



[SIMPLIS TUTORIAL](#) [WHAT IS SIMPLIS?](#) [ADVANCED SIMPLIS TRAINING](#) [SIMPLIS PARTS](#)

[SIMPLIS ANALYSIS MODES - OVERVIEW](#) [DVM - DESIGN VERIFICATION MODULE](#)

[SYSTEMDESIGNER](#) [USER MANUAL](#) [SIMPLIS REFERENCE](#) [SIMETRIX SCRIPT MANUAL](#)

[SIMPLIS VERILOG HDL](#)

[Home](#) / [Advanced SIMPLIS Training](#) / [Module 3 - Running SIMPLIS Simulations](#) / **3.1 SIMPLIS Multi-Step Analysis**

3.1 SIMPLIS Multi-Step Analysis

The ability to step or sweep parameters enables you to quickly verify a design is working as expected. In this topic, you will setup and sweep the **RLoad** parameter and then learn how to nest a sweep of the input voltage with the output load. In this nested multi-step simulation you will verify the converter performance over both the input voltage range and the output load range.

To download the examples for Module 3, click [Module_3_Examples.zip](#)

In this topic:

- [Key Concepts](#)
- [What You Will Learn](#)
- [Getting Started](#)
- [Discussion](#)
- [How The Multi-Step Analysis Works](#)
- [Exercise #1: Verify R3 Value](#)
- [SIMPLIS Multi-Core Capability](#)
- [Exercise #2: Enable Multi-Core](#)
- [Using DVM to Make Nested Multi-Step Sweeps](#)
- [Exercise #3: Converting A Schematic to DVM](#)
- [Discussion: Exercise #3](#)
- [Multi-Step Configuration File](#)
- [DVM Testplans](#)
- [Determining The Stepped Parameter Values for a Curve](#)
- [Conclusions and Key Points to Remember](#)

Key Concepts

Advanced SIMPLIS Training

[Course Outline](#)

[Installing the Training Course License](#)

[Getting Started](#)

[Module 1 - Overview of the SIMPLIS Environment](#)

[Module 2 - Advanced SIMPLIS](#)

[Module 3 - Running SIMPLIS Simulations](#)

[3.0 A Look Under the SIMPLIS Hood](#)

[3.1 SIMPLIS Multi-Step Analysis](#)

[3.2 SIMPLIS Monte Carlo Analysis](#)

[3.3 Loading a Schematic with Component Values](#)

[Module 4 - Introduction to](#)

This topic addresses the following key concepts:

- Design parameters can be swept with a linear or decade sweep, or over an explicit range of values.
- Multiple processor cores can be used to reduce the time required to execute a Multi-Step simulation.
- Multiple parameters can be stepped in nested sweeps by using DVM.

What You Will Learn

In this topic, you will learn the following:

- How a parameter can be stepped through a list of values, with one simulation executed per value.
- How parameters can swept over a range of values using a linear or decade sweep to determine the step size.
- How to setup and run multi-step simulations.
- How to use multiple processor cores to reduce the time required to execute a Multi-Step simulation.

Getting Started

1. Close the waveform viewer if it is open.
2. Open the schematic **3.1_SelfOscillatingConverter_POP.sxsch**.
3. From the schematic menu, select **Simulator > Setup Multi-step...**

Result: The Define SIMPLIS Multi-Step Analysis dialog opens:

Define SIMPLIS Multi-Step Analysis

Parameter name
Enter the parameter name in the box below. For example, to sweep a resistor with parameter value {Rload}, enter Rload in the box below.
Parameter name: RLoad

Step parameters
Start value: 1k
Stop value: 2k
Number of steps: 10

Step type
☐ Decade ☐ Group digital curves
☐ Linear
☒ List **Define List...**

Multi-core
Number of cores: 1 ☐ Show console for each process
Number of physical cores: 4 Number of cores allowed by license: 16

Options
☐ Save state If checked, the simulator state will be saved for each step allowing snapshots and initial conditions to be applied to subsequent multi-step runs

Run Ok Cancel Help

4. The dialog is already prepared to step the parameter **RLoad** through a list of values. To view the list of values, click on the **Define List...** button.

Result: The Define List dialog opens, displaying the three RLoad step values: 5, 2.5, and 4.2.

[Modeling](#)

[Module 5 -
Parameterization](#)

[Module 6 - Modeling](#)

[Applications](#)

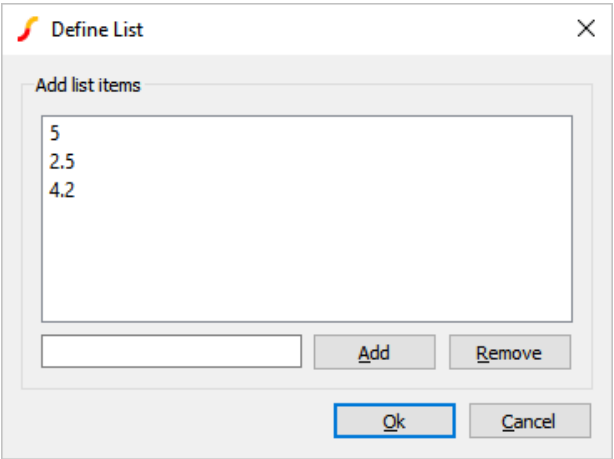
[Handout A: Useful
Keyboard Shortcut
Keys](#)

[Handout B: Training
Keyboard Shortcut
Keys](#)

[Handout C:
Schematic Tips and
Tricks](#)

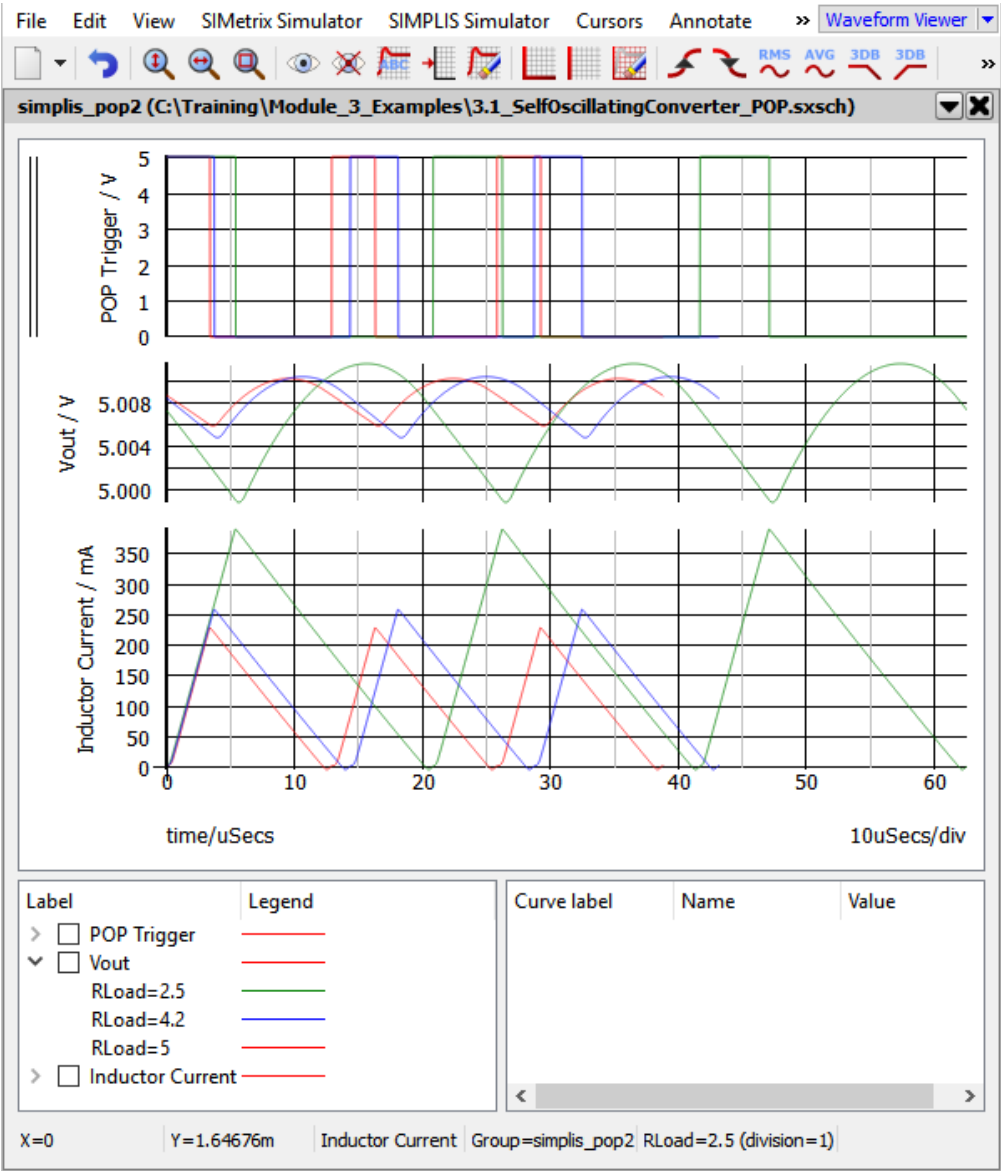
[Handout D: SIMPLIS
PWL R, L, C](#)

[Handout E: SIMPLIS
VPWL and IPWL
Resistors](#)



- 5. Click **Ok** on the Define List dialog.
- 6. Click **Run** on the Define SIMPLIS Multi-Step Analysis dialog.

***Result:** The Multi-Step simulation runs and the periodic operating point waveforms for each RLoad value are displayed on the waveform viewer: Note that the converter switching frequency, peak inductor current, and output ripple changes for each simulated RLoad value.*



Discussion

The SIMPLIS Multi-Step Analysis allows you to quickly verify how a design responds to parameter changes. In the next topic, [3.2 SIMPLIS Monte Carlo Analysis](#), you will learn how the SIMPLIS Monte Carlo Analysis allows you to vary multiple design parameters with statistical distributions.

In this example, three **RLoad** values which couldn't be defined as a linear or decade sweep were chosen. The List sweep option allows you to choose any list of values. The SIMPLIS Multi-Step Analysis allows you to sweep parameters over linear or decade steps as well. More information on the different sweep types can be found in the help topic linked to by the Help button on the Define SIMPLIS Multi-Step Analysis dialog.

How The Multi-Step Analysis Works

You may recall from the topic [3.0.1 What Happens When You Press F9?](#) that the **RLoad** variable is set in the F11 window:

```
.var RLoad=2.5
```

When you ran the Multi-Step Analysis, the **RLoad** value defined in the F11 window was overwritten by the step value defined in the Define SIMPLIS Multi-Step Analysis dialog. For each step value, the netlist preprocessor:

- Includes the stepped parameter value, in this case **RLoad**.
- Pre-processes the netlist
- Launches the SIMPLIS simulator on the resulting deck.

In this example, three individual deck files were created, and each new deck file over-writes the previous deck file. This example steps the **RLoad** parameter over three values in the order - **5** , **2.5**, **4.2**. In the last deck file, **R3** will have a value of 4.2Ω. In the next exercise you will verify the last value of R3 is 4.2Ω.

Exercise #1: Verify R3 Value

1. From the menu bar, select **Simulator > Edit Netlist (after preprocess)** .
Result: The Deck file opens in the netlist editor.
2. To search for the text **4.2**, follow these steps:
 - a. Use the shortcut key **Ctrl+F** to open the search dialog.
 - b. Type **4.2**
Result: the deck file scrolls to line 110 where the value of R3 is 4.2 Ω.

```
109 R2 35 0 1.5
110 R3 41 0 4.2
111 R2 0 28 150
```

SIMPLIS Multi-Core Capability

The SIMetrix/SIMPLIS Pro and Elite licenses can use multiple processor cores to simulate the stepped parameter values. Using multiple cores allows multiple step values to be simultaneously simulated. The number of cores allowed by the licenses are:

License	Number of Physical Cores
Pro	4
Elite	16

In addition to the maximum number of cores allowed by your license, you are obviously limited by the number of cores your computer has. For this limitation, SIMetrix/SIMPLIS only uses the physical cores, not the hyper-

threaded cores. In the next exercise you will run a multi-core, multi-step simulation.

Exercise #2: Enable Multi-Core

This exercise assumes your computer has more than one physical core and that you are using a SIMetrix/SIMPLIS Pro license.

1. Close the waveform viewer.
2. From the schematic menu, select **Simulator > Setup Multi-step...**

Result: The Define SIMPLIS Multi-Step Analysis dialog opens:

Define SIMPLIS Multi-Step Analysis

Parameter name
Enter the parameter name in the box below. For example, to sweep a resistor with parameter value **{Rload}**, enter **Rload** in the box below.
Parameter name:

Step parameters
Start value:
Stop value:
Number of steps:

Step type
☐ Decade ☐ Group digital curves
☐ Linear
☒ List

Multi-core
 Number of cores ☐ Show console for each process
Number of physical cores: 4 Number of cores allowed by license: 16

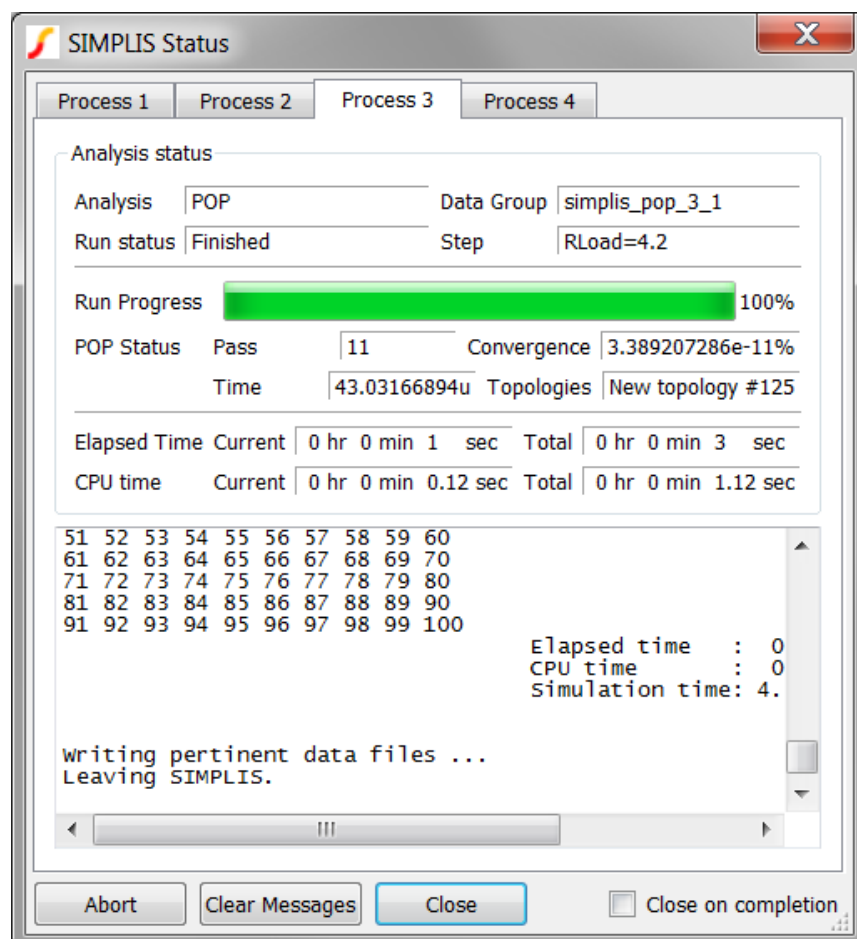
Options
☐ Save state If checked, the simulator state will be saved for each step allowing snapshots and initial conditions to be applied to subsequent multi-step runs

3. Change the **Number of cores** to **4**, the maximum allowed by the Pro license.

Note: If your machine has either a single core or two cores, the dialog will limit your **Number of cores** entry. If you have a single core, you can skip this exercise.

4. Click **Run**.

Result: The Multi-Step simulation runs on the maximum number of physical cores. Assuming you have a 4 Core machine, all 3 step values are executed simultaneously using 3 SIMPLIS processes, one per processor core. The SIMPLIS Status window has four tabs, one for each SIMPLIS process. The fourth tab is empty as no step value was run on this process:



In this exercise, a very small circuit was simulated, yet you probably noticed a significant speed improvement in the time required to execute the Multi-Step Analysis. As circuit size grows, this effect becomes dramatic, and in experiments, speed improvements of approximately 3.5 times when using 4 cores vs. a single core have been found.

Using DVM to Make Nested Multi-Step Sweeps

The Design Verification Module (DVM) makes running multi-step simulations on multiple parameters very easy. In this exercise you will create and run a DVM testplan which runs a nested parameter multi-step simulation sweeping two variables, each having 3 steps.

Exercise #3: Converting A Schematic to DVM

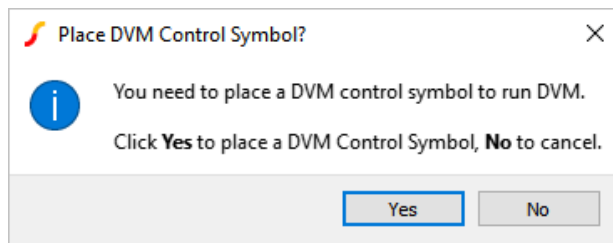
The DVM schematic

3.2_SelfOscillatingConverter_POP_DVM_basic.sxsch is electrically identical to the schematic used in the earlier example. The only difference between this schematic and **3.1_SelfOscillatingConverter_POP.sxsch** is **fixed probe measurements** have been added to the probes. DVM automatically makes these measurements on each parameter step and outputs the measurements to the DVM report.

To run this example circuit, follow these steps:

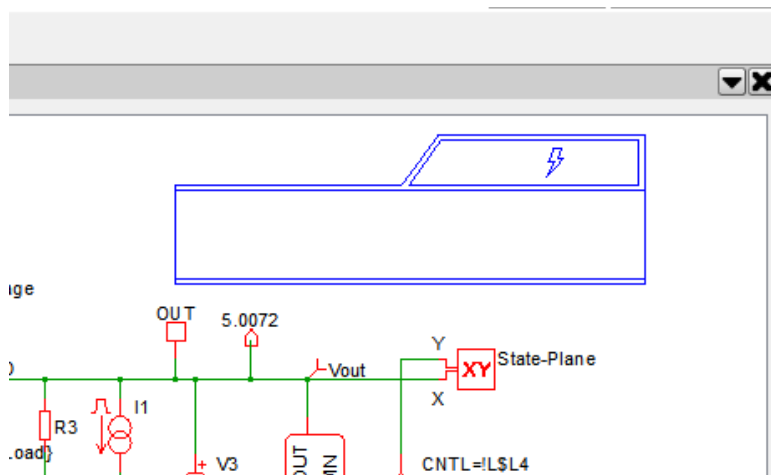
1. Open schematic
3.2_SelfOscillatingConverter_POP_DVM_basic.sxsch.
2. From the menu bar, select **DVM > Built-In Testplans > Multi-Step > Run Multi-Step Using DVM Control Symbol Parameters**.

Result: A DVM control symbol is necessary for any DVM simulation.
A dialog asking you to place a control symbol will appear:



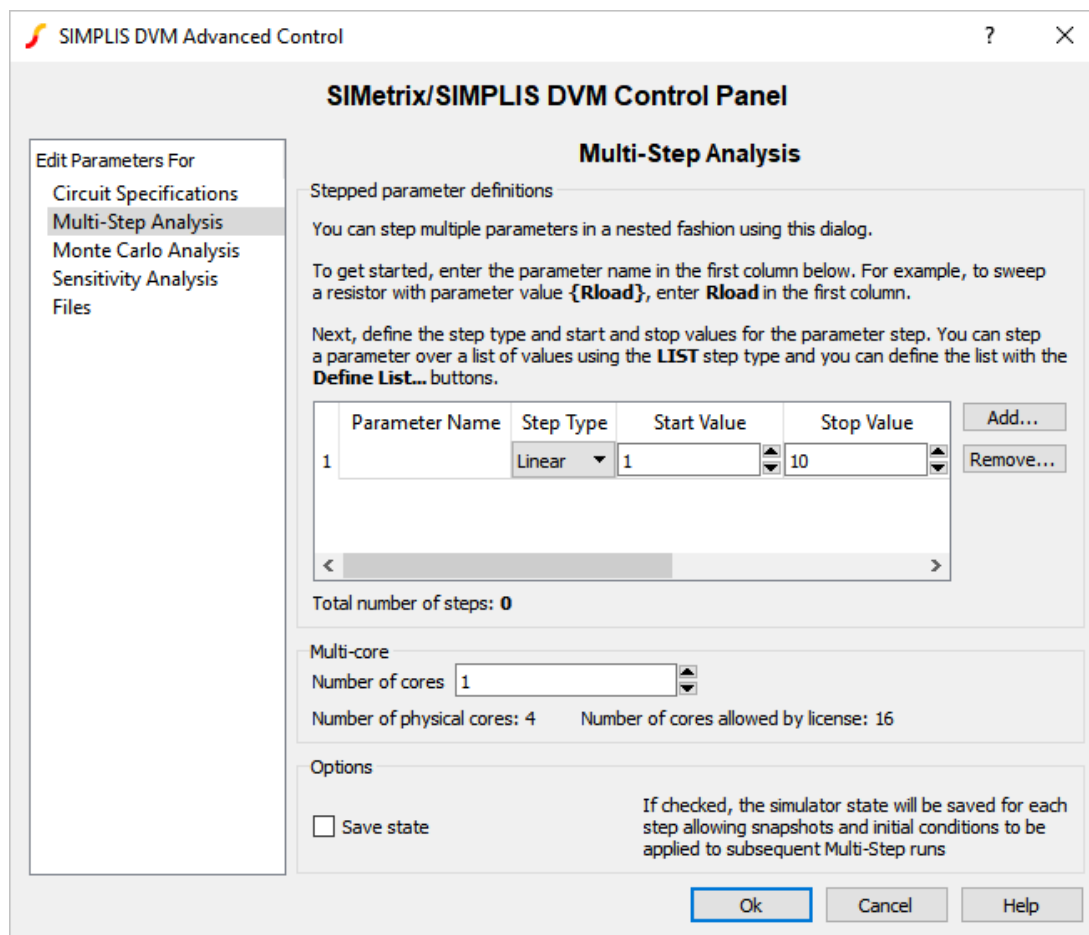
3. Click **Yes**.

Result: Focus returns to the schematic viewer with a DVM control symbol attached to your mouse.



4. Place the DVM control symbol on the schematic.

Result: The DVM control symbol stores specification information for the schematic, including the stepped parameters. While the DVM control symbol has several forms, the one used in this example is the most basic version. The control symbol will be opened to the Multi-Step tab, where you will need to enter your multi-step parameter values:



5. Fill in the first parameter to match what you simulated in Exercise #2:

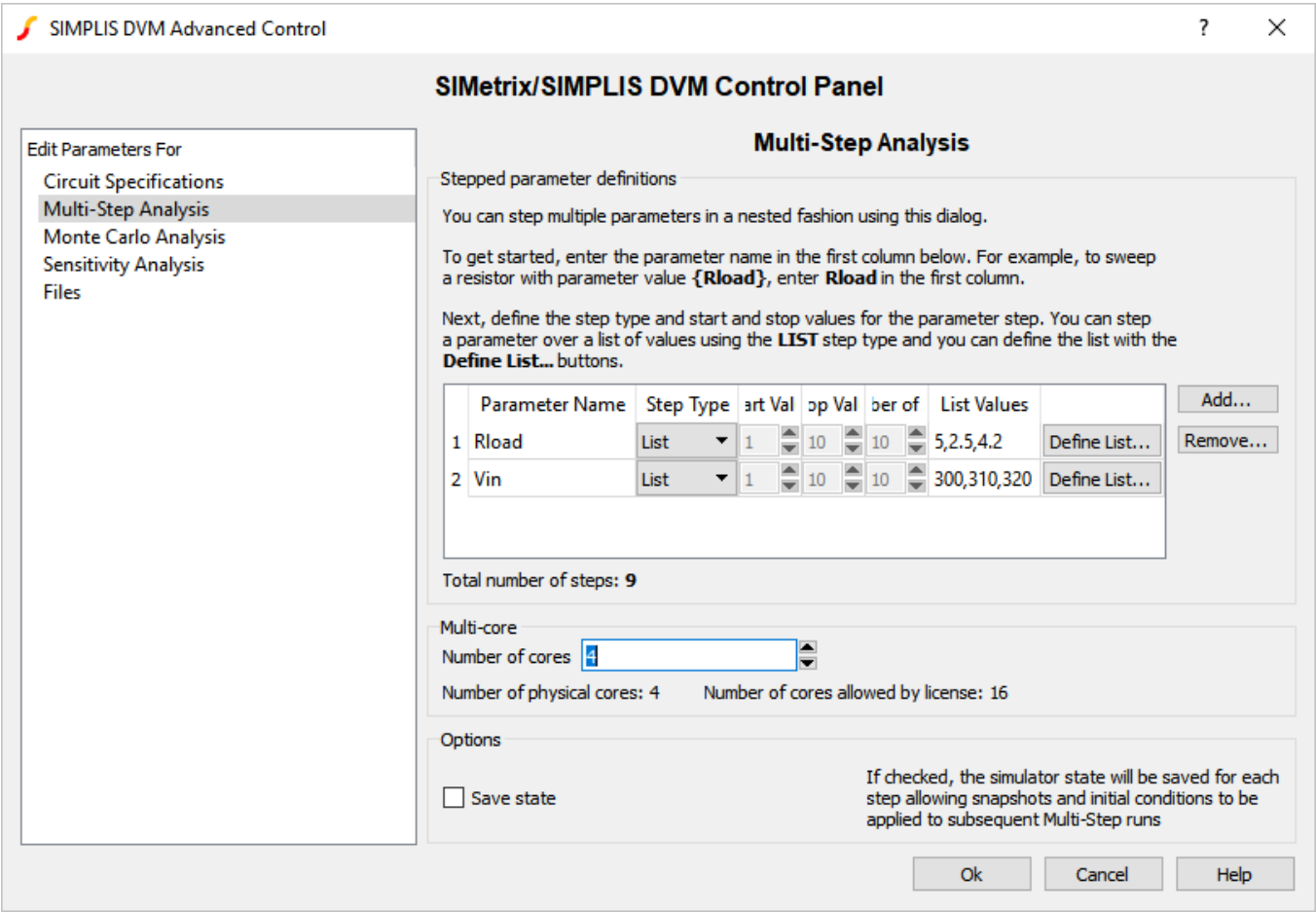
- Parameter Name - **Rload**
- Step Type - **List**
- List Values - **5, 2.5, 4.2**

6. Click **Add...** to add a second parameter:

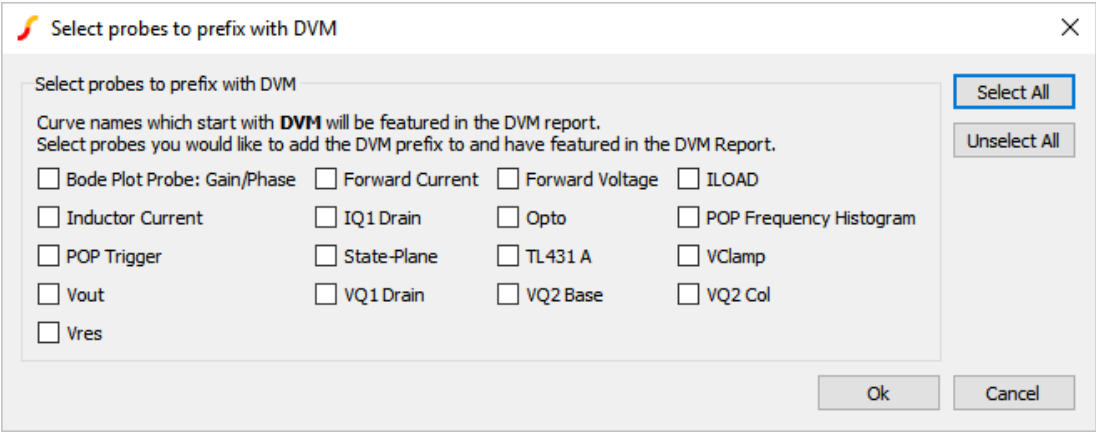
- Parameter Name - **Vin**
- Step Type - **List**
- List Values - **300, 310, 320**

7. Increase the **Number of cores** to **4**.

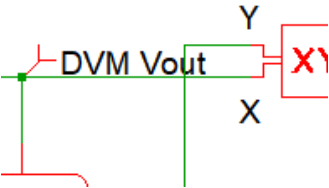
Result: The control symbol Multi-Step should look like the following:



8. Click **Ok**.
***Result:** Probes whose names start with **DVM** will be output to the DVM report. Since there are none with a **DVM** prefix, a prompt to rename probes will appear:*

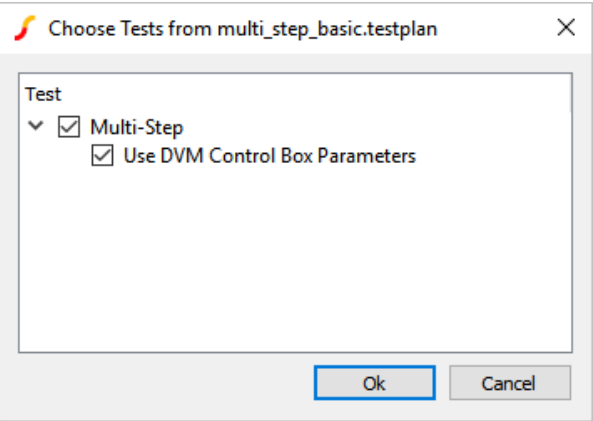


9. Select the **Inductor Current**, **POP Trigger**, and **Vout** probes and click **Ok**.
***Result:** The selected probes will be renamed on the schematic and a description of changes made will appear on the command shell:*



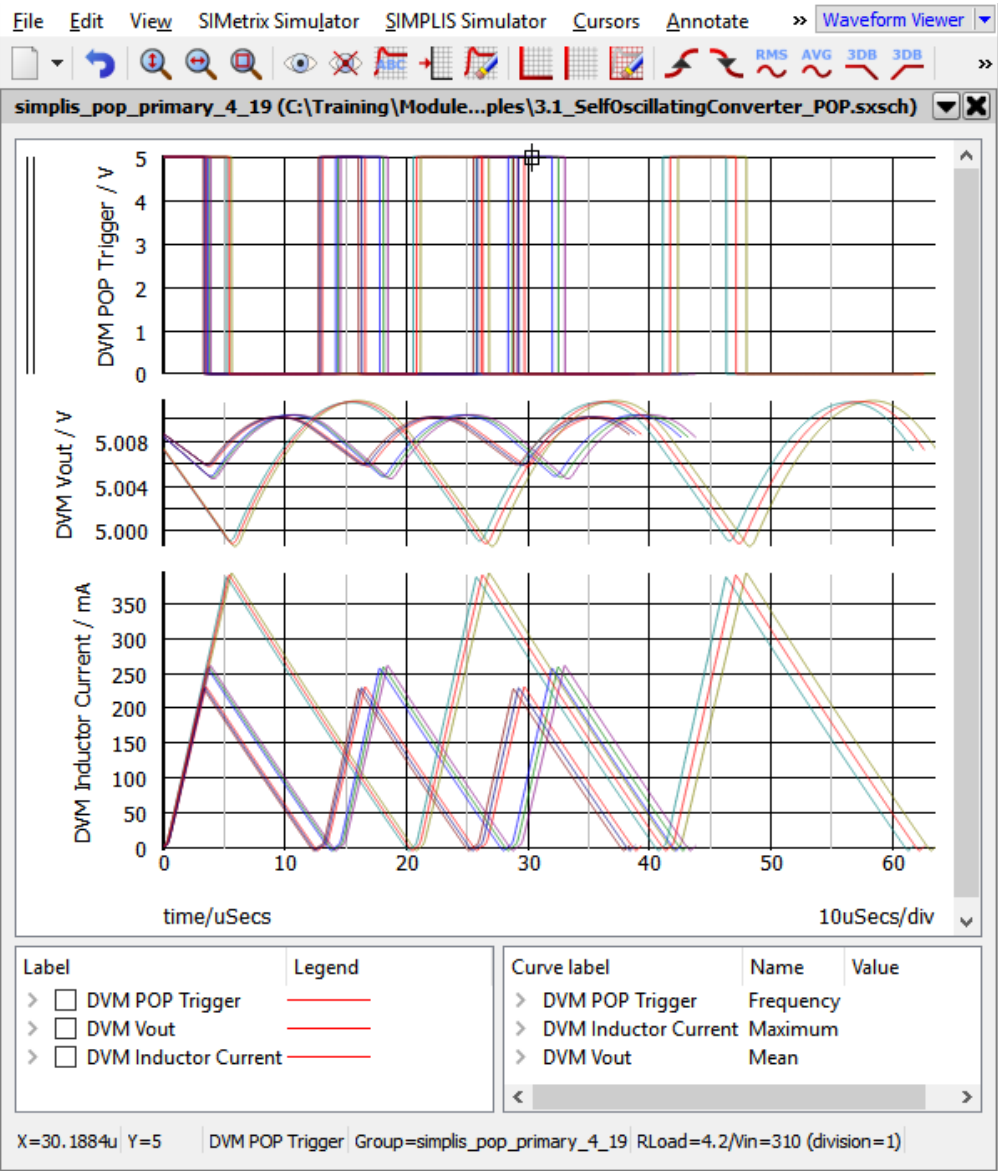
```
Renamed probe (Inductor Current) to (DVM Inductor Current)
Renamed probe (POP Trigger) to (DVM POP Trigger)
Renamed probe (Vout) to (DVM Vout)
```

10. Select the only test in this testplan:



11. Click **Ok**.

Result: DVM creates the multi-step configuration file from the control symbol-defined parameters (will overwrite any existing configuration file), and outputs the following curves before opening the report.



The real power of DVM lies in the HTML-formatted report generated after each test suite runs. DVM automatically makes each probe measurement on each simulation step, and outputs the data to the report. In the report,

the stepped parameter values and the measurements are shown in a single table:

MULTI-STEP RESULTS & PARAMS						
	Rload	Vin		Avg(Vout)	Frequency(POP Trigger)	Max(Inductor Current)
MIN	2.5	300	MIN	5.0065859	47.32187k	226.30357m
MAX	5	320	MAX	5.0084307	78.538875k	391.31527m

Run #	Rload	Vin	Run #	Avg(Vout)	Frequency(POP Trigger)	Max(Inductor Current)
1	5	300	1	5.0083576	76.543k	229.22747m
2	2.5	310	2	5.0066451	48.058526k	388.24584m
3	4.2	320	3	5.0081415	70.639311k	255.85801m
4	2.5	300	4	5.0065859	47.32187k	391.31527m
5	2.5	320	5	5.0067022	48.762278k	385.38021m
6	4.2	300	6	5.0080592	68.745913k	259.36442m
7	4.2	310	7	5.0081012	69.716097k	257.5494m
8	5	310	8	5.0083949	77.566748k	227.71321m
9	5	320	9	5.0084307	78.538875k	226.30357m

Discussion: Exercise #3

There are a few items that should be pointed out about the schematic in it's current state.

Multi-Step Configuration File

Because the Multi-Step Configuration file is created in the process described in Exercise #3, a non-DVM multi-step simulation can be conducted by clicking menu item **Simulator > Run Multi-step** .

DVM Testplans

DVM runs tests defined in a testplan file. Testplans are merely tab separated text files where each row represents a test to be executed, and each column performs an action during the test. To run a DVM multi-step simulation, you enter Multi-Step in the Analysis column. The Multi-Step analysis keyword tells DVM to use the parameter definitions stored on the DVM Control Symbol. In this example, the POP analysis parameters were previously defined with the **Simulator > Choose Analysis...** menu. DVM can also change the analysis parameters on a test-by-test basis. The testplan you just ran is shown below. It has one test which runs the multi-step simulation with the parameters you defined on the DVM control symbol.

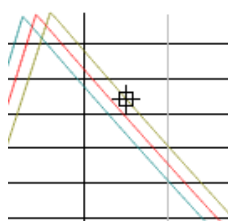
```
***
*** multi_step_basic.testplan
***
*?@ Analysis
***
Multi-Step
*** Documentation: http://www.simplis.com/documenta
```

Determining The Stepped Parameter Values for a Curve

The default SIMetrix/SIMPLIS configuration has eight waveform colors. Once the number of steps exceeds 8, the colors repeat, and some curves will have the same color, making it difficult to determine which is which. To

determine which curve corresponds to which stepped parameter:

1. Open the waveform tab containing the multi-step waveforms.
2. Hover your mouse over the target curve until you see the square box:



3. Look at the far right side of the status bar:

Curve label	Name	Value
> DVM POP Trigger (si...	Frequency	
> DVM Inductor Curre...	Maximum	
> DVM Vout (simplis_...	Mean	

0) Group=simplis_pop_primary_4_20	RLoad=2.5/Vin=300 (division=3)
-----------------------------------	--------------------------------

Conclusions and Key Points to Remember

- The SIMPLIS Multi-Step analysis allows you to step a parameter through a list, or to sweep a parameter with linear or decade steps to evaluate the circuit performance.
- Multi-Core capability greatly reduces the time required to execute Multi-Step simulations.
- DVM allows you to quickly and easily step parameter values and collect probe measurements from each simulation step.