

GeckoCIRCUITS

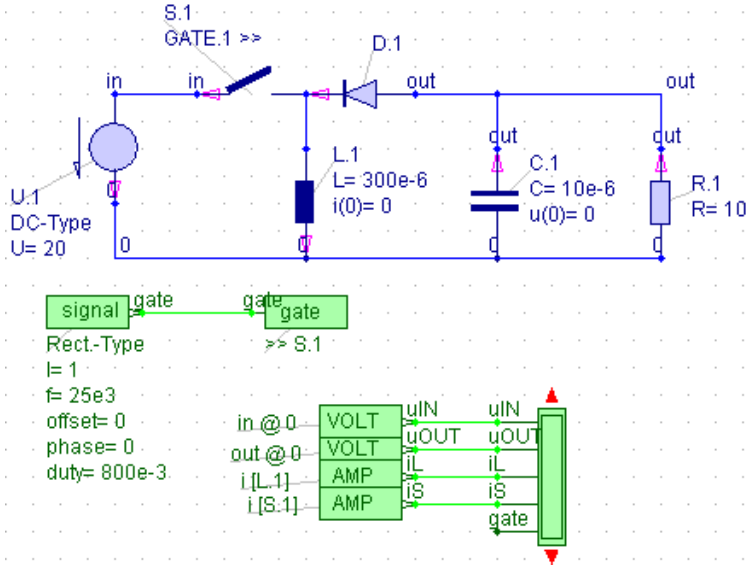
Beginner's Tutorial



User

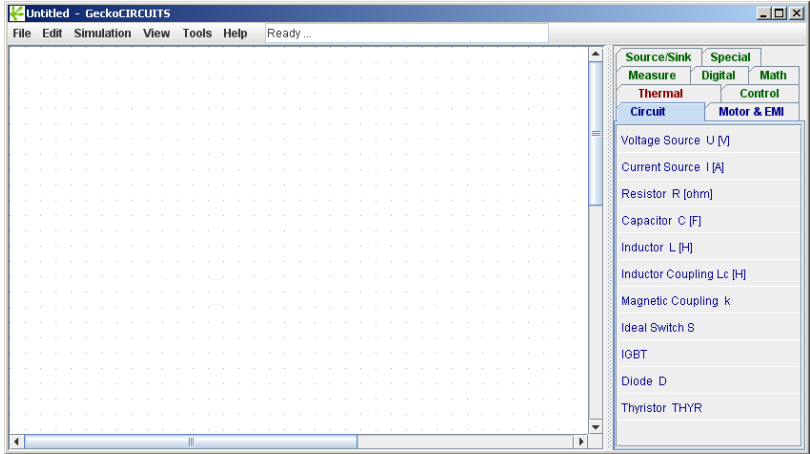
This tutorial is written for users with little or no experience in power electronics simulation. The simulation software GeckoCIRCUITS is explained in detail. Thermal problems, where

GeckoCIRCUITS is especially powerful, are not discussed here in order to keep the tutorial simple.



Buck-boost converter with simple control.

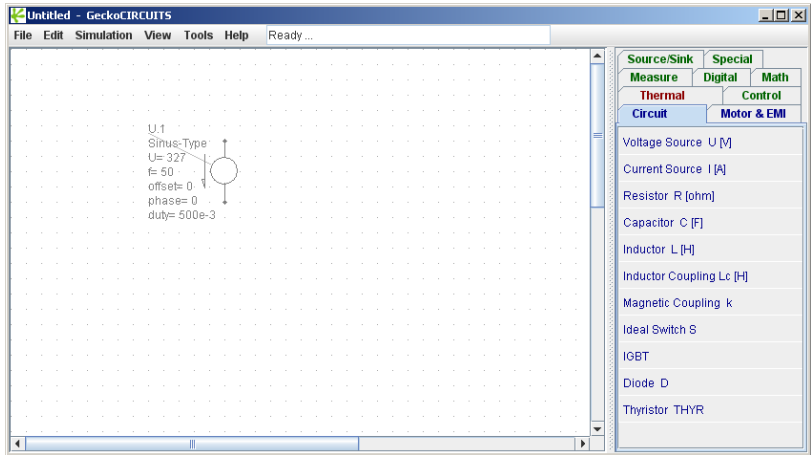
Power Circuit → Blue
Control → Green
Thermal → Red



Circuit simulator GeckoCIRCUITS after starting the program. Depending on the tabs at the right

side one can select components. The power circuit components will be displayed in blue, the control

components in green and thermal components in red.

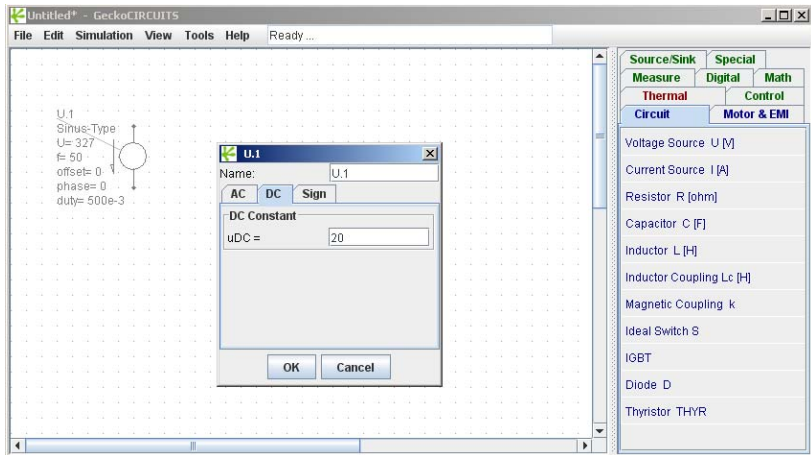


Editing Mode → Gray

Click with the mouse (1x left button) onto *Voltage Source U [V]* located in the blue tab. Afterwards, move the mouse pointer into the worksheet.

The selected voltage source moves with the mouse pointer, and is in the editing mode (gray). The voltage source can be put down to its final location by clicking the

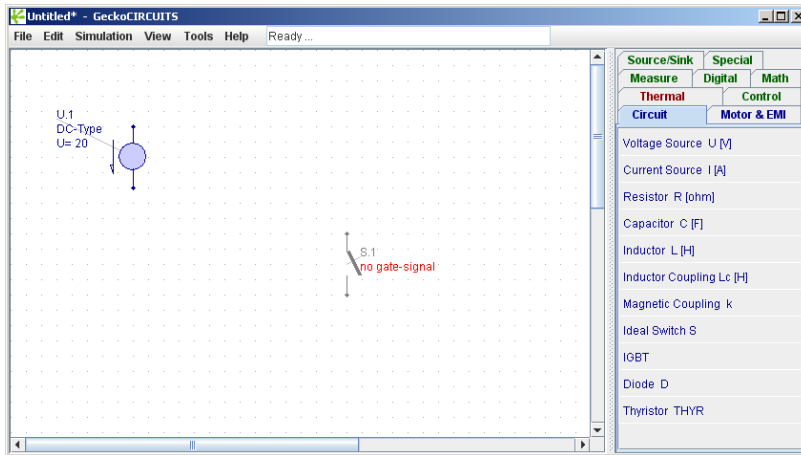
mouse (1x left button). Then, the voltage source exits its editing mode (gray) and is displayed in blue.



Double-click (2x left button) with the mouse-pointer onto the component's symbol to open the

parameter dialog. Sources are set to AC-sources by default. By clicking the *DC-Tab* of the

parameter dialog, the source becomes a DC voltage source. Set the DC-value of the source to 20V.



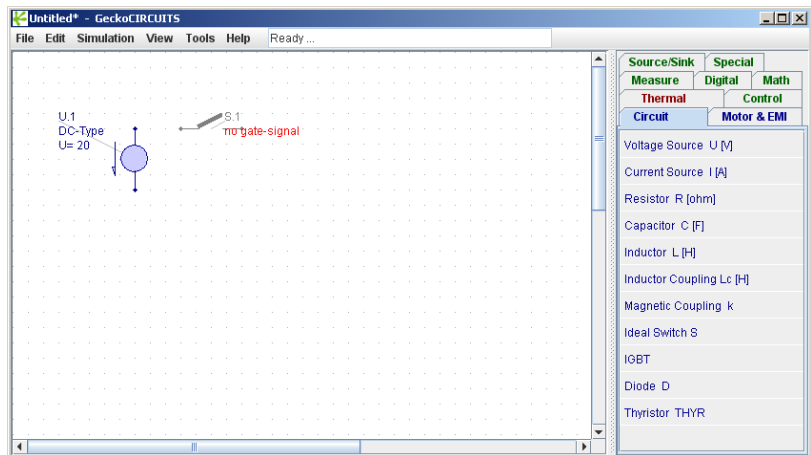
- Ideal Switch:**
- Bidirectional
 - $U_{on} = 0\text{ V}$
 - $r_{on} = 10\text{m}\Omega$
 - $r_{off} = 10\text{Meg}\Omega$

Clicking (1x left) *Ideal Switch S* (power circuit, blue tab) provides an ideal switch with bidirectional behaviour, ON-voltage drop (default: 0V), ON-resistance

(default: 10mΩ) and OFF-resistance (default: 10MegΩ). These parameters can be changed by the user via the component's parameter dialog. The parameter

dialog can be opened by double-clicking (2x left) the component's symbol after the switch has been put down.

Rotate power circuit component in editing mode (gray)



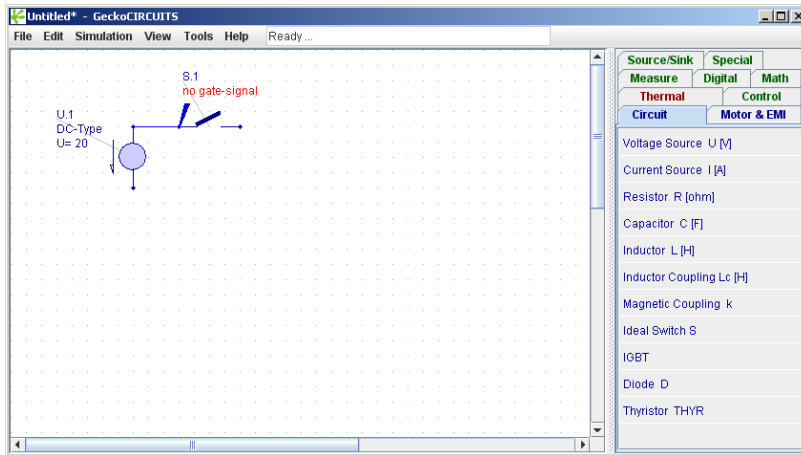
If power circuit components are in their editing mode (gray), they can

be rotated by 90° with each mouse-click (1x right).



The ideal switch (default-name: S.1) is put down with one mouse-

click (1x left), and exits its editing mode.



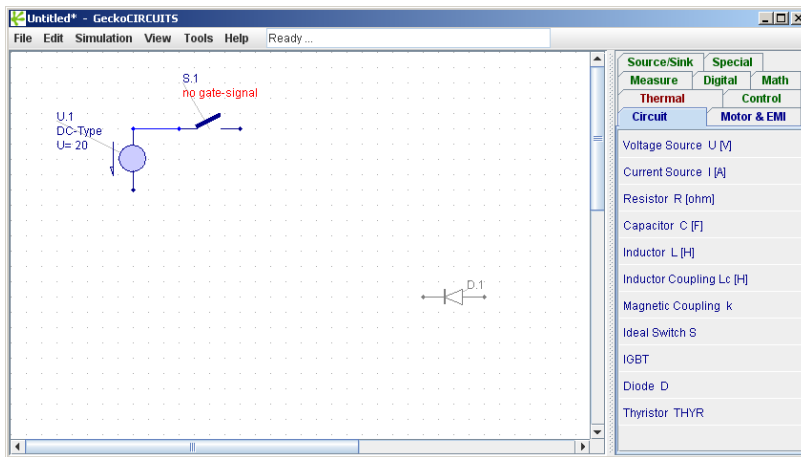
**Blue Tab
& Mouse-Click (1x right)
→ Blue Pen for Drawing
Connections**

**Blue connections can only
connect blue components**

If the blue tab is active (power circuit) with no component in editing mode, a blue pen for drawing connections can be activated by a mouse-click (1x

right). There is a node at start- and end-point of each connection where other connections or input/output-ports of components can be attached. Blue connections

can only connect (blue) power circuit components. A second mouse-click (1x right) deactivates the drawing pen.

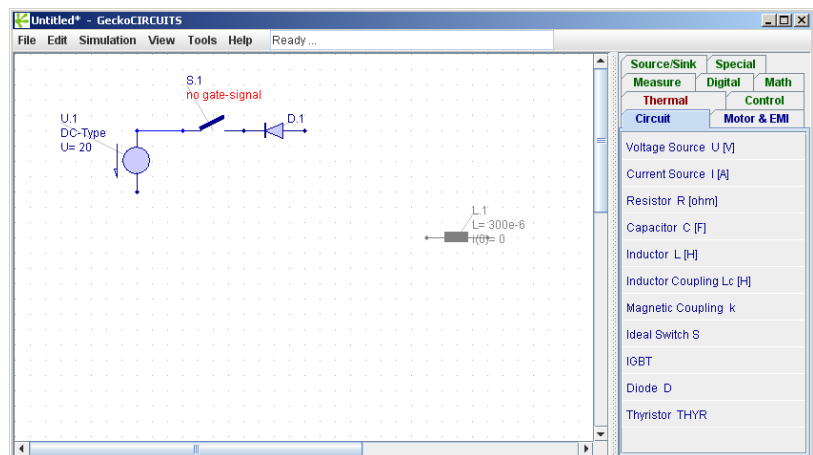


Clicking (1x left) *Diode D* in the blue tab provides a diode with ON-voltage drop (default: 0.6V), ON-

resistance (default: 10mΩ) and OFF-resistance (default: 10MegΩ). These parameters can be modified

in the tab *Characteristic* of the diode's parameter dialog.

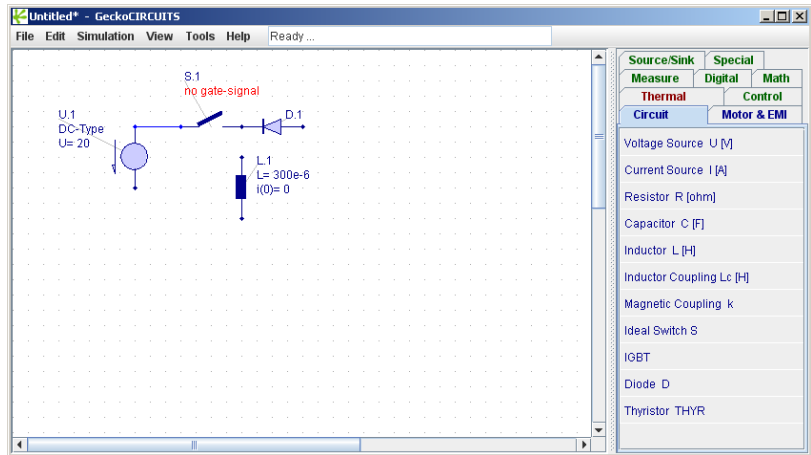
Inductor L gives higher simulation speed, but in case of magnetic coupling Inductor Coupling Lc has to be employed



Now select *Inductor L [H]* from the blue tab. If you want to couple inductors magnetically (e.g. modelling transformers) you have

to employ *Inductor Coupling Lc [H]* instead. The component *Magnetic Coupling k* would provide the coupling. The two components

Inductor L [H] and *Inductor Coupling Lc [H]* are equivalent, but using *Inductor L [H]* might improve simulation speed.



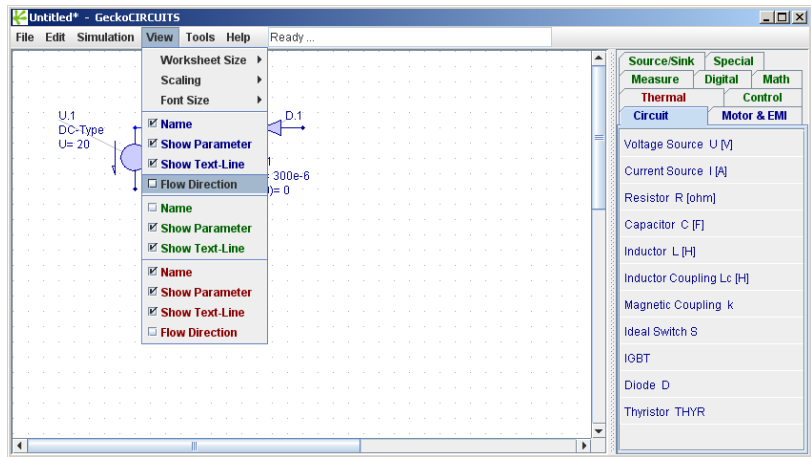
Rotate the inductor by 90° via mouse-click (1x right) in editing mode (gray). Put down the

inductor by mouse-click (1x left). The editing mode is then exited automatically and the inductor

symbol changes its color from gray to blue.

**Menu View >>
Flow Direction:**
Visualize flow direction in power circuit components

→ Current measurement
→ Initial conditions of L, C



Visualize the flow direction in power circuit components via submenu View >> Flow Direction.

This is especially of advantage for current measurement and for setting initial conditions of

inductors (initial current) and capacitors (initial voltage).



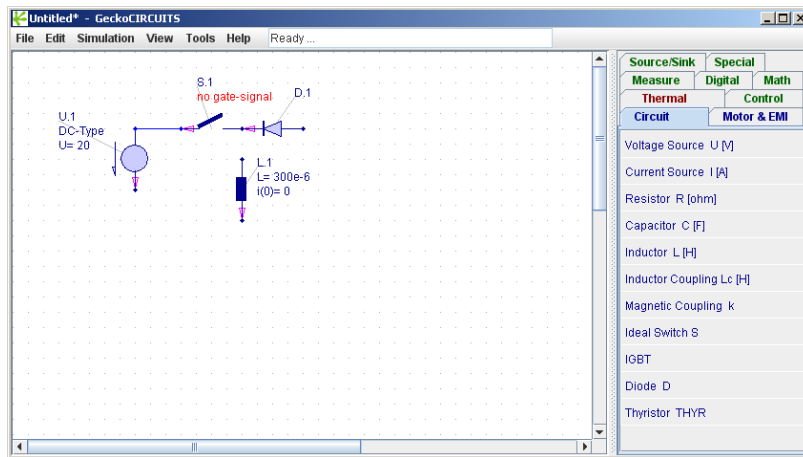
180°-Rotation:

- Mouse-click 1x left → Entering editing mode (gray)
- Mouse-click 1x right → Rotation 90°
- Mouse-click 1x right → Rotation 90°
- Mouse-click 1x left → Component put down (changes to blue)

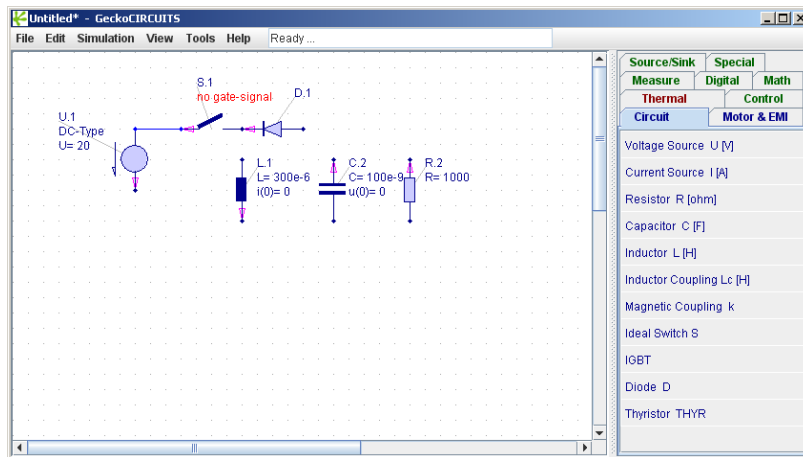
Mouse-click (1x left) the inductor to bring it into its editing mode (gray). Then, rotate the inductor by 180°

via two mouse-clicks (2x right). Put the inductor down by one mouse-click (1x left) which makes the

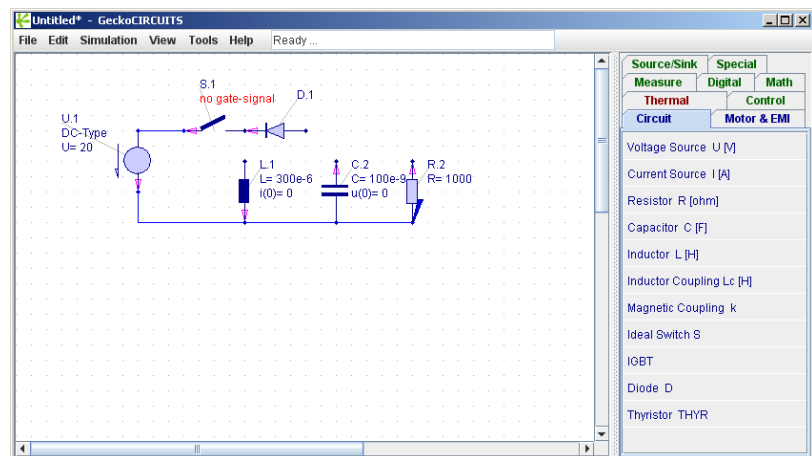
inductor exit its editing mode (changing from gray to blue again).



Visualization of the flow direction: direction of the current flow as
 The magenta arrows show the calculated.



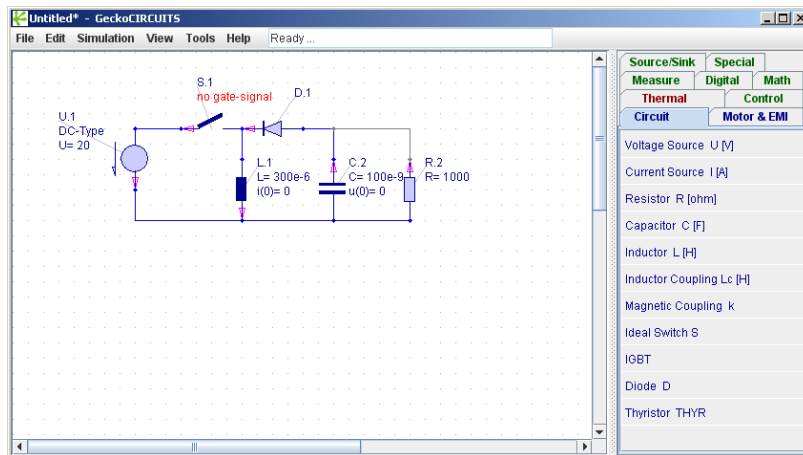
Select a capacitor and a load resistor.



A mouse-click (1x right) gives the drawing pen for drawing connections. Here, the blue tab for

power circuit components has to be active in order to get the blue pen. Only the blue pen can realize

(blue) connections for power circuit components.



Connections in the editing mode:

- Deleting
- Copying

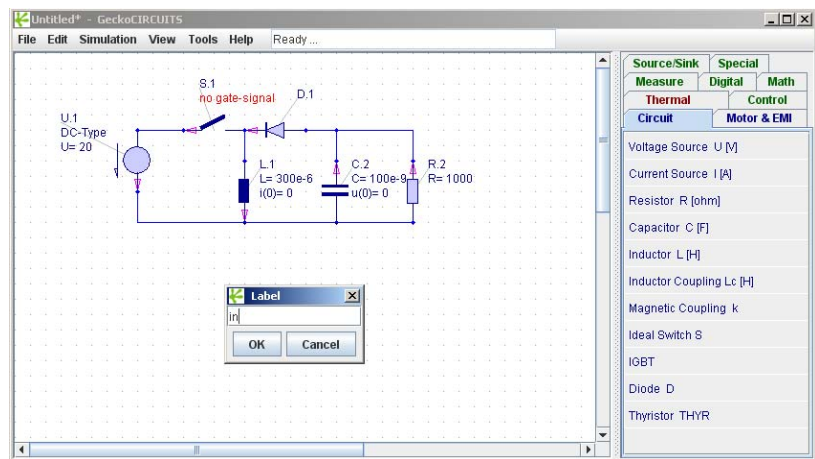
Connections can be changed into editing mode (gray) by one mouse-

click (1x left). Connections cannot be rotated but they can be deleted

and copied like components (see menu Edit).

Connections & Input/Output-Ports of Components

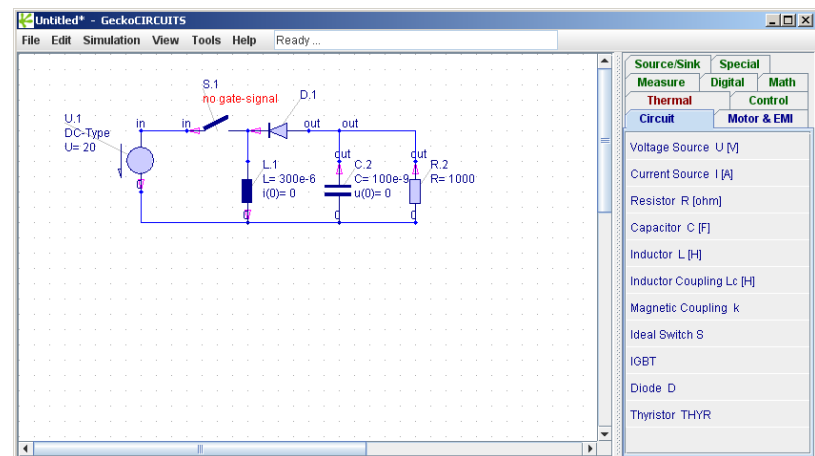
→ Double-click nodes to define node-labels

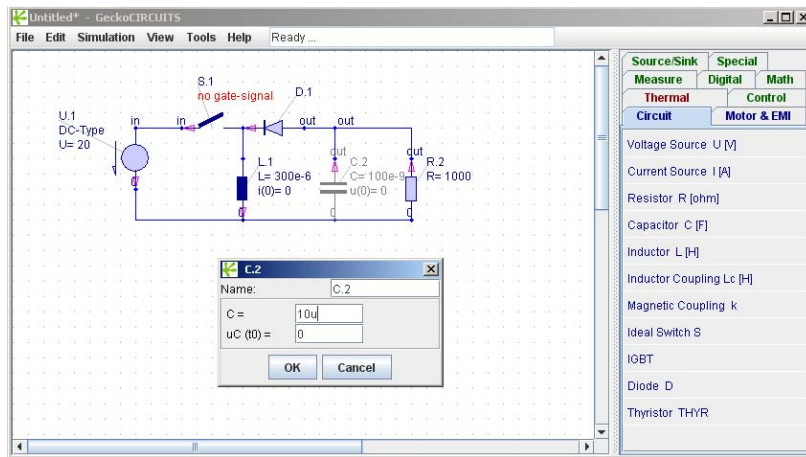


If you double-click connections or input/output-ports of components,

a parameter dialog will open and you can define a node-name.

Node-names are important for e.g. voltage measurement (see later).



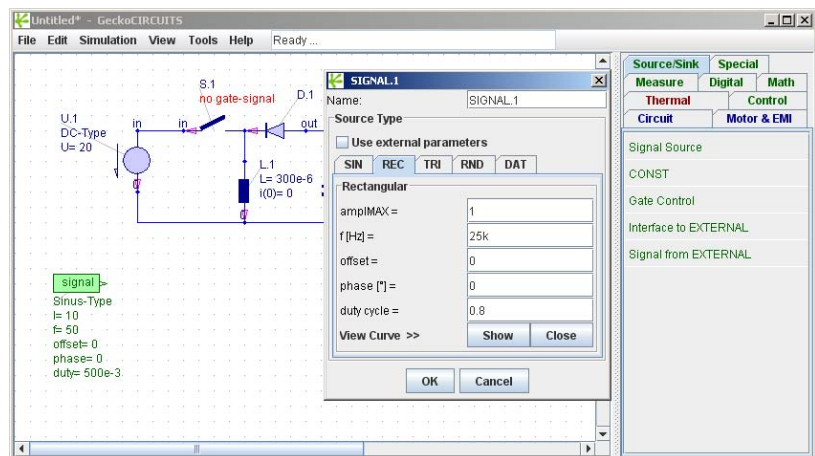


Double-click symbols of R and C to open parameter dialogs for changing component values. Instead of typing $10e-6$ or 0.00001

you can alternatively write $10u$ (also valid abbreviations: p, n, u, m, k). You could also define an initial voltage of the capacitor. This

**Simplified input of values:
Instead of typing
 $10e-6$ or 0.00001
just write: $10u$**

is of advantage e.g. in case of boost-converters, if you would like to omit the simulation of pre-loading.

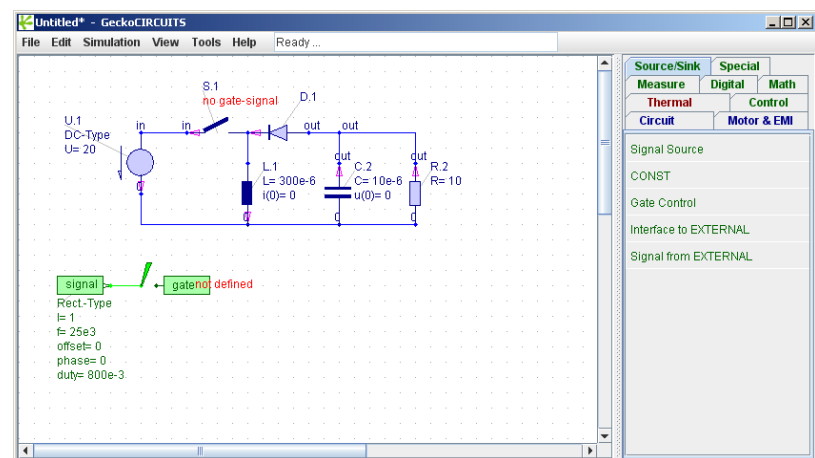


For modelling the control circuit you have to activate one of the green tabs, e.g. *Source/Sink*. By mouse-clicking (1x left) of *Signal Source*, moving the mouse-pointer

into the worksheet, and putting the selected component at its location (mouse-click 1x left to exit the editing mode), you get a general signal-source. Open the parameter

dialog via double-clicking. By selecting the tab *REC* the signal-source will provide a rectangular signal.

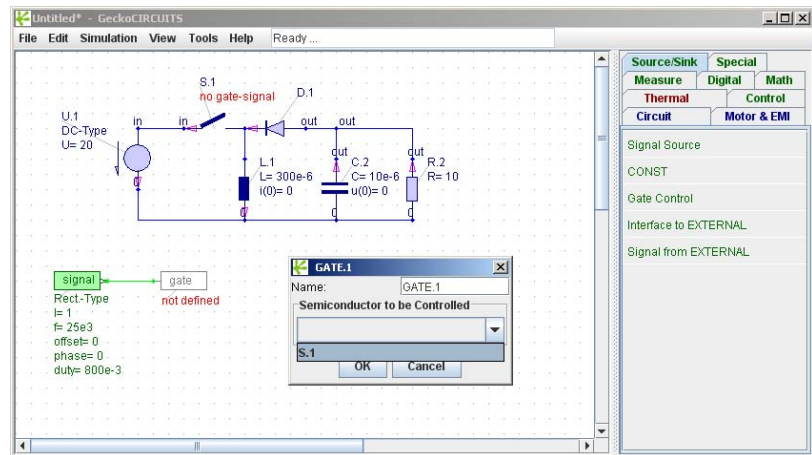
Control active power switches (blue) via the green Gate-Control block



Get the control block *Gate Control* from the tab *Source/Sink* in order to be able to control active switches in the power circuit. This is the only way to control active

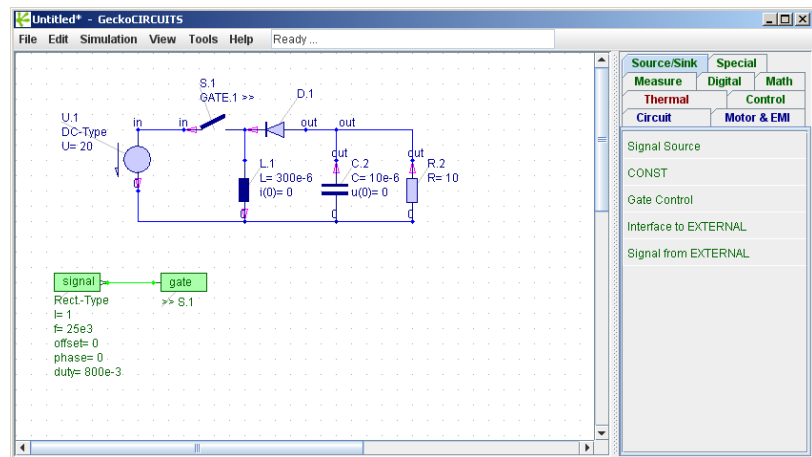
switches. Because the green tab is active, you can get the green drawing pen by one mouse-click (1x right). With the green pen only connections between green

control-blocks can be drawn. Deactivate the green pen by one second mouse-click (1x right).



Double-click the *Gate Control* block to open its parameter dialog. Here, you can select a certain

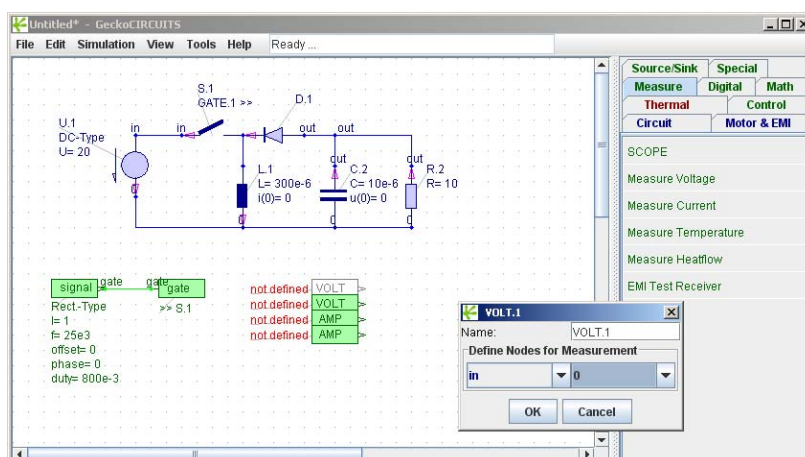
active power switch to be controlled by this block.



After successful selection, the red text *not defined* of *Gate Block* changes to the name of the

selected power switch: *>> S.1*. The red text *no gate-signal* of the selected power switch changes to

the name of the controlling block: *GATE.1 >>*.

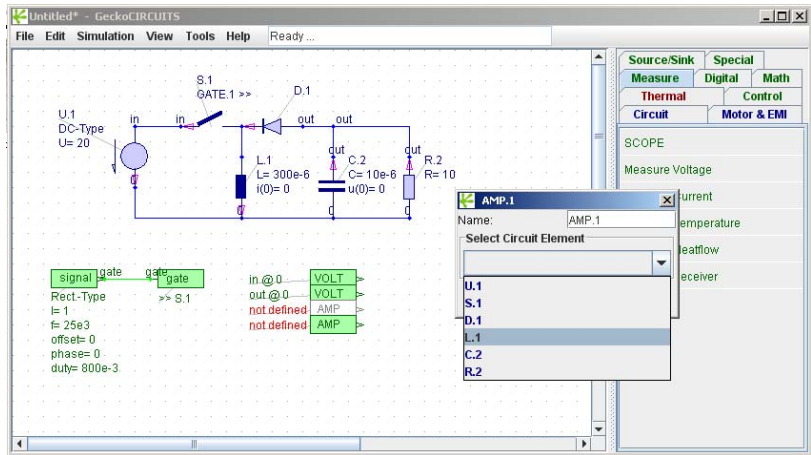


Measure all currents and voltages of the power circuit (blue) via green measurement blocks *AMP* and *VOLT*

To measure currents and voltages you need to employ the control blocks *Measure Voltage* and *Measure Current* in the green tab

Measure. If you double-click the voltage measurement block, the according parameter dialog allows the selection of two nodes, if you

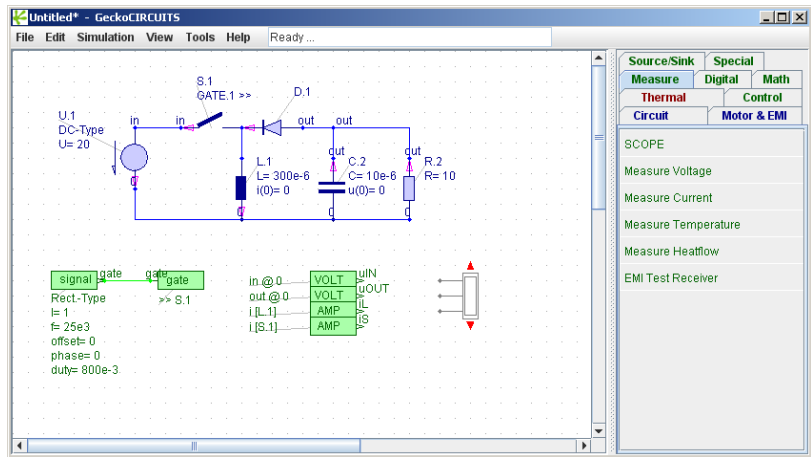
already have defined node-names. If there are no node-names defined, you cannot select and measure voltages.



Double-clicking of the current measurement block opens the according parameter dialog. Here,

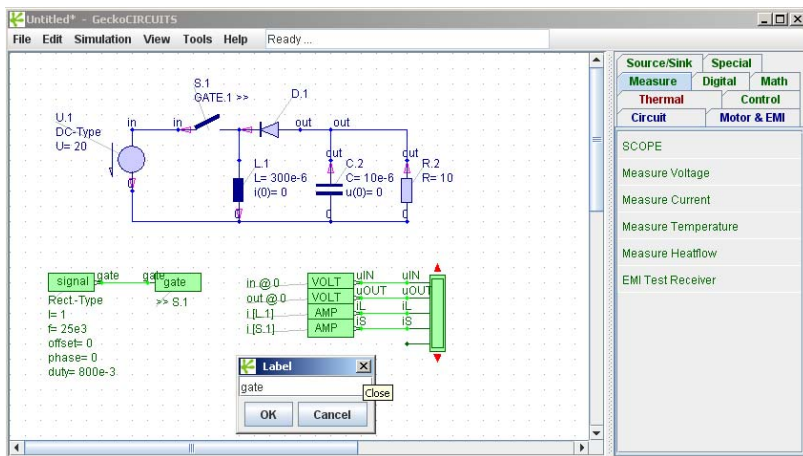
all power circuit components are listed and one can be selected.

Use SCOPE for visualization of simulated time-behavior



Use the signal block *SCOPE* of the green tab *Measure* for

visualization of the simulated time-behavior.

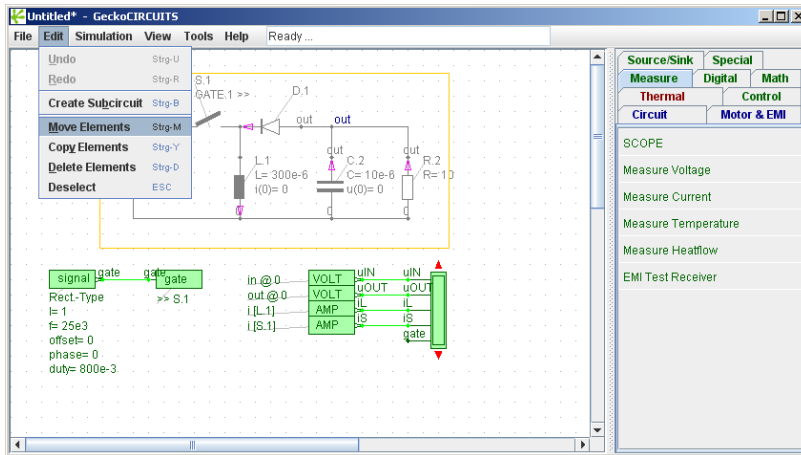


Node-names of the input-ports of the SCOPE are used to name the displayed curves

The number of input-ports of the *SCOPE* can be modified by clicking one of the red triangles (at

top and bottom of the *SCOPE*-symbol). If the input-ports have

names, these names are used to label the displayed curves.



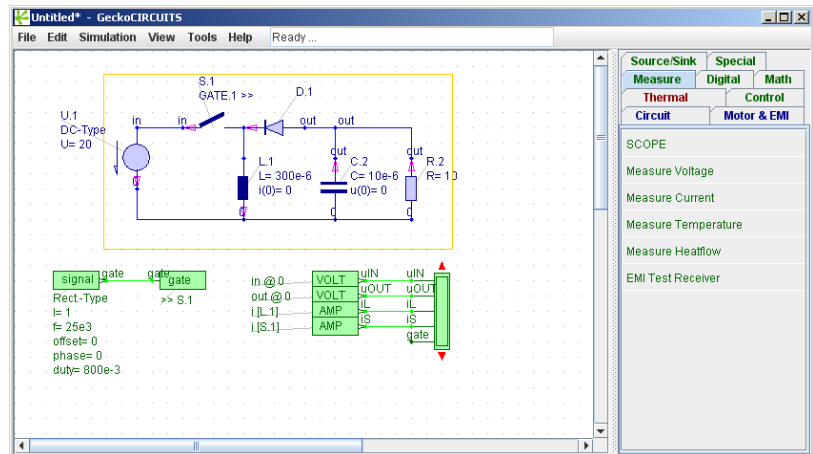
Alternatively: Define connections via identical node-names

Nodes are also connected if they have identical node-names (e.g. the green node-name *gate* in screenshot above). Instead of drawing a (green) connection from the signal-block output port to the

SCOPE input port, you can simply name the according SCOPE input port. If you click with the mouse into the worksheet (1x left), and then drag the mouse pointer (keep left mouse-button pressed while

moving), you can define a yellow rectangle. All components internal of this rectangle change into editing mode (gray) if you release the mouse button.

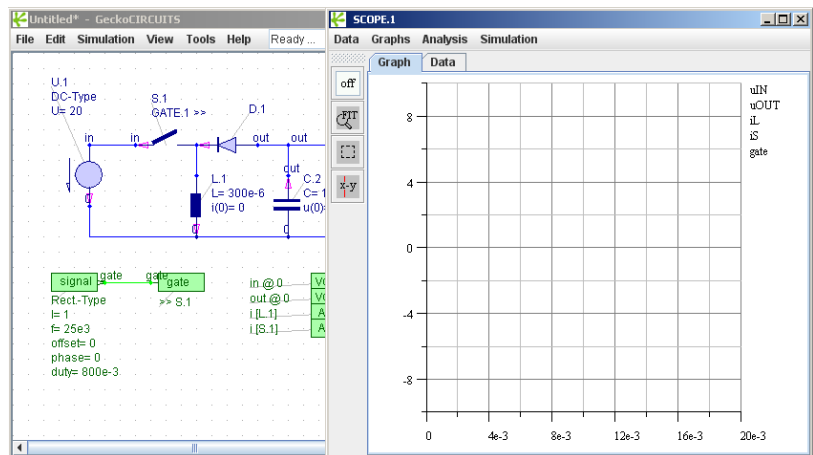
Yellow Selection Rectangle → Put component group into editing mode



Employing the yellow selection rectangle is an easy way to work on a component group efficiently,

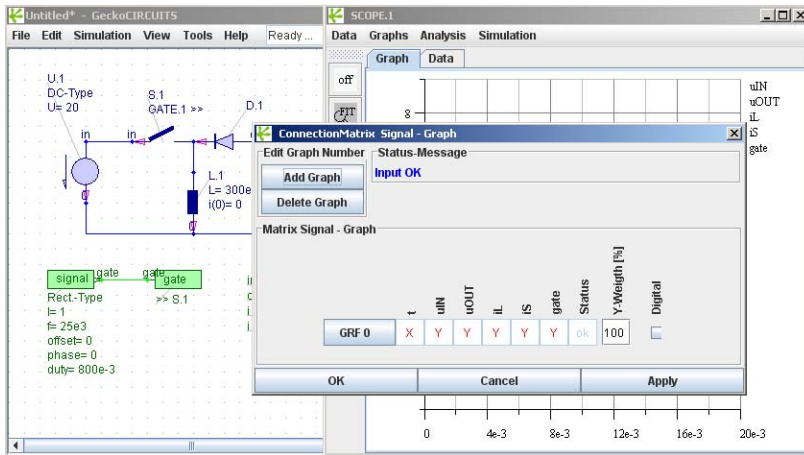
e.g. moving components via the menu Edit >> Move Elements, or

copying and deleting whole component groups.



Double-click the SCOPE-block to open a scope-window. All signals

at the SCOPE input ports can be displayed.

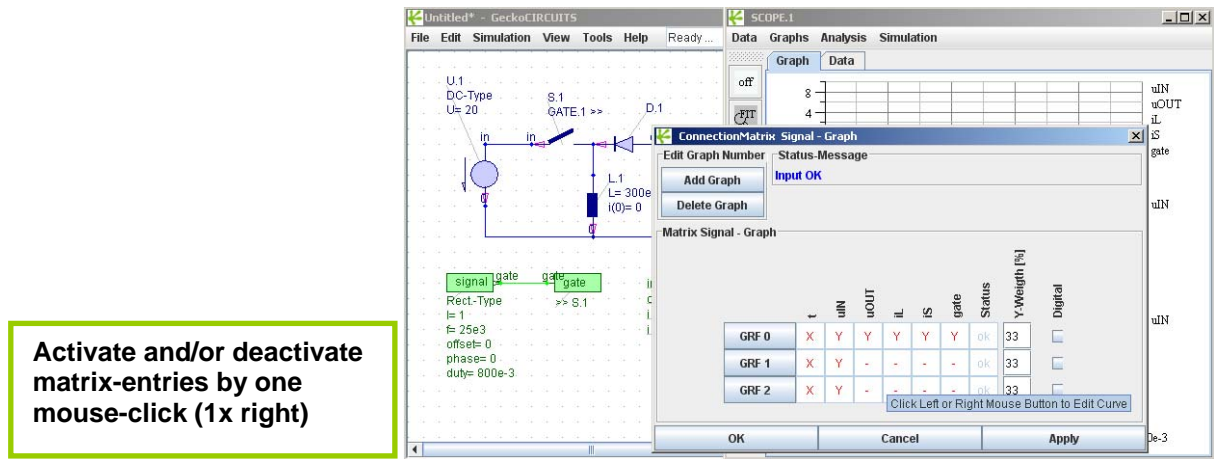


Use the scope-window menu **Graphs >> Signal-Graph** to open the **ConnectionMatrix-Dialog**

Open the dialog *ConnectionMatrix* via the scope-menu **Graphs >> Signal-Graph**, where the number

of displayed graphs can be set (via buttons *Add Graph* and *Delete Graph*). All SCOPE input signals

can be attached to each graph via the matrix structure of this dialog.

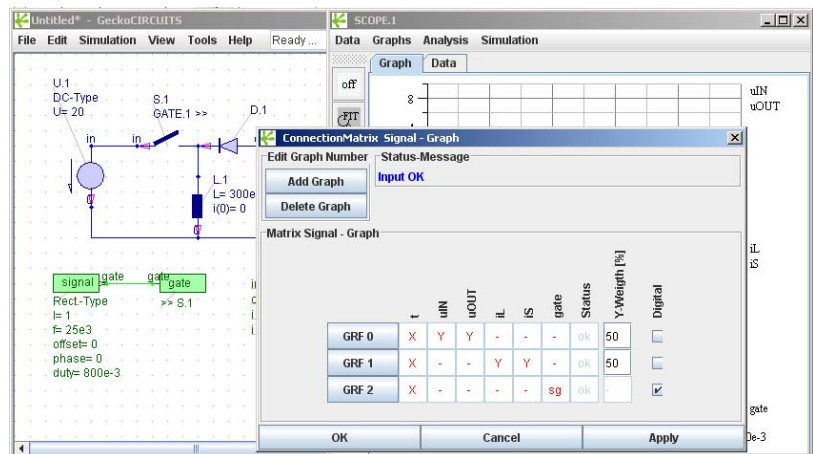


Activate and/or deactivate matrix-entries by one mouse-click (1x right)

Click button *Add Graph* twice to create two more graphs in this scope-window. In the matrix there are now three graphs, furthermore

three signals with entry 'X' plus five signals with entry 'Y' (uIN, uOUT, iL, iS, gate). By mouse-clicking (1x right) these matrix-

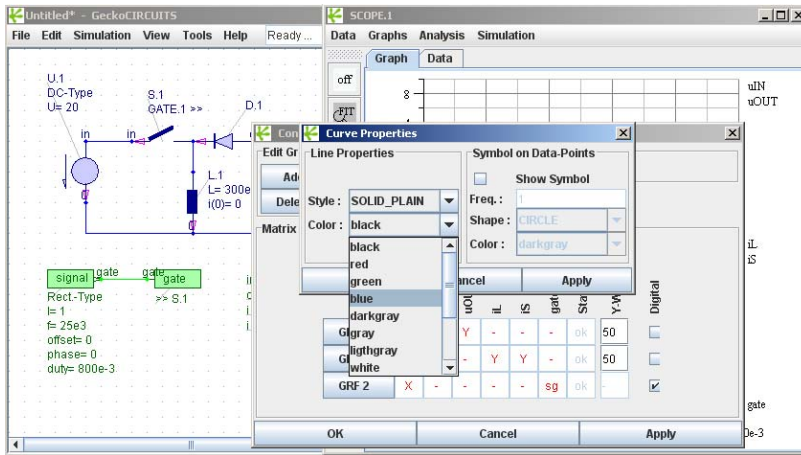
entries can be changed quickly (active → 'Y', inactive → '-').



Set the top graph to display two voltage curves uIN and uOUT, and the second graph to display two current curves iL and iS. The third

graph has to be set as *Digital* (see according checkbox). This is very helpful if a number of digital signals should be displayed which

is a popular option with many modern measurement devices.

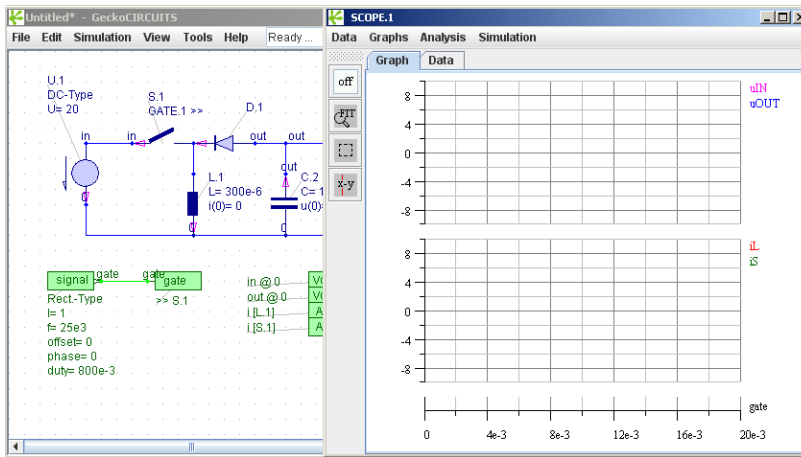


**Matrix-Entry 'Y':
Mouse-click (1x left) →
Dialog for displaying a
single curve**

If you click (1x left) the matrix-entry 'Y', a dialog for editing the curve properties will open. There, you

can define color, line style and/or symbol points of the curve. Optionally, you can change the

properties of the x- and y-axis by clicking the graph buttons (e.g. GRF 1).

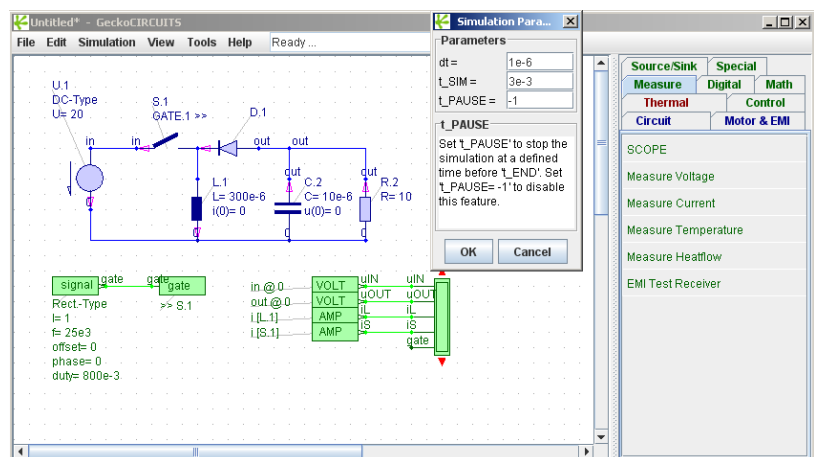


After all properties and parameters have been set, the three graphs (including the third one which has

been set to *Digital*) are displayed. The curve-names (defined by the node-names at the SCOPE input

ports) are shown in the colors selected by the user.

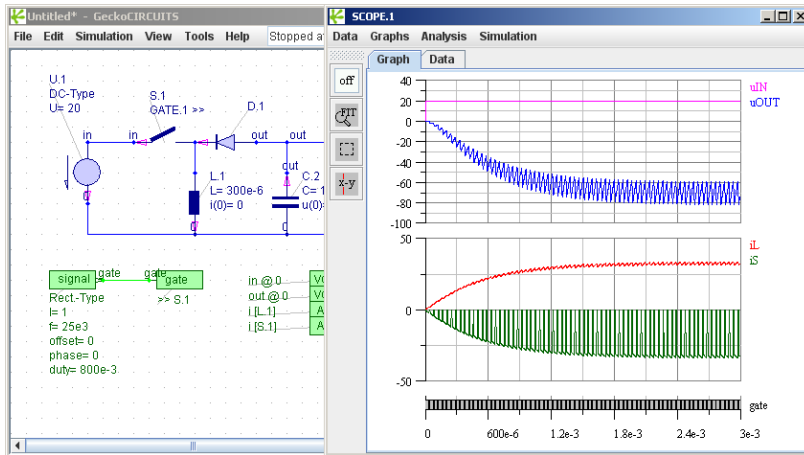
Menu Simulation >> Parameter opens dialog for numerical step-width and simulation time



Open the dialog *Simulation Parameters* via menu Simulation >> Parameter for defining the constant numerical step-width ($dt = 1e-6$) and the duration of the simulation ($t_SIM = 3e-3$).

Optionally you can stop the simulation at a defined time by setting t_PAUSE different from -1 . E.g. $t_PAUSE = 0.001$ would stop the simulation after one third of its total duration. Then, the user could

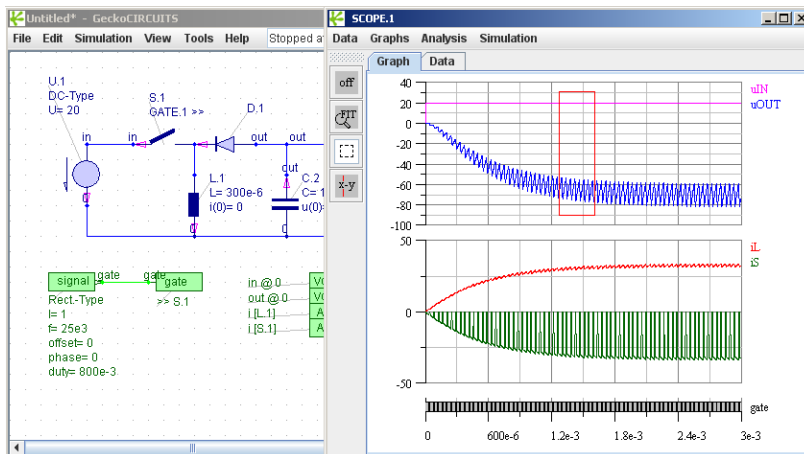
change parameters like load resistor and proceed via menu Simulation >> Continue in order to investigate a load-change.



For the numerical step-width dt (defined in the dialog *Simulation >> Parameter*) you should select a value that is significantly lower than the smallest time-constant of

the system. In case of this DC-DC converter the smallest time to consider is defined by the switching frequency as $T_s = 1/f_s = 1/25\text{kHz} = 40\mu\text{s}$. Therefore, the

maximum value of dt should be around $40\mu\text{s}/100 = 400\text{ns}$. Start the simulation via submenu Simulation >> Init & Start.

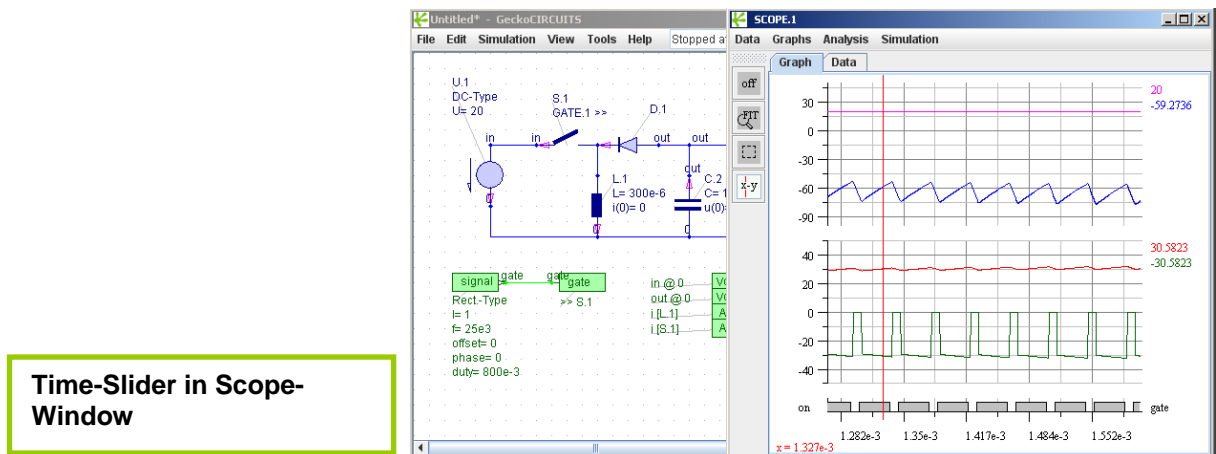


Zoom-Button in Scope-Window

Click the *Zoom-Symbol* (third symbol from top at the menu left-

hand side in the scope-window). Draw a red rectangle to define the

zoom-region by dragging the mouse pointer on the graph.

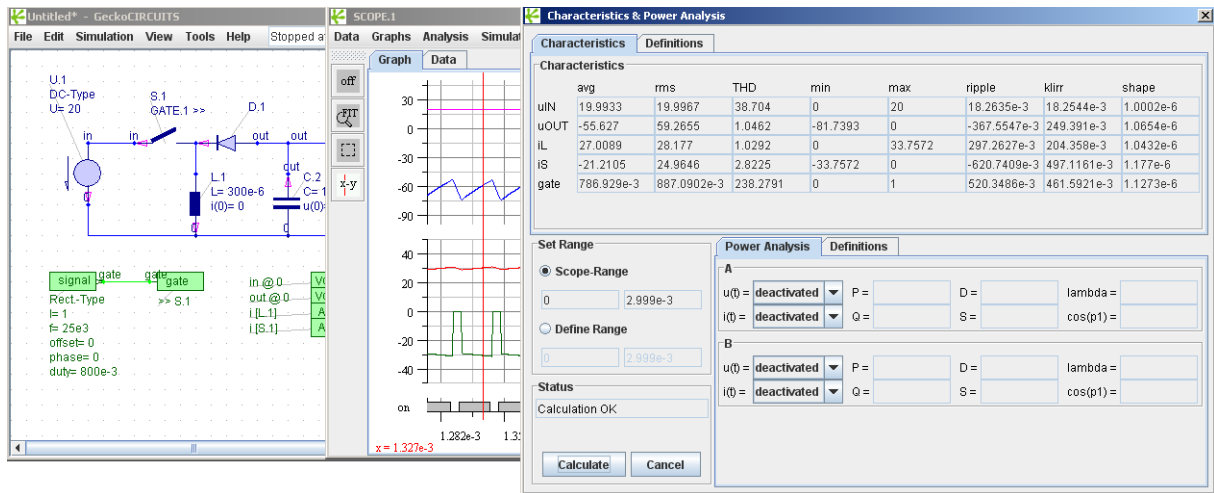


Time-Slider in Scope-Window

The zoomed curves are auto-scaled. By clicking the *Slider-*

Symbols (fourth symbol from top at the menu left-hand side) you

activate the time-slider to show values of the curves.



Use the scope-menu Analysis >> Calc. AVG, RMS to calculate important curve properties like avg- and rms-value. Proceed with the simulation via scope-menu Simulation >> Continue. Before

proceeding parameters can be modified. If, e.g., the load resistor is changed, a load-step-response can be simulated. If you want to save simulation-data in form of an ASCII-file (scope-menu Data >>

Write Data to File), activate data-saving *before* starting the simulation via scope-menu Data >> Activate Data Saving.

Activate scope-menu Data >> Activate Data Saving before simulation

→ Save simulation results as ASCII-file

Information / Feedback

Author: Uwe Drogenik

Date: Nov. 7, 2008

Contact: Uwe Drogenik
Power Electronic Systems Laboratory
ETH Zentrum, ETL H13
CH-8092 Zurich, Switzerland

Phone +41-44-632 4267
Fax +41-44-632 1212
Email drogenik@lem.ee.ethz.ch
uwe.drogenik@gecko-research.com