cadence®

Simulating an SMPS Design using Capture-PSpice Flow

Product Version 17.4-2019
October 2019

Document Last Updated: June 2020

Cadence Design Systems, Inc. (Cadence), 2655 Seely Ave., San Jose, CA 95134, USA.

Contents

Preface

This tutorial provides an overview of OrCAD® Capture - PSpice® flow using a Switched-Mode Power Supply (SMPS) design. In this tutorial, you will configure the design for simulation, simulate the design using PSpice, and then use the Advanced Analysis option to verify stability and yield of the design.

Audience

This tutorial is designed for:

- PCB designers using OrCAD products to design and simulate a circuit design
- First-time users of the Capture - PSpice flow

Prerequisites

To perform the tutorial tasks, you need to have following Cadence® products installed:

- Capture or Capture CIS
- PSpice AD
- PSpice Advanced Analysis

Design and Library Files

Extract the `project.zip` file located at `<Cadence Installation>/doc/pspcaptut/examples/`. The `project.zip` archive contains library files and design files required to run the tasks in this tutorial.

PSpice Tutorial Libraries

- `CIS_PARTLIB`
- `PWMCON`
- `TECCI_CORE`

Enabling the SMPS design for Simulation

Before simulating a design, you need to enable the design for simulation.

Objective

Open the SMPS design in OrCAD Capture

Opening the SMPS design in Capture

To open the design, `demo_smps_1.dsn` in Capture CIS, do the following:

1. Choose *Cadence PCB 17.4-2019 – Capture CIS 17.4* from the Start menu.

   If prompted, from the 17.4 CaptureCIS Product Choices dialog box, choose *OrCAD PSpice Designer Plus* and click *OK*.

   The OrCAD Capture CIS window opens.

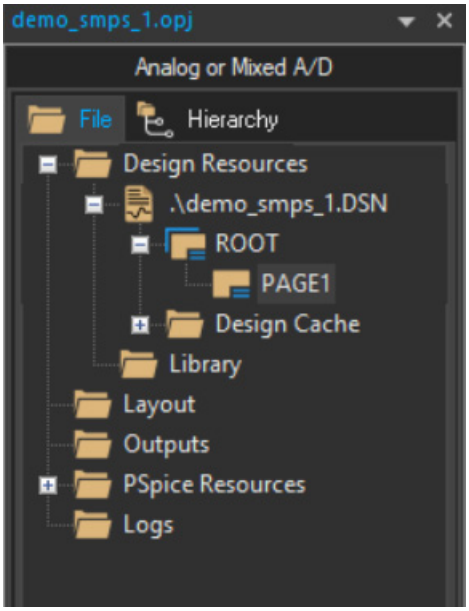
2. Choose *File – Open – Design*, browse to `DEMO_SMPS_1.dsn` from the project files of this tutorial, and click *Open*.

   The project manager window opens.

Observe the project manager window.

The project type is specified as A*nalog or Mixed A/D* below the title bar of the Project Manager window. You can simulate analog or mixed signal circuits in PSpice.

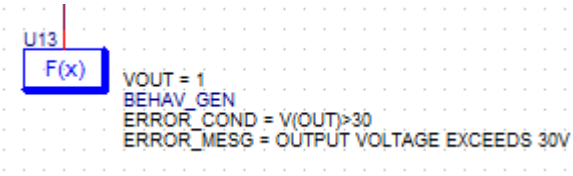
3. Under *Design Resources*, expand `demo_smps_1.DSN` and `ROOT`. Double-click *PAGE1* to open the schematic page.



The design contains a hierarchical block *pwmcon* with a pulse-width modulator circuitry controlled by feedback from `18V` output. A ferrimeter transformer, designed using Magnetic Parts Editor, is used in the design. A high voltage is switched through the primary winding of the transformer. The secondary winding of the transformer is connected to a rectifier and a filter.

Observe the functional component *BEHAV_GEN*. Error condition is specified as `V(OUT)>30` to ensure that the simulation of the design stops if *V(OUT)* exceeds `30V`.

Note: You can specify warning conditions by specifying a value for the *WARN_COND* attribute. In case of warning conditions simulation continues after displaying the specified warning message.



4. Select *Place – Part*, press *P*, or click the Place part icon.

   The *Place Part* pane opens.

5. Click the *Add Library* icon.

   The Browse File dialog box opens.

6. Browse to `<installation_directory>\tools\capture\library\pspice\advanls\pspice_elem.olb`.

7. Select `pspice_elem.olb` and click *Open*, or double-click `pspice_elem.olb`.

   The `PSPICE_ELEM` library appears in the *Libraries* list box.

8. Search for `Variables` from the *Part* list box.

9. Click the *Place Part* icon or press *Enter*.

   The part symbol is attached to the pointer.

10. Click the schematic page where you want to place this component.

11. Right-click and select *End Mode* or press *Esc*.

What's Next

Next, you will create a simulation profile and run a PSpice simulation on the SMPS design.

Recommended Reading

For more information on opening a project, adding properties in a schematic, and adding variables block in a schematic, see chapters *Working with Projects* and *Working with Properties* in *OrCAD Capture User Guide*.

Setting Up and Running a PSpice Simulation

In this chapter, you will create a simulation profile for the SMPS design to run a transient analysis in PSpice.

Objectives

- Create a PSpice Simulation Profile using OrCAD Capture
- Simulate a design using PSpice

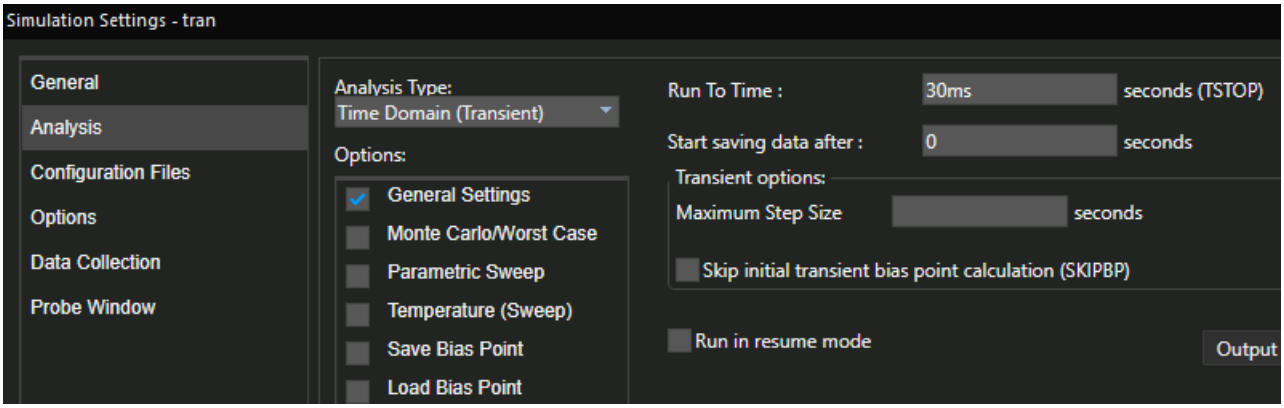
Creating a PSpice Simulation Profile

To create a new Simulation Profile in OrCAD Capture, perform the following steps:

1. Choose *PSpice – New Simulation Profile*.
2. Enter the Name as `trans`.
3. Ensure *Inherit From* is *none*.
4. Click *Create*.
5. Select *Analysis* in the Simulation Settings dialog box.

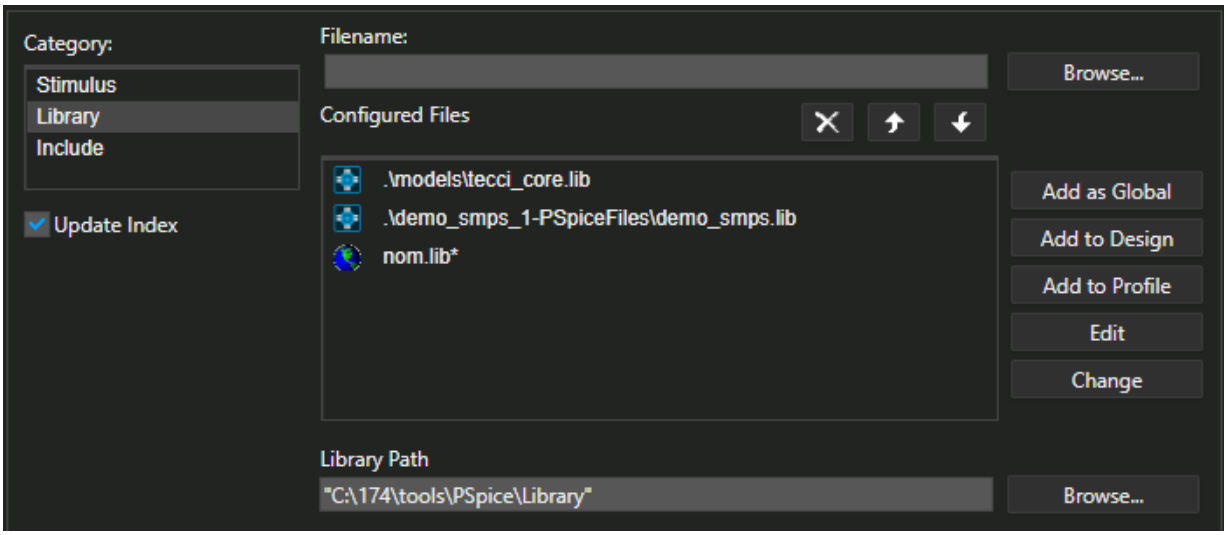
6. To specify a transient analysis to run for `30ms` starting from the `0s`, do the following:

- ❑ Choose the *Analysis Type* as *Time Domain (Transient)*.
- ❑ In *Run To Time*, enter `30ms`.
- ❑ In *Start saving data after*, enter `0`.
- ❑ Ensure that *Skip the initial transient bias point calculation (SKIPBP)* is not selected.

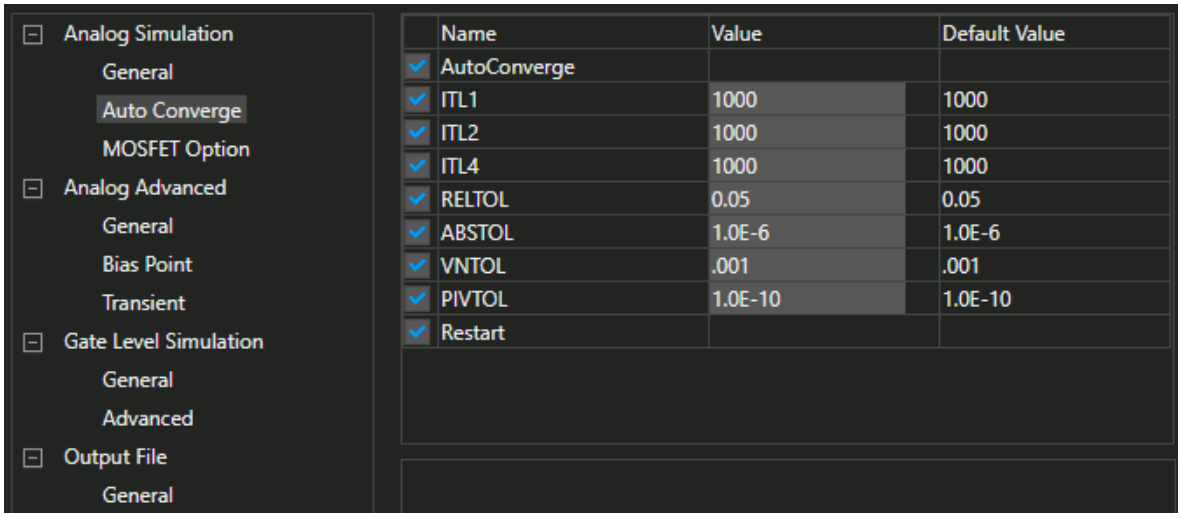


7. Select *Configuration Files*.
8. Ensure that `tecci_core.lib` and `demo_smps.lib` are listed under *Configured Files* for the *Library* Category.

   If required, browse to the library files in the `models` folder of the project directory and add them using *Add to Design*.



9. Select *Options*.
10. Under *Analog Simulation*, select *Auto Converge* and then set *AutoConverge*.



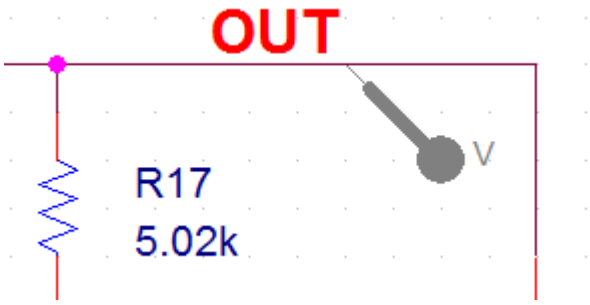
When you select `AutoConverge`, PSpice uses relaxed limits for some of the options, such as `ITL1` and `RELTOL`, to adjust and run the simulation to achieve convergence.

11. Click *Apply* to save changes.
12. Click *OK*.

Simulating the Design using PSpice

Perform the following steps in OrCAD Capture to perform simulation:

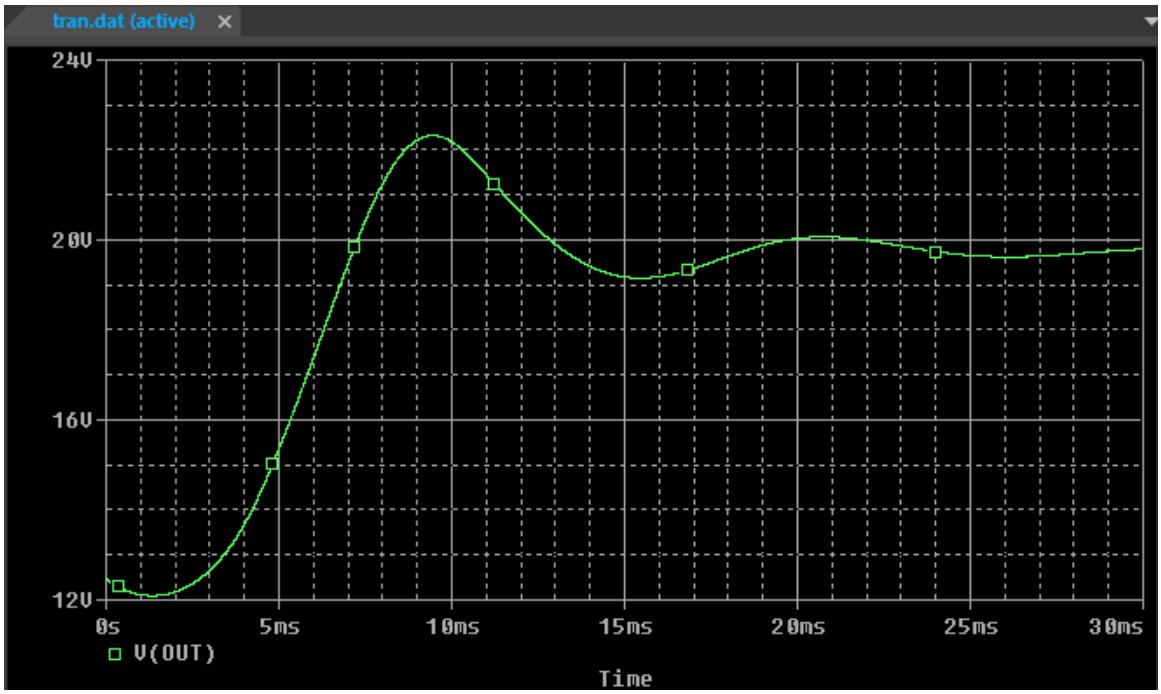
1. Place a voltage probe on the OUT net: choose *PSpice – Markers – Voltage Level* and click on the `OUT` net.



2. Choose *PSpice – Run* or click to run the simulation.

   If required, click *Yes* in the Undo Warning dialog box. Close the Simulation Message Summary dialog box.

   The simulation result is displayed in the PSpice probe window.



What's Next

Next, you will verify the stress levels of components in SMPS using smoke analysis of Capture - PSpice Advanced Analysis flow and then correct the stress levels for the components based on the analysis result.

Recommended Reading

For more information on creating a simulation profile, running a PSpice simulation on any design, and understanding convergence options in PSpice, see *PSpice User Guide*.

Verifying Stress Levels of Components in SMPS

Run smoke analysis to identify components stressed due to power dissipation, increase in junction temperature, secondary breakdowns, or violations of voltage / current limits. In this chapter, you will perform Smoke analysis based on the transient profile to identify and correct components that are stressed.

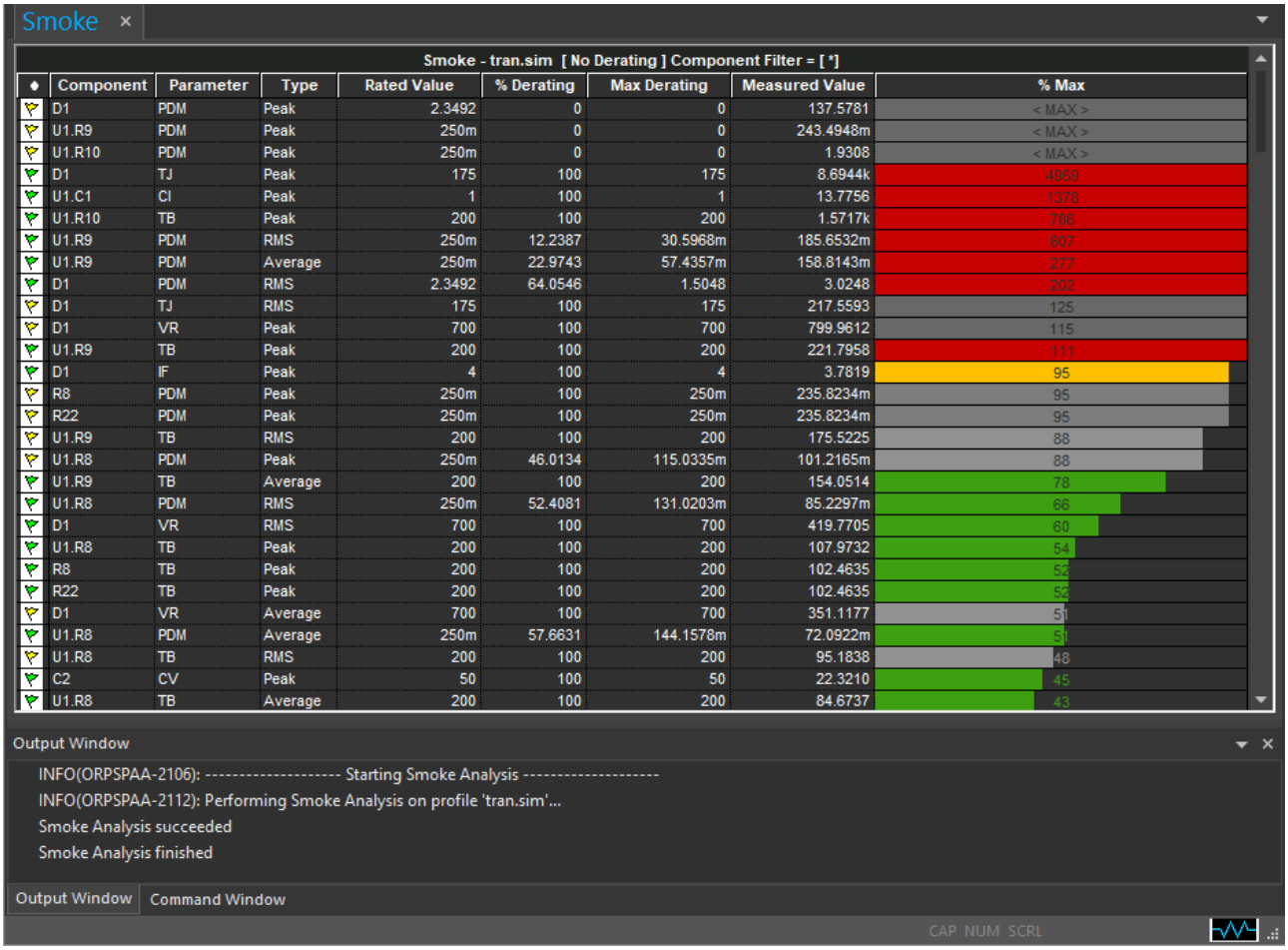
Objectives

- Identify components under stress by running Smoke Analysis
- Correct stress levels using PSpice Advanced Analysis results

Identifying Stressed Components: Smoke Analysis using PSpice Advanced Analysis

1. In Capture, choose *PSpice – Advanced Analysis – Smoke*.

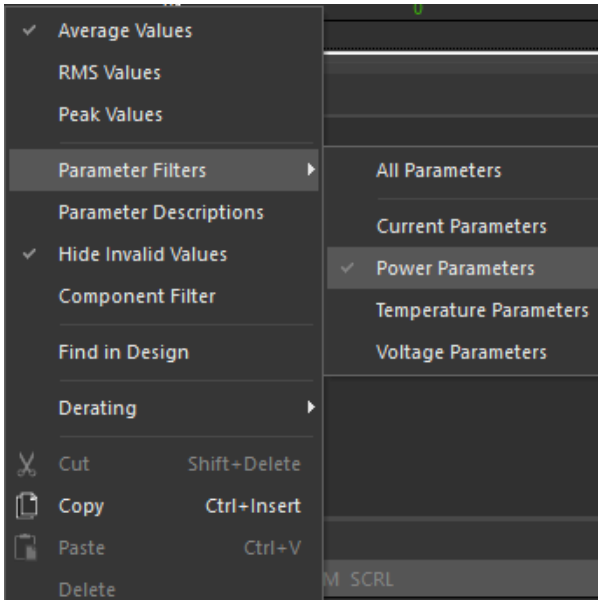
The PSpice Advanced Analysis window opens with *Smoke* tab displayed.



You can right-click in the result pane of the Smoke Analysis window and choose options to see only specific measurements, such as RMS, average, or peak values. You can also choose to view specific parameters.

2. Right-click to open the context menu.
3. Ensure that only *Average Values* is selected.
4. From the context menu choose *Parameter Filters*.
5. Ensure that only *Power Parameters* is selected.

6. Choose *Hide Invalid Values*. This ensures that invalid values are not displayed.



Observe the changes in the result pane of the Smoke Analysis window.

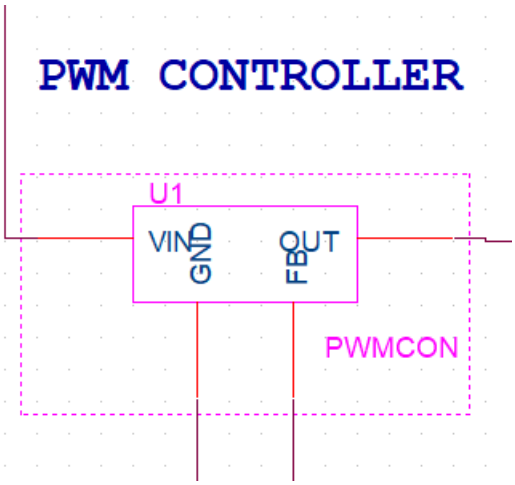
Smoke - tran.sim [No Derating] Component Filter = [*]

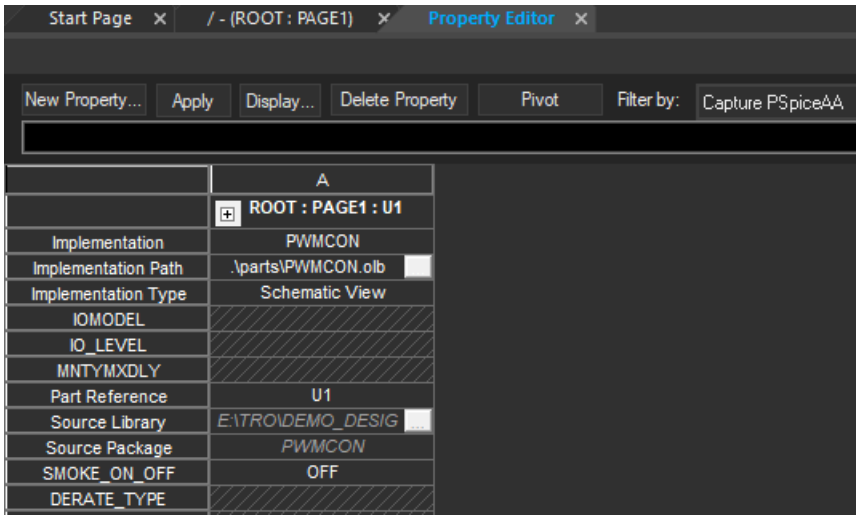
Component	Parameter	Type	Rated Value	% Derating	Max Derating	Measured Value	% Max
U1.R9	PDM	Average	250m	22.9743	57.4357m	158.8143m	277
U1.R8	PDM	Average	250m	57.6631	144.1578m	72.0922m	51
R17	PDM	Average	250m	100	250m	55.1317m	23
R8	PDM	Average	250m	100	250m	50.3793m	21
R22	PDM	Average	250m	100	250m	50.3793m	21
D1	PDM	Average	2.3492	100	2.3492	170.9602m	8
R18	PDM	Average	250m	100	250m	7.5774m	4
U1.R11	PDM	Average	250m	84.7594	211.8985m	4.3515m	3
U1.R4	PDM	Average	250m	86.4985	216.2462m	3.7949u	1
U1.R10	PDM	Average	250m	86.4304	216.0759m	174.0814u	1
U1.R5	PDM	Average	250m	86.4990	216.2475m	2.4785u	1
U1.R6	PDM	Average	250m	86.4900	216.2250m	24.9999u	1
U1.R7	PDM	Average	250m	86.5000	216.2500m	10.5284n	1
U1.R3	PDM	Average	250m	86.4985	216.2462m	3.7949u	1
R16	PDM	Average	250m	100	250m	2.2728m	1
R20	PDM	Average	250m	100	250m	1000.0000u	1
U1.R1	PDM	Average	250m	86.5000	216.2500m	0	0
U1.R2	PDM	Average	250m	86.5000	216.2500m	0	0

In the current schematic, the component `U1.R9` is under stress, as shown by the red color row.

Correcting Stress Levels using PSpice Advanced Analysis

1. In the Capture schematic, double-click PWM Controller (`PWMCON`) and change the value of *SMOKE_ON_OFF* and to `OFF`.





2. Save the schematic.
3. If you change the value of a component used in schematic, re-run the PSpice simulation before you run the smoke analysis.
4. Run smoke analysis again (*PSpice – Advanced Analysis – Smoke*).

The smoke analysis results show that stress has been removed after disabling the *SMOKE_ON_OFF* property. The *SMOKE_ON_OFF* property on `PWMCON` is changed to `OFF` to discard smoke analysis on the hierarchical block.

What's Next

Next, you will create measurement expressions for the SMPS design.

Recommended Reading

For more information on Smoke Analysis, see the chapter on Smoke in *PSpice Advanced Analysis User Guide*.

Evaluating Stability and Optimization: Creating Measurements

In this chapter, you will create measurements to evaluate the stability and optimization of a design using Capture - PSpice Advanced Analysis flow.

Objectives

- Create measurements

Creating Measurements

Create the following measurements to evaluate the characteristics of the waveform generated using PSpice:

- `Max_XRange(V(OUT),25m,30m)`
- `Min_XRange(V(OUT),25m,30m)`

To create the above measurements:

1. Open the PSpice Probe window.

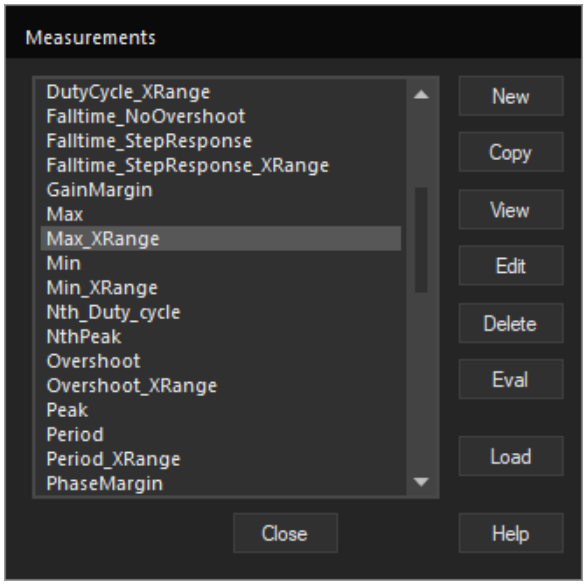
   The Probe window is displayed whenever you run a simulation.

2. In the Probe window, choose *Trace – Measurements*.

   The Measurements dialog box appears.

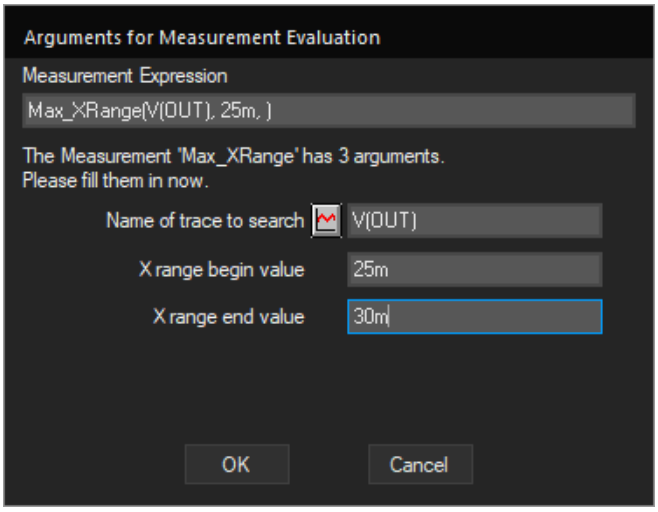
   Note: To know more about different measurement expressions, see *Measurement Expressions* chapter in *PSpice User Guide*.

3. Choose *Max_XRange* in the Measurements dialog box.



4. Click *Eval*.

5. Specify the trace value as `V(OUT)` and the max and min values for the XRange as `25m` and `30m`, respectively.

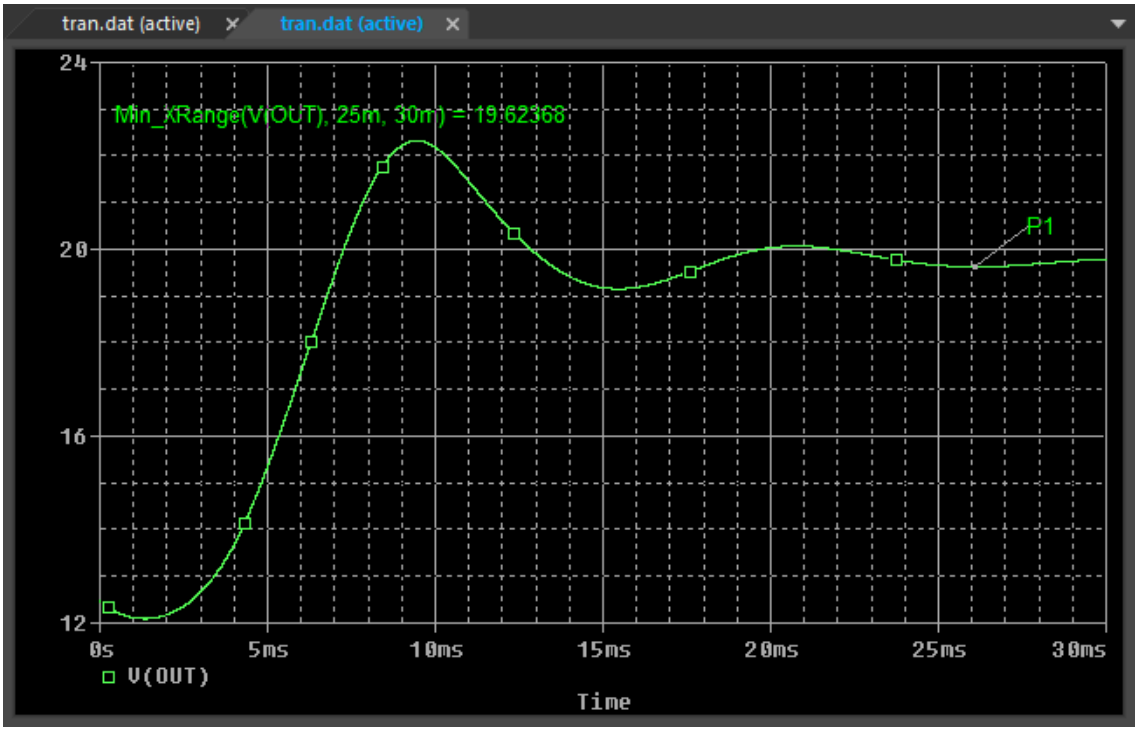


6. Click *OK*.

7. Click *OK* in the Display Measurement Evaluation message box after viewing the results.

8. Click *Close* in the Measurements dialog box.

9. Similarly add *Min_XRange*.



What's Next

Next, you will verifying design stability and yield and then optimize the design using various advanced analyses.

Recommended Reading

For more information on creating measurement expression and setting tolerances, see *PSpice User Guide*.

Verifying Design Stability and Yield

PSpice Advanced Analysis is a set of advanced tools that augment the classic PSpice functionality with capabilities that include Smoke, Sensitivity, Monte-Carlo, Optimizer, and Parametric Plotter.

In this chapter, use these advanced analysis tools to verify the stability of the SMPS design and optimize it.

Objectives

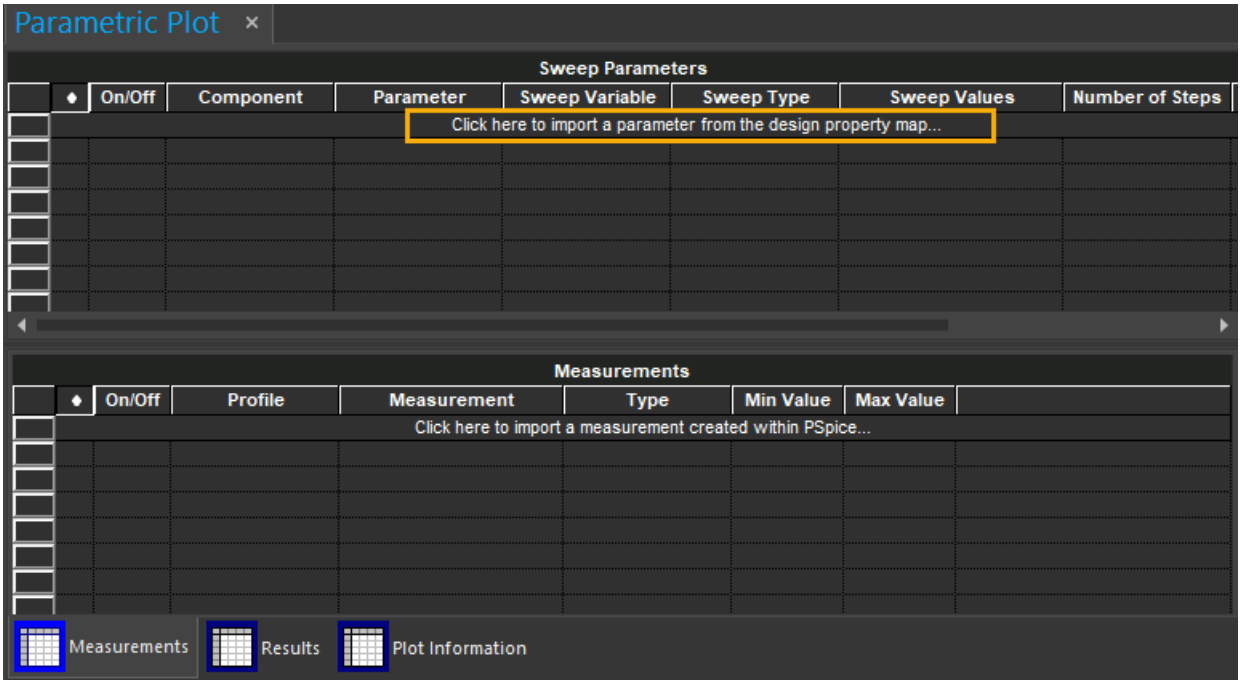
- Run Parametric Plotter
- Run Monte-Carlo
- Run Optimizer and Sensitivity Analysis

Running Parametric Plotter

In Parametric Plotter, you analyze sweep results from multiple parameters and you can sweep any number of design and model parameters (in any combinations) and view results in Plot/ Probe in tabular or plot form. You will run Parametric Plot analysis to ensure that the design is stable for a range of load and fluctuations in the line voltages.

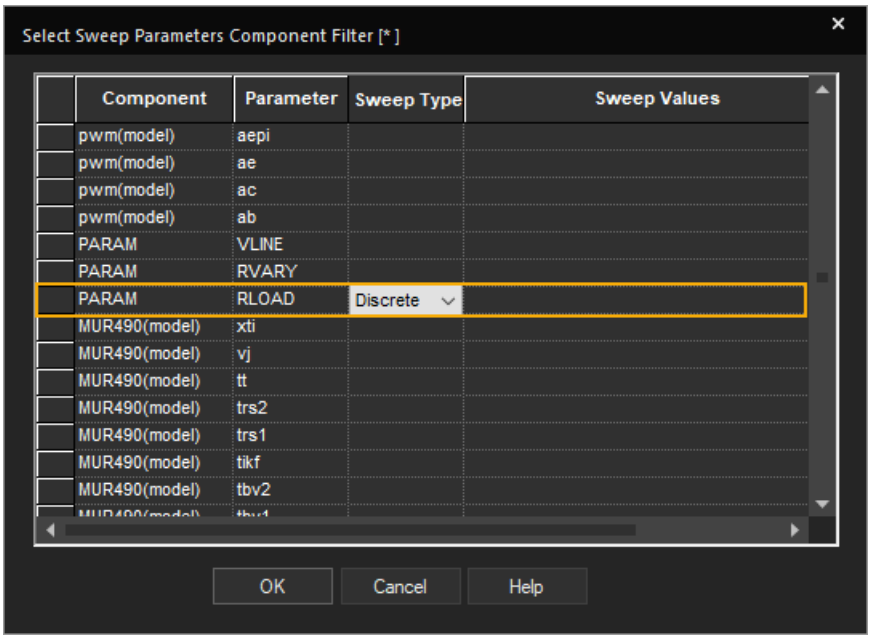
1. Choose *PSpice – Advanced Analysis – Parametric Plot* from OrCAD Capture to start Parametric Plotter.

2. In the *Sweep Parameters* section of the Parametric Plot window, click *Click here to import a parameter from the design property map*.



3. In the Select Sweep Parameters Component Filter window, scroll down to the Parameter `RLOAD` and click in the *Sweep Type* column for this parameter.

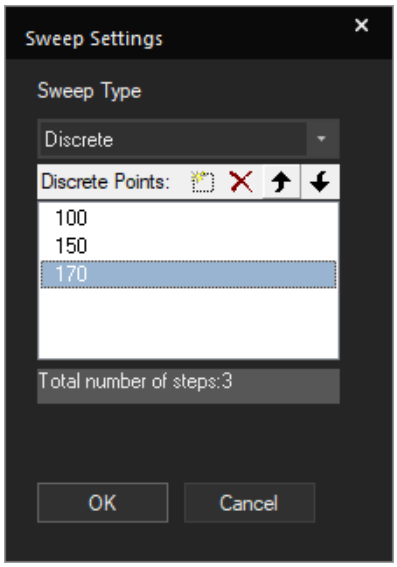
4. Choose `Discrete` from the list.



5. Click in the *Sweep Values* column to open the Sweep Settings dialog box.

6. Specify the values as `100`, `150`, and `170`.

   To specify a value, click *New* ( ) and type the value. Similarly, to delete a value, select it and click ( ).

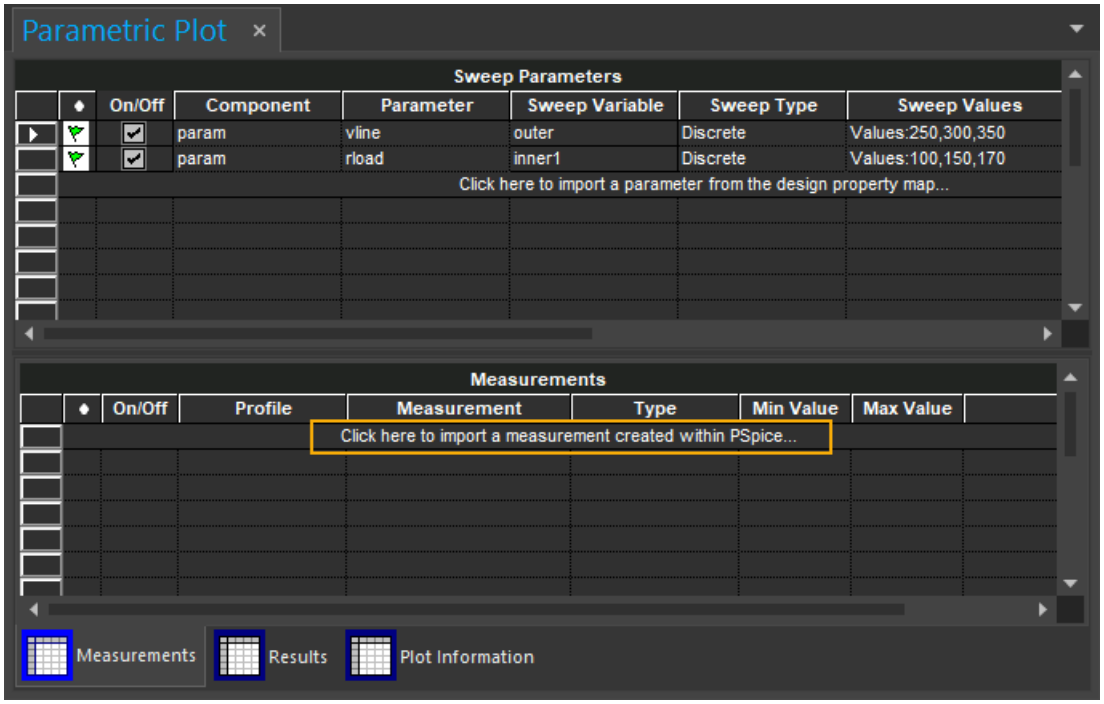


7. Click *OK* to close the Sweep Settings dialog box.
8. Similarly, enter `250`, `300`, and `350` as discrete values for *Parameter* `VLINE`.
9. Click *OK* in the Select Sweep Parameters Component Filter window.

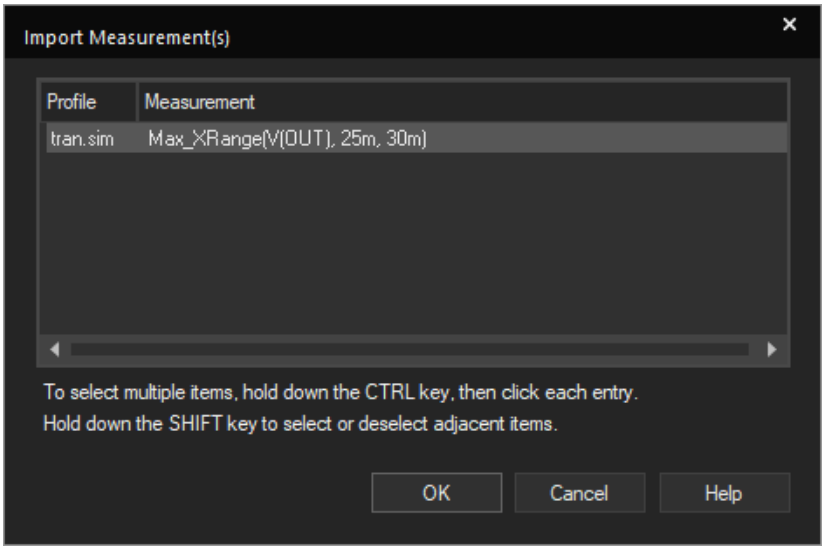
You will simulate the design for different values of `RLOAD` and `VLINE` to observe the impact of this variation on the output voltage, `Vout`.

To simulate and observe variations for different values, import the measurements created in PSpice using Import Measurement(s) window.

1. In the *Measurements* section of the Parametric Plot window, click *Click here to import a measurement created within PSpice*.



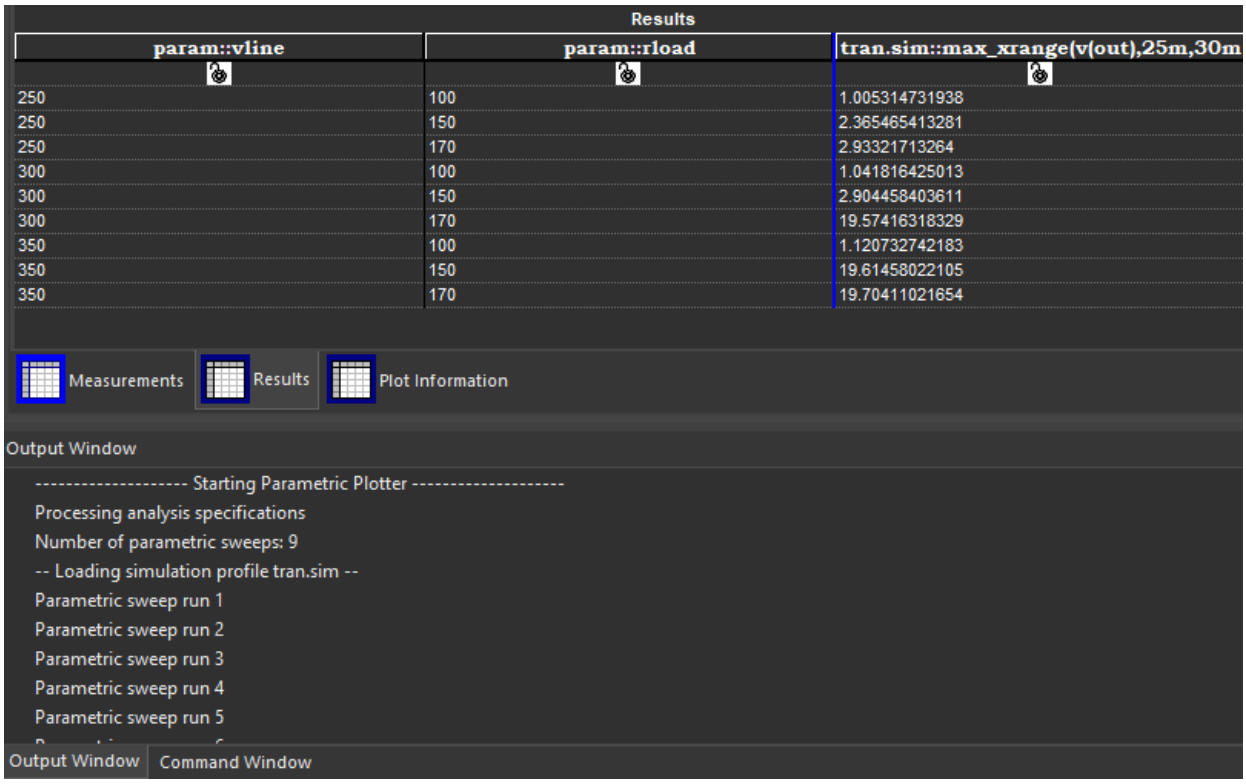
2. Select the measurement, `Max_XRange` and click *OK*.



3. To run the analysis, select *Run – Start Parametric Plot*.

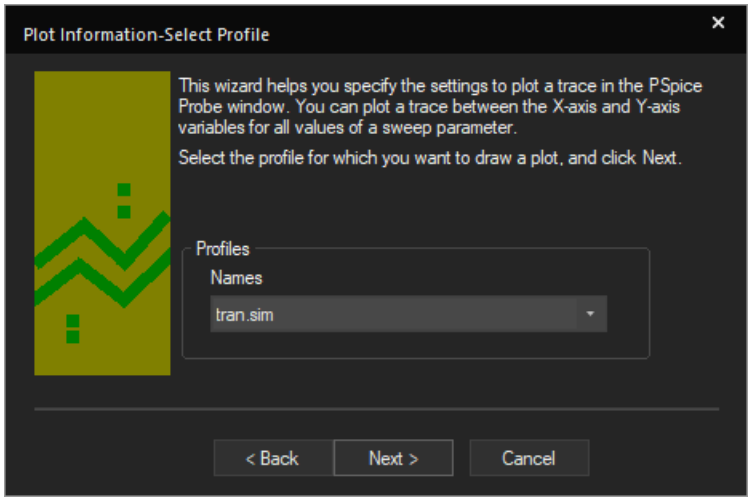
4. To view measurement results, click the *Results* tab.

   The *Results* tab that lists the values for the parameters and the measurement results for each value.



5. To view the graph of the results, click the *Plot Information* tab.
6. Click the label, *Click here to add plot*.

7. In the Plot Information- Select Profile page of the wizard, choose `tran.sim` and click *Next*.



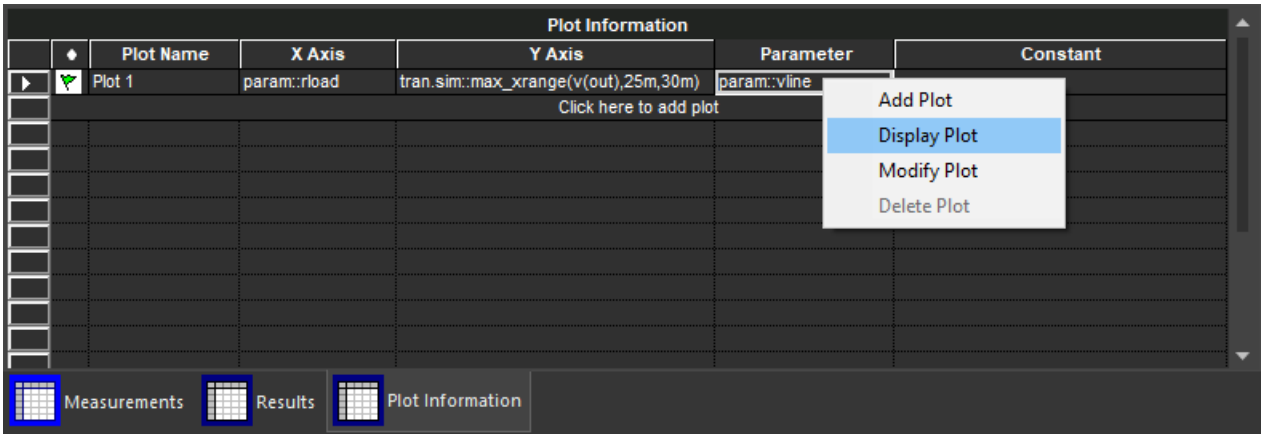
8. In the Plot Information-Select X-Axis Variable page, choose `param::rload` and click *Next*.

9. In the Plot Information-Select Y-Axis Variable page, choose `tran.sim::max_xrange(V(out),25m,30m)` and click *Next*.

10. In the Plot Information-Select Parameter page, choose `param::vline` and click *Finish*.

    The details you entered are added to the first row of the *Plot Information* tab.

11. Right-click this row and choose *Display Plot*.



From the result of the parametric plotter analysis observe that:

- For VLINE250

  VOUT changes abruptly from 1.2V to 19V when RLOAD changes from 100 ohms to 150 ohms, but when *RLOAD* changes from 150 ohms to 170 ohms the `Vout` remains constant.

- For VLINE300

  VOUT changes abruptly from 1.2V to 19V when RLOAD changes from 100 ohms to 150 ohms, but when RLOAD changes from 150 ohms to 170 ohms the `Vout` remains constant.

- For VLINE350

  VOUT remains fairly stable whenever RLOAD is changed.

Therefore, you can conclude that if VLINE is 350V, the design is fairly stable and the initial value of 350V for VLINE is OK for this SMPS design.

Calculating Yield by Running Monte-Carlo

Monte Carlo analysis calculates the circuit response to changes in part values by randomly varying all model parameters for which a tolerance is specified. This provides statistical data on the impact of a device parameter's variance. Monte Carlo analysis is frequently used to predict yields on production runs of a circuit.

There are two ways to run Monte-Carlo Analysis and calculate the yield:

- Using PSpice
- Using PSpice Advanced Analysis

Before you start the analysis, in the schematic design:

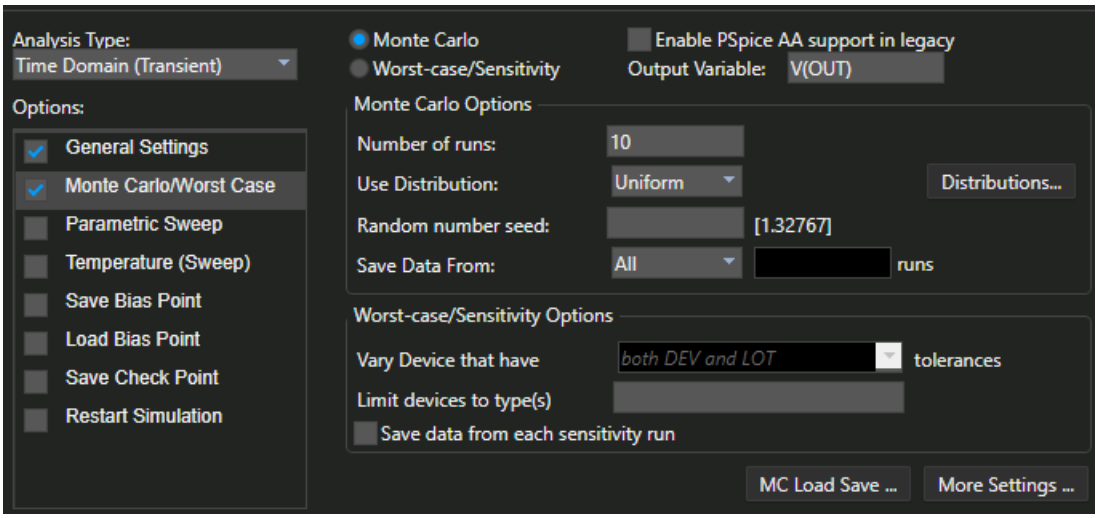
- Set the *TOL_ON_OFF* property to `OFF` on the `PWMCON` part to ignore tolerance for hierarchical components before you run Monte-Carlo Analysis.
- Set the value of `RTOL` to 10 in the `Advanced Analysis Properties` variable (added from the `pspice_elem.olb` library).

Running Monte-Carlo using PSpice

For the SMPS design, run the Monte Carlo Analysis in Time Domain to calculate the yield:

1. In Capture, choose *PSpice – Edit Simulation Profile*.

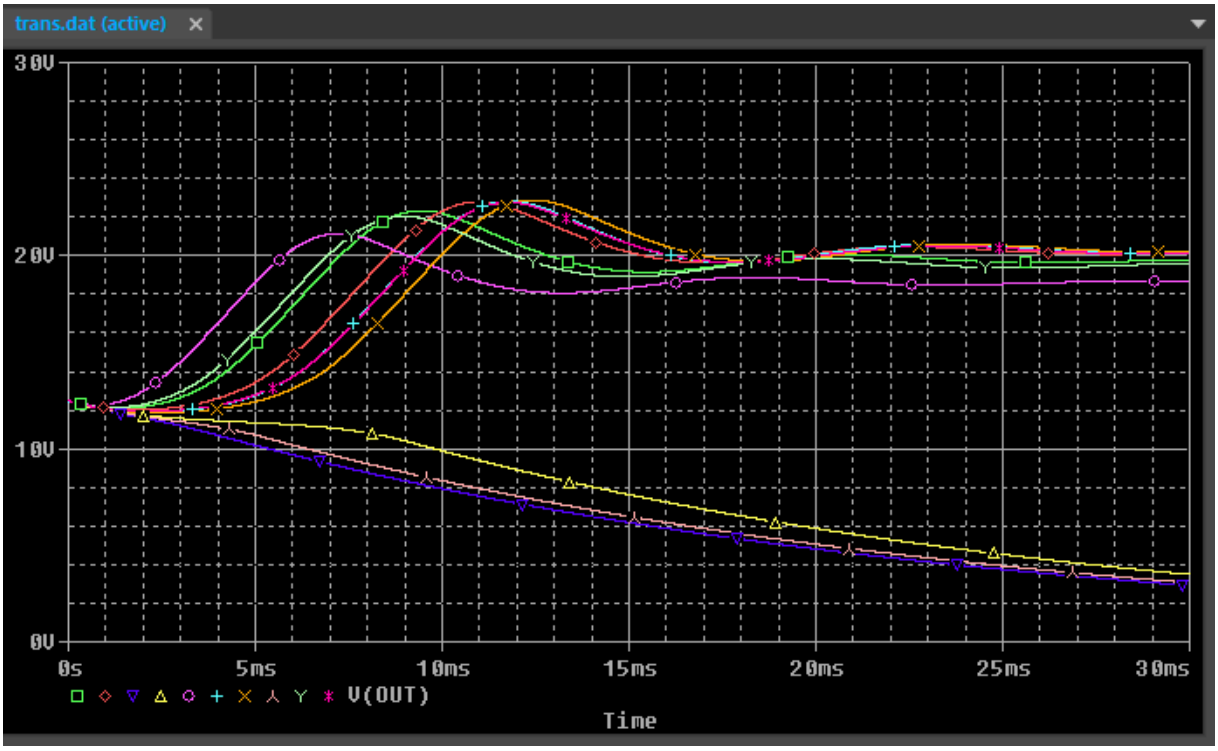
2. Under *Options*, select *Monte Carlo/Worst Case*.
3. Set *Output Variable* to `V(OUT)` and *Number of runs* to `8`.



4. Click *Apply* and then click *OK* to save the settings and close the Simulation Settings dialog box.

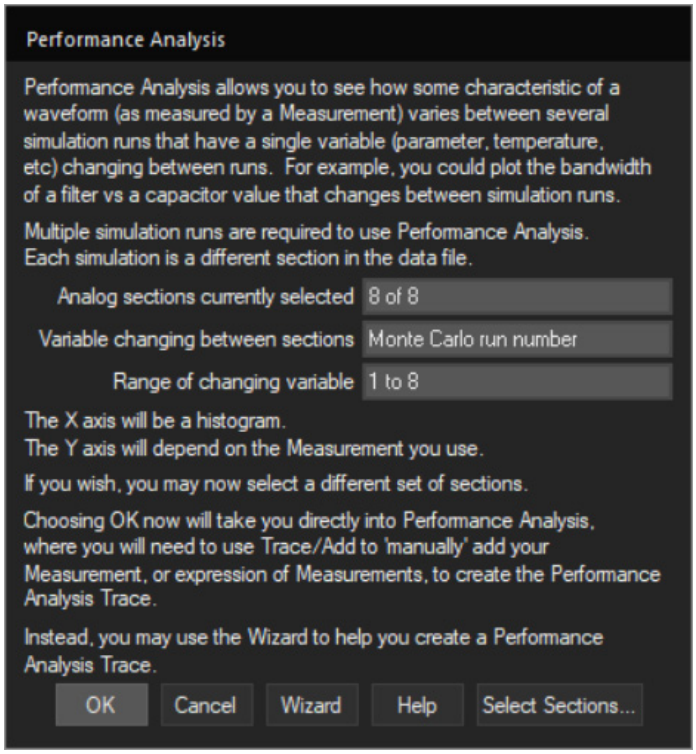
5. Run the PSpice simulation.

The PSpice probe window displays the simulation result.



6. In PSpice, choose *Trace – Performance Analysis* to compare the different waveforms generated using Monte Carlo Analysis.

The Performance Analysis dialog box appears.

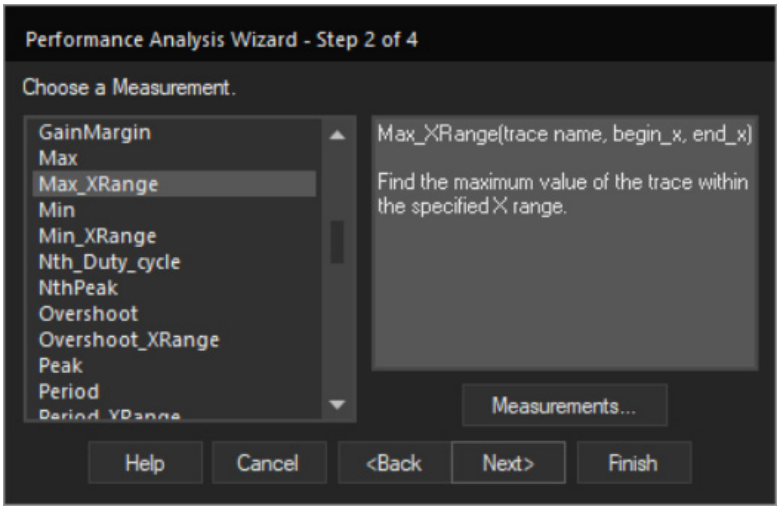


7. Click *Wizard* in the Performance Analysis dialog box.

   Using this wizard, you will create a plot to calculate the yield of the design.

8. Click the *Next*.

9. In the *Choose a Measurement* page, choose `Max_XRange` and then click *Next*.



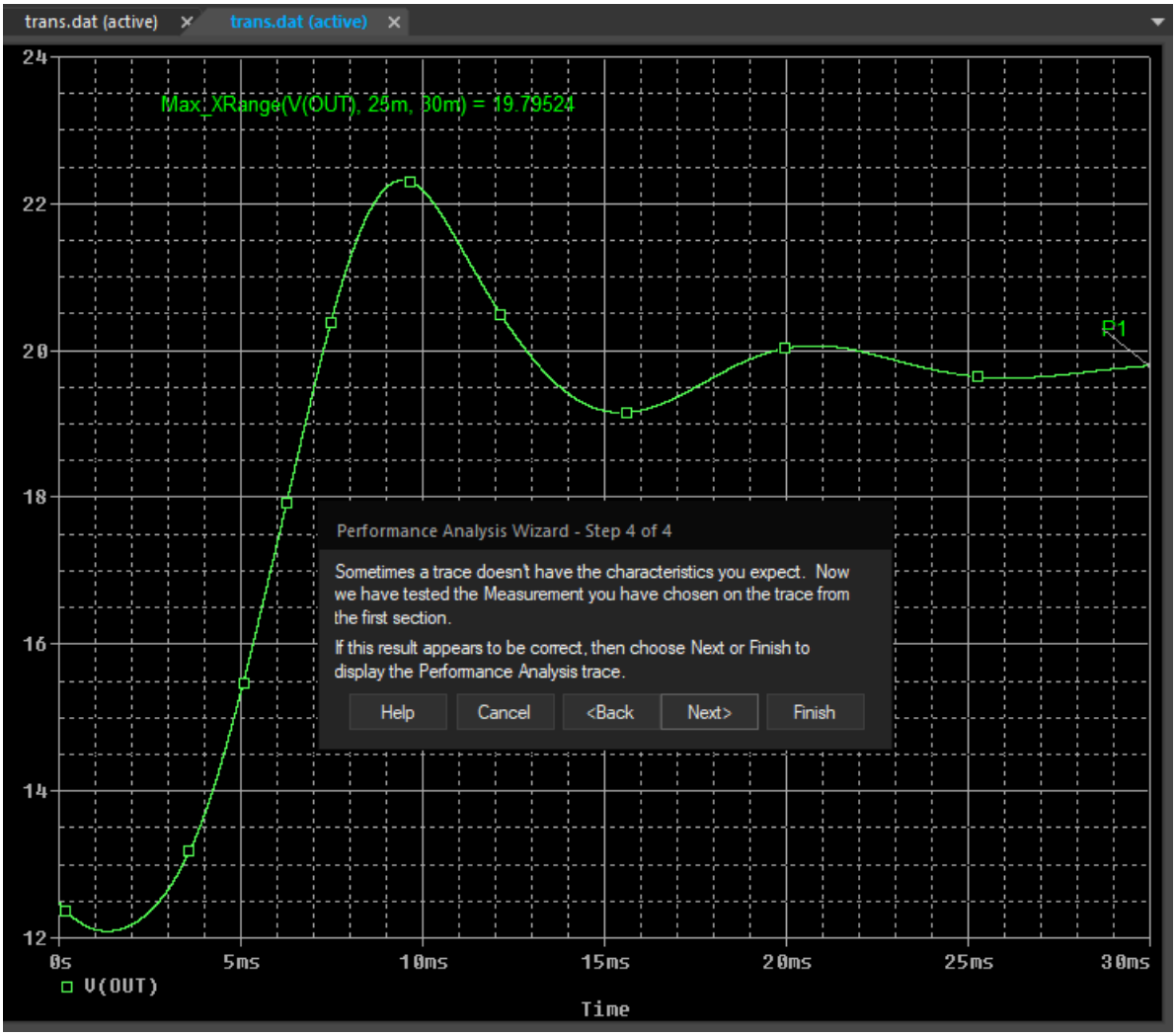
10. In *Name of trace to search* enter `V(OUT)`.

11. In *XRange begin value* enter `25m`.

12. In *XRange end value* enter `30m`.

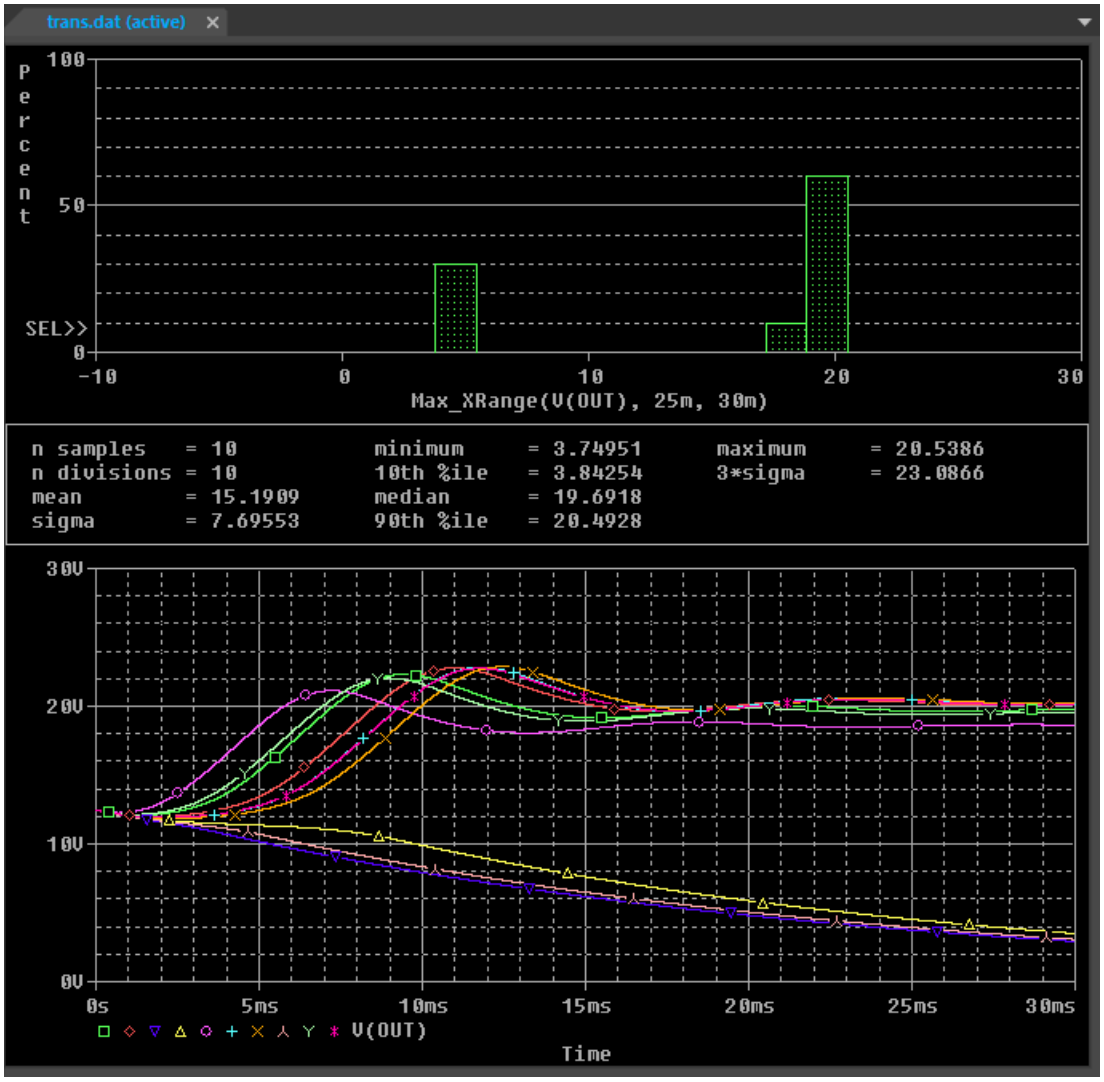
13. Click *Next*.

The wizard displays the `Max_XRange` trace for `V(OUT)` for the first run. This is done to test the measurement and you can verify if the result is correct.



14. Click *Finish*.

A plot of Max_XRange(V(OUT), 25m, 30m) vs V(OUT) occurrence percent appears.



Using this plot in the PSpice Probe, you can calculate the yield of the SMPS design.

Running Monte-Carlo using PSpice Advanced Analysis

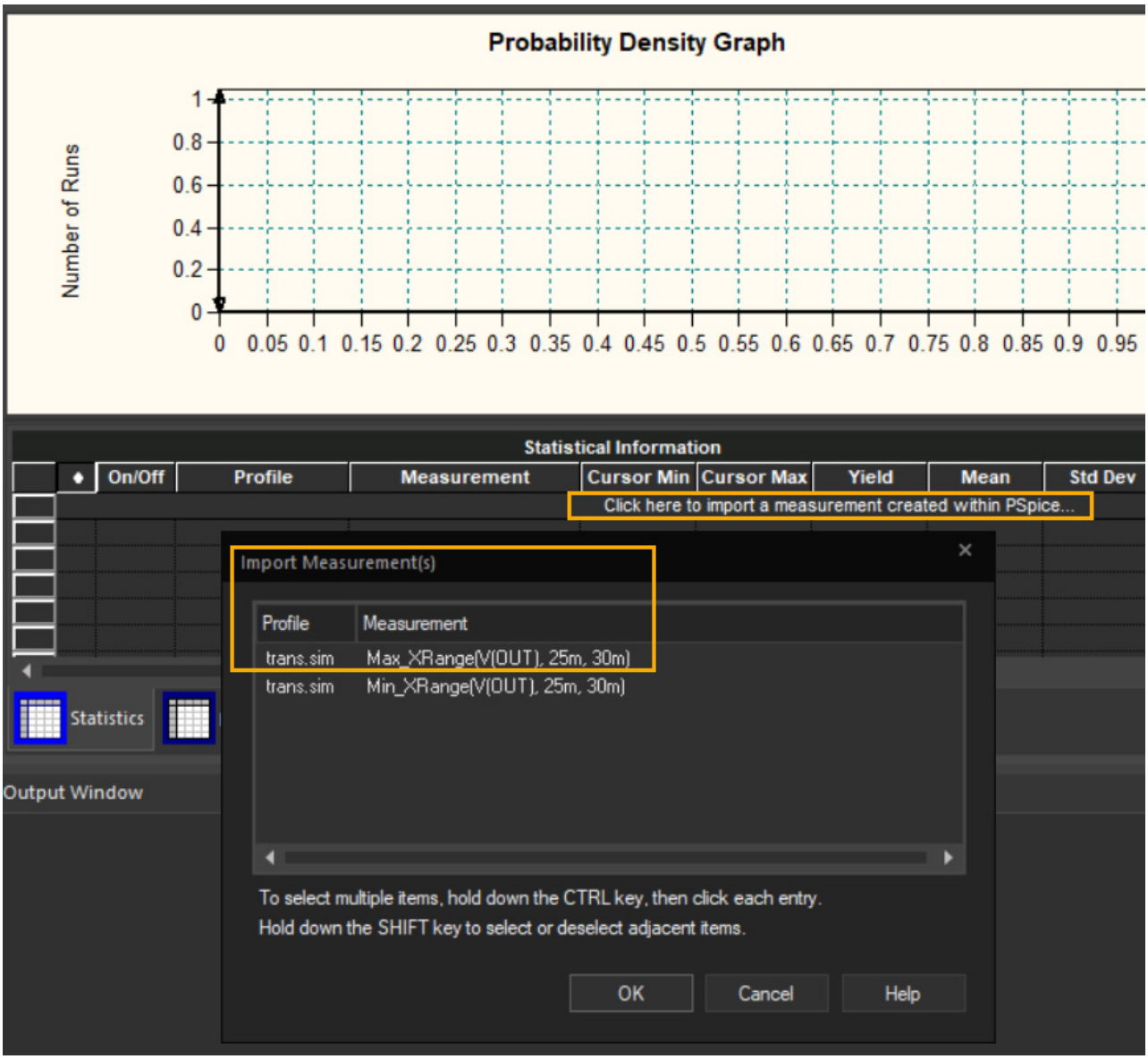
Before you start the advanced analysis, in the schematic design:

- Ensure that the *TOL_ON_OFF* property to `OFF` on the `PWMCON` part to ignore tolerance for hierarchical components before you run Monte-Carlo Analysis.
- Set the value of `LTOL` to 10 in the `Advanced Analysis Properties` variable (added from the `pspice_elem.olb` library).

To run Monte-Carlo using PSpice Advanced Analysis, do the following:

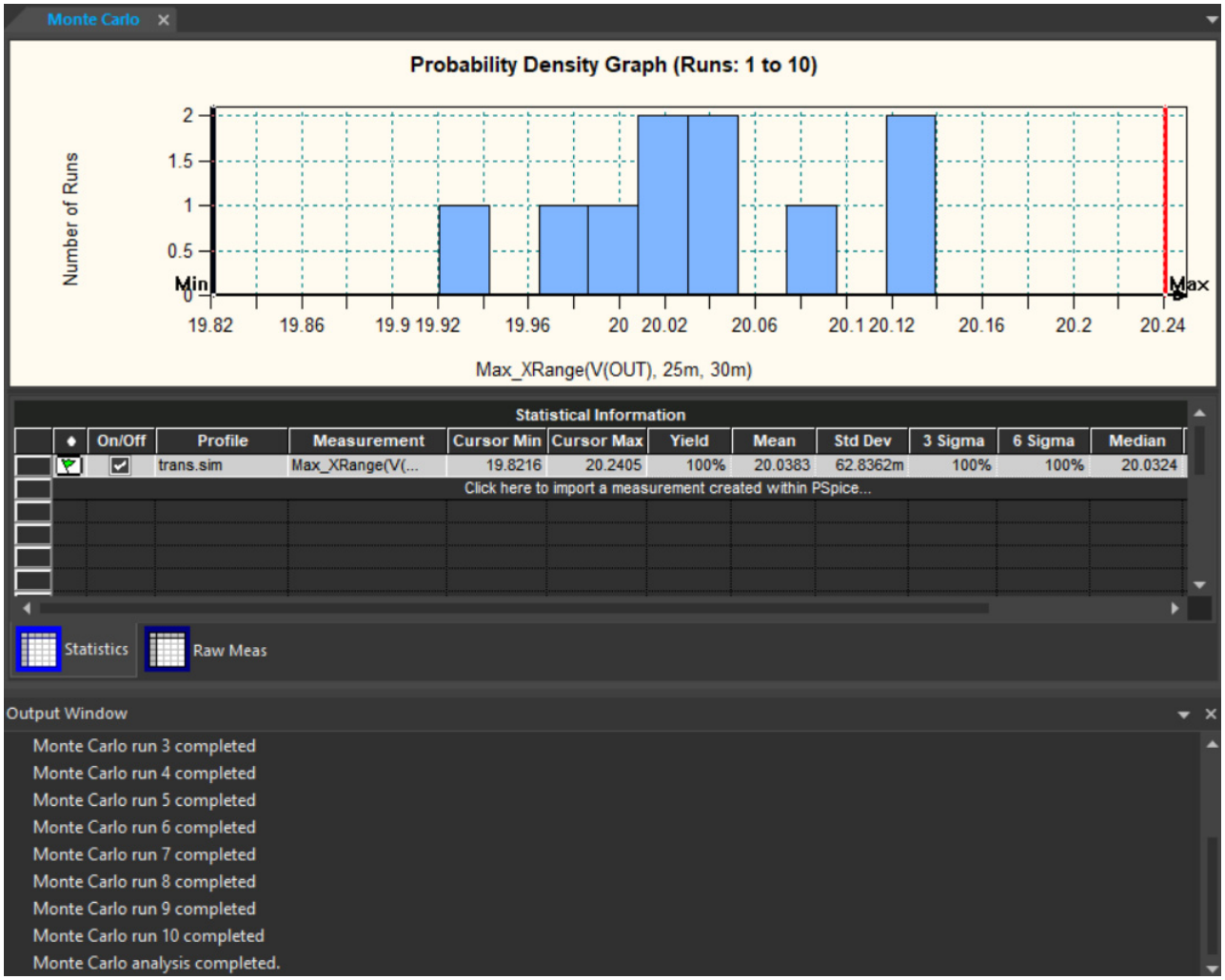
1. In Capture, choose *PSpice – Advanced Analysis – Monte Carlo*.
2. Click the text, *Click here to import a measurement created within PSpice*.

3. Choose `Max_XRange` from the Import Measurement(s) dialog box and click *OK*.



4. Choose *Run – Start Monte Carlo* to run the Monte Carlo analysis.

5. As Monte Carlo Analysis is completed, *Probability Density Graph* is displayed.

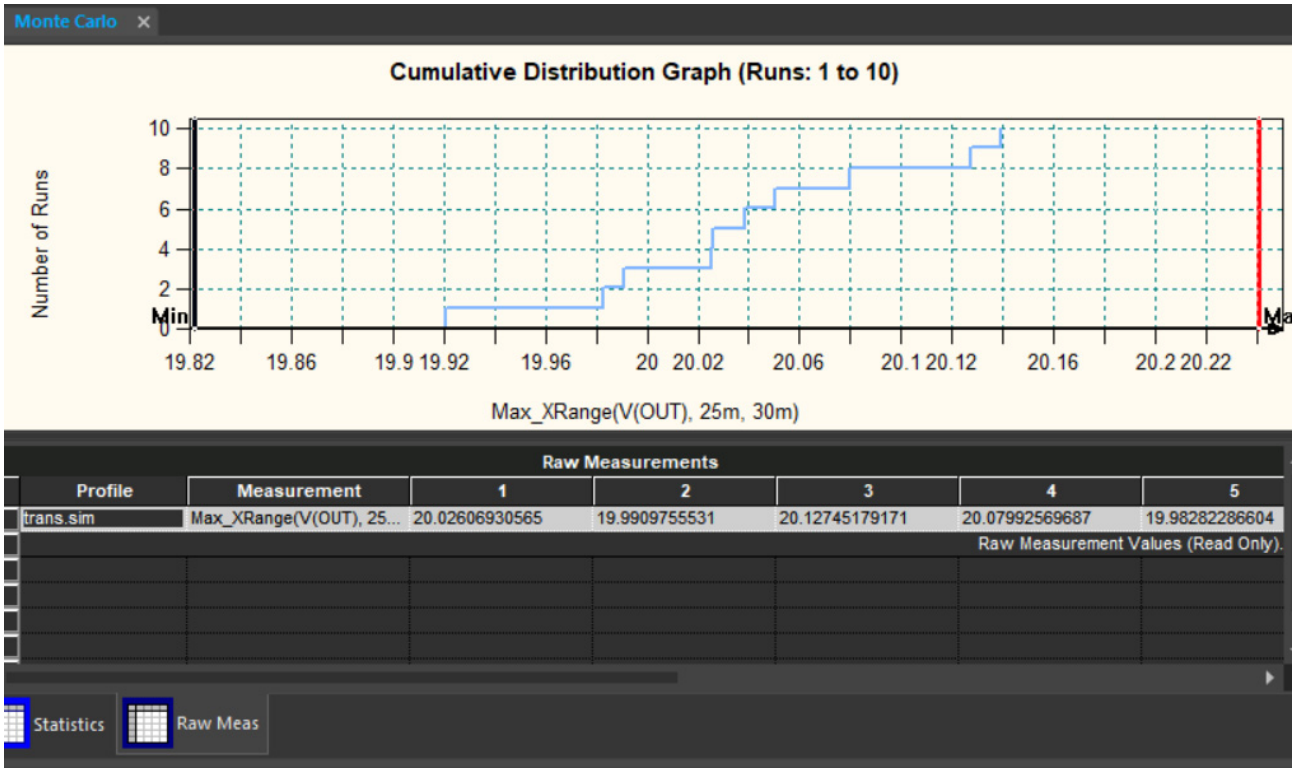


In the Probability Density Graph, every Monte Carlo Analysis run is within the Cursor Minimum value and Cursor Maximum value, which concludes that the yield of the SMPS design is 100%. The yield information is shown in the Statistical Information tab.

6. Right-click this graph and choose *MC Graph (PDF/CDF)* to view data in a Cumulative Distribution Graph.

7. Click the *Raw Measurements* tab.

This tab displays the measurement data for every run of the simulation.

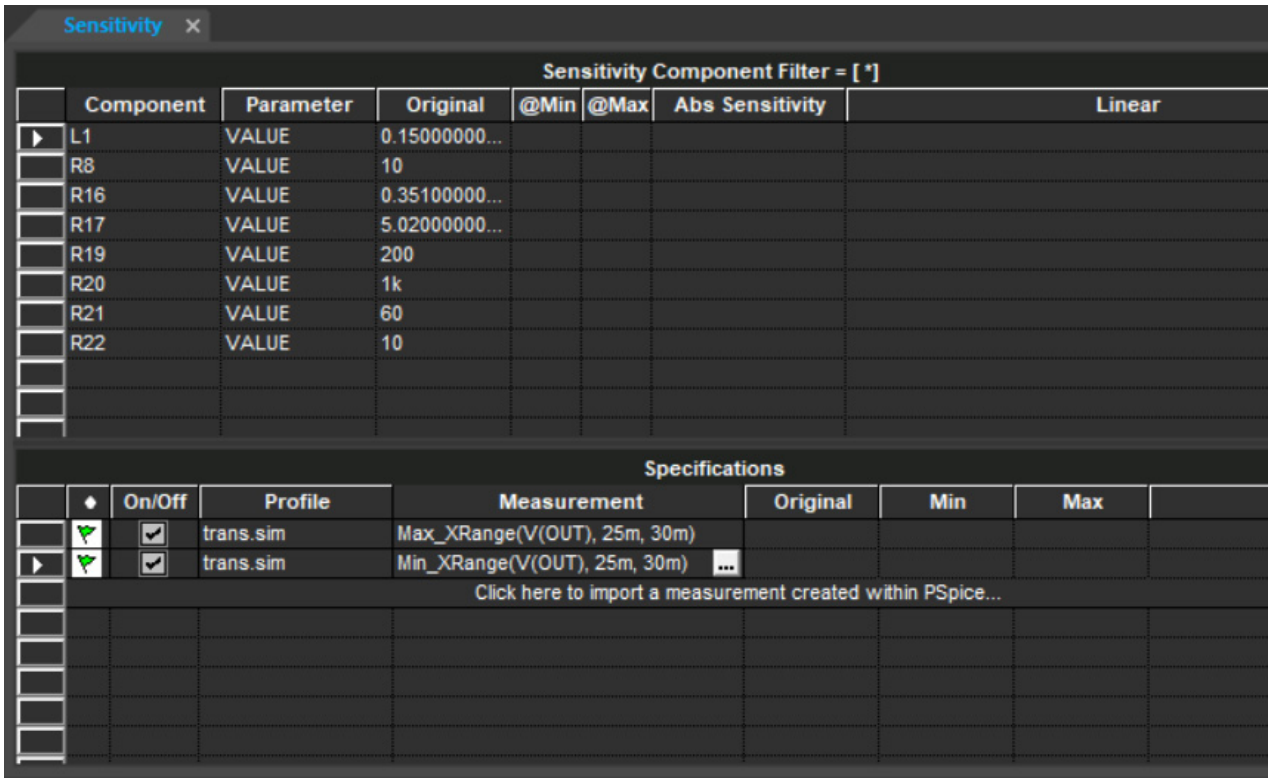


Running Sensitivity and Optimizer Analysis using PSpice Advanced Analysis

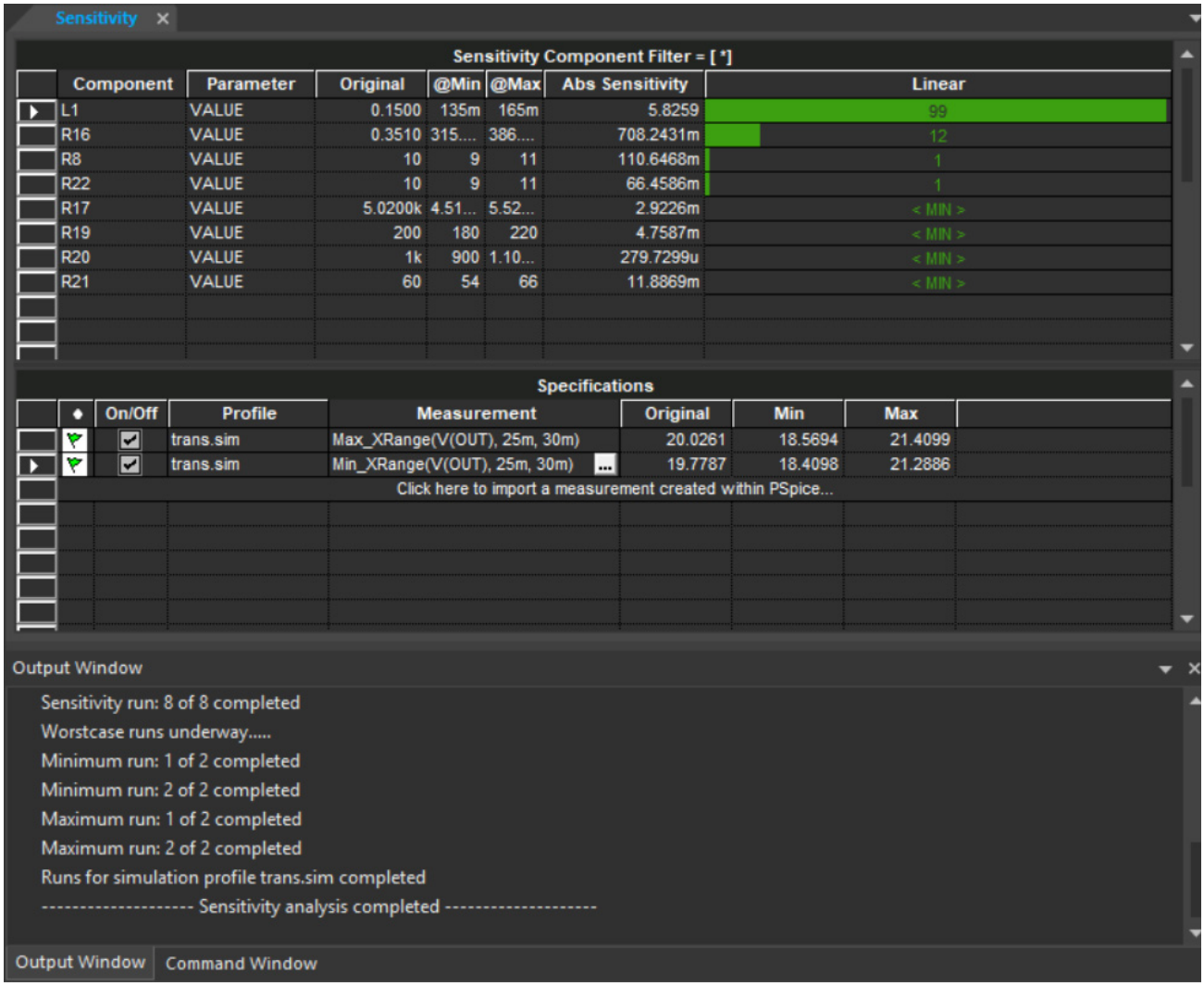
Optimizer is a design tool for optimizing analog circuits and their behavior. It helps you modify and optimize analog designs to meet your performance goals. Optimizer fine tunes your designs faster than trial and error bench testing methods. Use Optimizer to find the best component or system values for your specifications.

Run Sensitivity Analysis, before running Optimizer, to identify the most sensitive component in the design.

1. In Capture, choose *PSpice – Advanced Analysis – Sensitivity* to run Sensitivity Analysis.
2. Click the text, *Click here to import a measurement created within PSpice*.
3. Choose `Max_XRange` from the Import Measurement(s) dialog box and click *OK*.
4. Similarly, import `Min_XRange`.



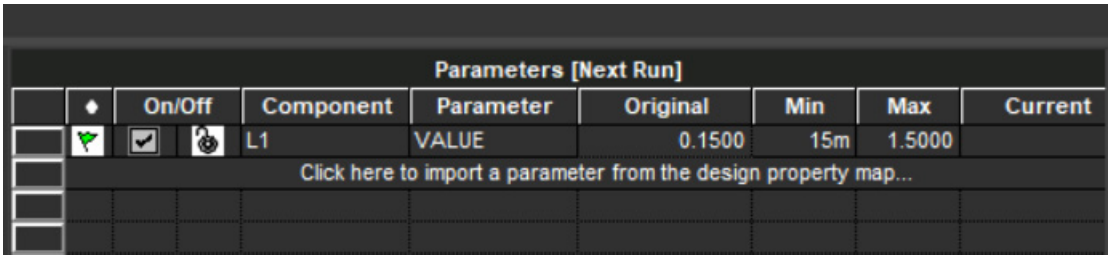
5. Run Sensitivity Analysis.



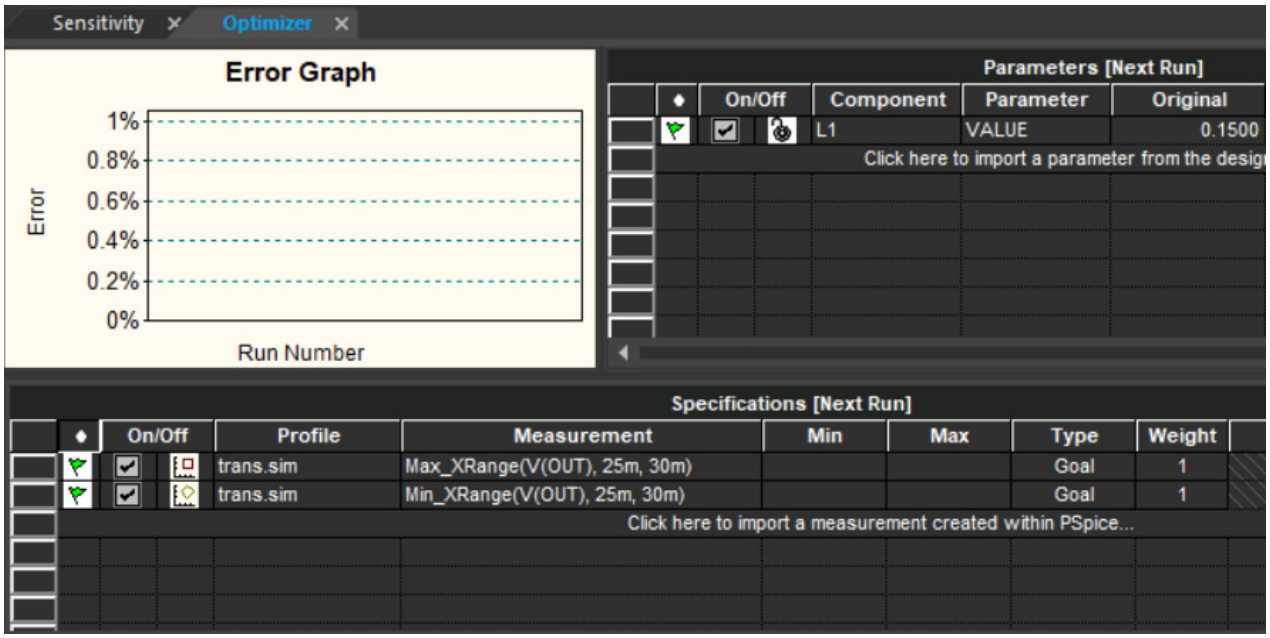
The results shows that L1 is the most sensitive component.

6. Right-click *L1* and choose *Send to Optimizer*.

L1 is added as the parameter in the *Parameters[Next Run]* section.



7. Click *Click here to import a measurement created within PSpice* to import the measurement `Max_XRange` in the *Specifications[Next Run]* section.
8. Choose `Max_XRange` from the Import Measurement(s) dialog and click *OK*.
9. Similarly, import `Min_XRange` in the *Specifications[Next Run]* section.



10. Specify goals by defining minimum and maximum measurement values for the imported measurements.

Measurement Type	Minimum Measurement	Maximum Measurement
`Max_Range`	`18`	`20.5`
`Min_Range`	`18`	`19`

11. Specify minimum and maximum for the component as well.

Component Type	Minimum Measurement	Maximum Measurement
L1	100m	250m

12. Run Optimizer Analysis.

From the Optimizer Analysis results, you can see that the optimized value of the L1 component is 149.82m for the goals defined in *Standard* tab.

Recommended Reading

For more information about various advanced analysis, such as, Monte Carlo Analysis, Parametric Plot Analysis, Optimizer Analysis, and Sensitivity Analysis, see PSpice Advanced Analysis User Guide and PSpice User Guide.