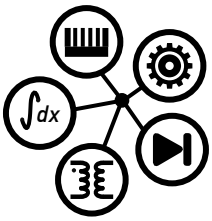

PLECS

*DEMO MODEL*

# Buck Converter with Parameter Sweep

Last updated in PLECS 4.3.1

# 1 Overview

This demonstration is based on the demo model “Buck Converter with Analog Controls” in the PLECS demo models library. It performs a parameter sweep by modifying the value of inductor $L_1$ from a simulation script.

# 2 Model

The schematic shows a buck converter with an analog proportional integral derivative (PID) controller. The inductor $L_1$ is labeled in the schematic below and can be simulated three times in an automated fashion, with a value of 50 $\mu$H, 100 $\mu$H, and 200 $\mu$H, respectively.

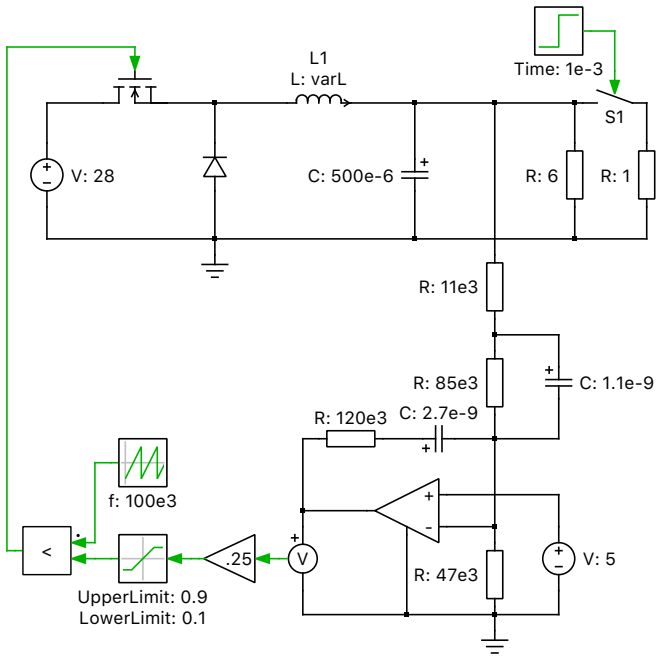


**Figure 1: Buck converter with analog controls**

# 3 Simulation

The simulation demonstrates the start-up of the converter and a load jump at 1 s. A single transient simulation can be performed where an inductance value of 50 $\mu$H is assigned to $L_1$ from the model initializations. Alternatively, by using the pre-defined script, three simulations are performed, one for each inductance value of $L_1$ described above. The result of each simulation is displayed as a new trace in the scope. Each trace is labeled with the corresponding inductance value. The script also analyzes the simulation result and prints the peak current value into the MATLAB or Octave console. To run the scripted simulation demonstration execute the following steps for either PLECS Blockset or PLECS Standalone:

- **Blockset:** Double-click on the subsystem block in the Simulink model to view and run the m-file `parameter_sweep_script.m` in the MATLAB editor which has the following content:

```
% create path to scope
scope = ('buck_converter_with_parameter_sweep/Circuit/Scope');

% clear all previous traces in scope 'Scope' in the current model
```

```
plecs('scope', scope, 'ClearTraces');

% parametric values to be swept
inductorValues = [50, 100, 200]; % in uH

for ix = 1:length(inductorValues)
  % set value for L1
  varL = inductorValues(ix) * 1e-6;
  % start simulation, return probed signal values to workspace using Output port '1'
  [t, x, y] = sim('buck_converter_with_parameter_sweep');
  % hold and label traces in scope
  plecs('scope', scope, 'HoldTrace', ['L=' mat2str(inductorValues(ix)) 'uH']);
  % find maximum current value and index
  [maxv, maxidx] = max(y(:,1));
  % Output maximum current values to MATLAB console
  fprintf('Max current for L=%duH: %fA at %fs\n', inductorValues(ix), maxv, t(maxidx));
end
```

The results are printed into the MATLAB console.

- **Standalone:** Select **Simulation scripts...** from the **Simulation** menu and run the following **Parameter Sweep** script:

```
% create simStruct with field 'ModelVars'
mdlVars = struct('varL', 50e-6);
simStruct = struct('ModelVars', mdlVars);

% clear all previous traces in scope 'Scope' in the current model
plecs('scope', './Scope', 'ClearTraces');

% parametric values to be swept
inductorValues = [50, 100, 200]; % in uH

for ix = 1:length(inductorValues)
  % set value for L1
  simStruct.ModelVars.varL = inductorValues(ix) * 1e-6;
  % start simulation, return probed signal values to workspace using Output port '1'
  out = plecs('simulate', simStruct);
  % hold and label traces in Scope
  plecs('scope', './Scope', 'HoldTrace', ['L=' mat2str(inductorValues(ix)) 'uH']);
  % find maximum current value and index
  [maxv, maxidx] = max(out.Values(1,:));
  % Output maximum current values to Octave Console window
  printf('Max current for L=%duH: %fA at %fs\n',
     inductorValues(ix), maxv, out.Time(maxidx));
end
```

The results are printed into the Octave console which can be accessed by selecting **Show Console** from the **Window** menu.

# 4 Conclusion

This model demonstrates using a simulation script to perform a parameter sweep of a physical circuit value in either PLECS Blockset or PLECS Standalone. This example code can be readily be adapted to other applications.

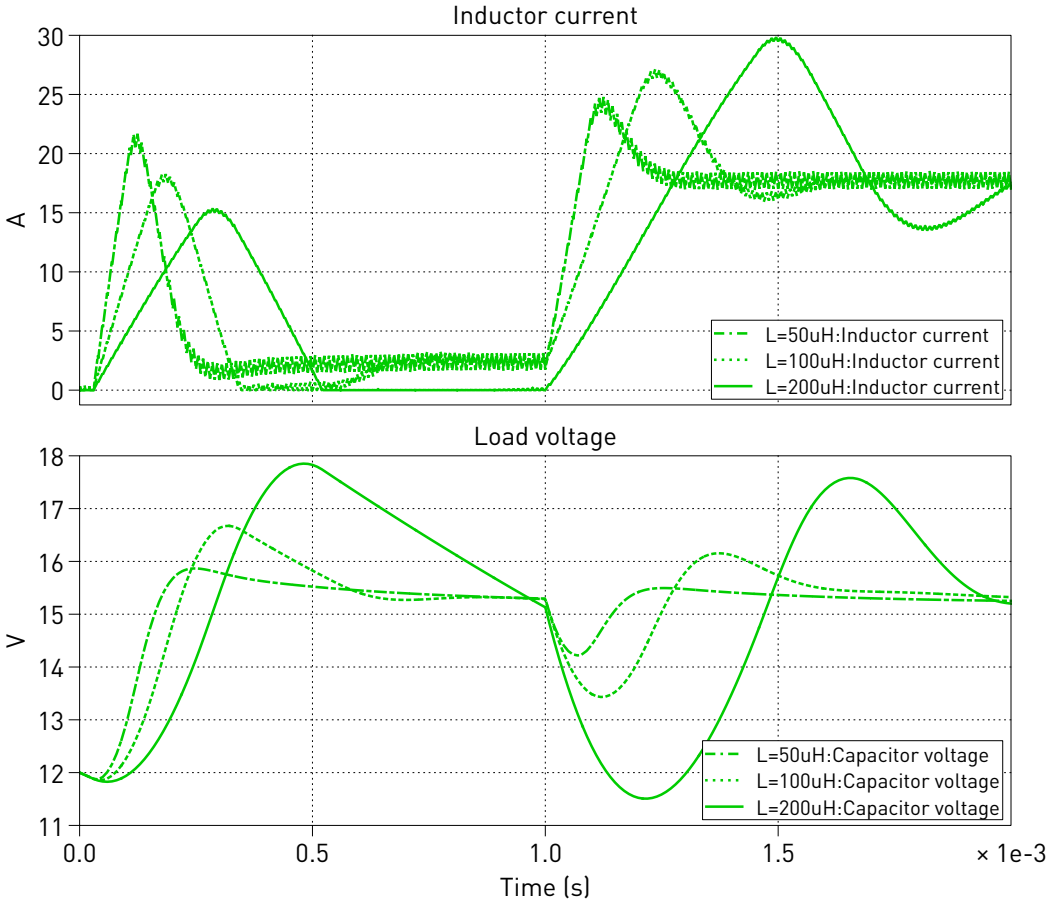


**Figure 2: Output results of three different inductance values from parameter sweep**

**Revision History:**

| | |
|---|---|
| PLECS 4.3.1 | First release |

**How to Contact Plexim:**


| | | |
|---|---|---|
| ☎ | +41 44 533 51 00 | Phone |
| | +41 44 533 51 01 | Fax |
| ✉ | Plexim GmbH<br>Technoparkstrasse 1<br>8005 Zurich<br>Switzerland | Mail |
| @ | `info@plexim.com` | Email |
| | `http://www.plexim.com` | Web |


*PLECS Demo Model*