

Simulating an SMPS Design using Capture-PSpice Flow

**Product Version 17.4-2019
October 2019**

Document Last Updated: June 2020

© 1999-2020 Cadence Design Systems, Inc. All rights reserved.

Portions © Apache Software Foundation, Sun Microsystems, Free Software Foundation, Inc., Regents of the University of California, Massachusetts Institute of Technology, University of Florida. Used by permission. Printed in the United States of America.

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

Product PSpice contains technology licensed from, and copyrighted by: Apache Software Foundation, 1901 Munsey Drive Forest Hill, MD 21050, USA © 2000-2005, Apache Software Foundation. Sun Microsystems, 4150 Network Circle, Santa Clara, CA 95054 USA © 1994-2007, Sun Microsystems, Inc. Free Software Foundation, 59 Temple Place, Suite 330, Boston, MA 02111-1307 USA © 1989, 1991, Free Software Foundation, Inc. Regents of the University of California, Sun Microsystems, Inc., Scriptics Corporation, © 2001, Regents of the University of California. Daniel Stenberg, © 1996 - 2006, Daniel Stenberg. UMFPACK © 2005, Timothy A. Davis, University of Florida, (davis@cise.ulf.edu). Ken Martin, Will Schroeder, Bill Lorensen © 1993-2002, Ken Martin, Will Schroeder, Bill Lorensen. Massachusetts Institute of Technology, 77 Massachusetts Avenue, Cambridge, Massachusetts, USA © 2003, the Board of Trustees of Massachusetts Institute of Technology. All rights reserved.

Trademarks: Trademarks and service marks of Cadence Design Systems, Inc. contained in this document are attributed to Cadence with the appropriate symbol. For queries regarding Cadence's trademarks, contact the corporate legal department at the address shown above or call 800.862.4522.

Open SystemC, Open SystemC Initiative, OSCI, SystemC, and SystemC Initiative are trademarks or registered trademarks of Open SystemC Initiative, Inc. in the United States and other countries and are used with permission.

All other trademarks are the property of their respective holders.

Restricted Permission: This publication is protected by copyright law and international treaties and contains trade secrets and proprietary information owned by Cadence. Unauthorized reproduction or distribution of this publication, or any portion of it, may result in civil and criminal penalties. Except as specified in this permission statement, this publication may not be copied, reproduced, modified, published, uploaded, posted, transmitted, or distributed in any way, without prior written permission from Cadence. Unless otherwise agreed to by Cadence in writing, this statement grants Cadence customers permission to print one (1) hard copy of this publication subject to the following conditions:

1. The publication may be used only in accordance with a written agreement between Cadence and its customer.
2. The publication may not be modified in any way.
3. Any authorized copy of the publication or portion thereof must include all original copyright, trademark, and other proprietary notices and this permission statement.
4. The information contained in this document cannot be used in the development of like products or software, whether for internal or external use, and shall not be used for the benefit of any other party, whether or not for consideration.

Disclaimer: Information in this publication is subject to change without notice and does not represent a commitment on the part of Cadence. Except as may be explicitly set forth in such agreement, Cadence does not make, and expressly disclaims, any representations or warranties as to the completeness, accuracy or usefulness of the information contained in this document. Cadence does not warrant that use of such information will not infringe any third party rights, nor does Cadence assume any liability for damages or costs of any kind that may result from use of such information.

Restricted Rights: Use, duplication, or disclosure by the Government is subject to restrictions as set forth in FAR52.227-14 and DFAR252.227-7013 et seq. or its successor.

Contents

<u>Preface</u>	5
<u>Audience</u>	5
<u>Prerequisites</u>	5
<u>Design and Library Files</u>	5
<u>PSpice Tutorial Libraries</u>	5
<u>Enabling the SMPS design for Simulation</u>	7
<u>Opening the SMPS design in Capture</u>	7
<u>What's Next</u>	9
<u>Recommended Reading</u>	10
<u>Setting Up and Running a PSpice Simulation</u>	11
<u>Creating a PSpice Simulation Profile</u>	11
<u>Simulating the Design using PSpice</u>	15
<u>What's Next</u>	16
<u>Recommended Reading</u>	16
<u>Verifying Stress Levels of Components in SMPS</u>	17
<u>Identifying Stressed Components: Smoke Analysis using PSpice Advanced Analysis</u> ..	17
<u>Correcting Stress Levels using PSpice Advanced Analysis</u>	20
<u>What's Next</u>	21
<u>Recommended Reading</u>	21
<u>Evaluating Stability and Optimization: Creating</u>	
<u>Measurements</u>	23
<u>Creating Measurements</u>	23
<u>What's Next</u>	25
<u>Recommended Reading</u>	25
<u>Verifying Design Stability and Yield</u>	27
<u>Running Parametric Plotter</u>	27
<u>Calculating Yield by Running Monte-Carlo</u>	36
<u>Running Monte-Carlo using PSpice</u>	36
<u>Running Monte-Carlo using PSpice Advanced Analysis</u>	42

Simulating an SMPS Design using Capture-PSpice Flow

<u>Running Sensitivity and Optimizer Analysis using PSpice Advanced Analysis</u>	46
<u>Recommended Reading</u>	49

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

Simulating an SMPS Design using Capture-PSpice Flow

Preface

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_smeps_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.

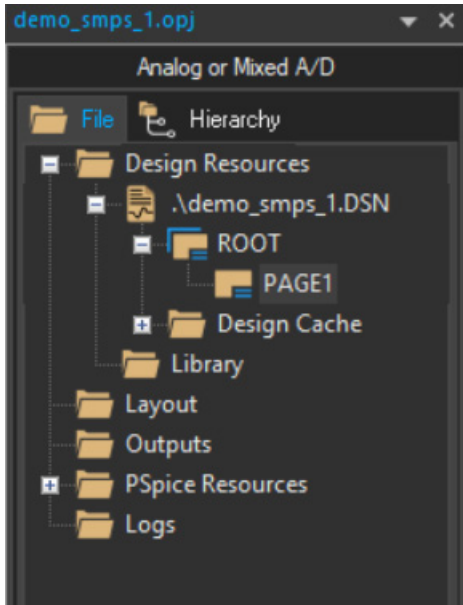
Simulating an SMPS Design using Capture-PSpice Flow

Enabling the SMPS design for Simulation

Observe the project manager window.

The project type is specified as *Analog 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_smeps_1.DSN` and `ROOT`. Double-click `PAGE1` to open the schematic page.



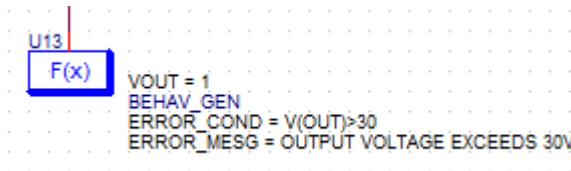
The design contains a hierarchical block *pwmcon* with a pulse-width modulator circuitry controlled by feedback from 18V output. A ferrimeter transformer, designed using Magnetic Parts Editor, is used in the design. A high voltage is switched through the primary winding of the transformer. The secondary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

Simulating an SMPS Design using Capture-PSpice Flow

Enabling the SMPS design for Simulation

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:

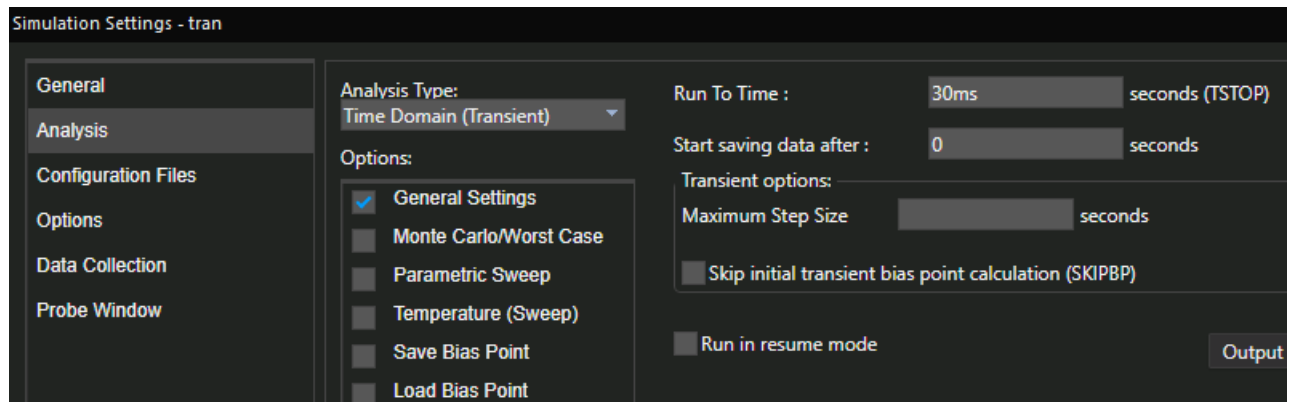
1. Choose *PSpice – New Simulation Profile*.
2. Enter the Name as `trans`.
3. Ensure *Inherit From* is *none*.
4. Click *Create*.
5. Select *Analysis* in the Simulation Settings dialog box.

Simulating an SMPS Design using Capture-PSpice Flow

Setting Up and Running a PSpice Simulation

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.



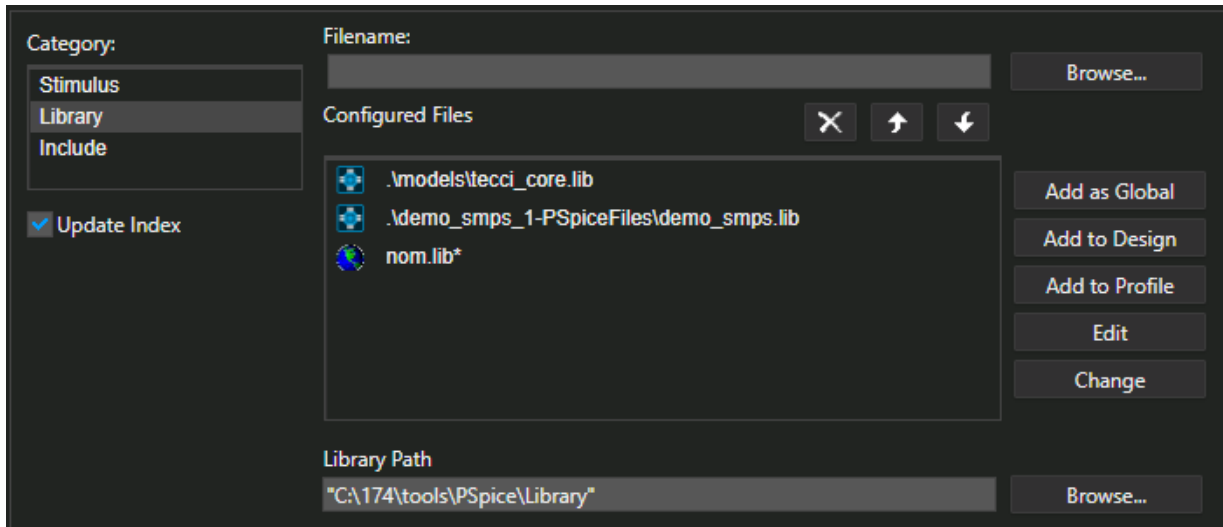
Simulating an SMPS Design using Capture-PSpice Flow

Setting Up and Running a PSpice Simulation

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*.

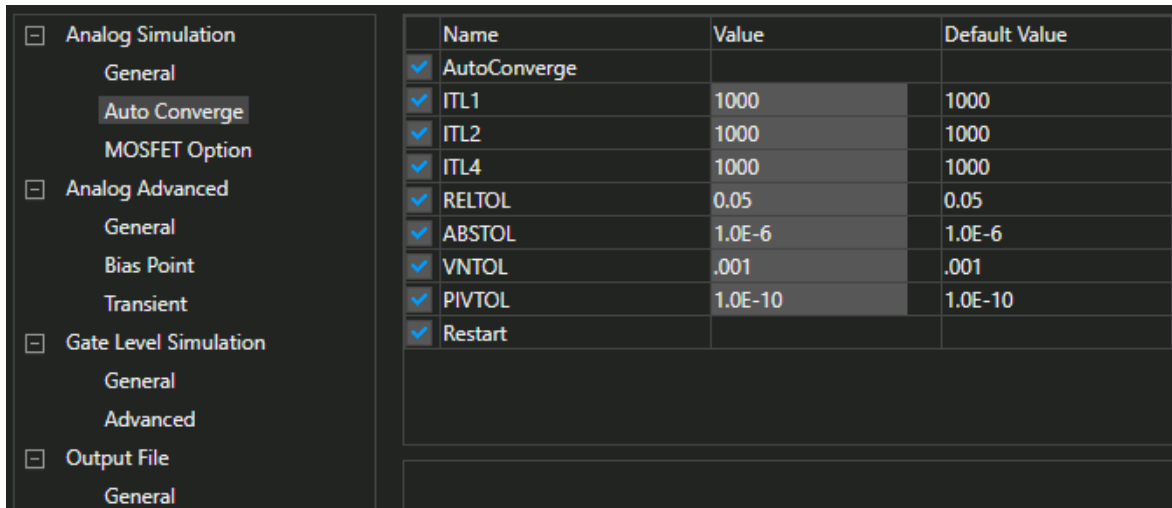


Simulating an SMPS Design using Capture-PSpice Flow

Setting Up and Running a PSpice Simulation

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.

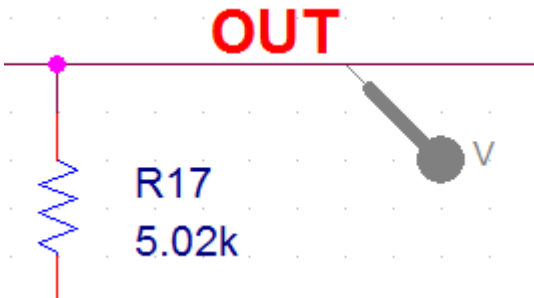
11. Click *Apply* to save changes.

12. Click *OK*.

Simulating the Design using PSpice

Perform the following steps in OrCAD Capture to perform simulation:

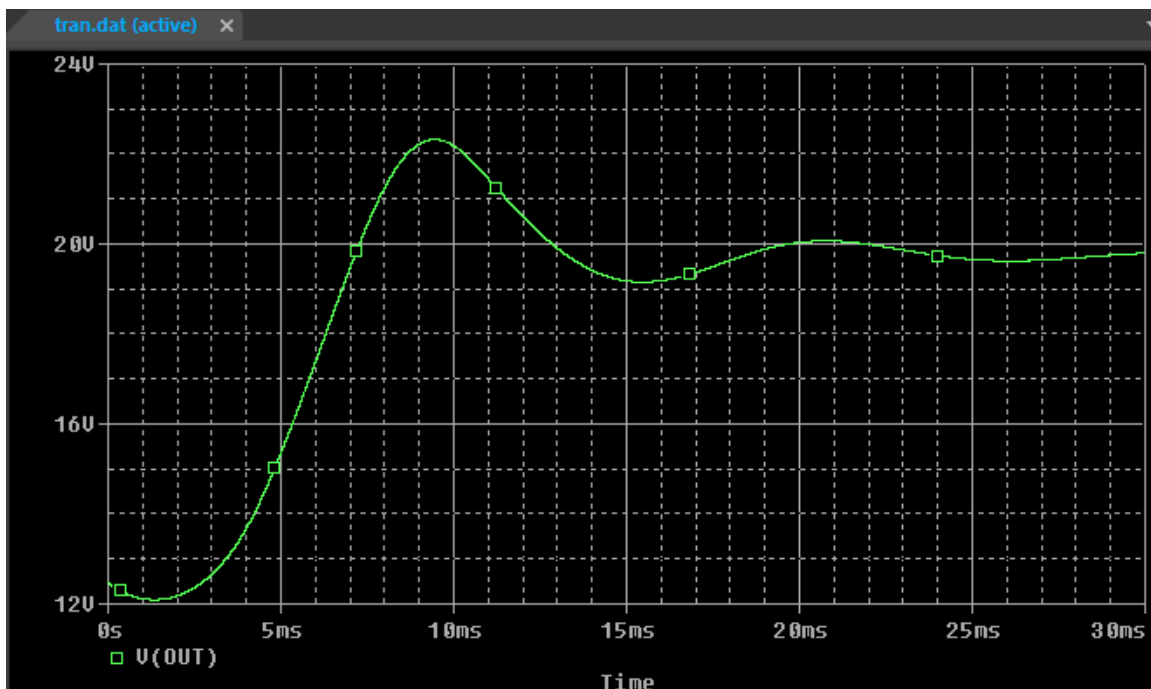
1. Place a voltage probe on the OUT net: choose *PSpice – Markers – Voltage Level* and click on the OUT net.



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

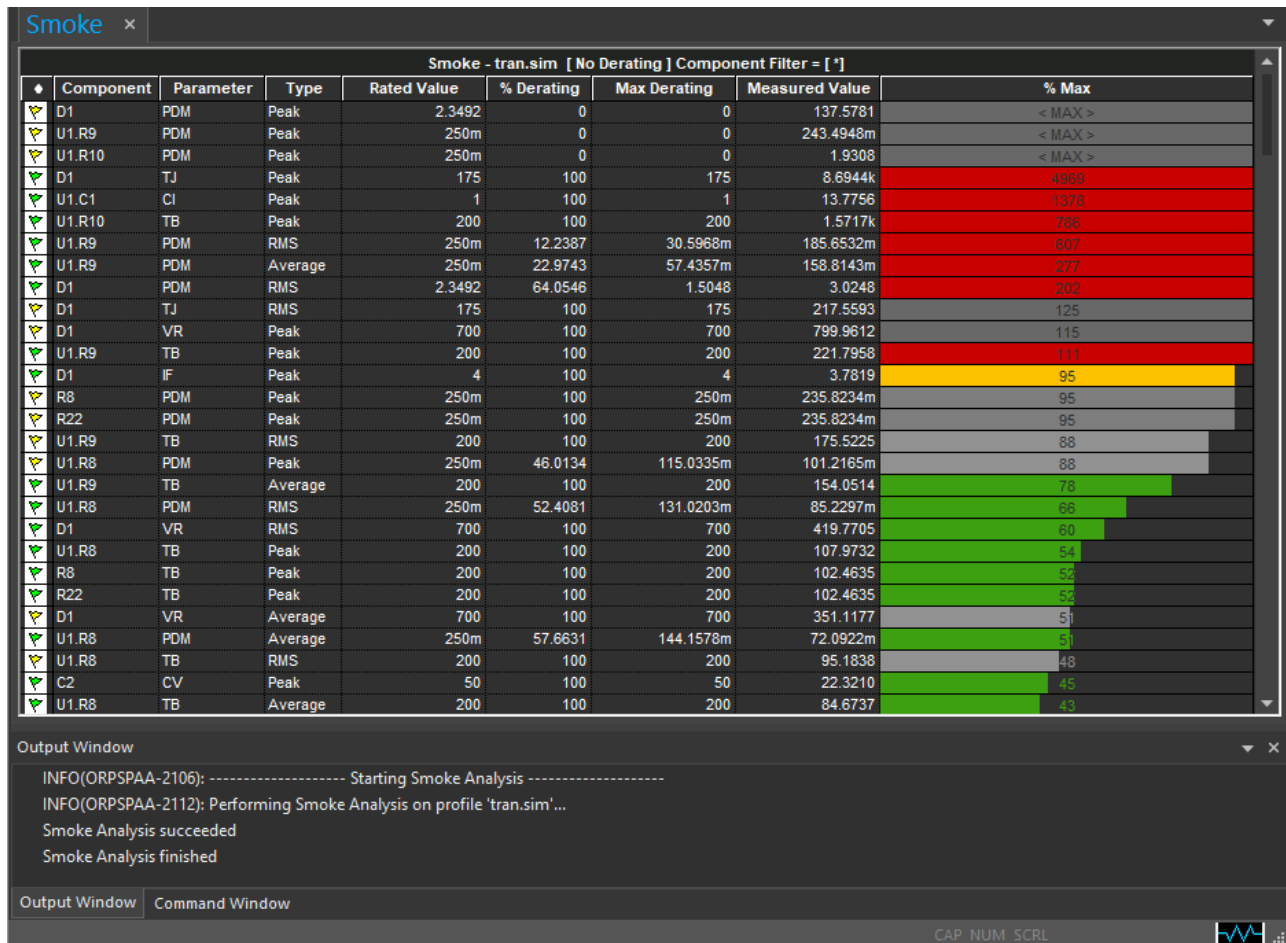
Identifying Stressed Components: Smoke Analysis using PSpice Advanced Analysis

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

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Stress Levels of Components in SMPS

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



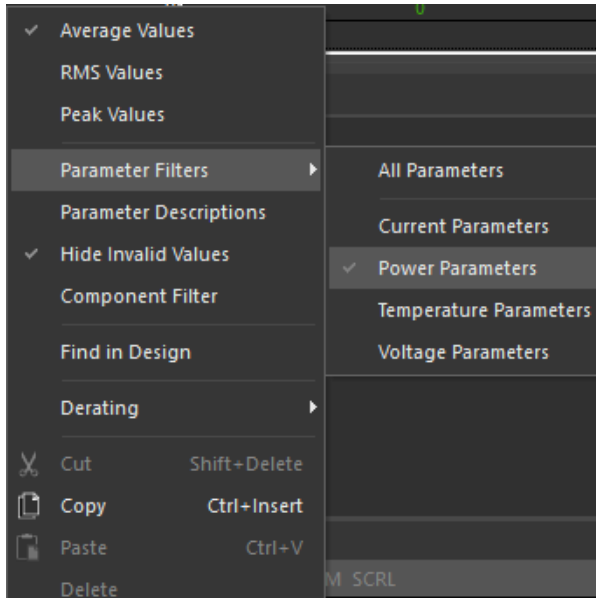
You can right-click in the result pane of the Smoke Analysis window and choose options to see only specific measurements, such as RMS, average, or peak values. You can also choose to view specific parameters.

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.

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Stress Levels of Components in SMPS

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



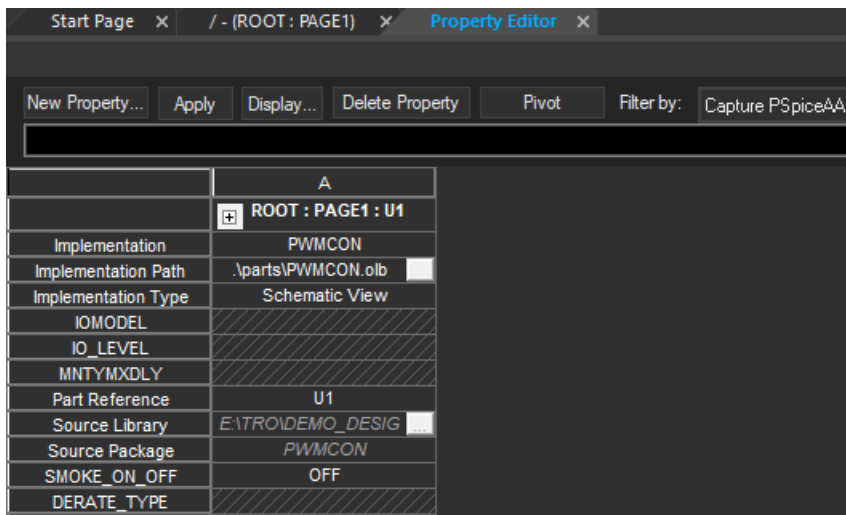
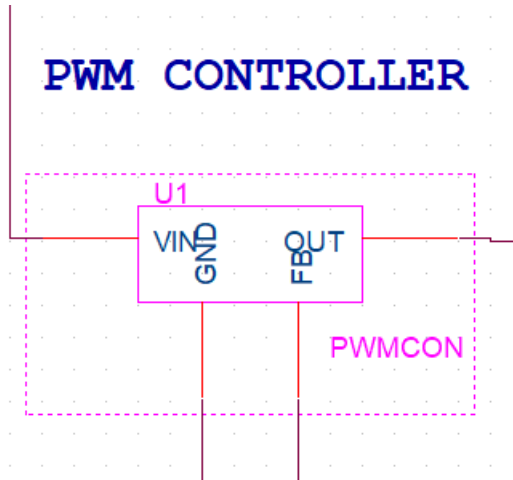
Observe the changes in the result pane of the Smoke Analysis window.

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*).

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Stress Levels of Components in SMPS

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*.

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Stress Levels of Components in SMPS

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*.

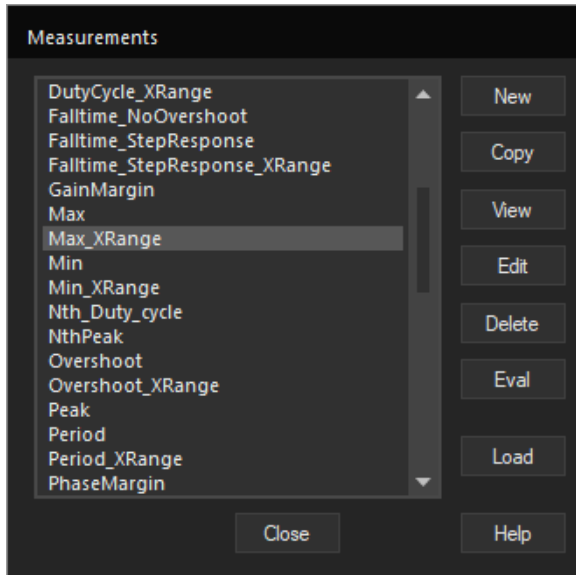
The Measurements dialog box appears.

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

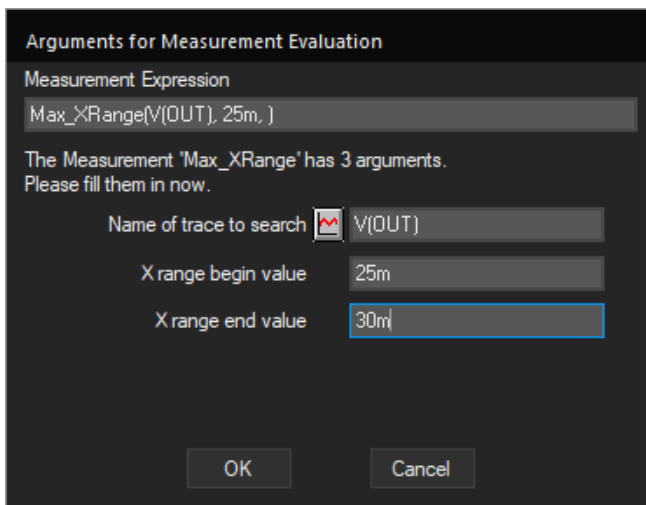
Simulating an SMPS Design using Capture-PSpice Flow

Evaluating Stability and Optimization: Creating Measurements

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.

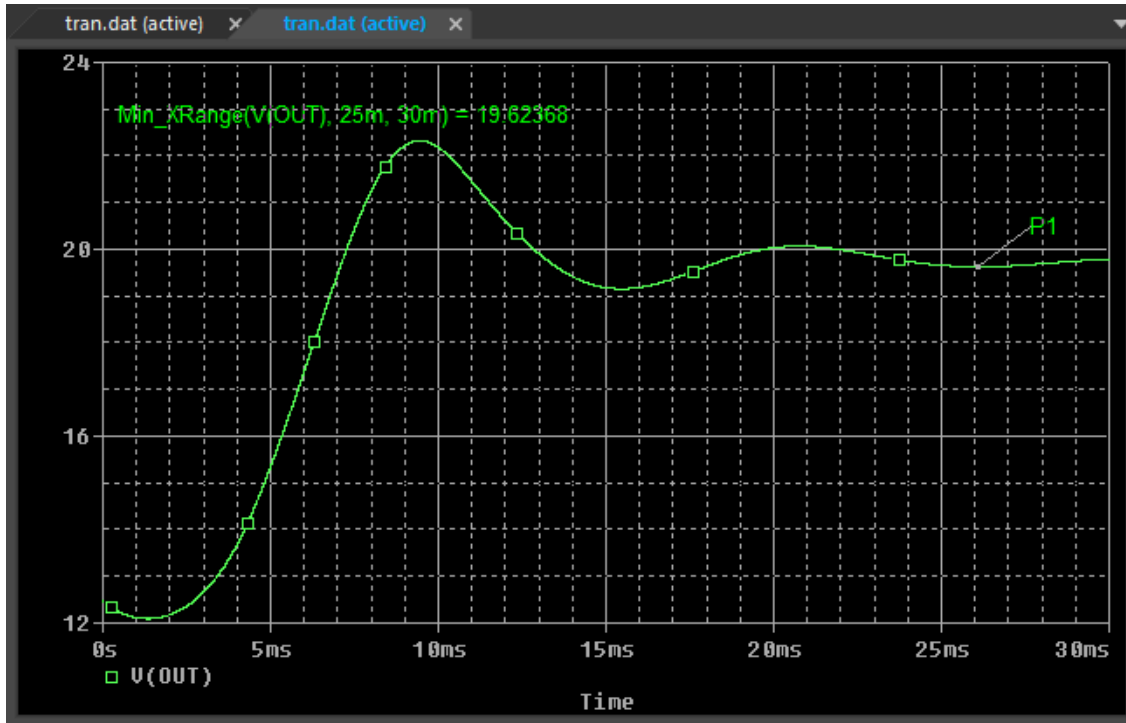


6. Click *OK*.
7. Click *OK* in the Display Measurement Evaluation message box after viewing the results.
8. Click *Close* in the Measurements dialog box.

Simulating an SMPS Design using Capture-PSpice Flow

Evaluating Stability and Optimization: Creating Measurements

9. Similarly add *Min_XRange*.



What's Next

Next, you will verify 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*.

Simulating an SMPS Design using Capture-PSpice Flow

Evaluating Stability and Optimization: Creating Measurements

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

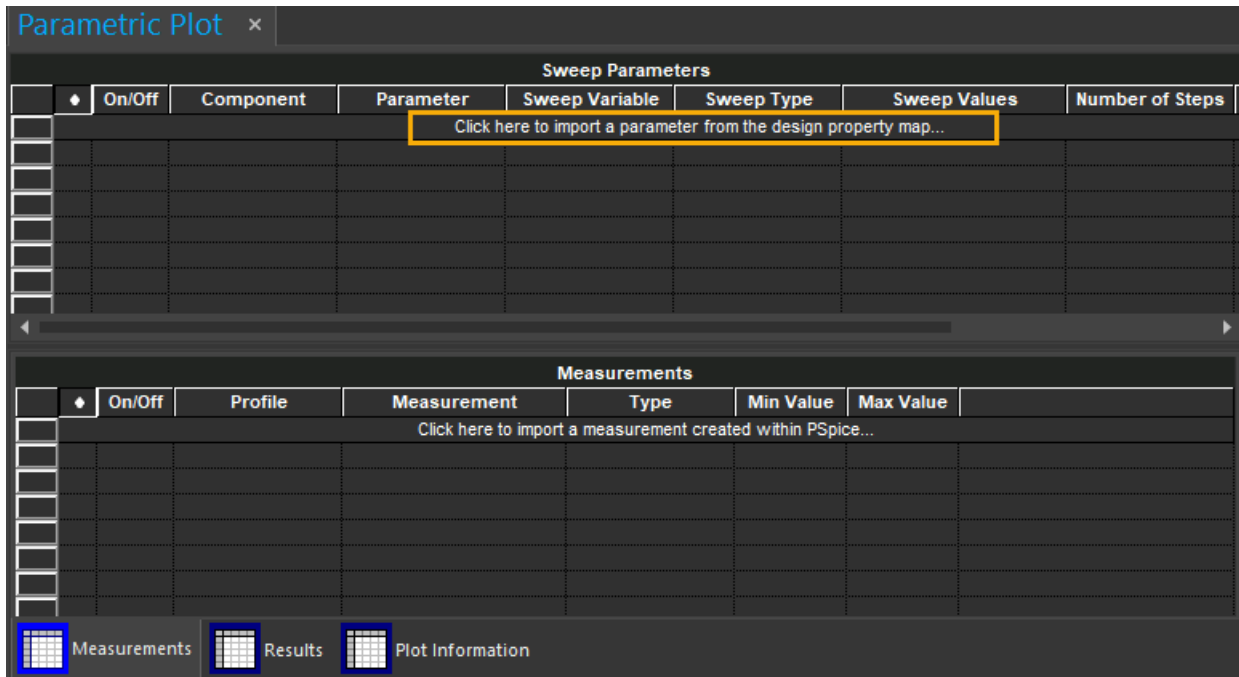
In Parametric Plotter, you analyze sweep results from multiple parameters and you can sweep any number of design and model parameters (in any combinations) and view results in Plot/Probe in tabular or plot form. You will run Parametric Plot analysis to ensure that the design is stable for a range of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

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

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

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

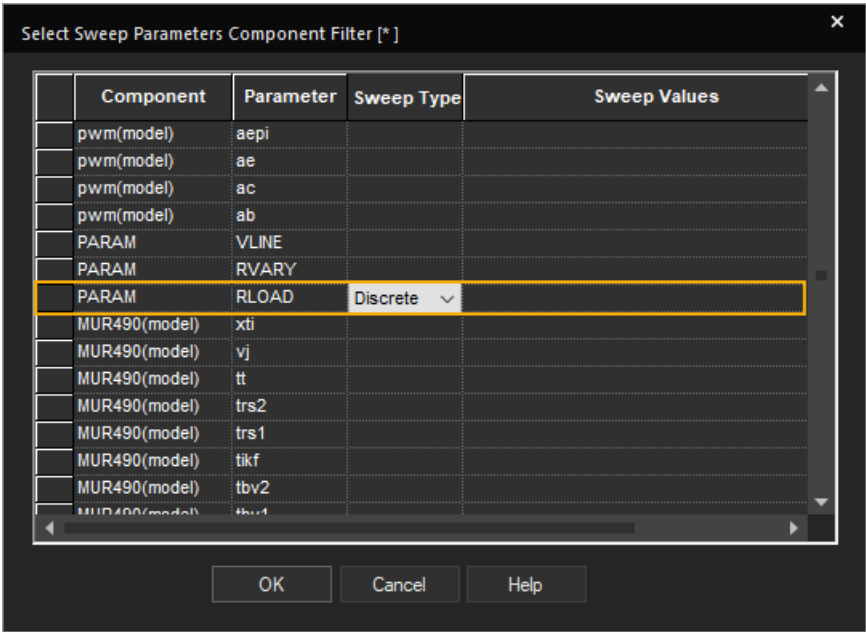


3. In the Select Sweep Parameters Component Filter window, scroll down to the Parameter **RLOAD** and click in the *Sweep Type* column for this parameter.

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

4. Choose `Discrete` from the list.





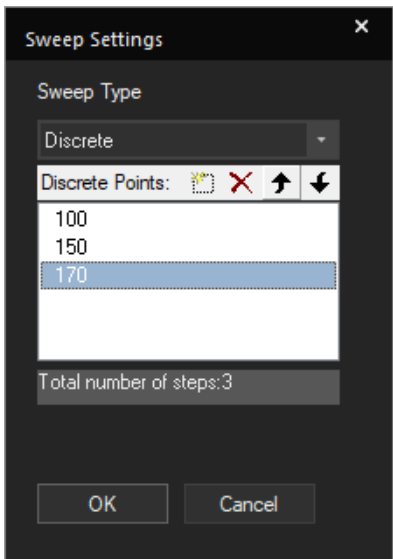
5. Click in the *Sweep Values* column to open the Sweep Settings dialog box.

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

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.

VLINE	Discrete	Values:250,300,350
RVARY		
RLOAD	Discrete	Values:100,150,170

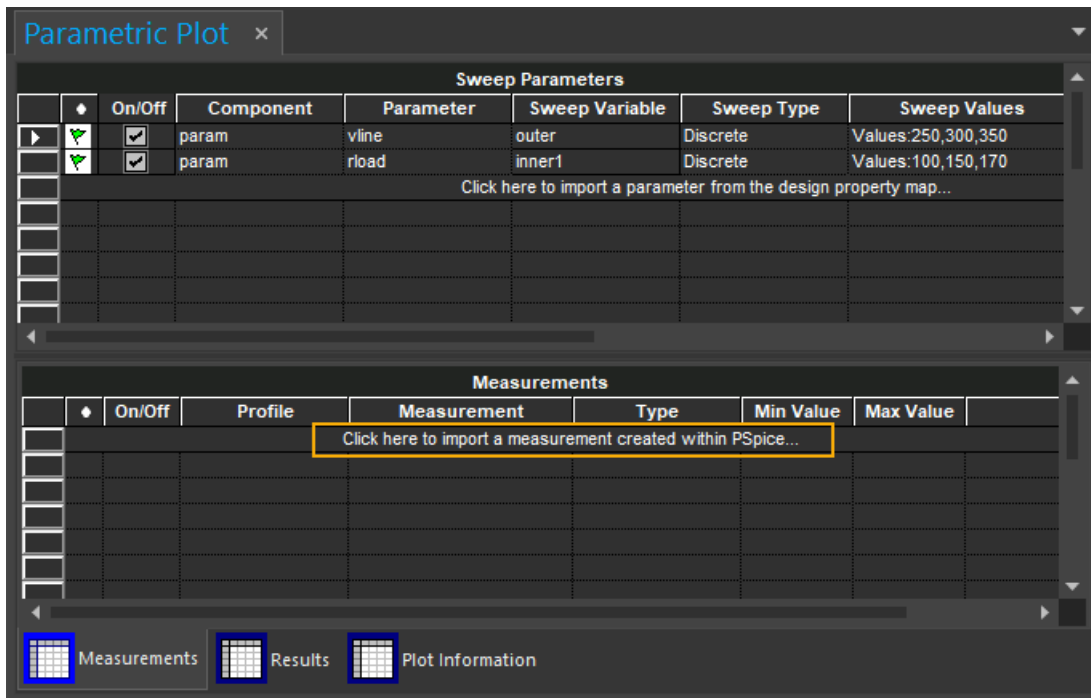
You will simulate the design for different values of `RLOAD` and `VLINE` to observe the impact of this variation on the output voltage, V_{out} .

Simulating an SMPS Design using Capture-PSpice Flow

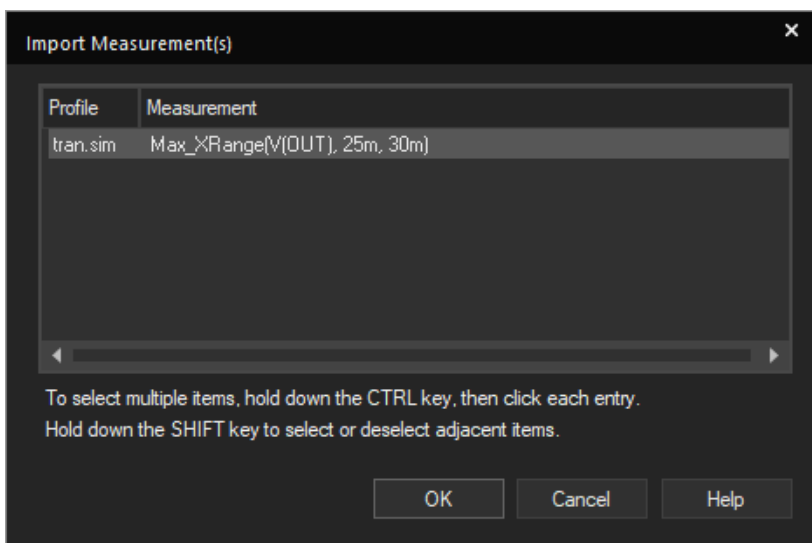
Verifying Design Stability and Yield

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*.




Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

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.

Results		
param::vline	param::rload	tran.sim::max_xrange(v(out),25m,30m
250	100	1.005314731938
250	150	2.365465413281
250	170	2.93321713264
300	100	1.041816425013
300	150	2.904458403611
300	170	19.57416318329
350	100	1.120732742183
350	150	19.61458022105
350	170	19.70411021654

 Measurements
  Results
  Plot Information

Output Window


```

----- Starting Parametric Plotter -----
Processing analysis specifications
Number of parametric sweeps: 9
-- Loading simulation profile tran.sim --
Parametric sweep run 1
Parametric sweep run 2
Parametric sweep run 3
Parametric sweep run 4
Parametric sweep run 5
        
```

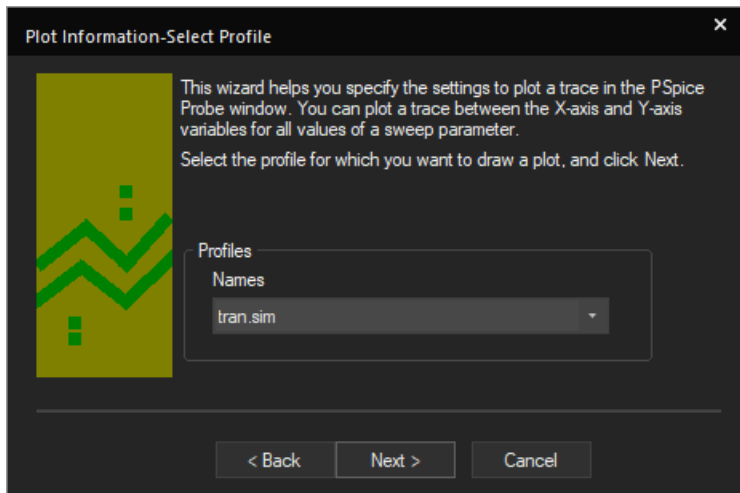
Output Window
 Command Window

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

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

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.

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

From the result of the parametric plotter analysis observe that:

- For VLINE250

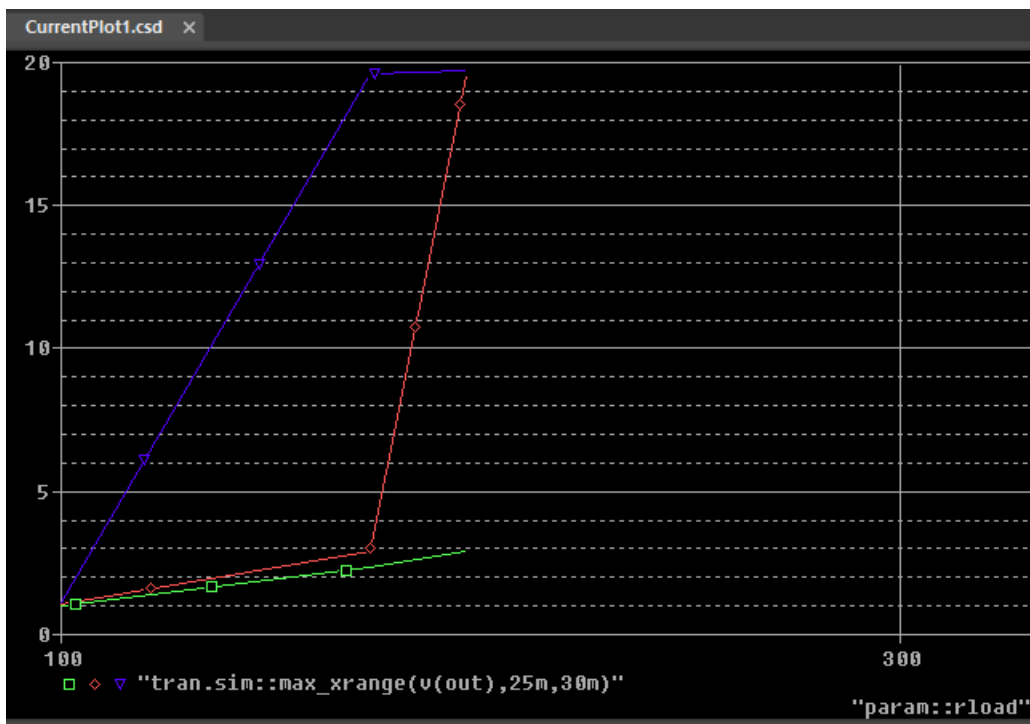
VOUT changes abruptly from 1.2V to 19V when RLOAD changes from 100 ohms to 150 ohms, but when *RLOAD* changes from 150 ohms to 170 ohms the *Vout* remains constant.

- 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

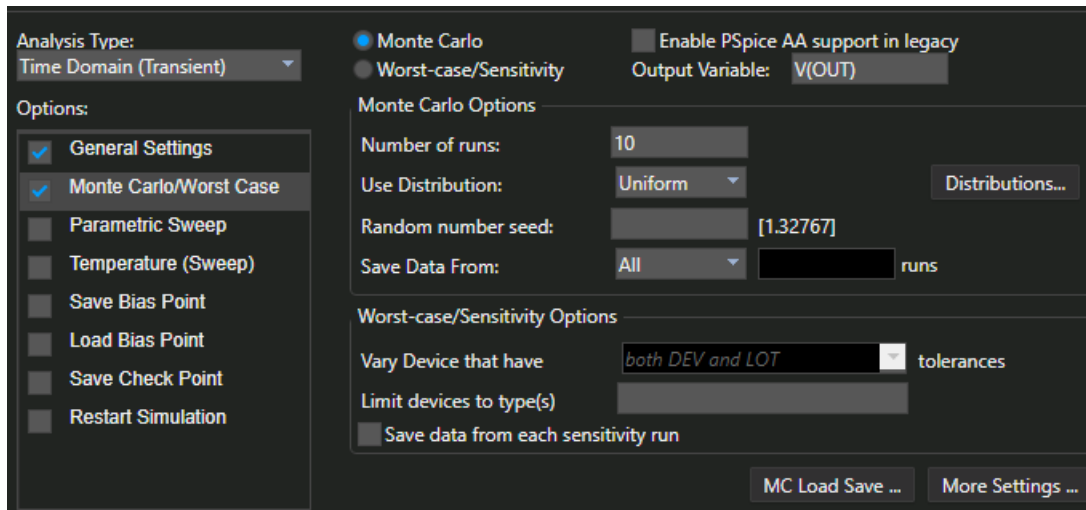
For the SMPS design, run the Monte Carlo Analysis in Time Domain to calculate the yield:

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

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

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



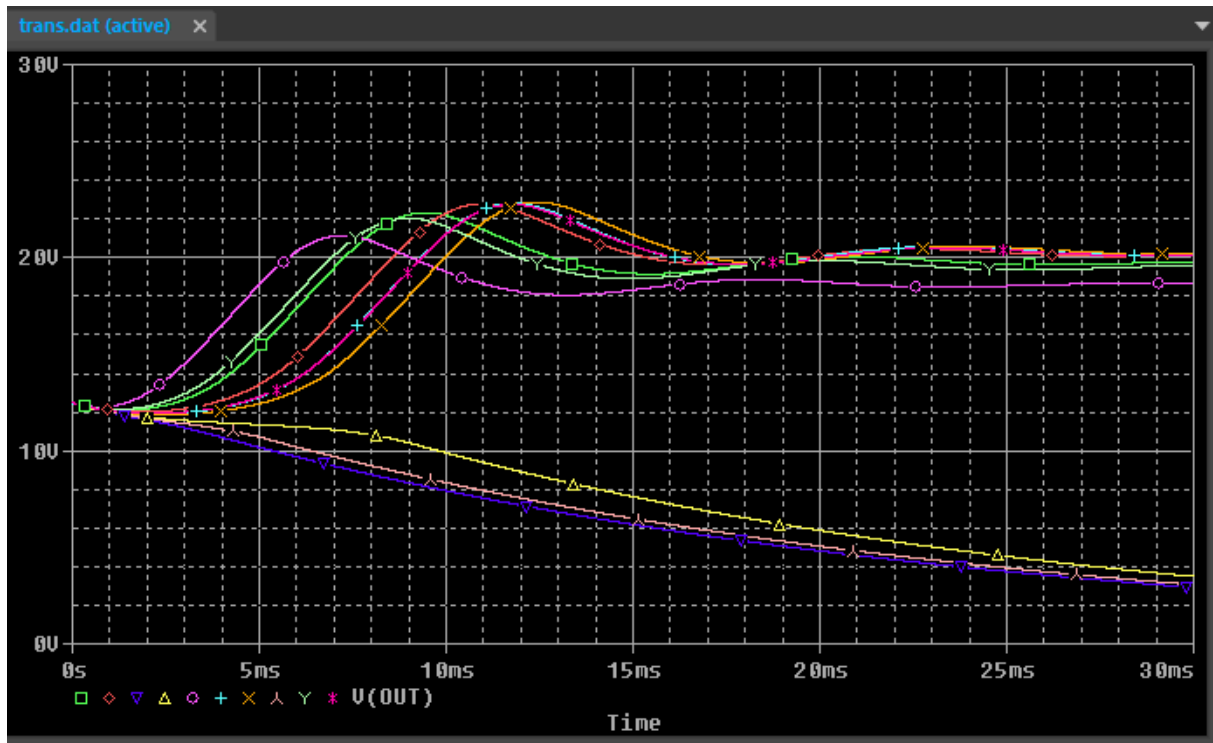
4. Click *Apply* and then click *OK* to save the settings and close the Simulation Settings dialog box.

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

5. Run the PSpice simulation.

The PSpice probe window displays the simulation result.

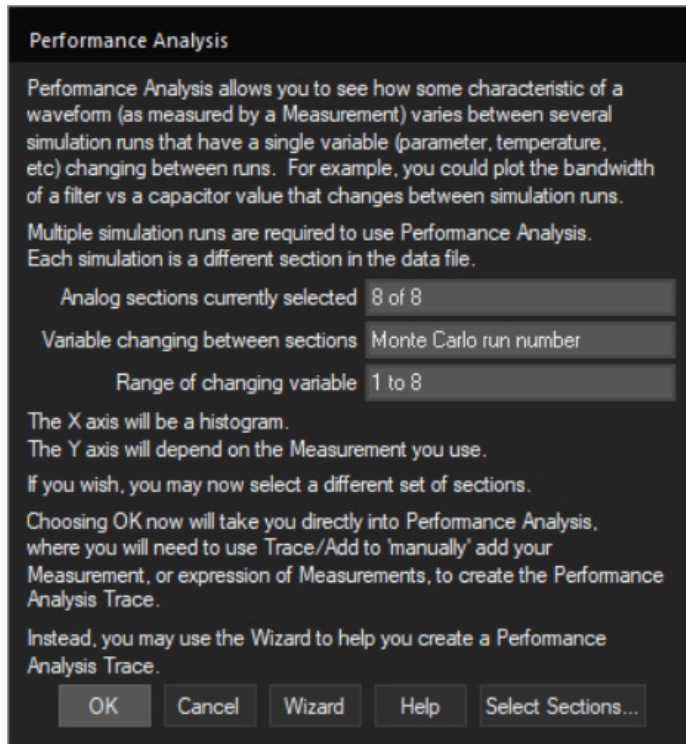


6. In PSpice, choose *Trace – Performance Analysis* to compare the different waveforms generated using Monte Carlo Analysis.

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

The Performance Analysis dialog box appears.

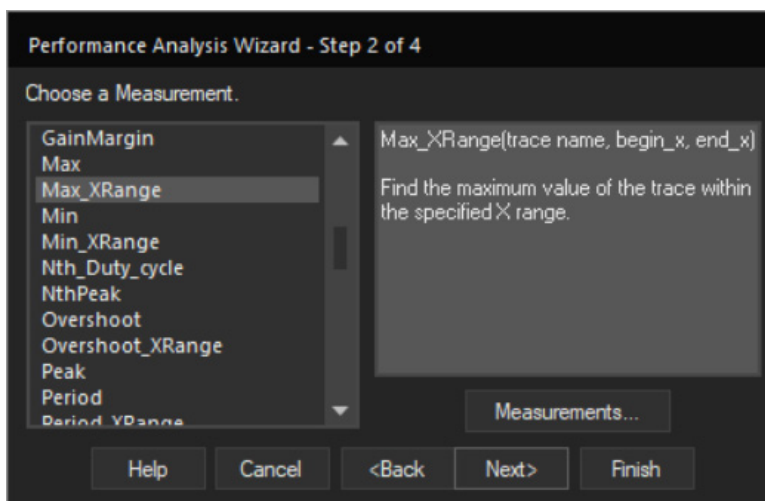


7. Click *Wizard* in the Performance Analysis dialog box.

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*.

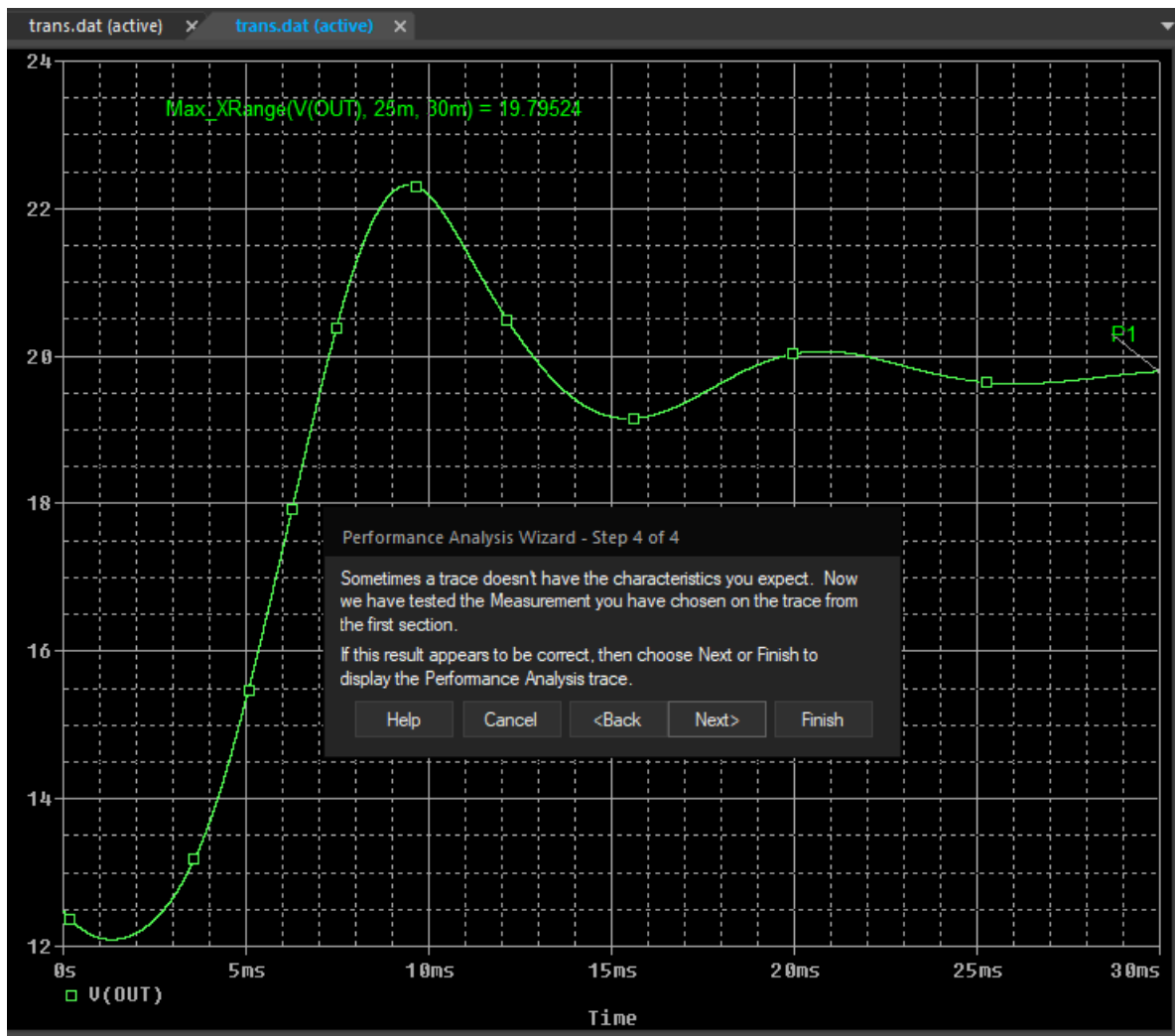


Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

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.

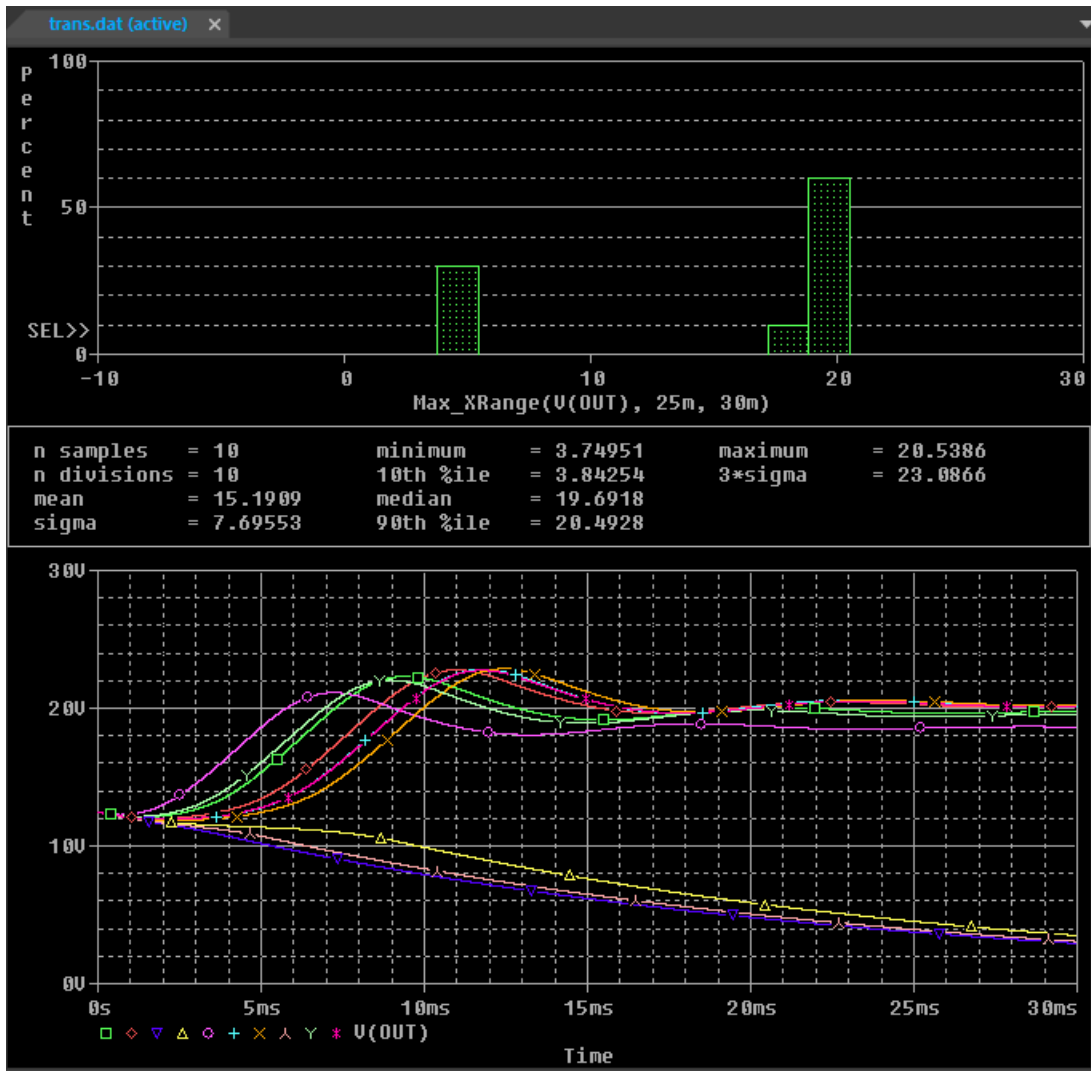


Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

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).

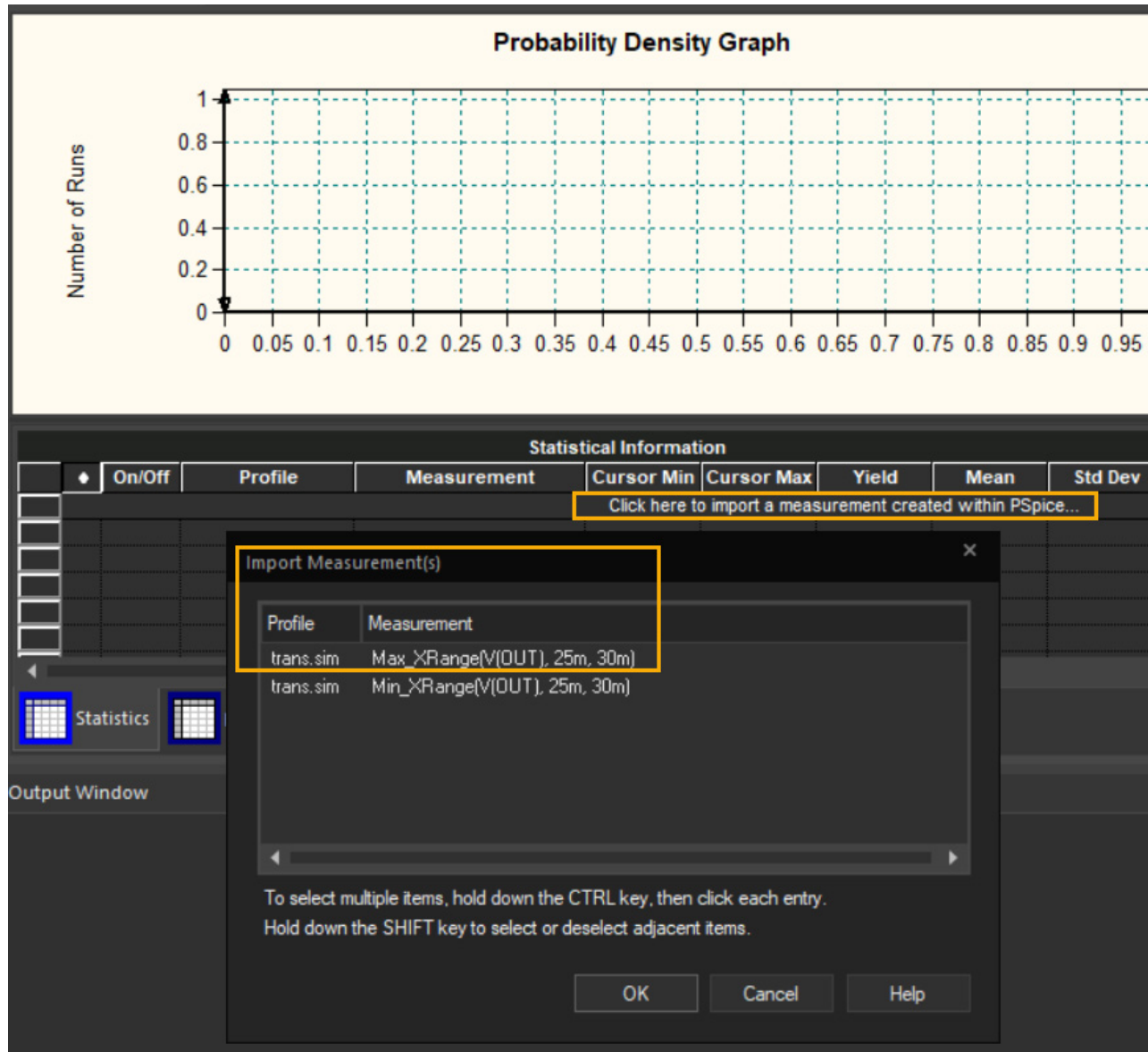
To run Monte-Carlo using PSpice Advanced Analysis, do the following:

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

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

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

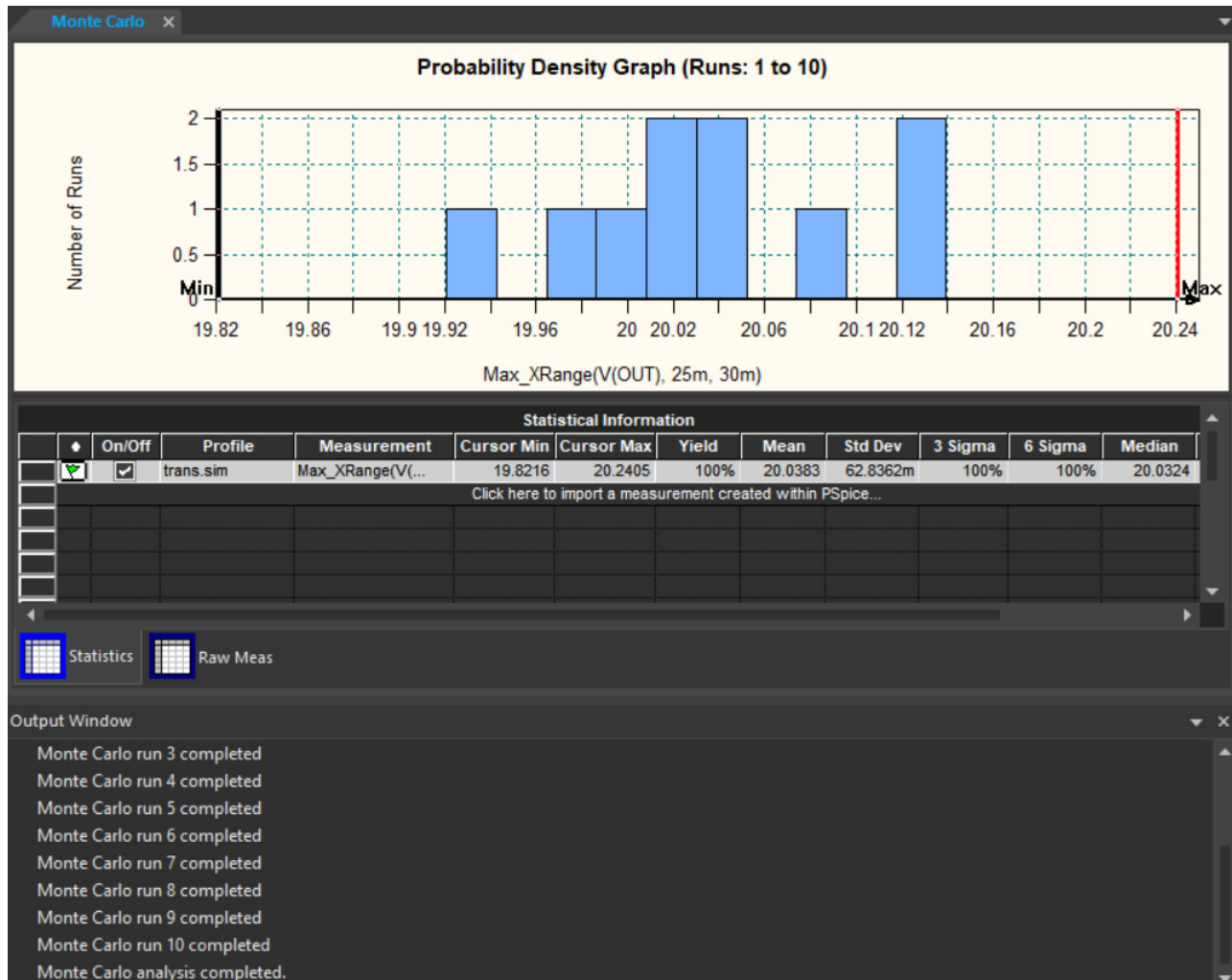


4. Choose *Run – Start Monte Carlo* to run the Monte Carlo analysis.

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

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

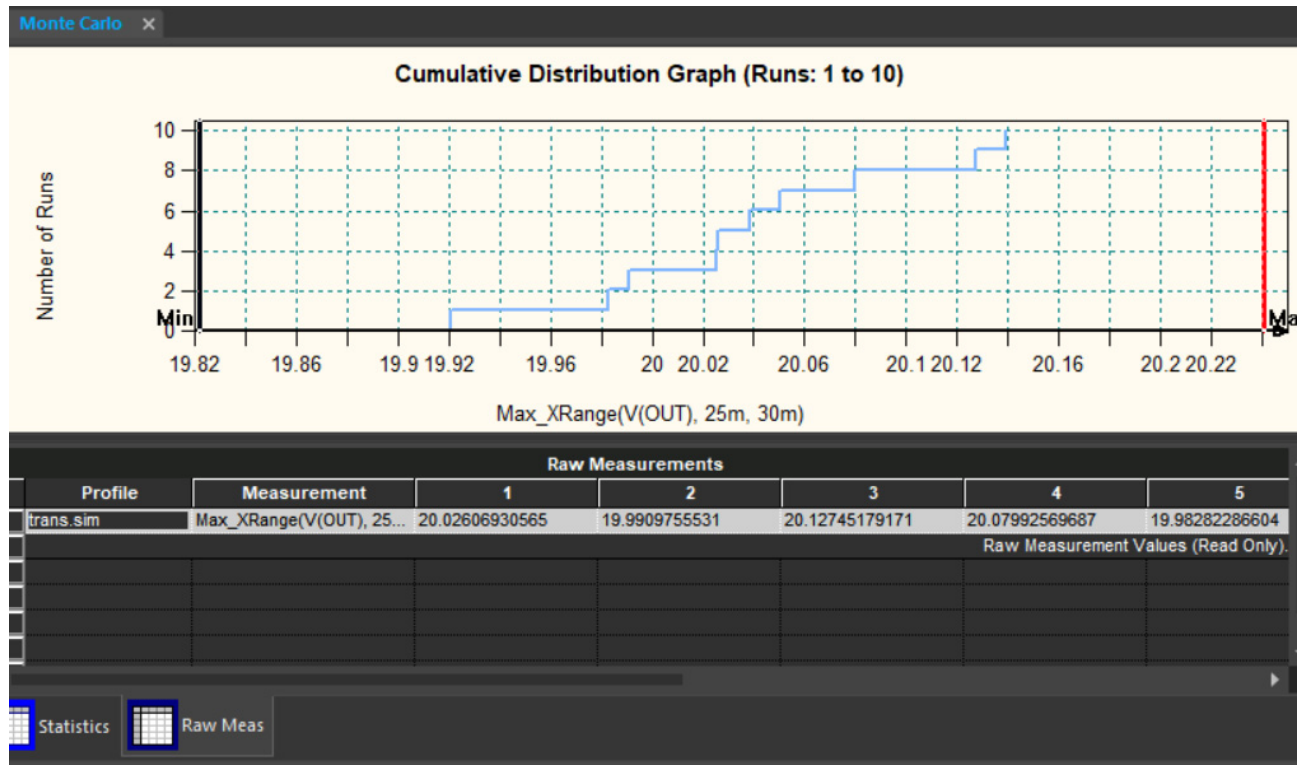


In the Probability Density Graph, every Monte Carlo Analysis run is within the Cursor Minimum value and Cursor Maximum value, which concludes that the yield of the SMPS design is 100%. The yield information is shown in the Statistical Information tab.

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.

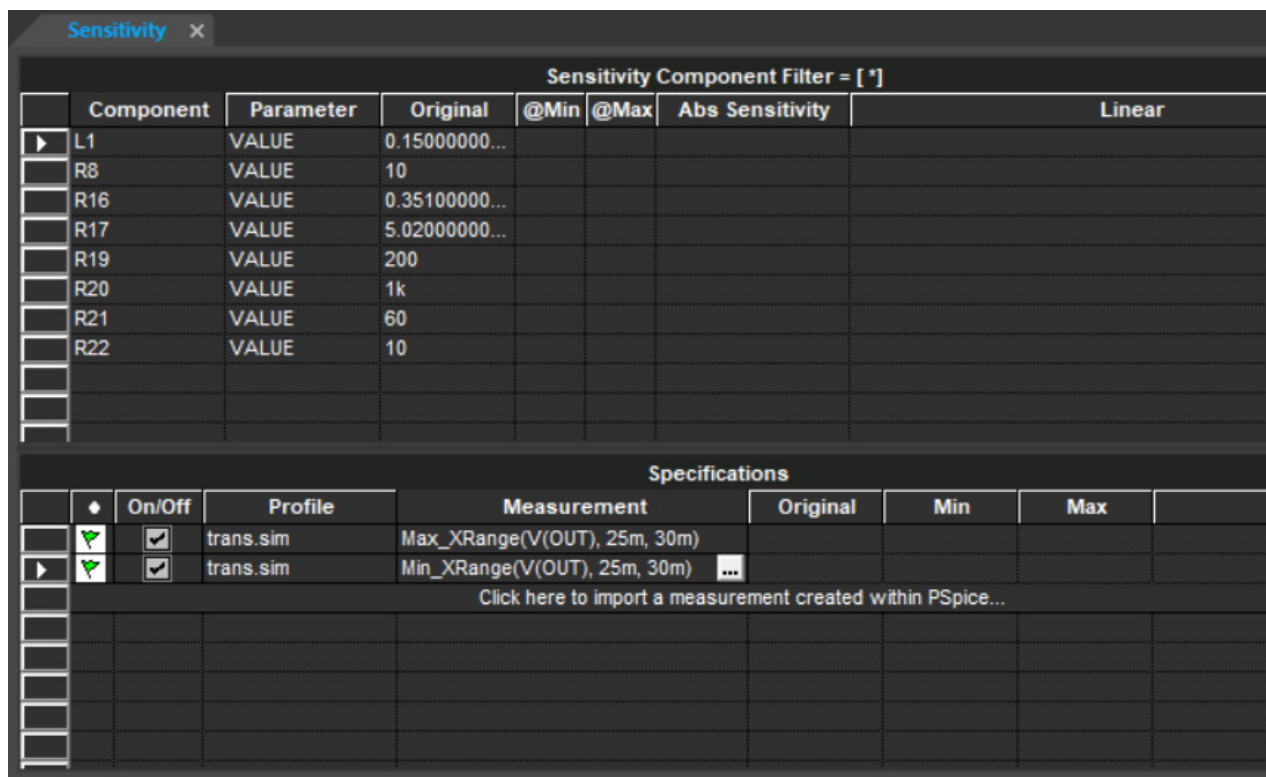


Running Sensitivity and Optimizer Analysis using PSpice Advanced Analysis

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 components 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`.



Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

5. Run Sensitivity Analysis.

Sensitivity Component Filter = [*]

Component	Parameter	Original	@Min	@Max	Abs Sensitivity	Linear
L1	VALUE	0.1500	135m	165m	5.8259	99
R16	VALUE	0.3510	315...	386...	708.2431m	12
R8	VALUE	10	9	11	110.6468m	1
R22	VALUE	10	9	11	66.4586m	1
R17	VALUE	5.0200k	4.51...	5.52...	2.9226m	< MIN >
R19	VALUE	200	180	220	4.7587m	< MIN >
R20	VALUE	1k	900	1.10...	279.7299u	< MIN >
R21	VALUE	60	54	66	11.8869m	< MIN >

Specifications

On/Off	Profile	Measurement	Original	Min	Max
<input checked="" type="checkbox"/>	trans.sim	Max_XRange(V(OUT), 25m, 30m)	20.0261	18.5694	21.4099
<input checked="" type="checkbox"/>	trans.sim	Min_XRange(V(OUT), 25m, 30m)	19.7787	18.4098	21.2886

Click here to import a measurement created within PSpice...

Output Window

```

Sensitivity run: 8 of 8 completed
Worstcase runs underway.....
Minimum run: 1 of 2 completed
Minimum run: 2 of 2 completed
Maximum run: 1 of 2 completed
Maximum run: 2 of 2 completed
Runs for simulation profile trans.sim completed
----- Sensitivity analysis completed -----
    
```

Output Window Command Window

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.

Parameters [Next Run]

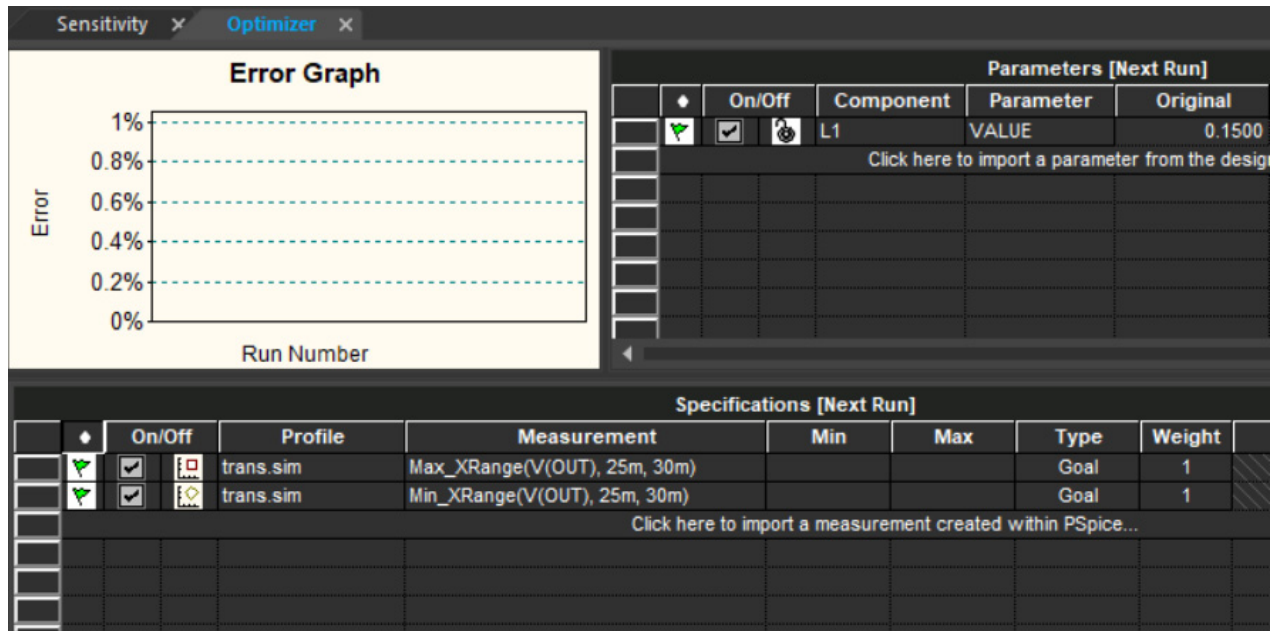
On/Off	Component	Parameter	Original	Min	Max	Current
<input checked="" type="checkbox"/>	L1	VALUE	0.1500	15m	1.5000	

Click here to import a parameter from the design property map...

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

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

Simulating an SMPS Design using Capture-PSpice Flow

Verifying Design Stability and Yield

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.