Simulation tools

F. Jenni
Paul Scherrer Institute, Villigen, Switzerland

Abstract
In the last two decades, simulation tools made a significant contribution to the great progress in development of power electronics. Time to market was shortened and development costs were reduced drastically. Falling costs, as well as improved speed and precision, opened new fields of application. Today, continuous and switched circuits can be mixed. A comfortable number of powerful simulation tools is available. The users have to choose the best suitable for their application. Here a simple rule applies: The best available simulation tool is the tool the user is already used to (provided, it can solve the task). Abilities, speed, user friendliness and other features are continuously being improved—even though they are already powerful and comfortable. This paper aims at giving the reader an insight into the simulation of power electronics. Starting with a short description of the fundamentals of a simulation tool as well as properties of tools, several tools are presented. Starting with simplified models of power semiconductors, a time-continuous Matlab/ Simulink model for an H-bridge is introduced. The second part starts with the hardware model of a buck converter and the design and simulation of an appropriate linear controller. A possible structure of a controller with limiters and additional voltage feedforward is shown. The text closes with a summary of present challenges in simulation.

1 Why simulations

Simulation is a very important means in the field of power electronics. Simulation leads to

- saving of development time,
- saving of costs ('burnt power circuits tend to be expensive'),
- better understanding of the function,
- testing and finding of critical states and regions of operation,
- fast optimization of system and control.

Today it is difficult to imagine the task of power electronics development without the help of simulation.

2 Principle of simulation in the past and today

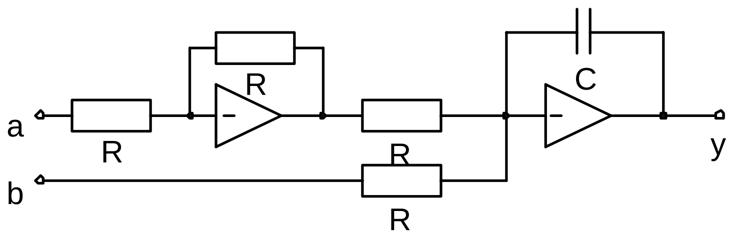
The key operation of all simulation is the solution of differential/integral equations. That can be done with analog or digital computers:

2.1 Analog computers

Simulation tools are quite old. For more than 50 years analog computers have been built in numbers for various simulation purposes. Essentially, they were based on the functions: addition, subtraction, and integration. That allowed the simulation of linear physical systems that could be described with linear integral equations. Multiplication, division, and other non-linear operations came later as addenda.

The 'program' of a simple linear equation on an analog computer looked as follows:

$$y = \int (a - b) dt$$



(The operational amplifier being ideal, i.e. no drift, and RC = 1)

The result of the integration on an analog computer is mathematically correct.

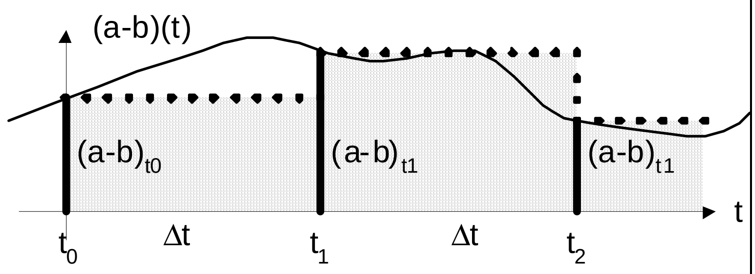
Depending on the time constants of the integration, a time-scaling of the equation was sometimes necessary. Given by the maximal output voltage of the amplifiers, the amplitudes of the signals had to be scaled too. It is obvious that in the end a rescaling of the output 'result' was also needed. Time-scaling and rescaling caused problems for beginners.

While analog computers were widely used in the past, most of them vanished after 1980.

2.2 Digital computers

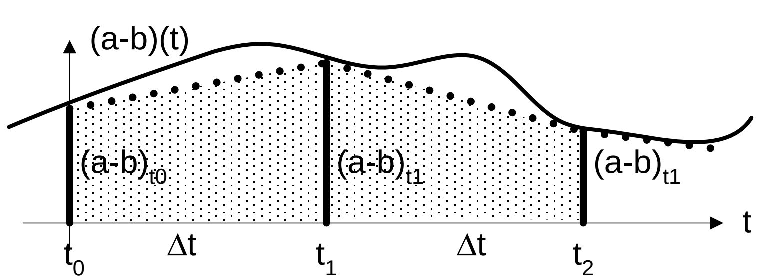
On a digital computer the integration can be represented by a simple summation according to:

$$y = \sum_i (a - b)_{ti} \Delta t$$



An improved integration looks as follows:

$$y = \sum_i \{(a - b)_{ti} + (a - b)_{ti+1}\} \frac{\Delta t}{2}$$



Besides the two shown fixed-step solvers (Δt = constant), solvers exist which adapt the time step to the function that has to be integrated. They are very useful for systems that contain switching actions.

Over all, a large number of integration methods ('solvers') exist. All of them have advantages and disadvantages. But in general, all of them compute a time-discrete summation only as an approximation of the integral. Therefore, the integration algorithm is very important for the result of a simulation. An inappropriate algorithm can lead to long simulation times and/or wrong results.

In spite of the problems of a correct integration, simulations on digital computers have a number of advantages:

– Easy implementation of non-linear functions (multiplication, division, complex functions, etc.).
– Comfortable input and output of model and data.
– No drift and offset problems.
– Translation of a graphically entered model into a mathematical description for the simulation by the computer: this has brought an extreme comfort in the last years.
– Some tools provide an automatic linearization of non-linear systems.
– No time and amplitude scaling necessary.

3 Simulation tools

The best available simulation tool is the tool you are used to!

(Provided, that it can solve the task.)

There are a number of powerful simulation tools available. All of them have advantages and disadvantages. They can be grouped into three parts; according to the way the system to be simulated is 'entered into the computer':

Mathematical input

In this case an exact mathematical description of the circuit to be simulated is entered into the machine. This can be done with various programming languages such as Basic, Fortran, etc., or the very practical Matlab. (Matlab with the toolbox 'Simulink' has an additional feature: the possibility to enter the mathematical description in graphic form.)

Input in Matlab:

```
% Calculated data
udc_in =   udc0;
uout_0 =  m0*udc_in
iout0    =  uout_0/Rm
```

Netlist input

This form of input was used, for example, by former versions of Spice. The physical elements of a circuit, i.e., resistors, capacitors, active elements were entered on the keyboard as a description list. This form of entering a circuit can still be used in Spice. But, it is no longer necessary. Nowadays, the circuit can be entered into the computer graphically.

Example:

```
R_R1         N2 N1  10
C_C4         N5 N6  1u
D_D1         N6 N1 D1N4007 100
+SIN 0 326.6 50 0 0 240
V_V12         N14 0
+SIN 0 326.6 50 0 0 0
R_RD         N16 N15  20m
D_D4         0 N6 D1N4007 100
```

Graphical input

This group contains the tools with the most comfortable form of entering the system into the machine. Most of today's simulation packages for power electronic provide graphical input. In the following, only such tools (beside Matlab-Simulink) will be discussed.

3.1 What can be expected from a good tool?

Some properties of a good tool are

- Comfortable, intuitive input of the circuit.
- Correct models for the elements (as simple as possible but, as good as necessary).
- Correct error messages.
- Robust execution of the simulation.
- Sophisticated integration algorithm (various algorithms to be chosen for the type of model).
- Good output of the simulation results (formats which can be exported to other programs).
- Support from the manufacturer.
- Portability of models from one program version to following ones.

3.2 Critical tasks and other critical points

Difficult for digital simulation tools are

- fast events like switching actions,
- differentiation.

Some tools do have electrical elements which are not well-enough modelled (some were even wrong….). To find 'bad' elements in a simulation can be very time-consuming.

3.3 A selection of simulation tools

There are a number of simulation tools available. All of them have advantages and disadvantages (one of the latter often being the cost). The following selection of tools provides graphical input.

a) PSpice

PSpice has been on the market for a long time. It started as a simulation tool for low-power electronic circuits. A large library with PSpice models for various electronic components exists. Further models can be added. Today it can simulate analog and digital circuits with a lot of features. The representation of numerical blocks and controllers is difficult.

Availability, costs:

- Industry from 7500 €
- Universities from 3000 €
- Student licences / demo version available (reduced model sizes).

Information:
www.logmatic.ch ; www.orcad.com ; www.orcadpcp.com

b) Matlab / Simulink / SimPowerSystems / PLECS

Matlab is a mathematical tool that has been established for a long time. Toolboxes for various applications exist. One of them is Simulink, a graphical tool for the entering of functions. Simulink itself can be expanded with another toolbox: SimPowerSystem. This toolbox is designed for the simulation of electrical power systems including power electronics. The elements of the various toolboxes can be combined.

Availability, costs (Matlab, Simulink and SimPowerSystem):

- Industry from 8400 €
- Universities from 2100 €
- Demo version/student licences available for a small amount (reduced model sizes).

Information:
www.mathworks.ch ; www.mathworks.com

An additional toolbox for the simulation of power electronics is PLECS. This is a fast and reliable power toolbox for Matlab.
www.plexim.com

c) PSIM

PSIM is one of the tools developed specifically for power electronics. Therefore, it is optimized for the tasks arising in this field. This results in fast simulation runs.

PSIM offers some add-ons, one of them being an interface to Matlab/Simulink. With that interface the full mathematical power of Matlab is accessible.

Availability, costs:

- Industry from 1700 €
- Universities from 280 €
- Demo version/student licences available for a small amount (reduced model sizes).

Information:
www.powersimtech.com ; www.powersys.fr

d) Simplorer

Basically Simplorer consists of four modelling languages:

- VHDL-AMS for analog-mixed-signal design.

– Circuit simulator for the simulation of power electronic circuits.
– Block diagram simulator for the simulation of controllers and similar tasks.
– State machine simulator for event-driven systems.

These features enable the engineer to choose the language most appropriate to the task. Simplorer can be interfaced to a number of other simulation tools.

Student version available (reduced model sizes)
Prices and conditions are unknown to the author.

Information:
www.ansoft.com

e) CASPOC

This tool is designed for the simulation of power electronics and electrical drives. It provides a large library of blocks for both topics. In addition, code in Pascal and C can be included.

A freeware version of CASPOC is available.
Prices and conditions are unknown to the author.

Information:
www.integratedsoft.com

f) Saber

Saber is a tool that has been developed for a wide range of applications, including power electronics. Saber can handle analog, digital, mixed and event-driven devices. It can be linked to digital simulations to handle models written in Verilog or VHDL.

Prices and conditions are unknown to the author.

Information:
www.avanticorp.com

4 Switches, semiconductors, and passive elements

4.1 Switches

In the original sense the word ‘switch’ is used for mechanical elements which open or close an electrical circuit. Some switches are activated manually, others by an electrical coil. The latter were called relay or contactors.

For all switching elements the following four states are interesting:

1. turned off
2. turn-on process
3. turned on
4. turn-off process.

The exact behaviour of switches can be rather complicated. In most cases the turned-off state (switch open) can be modelled by an infinite resistance. Turned on (switch closed) can be represented by a small resistance.

The turning on and off processes include a delay between the control signal and the real opening or closing of the contacts. In particular, with higher voltages arcing can occur. Switching actions often consist of a number of on/off's due to bouncing of the contacts. The effects of arcing and bouncing are reduced with RC snubbers.

4.2 Semiconductors

The word 'power electronics' is derived from the use of 'electronic elements' for power applications. In small- to medium-sized power supplies the semiconductors are diodes, Field Effect Transistors (FET) and Insulated Gate Bipolar Transistors (IGBT). For high power applications thyristors and thyristors with Gate Turn Off capability, GTOs and IGCTs are used.

In the field of power electronics the power semiconductors have to be operated in the 'switching mode' to avoid excessive losses and damage: they are either turned on or off. Compared to mechanical switches, semiconductors are much faster and are capable of an 'unlimited' number of switching actions. The four states of a semiconductor will be discussed briefly.

– In the turned-off state a semiconductor is well represented by a very high resistance. A small leakage current can exist. But, this current is usually neglected. The limited voltage-blocking capacity has to be taken into account.

– While turning on (an action that lasts some 10 ns for FETs and about 1 μs for IGBTs) current through and voltage across the element occur. This leads to the 'turn-on losses'. There is a small delay between the turn on signal and the actual turning on. Beginning and ending of the turn-on process are gradual.

– Turned on, the FET represents a simple resistor. The IGBT can be modelled with a resistor in series with a voltage source. In this state the conducting losses are generated. The current-carrying capacity of the elements is limited.

– Turning off is a transient state with a similar duration to the turn-on process. Current through and voltage across the element occur for a short time simultaneously, which leads to the 'turn-off losses'. Especially for IGBTs the end of the turn-off process is gradual which leads to the current tail. A turn-off delay occurs between the control signal and the falling off of the current.

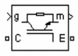
For an exact model of the semiconductor, internal capacitances and stray inductances have to be considered too. Depending on the quality of the data sheets of the semiconductor it is often difficult to find all the mentioned date. Depending on the simulation tool used, more or less data for the description of the elements is needed.

As a general rule: The simpler the model the faster the simulation.

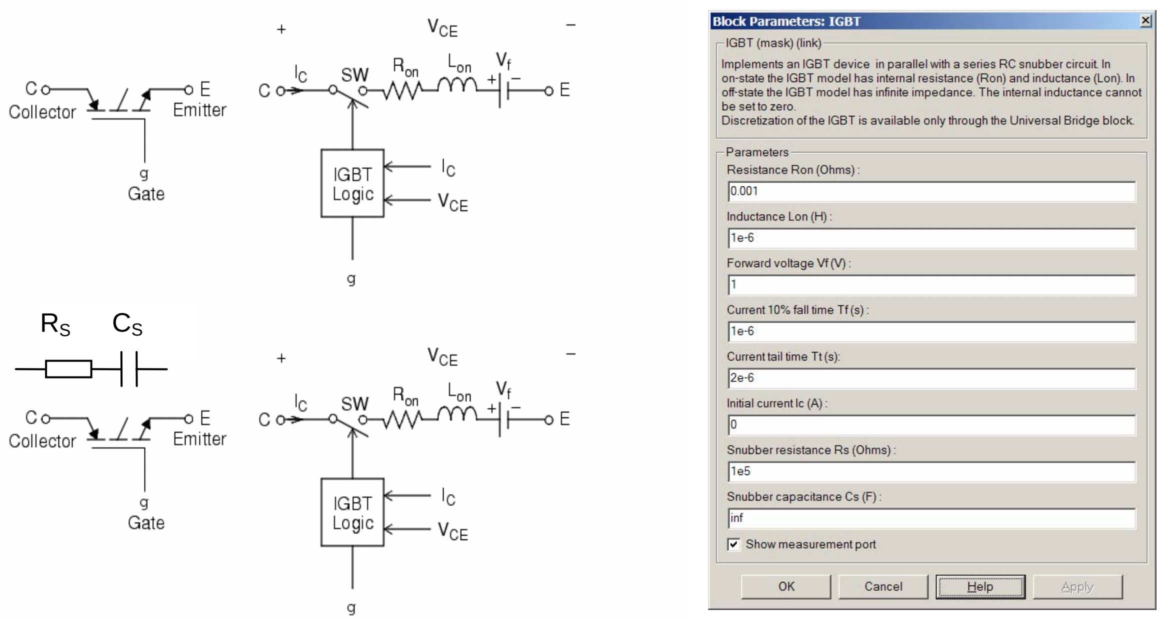
It is part of the art to find the simplest model that fulfils the needs of a simulation.

As an example the block parameters for an IGBT are shown.

Description



The IGBT block implements a semiconductor device controllable by the gate signal. The IGBT is simulated as a series combination of a resistor Ron, inductor Lon, and a DC voltage source Vf in series with a switch controlled by a logical signal (g > 0 or g = 0).



4.3 Snubbers

Because of the fast turn-off capability of switches and power semiconductors, the inductances in series to the elements are very critical. Most elements do also have an internal inductance. These inductances are the reason for the need of RC snubbers connected across the semiconductor. The modelling of inductance and snubber are critical in simulations.

4.4 Passive elements

In power supplies, as in other power electronic circuits, passive elements are needed. Most real passive elements have a non-ideal behaviour. The non-ideal effects are

Resistors: The models are usually pure ohmic. Real elements can be

– temperature-dependent

– frequency-dependent (skin effect, proximity effect)

– inductive.

Capacitors: They show

– current-dependent losses (‘Equivalent Serial Resistance ESR’)

– voltage-dependent losses (especially electrolytic capacitors)

– serial inductance.

Inductances: These can be very demanding elements. They are

– ohmic (losses!)

– frequency-dependent (skin and proximity effect leading to increase of the resistance; decrease of the inductance)

– amplitude-dependant (saturation)

– capacitive (especially for higher frequencies).

For larger magnets these effects can be very troublesome. It can be necessary to generate models which take some of these effects into account (RLC ladders).

Coupled inductances (transformers): It is important to understand the definition of the model. In some simulation programs the models did not operate correctly in the past.

5 Fast models for switched power stages

The core of a magnet power supply consists of the power semiconductors. In case of a buck converter this is typically a single FET or IGBT and a diode; in the case of a 4Q converter there are 4 FETs or IGBTs with their associated diodes. Depending on the goal of the simulation the power part can be modelled differently:

Electrical circuit design:

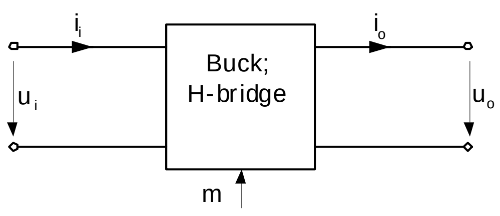
For the design of the circuit a good model of the semiconductors is necessary. Therefore the semiconductor models, as discussed before, have to be implemented in the circuit. The simulation results of such a model are good. But, the simulation of an exact switched model is time-consuming, on account of the switching actions.

Controller design:

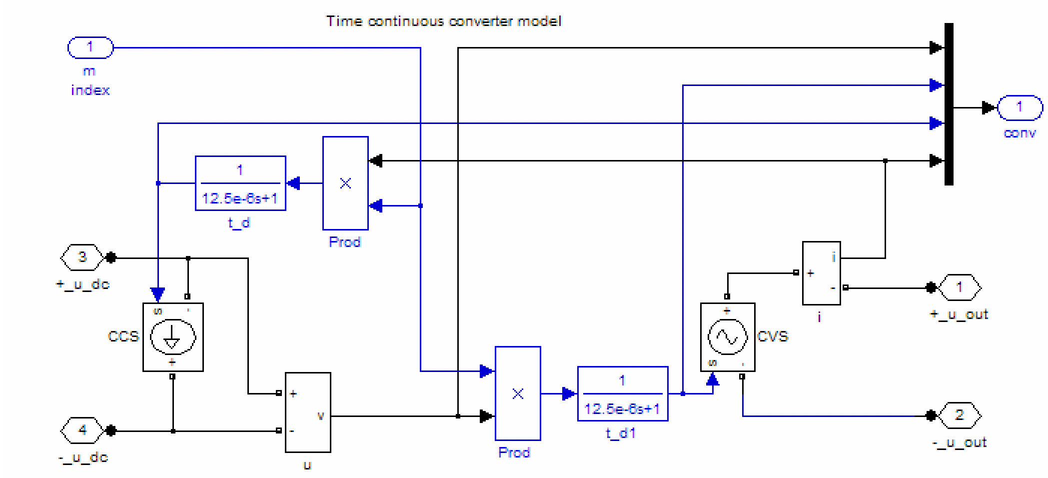
The design of the controller starts when the electrical design is finished. Longer time intervals are being investigated. Hence, a 'faster' model is desirable.

For the controller design, the frequency components of the switching frequency are filtered so well that they are no longer of interest. Therefore a time-continuous description of the power semiconductor part is interesting—time-continuous simulations are fast.

The ***time-continuous description*** of H-bridge and buck converters can be done with two mathematical equations (i: input; o: output; m: modulation index):

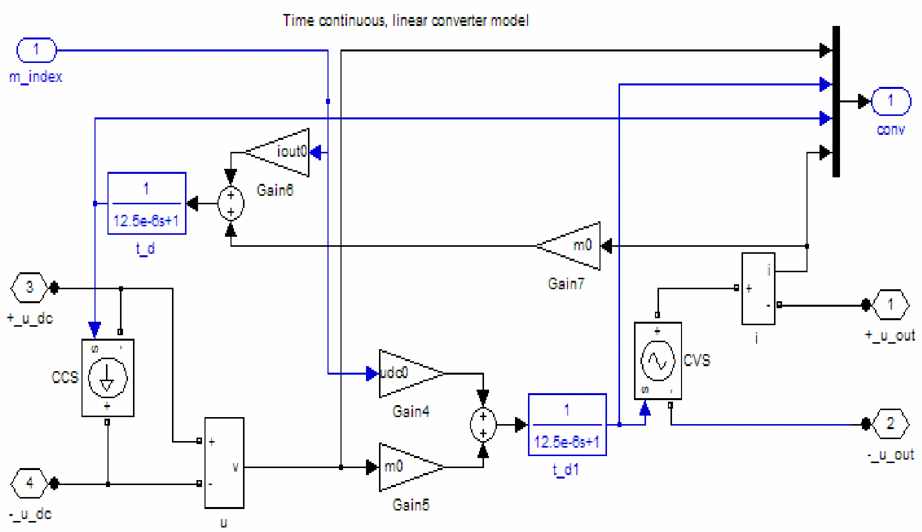
	$u_o = m \cdot u_i$ $i_i = m \cdot i_o$	$buck: 0 \le m \le 1$ $H\text{–}bridge: -1 \le m \le 1$

In Matlab /Simulink the following structure for the converter results:



For a ***linear, time-continuous description*** of the equations, the continuous model has to be linearized (index 0: point of operation):

Non-linear	linear
$u_o = m \cdot u_i$	$\Delta u_o = \Delta m \cdot u_{i,0} + m_0 \cdot \Delta u_i$
$i_i = m \cdot i_o$	$\Delta i_i = \Delta m \cdot i_{o,0} + m_0 \cdot \Delta i_o$



With the linear model a direct state-space description of the system is possible. This description is necessary for the design of state-space controllers. Before simulation with this model the steady-state values have to be specified:

- input voltage: u_{dc0}
- modulation index: m_0
- output current: i_{out0} .

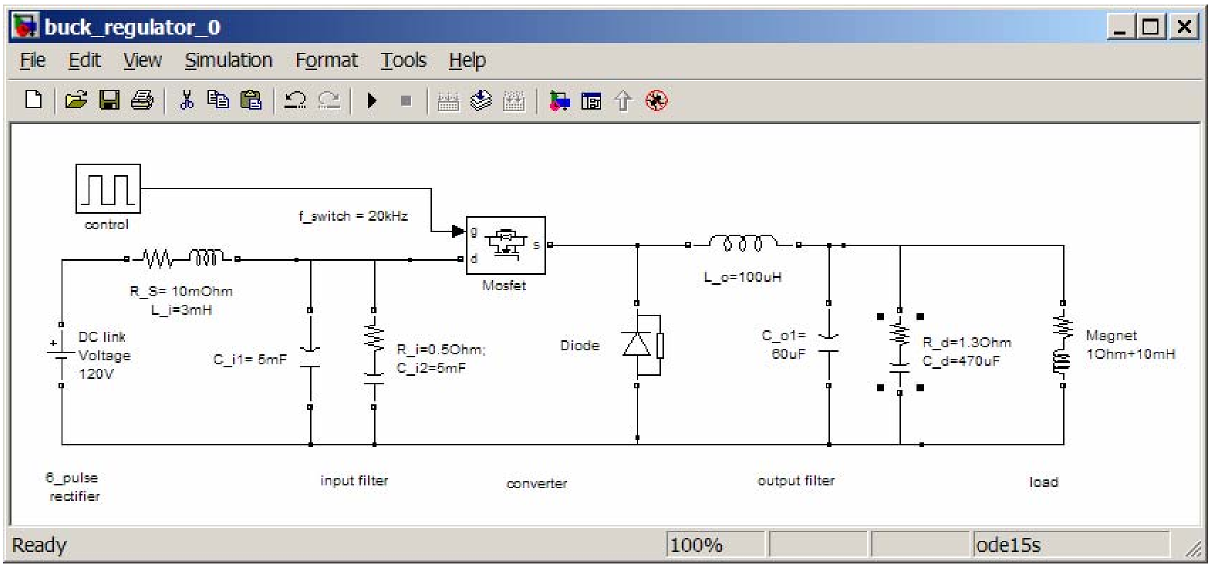
Literature:

Various descriptions of pulse-width modulation and continuous descriptions of converters can be found in the book *'Steuerverfahren für selbstgeführte Stromrichter'*, Felix Jenni and Dieter Wüest, vdf-Verlag Zürich; ISBN 3-7281-2141-X.

6 Circuits and modelling

For the discussions a buck converter with input and output filter will be used as a model.

6.1 'Model power supply'



Converter data

100 V/100 A at the load, switching frequency 20 kHz (above the audible range).

The 3 dB frequency of the input filter is chosen to be 30 Hz: L_i = 3 mH/C_i1 + C_i2 = 10 mF. Although the ripple frequency of a six-pulse rectifier is 300 Hz, also 50 Hz have to be expected in the dc-link voltage, due to voltage asymmetry in the mains.

The maximum ripple of the current through L_o is chosen to be 15% of the maximum dc-current, i.e. 15 App. The 3 dB frequency of the output filter should be one decade below the switching frequency, i.e. 2 kHz: L_o1 = 100 µH/C_o1 = 60 µF.

The input and output filter are both damped with small damping resistors in series with capacitors.

Both filters are coupled through the FET.

Semiconductors and snubbers

The models of FET and diode contain serial inductances as well as parallel RC snubbers.

With the rule of thumb '1m of wiring introduces an inductance of 100 nH…1 µH 10 nH are introduced as serial inductances for both elements.

For the FET the snubber resistor value is chosen for a discharge current of the magnitude of the nominal current, i.e. 120 V/100 A ~ 1 Ω (in reality the current will be less, due to the finite switching time of the semiconductor). At turn-off the snubber resistor is responsible for the over voltage across the FET: the smaller the resistor the better.

For the simulation, the snubber capacitor is now chosen in such a way that the wave impedance of the LC circuit is smaller than the snubber resistor:

$$Z_W = \sqrt{L_S / C_S} < R_S \rightarrow C > L_S / R_S^2 \ .$$

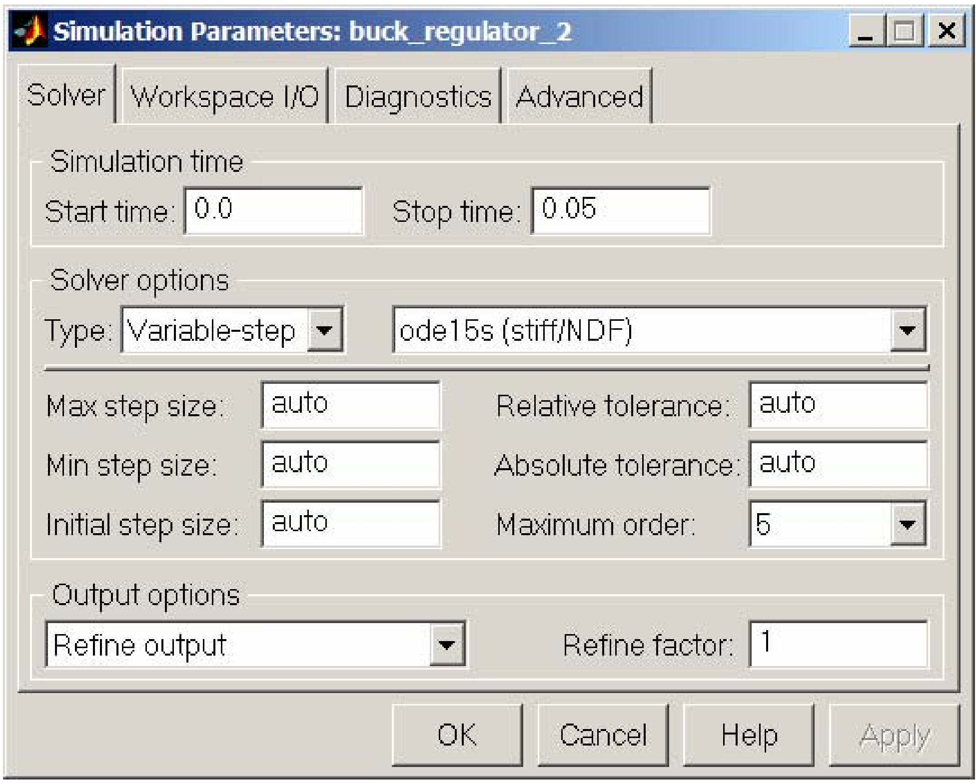
Therefore in our model the snubber capacitor has to be greater than 10 nF. Here 100 nF was chosen. (Beware: the bigger the capacitor, the more losses occur in the snubber resistor.) The same snubber was chosen for the diode.

6.2 Simulation of the model power supply

For the simulation of a circuit the simulation tool has to be set up.

Simulation time:

As a first approach for the simulation time, approximately 3 to 5 times the largest time constant of the system is recommended.



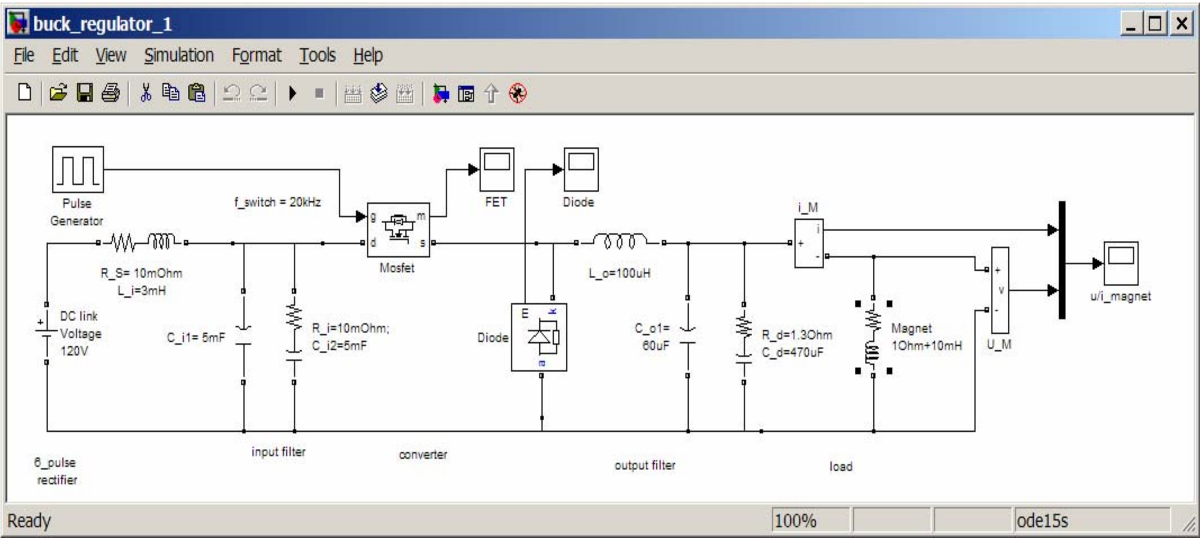
Integration algorithm

For a model that contains switching actions an algorithm with adaptive step size can speed up the simulation very much. In Matlab/Simulink there are several algorithms with this capability.

Usually, the other parameters of the simulation tool can be left at their default values.

Measurements

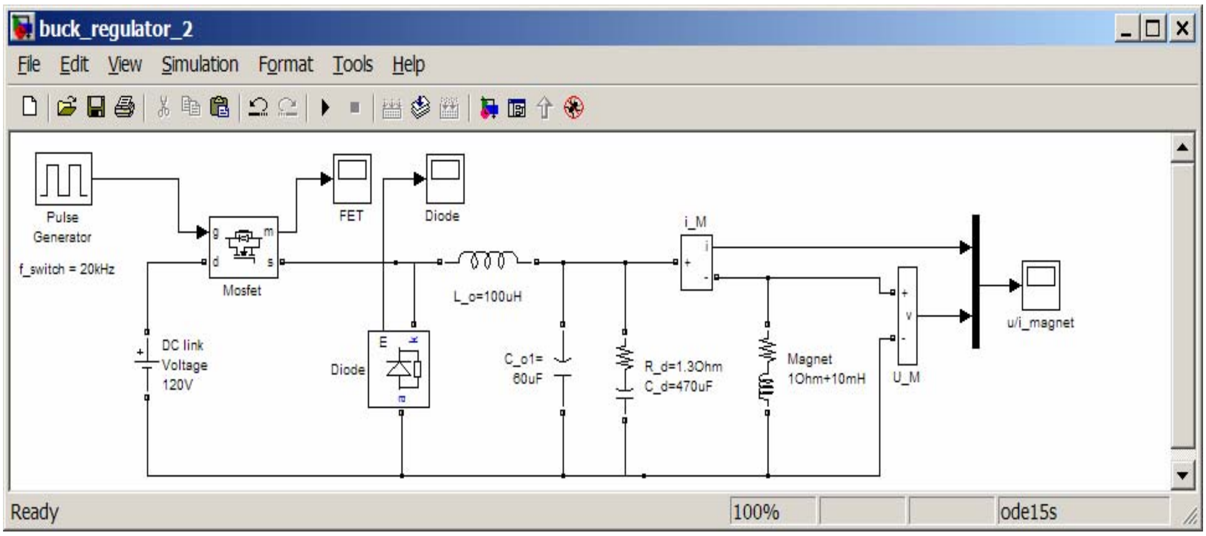
Depending on the simulation tool, the data to be measured has to be specified. In Matlab/Simulink this is done with 'scopes' or 'data export to workspace'.



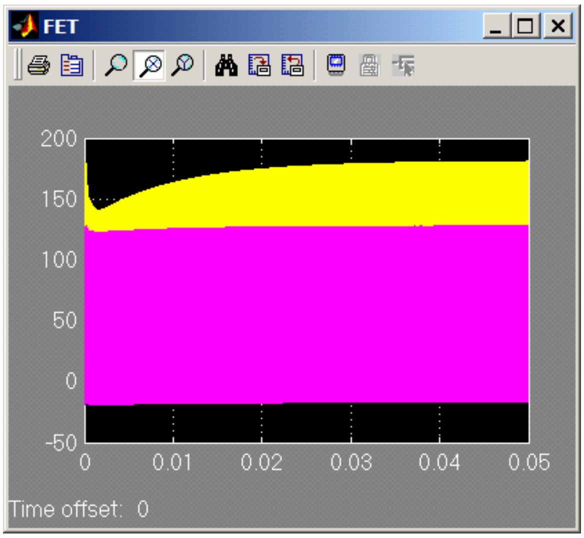
6.3 Electrical behaviour of a reduced model

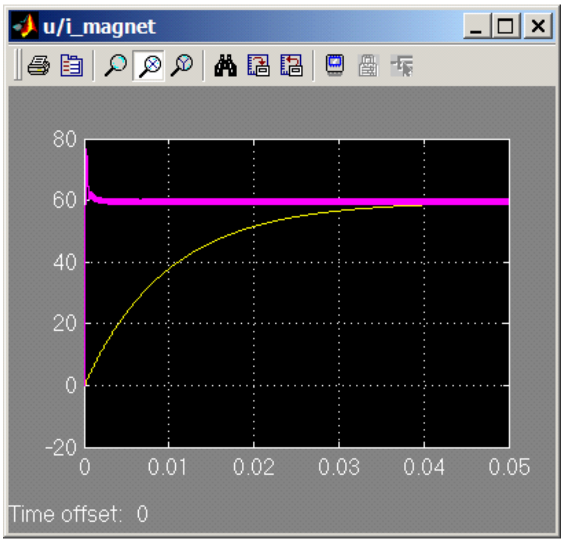
After the model is built simulation can start. This will be first demonstrated for the output part of the converter only. The dc-link filter is replaced by a stable voltage source only.

The largest time constant is that of the load: $\tau = L/R = 10$ ms → chosen simulation time 50 ms.



The FET and load currents and voltages look as follows:





The interpretation of the FET voltage and current is difficult in the chosen time-span. But, the load data seems to be correct: after an overshoot the voltage reaches a stable value and the current rises according to an e-function.

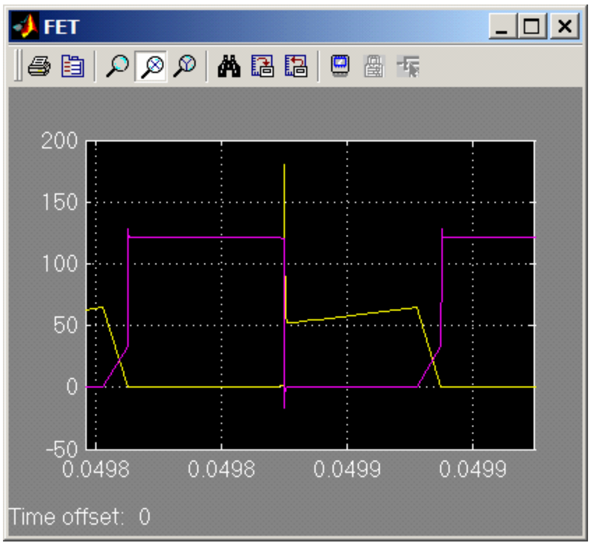
Zoomed data:

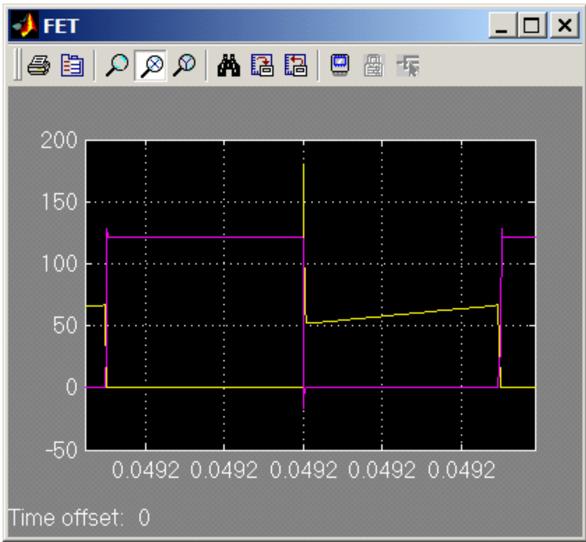
With the zoom function a better view of details is easily possible. The plot shows the voltage across and the current through the FET. Of special interest are

– The current peak caused by the snubber network.

– The current rise during the on-time of the FET.

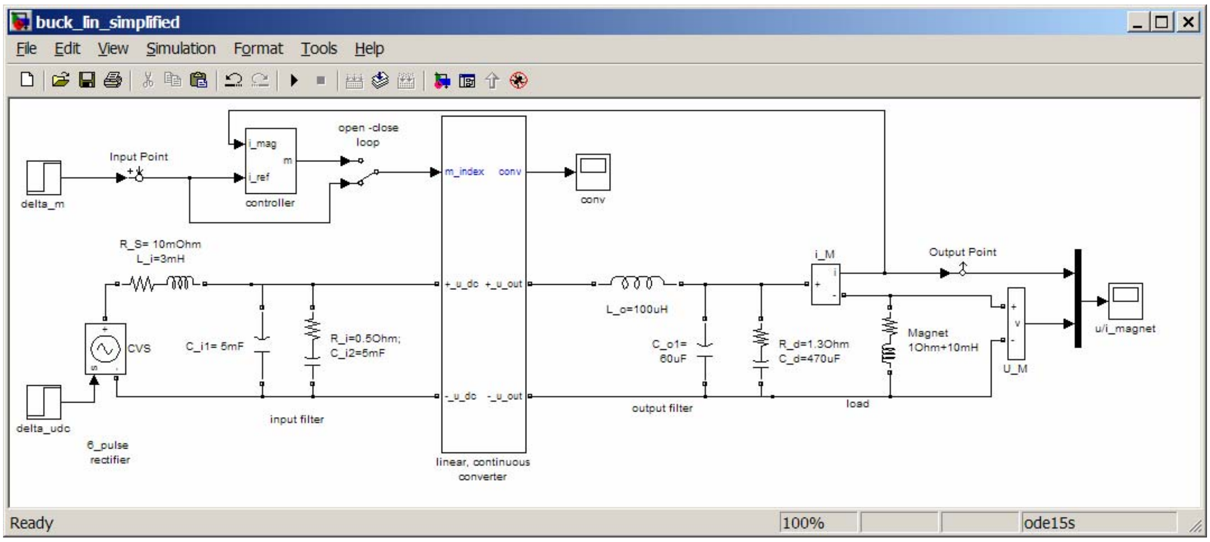
The left-hand picture shows also a problem of insufficient solver resolution: voltage and current are not correct in the region of the switching. In the right simulation the maximum time step has been reduced until correct waveforms were reached.

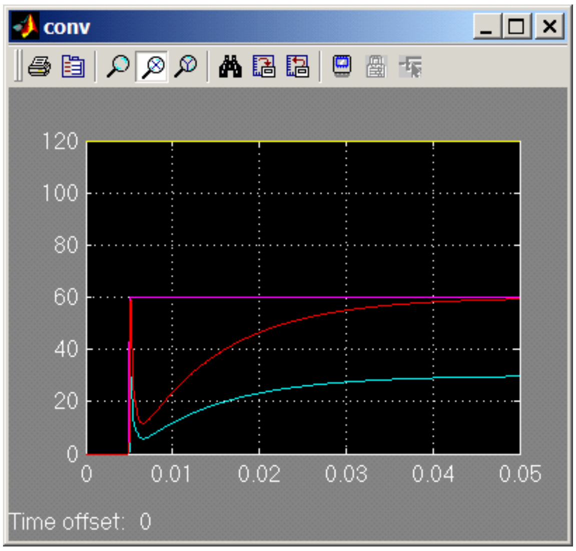


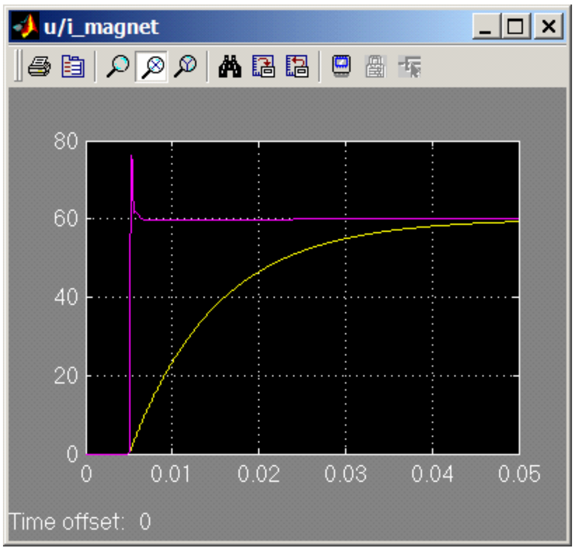


6.4 Time continuous model

For the controller design and considerations of the behaviour of the controlled circuit, it is preferable to have a faster simulation than a switched model can provide. For that purpose the linear model of the combination FET and diode is very helpful:







Simulation results:

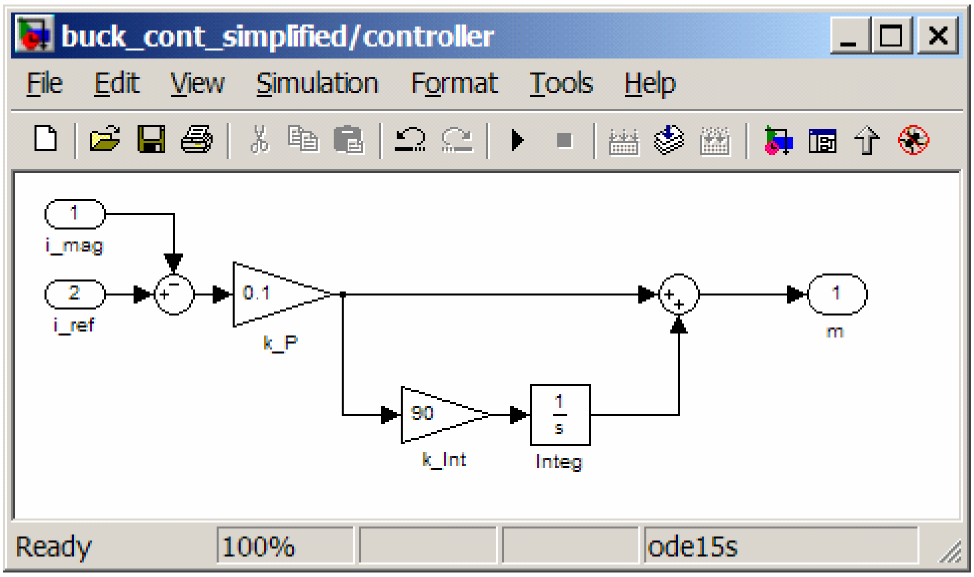
The left picture shows the internal properties of the converter: input voltage and current as well as output voltage and current.

The magnet data can be compared with the simulation of the switched converter: The load current looks identical in both cases. But, the magnet voltage of the switched model has an additional voltage ripple. This ripple, caused by the switched voltage at the filter input, does not exist in a continuous model.

7 Controller design

The design of controller is a wide field: PID controller, state controller, linear and non-linear controller, adaptive controller and so on.

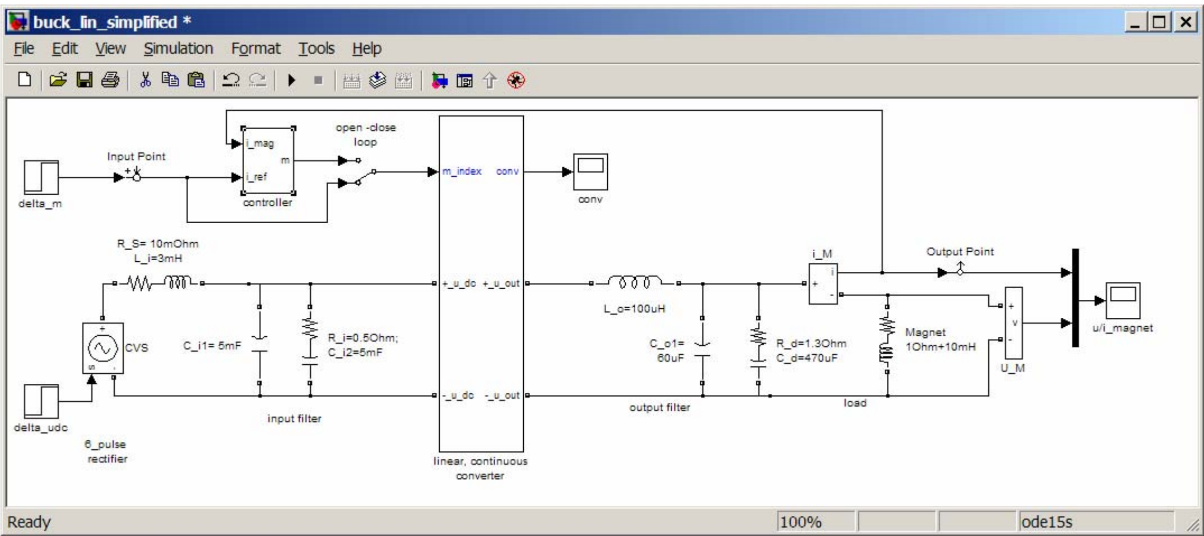
From practical usage, a linear PID controller design will be discussed in the following. As an example a power supply controller with additional feedforward, adaptation of a coefficient, d*i*/d*t* limiter and further features will be presented.



7.1 Controller design with the help of the linear model

The controller design will be made with the help of a Bode diagram. For that purpose a linear representation of the circuit, together with the PWM modulator, is necessary. Some programs are capable of generating such a model automatically. For the demonstration the already presented linear Matlab/Simulink model will be used.

With this linear model the output-to-input frequency response can be analysed in various points of operation. The result for a modulation index of $m = 0.5$ is shown as a Bode plot.



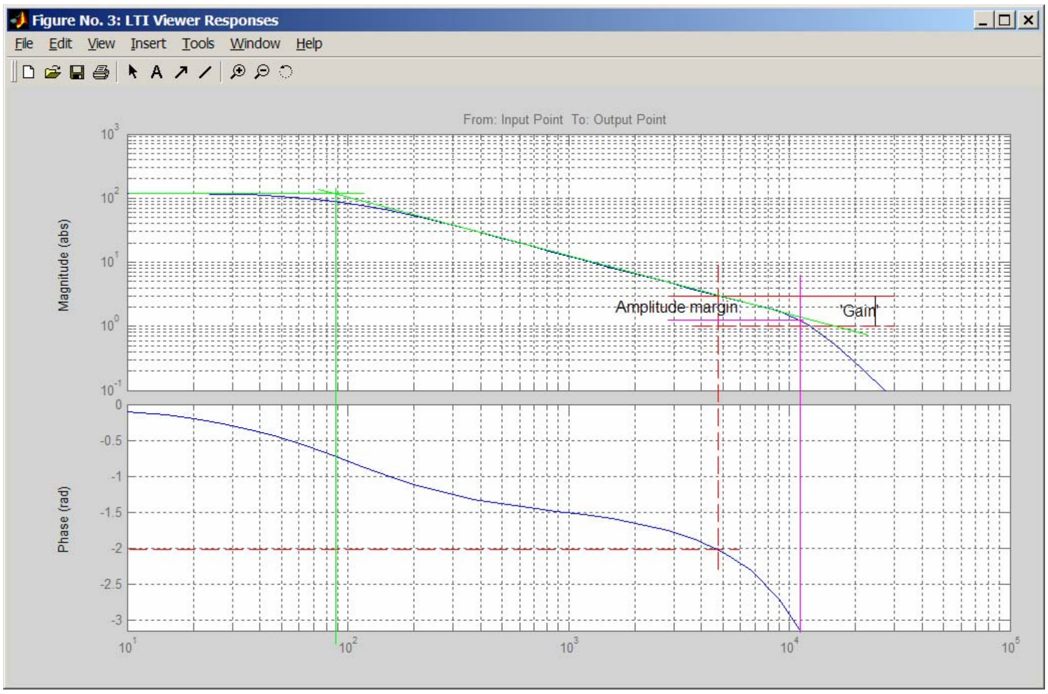
With help of the Bode diagram and the rules of Bode, the layout of a PI controller is a simple task (with a bit of routine…). (No PID if ever possible—they suffer from the amplified noise.)

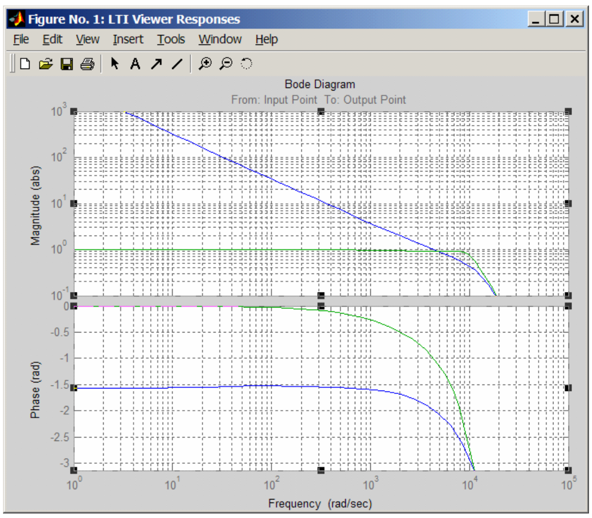
The behaviour of the model with the designed controller can be demonstrated in the Bode diagram or as a step response.

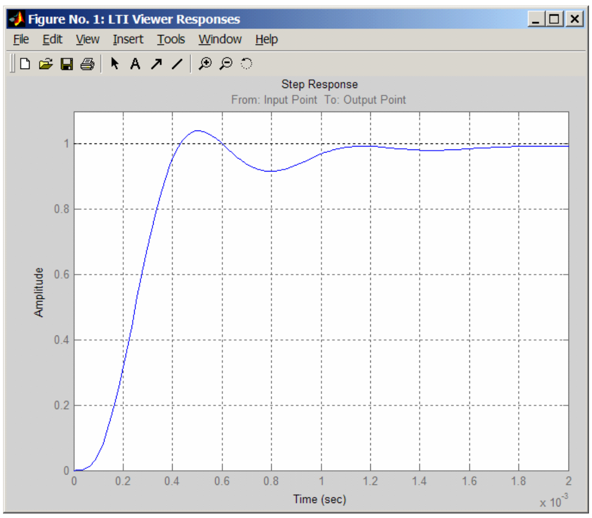
For a fast controller with minor overshoot:

Phase margin: $> \pi/3$ (60 degree); Amplitude margin: > 2 (6 dB).

With this rule the *P* gain (red lines) is 1/3 and the integrator constant 90 (green lines).



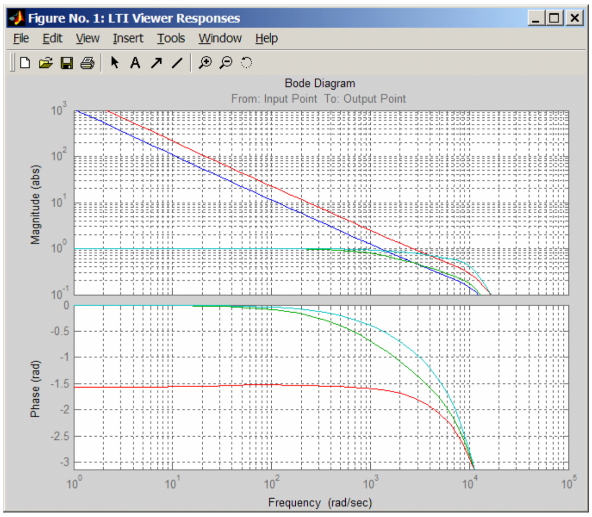


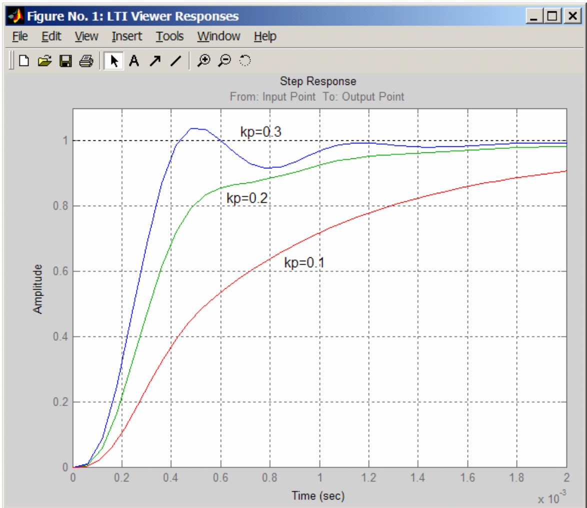


For a controller with no overshoot:

Phase margin: $\pi/3 < \varphi < \pi/2$; Amplitude margin: > 2 (6 dB).

With this rule the *P* gain is between 0.1 and 0.3 and the integrator constant does not change.

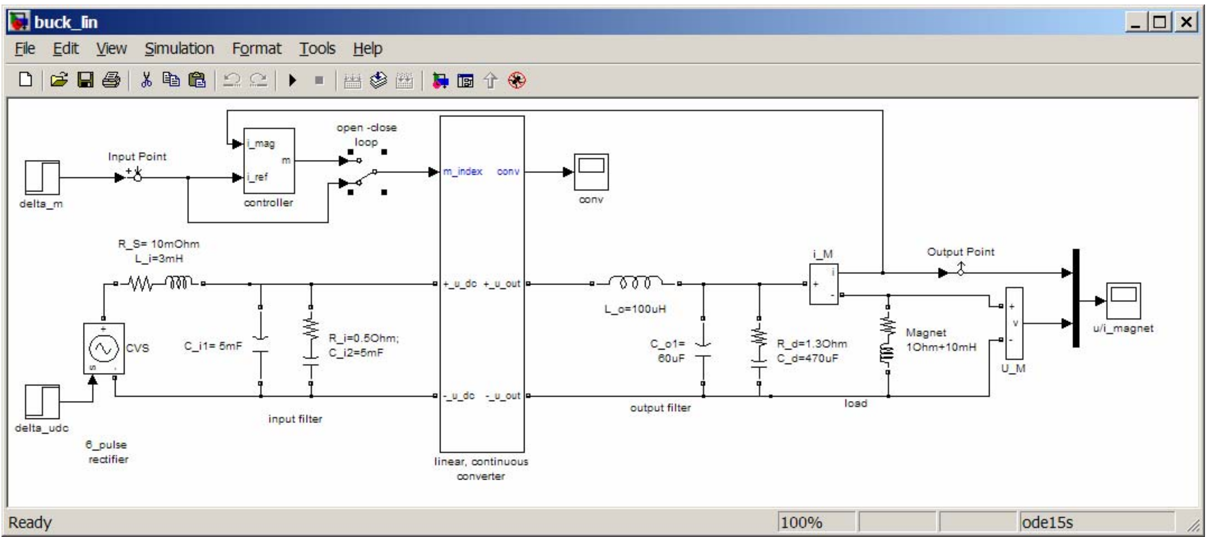




The designed controller can now be implemented in the nonlinear model.

7.2 Controller for the converter with input filter

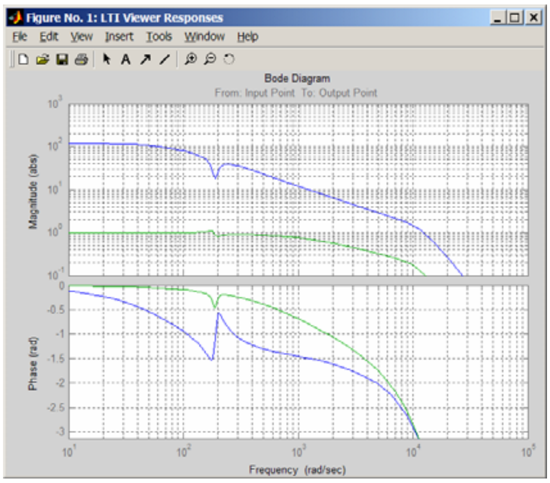
In the same way as for the reduced model a controller can be designed for the converter with input filter:

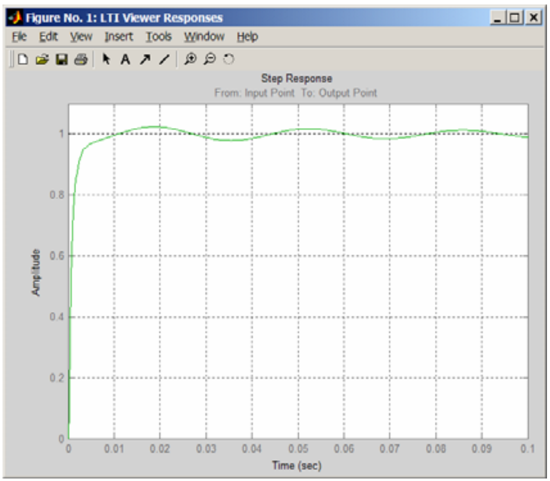


The result of the simulation shows that the system is stable, but the oscillation of dc-link voltage can not be entirely suppressed in the load current, because of the limited amplification.

(Controller: *P* gain: 0.1; Integration constant: 90.)

For an additional suppression of the influence of any dc-link voltage variation, a voltage feed-forward is added in the next demonstration.



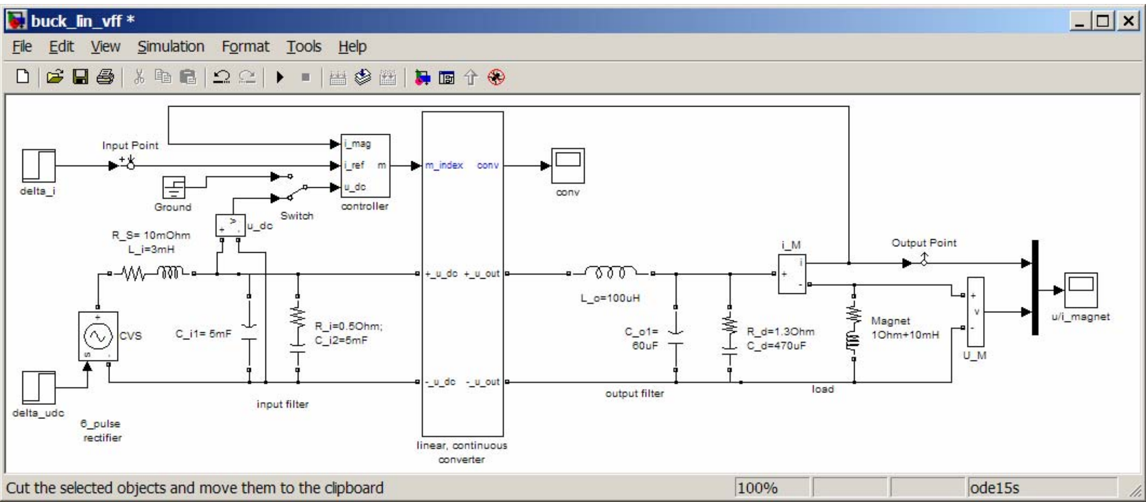


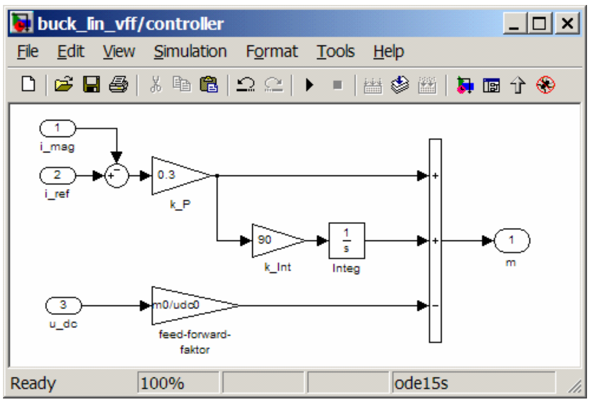
7.3 DC-link voltage feedforward

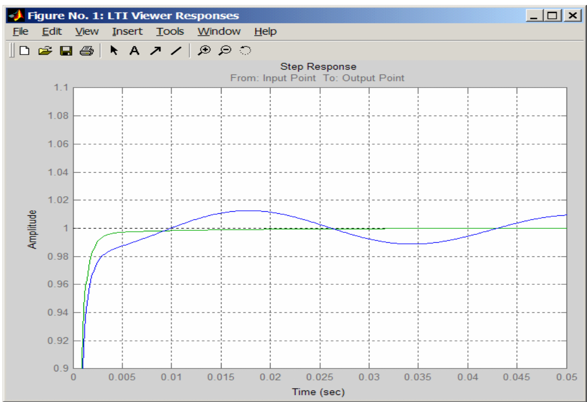
For the voltage feedforward the equation for the linearized converter is used:

$$\Delta u_o = \Delta m \cdot u_{i,0} + m_0 \cdot \Delta u_i = 0$$

$$\longrightarrow \Delta m = -\frac{m_0}{u_{i,0}} \Delta u_i \quad .$$



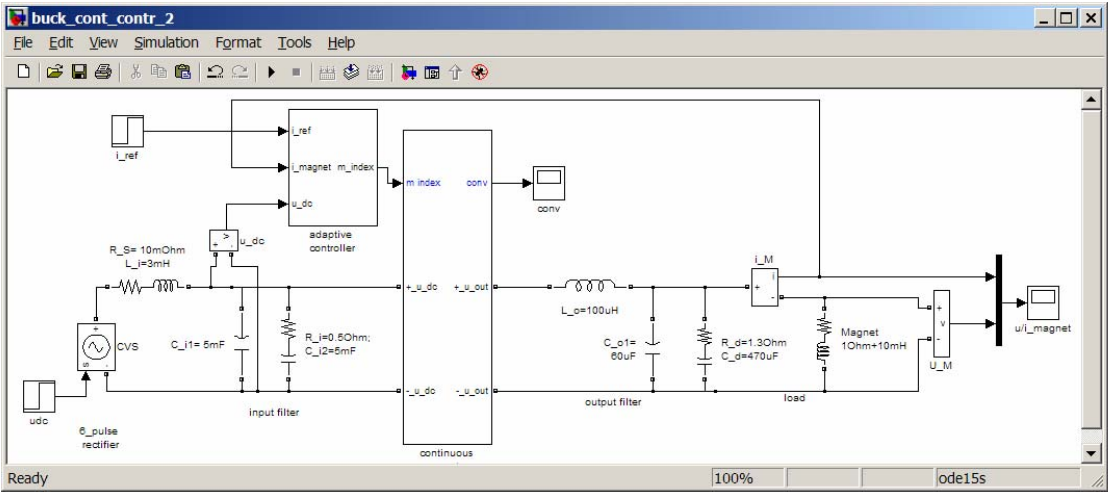


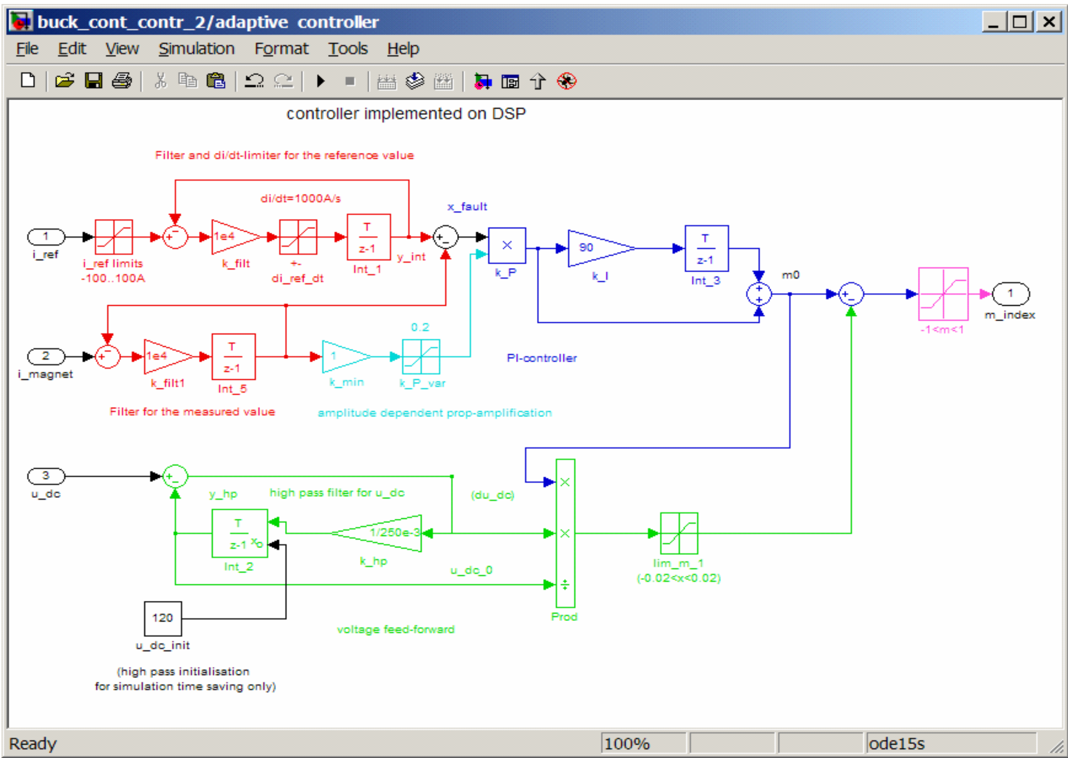


8 'Full' controller

Up to this point the controller parameters have been tested on a linear model with no limits and restrictions.

The following example shows a simplified controller as it is implemented in a large number of power supplies at PSI. This model will be used for demonstrations.





9 Present challenges in simulation

Simulation tools have made big advances in recent years. Nevertheless, further improvement and extension of the capabilities were welcome. Depending on the tool used, some of the following 'wishes' have already been fulfilled:

A) Core of a simulation tool

- Intuitive usage (most engineers do not like manuals)
- Libraries of correct models and elements (as simple as possible)
- Fast models/high simulation speed
- Macros and sub-circuit capabilities
- Good access to all properties of a simulation (at the end of a simulation and during simulation)
- Overall loss calculation of an entire model.

B) Environment

- Toolboxes for additional tasks (analysis, simulation of non-power-electronics tasks, etc.)
- Open to other programs (data import and export)
- Automatic code generation of blocks of a simulation for implementation on other processors (e.g., the implementation of a simulated controller on a DSP in C)
- Extended online help
- User support by the manufacturer of the tool
- Transferability of models between various versions of the (same) simulation program
- Lower prices.