion

- After performing the examples in this quickstart guide, you are now able to perform first simulations with PSpice.
- This guide is only intended to demonstrate the most important steps necessary to run simulations with PSpice.
- The quickstart guide is not meant to replace any PSpice training as delivered by FlowCAD or Cadence. It does not replace carefully studying the extensive documentation shipped with the software.
- If PSpice models can not be found in the standard library, you can always ask the manufacturer directly. PSpice allows to add models in the Simulation Settings of the simulation profile in the Configuration Files tab. In addition, PSpice offers a powerful Model Editor which allows to create own models just based on data sheets.
- PSpice allows to simulate also mechanical or thermal elements, e.g. a shock absorber of an vehicle or the temperature distribution on a PCB. You only need the electrical replacement models for these types of simulation. Those are not part of the standard libraries shipped by Cadence.
- PSpice can handle digital and analogue signals coexisting in the same circuit by running a digital (event driven) and analogue simulation at the same time in one simulation.
- Using the Smoke analysis, PSpice can check if your components are used below their maximum operating conditions. PSpice identifies components which are stressed and improves the reliability of your circuit.



PSpice Extensions

- Using the PSpice Advanced Analysis Option will allow you to extend the basic analogue / digital simulation features into a tool which helps you to improve performance, productivity, predictability, cost and the quality of your circuits, by using features like Sensitivity Analysis, Parametric Plots, extended Monte Carlo Analysis and the Optimizer.
- PSpice provides a system-level simulation solution, thanks to the possibility to abstract the functionality of models like microcontroller, state-machines, etc. using C/C++, SystemC and Verilog-A programming languages. This would allow 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 and MATLAB / Simulink. For this, Cadence offers an interface (SLPS) (Simulink-PSpice), which enables a co-simulation with MATLAB. You can also send simulation data to MATLAB for post-processing and use MATLAB functions in the PSpice environment.



Appendix



System Requirements

Operating System	Windows 11 Professional and Enterprise; Windows 10 (64-bit) Professional and Enterprise, including Dark Theme mode Windows Server 2016 (All Service Packs) Windows Server 2019
Recommended Hardware	Intel Core™ i7 4.30 GHz or AMD Ryzen™ 7 4.30 GHz with at least 4 cores 16 GB RAM 50 GB free disk space (SSD recommended) 1920 x 1200 display resolution with true color (at least 32-bit color) A dedicated graphics card supporting OpenGL, min 2 GB (with additional support for DX11 for 3D canvas) Broadband Internet connection for some services
In General	Above mentioned hardware specifications are to be understood as a minimum requirement. Depending on the task at hand, hardware must be adapted accordingly “upwards”.



Installation

- To run this tutorial successfully, it is expected, that the software is installed correctly on your machine and you have access to your PSpice license.
- Or you have installed and activated the OrCAD Trial Version 22.1.
- An installation guide of the software can be found at <https://www.flowcad.com/en/download.htm>.
- This tutorial is based on the SPB 22.1 release of the software.
 - It will run with earlier versions of the software, but you may observe differences in some screenshots.



Contact us / Kontakt zu FlowCAD

Please do not hesitate to contact us.

Für weitere Fragen und Informationen stehen wir gerne zur Verfügung.

FlowCAD Deutschland

Mozartstr. 2
85622 Feldkirchen bei München
T +49 89 45637-770
info@FlowCAD.de



FlowCAD Schweiz

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



FlowCAD Polska

ul. Sąsiedzka 2A
80-298 Gdańsk
T +48 58 727 90 90
info@FlowCAD.pl





Follow Us



» [FlowCAD.com](https://www.flowcad.com)



» [FlowCAD.com/
newsletter](https://www.flowcad.com/newsletter)



» [youtube.com/
FlowCAD](https://www.youtube.com/FlowCAD)



» [linkedin.com/
FlowCAD](https://www.linkedin.com/FlowCAD)



» [twitter.com/
FlowCAD](https://twitter.com/FlowCAD)



» [instagram.com/
FlowCAD](https://www.instagram.com/FlowCAD)



» [facebook.com/
FlowCAD](https://www.facebook.com/FlowCAD)



» [xing.com/
FlowCAD](https://www.xing.com/FlowCAD)

Don't forget to subscribe, share and like!



FlowCAD