Technische Hochschule Ingolstadt

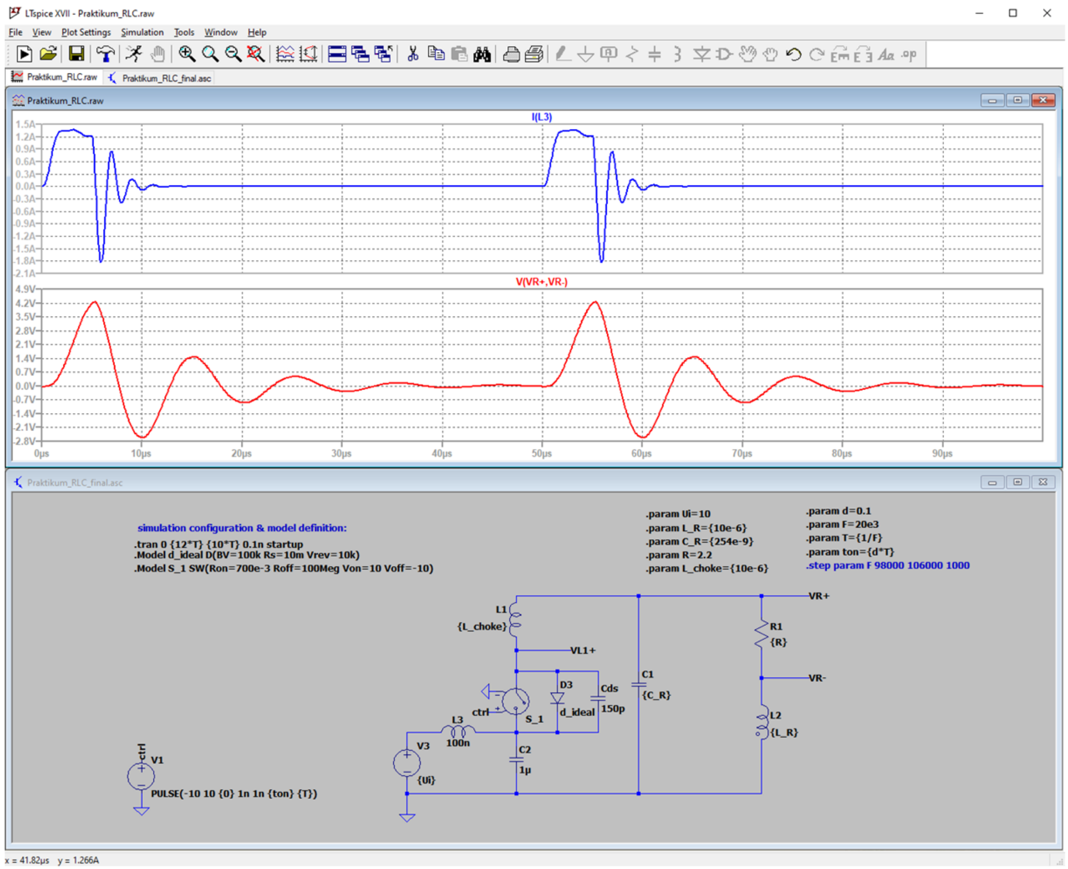
An Introduction to LTspice

Hannes Schwan

10.05.2024

What is LTspice?

- LTspice® is a free high performance SPICE simulator software, including a graphical schematic capture interface.
- Schematics can be probed to produce simulation results easily explored through LTspice's built-in waveform viewer.
- Please download it from: https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html



realistic simulation of a RLC (Resistor-Inductor-Capacitor) resonant circuit:

I. configuration of Ltspice (solver, simulation configuration, appearance etc.)

II. realizing circuit schematics with parametrized components

III. realizing component models (diodes, switches etc)

IV. transient analysis of the built up RLC circuit

V. AC analysis of the built up RLC circuit

Agenda

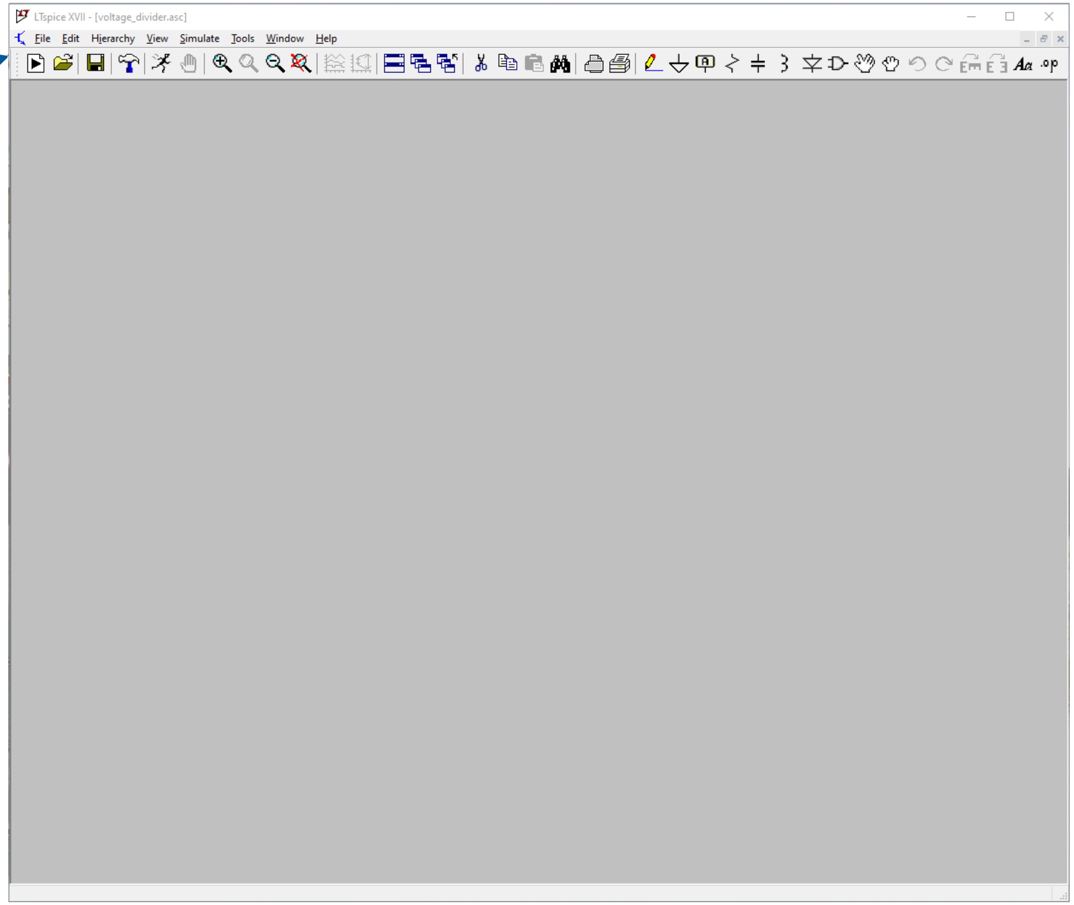
realistic simulation of a RLC (Resistor-Inductor-Capacitor) resonant circuit:

I. configuration of Ltspice (solver, simulation configuration, appearance etc.)
II. realizing circuit schematics with parametrized components
III. realizing component models (diodes, switches etc)
IV. transient analysis of the built up RLC circuit
V. AC analysis of the built up RLC circuit

I. Configuration of Ltspice

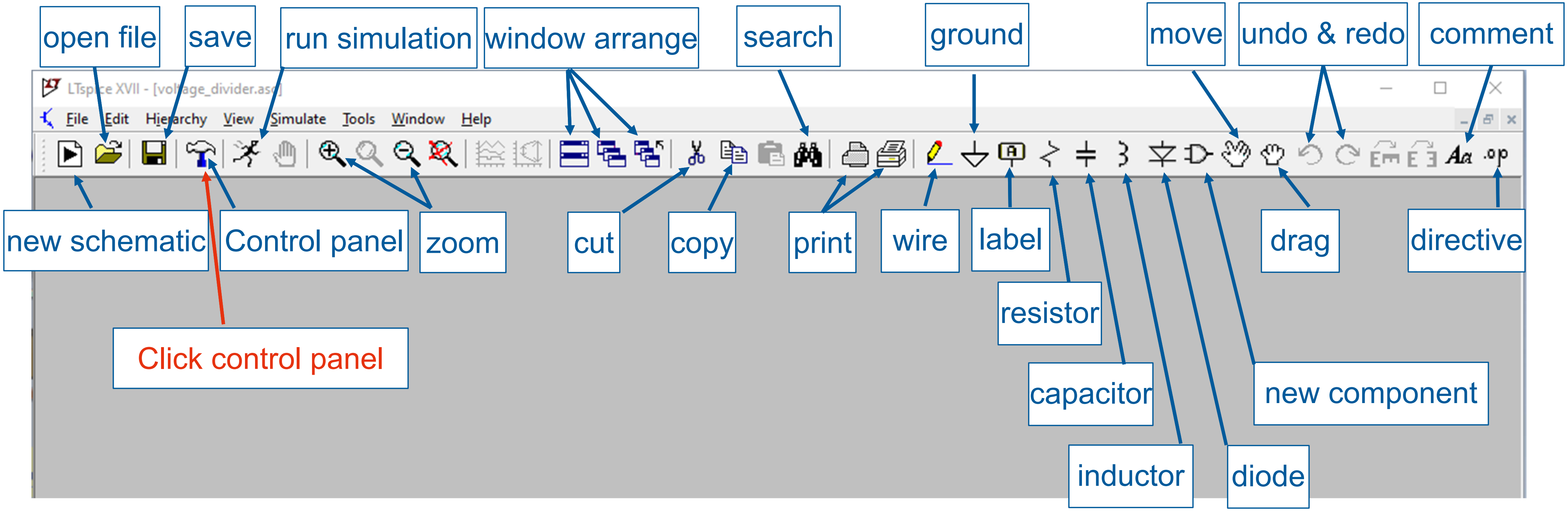
New Schematic

- Open Ltspice
- Click “File”
- Click “New Schematic”
- Click “File”
- Click “Save As”
- Name the schematic: voltage_divider
- Click “Save”



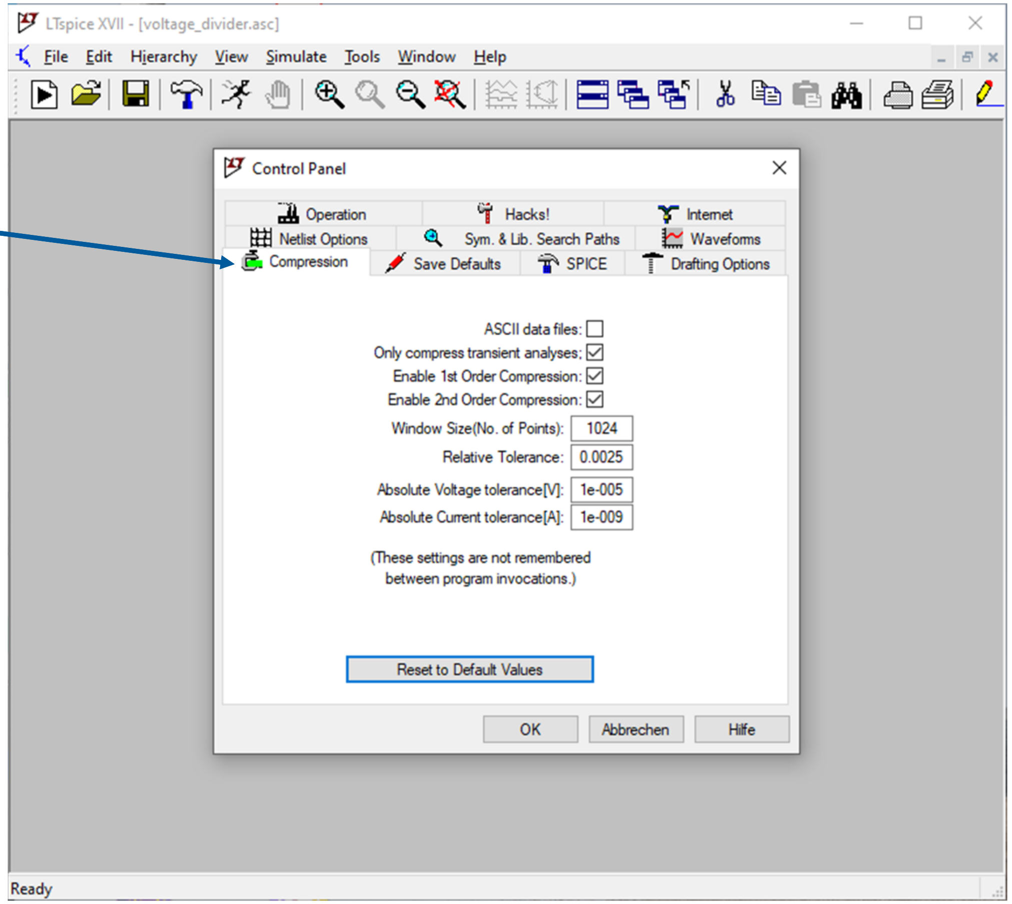
I. Configuration of Ltspice

Toolbar



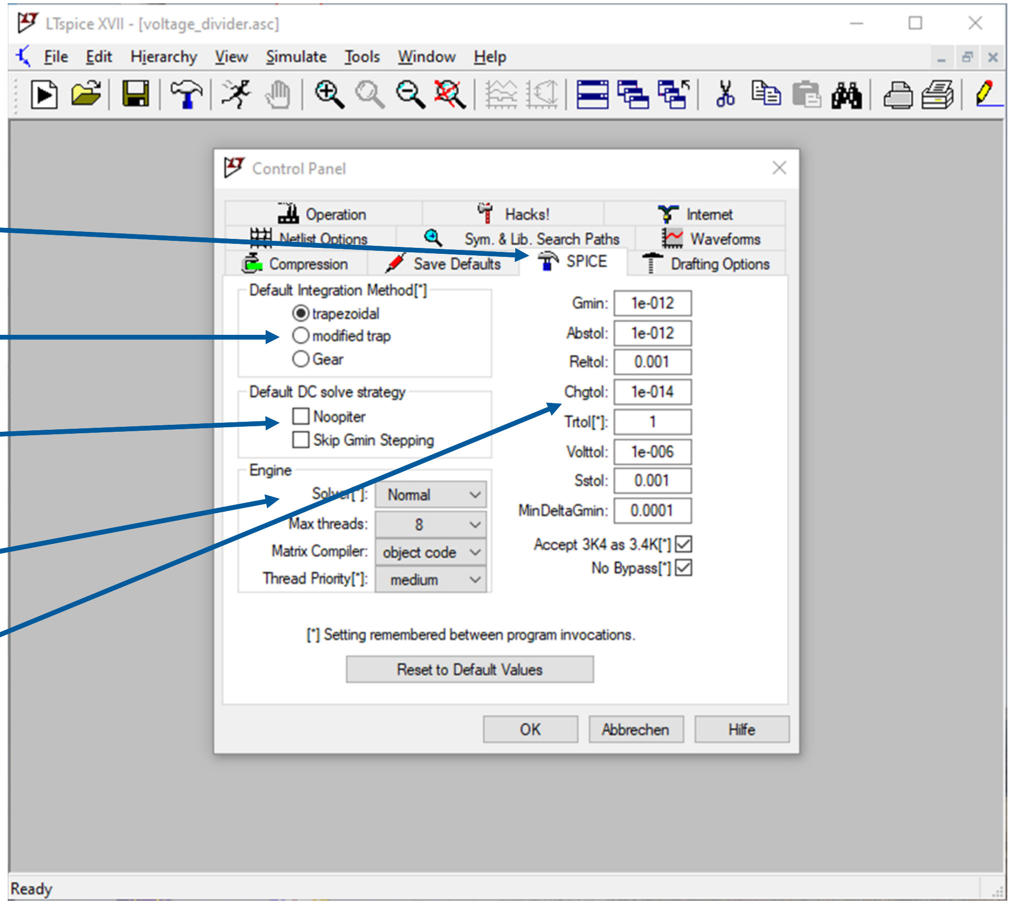
Compression settings of simulation data:

- intern results are compressed and rounded to the tolerances given
- low tolerances result in more correct results but extend simulation time!



<u>SPICE settings:</u>

- solver method: different solver algorithms can be more accurate or faster
- DC solve strategy
- solver and hardware related settings: the more cores the more threads are possible for faster simulation
- tolerances of node solving



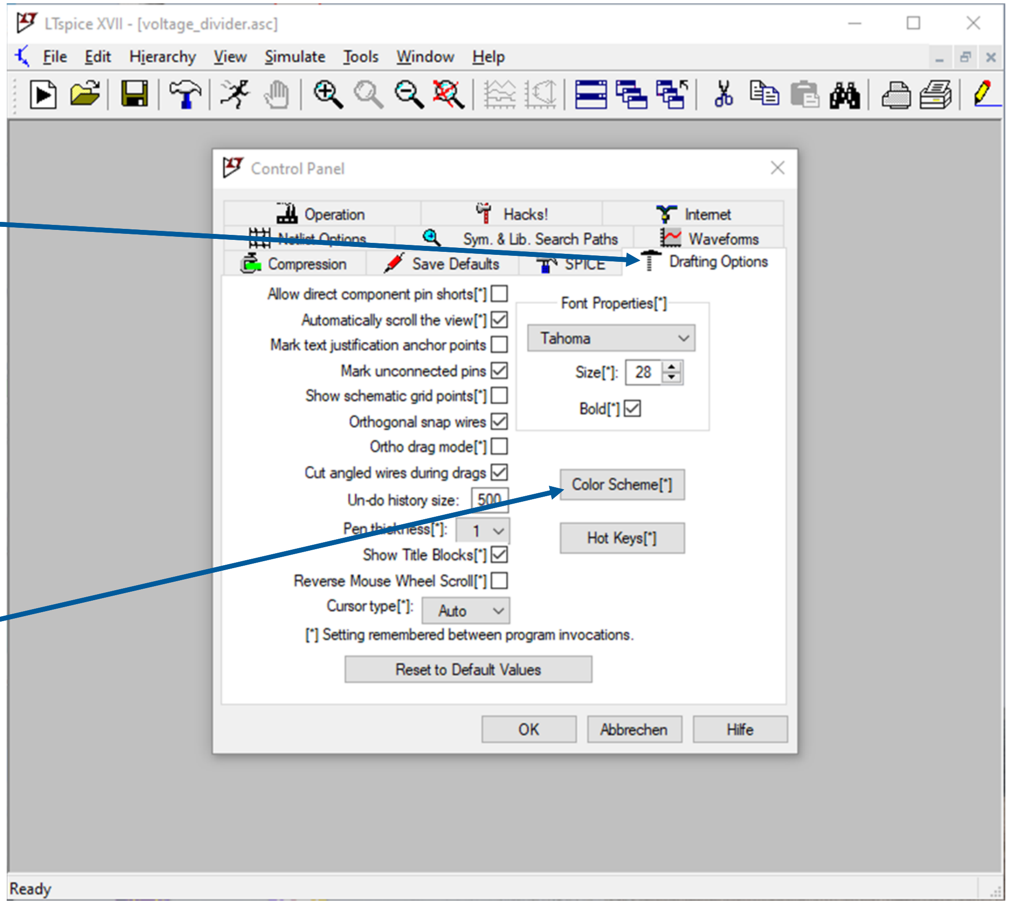
I. Configuration of Ltspice

Control Panel: Drafting Options

Options for drafting and appearance of the schematic:

- show grid and unconnected wires
- drag options of wires
- font and font size selection
- etc..

Click: “Color Scheme”



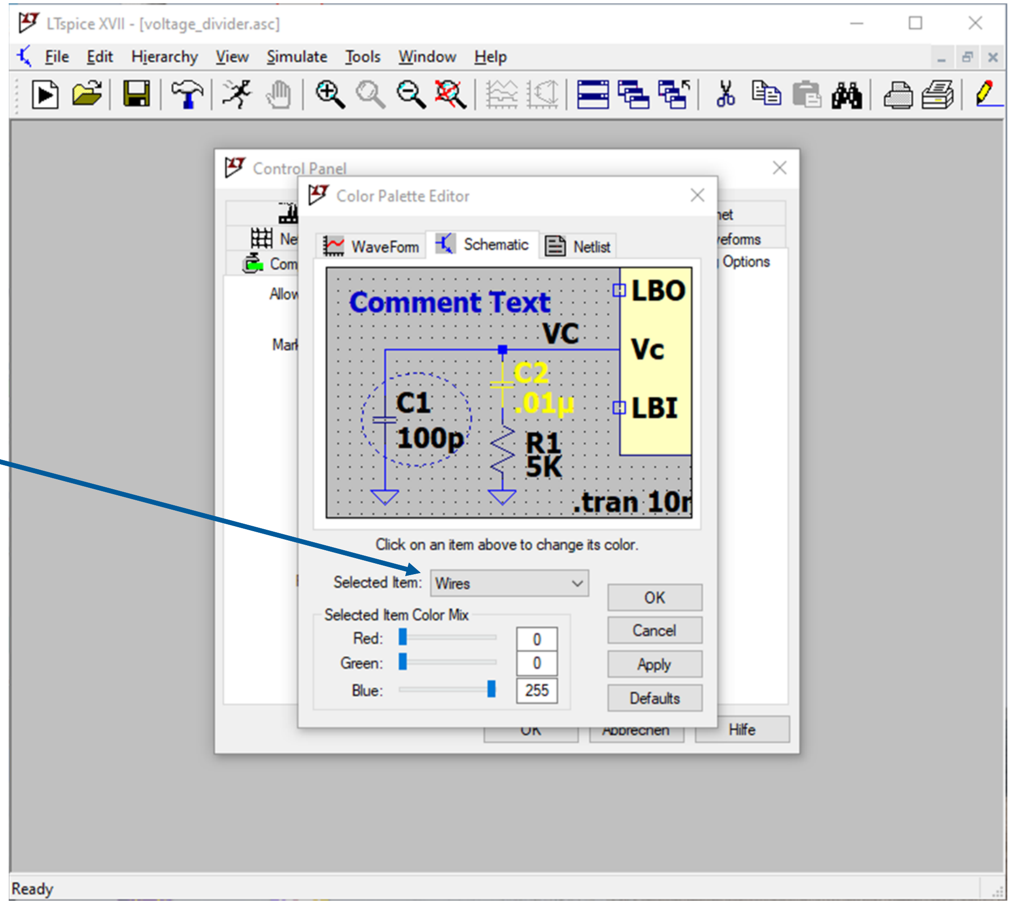
I. Configuration of Ltspice

Control Panel: Drafting Options

<u>Color Palette Editor - Schematic:</u>

change colour of:

- wires
- backgrounds
- text: comments, directives
- components
- etc.

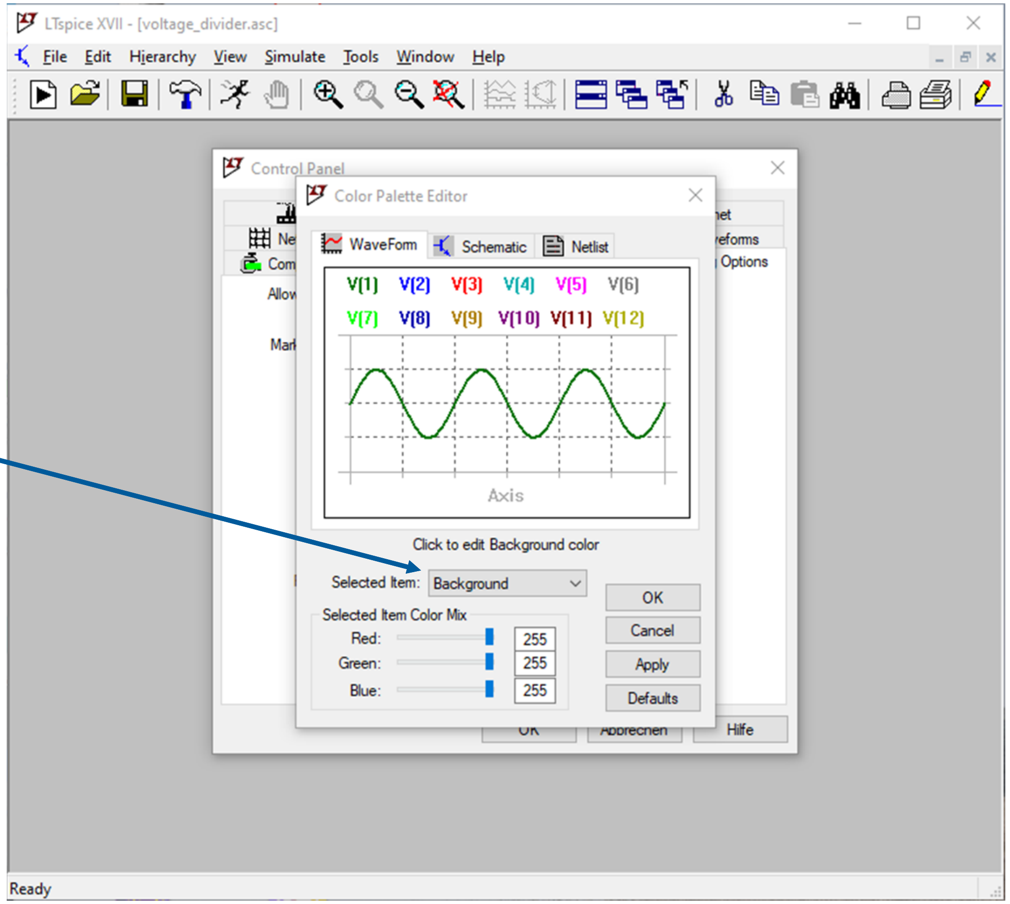


Color Palette Editor - WaveForm:

change colour of:

- traces
- axis
- grid
- background

  (please change to white 255-255-255 now!)
- etc.

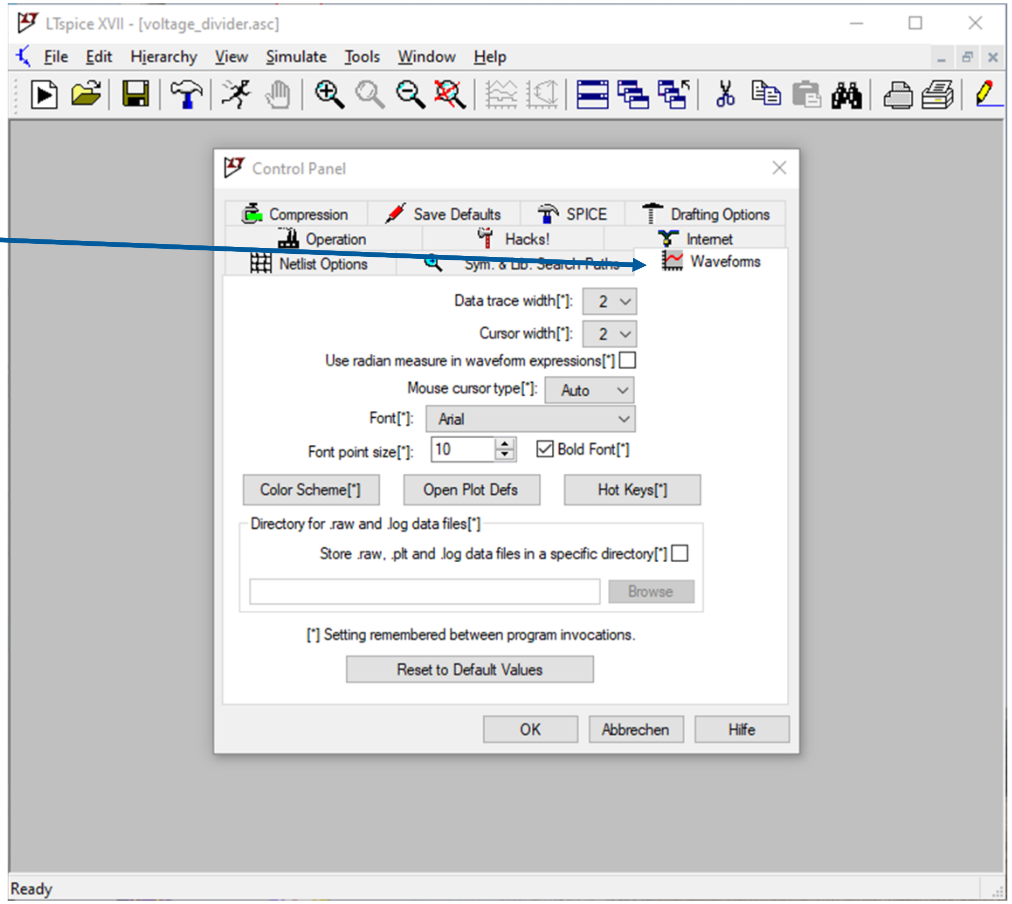


Waveforms:

change:

- trace width and cursor width of waveforms
- cursor type in waveform viewer
- font and font size of waveform viewer
- etc.

(please copy the shown settings now!)



Agenda

realistic simulation of a RLC (Resistor-Inductor-Capacitor) resonant circuit:

I. configuration of Ltspice (solver, simulation configuration, appearance etc.)

II. realizing circuit schematics with parametrized components

III. realizing component models (diodes, switches etc)

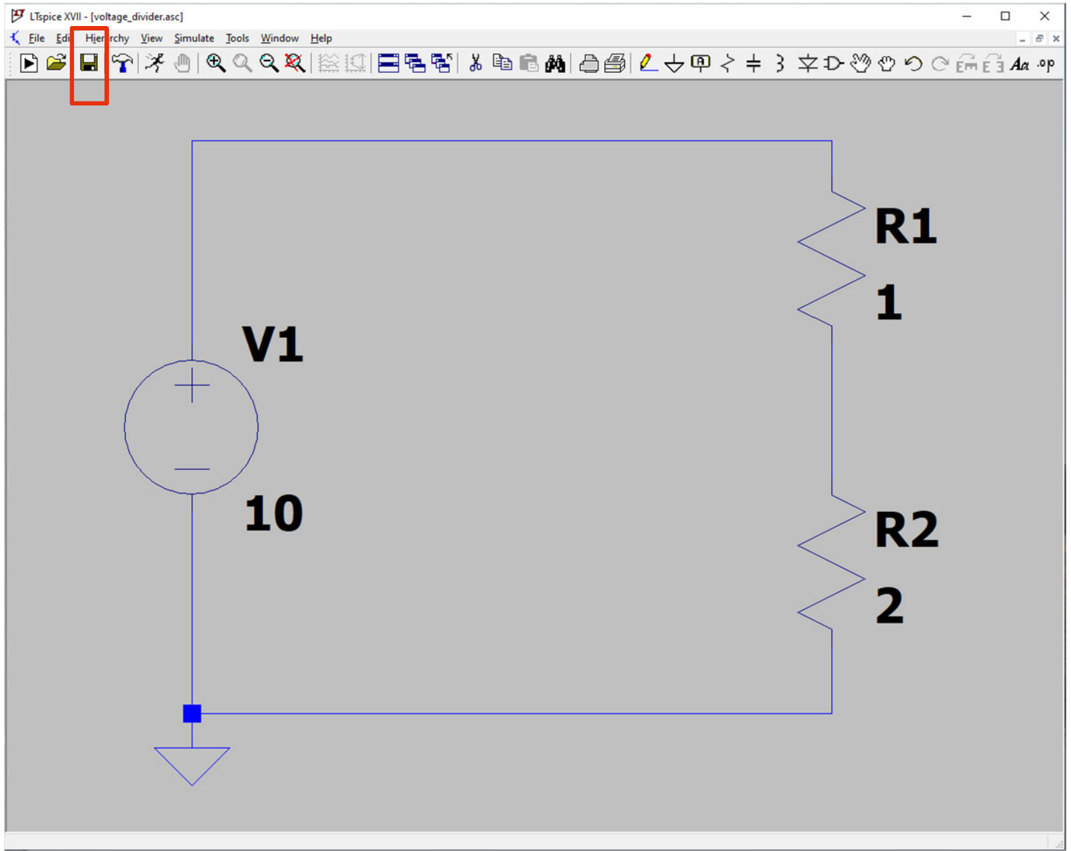
IV. transient analysis of the built up RLC circuit

V. AC analysis of the built up RLC circuit

II. Realizing circuit schematics with parametrized components

Draw schematic

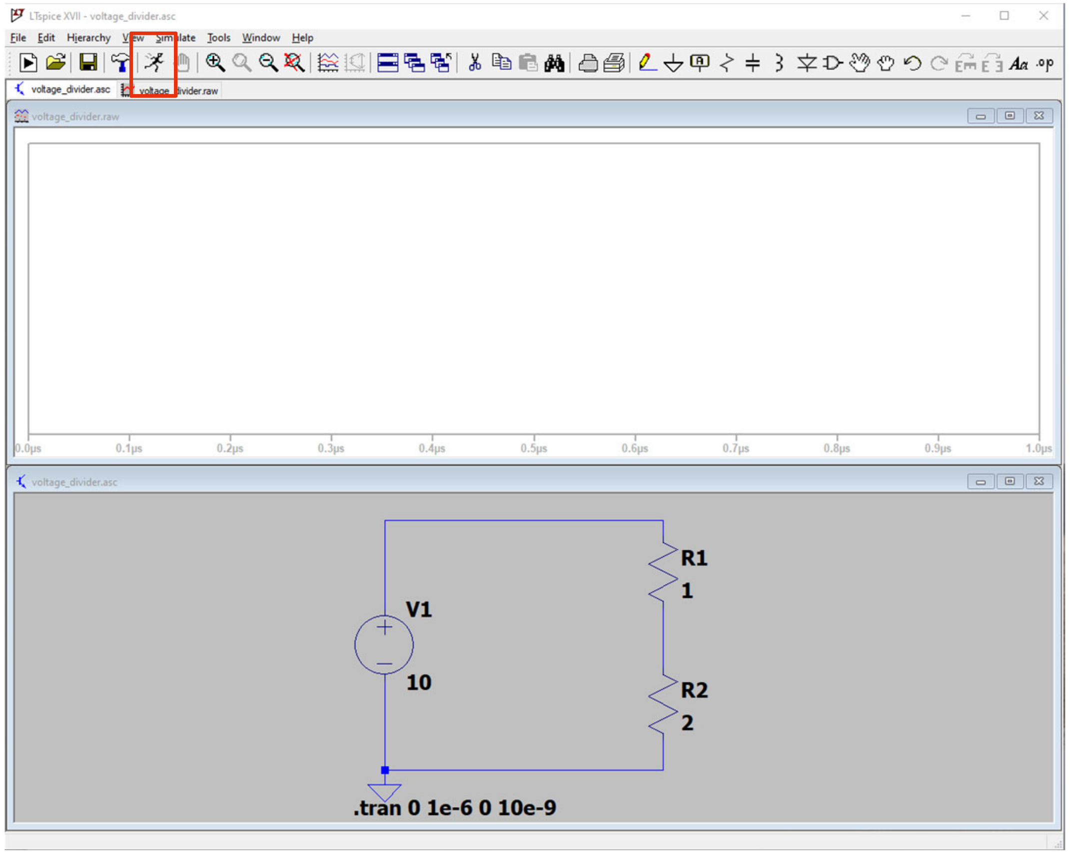
- Place two resistors
- Right click: R1 put 1Ohm, R2 put 2Ohm
- Place a voltage source
- Right click: V1 put 10V
- Connect the components with wires
- Place a ground
- Click “Save”



II. Realizing circuit schematics with parametrized components

Simulation configuration: transient analysis

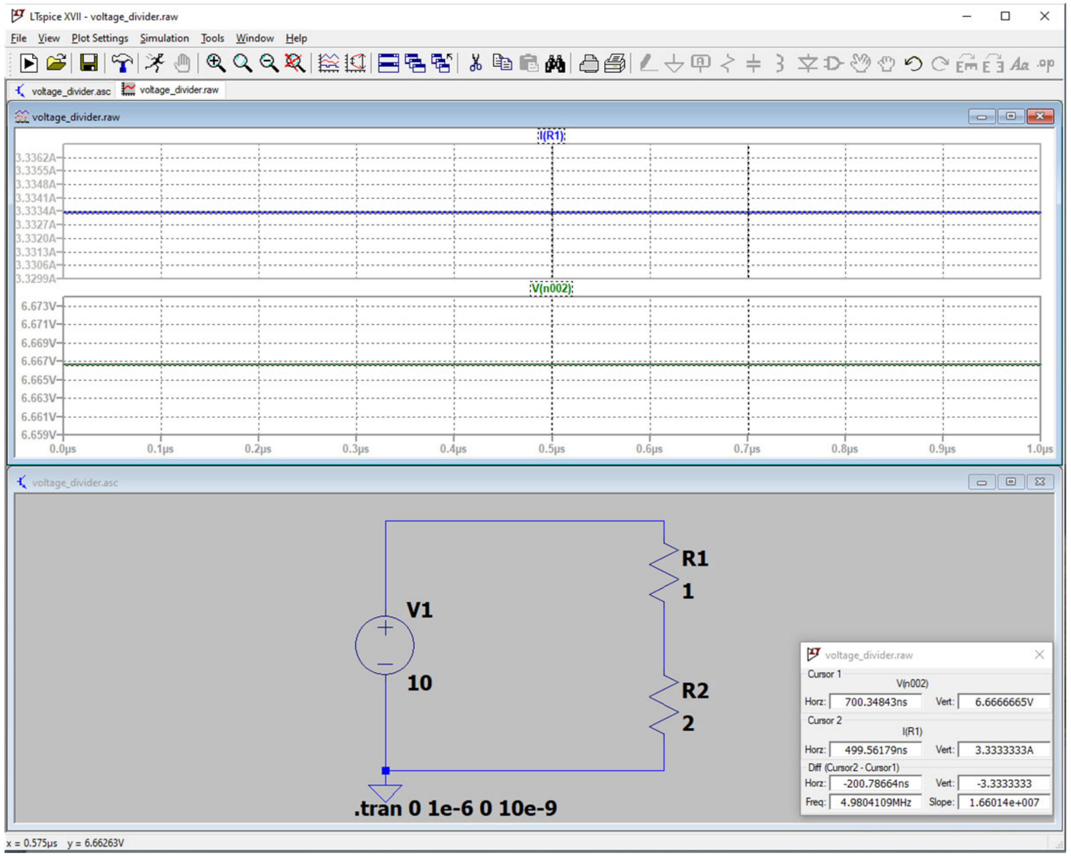
- Click “run”
- Transient analysis:
  - Stop time: 1e-6
  - Start time: 0
  - Max. time step: 10e-9
- Click “ok”
- Click “run”



II. Realizing circuit schematics with parametrized components

Waveform analyzing

- Click on wire between R1 and R2 to measure the voltage at this node versus ground
- Click on the voltage name in the waveform viewer -> read time and voltage value from the pop-up
- Drag the cursor to get the voltage reading vs. time
- Right click waveform viewer
- Click "Add Plot Pane"
- Click on R1 to get the current through the circuit
- Click on the current name in the waveform viewer
- A second cursor appears! The pop up also shows differences in time and voltage of both cursors.

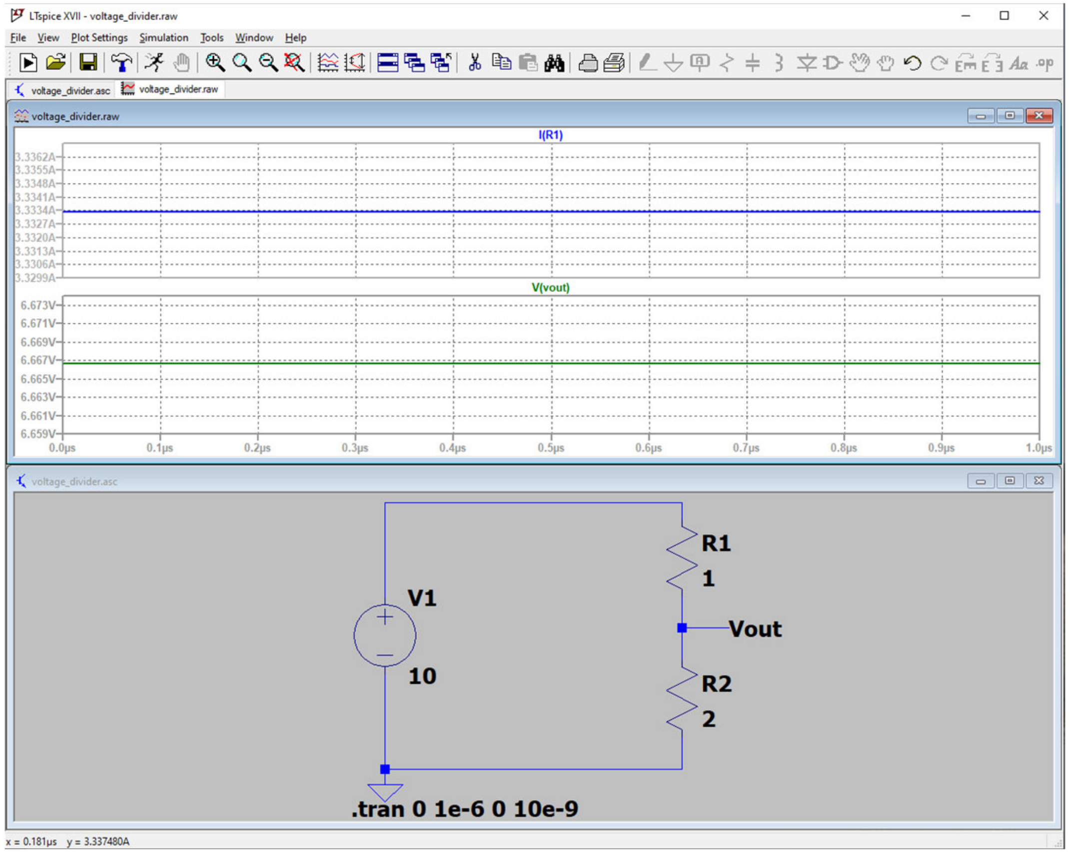


II. Realizing circuit schematics with parametrized components

Labeling

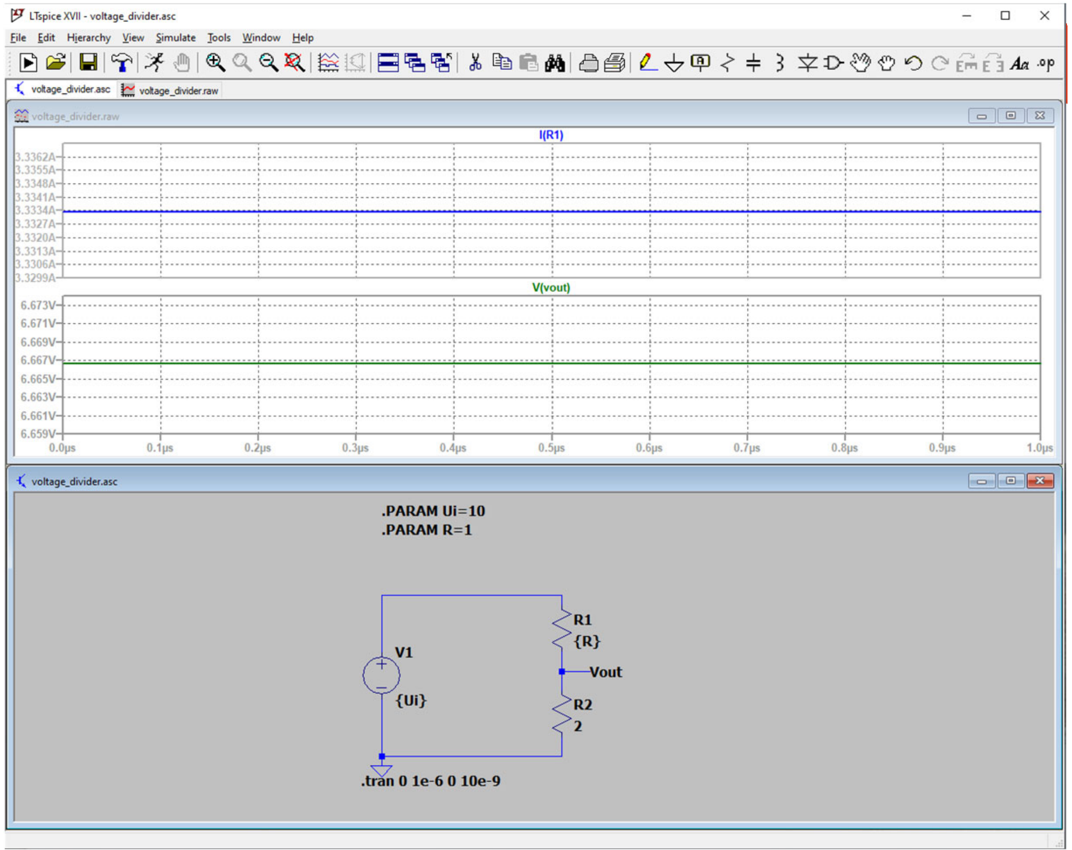
- Draw a wire from the node between R1 and R2
- Click “label”
- Give label name Vout
- Place label at wire end
- Cut the voltage across R2 from the viewer
- Right click to escape cut mode!
- Click “run”
- Click label Vout
- Drag Vout to second plot pane
- Advantage of Labeling: changes in the schematic

take effect on node names but not on labels!



Parametrizing

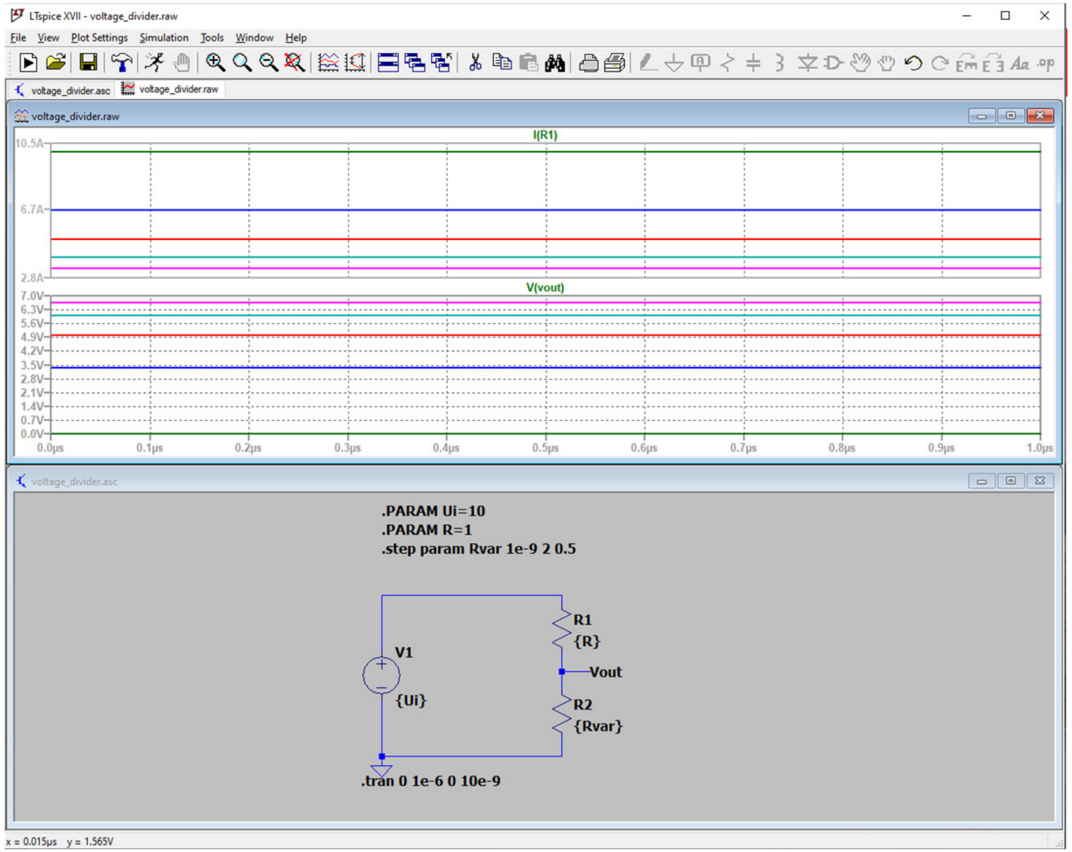
- Right click V1
- Give DC value: {Ui} <- brackets needed
- Click ok and into the schematic window
- Click ".op" to add directive
- Type .PARAM Ui=10
- Click ok and place the directive in the schematic
- Give R1 the value {R} with 1Ohm



II. Realizing circuit schematics with parametrized components

Stepping

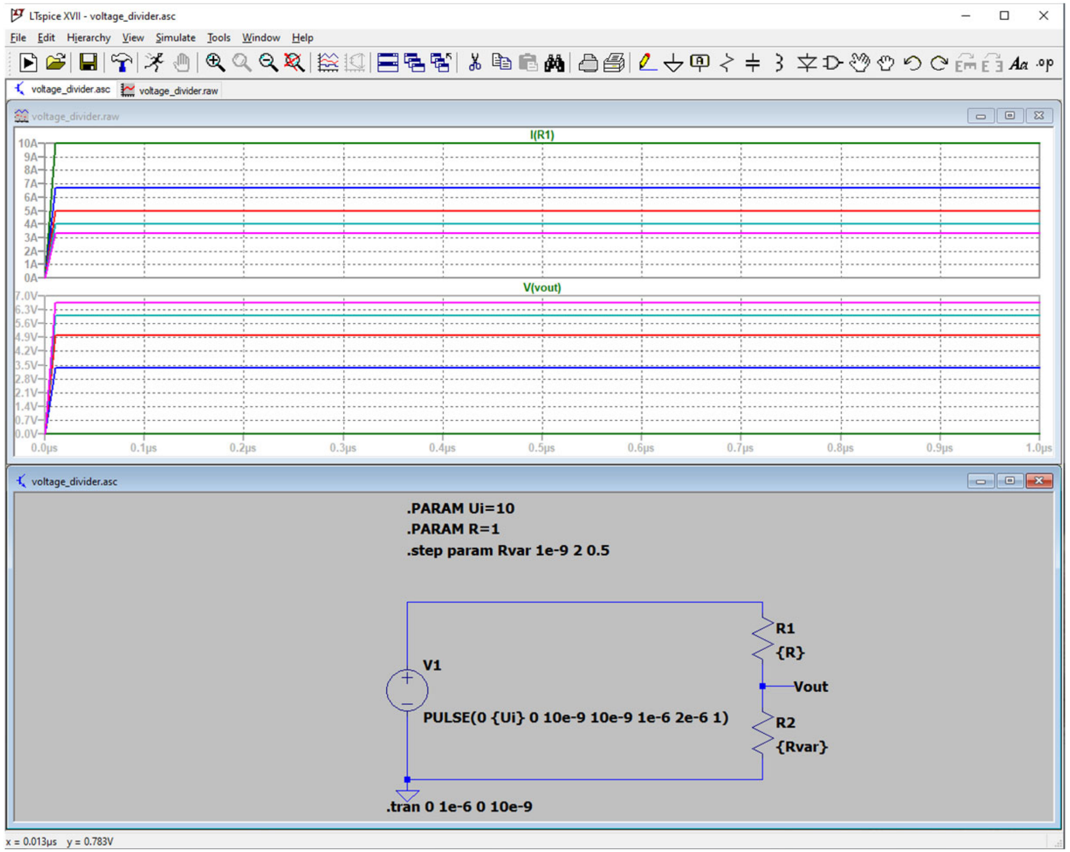
- Click “.op” to add directive
- Type .step param Rvar
- Click ok and place the directive in the schematic
- Right click on the directive
- Give Rvar these values:
  - Start value: 0
  - Stop value: 2
  - Increment: 0.5
- Click ok and give R2 the value {Rvar}
- Click “save” and “run”



II. Realizing circuit schematics with parametrized components

Voltage Source Configuration

- Right click V1
- Click “Advanced”
- Click “PULSE”
  - Vinitial: 0
  - Von: {Ui}
  - Tdelay: 0
  - Trise: 10e-9
  - Tfall:10e-9
  - Ton: 1e-6
  - Tperiode: 2e-6
  - Ncycles: 1
- Click ok and “run”

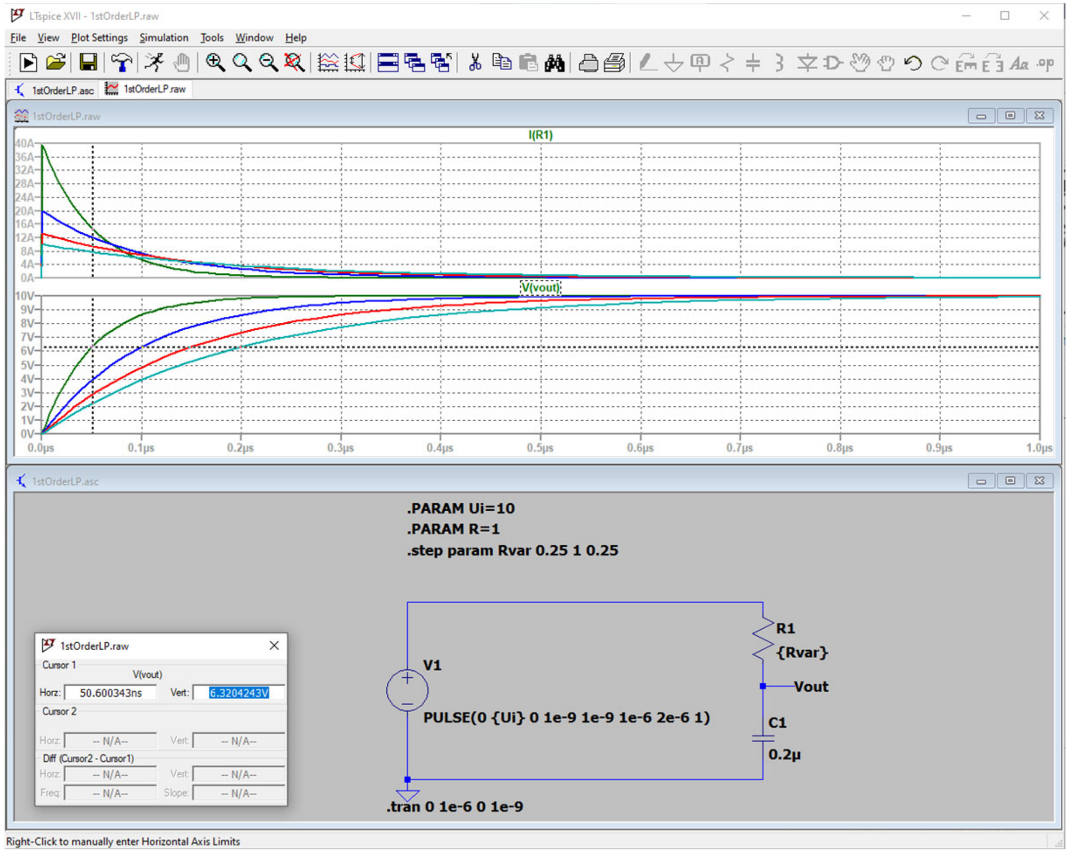


II. Realizing circuit schematics with parametrized components

Exercise Low Pass Filter

<u>Exercise:</u>

- Create new schematic: 1stOrderLP.asc
- Built the schematic as shown right.
- Add a capacitor C1 with the value of 0.2uF
- Simulate Vout for stepping R1 from 0.25Ohm to 1Ohm in 0.25Ohm steps



Exercise Low Pass Filter

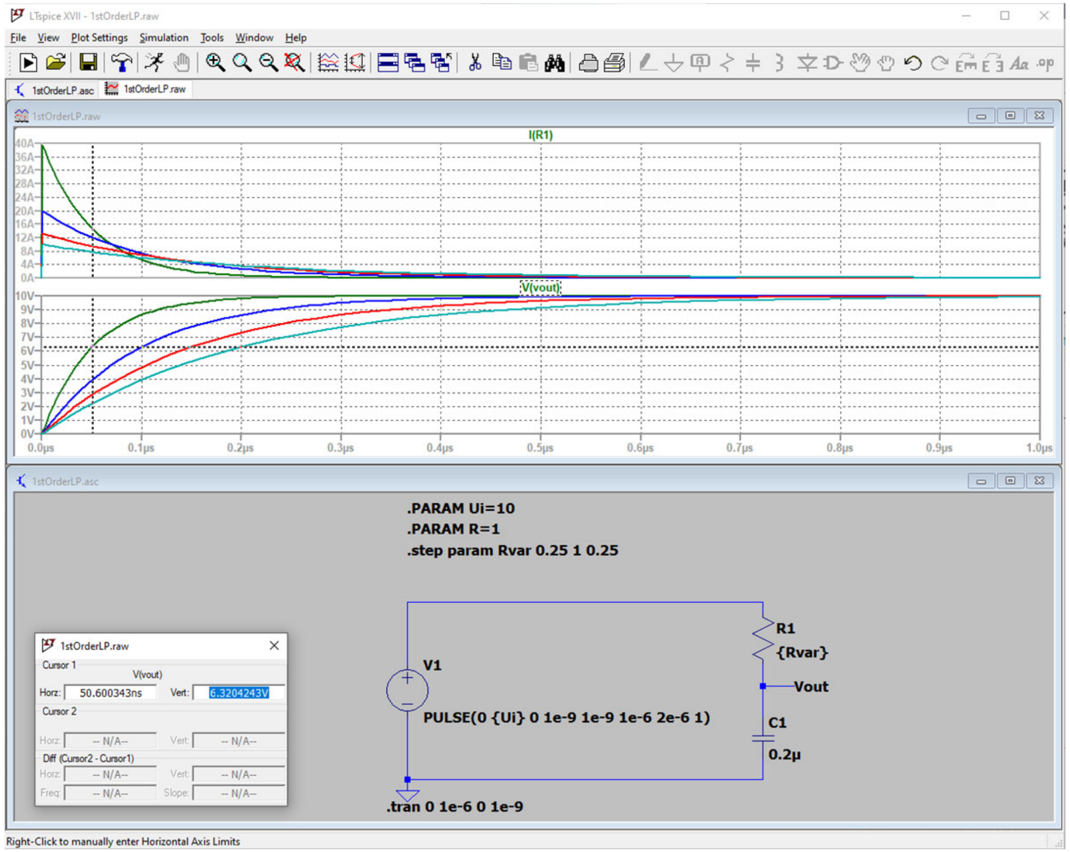
Exercise:

$V_{out} = U_i * (1 - e^{\frac{-t}{\tau}})$ with $\tau = RC$

Measure τ of the charging curves and write down!

(τ = time at which V_{out} =0.632*U_i)

- 1. τ =50.6ns ≈ 0.25Ohm*0.2uF
- 2. τ =100.3ns ≈ 0.50Ohm*0.2uF
- 3. τ =150.1ns ≈ 0.75Ohm*0.2uF
- 4. τ =200.7ns ≈ 1.00Ohm*0.2uF



Agenda

realistic simulation of a RLC (Resistor-Inductor-Capacitor) resonant circuit:

I. configuration of Ltspice (solver, simulation configuration, appearance etc.)
II. realizing circuit schematics with parametrized components
III. realizing component models (diodes, switches etc)
IV. transient analysis of the built up RLC circuit
V. AC analysis of the built up RLC circuit

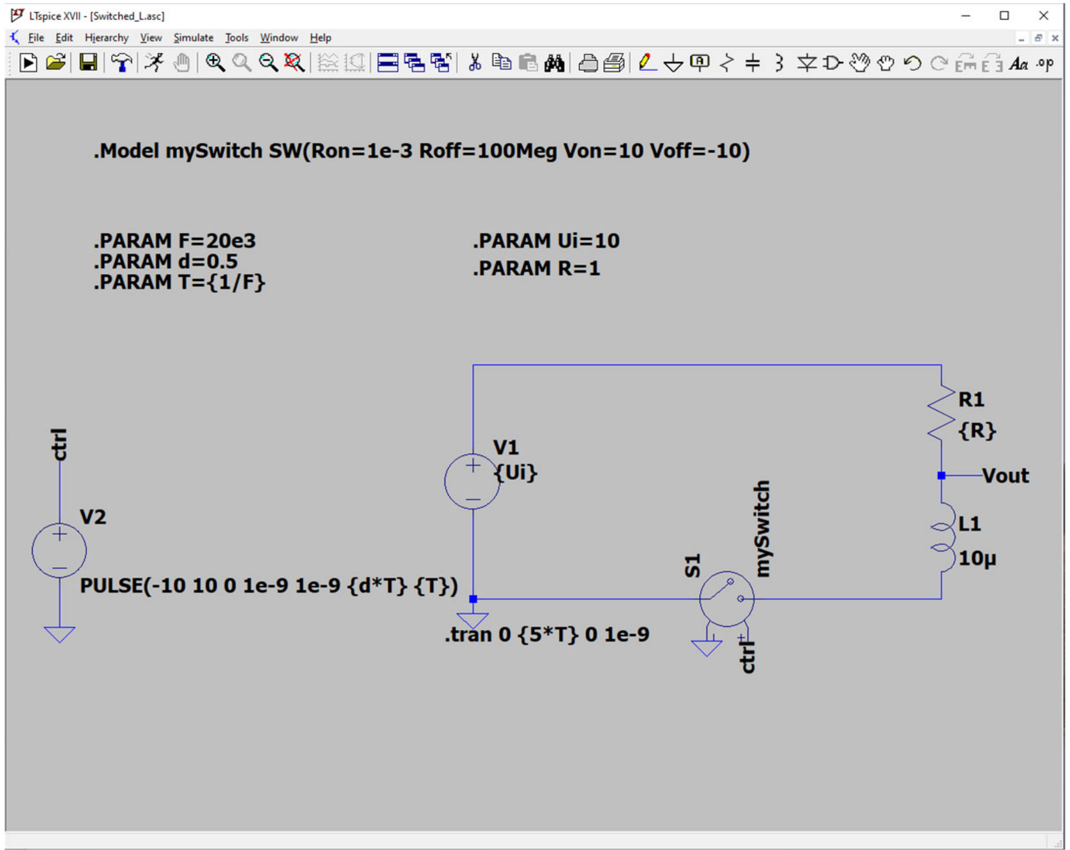
III. Realizing component models

Switch Model

- Create new schematic “Switched_L.asc” as shown on the right.
- Click “component”
- Add an ideal switch “sw” to the schematic
- Add a label “ctrl” to the + connection and Ground to the – connection of the switch
- Add a voltage source with PULSE config.
  - -10, 10, 0, 1e-9, 1e-9, {d*T}, {T}
  - add the variables F=20e3, d=0.5, T={1/F}
- Change stop time to {5*T}
- Add directive “.op”:

.MODEL mySwitch SW(Ron=1e-3 Roff=100Meg Von=10 Voff=-10)

- Change value of S1 from SW to mySwitch

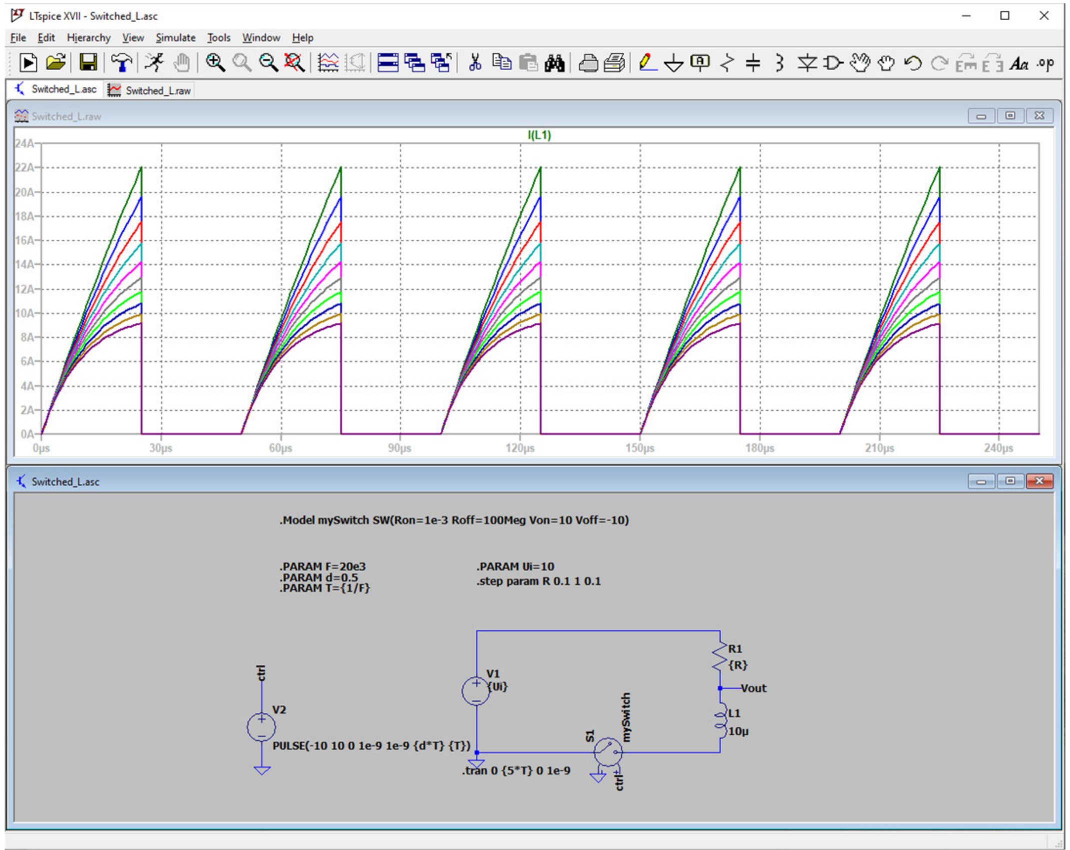


III. Realizing component models

Exercise Switch Model

<u>Exercise:</u>

- Simulate I(L1) for stepping R1 from 0.1Ohm to 1Ohm in 0.1Ohm steps
- Explain results
- Why is the voltage across L1 in Mega Volt at switch turn off?



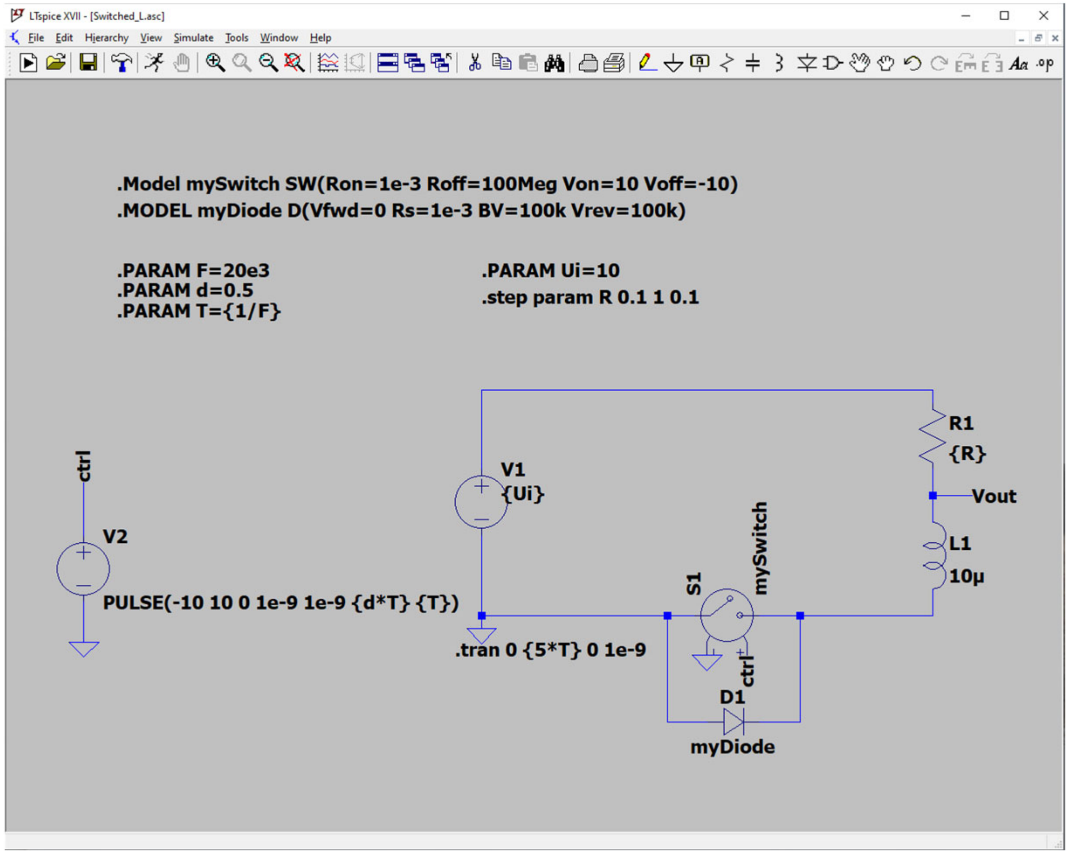
III. Realizing component models

Diode Model

- Modelling of the switch's body diode (MOSFET)
- Add a "Diode" in parallel to the switch as shown
- Add a diode model:

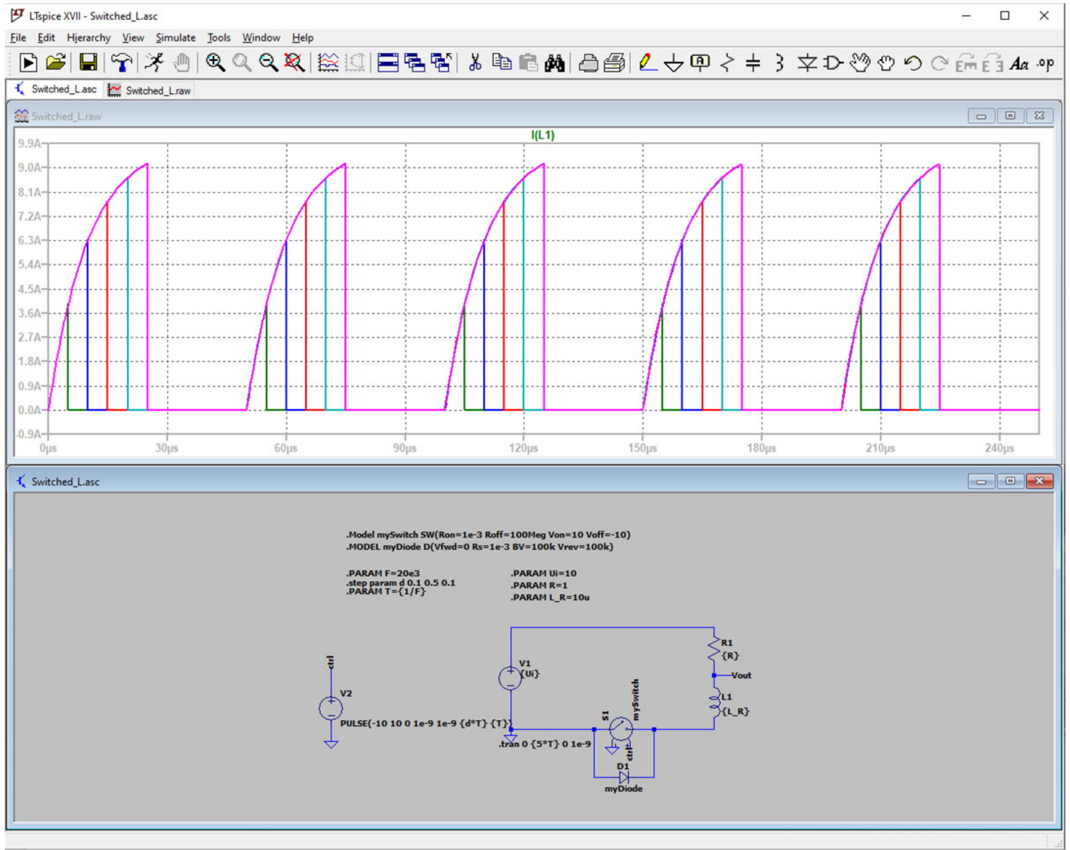
  .MODEL myDiode D(Vfwd=0 Rs=1e-3 BV=100k

  Vrev=100k)
- Add the model to the D1's value by right clicking "D"
- Click save



Exercise I:

- Add parameter “L_R=10u” and configurate the value of L1
- Change the parameter of R to 1Ohm
- Simulate I(L1) for stepping “d” from 0.1 to 0.5 in 0.1 steps
- Explain results, what does “d” do?



realistic simulation of a RLC (Resistor-Inductor-Capacitor) resonant circuit:

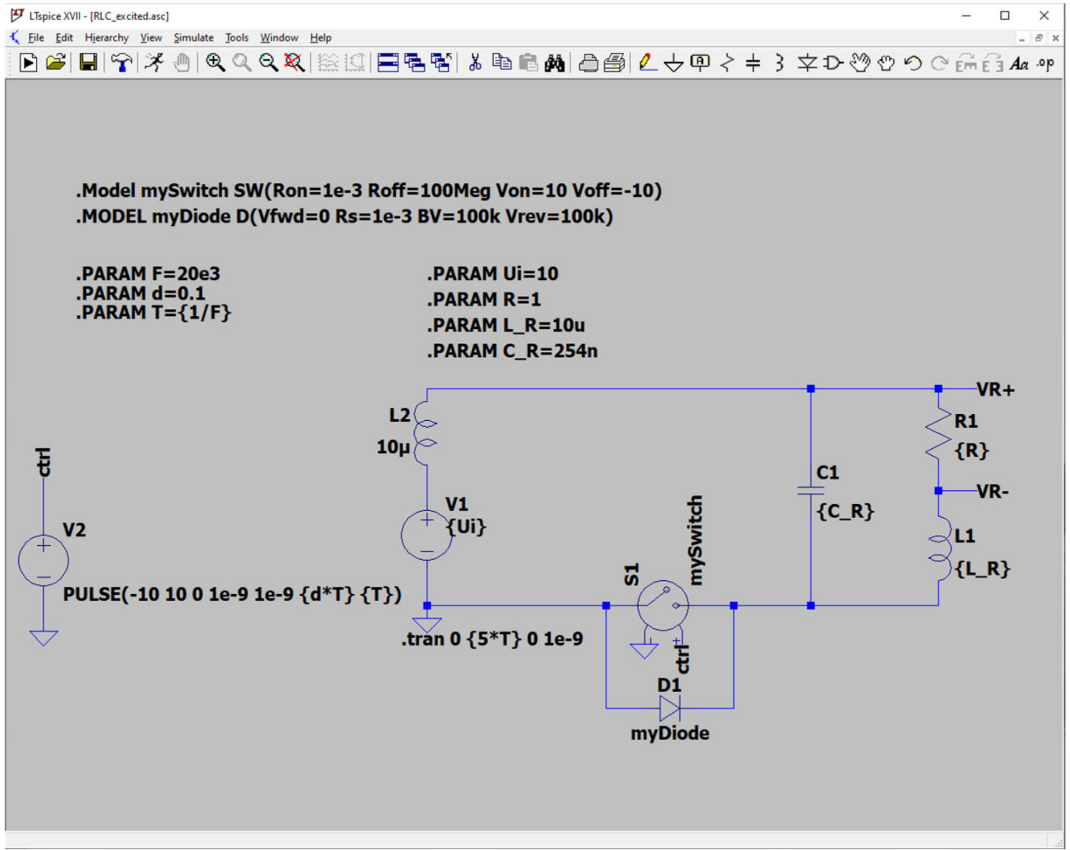
I. configuration of Ltspice (solver, simulation configuration, appearance etc.)
II. realizing circuit schematics with parametrized components
III. realizing component models (diodes, switches etc)
IV. transient analysis of the built up RLC circuit
V. AC analysis of the built up RLC circuit

IV. Transient analysis of the built up RLC circuit

RLC resonant circuit

Exercise:

- Save and save the schematic as RLC_excited.asc
- Add a capacitor C1 in parallel to R1 and L1
- Give C1 the value C_R with the parameter C_R=254n
- Give the nodes above and below R1 the names VR+ and VR-
- Adjust a duty cycle d of 0.1
- Add an Inductor L2 in series to V1 and give a value of 10uH
- What is L2 for?



IV. Transient analysis of the built up RLC circuit

RLC resonant circuit

Exercise:

1. $$v_{C1} = -v_{R1} - v_{L1} = -i \cdot R_1 - L_1 \cdot \frac{di}{dt}$$

2. $$i = C_1 \cdot \frac{dv_{C1}}{dt}$$

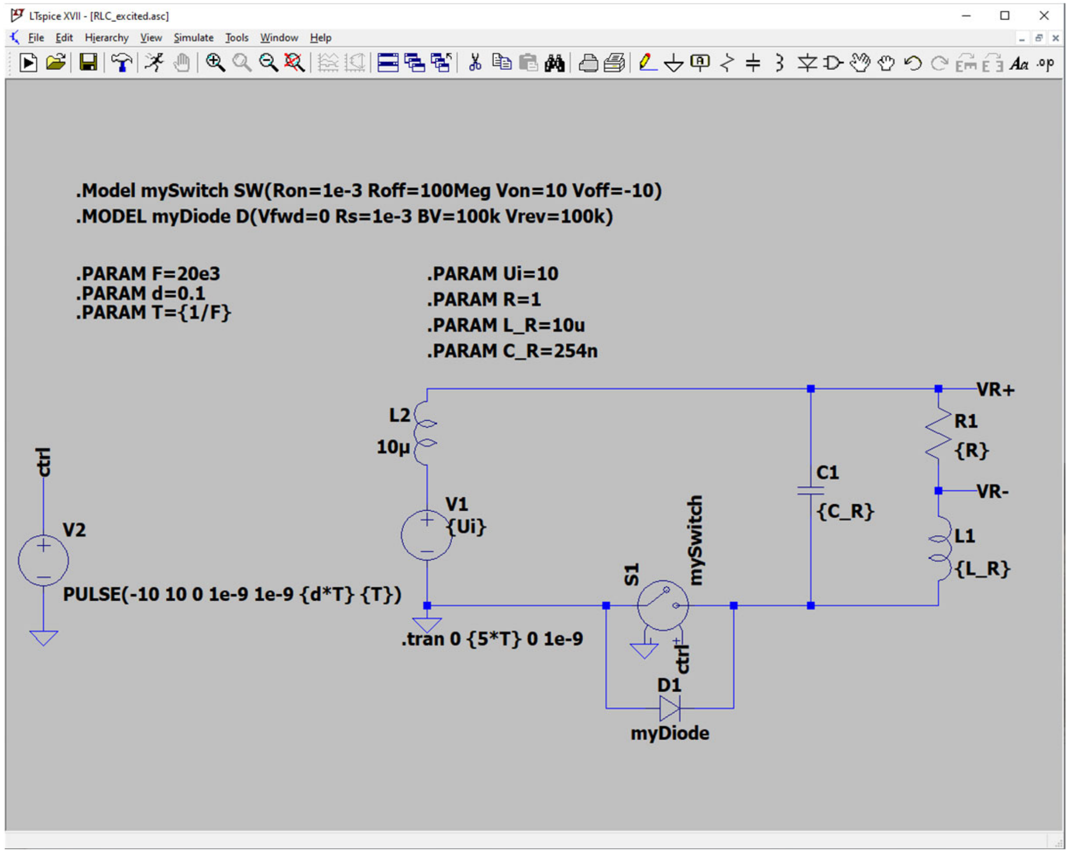
1. in 2. $$i = C_1 \cdot \frac{d}{dt}\left(-i \cdot R_1 - L_1 \cdot \frac{di}{dt}\right)$$

$$0 = \frac{d^2 i}{dt} + \frac{R_1}{L_1} \cdot \frac{di}{dt} + \frac{1}{L_1 C_1} \cdot i$$

With $\frac{R_1}{L_1} = 2\delta$ and $\frac{1}{L_1 C_1} = \omega_0^2$

δ= Damping factor

ω_0= $2\pi f_0$ resonant frequency in angular time



IV. Transient analysis of the built up RLC circuit

RLC resonant circuit

Exercise:

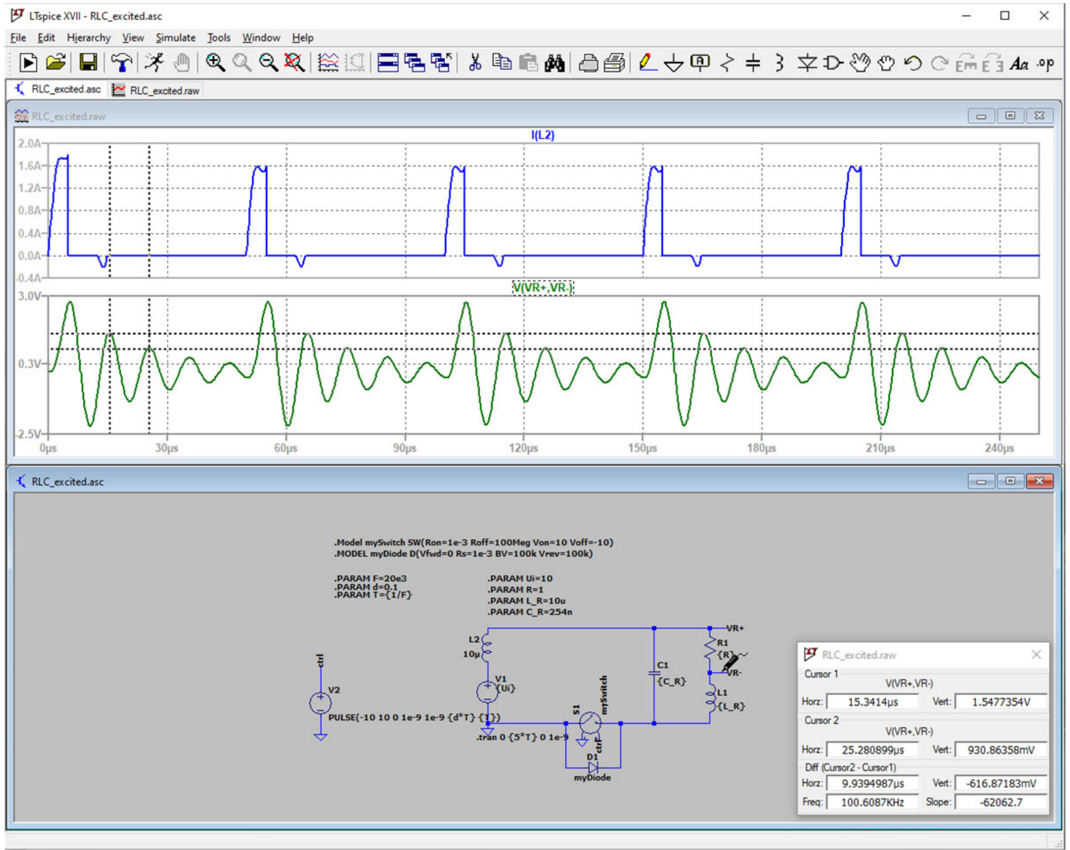
$$0 = \frac{d^2 i}{dt} + 2\delta \cdot \frac{di}{dt} + \omega_0^2 \cdot i$$

Logarithmic Decrement

$$\delta = \frac{R_1}{2L_1} = \frac{\Lambda}{T} = \frac{1}{T} \cdot \ln\left(\frac{A(t_1)}{A(t_1 + T)}\right)$$

$$\delta_{opt} = \frac{1\Omega}{2 \cdot 10\mu H} = 50000 \cdot \frac{1}{s}$$

$$\delta_{sim1} = \frac{1}{9.9395\mu s} \cdot \ln\left(\frac{1.54774V}{0.930864V}\right) = 51153 \cdot \frac{1}{s}$$



IV. Transient analysis of the built up RLC circuit

RLC resonant circuit

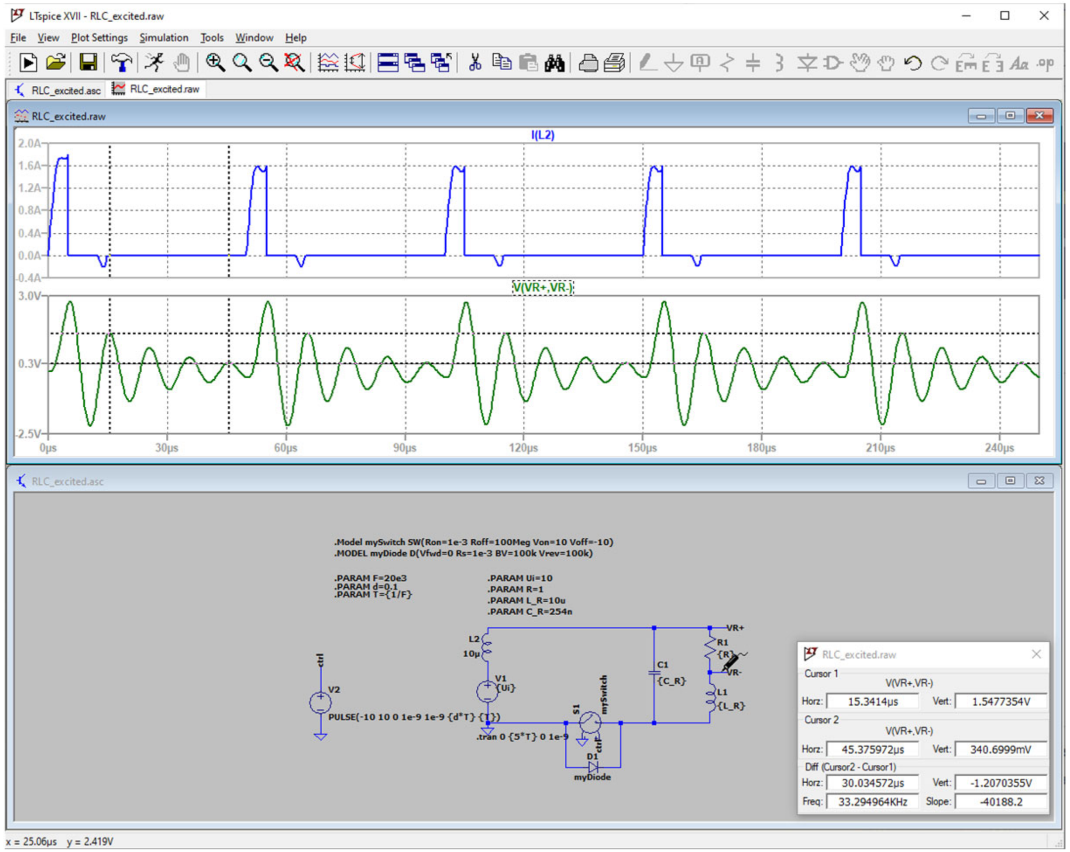
Exercise:

$$\delta_{opt} = \frac{1\Omega}{2 \cdot 10\mu H} = 50000 \cdot \frac{1}{s}$$

$$\delta_{sim1} = \frac{1}{9.9395\mu s} \cdot \ln\left(\frac{1.54774V}{0.930864V}\right) = 51153 \cdot \frac{1}{s}$$

$$\delta_{sim2} = \frac{1}{30.034572\mu s} \cdot \ln\left(\frac{1.54774V}{0.3407V}\right) = 50394 \cdot \frac{1}{s}$$

-> for non-linear systems, measuring over more periods gives a more accurate value!



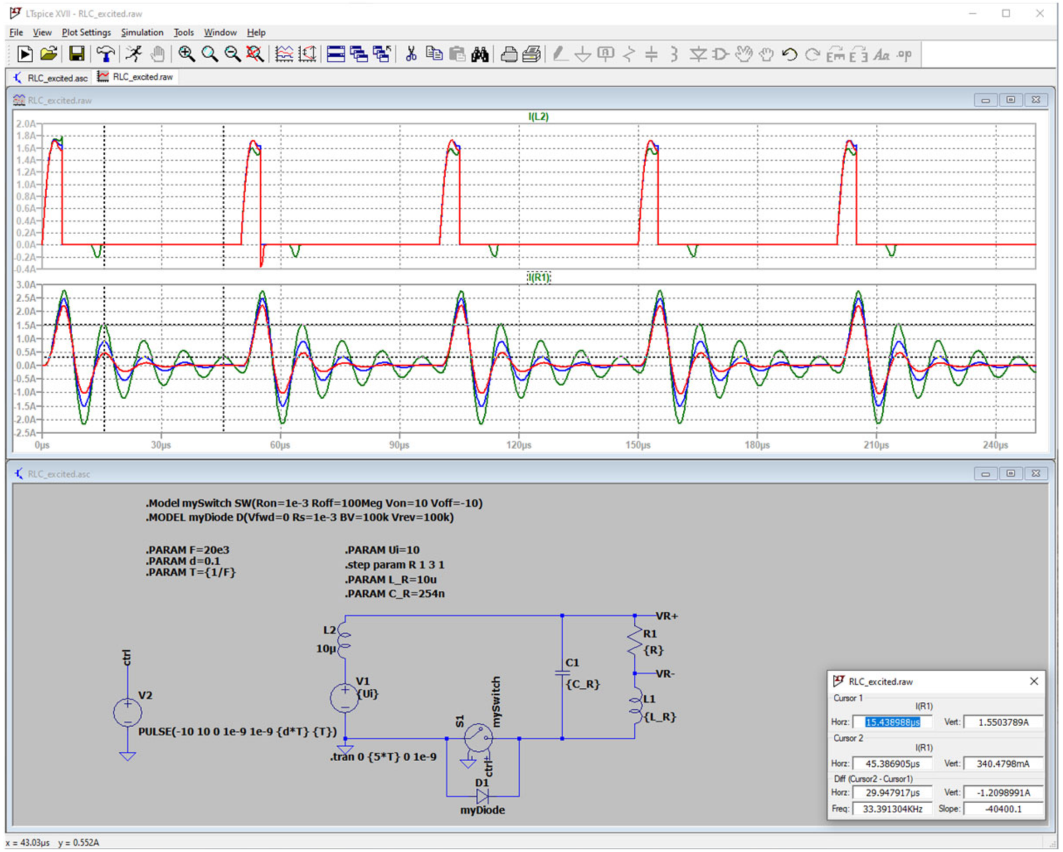
IV. Transient analysis of the built up RLC circuit

RLC resonant circuit

Exercise:

R_1	L_1	δ_{opt}	δ_{sim}
1	5µH	0.100*10^6 1/s	0.100145*10^6 1/s
1	20µH	0.025*10^6 1/s	0.025035*10^6 1/s
1	10µH	0.050*10^6 1/s	0.050030*10^6 1/s
2	10µH	1.0*10^6 1/s	0.997085*10^6 1/s
3	10µH	1.5*10^6 1/s	1.481884*10^6 1/s

$$\delta = \frac{R_1}{2L_1} = \frac{\Lambda}{T} = \frac{1}{T} \cdot \ln\left(\frac{A(t_1)}{A(t_1 + T)}\right)$$



realistic simulation of a RLC (Resistor-Inductor-Capacitor) resonant circuit:

I. configuration of Ltspice (solver, simulation configuration, appearance etc.)

II. realizing circuit schematics with parametrized components

III. realizing component models (diodes, switches etc)

IV. transient analysis of the built up RLC circuit

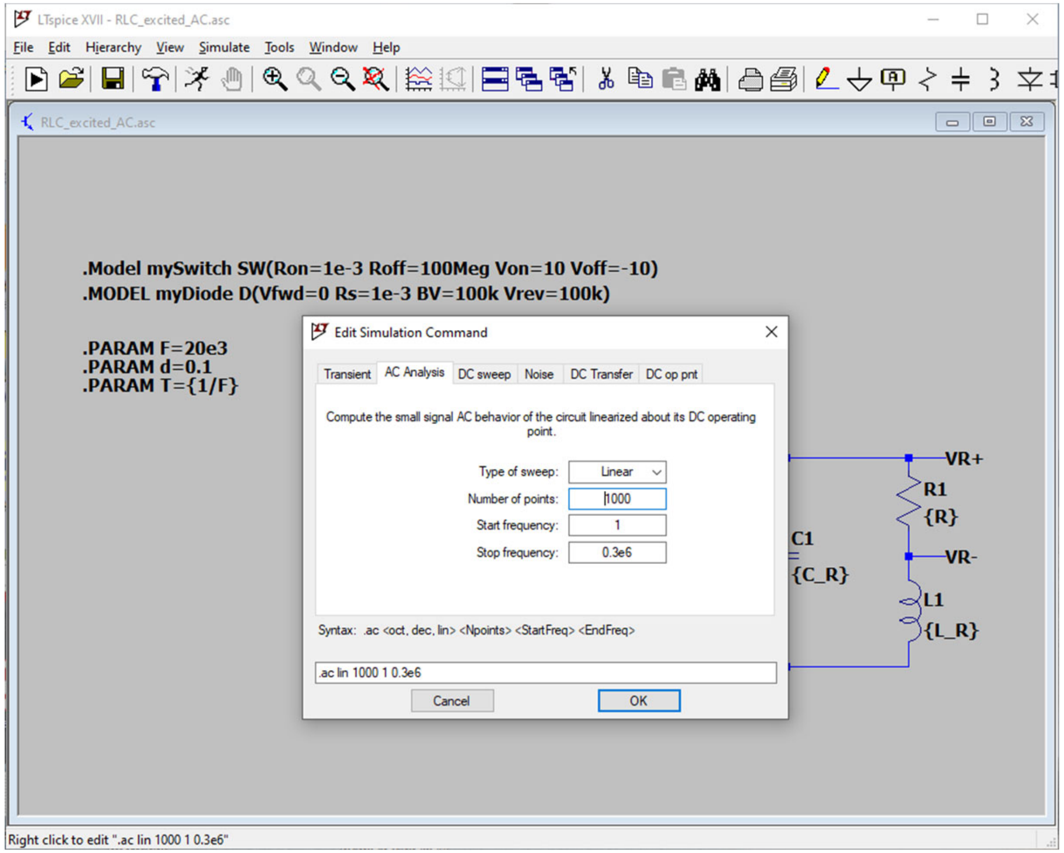
V. AC analysis of the built up RLC circuit

V. AC analysis of the built up RLC circuit

RLC resonant circuit

<u>Exercise:</u>

- Save and save the schematic as RLC_excited_AC.asc
- Replace the switch and body diode by a wire
- Delete V2 and replace L2 and V1 with a current source
- Right click the current source I1
- Click “Advanced”, put 1 Ampere as AC Amplitude
- Right click .tran operation and go to AC Analysis
- Type of sweep: linear, points: 1000,
- start freq: 1Hz, stop freq: 300kHz



V. AC analysis of the built up RLC circuit

RLC resonant circuit

Exercise:

- Run simulation and display V(vr+)
- Right Click the trace name V(vr+)
- Divide V(vr+) bý I(I1) to get impedance
- Simulate for R=1Ohm, 4Ohm and 7Ohm

R_1	L_1	C_1	f_{opt}	f_{sim}
1	10µH	254nF	99.55kHz	99.6454kHz
4	10µH	254nF	94.65kHz	96.71kHz
7	10µH	254nF	82.88kHz	78.91kHz

$$\omega = \sqrt{\omega_0^2 - \delta^2} \qquad f = \frac{\sqrt{\frac{1}{LC} - \left(\frac{R}{2L}\right)^2}}{2\pi}$$