Design Optimization with Curve-Fit
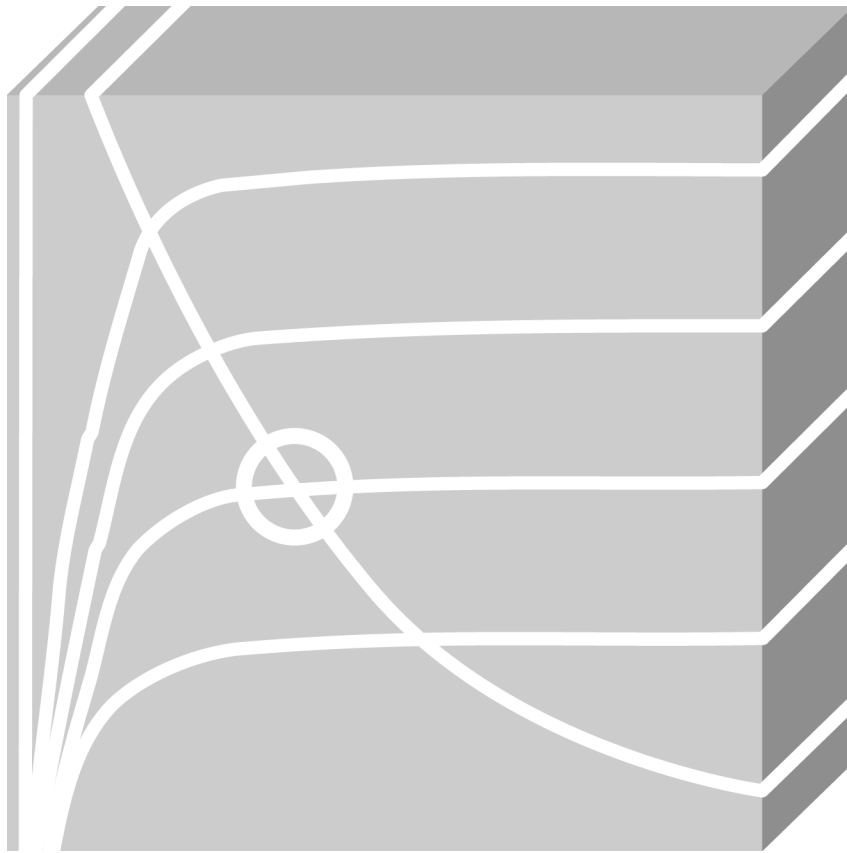
Table of Contents

1   Introduction

This document gives an overview how a user can use the option Optimizer of the advanced analysis package to optimize a circuit. It is also explained how optimizer works and what can be configured.

There are many times, that your designed circuit behaves as you expect, but not as precise as you need. In order to increase such accuracy, you can order PSpice to automatically calculate the value of different components to reach such accuracy. This information is given though reference files, where the value of the curves are specified.

There are other scenarios, where Optimizer is very powerful and useful:

- Increase the accuracy of PSpice Models.
- Calculate the value of components for new specifications. It means, that you don't have to calculate those new values manually, but PSpice calculate them for you.

For such calculations, it is possible to use curve-fit (as the example below) or you can use measurements (goals / specifications for your circuit).

2   Optimizer

Optimizer is a design tool for optimizing analog circuits and their behaviour. It helps you modify and optimize analog design to meet your performance goals. For your understanding, optimization is the process of fine-tuning a design by varying user-defined design parameters between successive simulations until performance comes close to the ideal one.

2.1   How to Use Optimizer?

Setting up the Optimizer consists of the following tasks:

- Open Optimizer in Advanced Analysis
- Select an engine
- Define Optimization parameters
- Set up circuit measurement or specifications (or curve-fitting)
- Run and let it to calculate the new value for the parameters

2.2   Which Engine Do You Have to Select?

Optimizer supports multiple engines:

- Modified LSQ Engine: to rapidly converge on an optimum solution
- Random engine: to pick up a starting point that avoids getting stuck in local minima when there is a problem converging
- Discrete engine: to pick up commercially available component values and run the simulation one more time with the selected commercial values

The normal flow in which these engines are used is Random Engine, followed by MLSQ Engine and finally the Discrete Engine.

In order to understand deeper the whole features that Optimizer supports, use the PSpice Advanced Analysis User Guide (page 83-147).

3 Design Example

This design is a two-pole multiple feedback active bandpass filter. The curve-fit specification is used to reach the desired response of the filter.
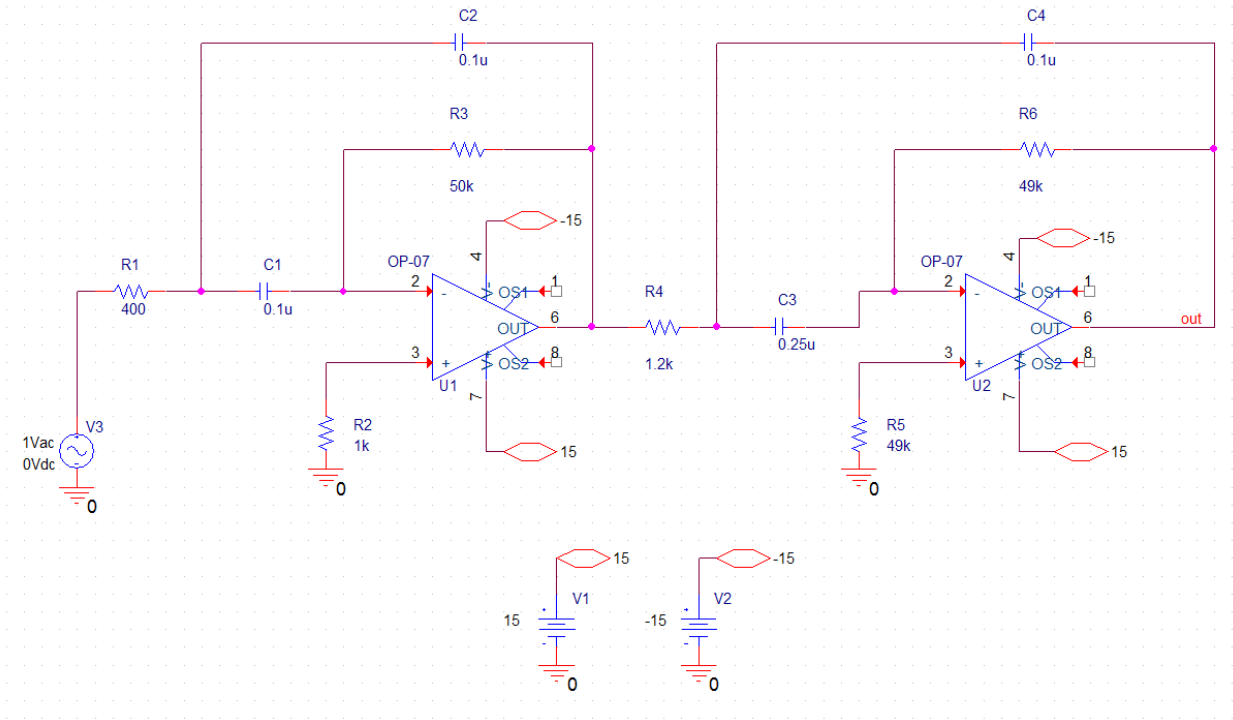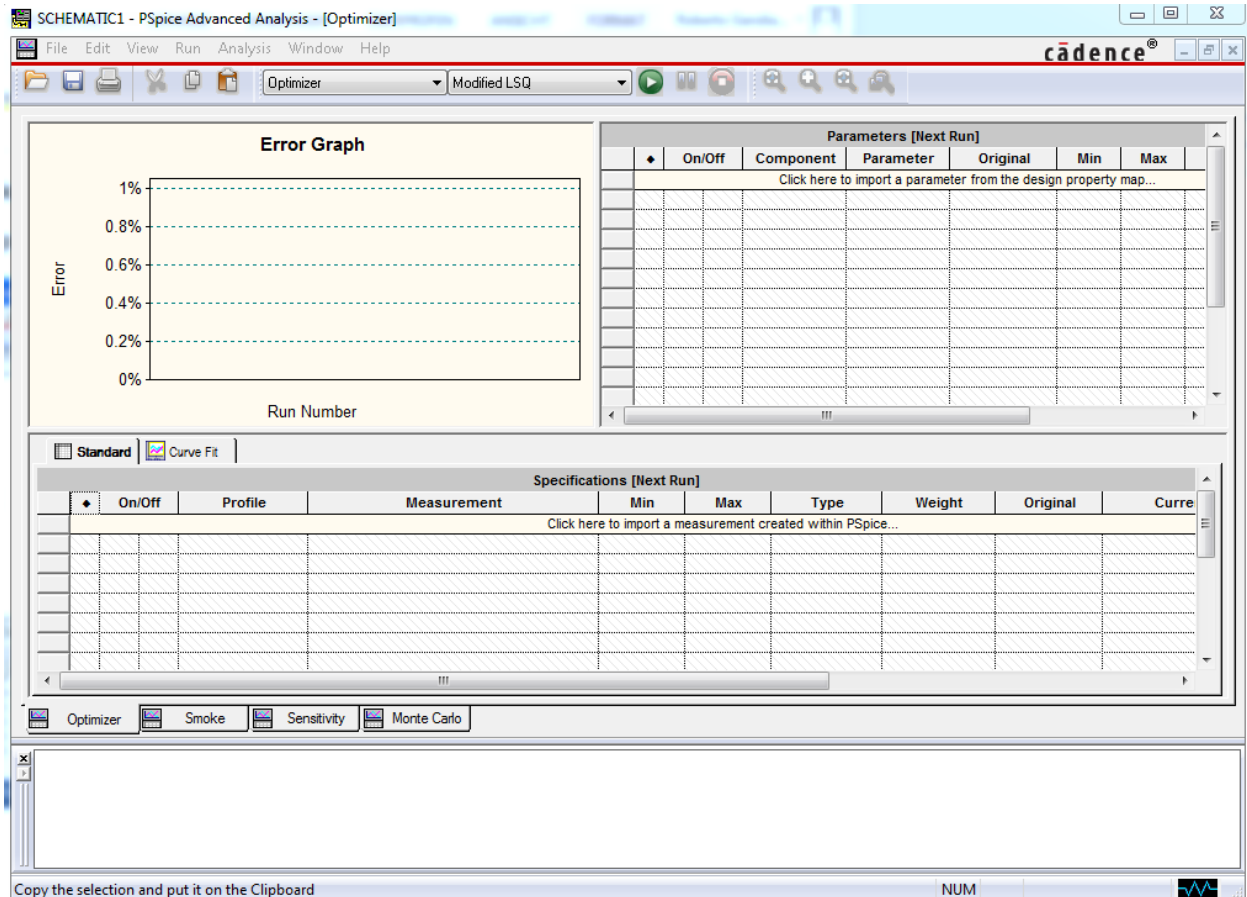

PSpice AD

Simulate this circuit in the frequency and add in the Probe Window two traces, the phase and the amplitude of the output voltage P(V(out)) and DB(V(out)). Use two Y-axis to visualize these curves appropriately.
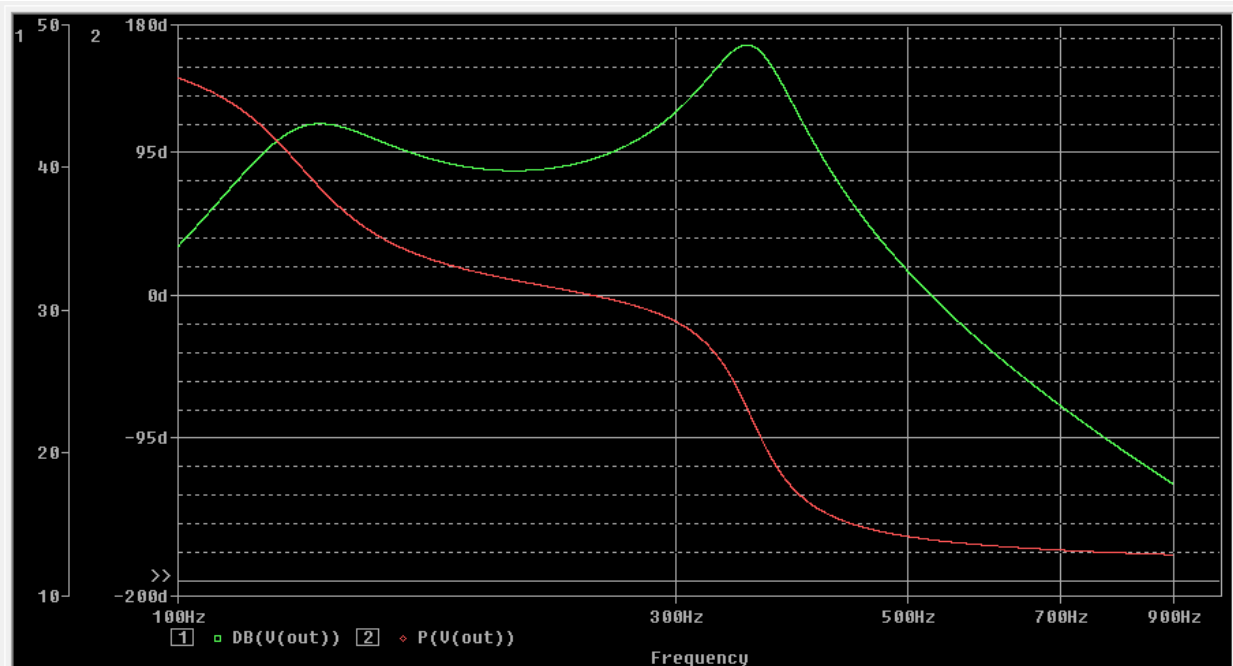
PSpice AA

To open Optimizer, click on **PSpice > Advanced Analysis > Optimizer**. Next window pops up:

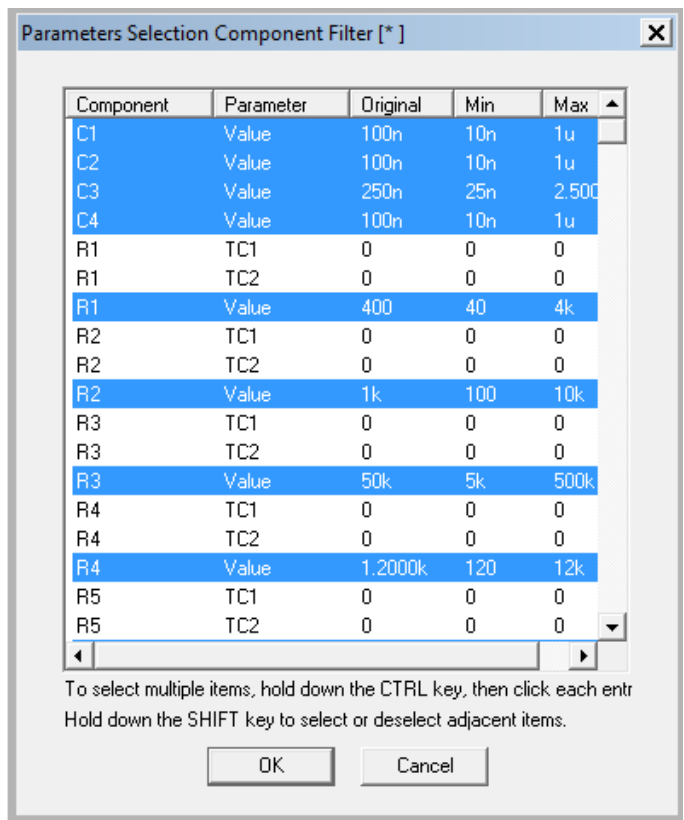The steps you have to follow to calculate the new component values are:

1. Select the engine you want to use for the calculation.
    –   **Modified LSQ Engine:** it uses both constrained and unconstrained minimization algorithms, which allows to optimize goals subject to nonlinear constraints.
    –   **Random Engine:** it applies a grid to the design space and randomly runs analysis at the grid points. It keeps track of the grid points already run so that it never runs a duplicate set of parameter values. Once it finishes its initial analysis, it reruns the best points so you can easily use them for Modified LSQ.
    –   **Discrete Engine:** it finds the nearest commercially available value for a component. The other engines calculate component values, but those values might not be commercially available.

    It is a conceptual engine, rather than a true engine in that it does not actually perform an optimization, it finds available values from lists. Once a value is selected, the engine makes a final run that lets you review the results in both the Optimizer and the output tools. If the results of the discrete analysis are not acceptable, the design can be optimized again to find another global minimum that might be less sensitive.

2. Specify component parameters.

   – Click on **click here to import a parameter form the design property map …**

	◆	On/Off	Component	Parameter	Original	Min	Max	Current
				Click here to import a parameter from the design property map...				

Parameters [Next Run]

   – In Parameters selection dialog window select all the capacitors and resistors, then click on **OK**.

Parameters Selection Component Filter [*]

Component	Parameter	Original	Min	Max
C1	Value	100n	10n	1u
C2	Value	100n	10n	1u
C3	Value	250n	25n	2.500
C4	Value	100n	10n	1u
R1	TC1	0	0	0
R1	TC2	0	0	0
R1	Value	400	40	4k
R2	TC1	0	0	0
R2	TC2	0	0	0
R2	Value	1k	100	10k
R3	TC1	0	0	0
R3	TC2	0	0	0
R3	Value	50k	5k	500k
R4	TC1	0	0	0
R4	TC2	0	0	0
R4	Value	1.2000k	120	12k
R5	TC1	0	0	0
R5	TC2	0	0	0

To select multiple items, hold down the CTRL key, then click each entr
Hold down the SHIFT key to select or deselect adjacent items.

OK   Cancel

– The original value, minimum and maximum are showed in next window:

	◆	On/Off		Component	Parameter	Original	Min	Max	Current
	▽	✔	🐞	R6	Value	49k	4.9000k	490k	
	▽	✔	🐞	R5	Value	49k	4.9000k	490k	
	▽	✔	🐞	R4	Value	1.2000k	120	12k	
▶	▽		🐞	R3	Value	50k	5k	500k	
	▽	✔	🐞	R2	Value	1k	100	10k	
	▽	✔	🐞	R1	Value	400	40	4k	
	▽	✔	🐞	C4	Value	0.1000u	10n	1u	
	▽	✔	🐞	C3	Value	0.2500u	25n	2.5000u	
	▽	✔	🐞	C2	Value	0.1000u	10n	1u	
	▽	✔	🐞	C1	Value	0.1000u	10n	1u	
				Click here to import a parameter from the design property map...					

Parameters [Next Run]

Minimum = original/10
Maximum = original*10

You can activate or deactivate a value for the analysis or even lock it to not change.

3. Add curve-fit specification
   – Select Curve-Fit in Optimizer window.

Standard	Curve Fit						
			Curve Fit [Next Run]				
On/Off	Profile	Trace Expression	Reference File	Ref. Waveform	Tolerance %	Weight	Error
			Click here to enter a curve-fit specification...				

– Click on **Click here to enter a curve-fit specification …,** the window New Trace Expression appears. Add P(V(out)) and DB(V(out)).

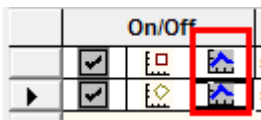| Standard	Curve Fit							
				Curve Fit [Next Run]				
On/Off		Profile	Trace Expression	Reference File	Ref. Waveform	Tolerance %	Weight	Error
✔	🔲	sim.sim	DB(V(out))				5	1
✔	◇	sim.sim	P(V(out))				5	1
			Click here to enter a curve-fit specification...					

– Select the Reference File where the curves are specified. This file should be saved as text file and follow this structure:

– In Ref.Waveform select the Phase and define a Tolerance of 5% and 1% for Weight.

- Weight: It is the importance of the specification. Change the number in the weight column if you want to emphasize the importance of one specification with respect to another.

- Tolerance: This value is used to calculate the error of the calculated parameters so that it is below the specified tolerance.

– The same for the amplitude in DB.

– Also select the dynamic viewing. It means, that in each iteration, you will see in PSpice how the curve looks like in comparison with the reference curve.

– After the configuration, optimizer should look like the image:

4. Run Optimizer

– After the run 10 it is possible to see, that the error is already 0% and the current values of the components are listed in the window.
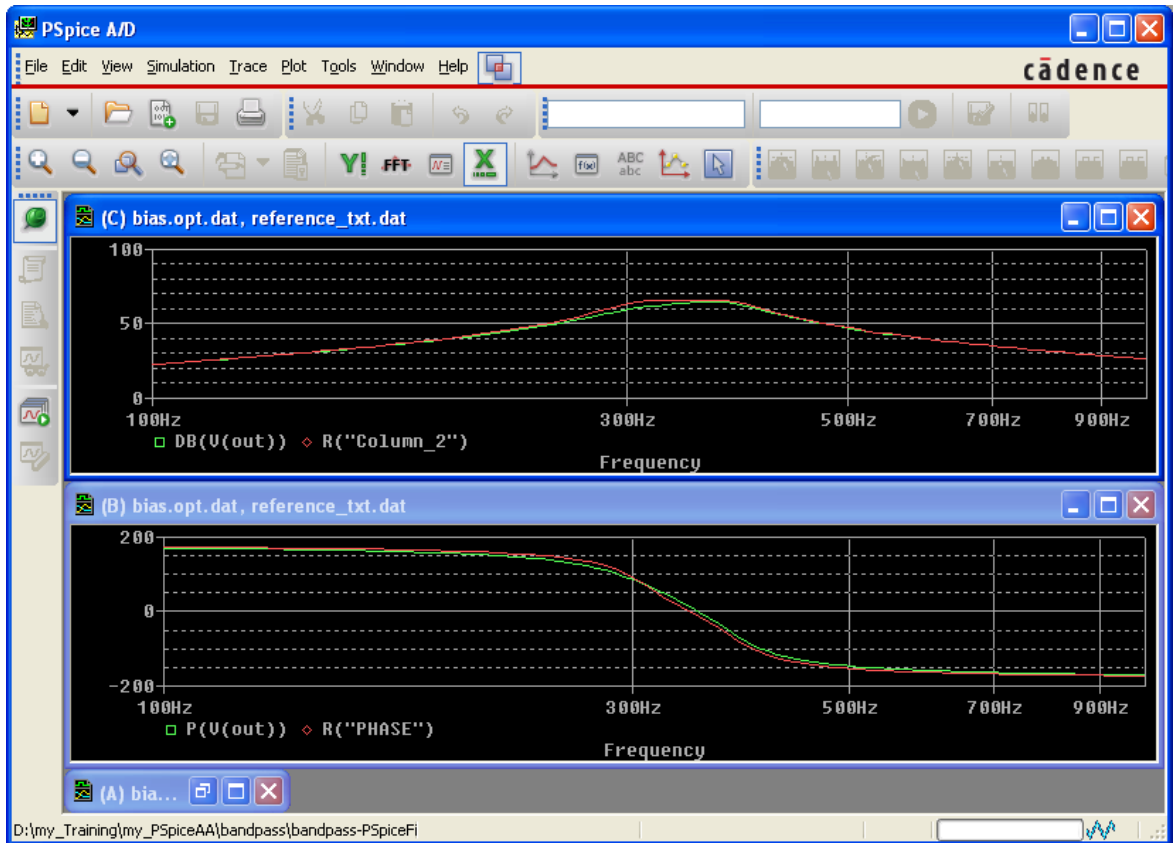
- In the probe window you can also see process of change of the curves. They agree quite well in comparison with the specifications.

 In case you want to see another run, click for example on Run 4 in **Error graph**.
  For that, select the specification for P(V(out)) > RMB click > View [Run#4].
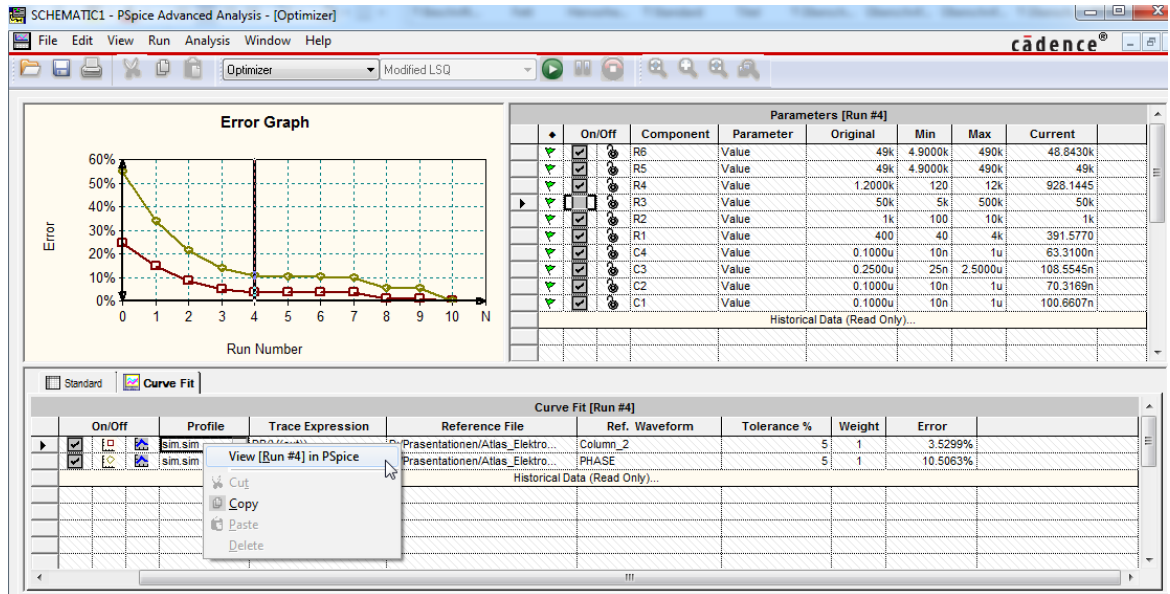
Finally, with help of the discrete engine you can find real values for the components.
Open the Profile Settings for Optimizer and select Discrete Engine:
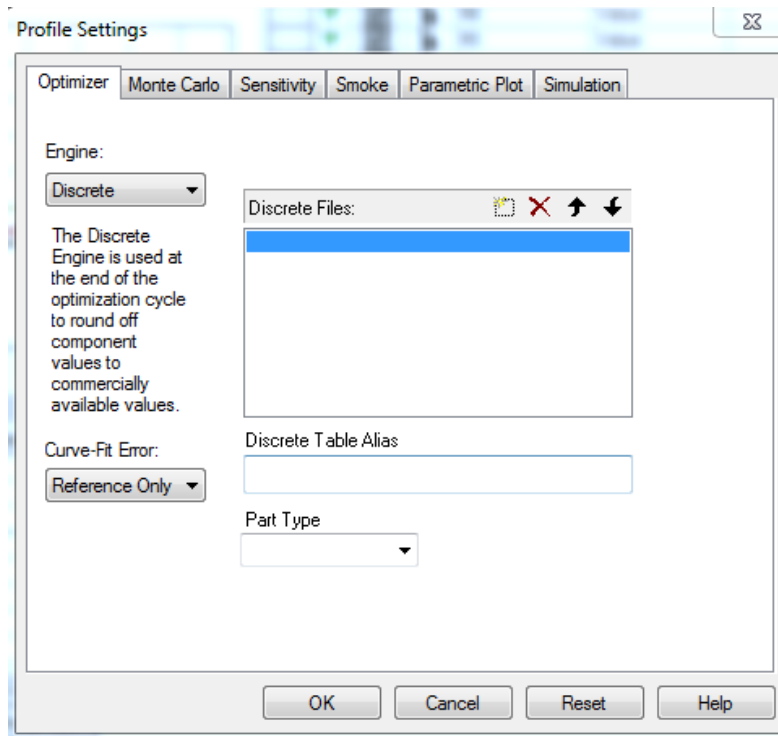
Before simulating, select under **Discrete Table** the value of the component and its tolerances you want the tool to take into account. Simulate again. After the simulation, real values are listed under current:
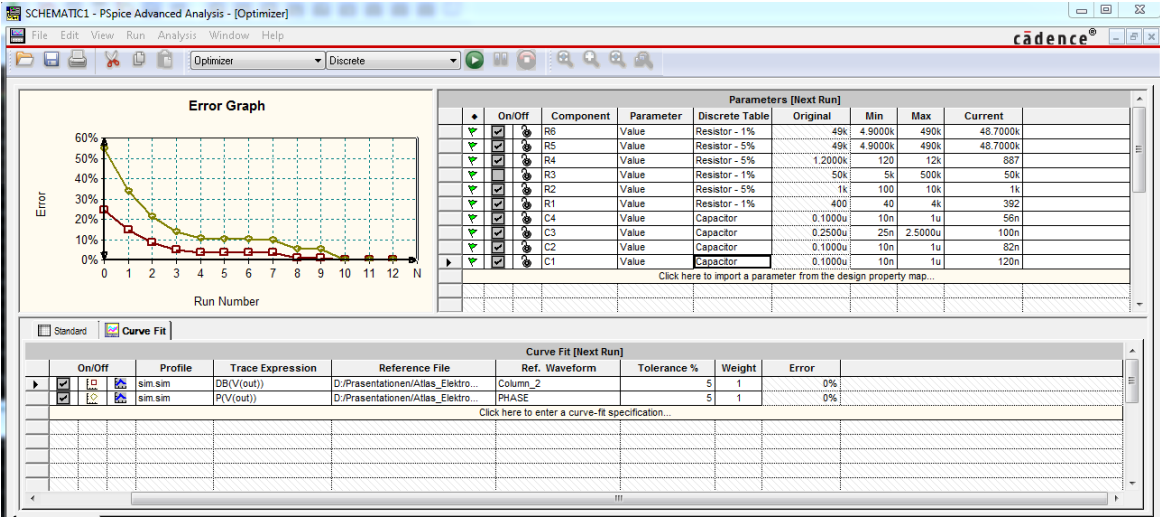
Note

For your information, the default discrete table values are located under **<installation>\ tools\pspice\library\discretetables.**

If you want to define your own discrete tables, you have to use this structure in a text file save with the extension .table:

```
res2to10%
START
10e-1
11e-1
12e-1
13e-1
15e-1
16e-1
18e-1
20e-1
22e-1
24e-1
27e-1
30e-1
33e-1
36e-1
39e-1
43e-1
47e-1
51e-1
```

Then you have to include your defined table in Optimizer clicking on **Edit > Profile Settings**: