CONCIRC Version 1.1
User's Manual

Institut für
Theoretische Elektrotechnik

www.tet.tuhh.de

Dr.-Ing. Heinz-D. Brüns, Dipl.-Ing. Angela Freiberg

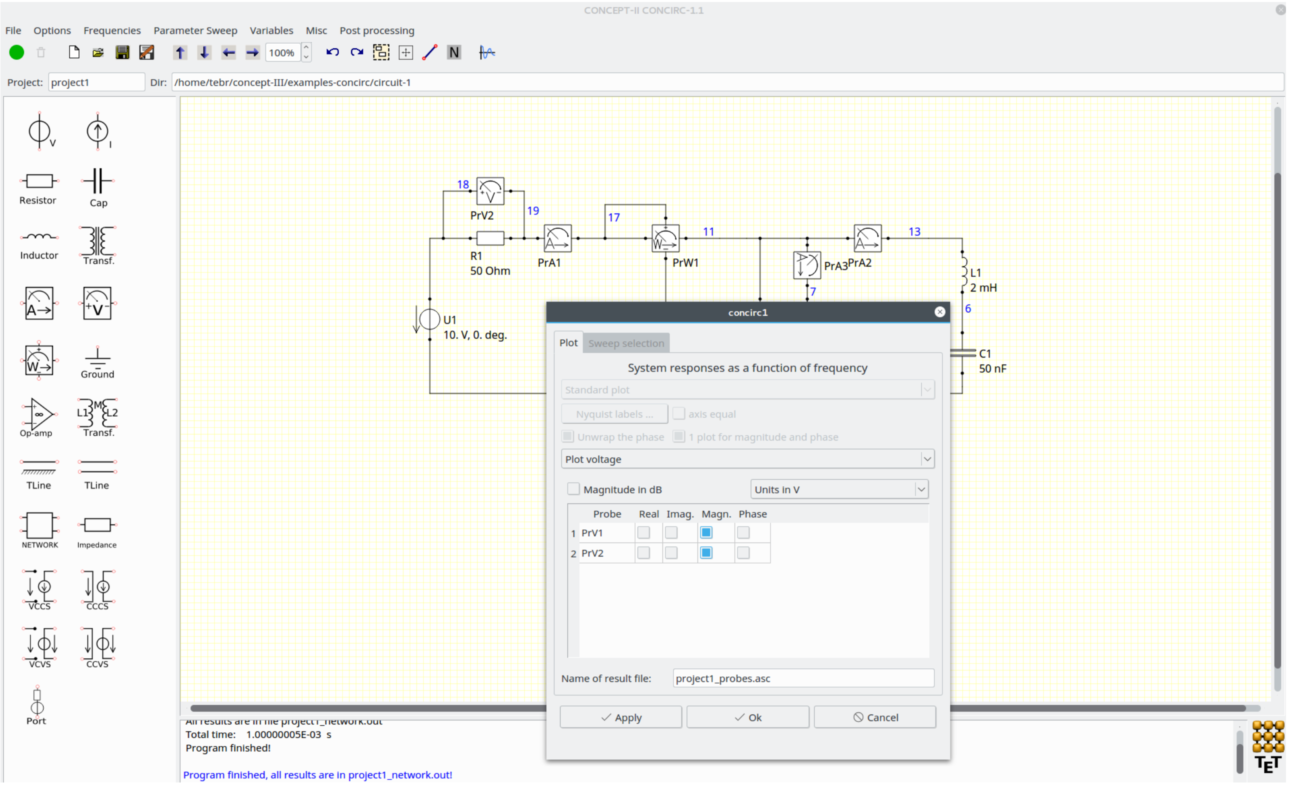
Technische Universität Hamburg-Harburg

- March 2020 -

A General Remarks

CONCIRC is under development at the Institut für Theoretische Elektrotechnik at the Technische Universität Hamburg-Harburg (http://www.tet.tuhh.de/). It is a simple network solver including all basic features necessary to support the students in their studies for the lecture „Electrical Engineering Fundamentals II". The students can use the program to get more insight into the specific behavior of many exercises. *CONCEPT-II* users can place lumped loads or Y-parameter matrices gained by a 3D full-wave field computation into equivalent circuits of extended models. Admittance-parameter matrices can be read in various formats. The program is easy to install. The graphing utility „Gnuplot" http://www.gnuplot.info is used for displaying voltages, currents or impedances as a function of frequency and needs to be installed separately (Note: Gnuplot is not part of *CONCIRC*). The program *CONCIRC* is based on the modified nodal analysis (MNA) [1].

B The GUI



Project: project1 Dir: /home/tebr/concept-III/examples-concirc/circuit-1

System responses as a function of frequency

Plot voltage

☐ Magnitude in dB Units in V

	Probe	Real	Imag.	Magn.	Phase
1	PrV1	☐	☐	☑	☐
2	PrV2	☐	☐	☑	☐

Name of result file: project1_probes.asc

```
All results are in file project1_network.out
Total time:    1.00000005E-03  s
Program finished!

Program finished, all results are in project1_network.out!
```

Fig. B.1: The graphical user interface of CONCIRC

The graphical user interface is depicted in Fig. B.1. The network elements as shown in the left vertical section can be arranged in the working area (region in light green) by drag and drop actions. Note that each equivalent circuit must include a ground, see symbol designated as "Ground" . Tool tips provide information about the circuit elements. Messages which are issued by the program can be inspected in the section below the working area.

The meaning of the icons in the top bar is indicated by tool tips that appear at mouse over. Several completely decoupled circuits with no conductive link can be specified in the working area at the same time.

: Lines can be drawn connecting the circuit elements in the working area

: The Gnuplot front end starts. This can be used for creating images of curves in various graphics file formats

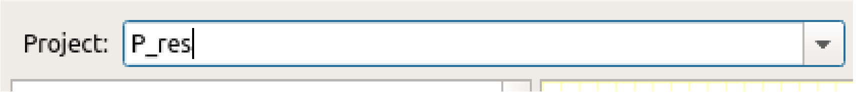
: Parts of circuits or complete circuits can be copied, deleted, rotated or moved (left click and drag, right click)

↑ ↓ ← → : Increase the working area in the indicated direction.

↶ ↷ : Undo, redo

100% : Scaling of the working area.

: If active, left click on an object and drag (the same action: right click on an object → pull-down menu → select **move** → drag)



Project: P_res

: Different Projects can be investigated in the same folder. Switch quickly between them.

B.1.1 Input- Output

Input field "Project" (see Fig. B.2): the default name of the input control file is *project1_network.cir* (ASCII format)

The default notation '*project1'* can be changed to an arbitrary name without blanks.

Normally the default name of the output file is *project1_network.out* (ASCII format). In the case of sweeps it is *project1_network.h5*. Quantities (voltages, impedances etc.) which are displayed on the screen are automatically stored in corresponding files. Such files are ready to be processed by the mentioned Gnuplot front end in order to create advanced graphical representations.

Example: *test_PrV1.as*c. We have '*test' as* name of the project, *PrV1*: voltage probe nnumber 1. The format of each .asc (ASCII) file is explained in the head section.

B.2 Working with the GUI

General procedure:

- Set up the equivalent circuit by drag and drop operations

  All lumped elements of a circuit have to be connected by lines. Right-click on an element provides the options according to Fig. B.2:

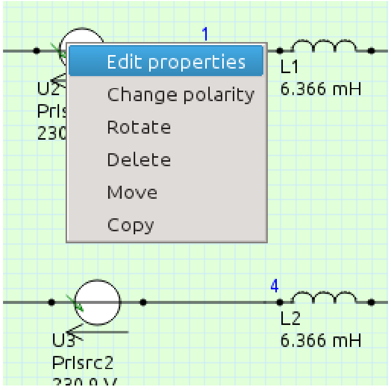
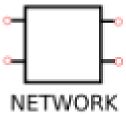


Fig. B.2: A right click onto an element in the working area provides a menu as depicted.

It is easy to move or rotate elements.

- 

: **Edit properties** → **Get port/ MTL file** → Load Y parameter matrix.

CONCEPT-II → Reduced Y parameter matrix, *red-y-mat.h5* (alternatively Touchstone format, *s-matrix snp*).

CONMTL →*project1_mtl_ymat.h5* , default name of the Y parameter matrix of a transmission line computed by the program *CONMTL*, see web page of the *Institut für Theoretische Elektrotechnik*. Notice: 'Port arrangement' , **(Multiconductor) Transmission line** . Do not forget to set the 'Number of ports' accordingly.

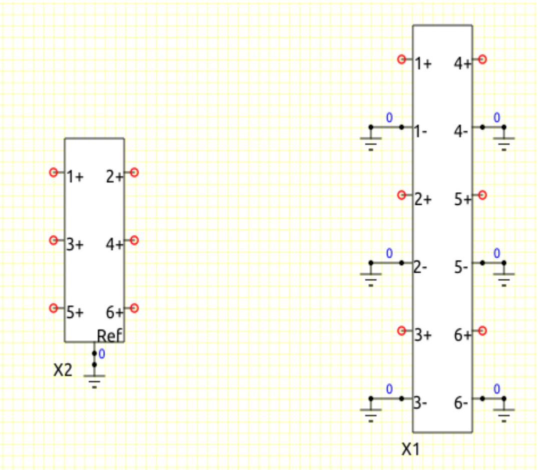
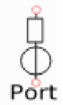
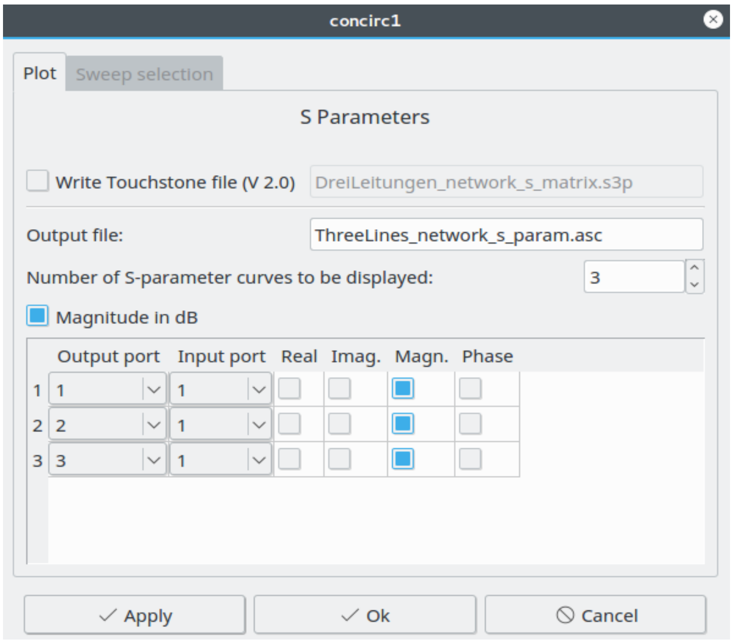


Fig. B.3: Example of a three-wire multiconductor transmission (MTL) line; there are three wires and a reference conductor, forming 6 ports. (1+: beginning of conductor 1, 2+ end of conductor 1; 3+,4+ beginning and end of conductor 2, etc.) The left symbol is the standard one for a MTL. The right symbol can also be used; see examples-concirc/Two-Port-Series-Connection.

- : Port symbol for the computation of S-parameters. Note that other sources are not allowed if a port symbol is specified in the circuit. No other quantities than S-parameters will be computed.



S Parameters

☐ Write Touchstone file (V 2.0) DreiLeitungen_network_s_matrix.s3p

Output file: ThreeLines_network_s_param.asc

Number of S-parameter curves to be displayed: 3

☑ Magnitude in dB

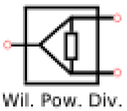
	Output port	Input port	Real	Imag.	Magn.	Phase
1	1	1	☐	☐	☑	☐
2	2	1	☐	☐	☑	☐
3	3	1	☐	☐	☑	☐

Fig. B.4: S-parameter selection

After the computation of S-parameters has finished a window according to

Fig. B.4 opens: Select the number and the desired port combination for plotting → click **Apply.**

An example is described in Sec. B.8.

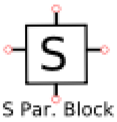
- 

: Symmetric Wilkinson Power divider. Implemented S parameter matrix:

$$[S]=\frac{-j}{\sqrt{2}}\begin{bmatrix}0 & 1 & 1\\ 1 & 0 & 0\\ 1 & 0 & 0\end{bmatrix} .$$

Input parameter: complex reference impedance.

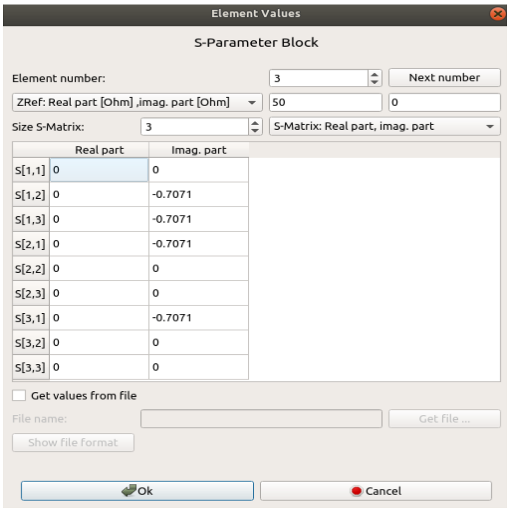
For a frequency dependent circuit notice *examples-concirc/Wilkinson* .



: S parameter matrices are directly implemented into the MNA procedure

(extended stamping). For a simple application of the element symbol see *examples-concirc/Power-combiner-circuits* , bottom circuit) .

Considering S parameters in this way is especially useful if a Y parameter representation does not exist. This is the case for the Wilkinson power divider for example.



Element Values

S-Parameter Block

Element number: 3

ZRef: Real part [Ohm] ,imag. part [Ohm]: 50, 0

Size S-Matrix: 3

S-Matrix: Real part, imag. part

	Real part	Imag. part
S[1,1]	0	0
S[1,2]	0	-0.7071
S[1,3]	0	-0.7071
S[2,1]	0	-0.7071
S[2,2]	0	0
S[2,3]	0	0
S[3,1]	0	-0.7071
S[3,2]	0	0
S[3,3]	0	0

☐ Get values from file

File name:

Fig. B.5: A right mouse click on the S parameter block symbol provides this window. The size of the S matrix is currently limited to 16 (sixteen ports). Frequency-dependent S parameters can be read from a file.

-
Set comment
: Right clicking on an empty region of the working area provides the possibility to write text.
- Specify the properties (values) of each element by right clicking onto the element → **Edit properties**.
- Note that clicking on a connecting line (a node) enables to choose the node voltage as output quantity.
- Assign the frequency sampling: **Frequencies**, the “Frequencies” window opens
- Note: Frequency dependent data for elements in the working area can be read from files (Examples: voltage source, current-controlled voltage source, impedance…) .

  In these files (ASCII format or .h5 format) the frequency has to be specified at the beginning of each line. The deviation of frequency samples taken from external files with respect to the internal frequency sampling is not allowed to be larger than a limit that can be set under the “*Frequencies*” input window (input field: “Allowed frequency deviation”, default is 10^{-4}).
- Start the simulation with button ●
- All computed results are stored in the file *project1_network.out* in case of no sweeps. In case of sweeps they are stored in the file *project1_network.h5* in HDF5 format. In case of S-parameters they are stored in *project1_network_s_param.h5.* ‘*project1*’ indicates the current project name. All frequency samples are contained in *netfreq.in.* Control files are *project1_network.cir* (necessary for using the GUI) and *project_network.net* which contains the netlist only. Both files can be used to start the program at the command line, see Sec. B.5.

- Postprocessing

  After the computer run has finished a window opens as illustrated in Fig. B.7 . “Pr” stands for probe, examples:
  - PrVn → Output of voltmeter with number n
  - PrIsrn → Current through voltage source with number n(activate by **Edit properties**)
  - PrAn → Output of ammeter with number n
  - PrPhin → Node potential of node with number n

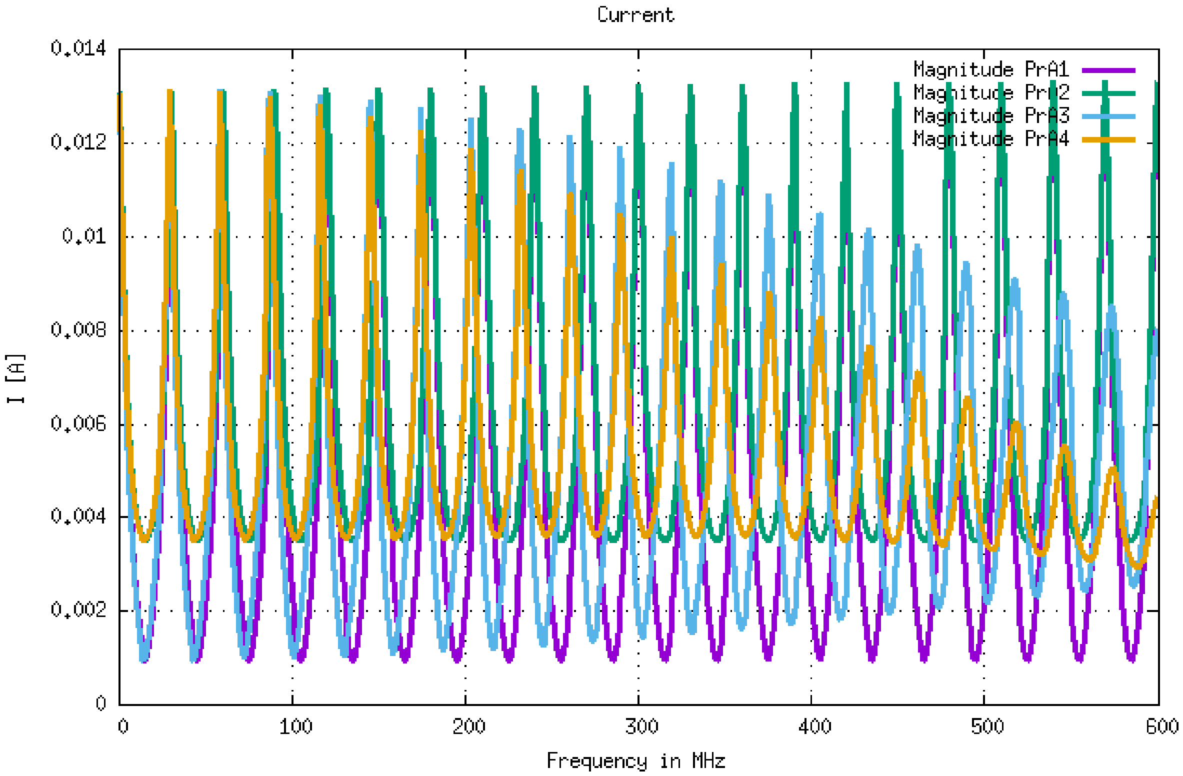
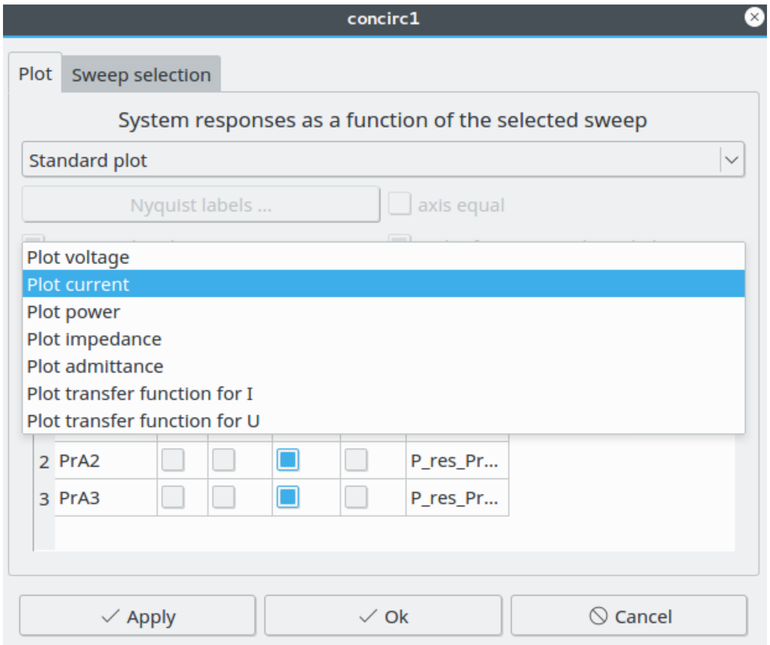


Fig. B.6: Example, demonstrating the output of 4 ammeters as a function f frequency.

- **Plot impedance** → Choose the corresponding voltage probe (*Probe U*) and current probe (*Probe I*). Similar: **Plot admittance**, **Plot transfer function for I**, **Plot transfer function for U.**
- All Plots can be produced as **Standard plot**, **Nyquist plot** or **Bode plot.** For an improved curve representation click on → the Gnuplot front end opens → line style, width, color, text size, ranges, log. scale etc. can be chosen and several output formats such as png are available. Real parts, imaginary parts, magnitudes and phases of system responses can be displayed as a function of frequency.



System responses as a function of the selected sweep

Standard plot

2	PrA2	□	□	☑	□	P_res_Pr...
3	PrA3	□	□	☑	□	P_res_Pr...

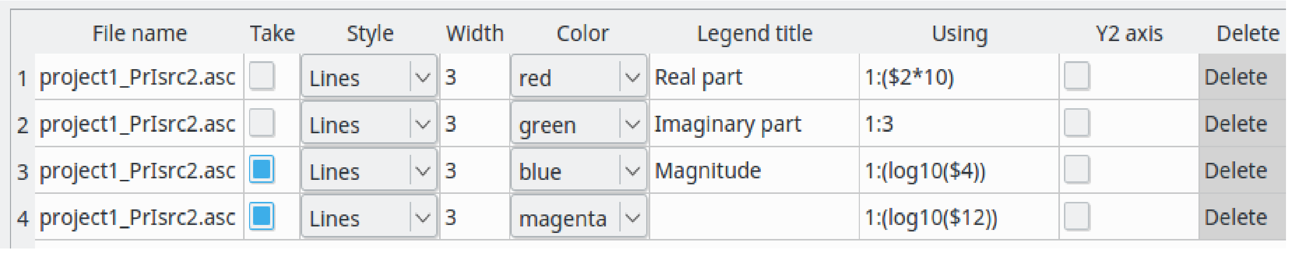
*Fig. B.7: Once the simulation has finished the post processing window appears (**Post processing** → **Show frequency domain values**).*

*Clicking on the top bar additional options are available: **Nyquist plot** and **Bode plot.** Especially for Nyquist plots various options exist for optimizing the curve representation.*

Fig. B.8 demonstrates some possibilities.

Hovering the mouse pointer over the ‘*Using’* input fields provides information about the columns that are contained in the file listed under ‘*File name’.*

In the example case four columns were chosen, only columns 4 and 12 will be considered for plotting, notice the check marks under ‘*Take*’. As columns 4 and 12 are representing the magnitudes, the corresponding logarithmic values will be shown as a function of frequency (the frequency is always listed in column 1). In the first row of Fig. B.8 all real parts, represented by column 2 (referred to as $2) are multiplied by a factor of 10 and in the third row the imaginary parts (column 3) are ready for plotting if the check marks under ‘*Take*’ were enabled. For more information on row editing notice the Gnuplot documentation.

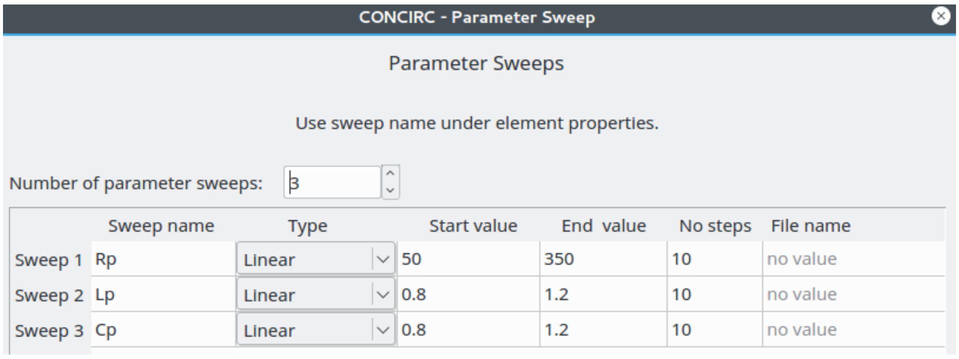


	File name	Take	Style	Width	Color	Legend title	Using	Y2 axis	Delete
1	project1_PrIsrc2.asc	□	Lines	3	red	Real part	1:($2*10)	□	Delete
2	project1_PrIsrc2.asc	□	Lines	3	green	Imaginary part	1:3	□	Delete
3	project1_PrIsrc2.asc	☑	Lines	3	blue	Magnitude	1:(log10($4))	□	Delete
4	project1_PrIsrc2.asc	☑	Lines	3	magenta		1:(log10($12))	□	Delete

Fig. B.8: Curve selection by using the Gnuplot front end of CONCIRC

B.2.1 Plot of sweep results

Before running the network simulation the parameter sweeps have to be defined:



CONCIRC - Parameter Sweep

Parameter Sweeps

Use sweep name under element properties.

Number of parameter sweeps: 3

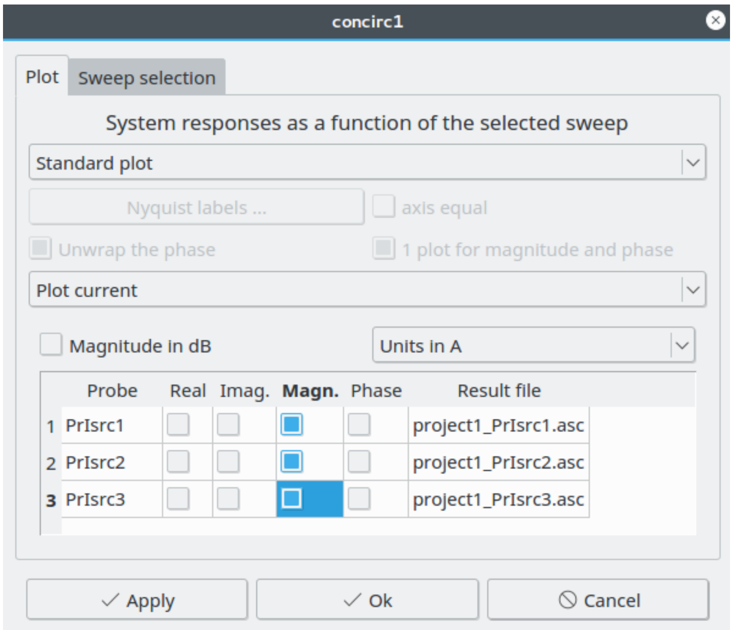
	Sweep name	Type	Start value	End value	No steps	File name
Sweep 1	Rp	Linear	50	350	10	no value
Sweep 2	Lp	Linear	0.8	1.2	10	no value
Sweep 3	Cp	Linear	0.8	1.2	10	no value

Fig. B.9: Here three parameter sweep are defined where each sweep comprises 10 steps

click on **Parameter sweep** → a window as illustrated in Fig. B.9 opens. By using the **Edit properties** function (right mouse click on elements in the working area) these sweep names can be assigned to any input value.

After the network computation has finished (may take a while due to a possibly huge number of data sets to be stored) the window according to Fig. B.10 opens, showing an example for the probes *PrIsc1, PrIsc2, PrIsc3* (**Plot current**). Clicking on **Apply** without taking further choices under the tab **Sweep selection** will display three curves showing the magnitude of the currents as a function of frequency. Each curve is referring to the corresponding first parameter value. This is the default setting.

If no frequency sweep is available, a parameter sweep has to be selected first under **Sweep selection** (see tab).



System responses as a function of the selected sweep

Standard plot

□ axis equal

□ Unwrap the phase

□ 1 plot for magnitude and phase

Plot current

□ Magnitude in dB

Units in A

	Probe	Real	Imag.	Magn.	Phase	Result file
1	PrIsrc1	□	□	☑	□	project1_PrIsrc1.asc
2	PrIsrc2	□	□	☑	□	project1_PrIsrc2.asc
3	PrIsrc3	□	□	☑	□	project1_PrIsrc3.asc

*Fig. B.10: Plot selection window. Can be opened by **Postprocessing** -> **Show frequency domain values***

Various curve representations are possible, click on the tab **Sweep selection.** A window as depicted in Fig. B.11 opens.

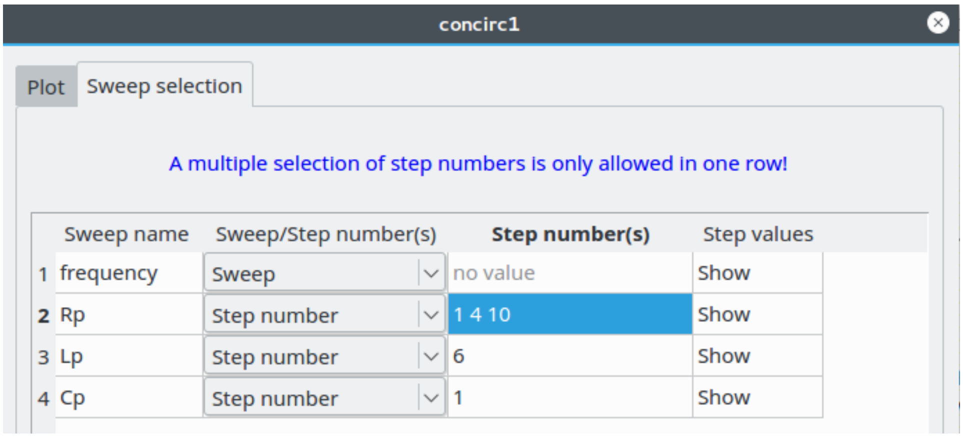
As can be seen, the frequency is selected for a sweep (column ‘*Sweep, step number(s)’). N*otice that:

- Any parameter variable that is listed could be chosen for a sweep
- Only one ‘*Sweep name’* can be chosen for a sweep at a time
- One further ‘*Sweep name’* can be chosen to include several discrete steps (integer numbers indicating the samples)

  The number of curves that will be displayed depends on the number of these steps and on the number of probes that were set to active according to Fig. B.10
- For a quick orientation a list of the step numbers will be shown when clicking on **Show** (end of each row).

As illustrated in the example, the parameter *Rp* is assigned to the steps 1 4 10, whereas *Lp* refers to a value according to step 6 and Cp refers to its first value at the same time.

Hence nine curves would be displayed as a function of frequency under the precondition that all probes are set to active according to Fig. B.10.



A multiple selection of step numbers is only allowed in one row!

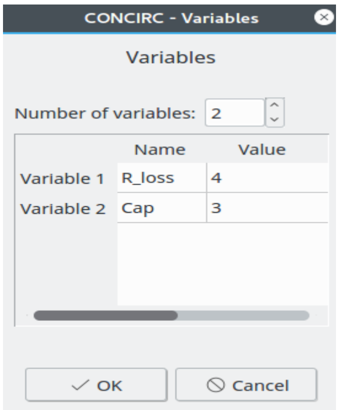
	Sweep name	Sweep/Step number(s)	Step number(s)	Step values
1	frequency	Sweep	no value	Show
2	Rp	Step number	1 4 10	Show
3	Lp	Step number	6	Show
4	Cp	Step number	1	Show

Fig. B.11: Note that only a single sweep can be selected at once.

For a sweep example refer to Sec. B.7.

B.3 Setting of Variable Names

Menu bar of the GUI: **Variables** → Variable names can be entered as demonstrated in Fig. B.12



Number of variables: 2

	Name	Value
Variable 1	R_loss	4
Variable 2	Cap	3

Fig. B.12: Example of two variable names

These names can be used to specify the numerical values of elements of the circuit in the working area (right mouse click, **Edit properties**). A single name can be used for several elements, the unit is taken as set for the corresponding element value.

B.4 Numerical Background

The program is based on the MNA (Modified Nodal Analysis) technique as originally introduced by Ruehli et al [1]. The code is based on the original work of S. Skibin [2].

B.5 Usage without GUI

Starting the executable at the command line is possible by this command:

```
concirc1   -nc   filename.cir
```

The .cir file is generated when using the graphical user interface (at least one previous successful run is useful for this application). Alternatively can be used:

```
concirc1  -nn  filename.net
```

The .net file is generated when using the graphical user interface (at least one previous successful run is useful for this application.) This file contains the net list in a *CONCIRC*-specific notation.

B.6 Transient responses

CONCIRC is working in the frequency domain. For time-domain system responses it is necessary to compute a series of sample frequencies (frequency loop) and to perform a subsequent inverse Fourier transform (IFT).

Note	Choosing an appropriate frequency sampling is essential for a successful simulation!

The whole solution procedure is as follows:

1. A special frequency sampling has to be applied: **Frequencies → Frequency sweep for an inverse Fourier transform**. The sampling scheme which has been proven to be successful is $f_0, 2f_0, 5f_0, \ldots, f_g$ with f_0 as the basic frequency.

   Carefully select the basic frequency which is related to the total duration T_{total} of the transients according to $f_0 = 1/(2 \cdot T_{total})$. Unfortunately this duration is not known in general when starting the numerical investigation and hence has to be approximated.

   The number of frequencies (*No. of frequencies*) determines the highest frequency f_g to be modeled according to the indicated sampling scheme.

   In the lower part of the input window suggestions are given for the maximum time duration of the overall system responses which could be covered by the chosen basic frequency and for the rise time that could be considered by the specified frequency interval (range between the basic frequency f_0 and the highest frequency f_g).

Case: Network without transmission lines

According to the experience gained from example networks the overall duration of the transients is determined by

- The largest time constant in a branch of simple R-L or R-C networks.

  For such a case T_{total} can be approximated by 10 to 20 times this largest time constant (RC, L/R). If the duration of the excitation function is much longer then the duration of the transient responses, the necessary basic frequency is mainly determined by the excitation function itself. For short pulses and quick rise times, respectively, a large upper frequency limit is required. This situation might lead to a large number of frequency samples in case of large value of T_{total} .

- The duration of an excited oscillation.

  To compute a highly damped oscillation a large sample width may be sufficient (relatively high basic frequency, relatively small number of frequency samples). On the other hand for a slightly damped oscillation (large Q-factor) a very dense frequency sampling may be required (very low basic frequency) in order to cover the long overall duration of the transient currents or voltages.

  It always recommended to observe the frequency domain system responses before performing an IFT. Criteria for sufficient frequency domain data:

Note

- At the basic frequency the currents or voltages should be not more than 10% away from their static values!
- The peaks of resonances should be covered in a relatively smooth way. The true peak values should be captured!

Case: Network without transmission lines

The required frequency sampling may be mainly determined by the length of the lines, their terminations and so on. For further details the reader is referred to the CONCEPT-II User's manual, section 5.2: *Key points for performing an IFT.*

2. Start the frequency domain computation of the system responses. Click on ●

3. Perform the IFT: **Post processing** → **Compute time-domain values** (see Fig. B.13).

4. The validity of the results have to be checked:

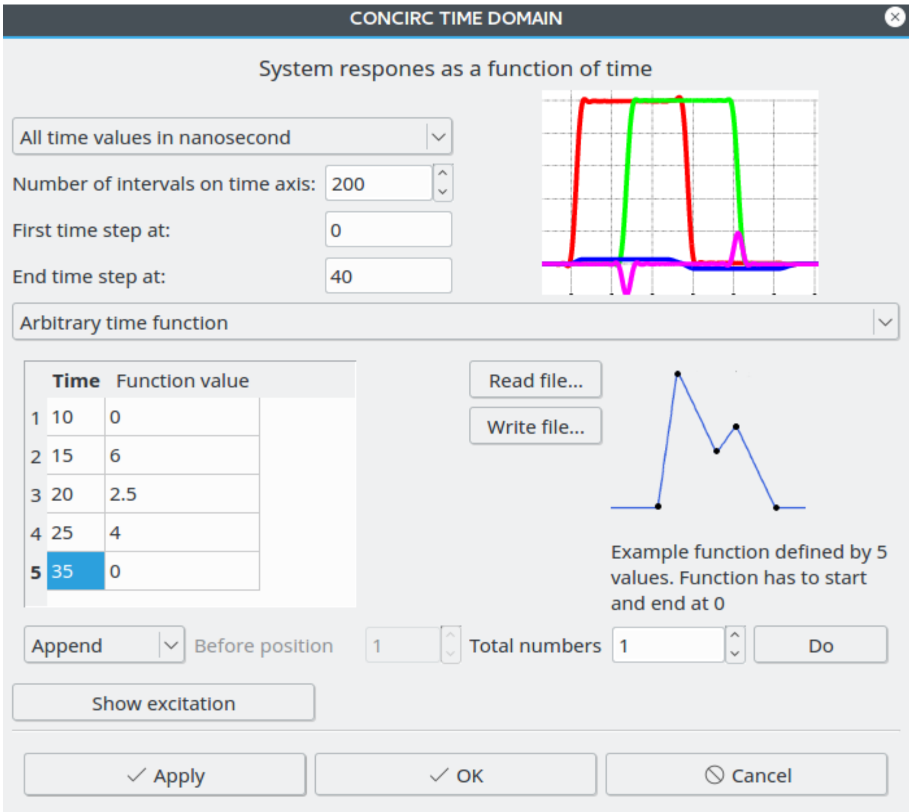
   It is important that all system responses start with zero amplitude. In order to judge this, it is recommended to start the time interval to be displayed significantly before the possible beginning of a physical quantity, beginning the time axis at a negative time is no problem. The curves must be zero before a signal rise is allowed from physical reasons. If this is not the case the basis frequency should be reduced (which automatically means that the number of frequency steps has to be increased to be able to model the same rise time as before). In case of small unphysical oscillations being observed before the start of a system response (Gibbs phenomenon) increase the highest frequency to be considered or lower the rise time if possible.

Note

Due to the sampling in the frequency domain a signal continuation in the time domain has to be excepted from mathematical reasons (see text books on system theory). Nonlinearities cannot be considered.

Simple time-domain analysis examples based on an IFT can be found in the folder *examples-concirc/IFT.*

a) The step response of an R-L-circuit is investigated in the subfolder *L-R-circuit-IFT*. A video explaining the modeling procedure can be found here: https://youtu.be/5uw9hWz-Pa3U (in German).



CONCIRC TIME DOMAIN

System respones as a function of time

All time values in nanosecond

Number of intervals on time axis: 200

First time step at: 0

End time step at: 40

Arbitrary time function

	Time	Function value
1	10	0
2	15	6
3	20	2.5
4	25	4
5	35	0

Example function defined by 5 values. Function has to start and end at 0

Fig. B.13: Input window for specifying the excitation function. Here: ***Arbitrary time function.*** *Precondition: the function is defined by piecewise linear sections and starts and ends at zero amplitude as demonstrated by an example function.*

Clicking on ***Show excitation*** *the time function will be displayed as rebuild by an IFT from the frequency sampling that has been entered under* ***Frequencies – Frequency sweep for an Inverse Fourier Transform (IFT)****.*

If the displayed function does not start at the specified time with zero amplitude, the frequency sampling might be insufficient. If strong hight frequency oscillations happen at locations of strong slopes the highest frequency should be increased...

b) A single-wire line over an ideal ground is considered in the sub-folder *Transient_line_IFT.*

The transmission line data are contained in the files *Single_wire_mtl_ymat.h5* (created by the program *CONMTL*) and *red-y-mat.h5* (created by the program *CONCEPT-II*). In *CONCIRC* the corresponding .h5 files can be loaded by means of the

NETWORK element ***(right mouse click*** → ***Edit properties*** → ***get port/MTL file ...).***

Images in .png format are provided for reference.

The two following examples demonstrate the frequency-domain application of *CONCIRC*.

B.6.1 Simple Circuit (*examples-concirc/circuit-1*)

The circuit to be computed is depicted in Fig. B.14 .

After the program run has finished the “System responses as a function of frequency” window opens immediately. Selecting **Plot current** will open a window showing the output data of the ammeters. As only a single frequency sample has been specified (88.1 kHz) only ASCII values appear:

```
#########################################################

#
# Current
#
Frequency: 88.1 kHz


Probe: PrA1
Magnitude: 0.100082 [A]


Probe: PrA2
Magnitude: 0.00466741 [A]


Probe: PrA3
Magnitude: 0.0999728 [A]
```

Notice how the watt-meter has been integrated into the circuit. The input impedance can be computed by clicking **Plot impedance** and selecting “Probe U” → **PrV1**, “Probe I” → **PrA1**

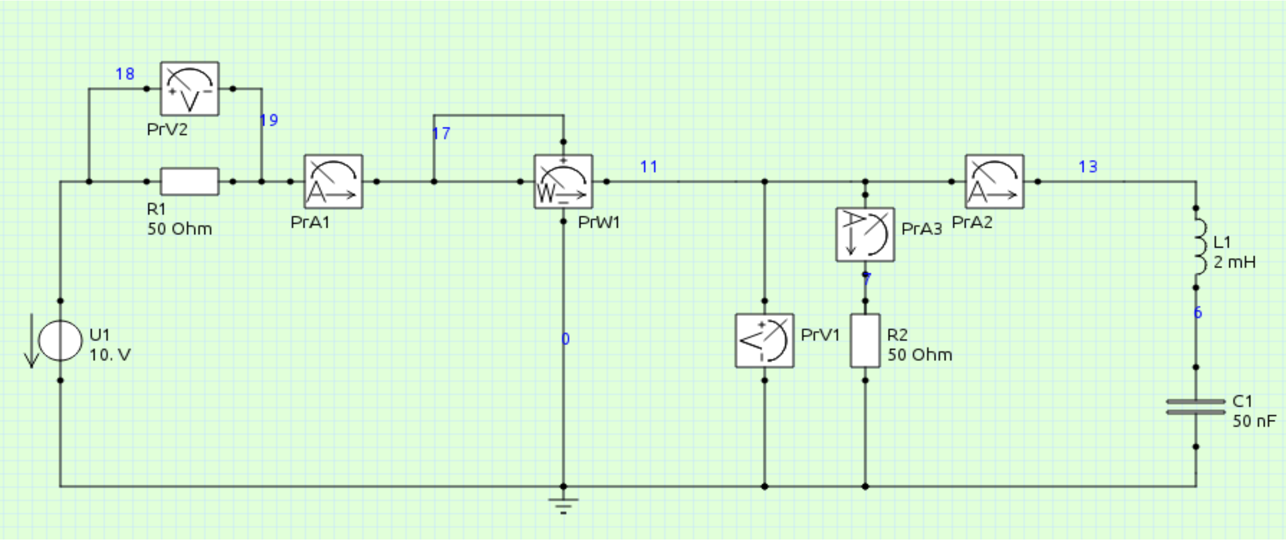


Fig. B.14: Simple circuit

B.6.2 Transformer (*examples-concirc/circuit-2*)

The investigated circuit is shown in Fig. B.15.

This time the input impedance shall be computed as a function of frequency: f = 40,...,500 Hz with 80 frequency steps (**Frequencies** → **Frequency sweep for arbirary interval**) . The data is gained by dividing the voltage of generator U1 by the source current PrIsrc1. The frequency dependent behavior is illustrated in Fehler: Referenz nicht gefunden. It is advantageous to ground both the primary side and the secondary side of the transformer.

In case of no solution of the equation system that has to be solved by the program (an error message will be displayed!) one could try for example to place an artificial resistor of high value a critical position of the circuit under consideration.

Notice that the voltage at node 2 has been selected to be scanned as a function of frequency (PrPhi1) . In general this can by achieved by a right click on a node → **Edit properties** …

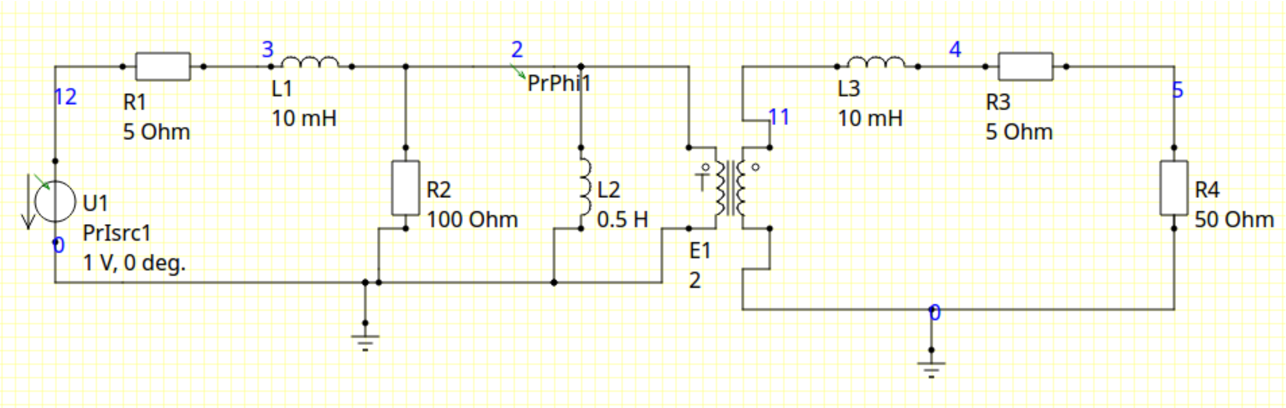


Fig. B.15: Simple circuit including a transformer

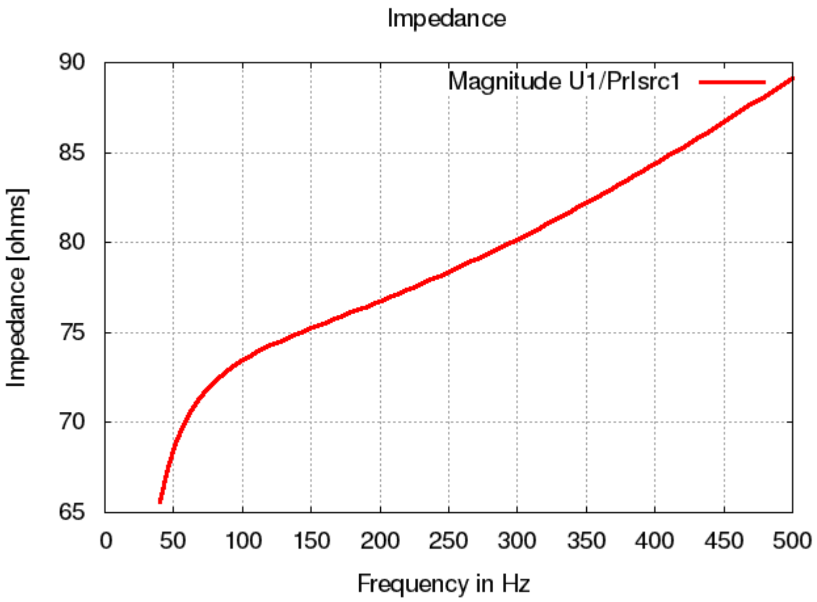


Fig. B.16: Input impedance of the transformer circuit according to Fig. B.15

B.7 Parameter sweep (*examples-concirc/ParallelResonance)*

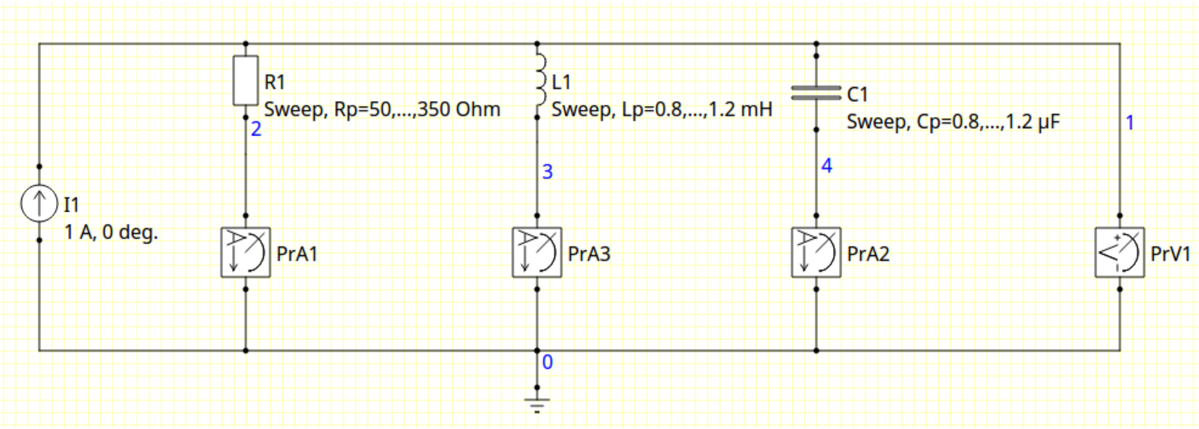
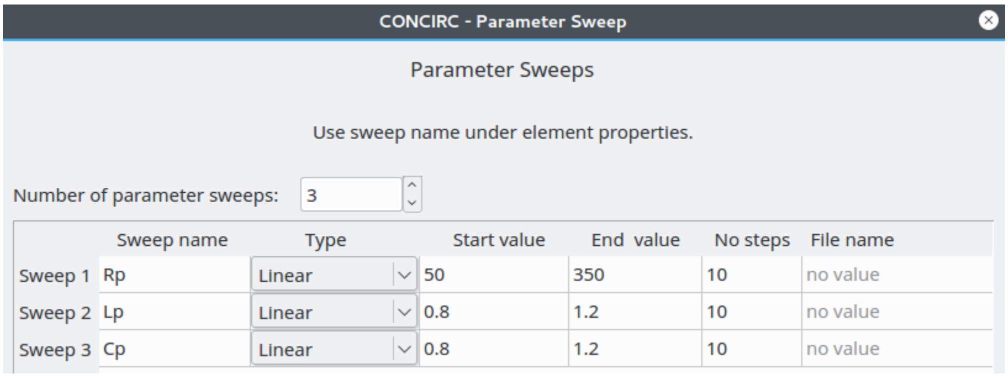


Fig. B.17: The investigated circuit

The equivalent circuit is shown in Fig. B.17. The frequency sampling ranges from 1 to 10 kHz with 91 samples (see file *netfreq.in*). The assumed entries for parameter sweeps can be observed when clicking on **Parameter sweep**, see Fig. B.18. Three sweeps are defined with 10 samples each. Note that in summary there are 91 x 10 x 10 x 10 = 91.000 data sets to be computed!



Parameter Sweeps

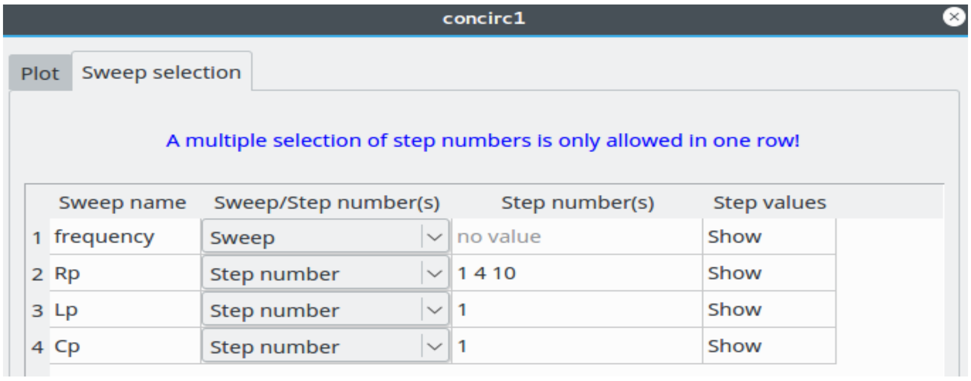
Use sweep name under element properties.

Number of parameter sweeps: 3

	Sweep name	Type	Start value	End value	No steps	File name
Sweep 1	Rp	Linear	50	350	10	no value
Sweep 2	Lp	Linear	0.8	1.2	10	no value
Sweep 3	Cp	Linear	0.8	1.2	10	no value

Fig. B.18: Three sweeps are specified. The fourth sweep is the frequency sweep not to be specified here.

After the computation has finished the voltage (output of probe PrV1) is selected to be displayed as a function of frequency (select under tab **Plot, Postprocessing** → **Show frequency domain values**). See Fig. B.19 for the entries to be made for curves as plotted to in Fig. B.20 .



concirc1

A multiple selection of step numbers is only allowed in one row!

	Sweep name	Sweep/Step number(s)	Step number(s)	Step values
1	frequency	Sweep	no value	Show
2	Rp	Step number	1 4 10	Show
3	Lp	Step number	1	Show
4	Cp	Step number	1	Show

Fig. B.19: Frequency sweeps for three different values of R1 (see equivalent circuit in Fig. B.17). For the capacitance C1 and the inductance L1 the initial values are taken (step number 1 each).

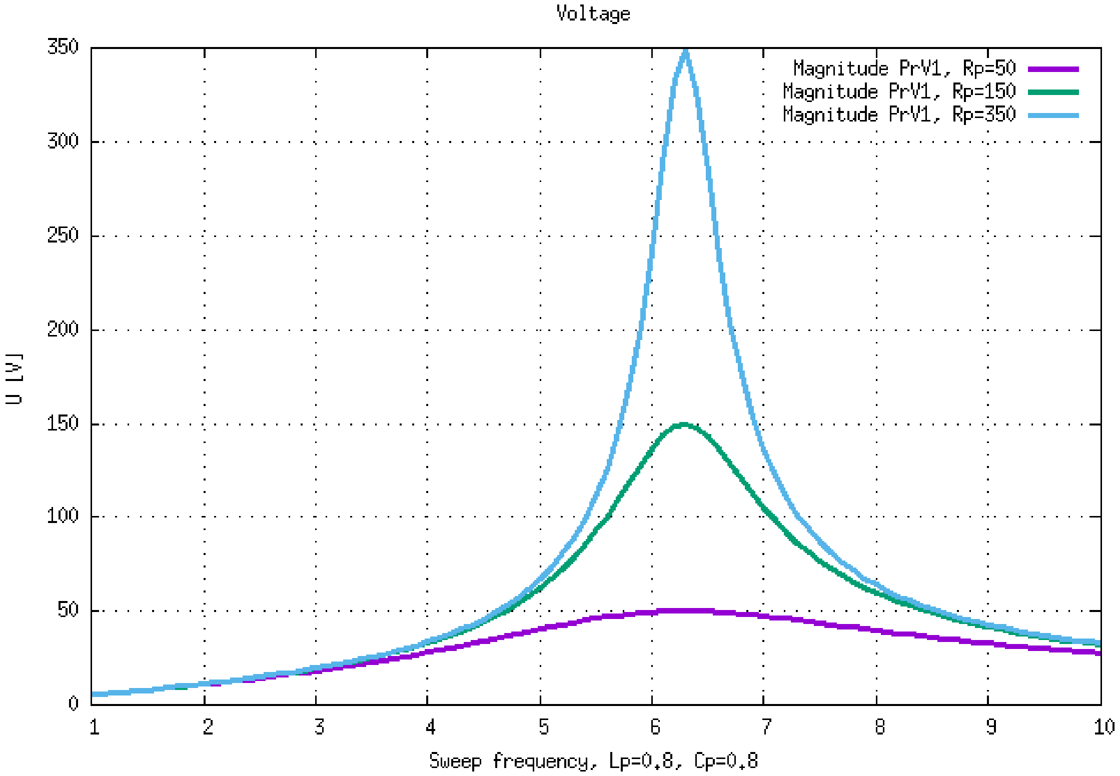
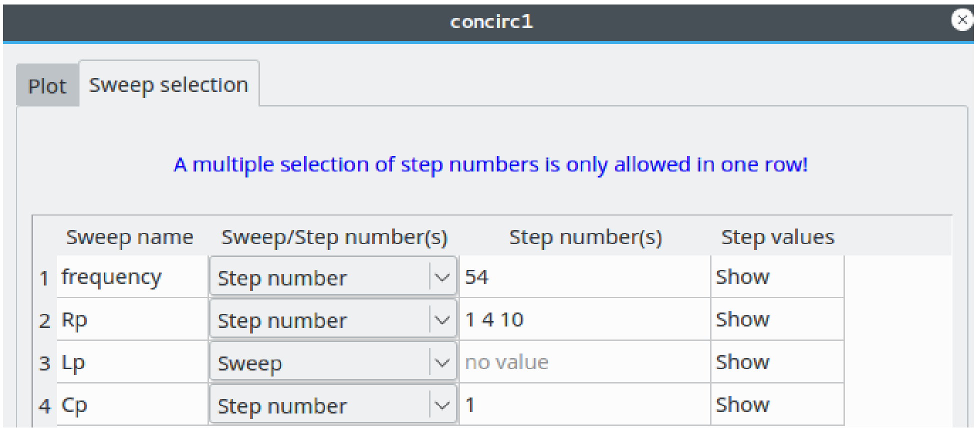


Fig. B.20: Voltage of a function of frequency. The curves refer to the three different resistors as indicated in top right corner (step 1: Rp=50, step 4: Rp=150, step 10: Rp=350, see also Fig. B.17).

Fig. B.21 demonstrates the entries that are necessary to get curves as depicted

in Fig. B.22. Here we have a constant frequency whereas the inductance is varying between 0.8 and 1.2 mH (see Fig. B.17).



A multiple selection of step numbers is only allowed in one row!

	Sweep name	Sweep/Step number(s)	Step number(s)	Step values
1	frequency	Step number	54	Show
2	Rp	Step number	1 4 10	Show
3	Lp	Sweep	no value	Show
4	Cp	Step number	1	Show

Fig. B.21: A sweep of Lp (see equivalent circuit in). The Frequency sample 54 is taken (click on ***Show****) corresponding to 6.3 kHz (see also Fig. B.17) . The parallel capacitance is set to its initial sample.*

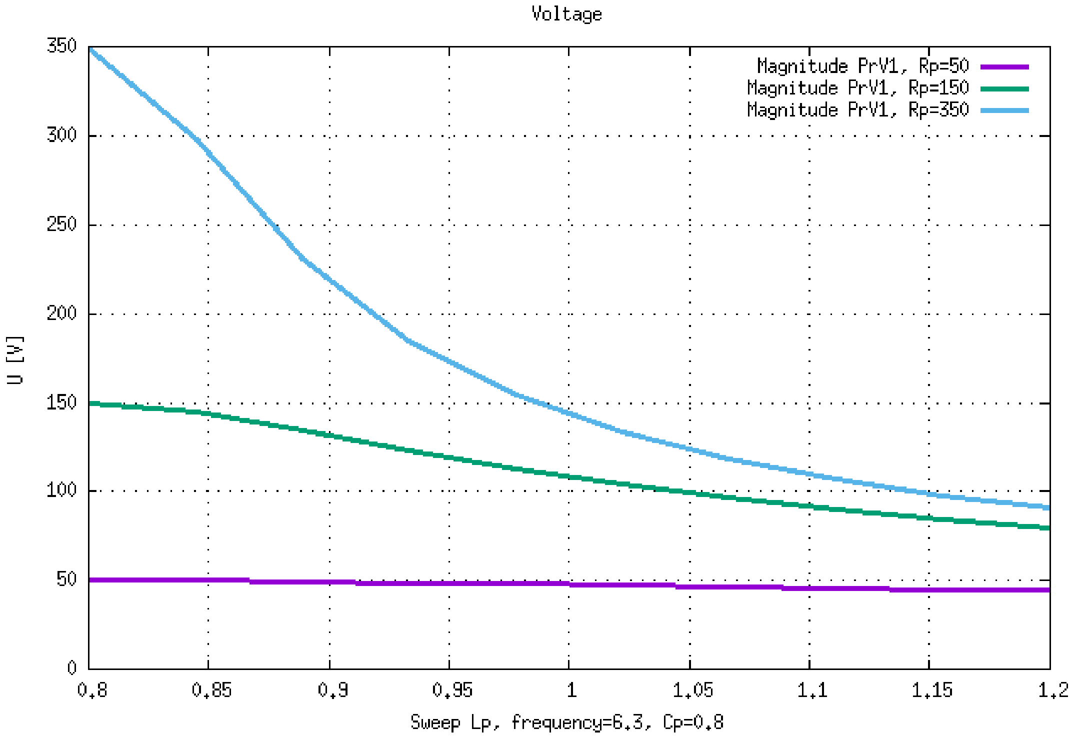
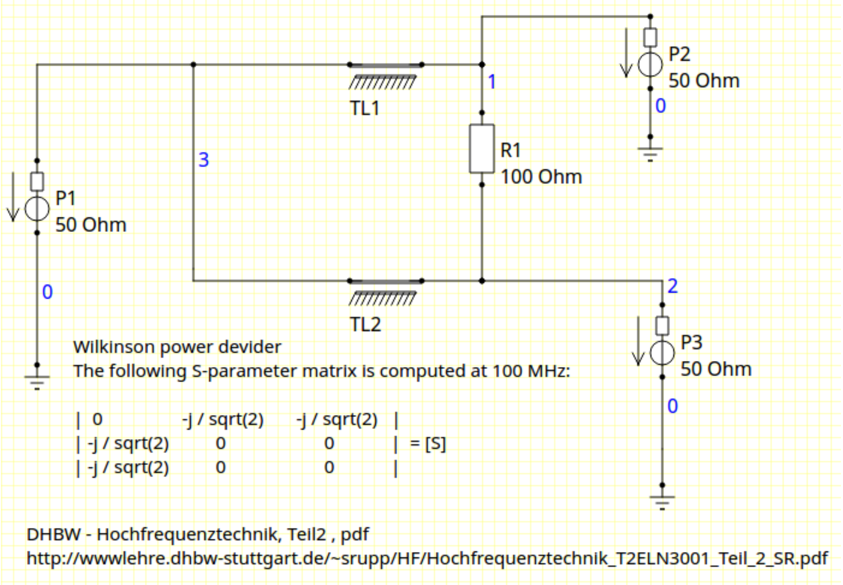


Fig. B.22: Voltage PrV1 according to Fig. B.17 as a function of inductance Lp. The curves refer to the three different resistors as indicated in top right corner (step 1: Rp=50, step 4: Rp=150, step 10: Rp=350) and a constant frequency of 6.3 kHz.

B.8 Wilkinson power divider (*examples-concirc/Wilkinson)*

The equivalent circuit is illustrated in Fig. B.23. The example is taken from [*] and S-parameters shall be computed.



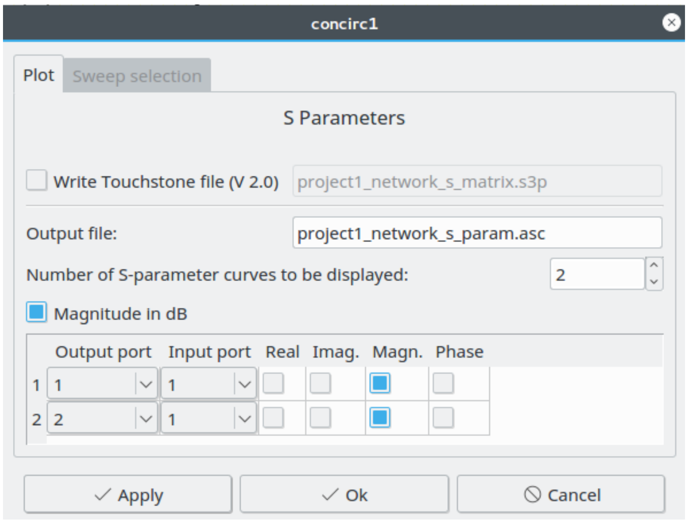
Wilkinson power devider
The following S-parameter matrix is computed at 100 MHz:

```
| 0           -j / sqrt(2)   -j / sqrt(2)  |
| -j / sqrt(2)     0              0        | = [S]
| -j / sqrt(2)     0              0        |
```

DHBW - Hochfrequenztechnik, Teil2 , pdf
http://wwwlehre.dhbw-stuttgart.de/~srupp/HF/Hochfrequenztechnik_T2ELN3001_Teil_2_SR.pdf

Fig. B.23: Computation of the S parameters of a Wilkinson power divider.

As can be seen, three ports have been placed (elements P1, P2, P3). For a frequency of 100 MHz the analytically computed S-parameters are known and shown in the figure as a 3 x 3 matrix.



S Parameters

□ Write Touchstone file (V 2.0) project1_network_s_matrix.s3p

Output file: project1_network_s_param.asc

Number of S-parameter curves to be displayed: 2

☑ Magnitude in dB

	Output port	Input port	Real	Imag.	Magn.	Phase
1	1	1	□	□	☑	□
2	2	1	□	□	☑	□

Fig. B.24: Selection of S-parameters to be displayed and stored

Start the computation by clicking on ●. After the computation has finished a window according to Fig. B.24 comes up (the window also opens by clicking on **Post processing → Show S-parameters**).

Clicking on **Apply** or **OK** provides the S-parameter curves S_11 and S_21 (here magnitudes) according to Fig. B.25.

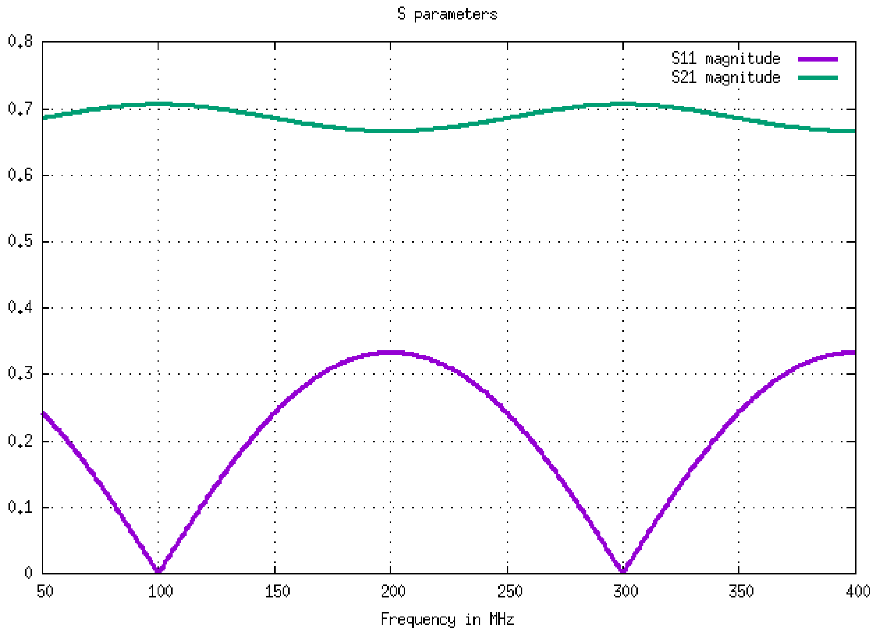


Fig. B.25: Magnitude of the S-parameters S_11 and S_21 as a function of frequency

[*]: DHBW -Hochfrequenztechnik, Teil 2, pdf

http://wwwlehre.dhbw-stuttgart.de/~srupp/HF/Hochfrequenztechnik_T2ELN3001_Teil_2_SR.pdf

B.9 Further examples

- See folder *examples-concirc/Constant-Elements*

  Complex constant elements. Specification of a frequency not necessary
- See folder *examples-concirc/LosslessTransformer*

  Computation of a transformer based on current controlled voltage sources (CCVS)
- Links to demo videos (German): https://youtu.be/scTEmUDtUJA

  https://youtu.be/-FlUSDN4rt8
- For CONCEPT-II users: Equivalent circuit based on the reduced y matrix: $CONCEPT/examples/Coupled_dipoles, see also CONCEPT-II Manual, Section 3.2.3 'Using red-y-matrix.h5 with the network tool CONCIRC'

- Combining with transmission line theory, see Section 5.3 ‘Multiconductor transmission line’

  of the CONMTL User’s Manual

C Licenses

CONCIRC uses the following 3rd-Party packages:

Package	License	Link
OpenBlas 0.2.10 (Linux)	BSD	http://www.openblas.net/
Qt 5.10.1	LGPL	https://www.qt.io/download-open-source/#section-2
The usage of the software is according to the GPL and the LGPL (see below)		
HDF5 1.8.20	BSD	http://www.hdfgroup.org/products/licenses.html
Inno Setup Compiler 5.5.9		http://www.jrsoftware.org/isinfo.php

For details on the respective licenses/copyrights see the files in the directory

$INSTALLATION_DIRECTORY/licenses. Text files of the GPL and the LGPL are included (gpl.txt, lgpl.txt)

[1] C.-W. Ho, A.E. Ruehli, P.A. Brennan: *The Modified Nodal Approach to Network Analysis*. IEEE Trans. on Circuits and Systems, June 1975, No. 6, pp 504 – 509

[2] S. Skibin: *Abstrahlung von Leitungen in numerischen Modellen mit Oberflächensimulation*, Diss. TUHH, 2005, Shaker Verlag