A Tutorial Introduction to Elements

The Free SIMPLIS Demonstration Version

Christophe Basso

Business Development Manager

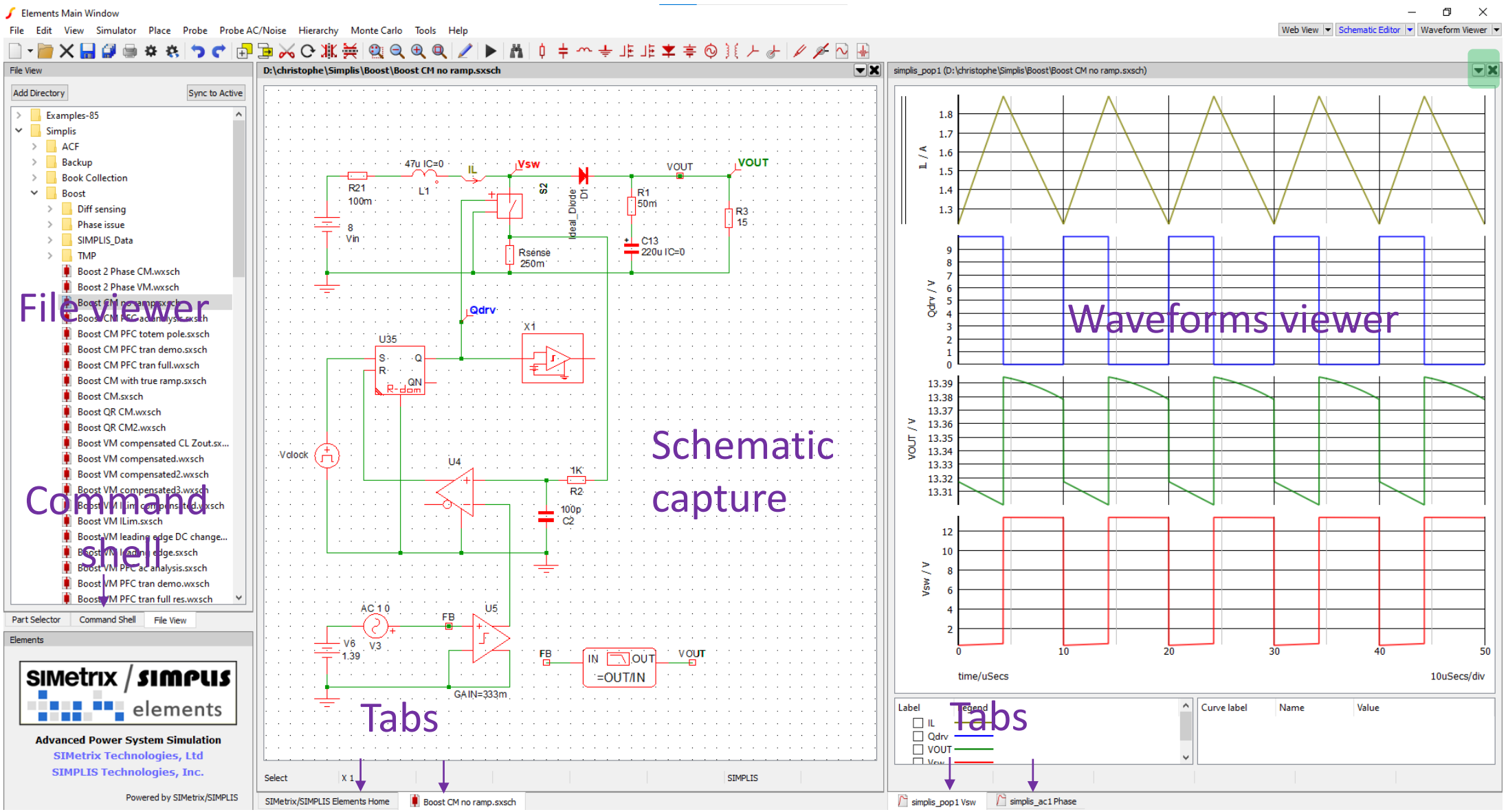
IEEE Senior Member

November 2022 – Rev 0.34

Agenda

- What is Elements?
- Running a Basic Simulation
- A Buck Converter
- Ac Linear Analysis with SIMPLIS
- Importing a SPICE Model
- The Ready-to-Use Template

Elements is the Demonstration Version of SIMPLIS



Download and install the demonstration version here:

https://www.simplistechnologies.com/product/elements

Home

SIMetrix/SIMPLIS Elements SIMulation of Piecewise LInear Systems

SIMetrix/SIMPLIS Elements is a free-to-download version of our software that offers full schematic capture and waveform viewing/analysis capability along with a host of documentation and training materials designed to help users get up-to-speed quickly with SIMetrix/SIMPLIS' simulation capabilities.

The latest version of SIMetrix/SIMPLIS Elements is **v8.50d**, released on October 5th, 2021.

SIMetrix/SIMPLIS Elements Installer (~109MB) DOWNLOAD

Elements is a free version of the program with no license or copying restrictions. Virtually all features are enabled but a circuit size limit applies. The limits for the Elements versions are generous enough for them to be used for real work and we are happy for users to do so.

System Requirements

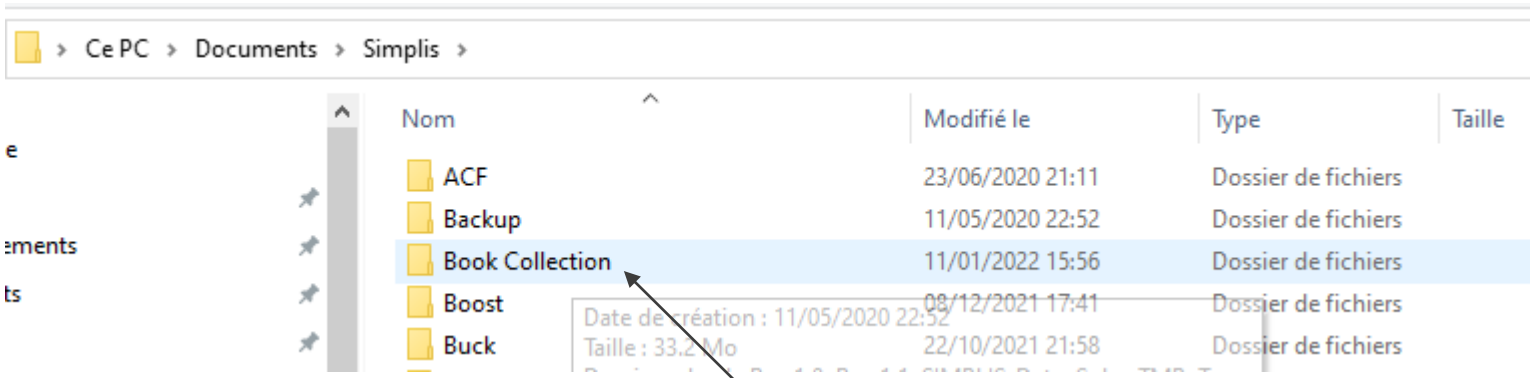
SIMetrix/SIMPLIS Elements requires Windows 10 **64 bit edition** (Home, Pro or Enterprise).

SIMetrix/SIMPLIS Elements Limitations

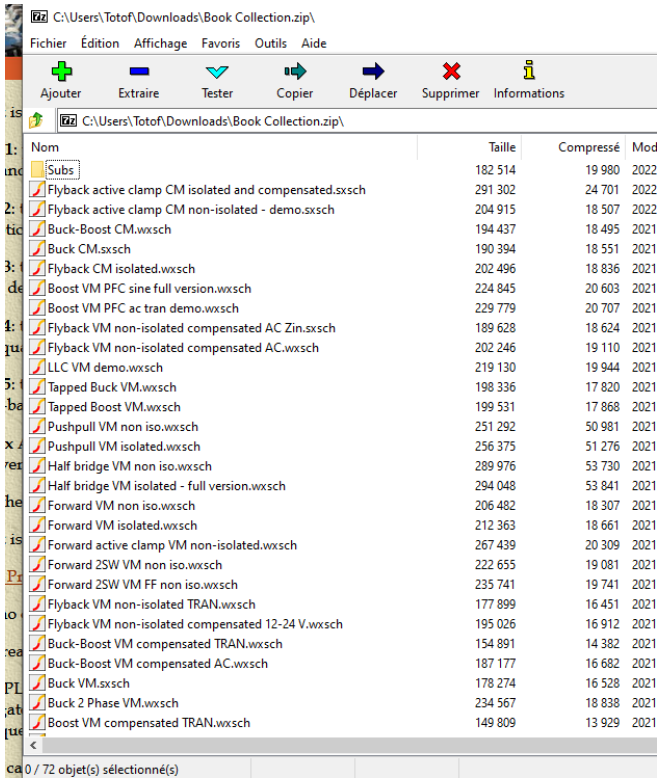
Then go to my webpage and download the ready-made templates:

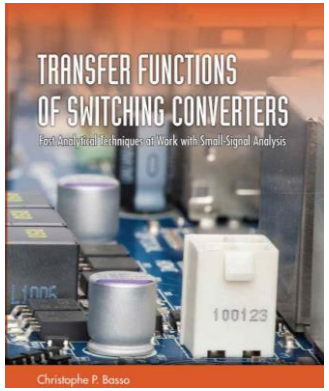
http://powersimtof.com/Downloads/Book/Book%20Collection.zip

Extract the files and place them, for instance, in a SIMPLIS directory created in <My Documents>



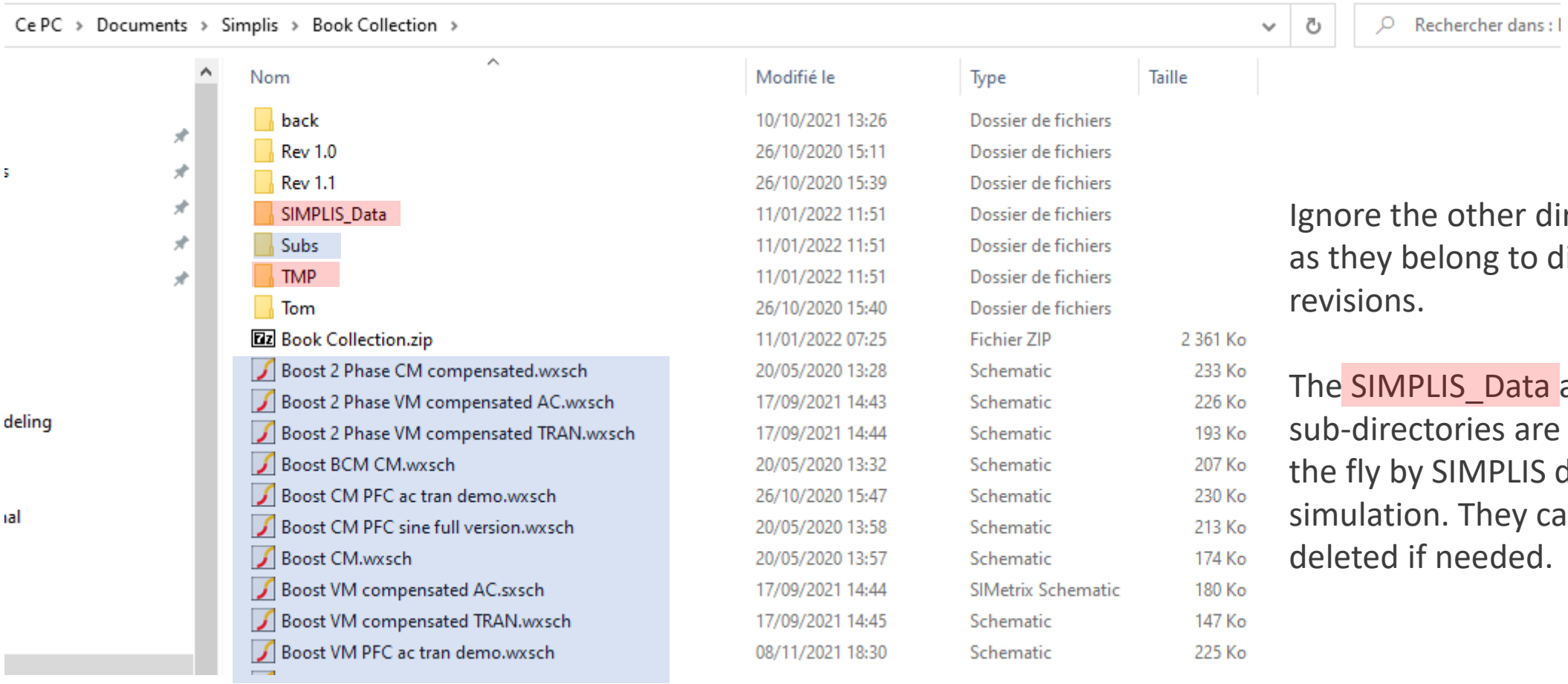
Extract the ZIP here





These files are examples I documented in my last book
Transfer Functions of Switching Converters – Faraday Press

Once installed, you have this structure:

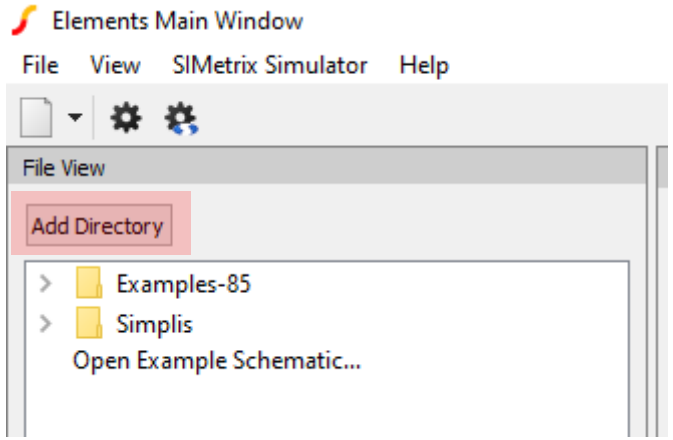


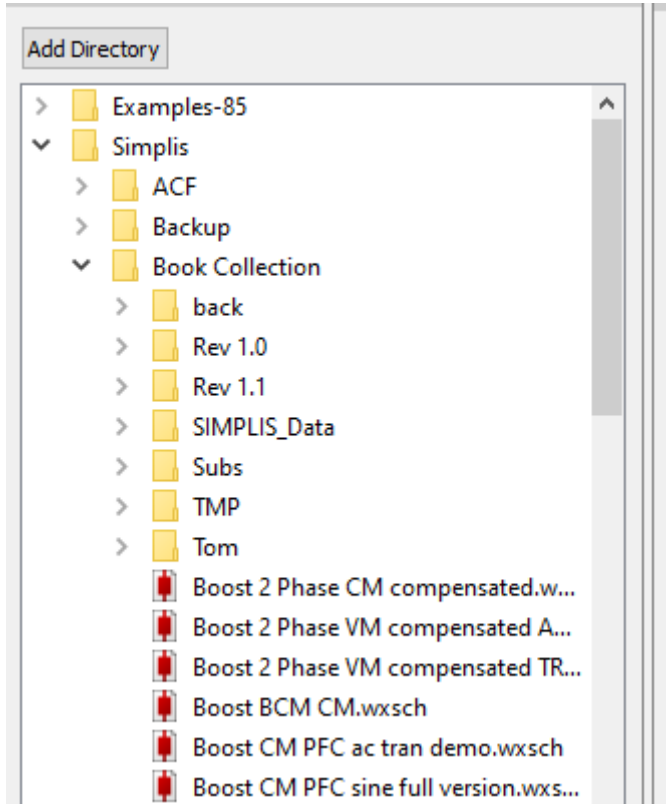
Ignore the other directories as they belong to different revisions.

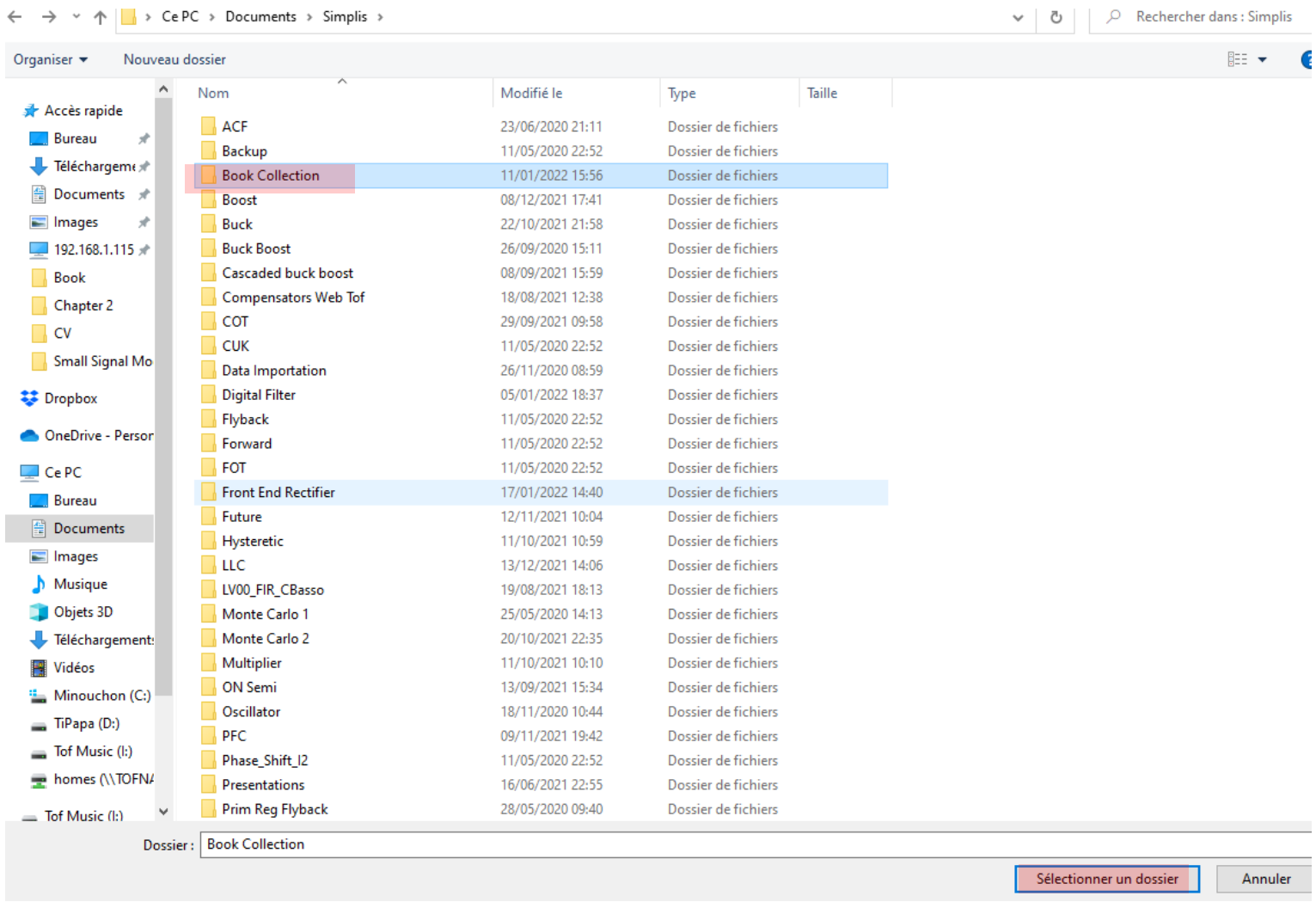
The SIMPLIS_Data and TMP sub-directories are filled on the fly by SIMPLIS during simulation. They can safely be deleted if needed.

I do not recommend to launch SIMPLIS by double-clicking on the file of interest. Open SIMPLIS and select the file of your choice instead.

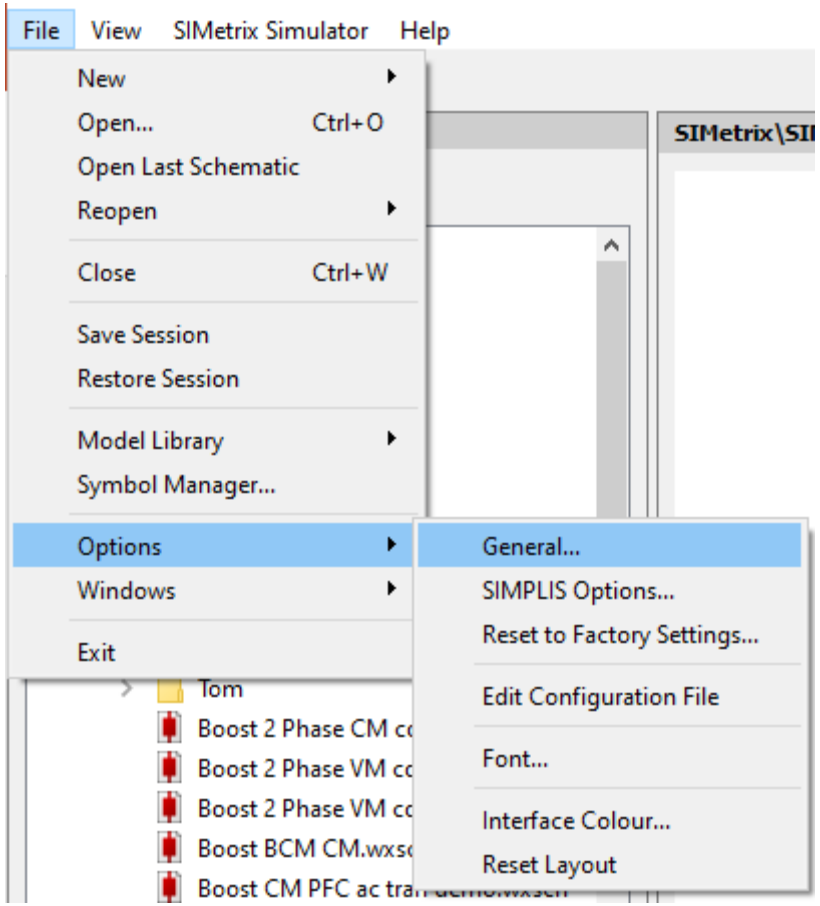
Once SIMPLIS Elements is installed, instruct the file viewer with the directory location:

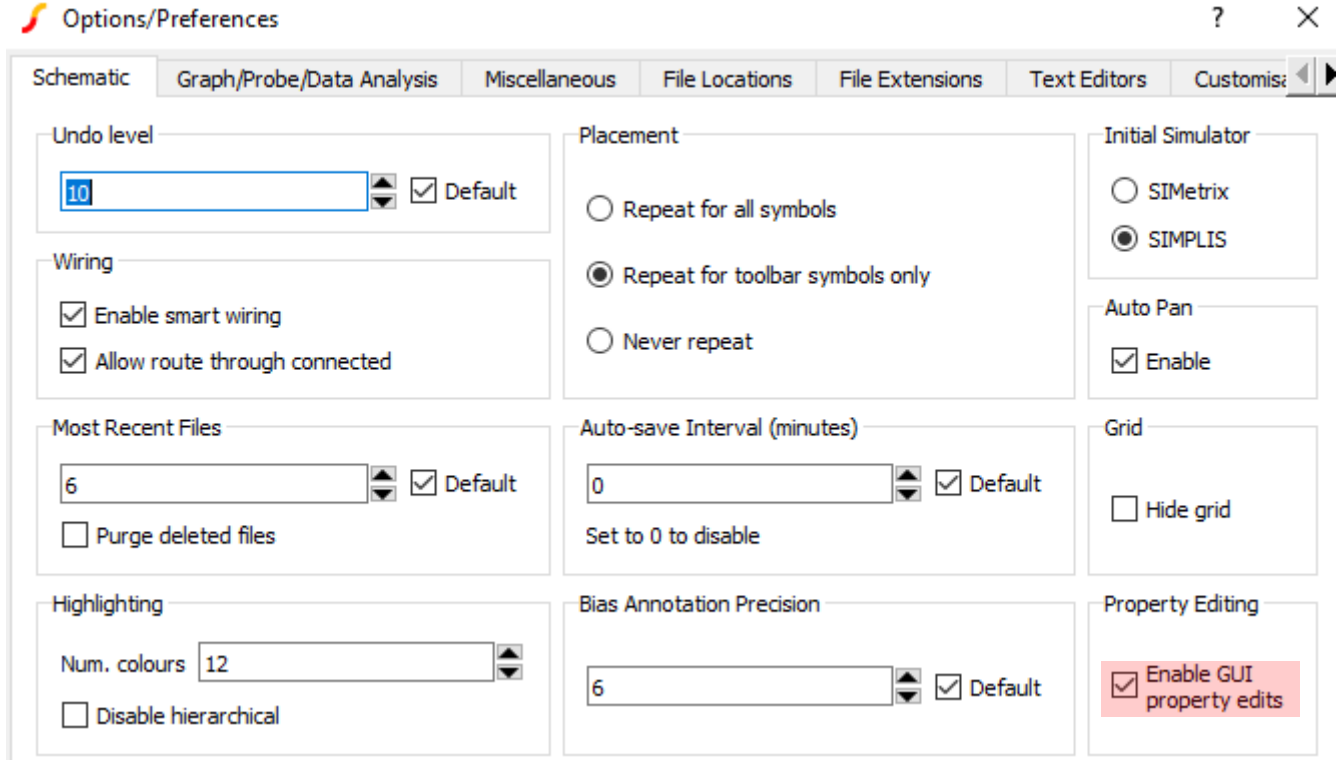




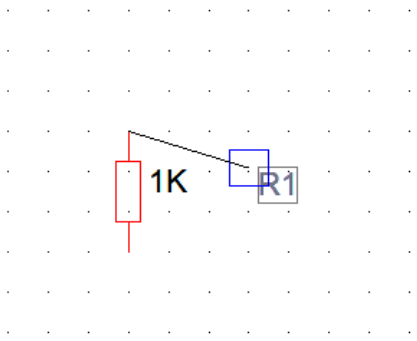


Then you can pick the file of your choice via this File View tab



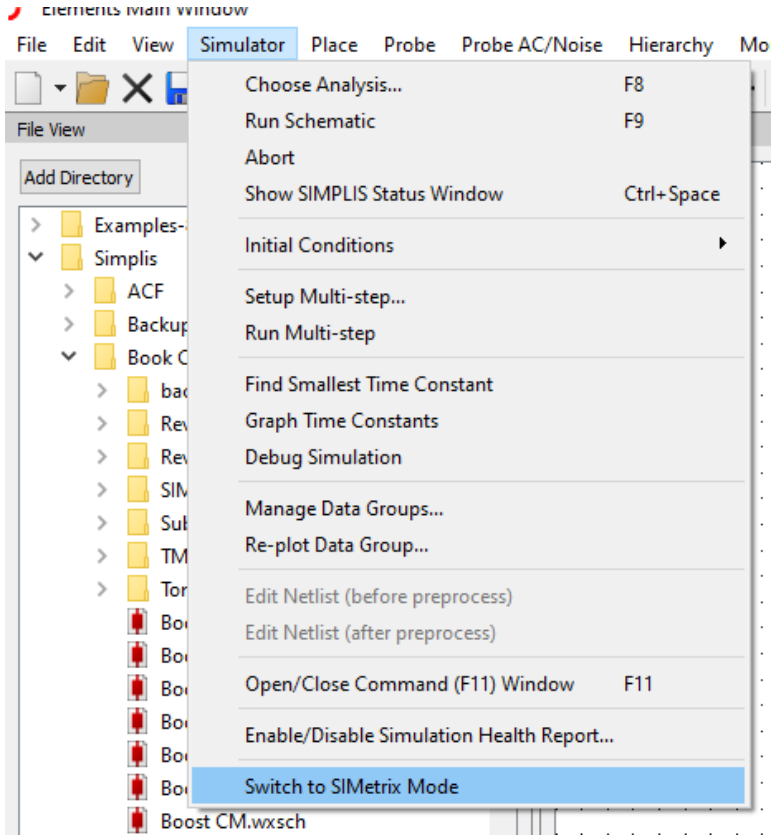


Edit property and move/align labels



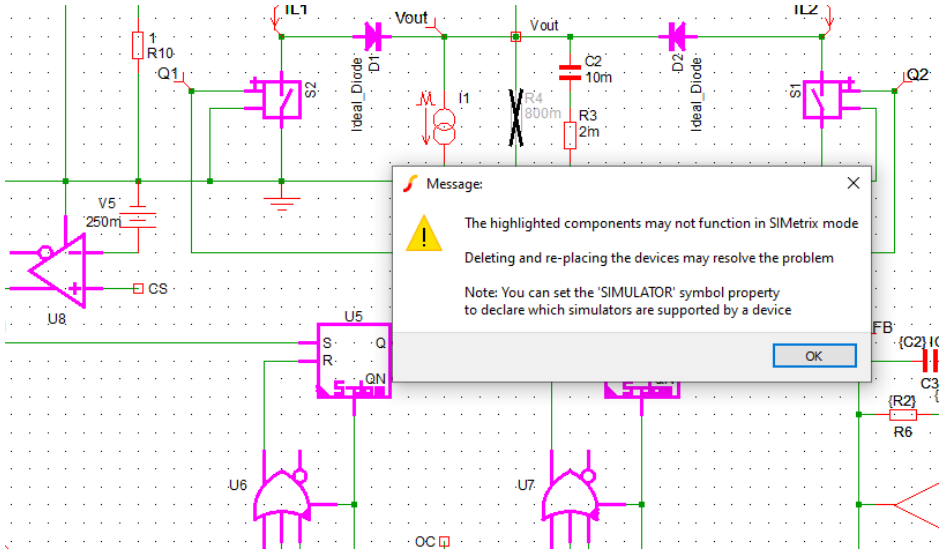
The program Elements gives access to two simulators:
SIMPLIS and SIMetrix

1. SIMetrix is a classical SPICE-based program and is compatible with many SPICE engines. It can perform ac and transient analyses as any SPICE package would do.
2. SIMPLIS is exclusively a time-domain simulator which can also simulate in ac and transient modes. SIMPLIS can extract the ac response from a switching circuit what SPICE cannot (easily) do.



You can switch between the two engines at any time

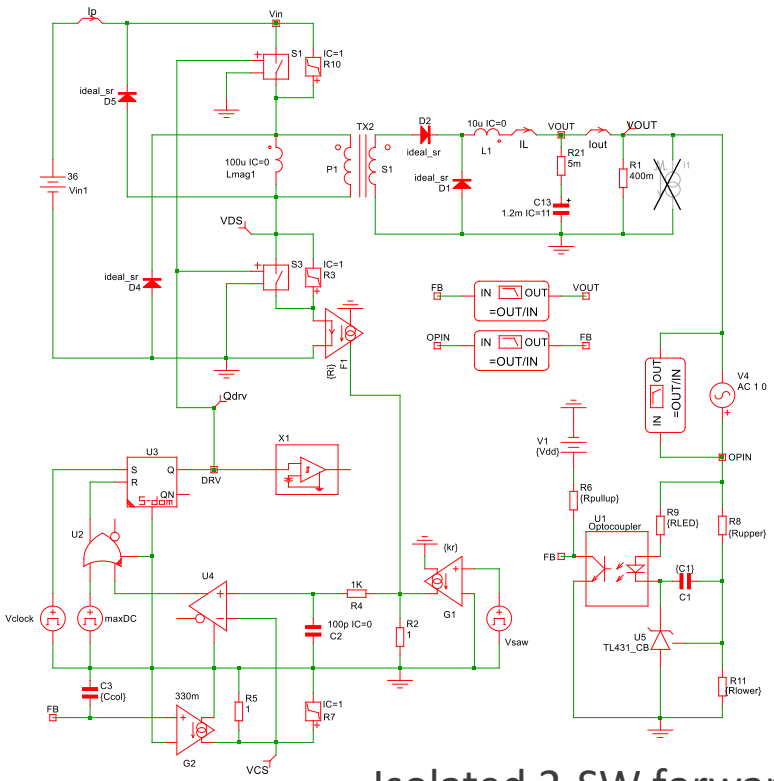
Some components are not compatible with both engines and the program warns you



SIMPLIS Circuit Size Limits

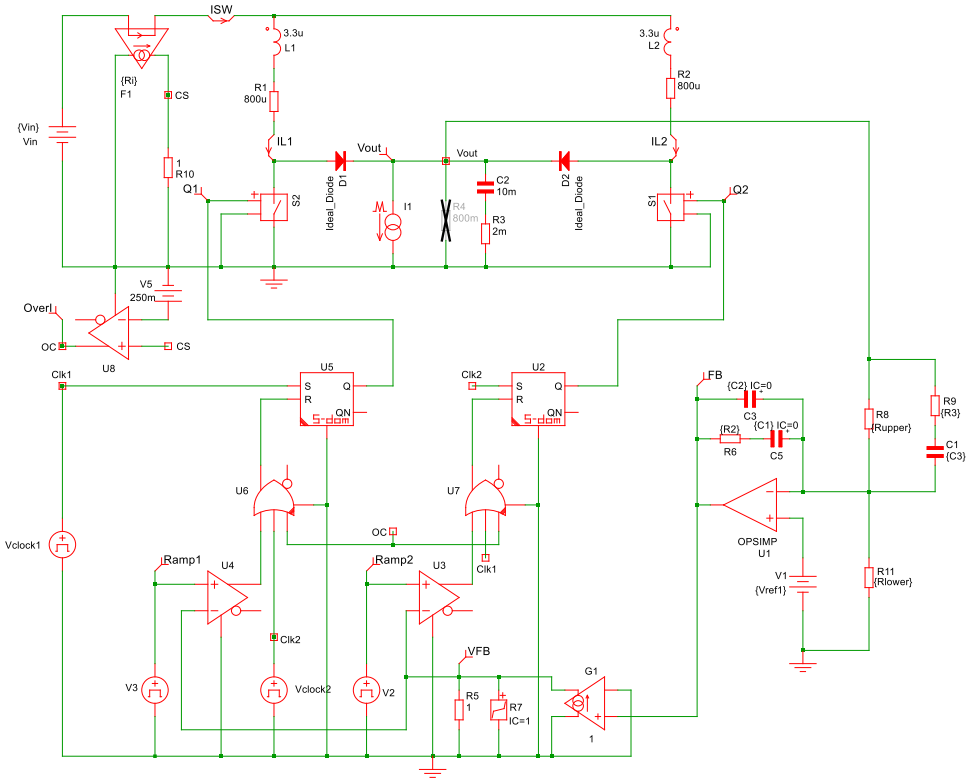
The exact limits for the SIMPLIS simulator are:

1. A total of no more than 15 state variables. A capacitor or an inductor each requires one state variable. Each time-varying or small-signal AC source requires one state variable, with the exception of SINusoidal or COSinusoidal sources, which require two state variables per source.
2. A total of no more than 10 capacitors or inductors combined.
3. A total of no more than six switches, simple or transistor.
4. A total of no more than six logic gates.
5. A total of no more than 26 "states." Each PWL element requires one state. Each switch requires one state. Each time-varying source requires one state. Each logic gate requires one state.
6. A total of no more than 100 new topologies. 100 topologies will be enough for simple switching circuits that use simple models only. More complex circuits or circuits that have more complicated models may exceed this limit



Isolated 2-SW forward

These are comfortable limits which let you simulate quite complex structures

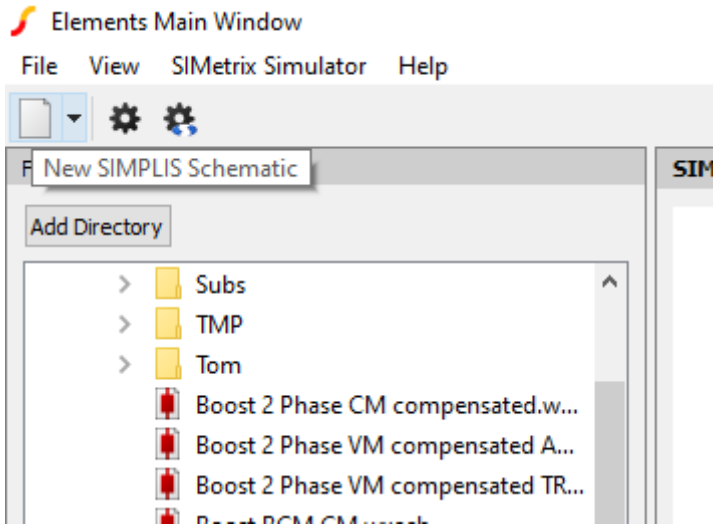


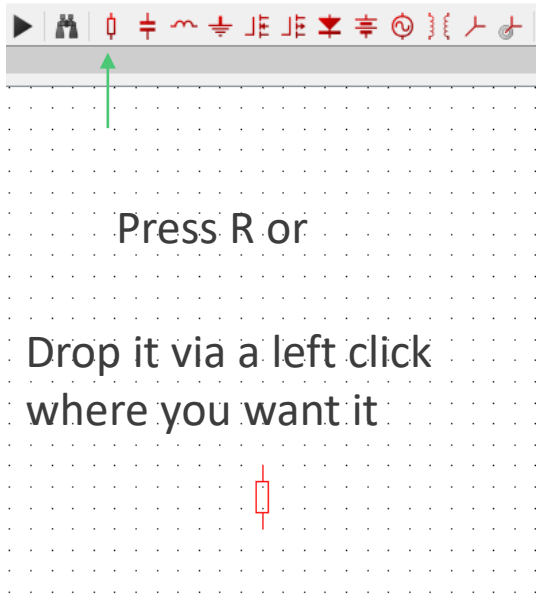
Compensated 2-phase boost

Agenda

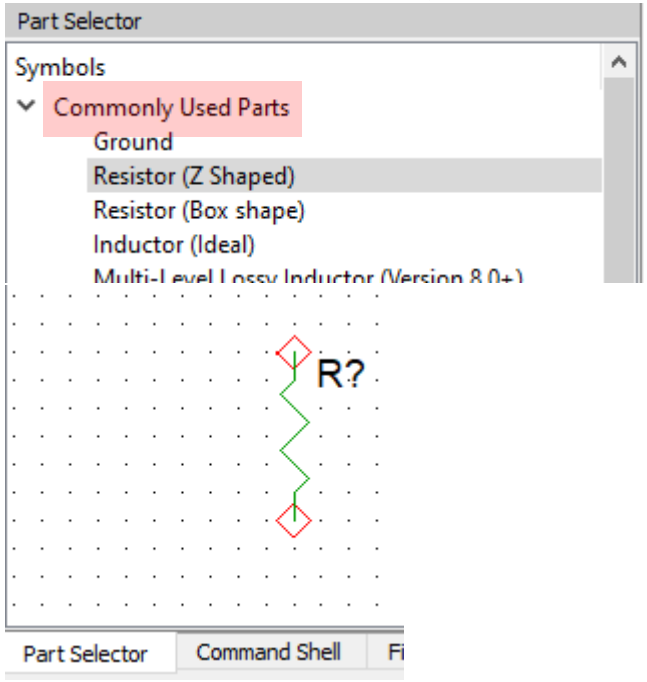
- What is Elements?
- Running a Basic Simulation
- A Buck Converter
- Ac Linear Analysis with SIMPLIS
- Importing a SPICE Model
- The Ready-to-Use Template

Let’s start with a simple simulation and a blank sheet





Box or Z-shape



Zoom-in and out by <CTRL> + central mouse wheel

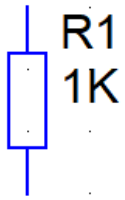
Hotkeys:

B → Probe
C → capacitor
D → diode
E → V-controlled V-source
F → I-controlled I-source
G → ground symbol
H → connecting port
I → current source
L → inductor
M → MOSFET
N → NPN bipolar
P → PNP bipolar
R → resistor
V → voltage source
W → waveform generator
Y → connecting port
Z → Zener diode

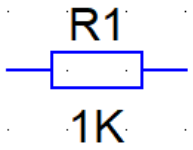
Zoom by <CTRL> + mouse wheel

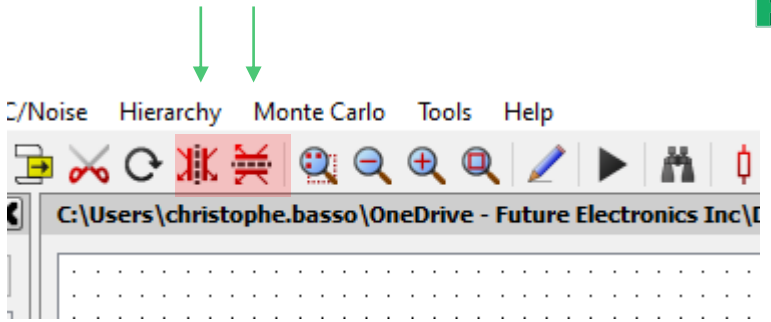
Panning with mouse: right-click and maintaining the pressure, the cursor turns to hand

Select the part
It becomes blue



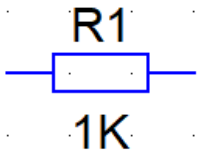
Rotate it with F5
Mirror it with F6

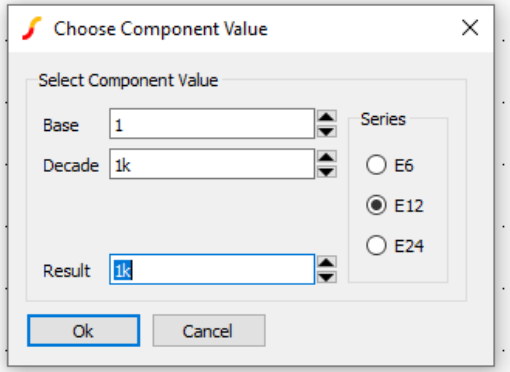




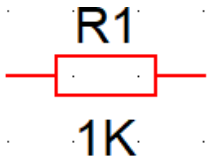
You can also use the toolbar buttons to rotate or mirror the part.

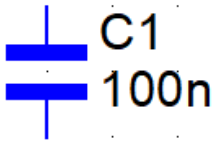
Change the component value
by double clicking



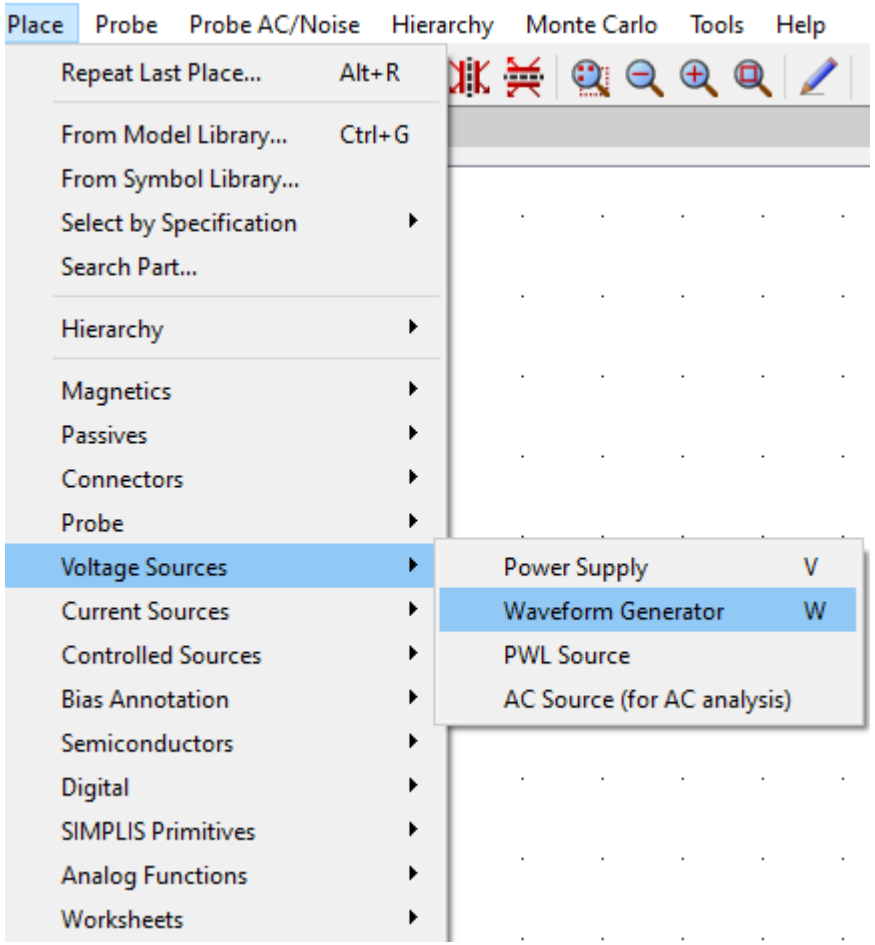


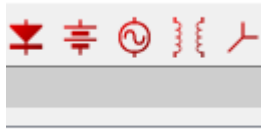
Press C or use toolbar

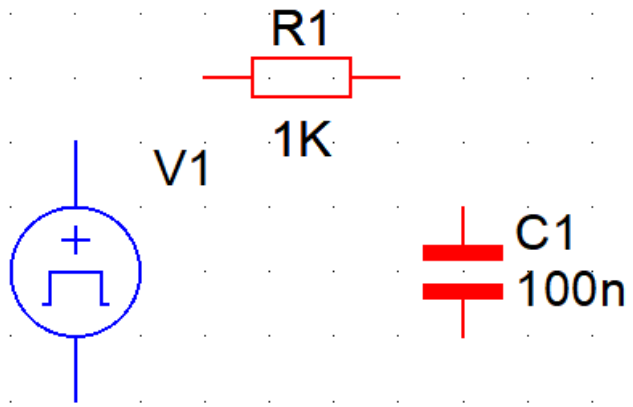


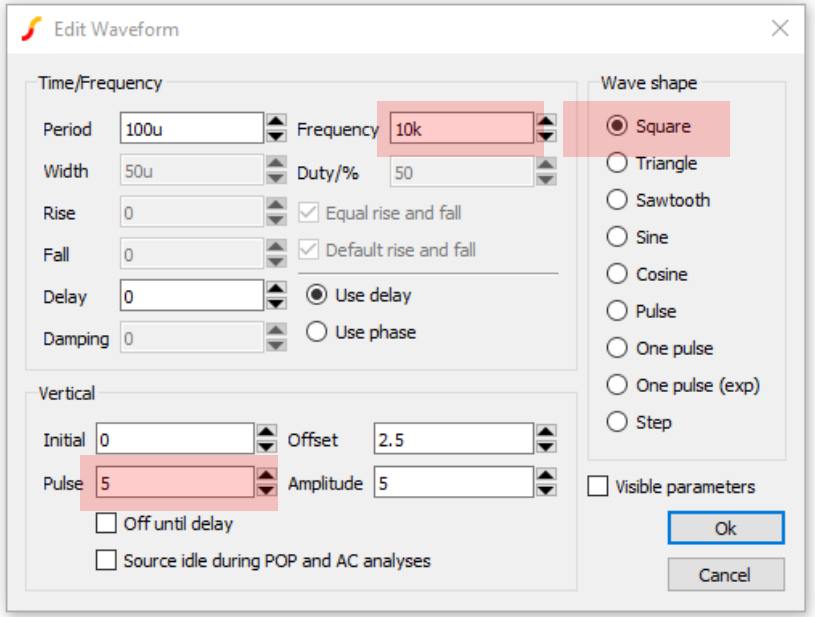


Add a waveform generator – press W or pick-up in the toolbar

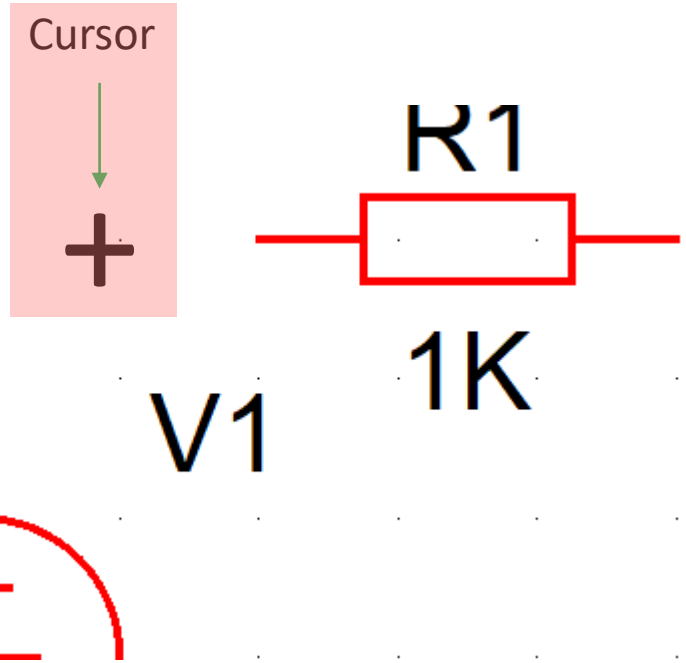




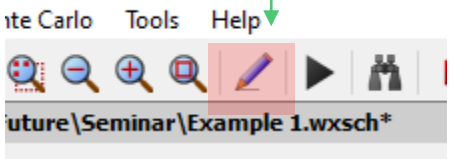


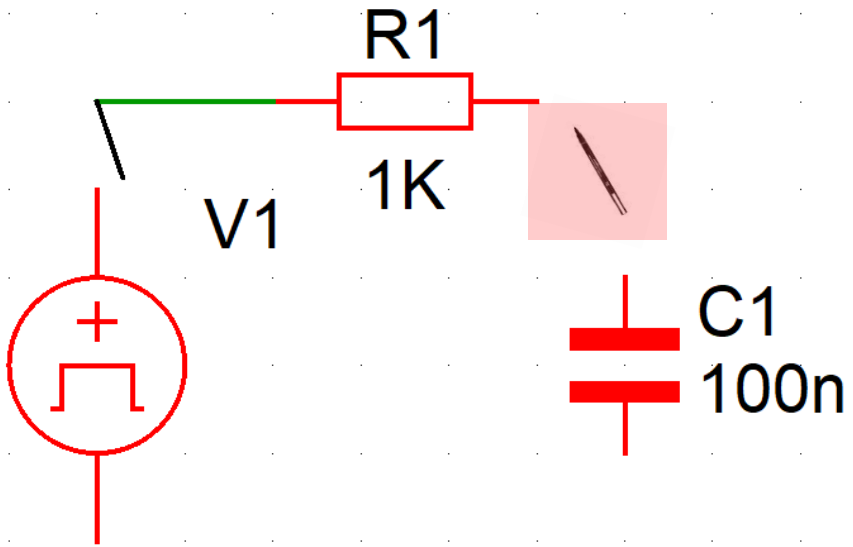


Double click

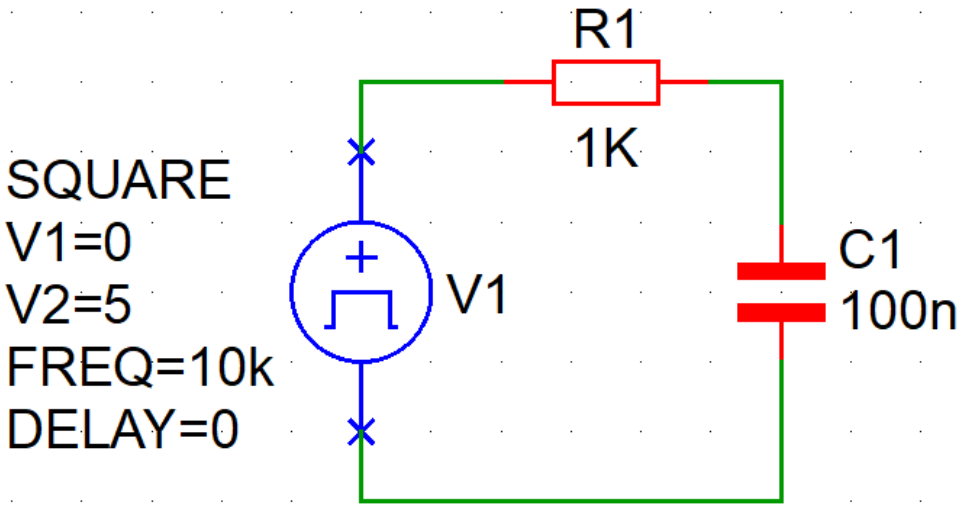


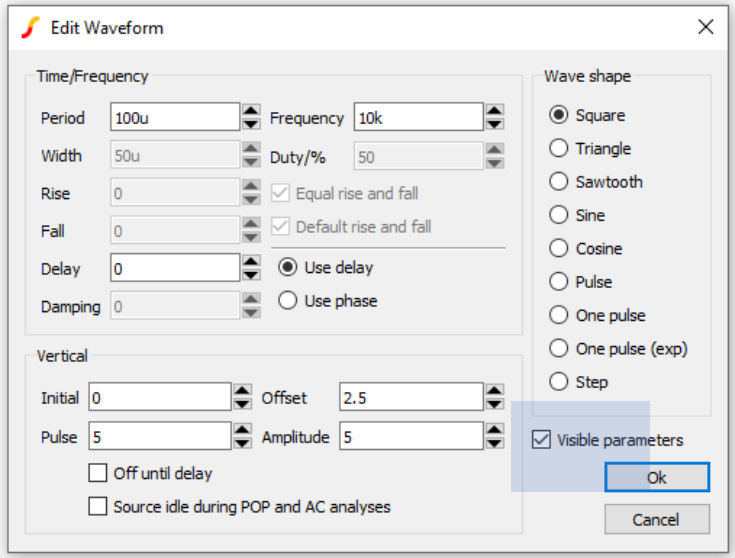
Approach a component connection or a node and the cursor turns into a pen. If it doesn't





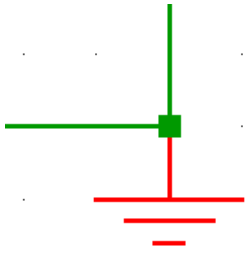
Double-click on the left mouse button also starts the pen



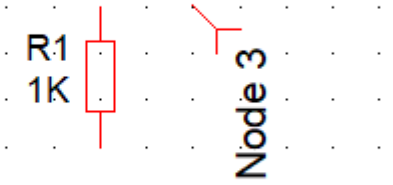


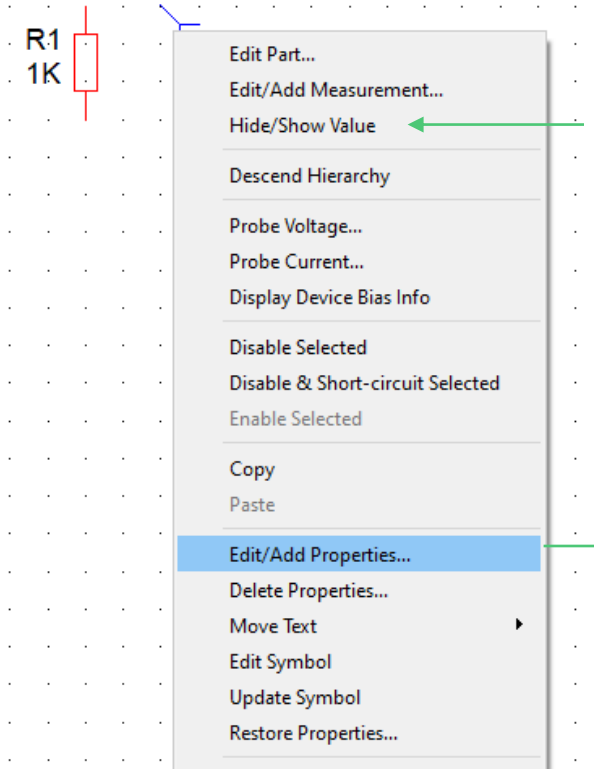
Add a ground by pressing G

Forgetting to place a ground is a classic!

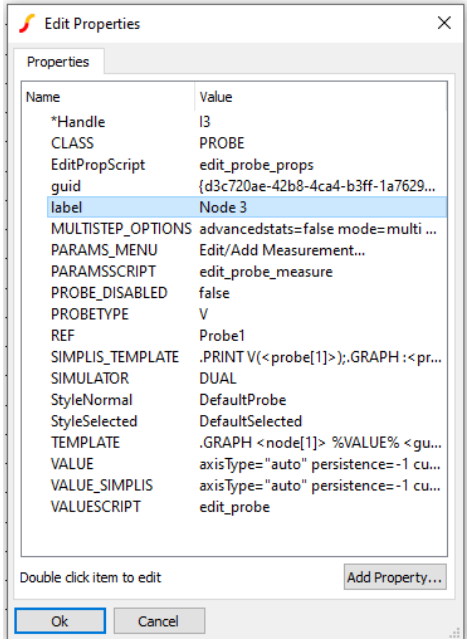


How to modify or rotate labels

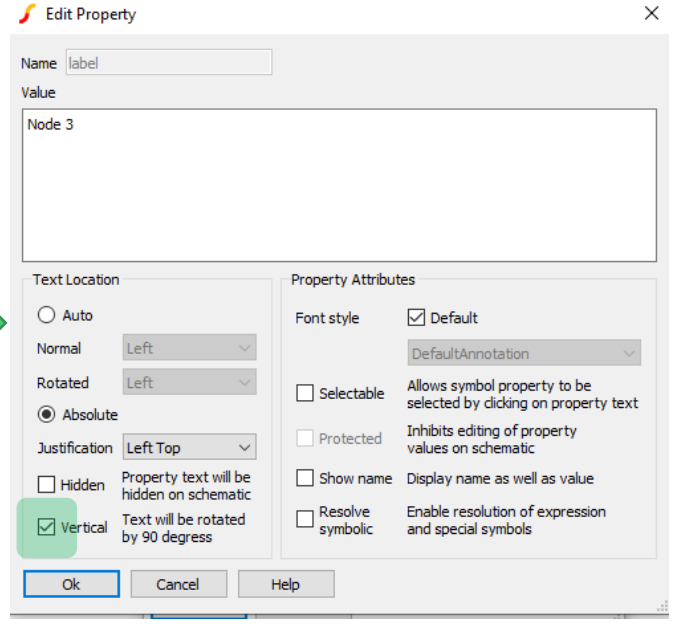


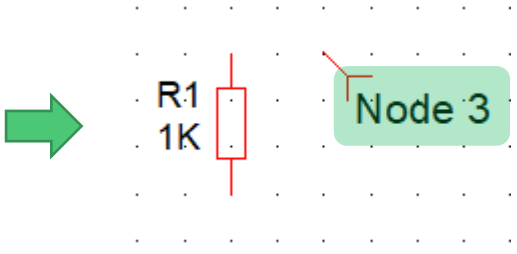


You can hide the value

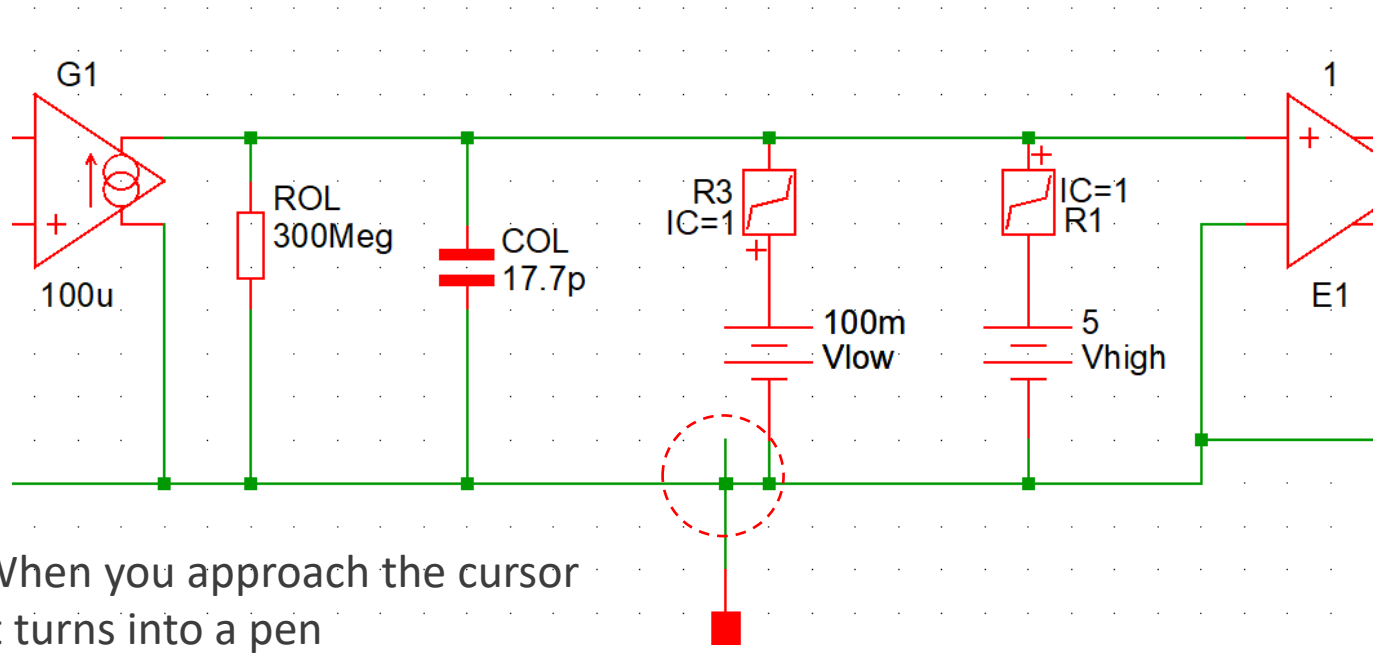


Double click

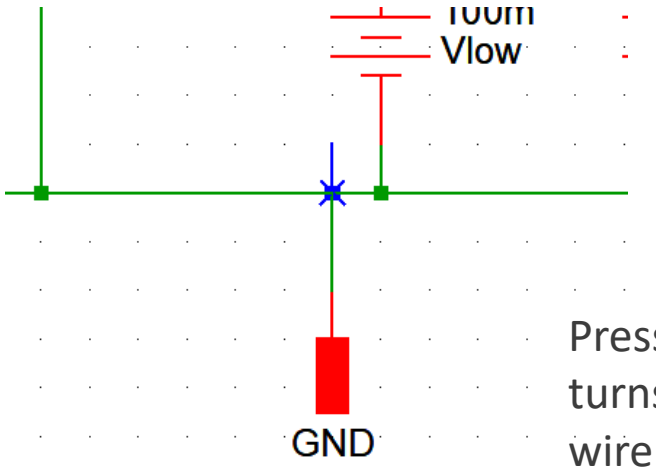




Sometimes you want to delete a tiny piece of wire
but you can't select it

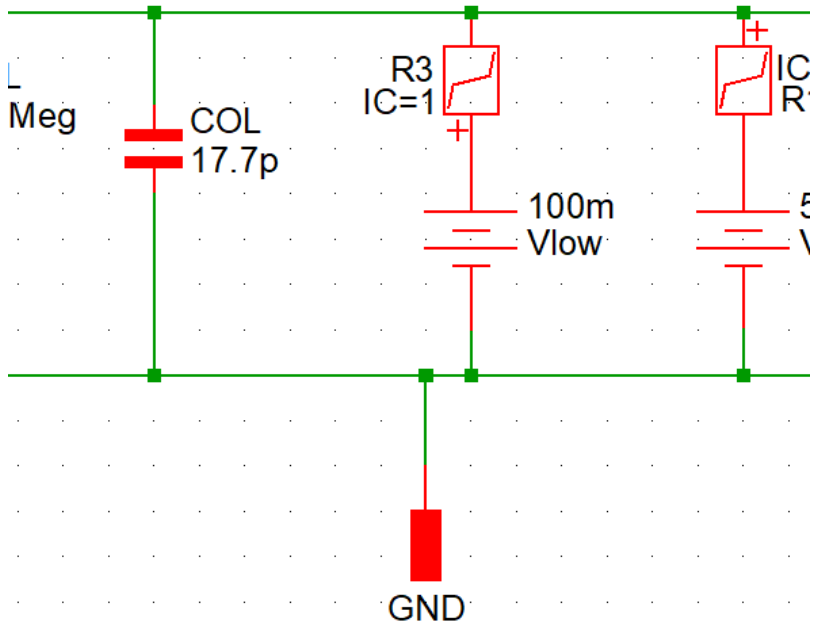


When you approach the cursor
it turns into a pen

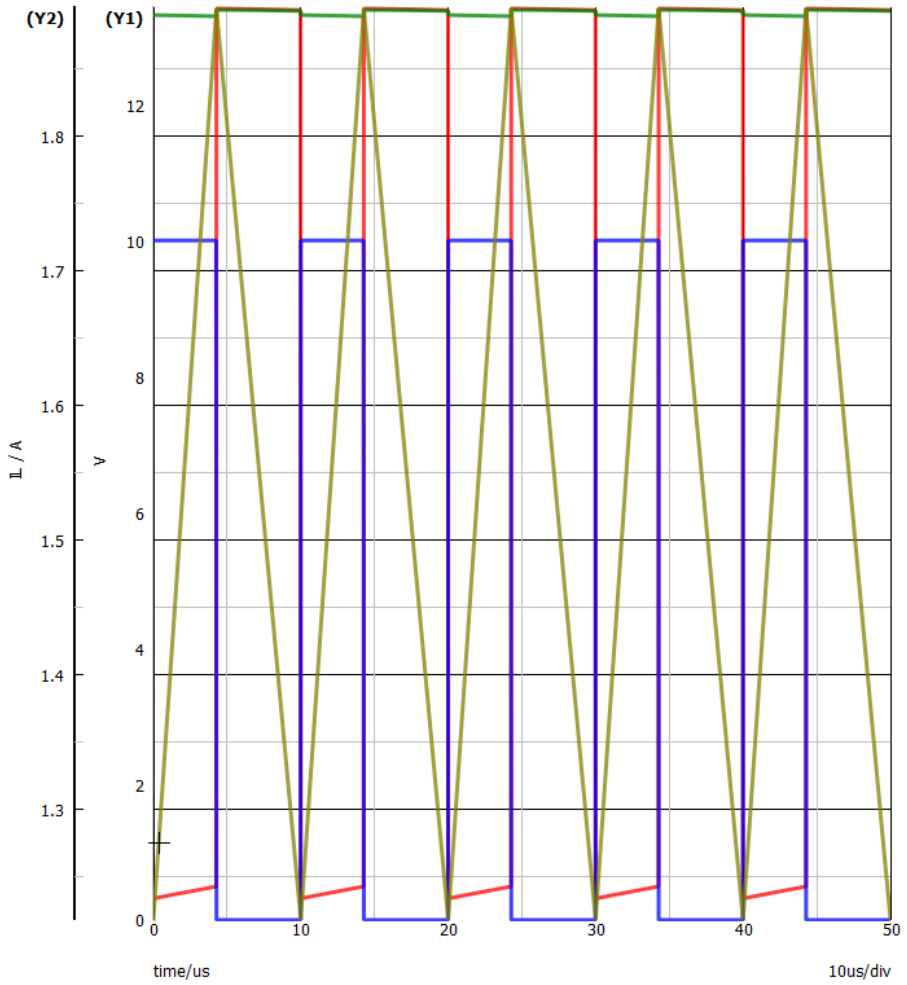


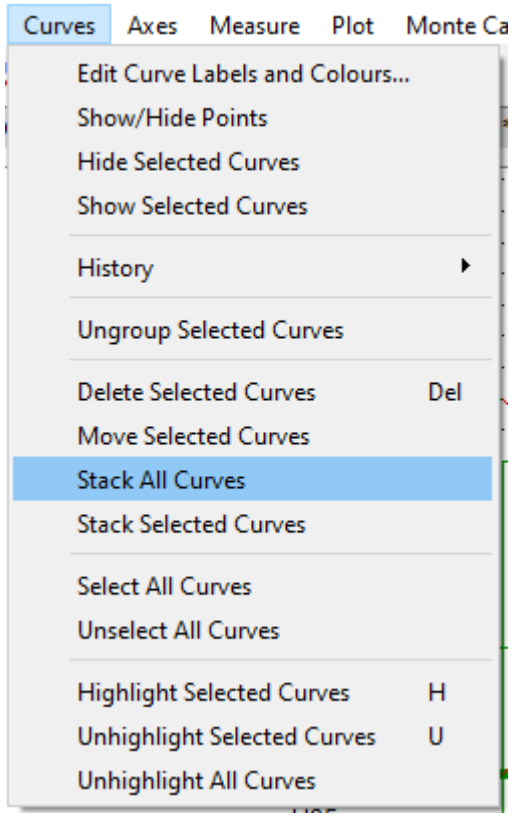
Press SHIFT and the cursor
turns into a cross: select the
wire which turns blue. Delete it.

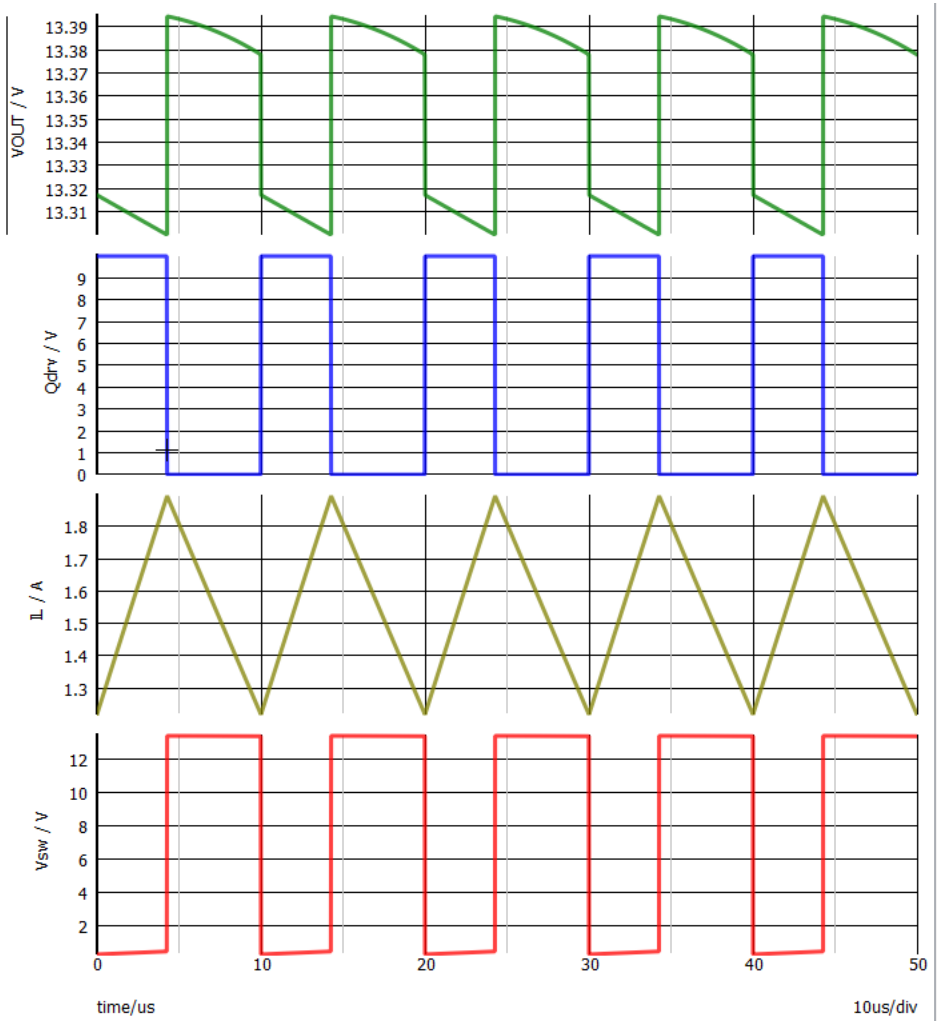
Voilà!



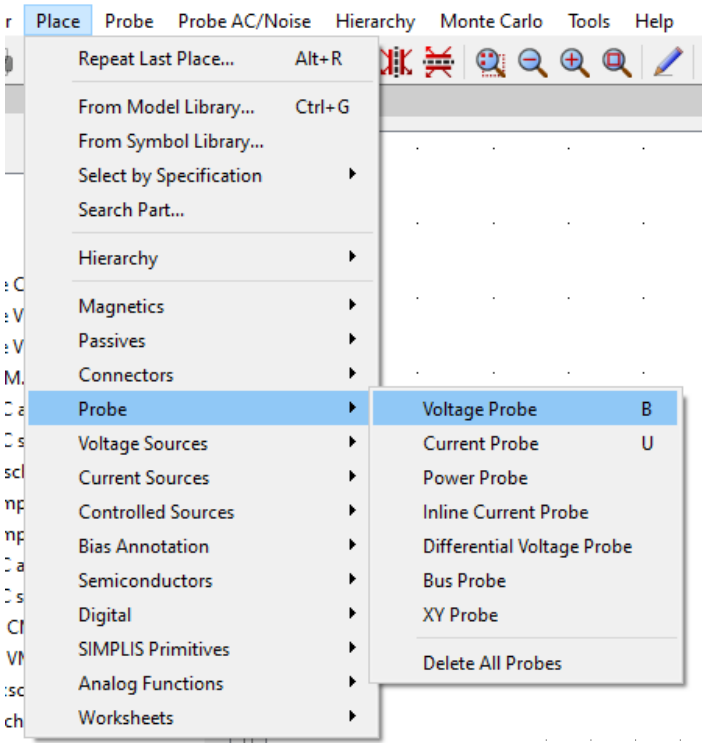
If you place probes without specifying axis information, by default, the viewer stacks up all the curves, ending in an ugly representation as below. To have them go to their own axis – what it should do by default in my opinion – open the Curves menu and choose “stack all curves”:

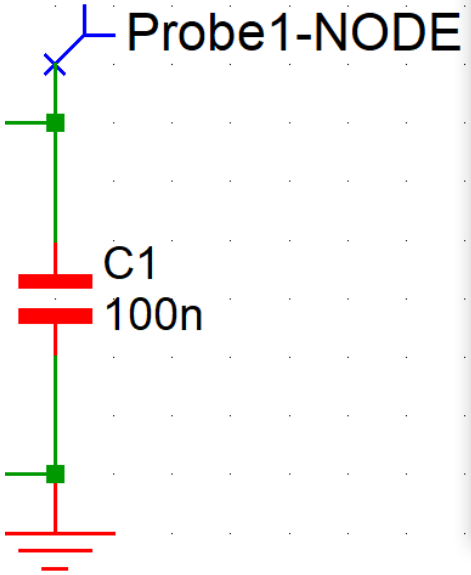


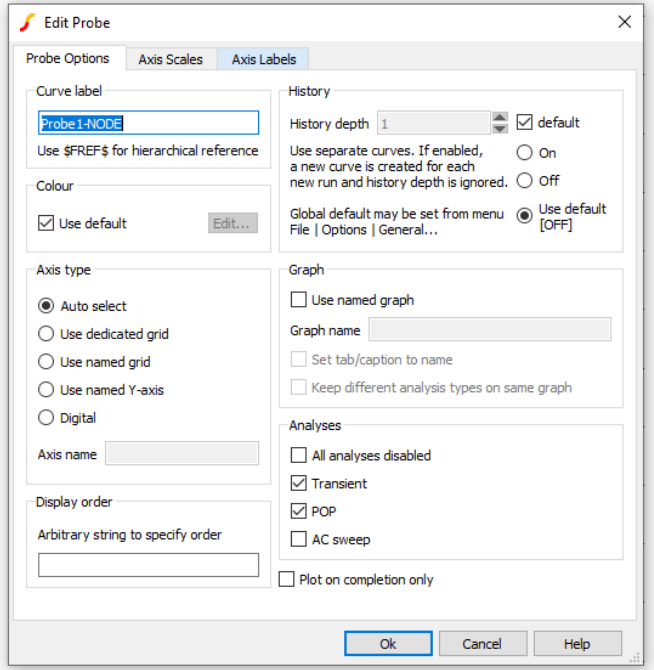


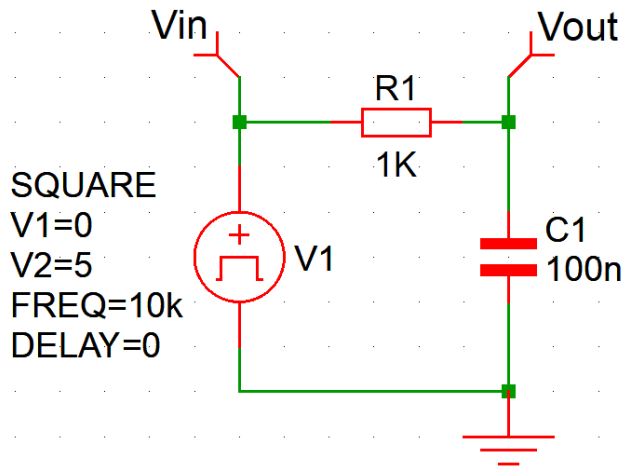


You place probes by pressing B on the keyboard:





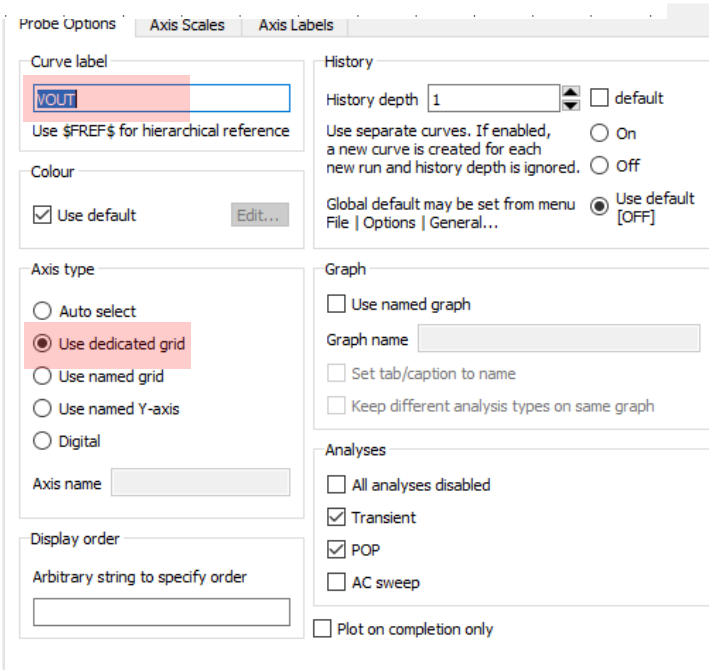




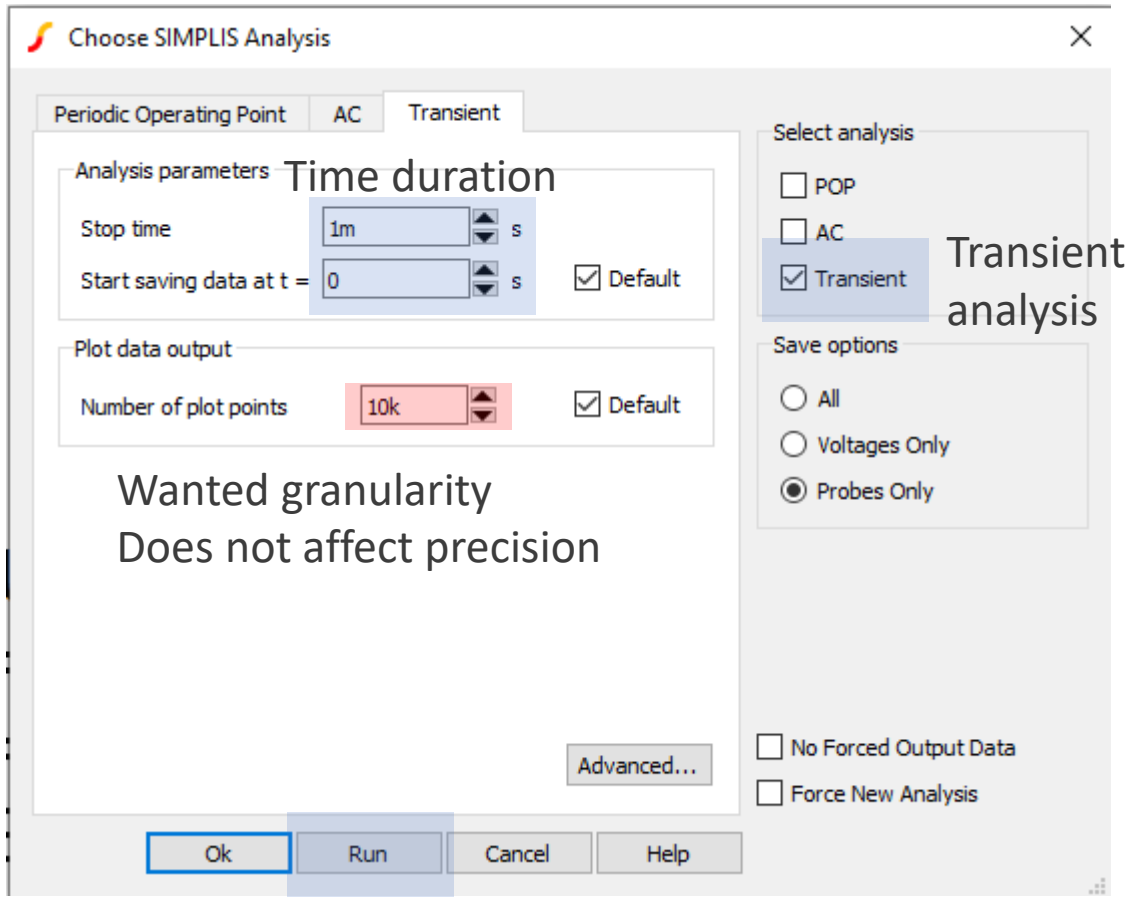
Label the probe

Tick this button

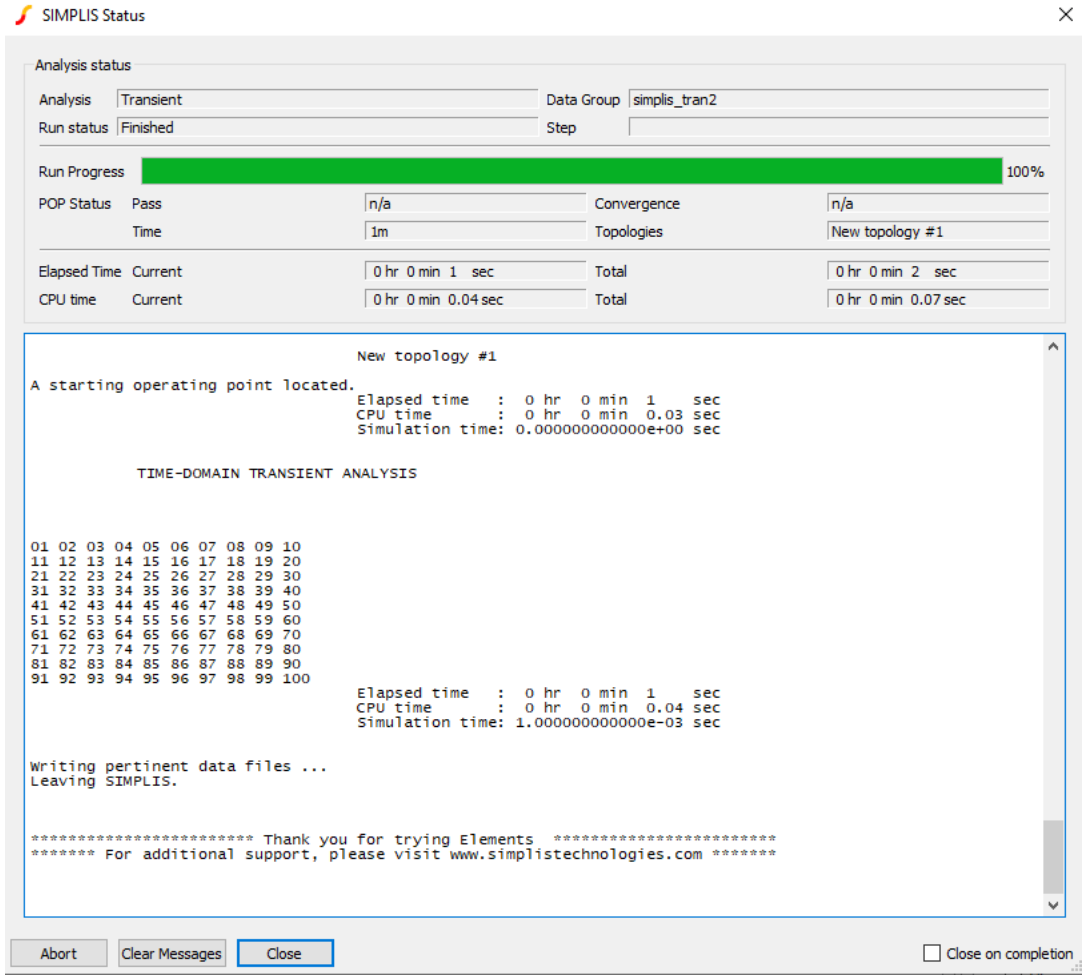
Enter the name



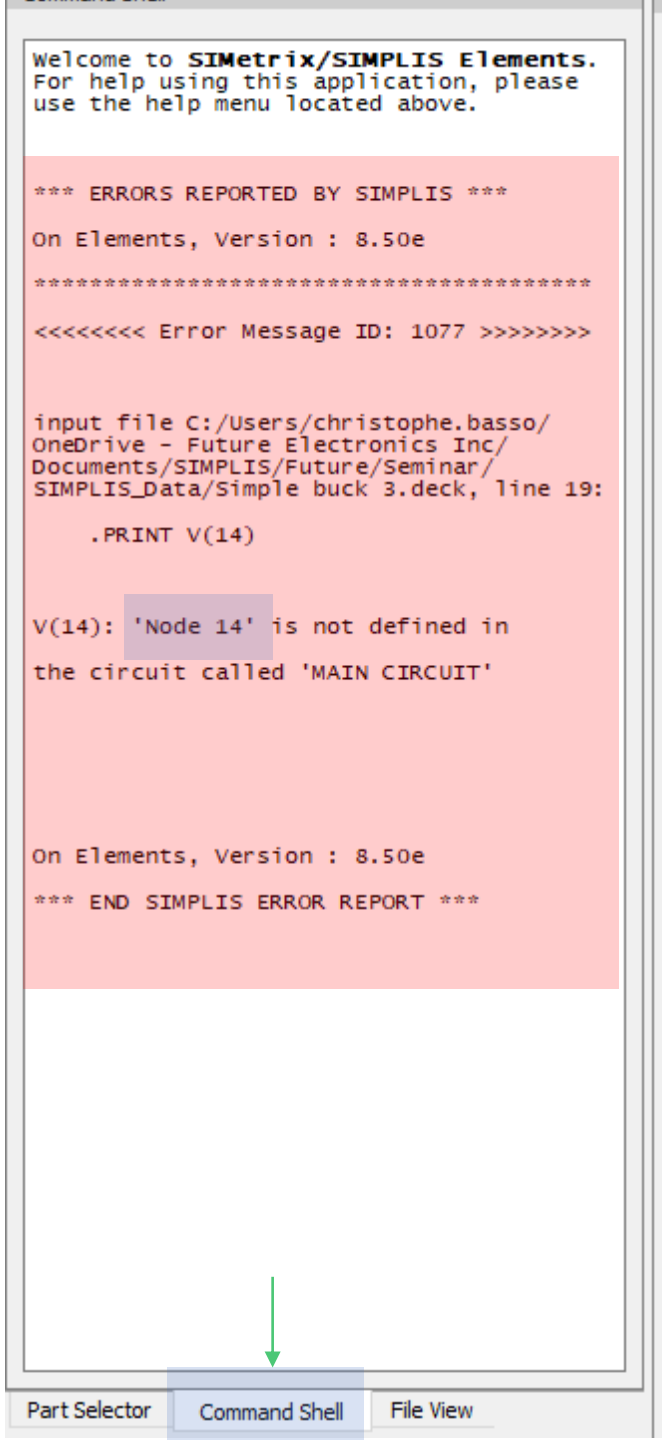
Choose the simulation parameters – press F8



Once the circuit is saved, go!



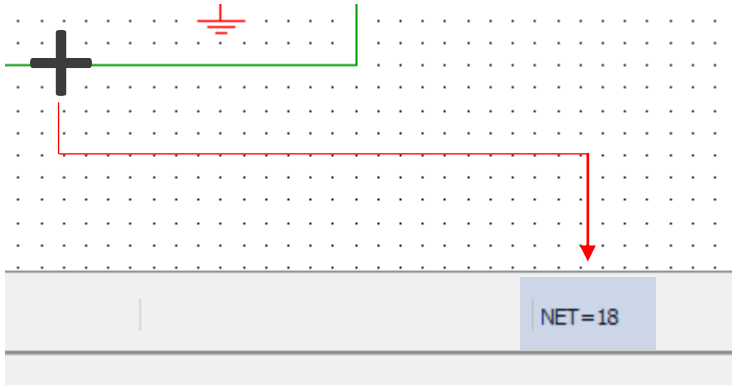
When everything is ok, the complete windows pops-up



In case simulation issues occur, look at the Command Shell window.

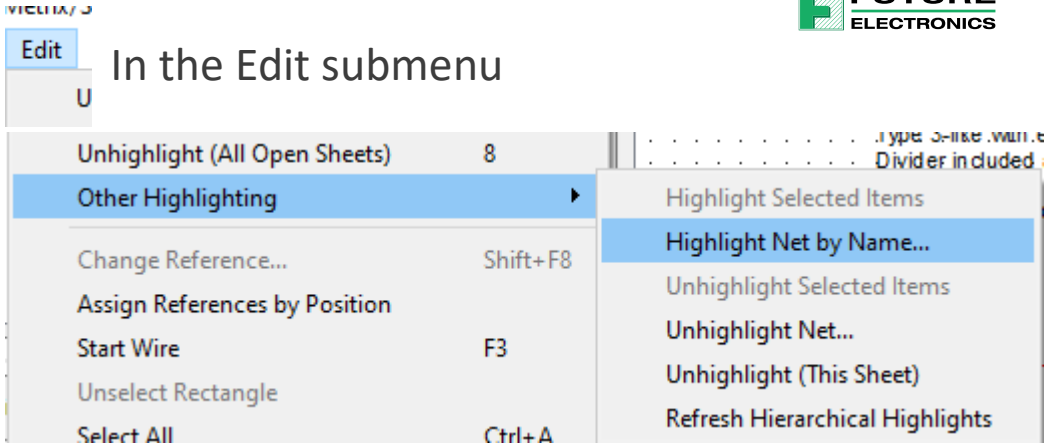
You can erase the window with <CTRL+A> and SHIFT+DEL

In case the error refers to a net or node number you have to find its position on the schematic diagram.

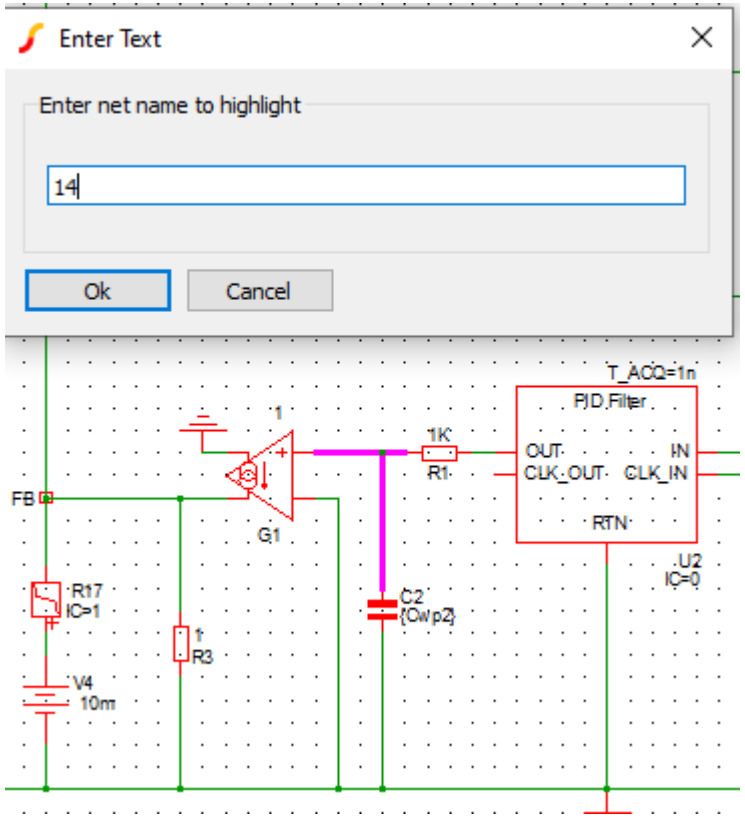


Fly over a (cuckoo's) net and see the number displayed in the low-side bar.

FUTURE ELECTRONICS

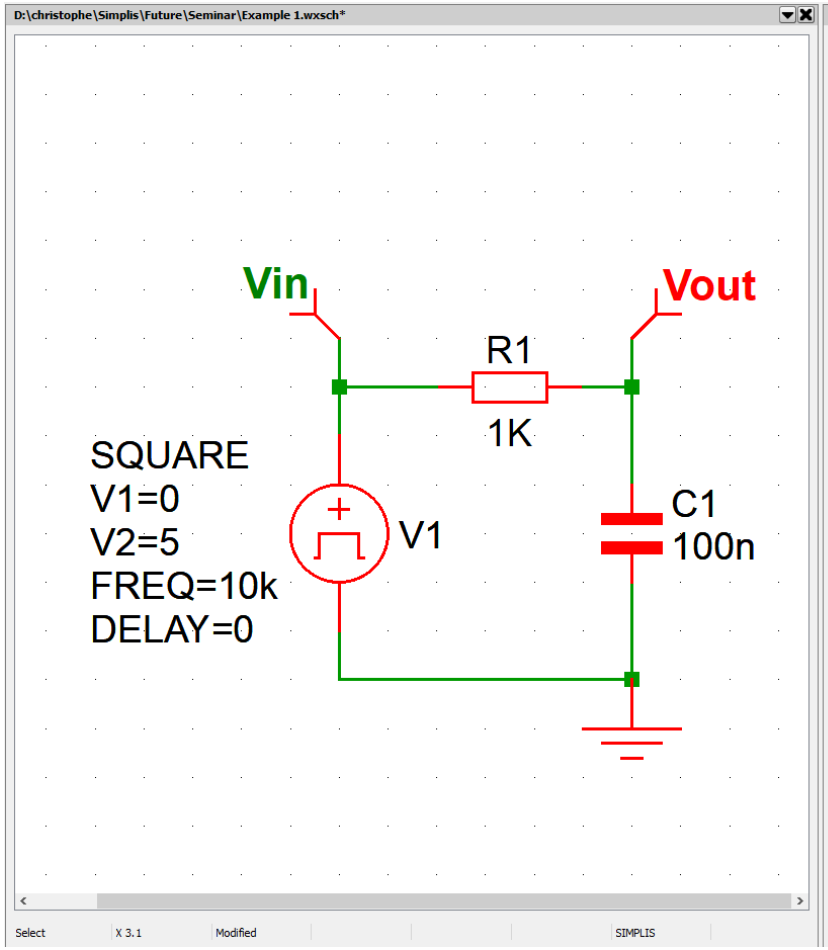


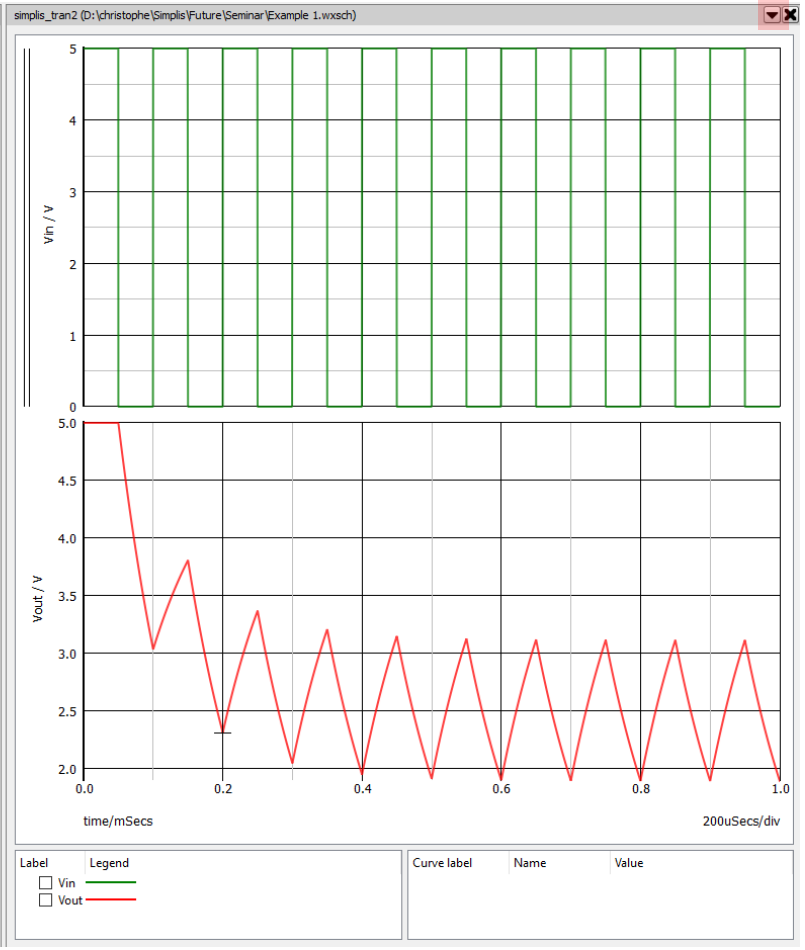
In the Edit submenu

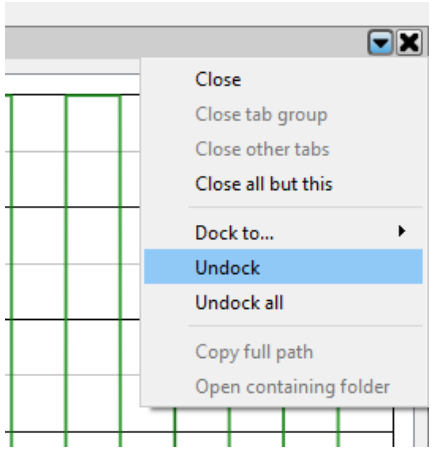


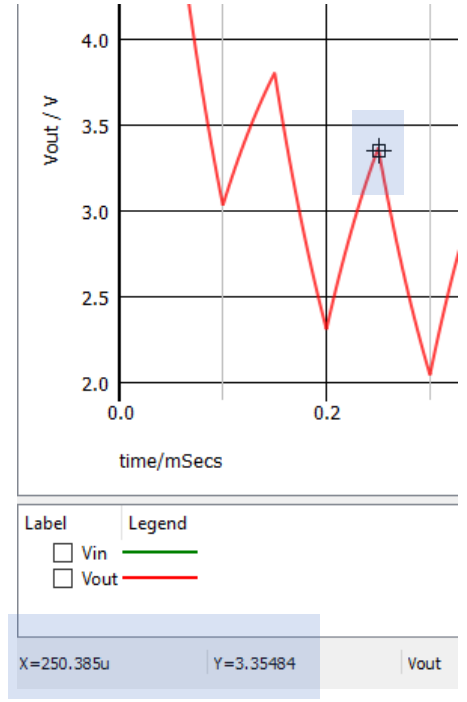
To erase the highlight, press 8 on the keypad

You can un-dock the result window





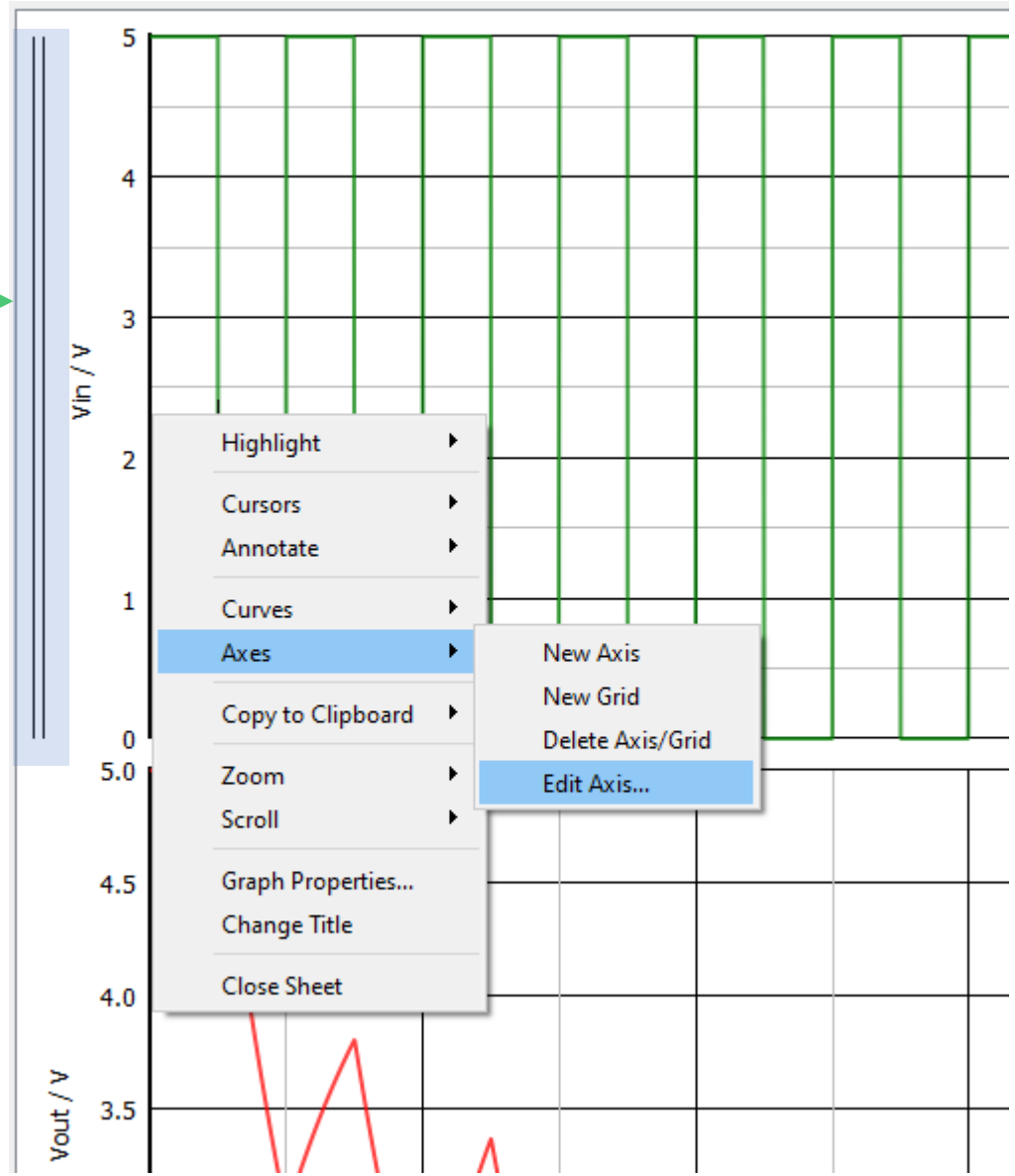


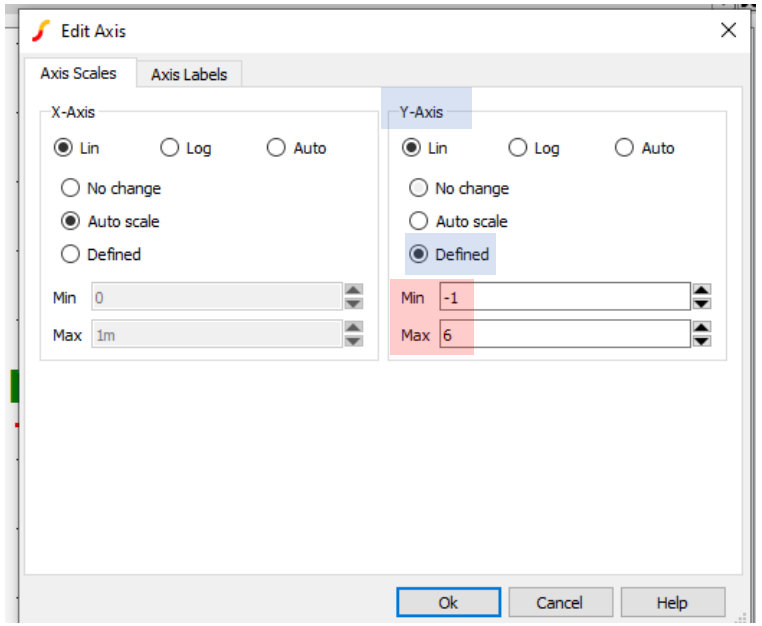


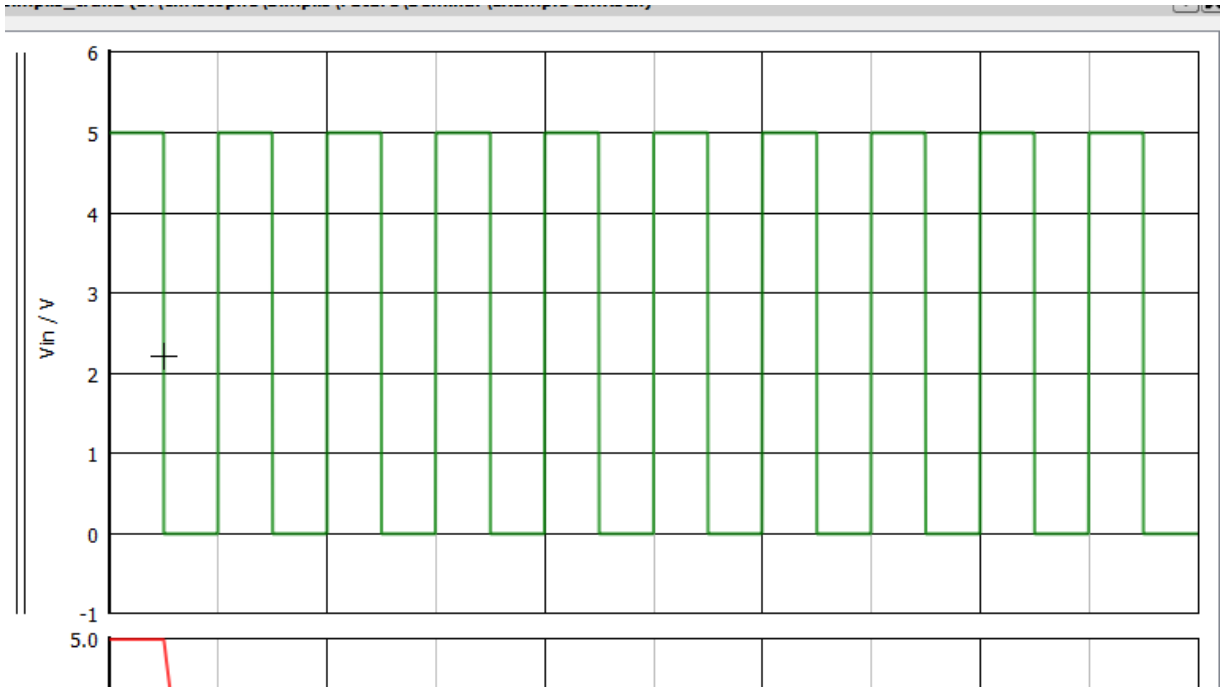
Fly the mouse over the waveform and see the XY values displayed

Selected sign

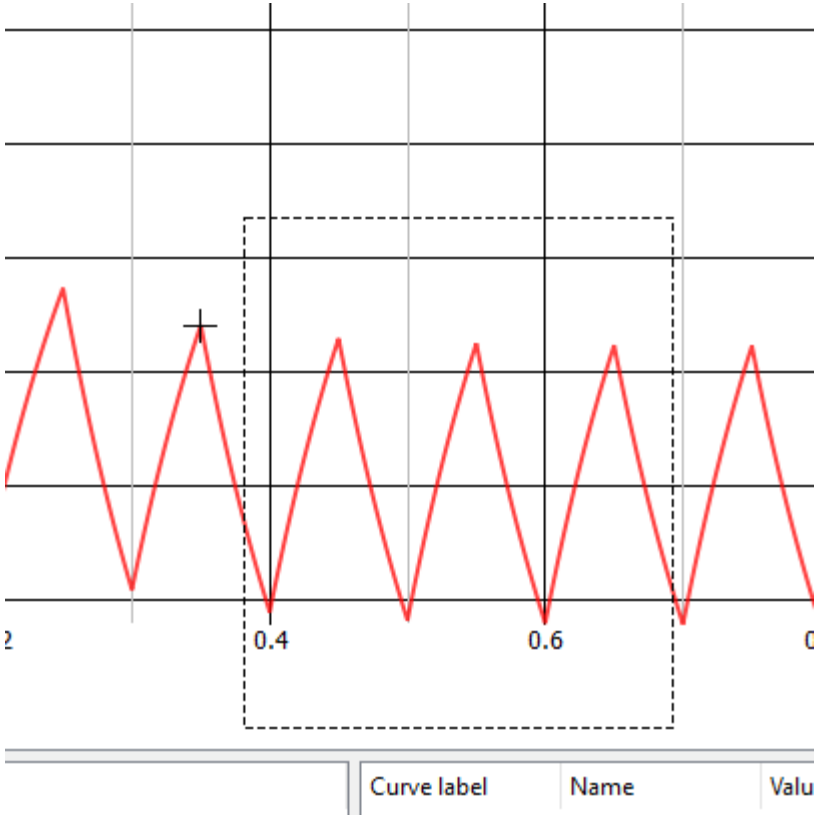
To change the axis







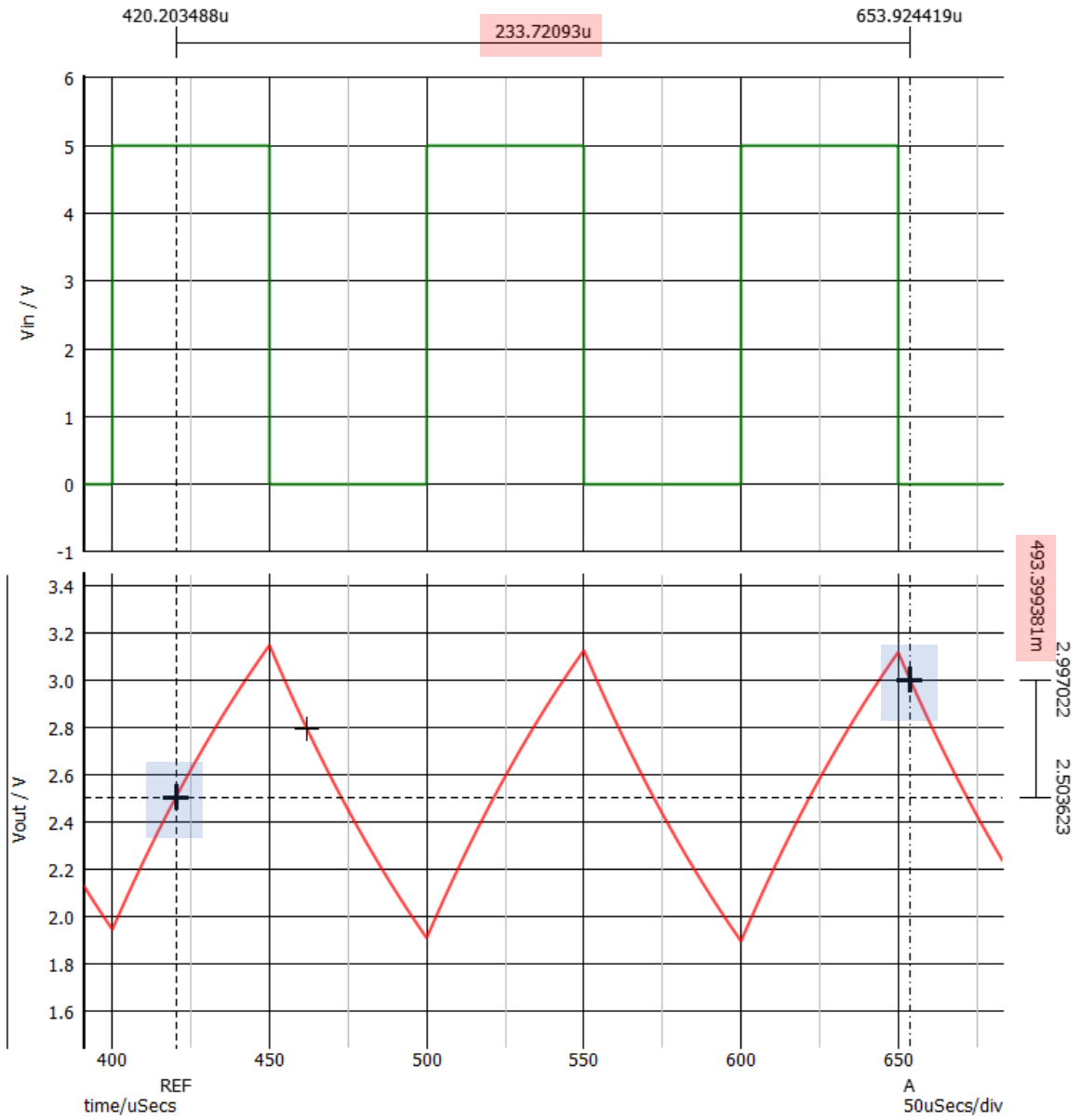
Zoom-in by dragging a window



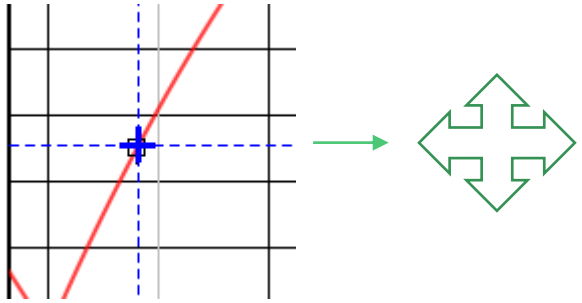
And call the cursors by pressing C

ΔX

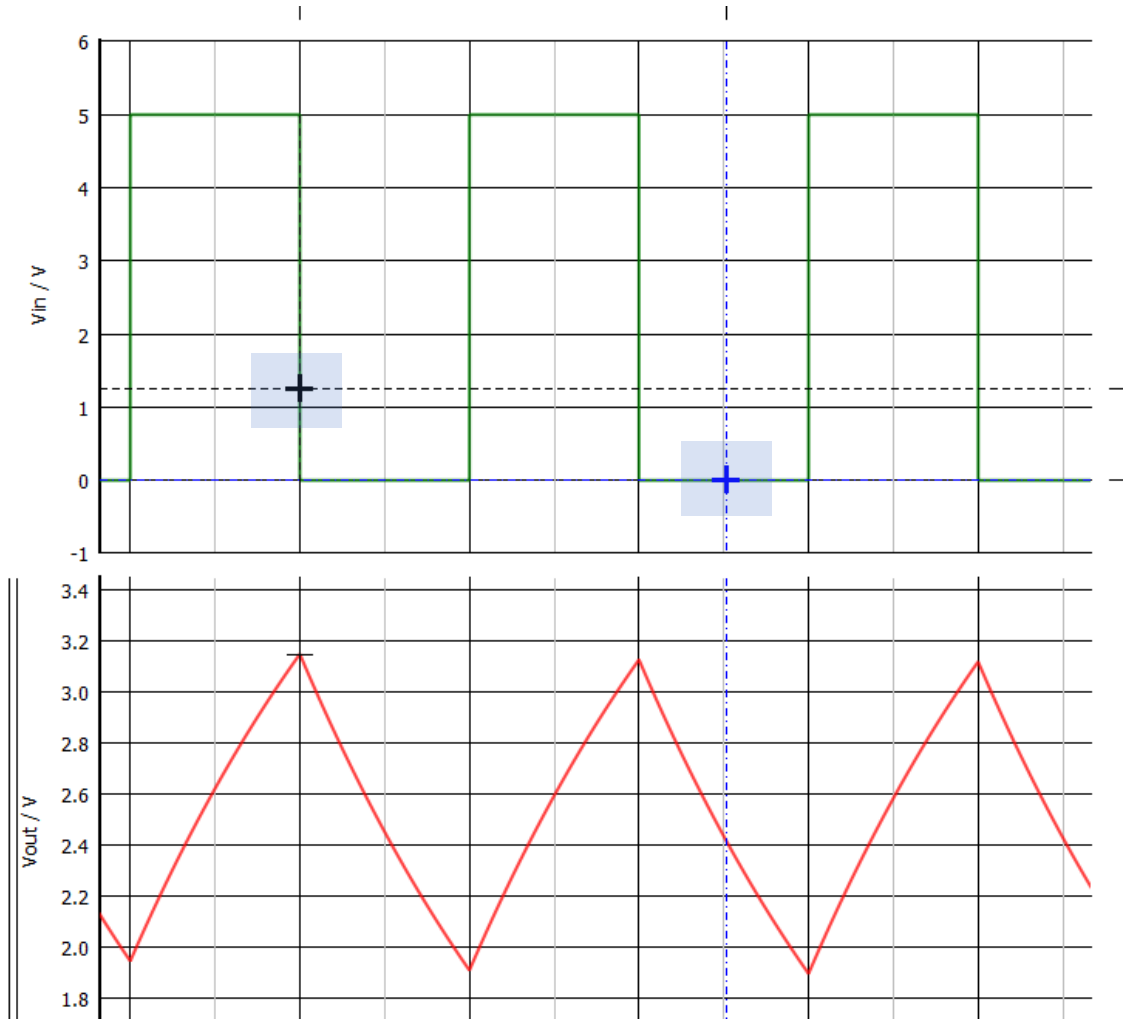
ΔY



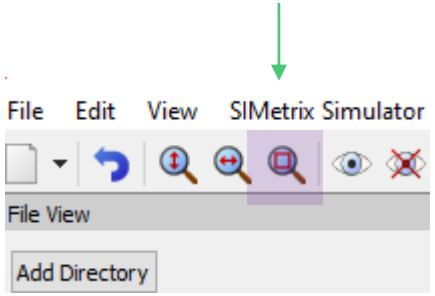
To change the cursors to another curve,
fly the mouse over the cursor cross until the symbol changes



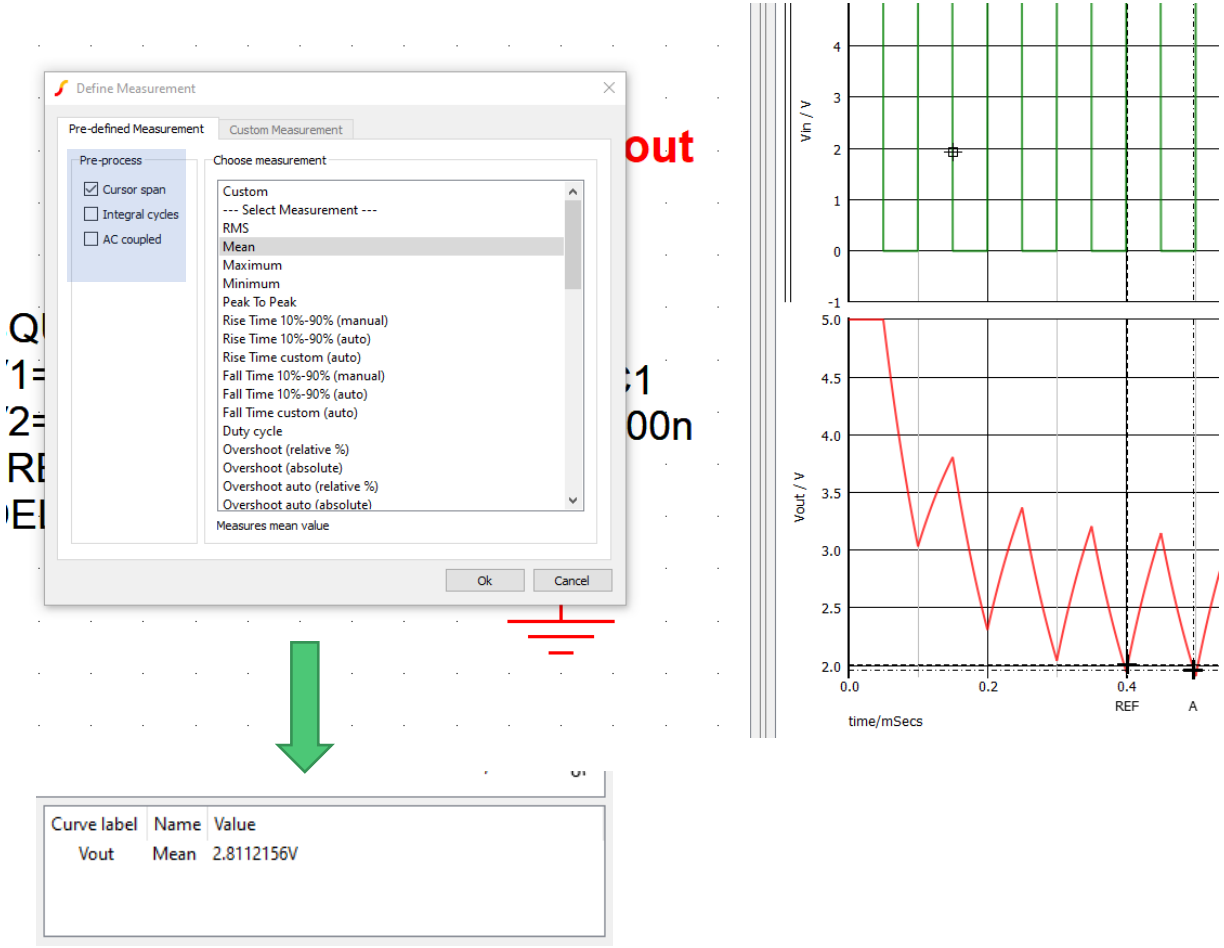
Then drop the cursor
to the other waveform
to explore it

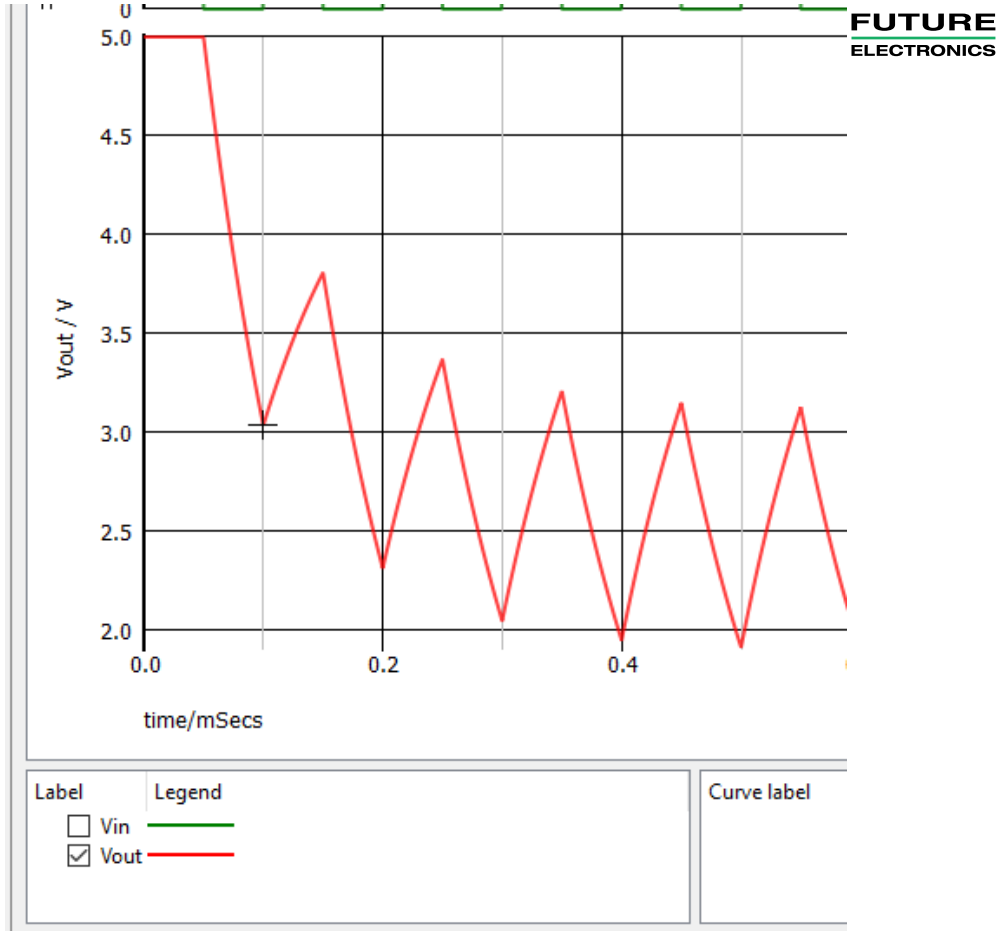


Go back to full span



To run a measurement on the waveforms, press F3

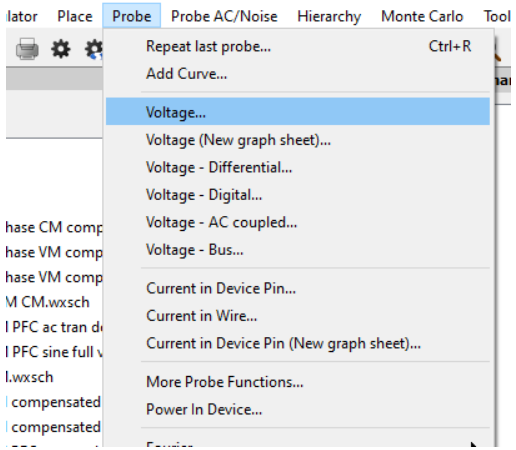


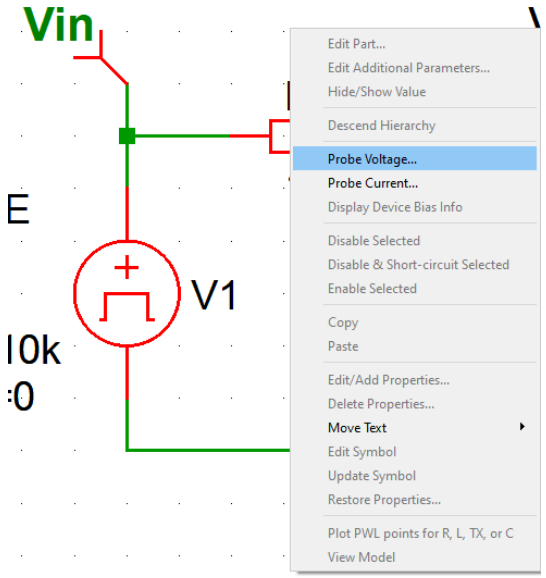


To delete a waveform, select it and press delete

Cursor span	Truncates the waveform data to the span defined by the current positions of the cursors. In other words, the measurement is performed on the range defined by the cursor positions.
Integral cycles	Truncates the waveform data to an integral number of whole cycles. This is useful for measurements such as RMS which are only meaningful if applied to a whole number of cycles.
AC coupled	Offsets the data by the mean value. This is equivalent to 'ACcoupling' the data.

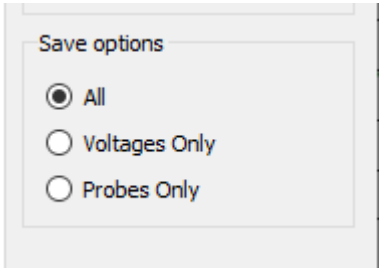
You can probe on the fly





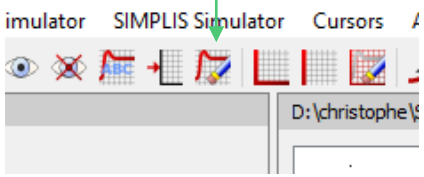
Or right click in the schematic

For on-the-fly probing don't forget to activate the All option in the sim window (F8)

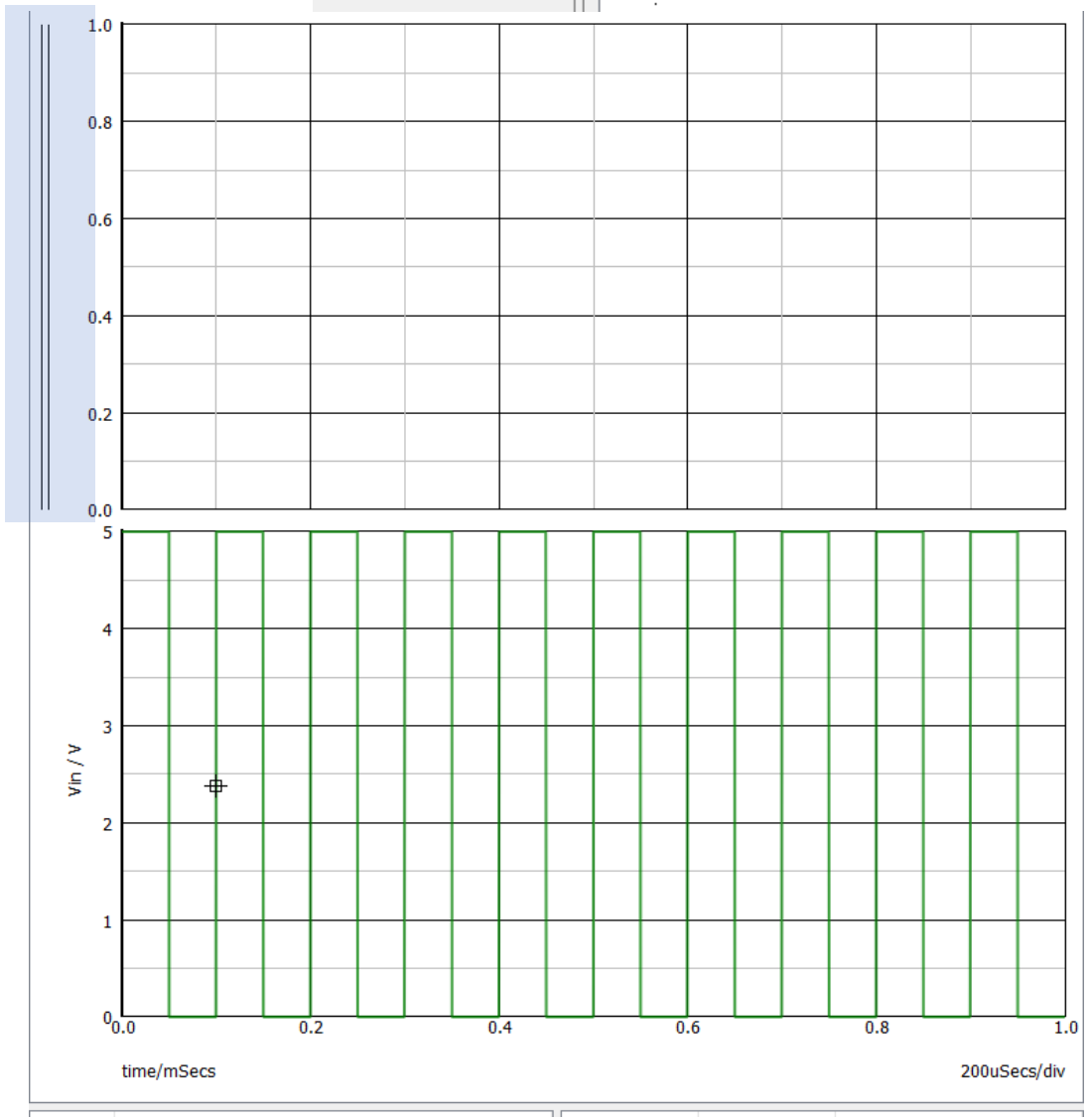


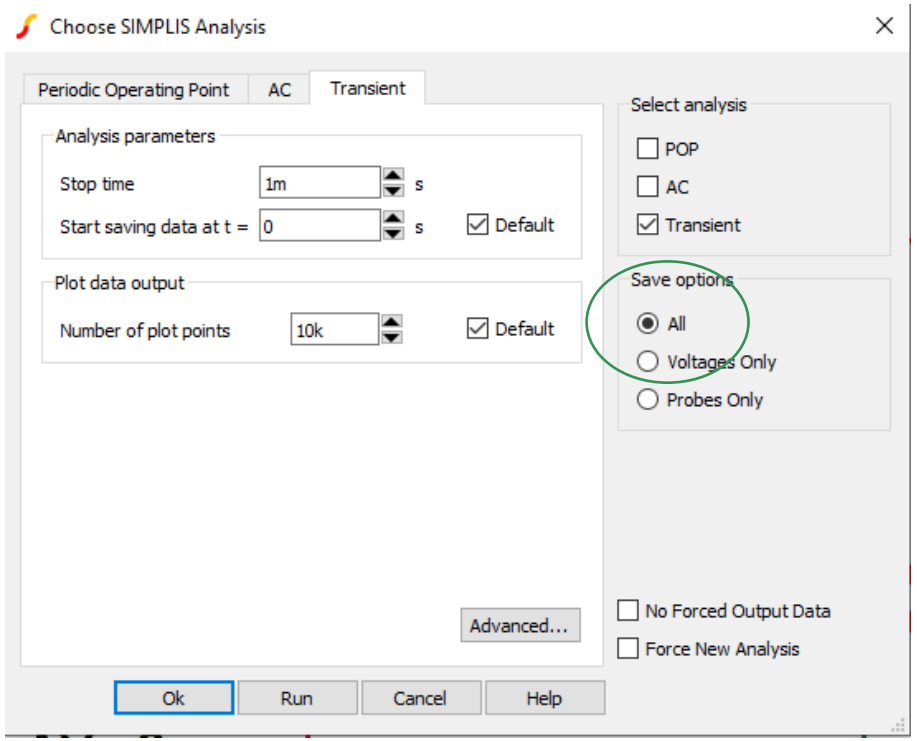
The probed signal will go to the newly-selected grid

New grid

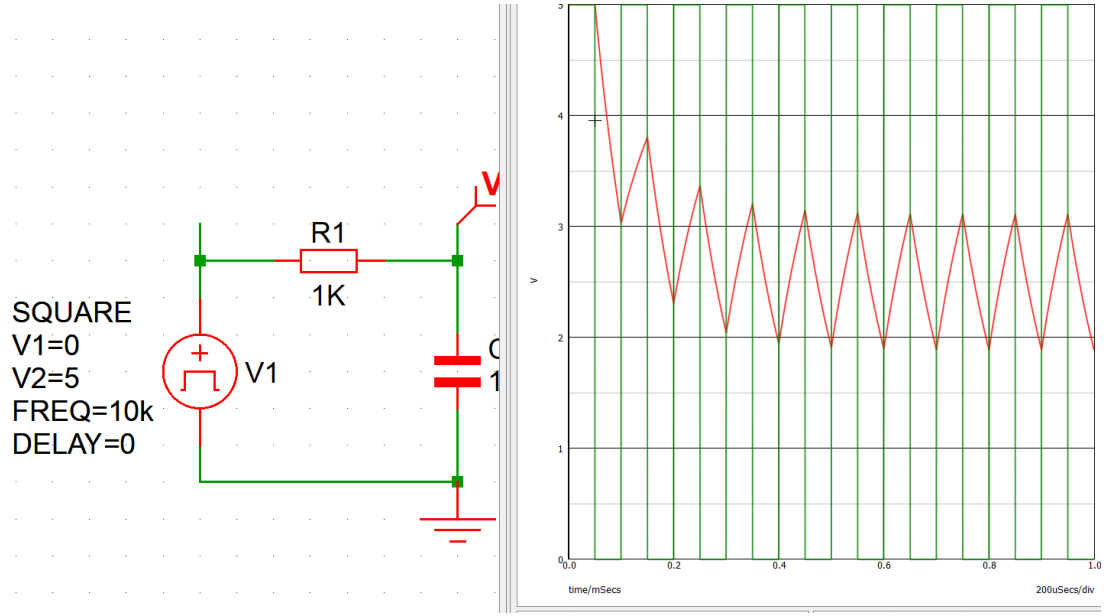


Selected

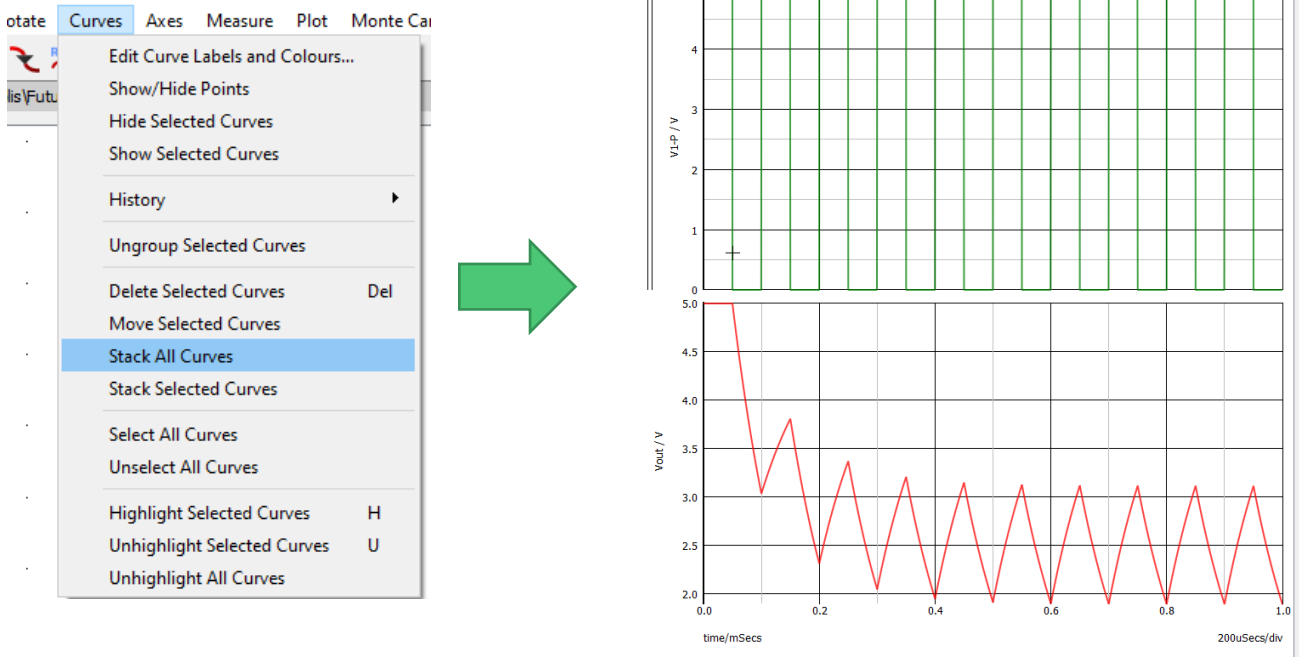




The waveforms are placed on a common grid



You can also select the All option and dynamically probe all main schematics nodes without placing probes first. Ok for debugging but I preferred well-labeled probes. Ticking All does not give access to subcircuits nodes, another statement is necessary.



In case you need to cross-probe voltages and currents in a subcircuit, you need to insert one of the below .KEEP combination in the main simulation window. Check the documentation here and watch out when using this command as the data file can quickly be gigantic!

The Concise Guide to .KEEP Statements

The following guidelines will be helpful to select .KEEP statements

Command	Action
.KEEP *V	Keeps all voltages at this hierarchical level.
.KEEP **V	Keeps all voltages at this hierarchical level and all lower levels.
.KEEP *I	Keeps all currents at this hierarchical level.
.KEEP **I	Keeps all currents at this hierarchical level and all lower levels.
.KEEP *I **V	Keeps all currents at this hierarchical level and all voltages at this hierarchical level and all lower levels.
Other combinations are possible using a space separated list of voltage and current declarations.	

CAUTION:

For large circuits, placing a .KEEP **V at the top level will save an enormous amount of data. Using .KEEP **V **I will save even more data, as all currents will be added to the data group.

CAUTION:

When you package your circuit for encryption and distribution, make certain you do not have .KEEP statements in the hierarchy. Any .KEEP statements will output simulation vectors to the data group, possibly exposing some internal intellectual property. This will be discussed in more detail in the 4.4 Protecting Your Intellectual Property - Model Encryption topic.

Press F11 while on the main schematic:

```
.param Rlower={2.5/lb}
.param fc=40
.simulator SIMETRIX
*.tran 1m
.ac dec 100 10m 1k
.simulator DEFAULT
```

Insert as shown

SIMetrix

```
.simulator SIMETRIX
*.tran 1m
.ac dec 100 10m 1k
.KEEP **I **V
.simulator DEFAULT
```

SIMPLIS

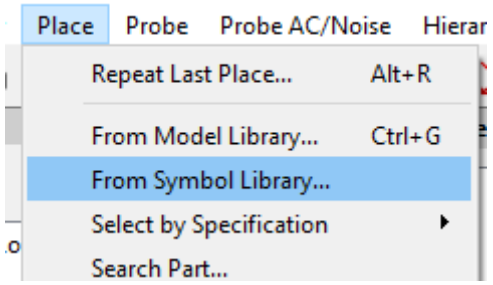
```
.simulator SIMPLIS
.ac DEC 200 10 100k
.KEEP **I **V
.print
+ ALL
.options
```

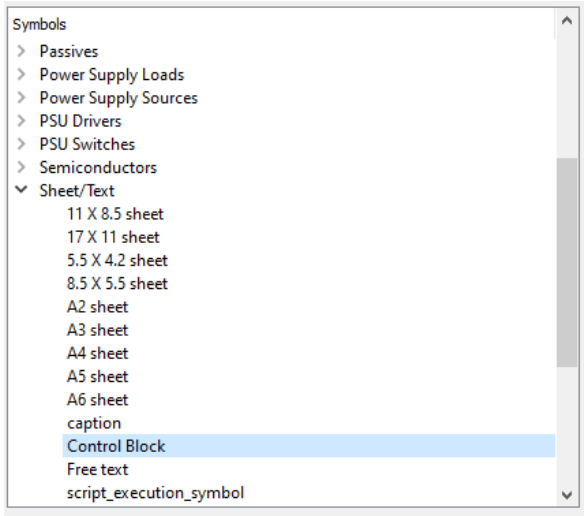
Saves all currents and all voltages at all levels

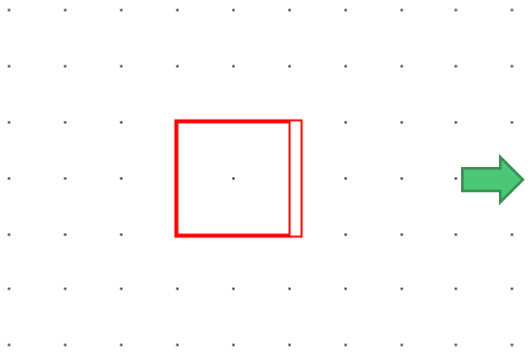
Agenda

- What is Elements?
- Running a Basic Simulation
- A Buck Converter
- Ac Linear Analysis with SIMPLIS
- Importing a SPICE Model
- The Ready-to-Use Template

Passing parameters from the schematic with a control block

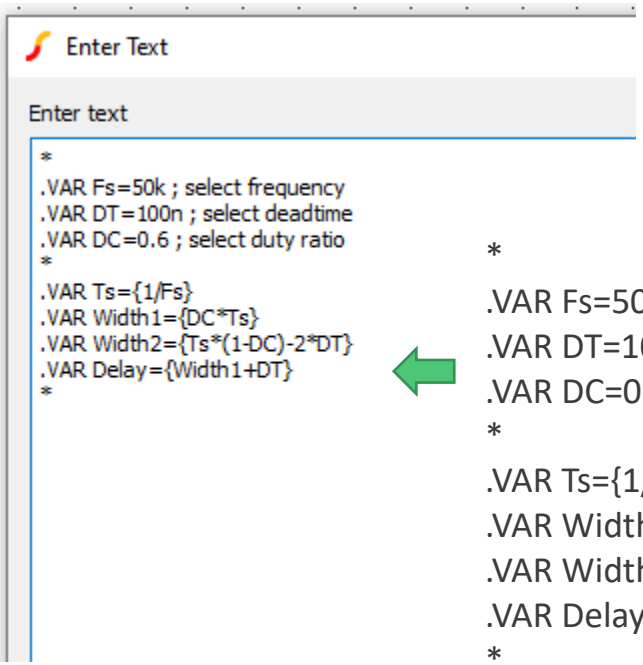






Double click
in the zone

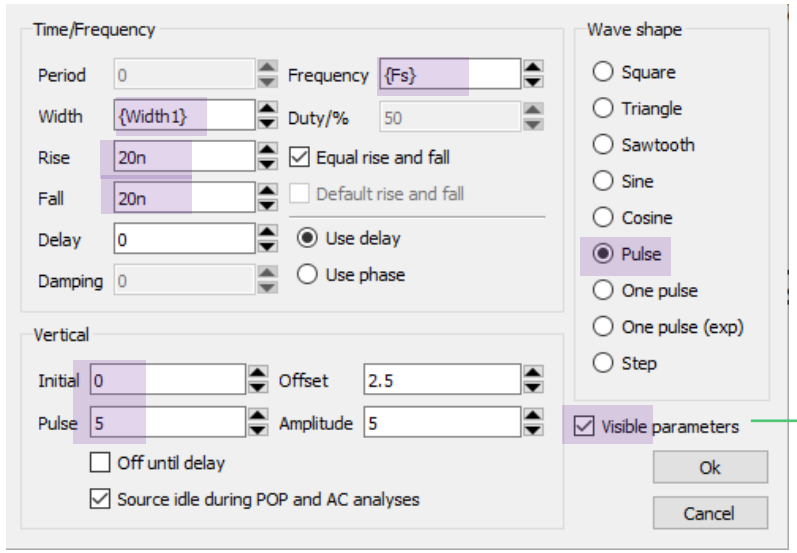
Text

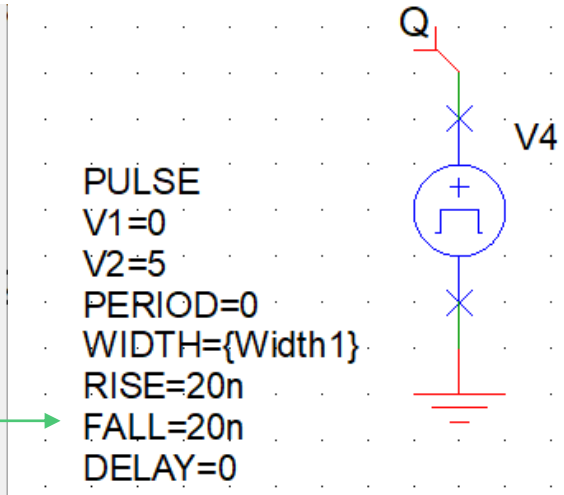


Copy/paste

```
*
.VAR Fs=50k ; select frequency
.VAR DT=100n ; select deadtime
.VAR DC=0.6 ; select duty ratio
*
.VAR Ts={1/Fs}
.VAR Width1={DC*Ts}
.VAR Width2={Ts*(1-DC)-2*DT}
.VAR Delay={Width1+DT}
*
```

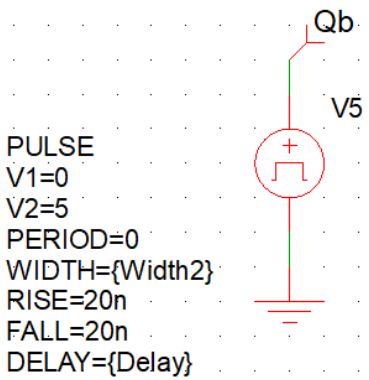
Now bring a generator by pressing W:

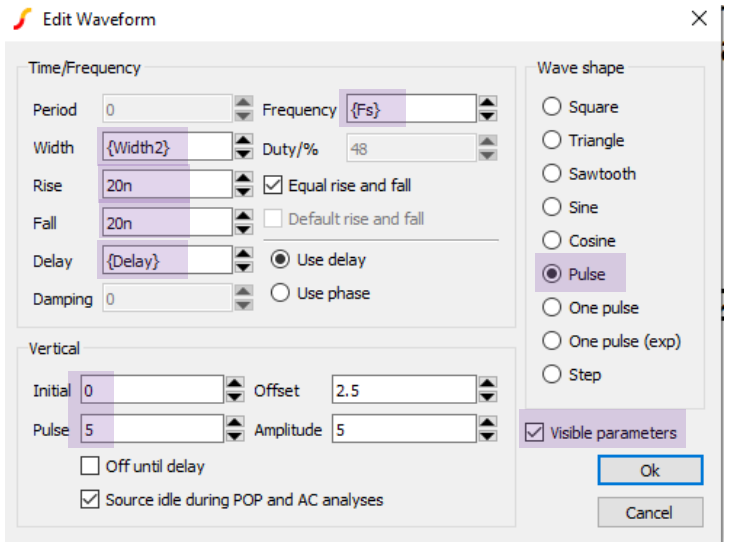




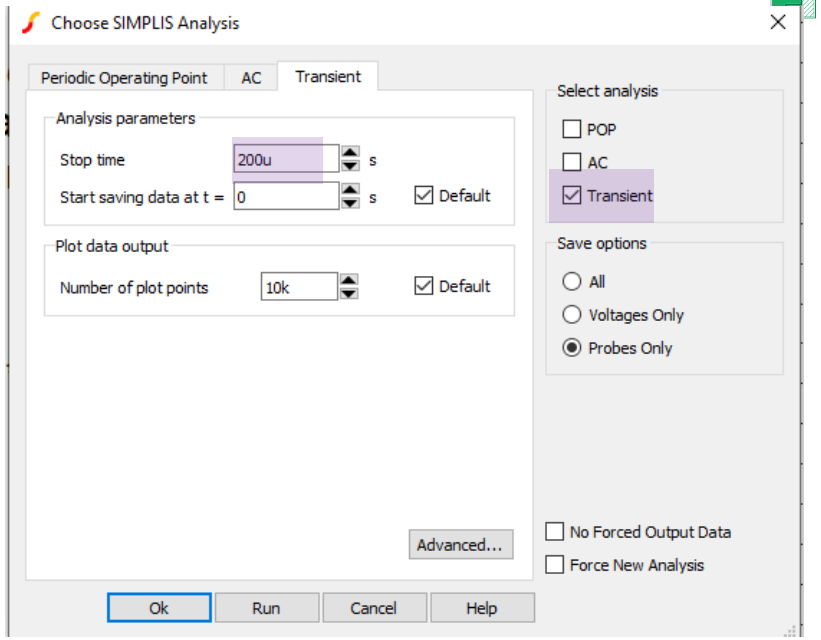
.VAR means local parameter, .GLOBALVAR means global parameters to reach subcircuits

Do the same for a second source

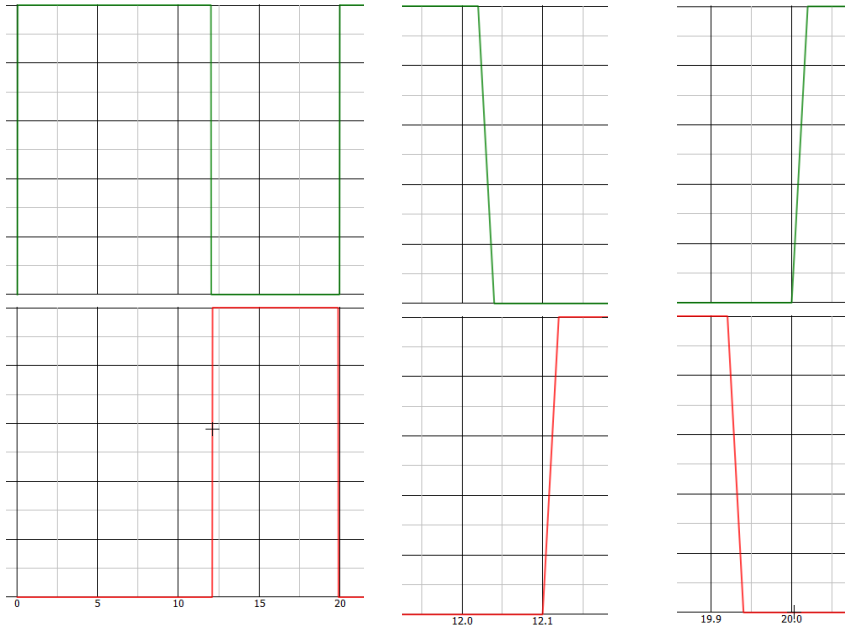




Press F8



Then press F9 to run:



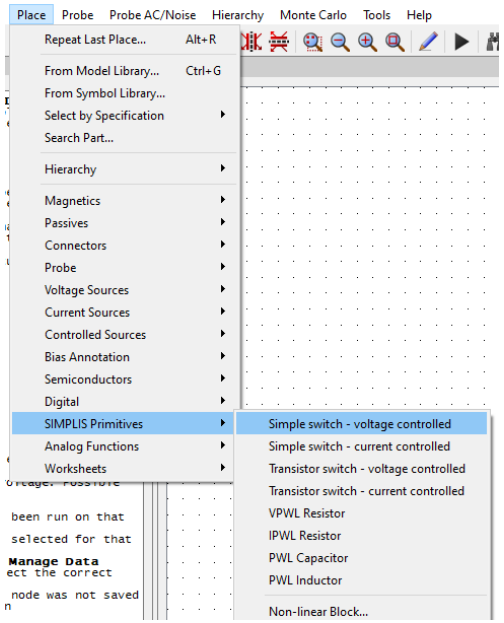
You could also pass parameters directly in the command window: press F11 to open the window, press F11 again to hide it

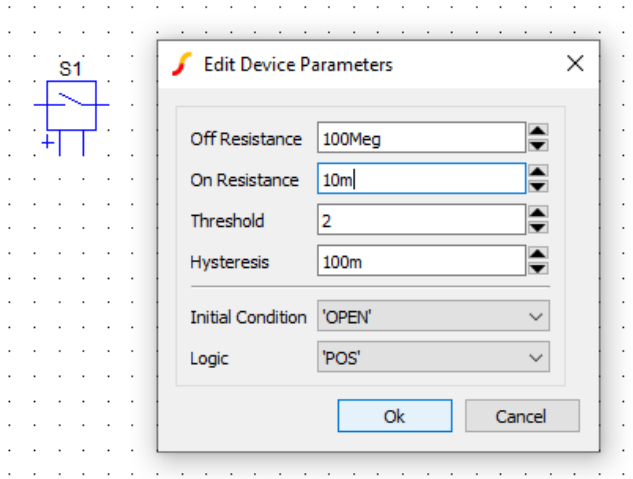
```
.simulator SIMPLIS
.options
+ PSP_NPT=10001
+ POP_ITRMAX=20
+ POP_OUTPUT_CYCLES=5
+ SNAPSHOT_INTVL=0
+ SNAPSHOT_NPT=11
+ MIN_AVG_TOPOLOGY_DUR=1a
+ AVG_TOPOLOGY_DUR_MEASUREMENT_WINDOW=128
.tran 200u 0

*
.VAR Fs=50k ; select frequency
.VAR DT=100n ; select deadtime
.VAR DC=0.6 ; select duty ratio
*
.VAR Ts={1/Fs}
.VAR Width1={DC*Ts}
.VAR Width2={Ts*(1-DC)-2*DT}
.VAR Delay={Width1+DT}
*

.simulator DEFAULT

```

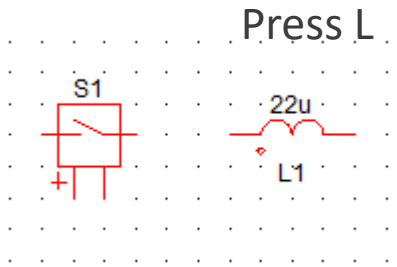
But passing controls from from the schematic is clearer and faster in my opinion.

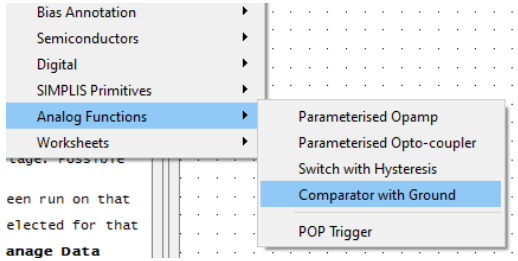
You've created two complementary waveforms with deadtime, well suited for sync buck of ½-bridge control.

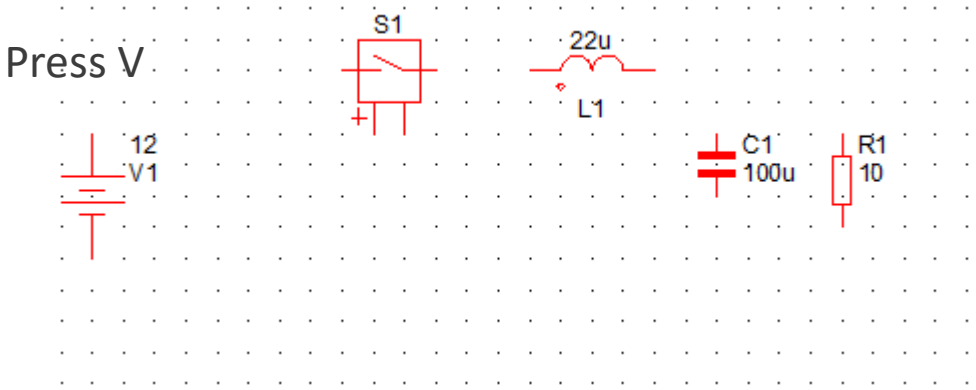
Let’ simulate a simple buck converter

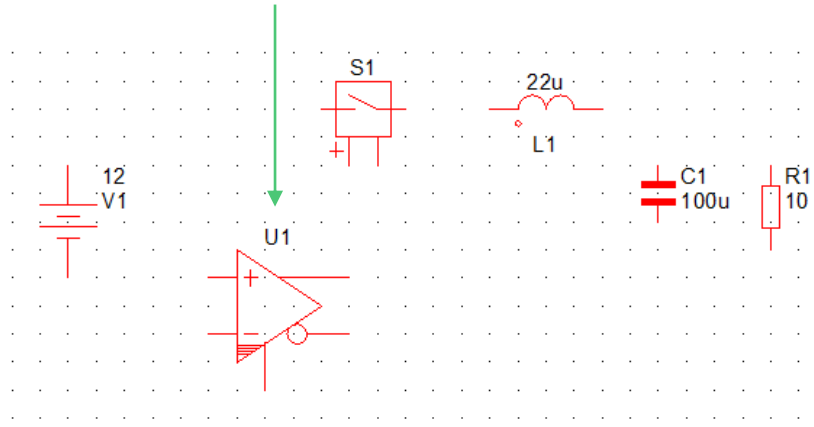




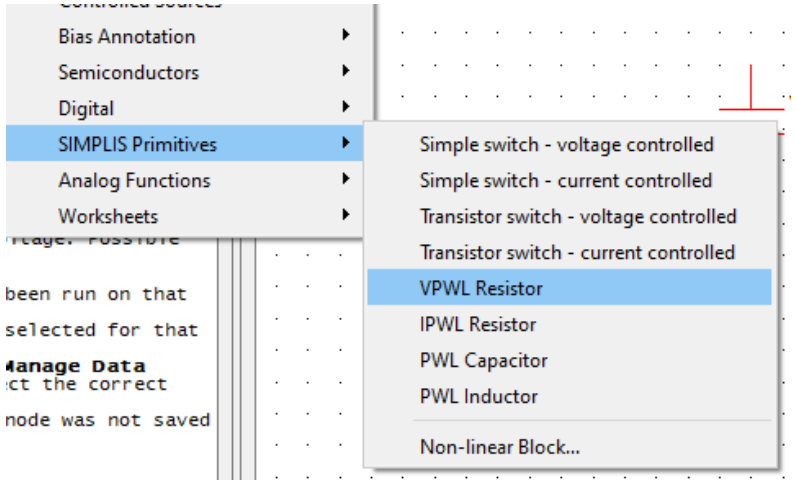


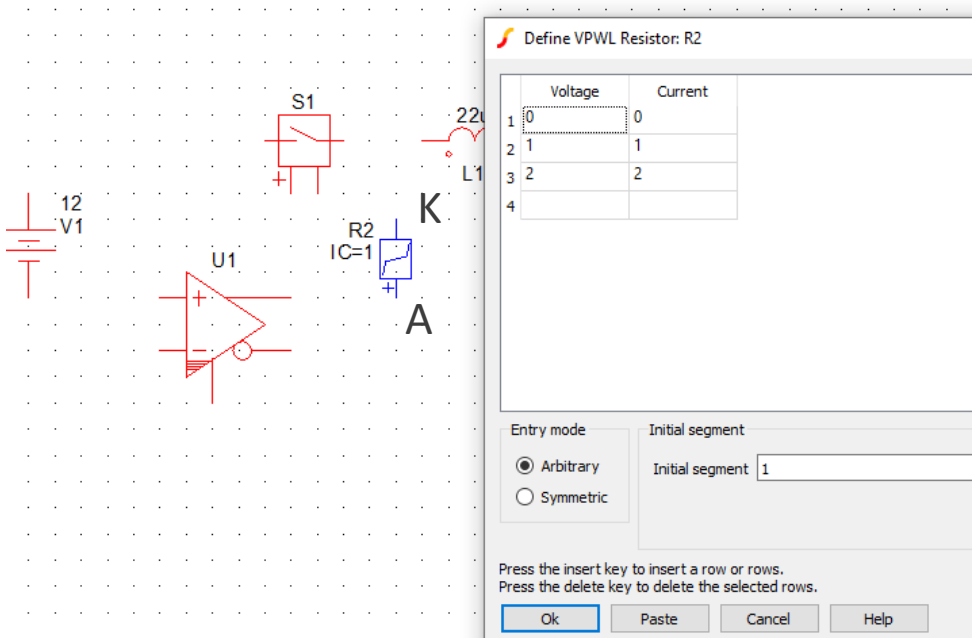


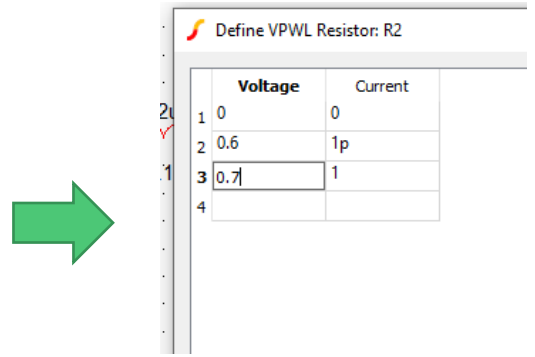


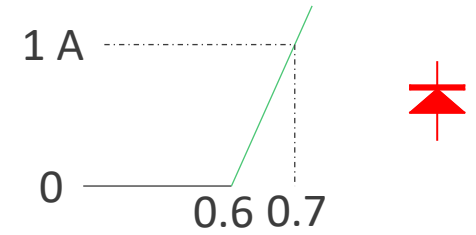


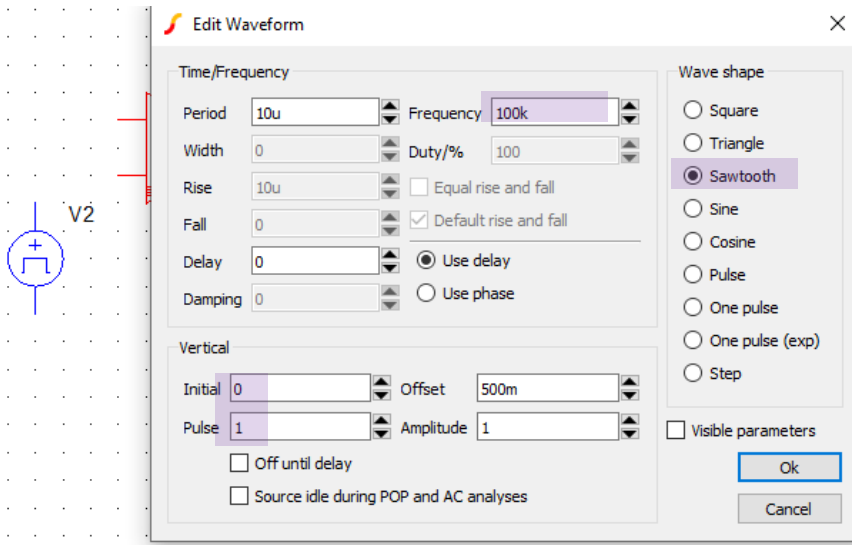
Simple buck 2



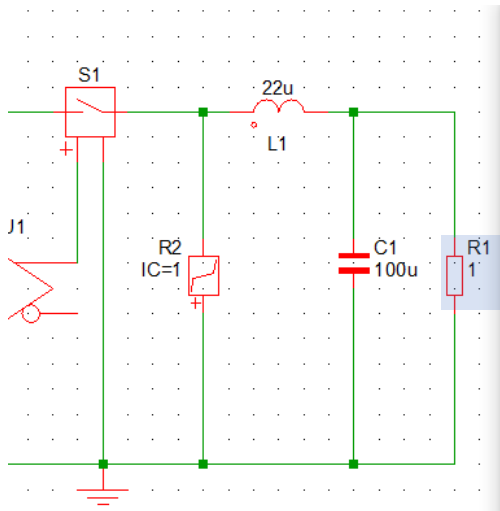


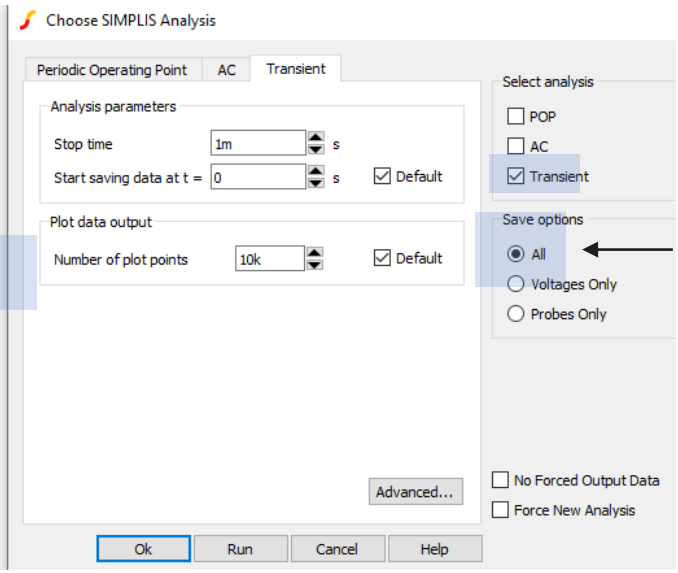


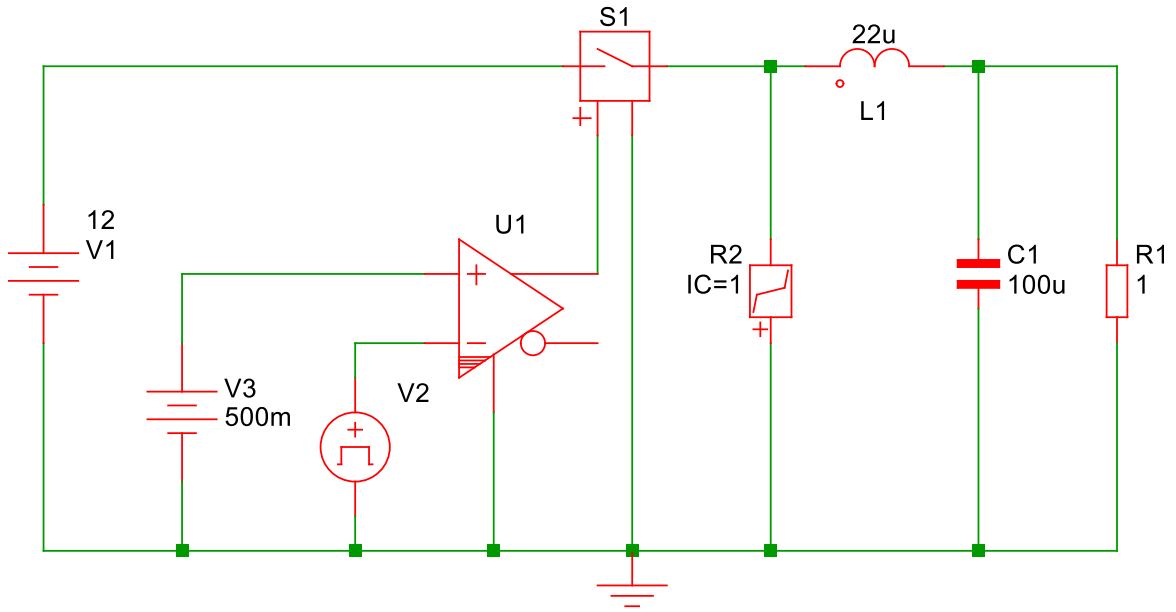


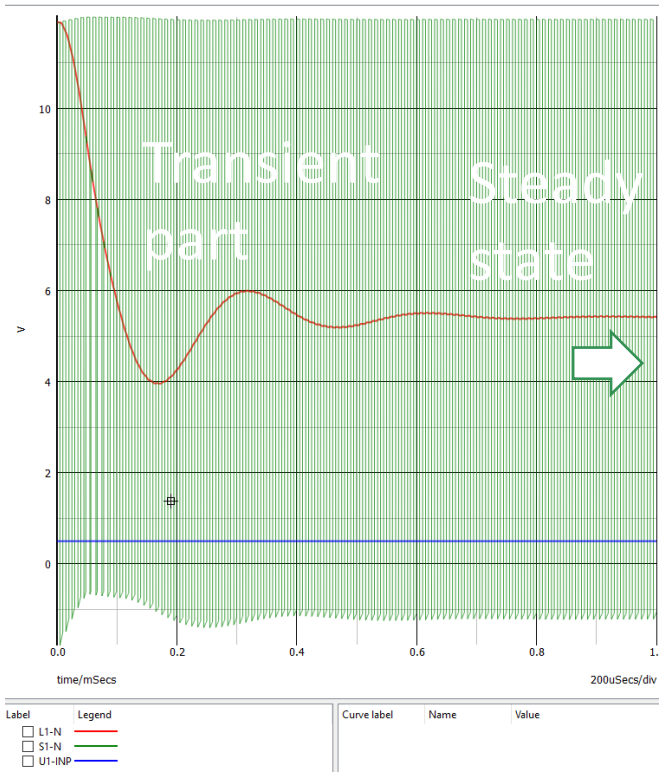


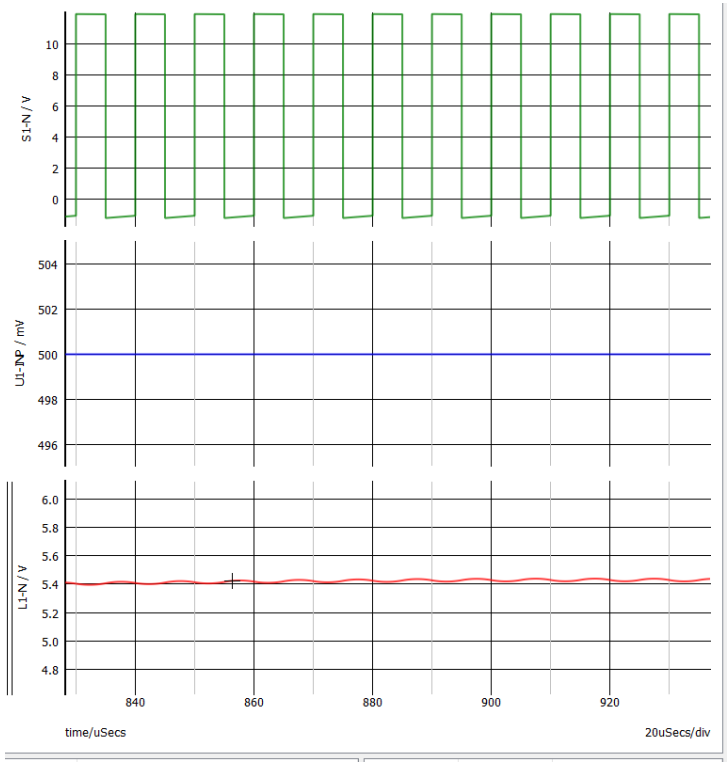
The 1-V sawtooth generator



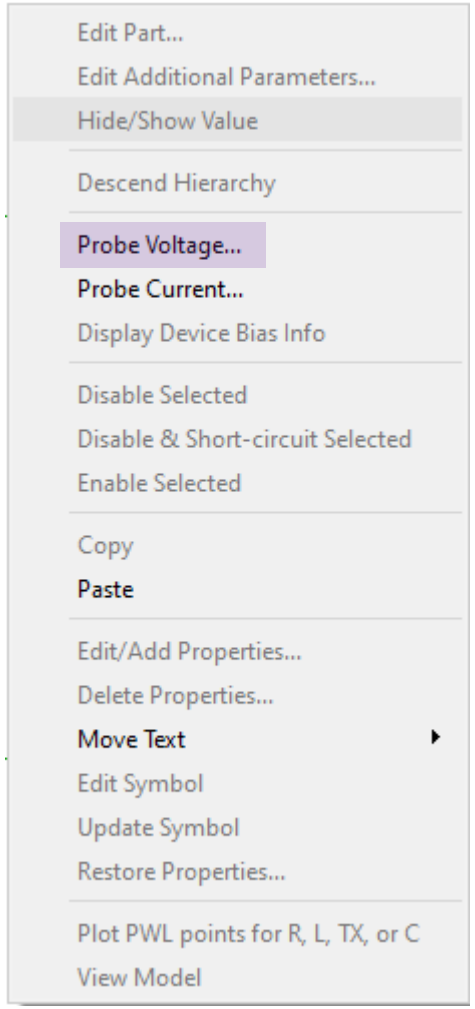






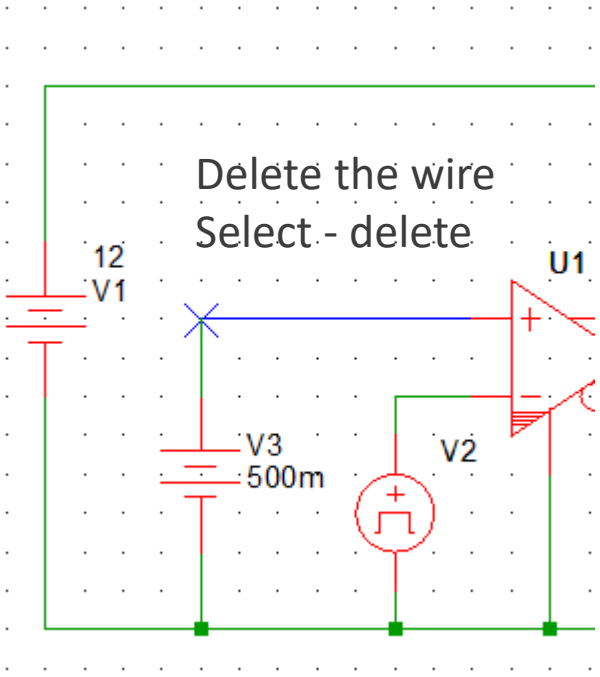


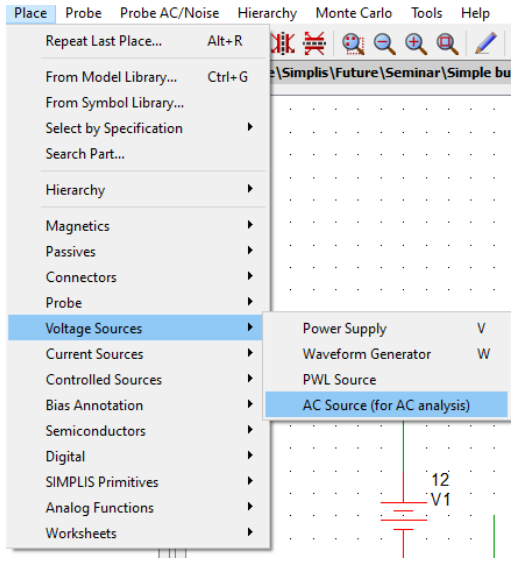
Stack and zoom-in

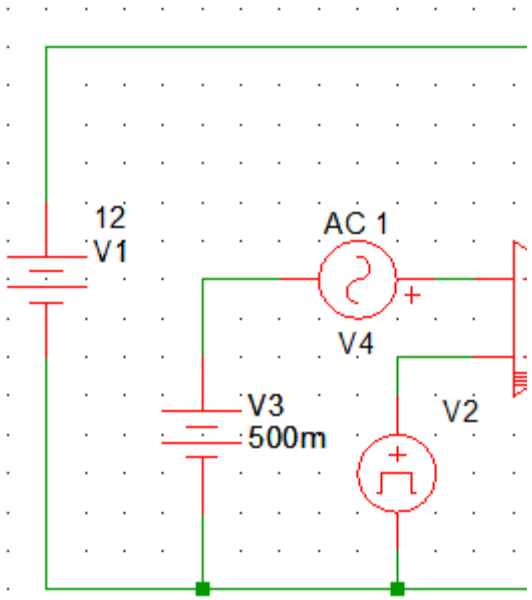


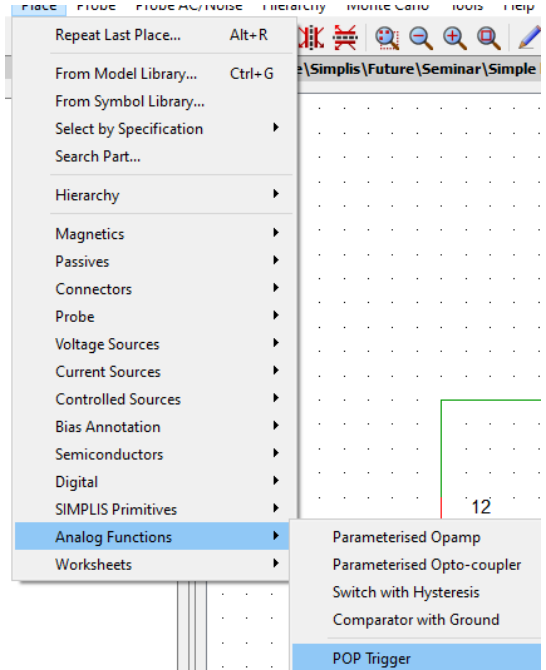
You can probe all nodes as you want and stack curves in the end

The strength of SIMPLIS is to extract the small-signal response from this switching circuit

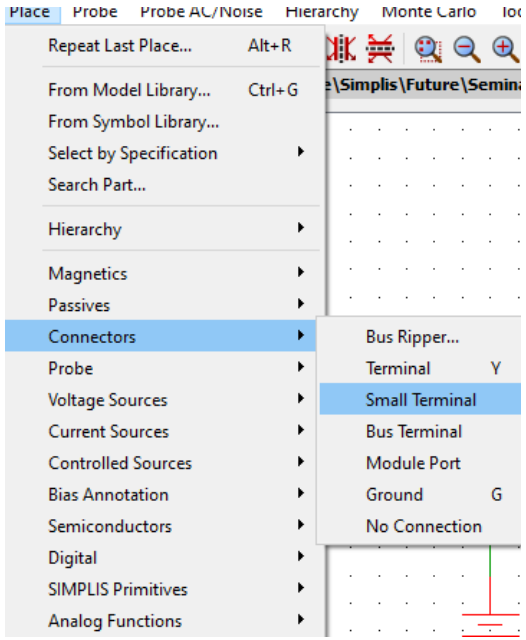


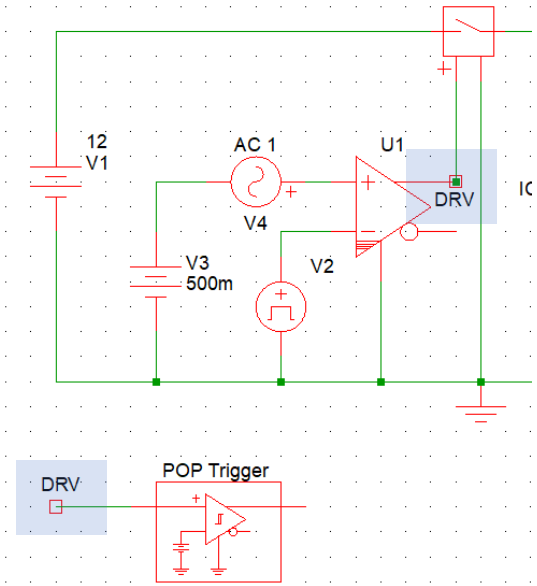




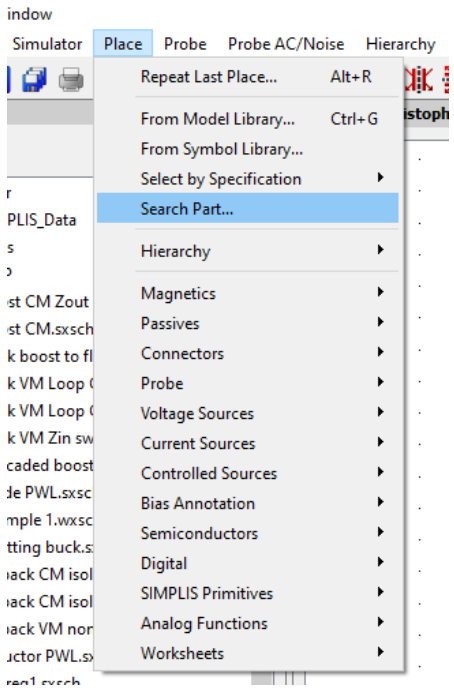


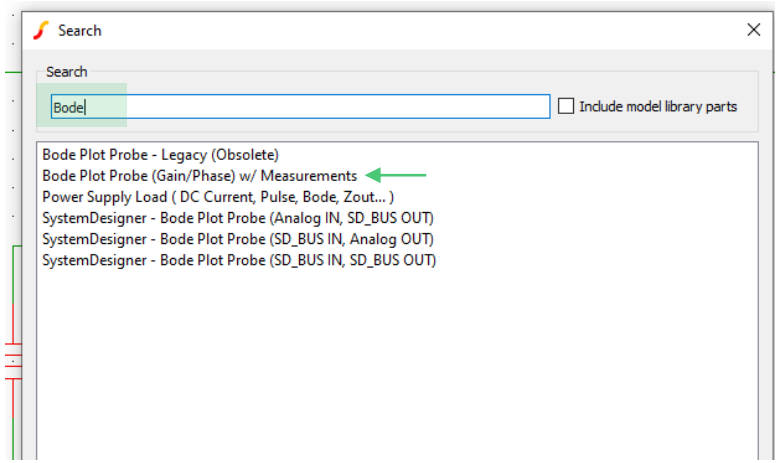
The POP for Periodic Operating Point instructs SIMPLIS on switching event

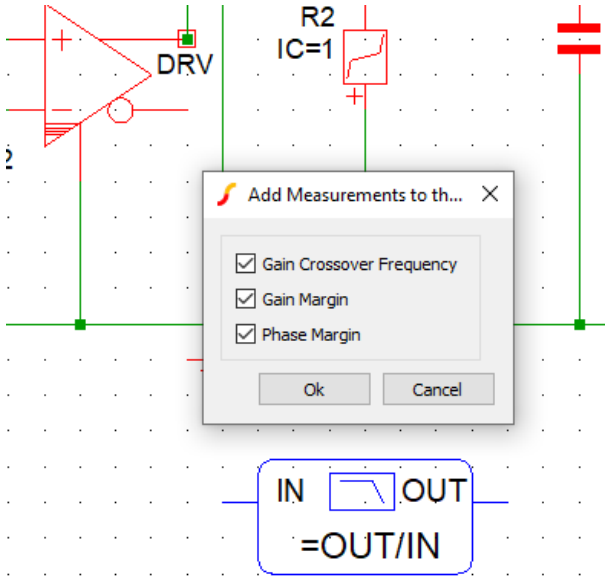




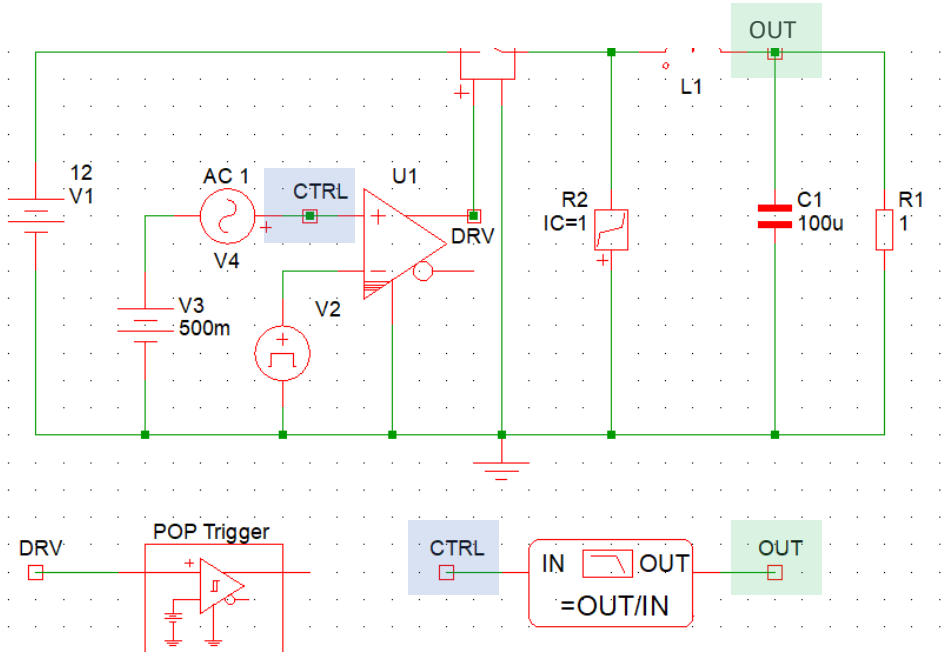
We now have a means to excite the converter, we need a means to reveal the Bode plot of the transfer function



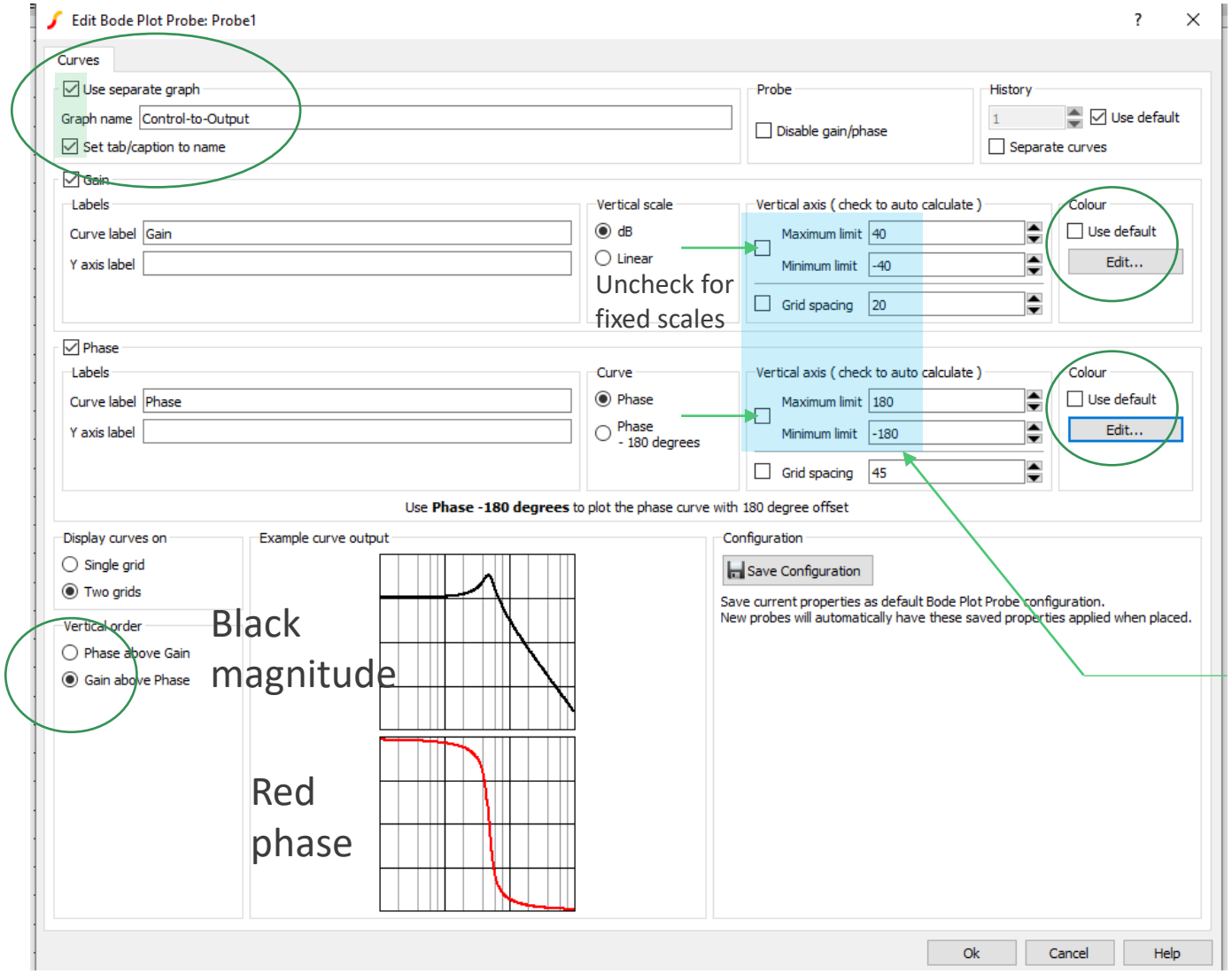




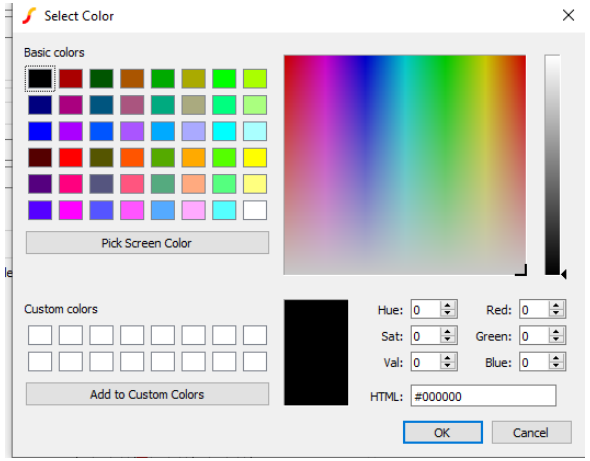
Add wires or small connectors



Enter a name and tick the two boxes

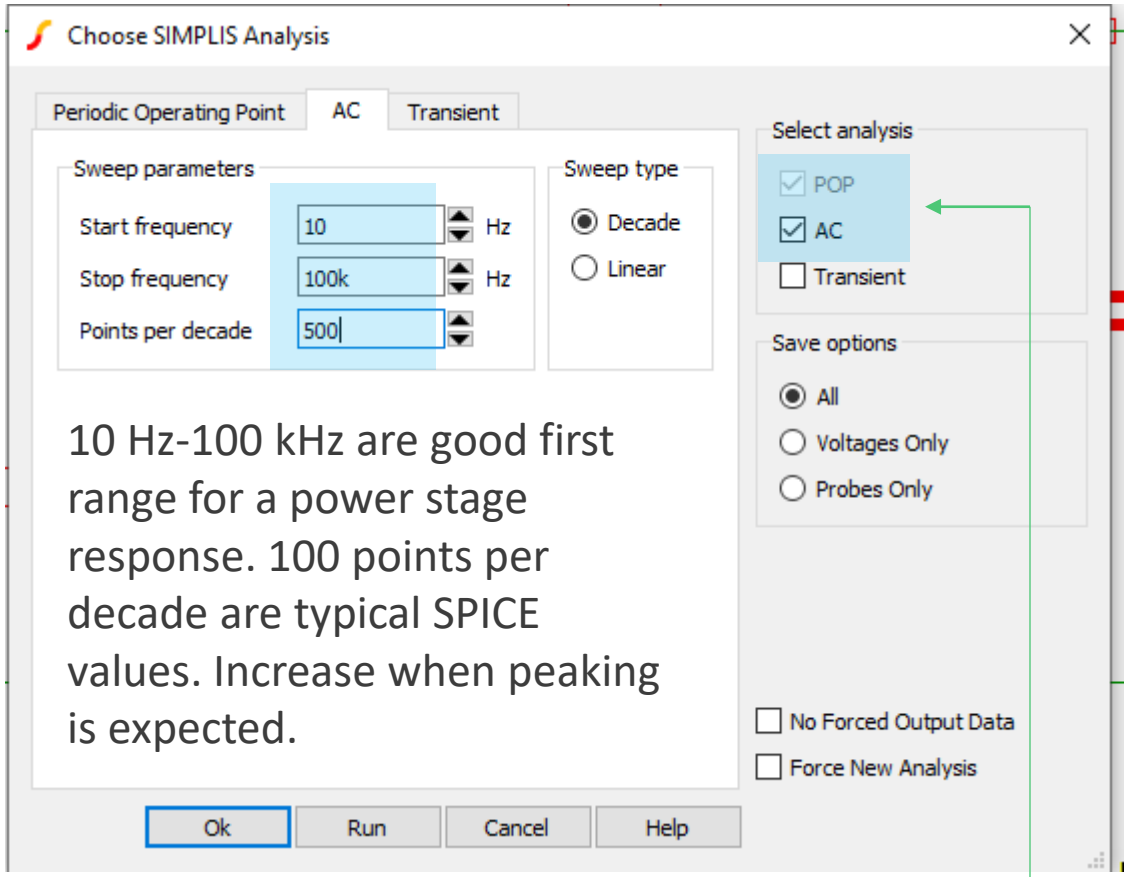


I like a black magnitude and a red phase curve :)



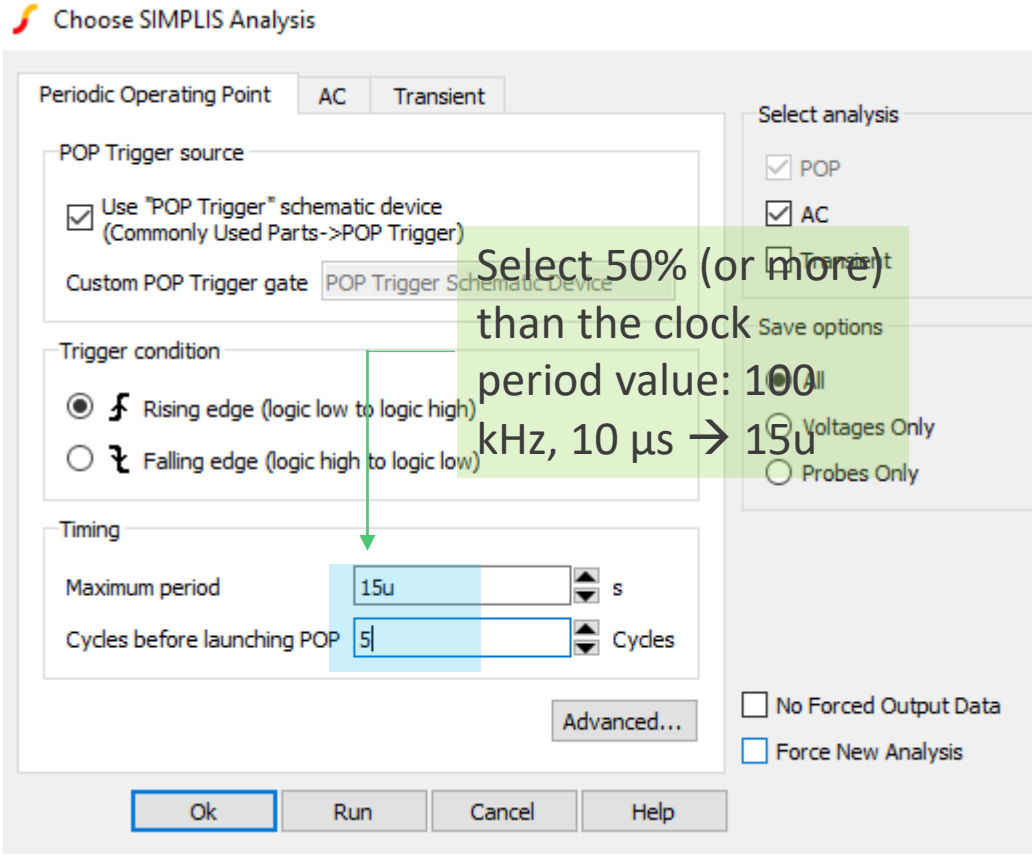
It is more comfortable to read when both magnitude and phase plots are symetrical with respect to the 0 point: 0 dB and 0°

Press F8 to access the simulation parameters



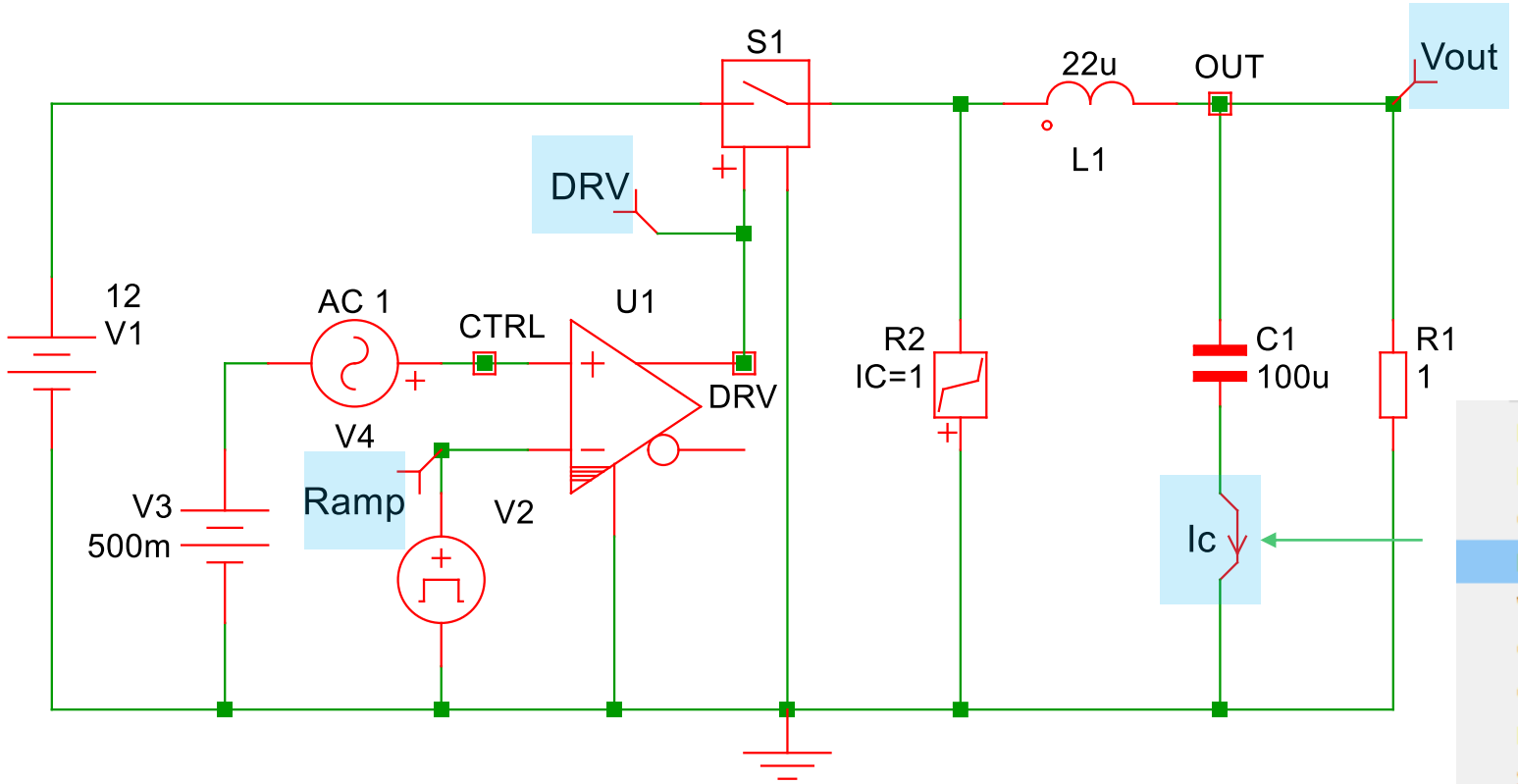
10 Hz-100 kHz are good first range for a power stage response. 100 points per decade are typical SPICE values. Increase when peaking is expected.

Tick ac analysis, POP will be checked automatically

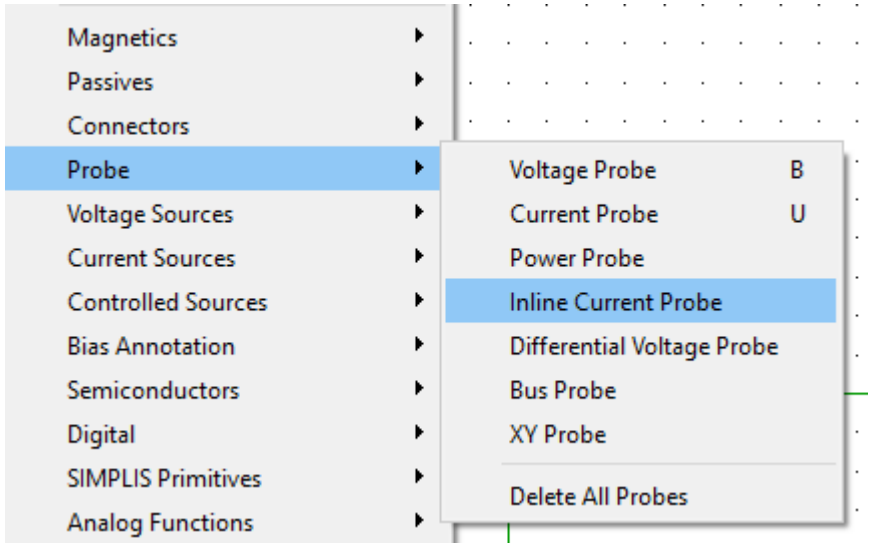


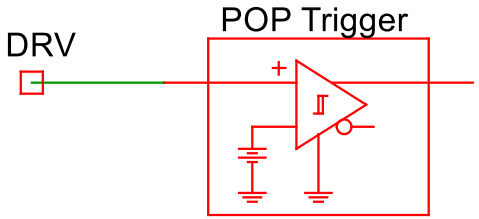
Select 50% (or more) than the clock period value: 100 kHz, 10 µs → 15u

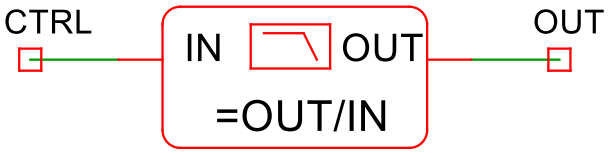
The periodic operating point or POP is a SIMPLIS proprietary algorithm which determines the steady-state operating point (average currents in capacitor and average voltages across inductors are all zero) in a record time. When POP is finished, the program launches the ac analysis.



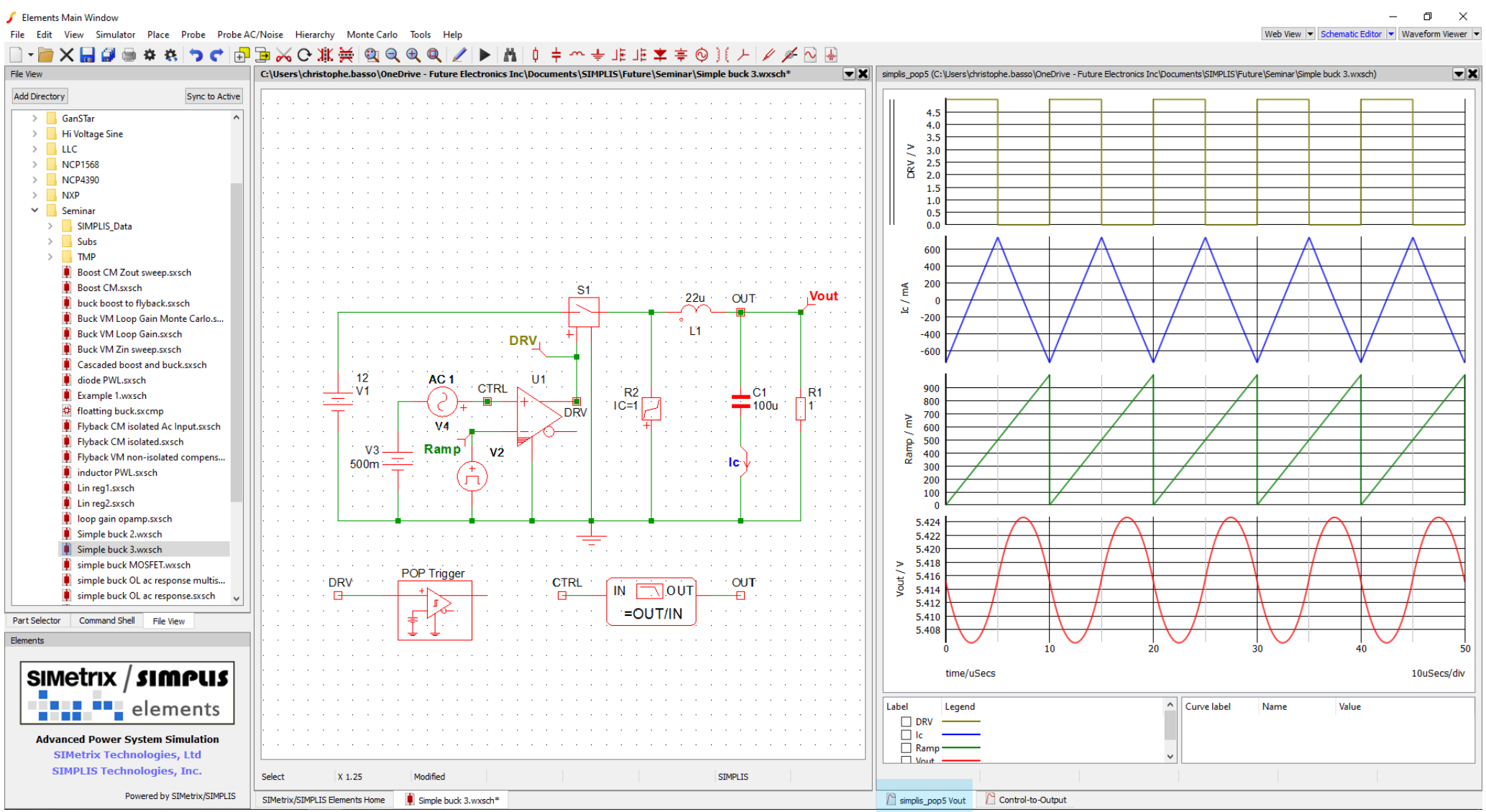
Add a few probes to visualize the waveforms after steady-state operation is done.



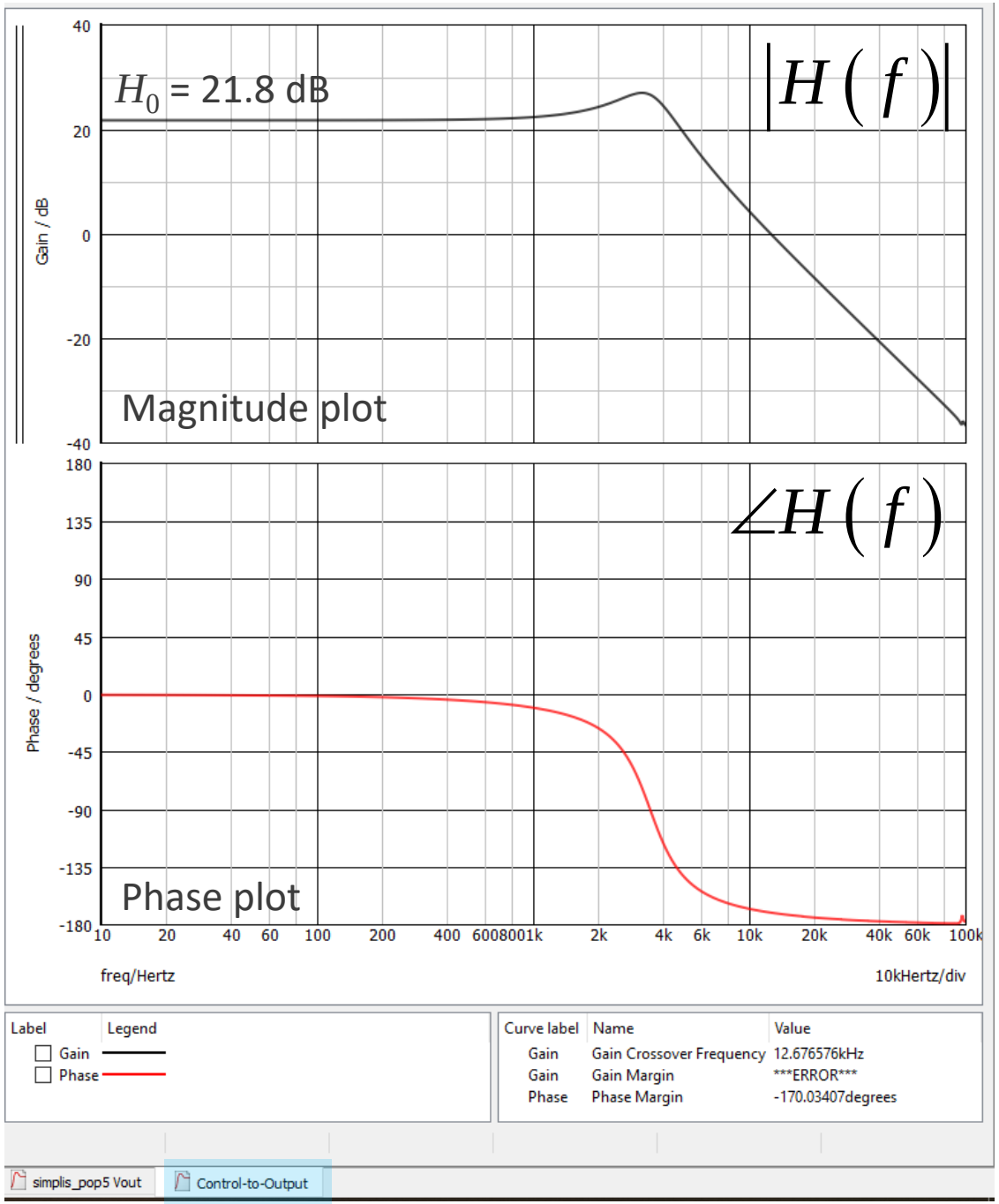




Ready to run the simulation! Press F9



This is the steady-state operating point: the converter is stabilized and the start-up sequence is long gone. The waveforms are the probes you placed



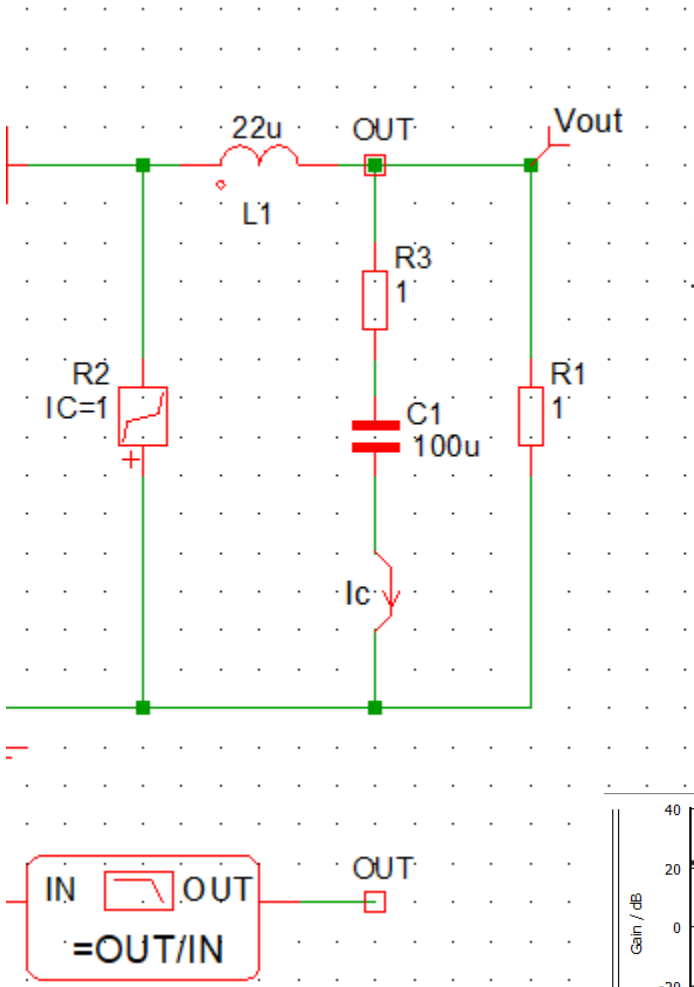
Quasi-static or dc gain value:

$$H(s) = H_0 \frac{N(s)}{1 + \frac{s}{\omega_0 Q} + \left(\frac{s}{\omega_0}\right)^2}$$

Input voltage, 12 V

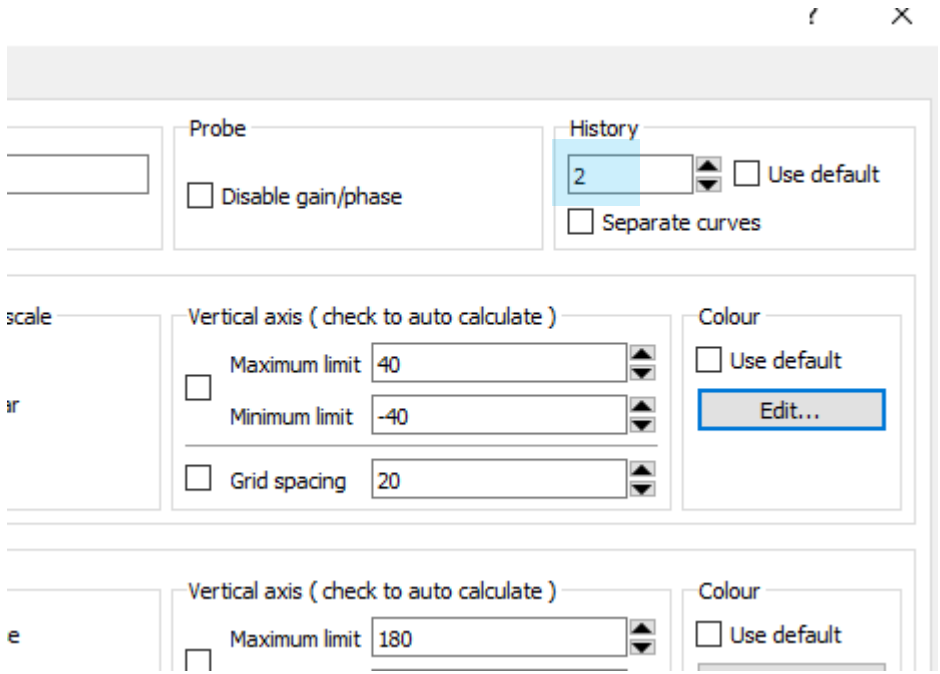
$$H_0 = \frac{V_{in}}{V_p} \frac{R_L}{R_L + r_L} \approx \frac{V_{in}}{V_p} \Rightarrow H_0 = 20 \log \frac{12\ \text{V}}{1\ \text{V}} \approx 21.6\ \text{dB}$$

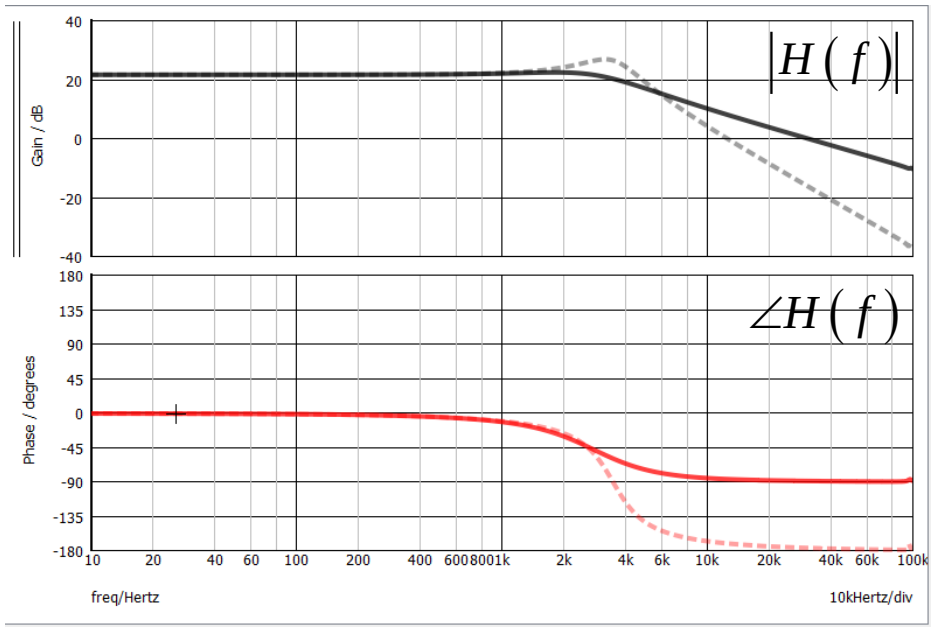
Peak amplitude of the sawtooth



Insert a resistor in the schematic

Select a remanence of 2

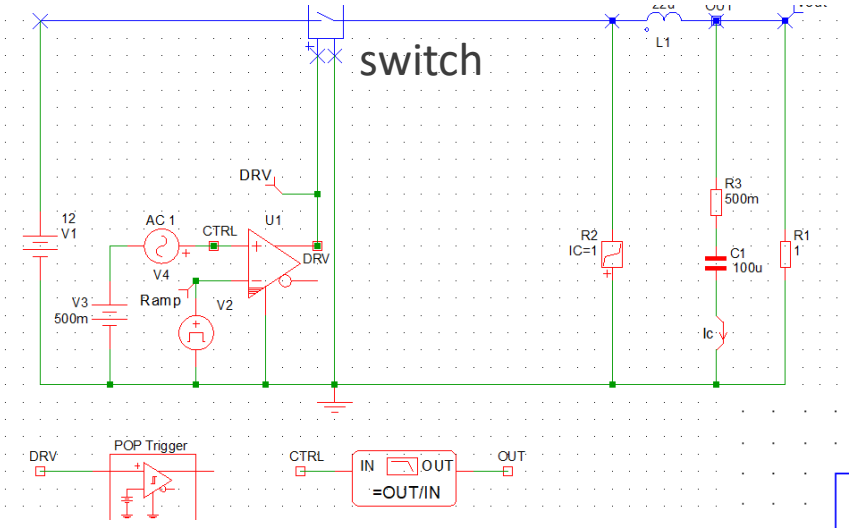




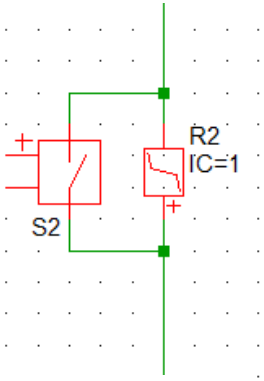
Change the series resistance value and see the different responses. The dashed lines are the first results.

Let's add some synchronous rectification

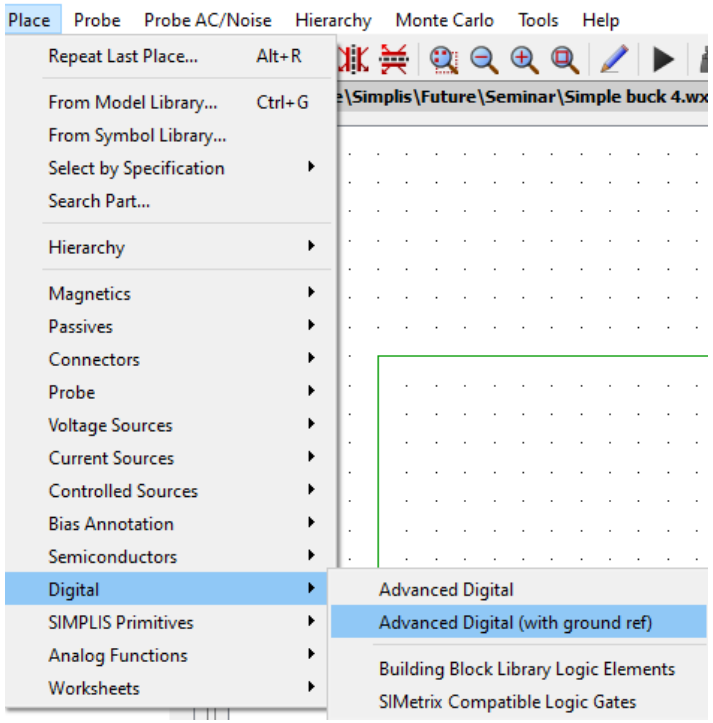
Make some space first

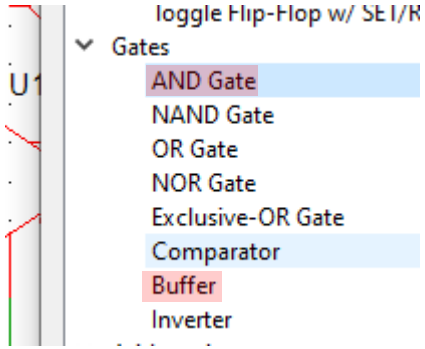


Copy the switch
and rotate it via F5.
Place it across the diode.



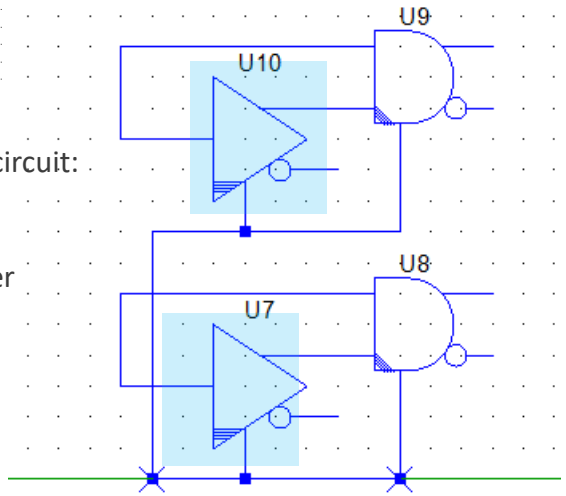
Grab a few gates to implement deadtime

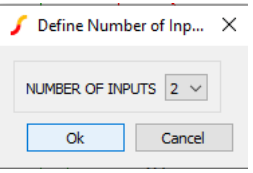


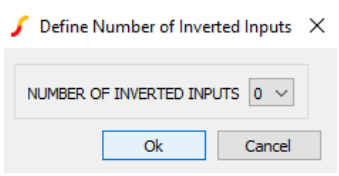


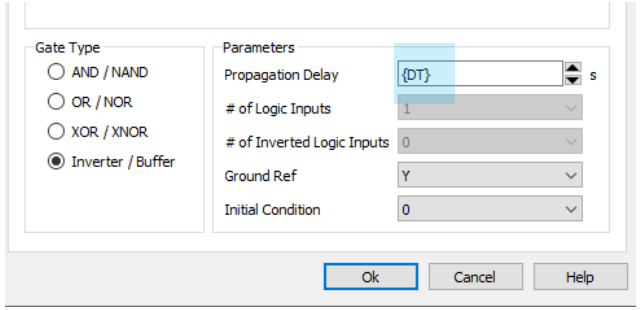
Capture this circuit:

Double-click
on each buffer

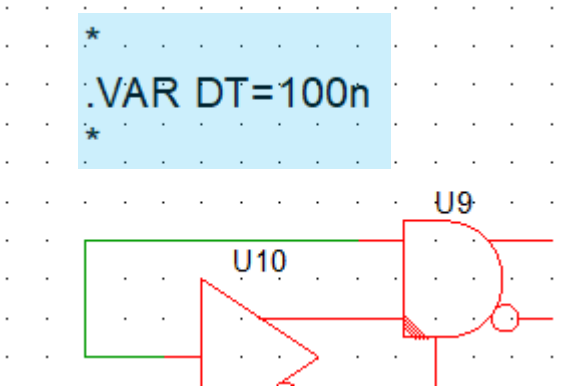




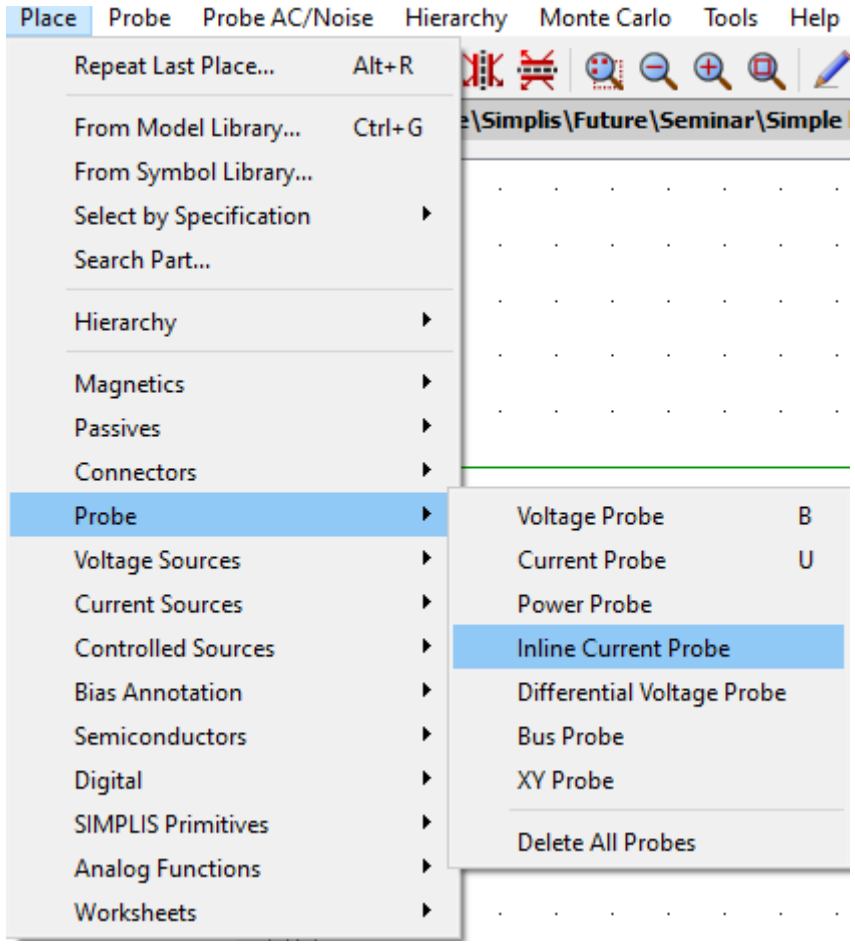


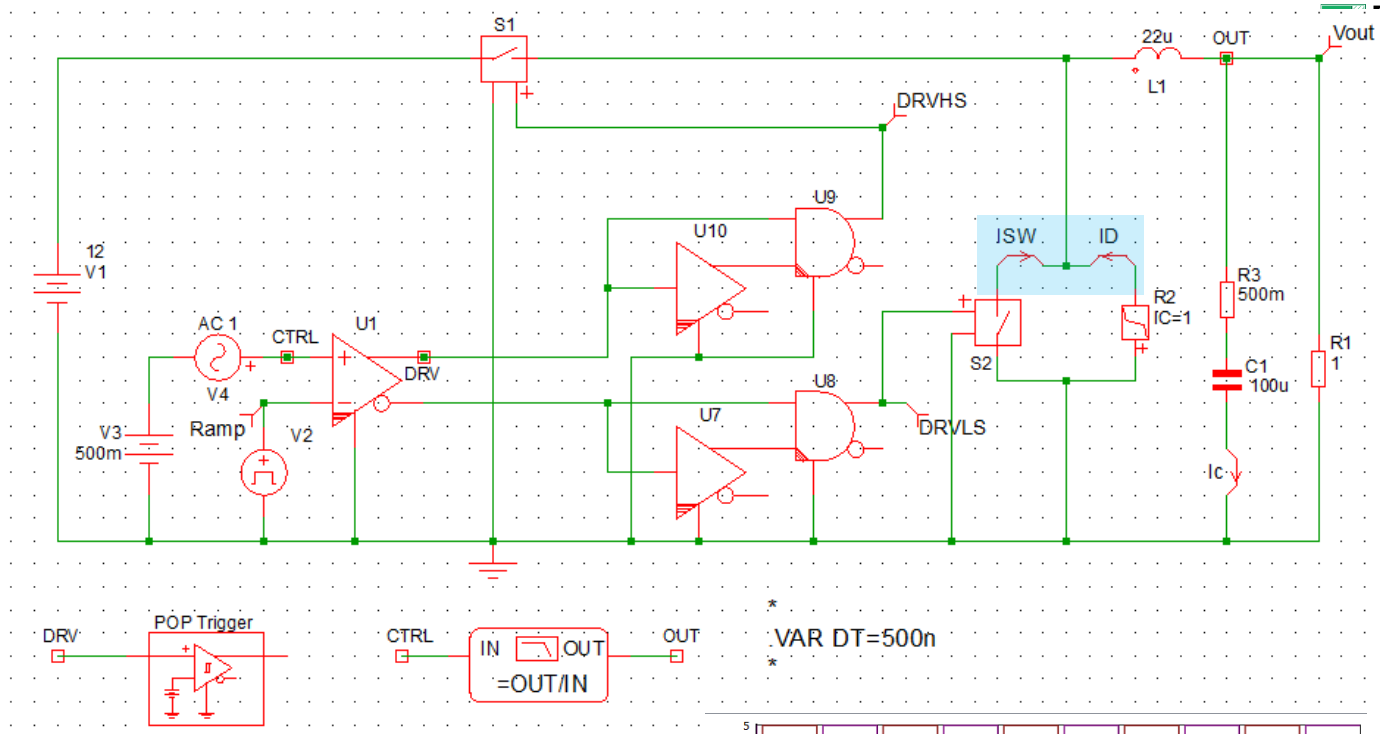


Insert a control block

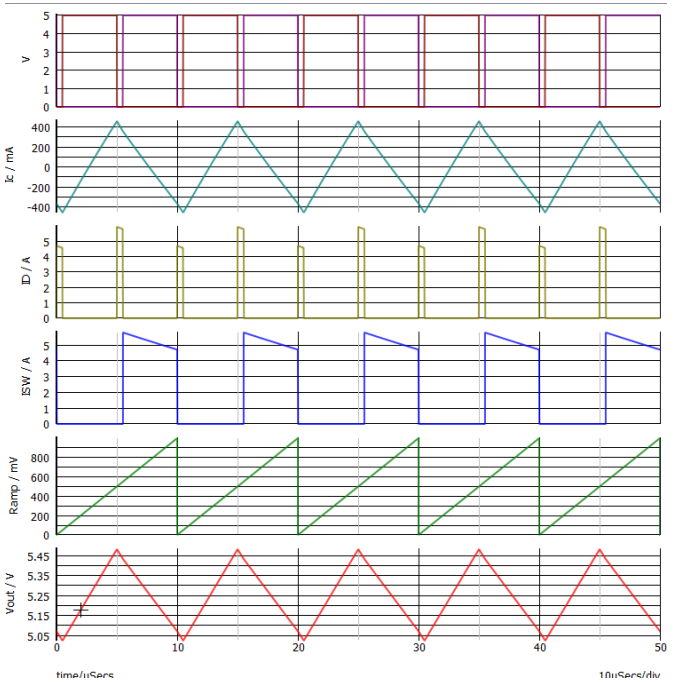


Finish the wiring and insert two current probes





Press F9 and see the waveforms and the effects of an exaggeratedly-large deadtime

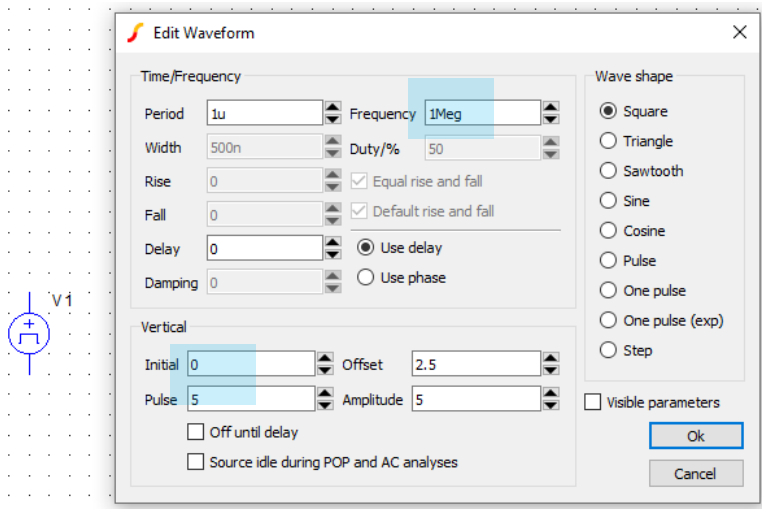


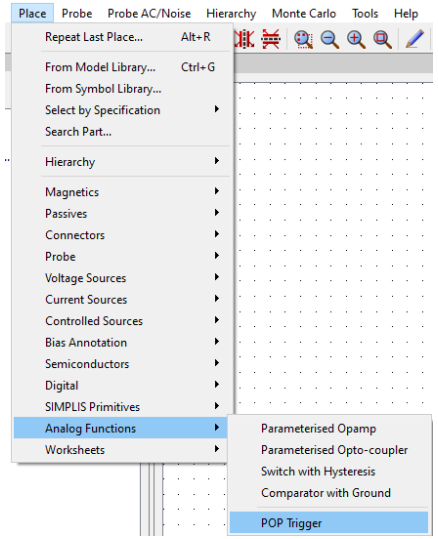
Agenda

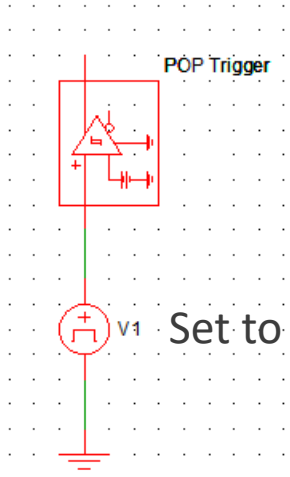
- What is Elements?
- Running a Basic Simulation
- A Buck Converter
- Ac Linear Analysis with SIMPLIS
- Importing a SPICE Model
- The Ready-to-Use Template

Ac analysis with SIMPLIS

SIMPLIS is a time-domain simulator therefore switching is needed to perform an ac analysis
→ “cheat” the simulator with a simple clock circuit



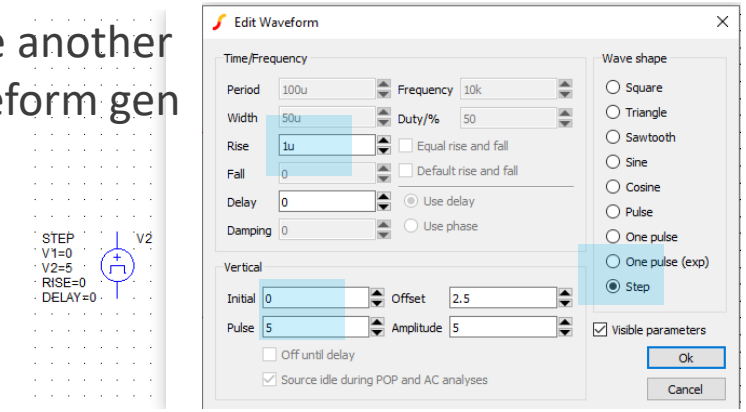


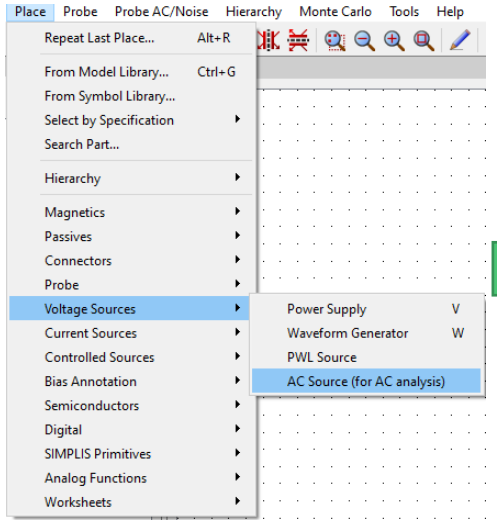


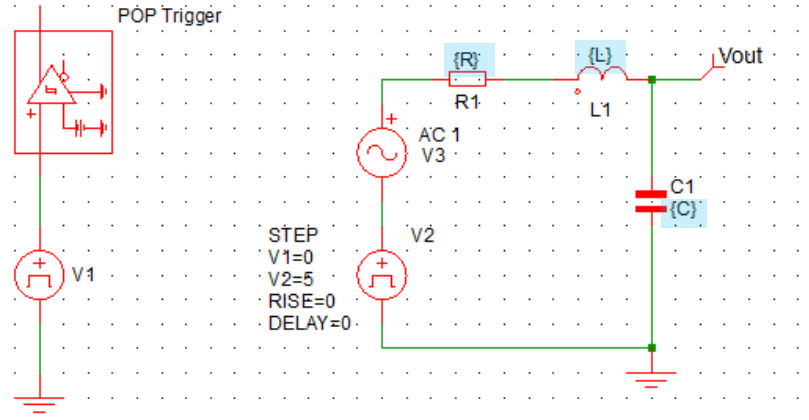
Set to 1 MHz

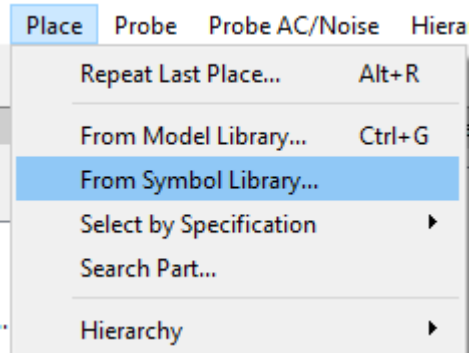
Add R, L and C with {parameters}

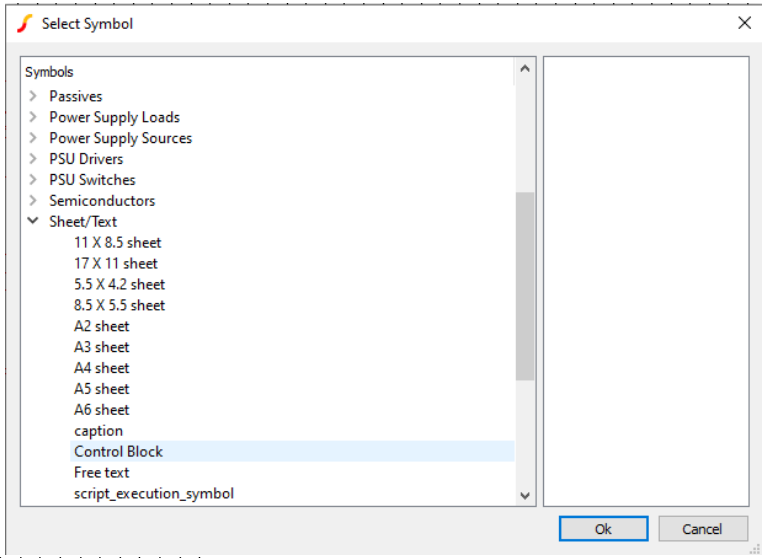
Place another waveform gen

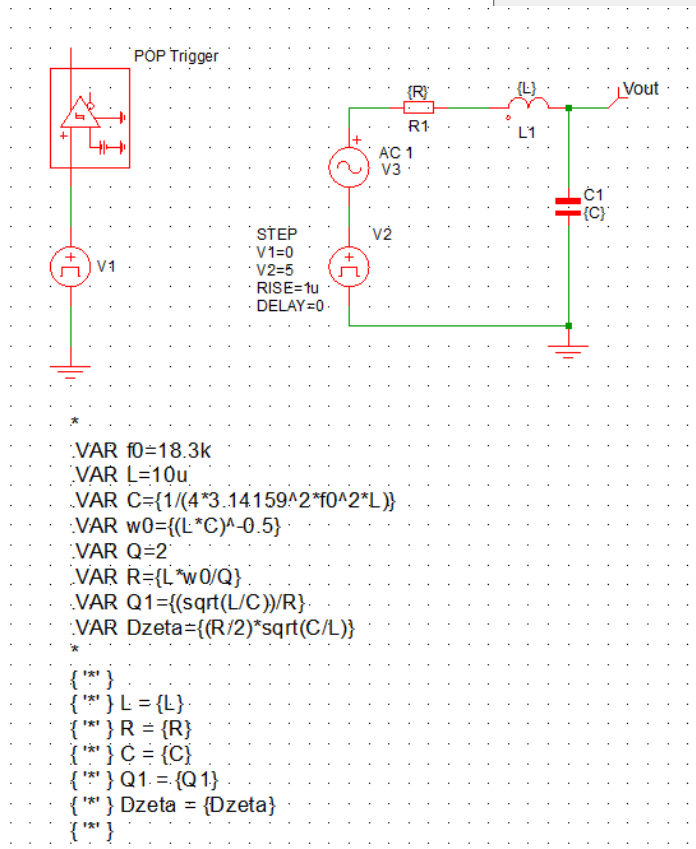












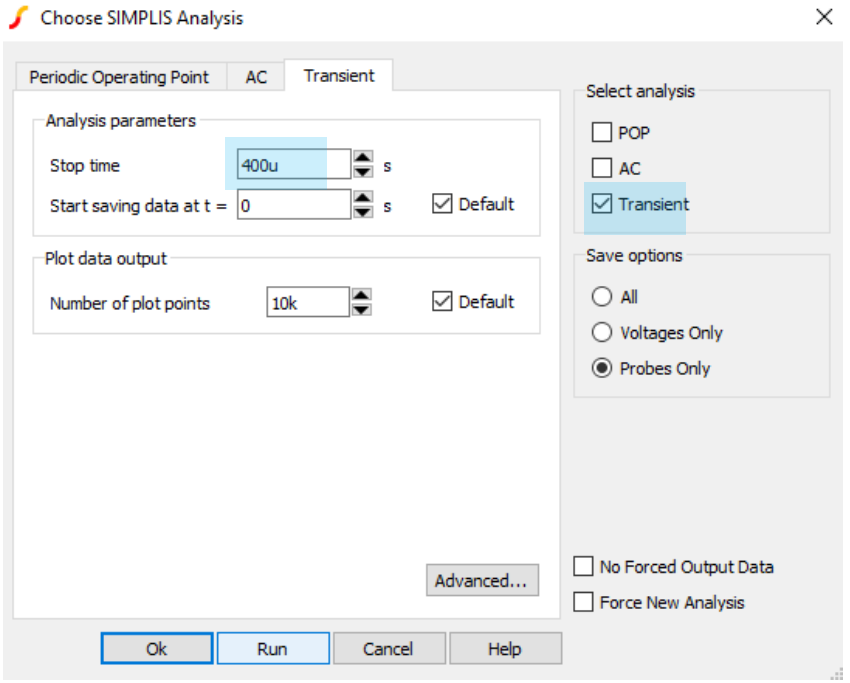
Copy/paste

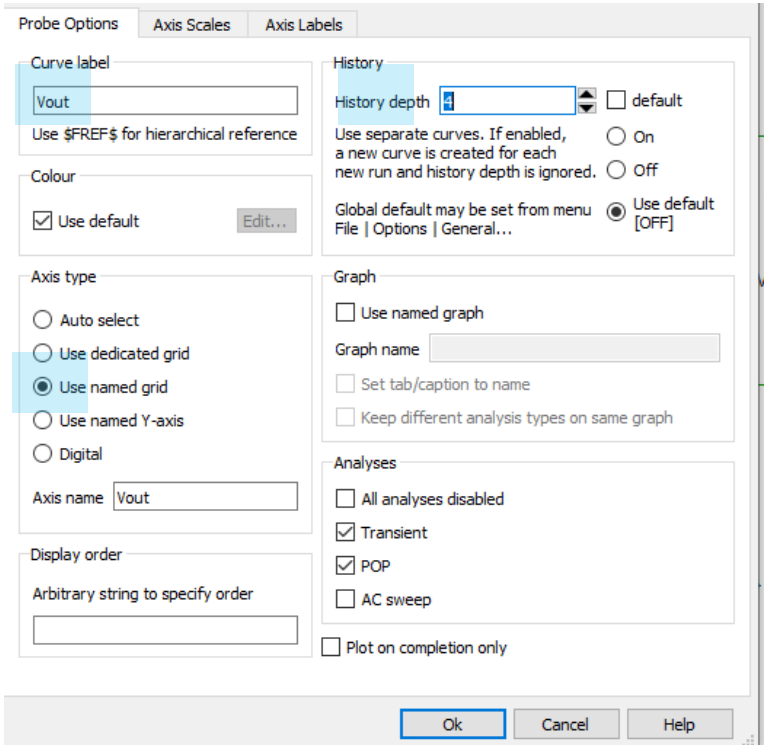
```
*
.VAR f0=18.3k
.VAR L=10u
.VAR C={1/(4*3.14159^2*f0^2*L)}
.VAR w0={(L*C)^-0.5}
.VAR Q=2
.VAR R={L*w0/Q}
.VAR Q1={(sqrt(L/C))/R}
.VAR Dzeta={(R/2)*sqrt(C/L)}
*
{ '*' }
{ '*' } L = {L}
{ '*' } R = {R}
{ '*' } C = {C}
{ '*' } Q1 = {Q1}
{ '*' } Dzeta = {Dzeta}
{ '*' }
```

Curly braces

Add these lines to display the calculated values

Press F8 for the simulation setup and select 400u for the analysis time

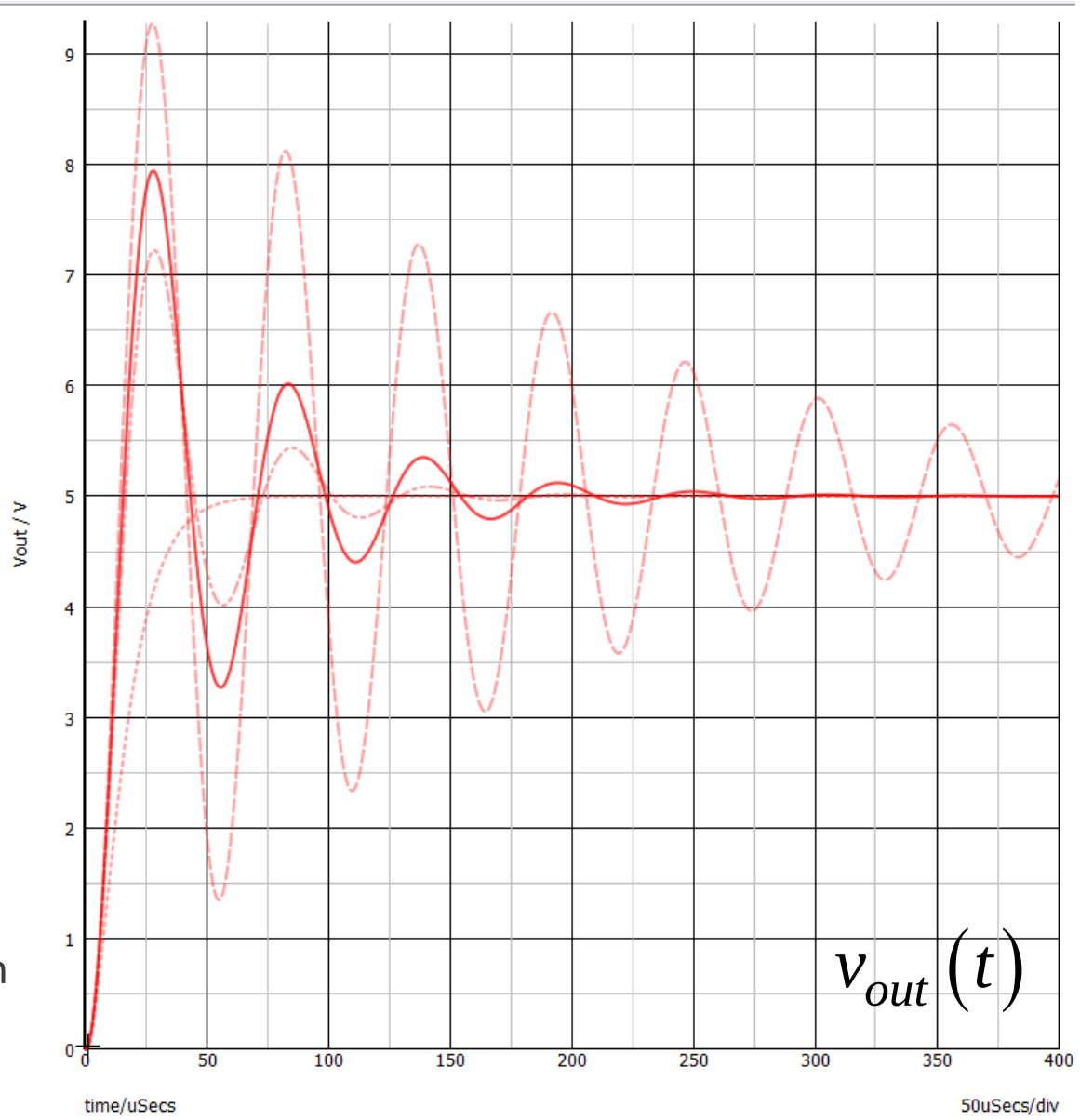




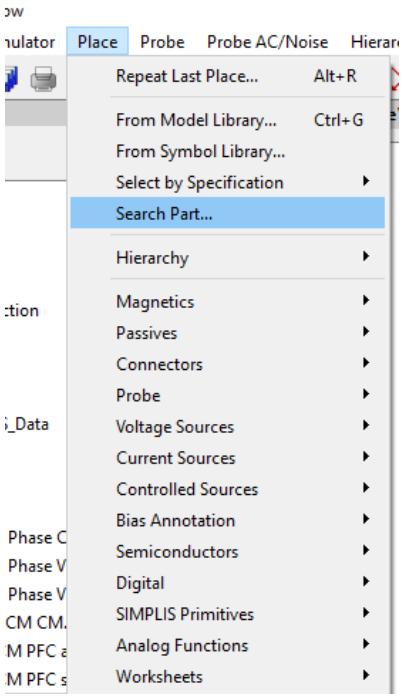
Add the V_{out} probe and select a persistence of 4

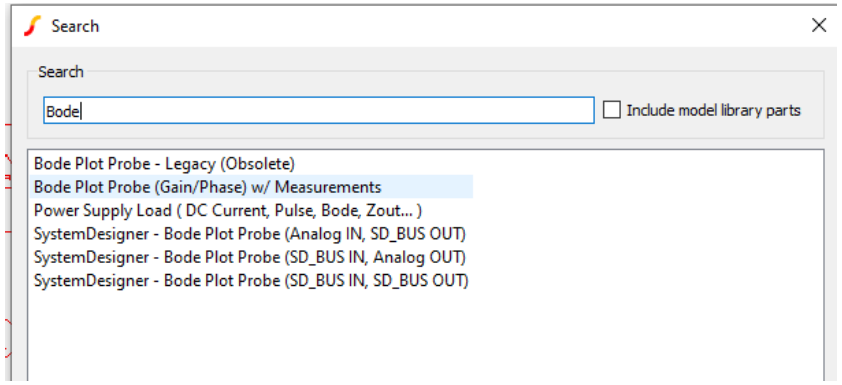
```
*
.VAR f0=18.3k
.VAR L=10u
.VAR C={1/(4*3.14159^2*f0^2*L)}
.VAR w0={(L*C)^-0.5}
.VAR Q=2
.VAR R={L*w0/Q}
.VAR Q1={(sqrt(L/C))/R}
.VAR Dzeta={(R/2)*sqrt(C/L)}
```

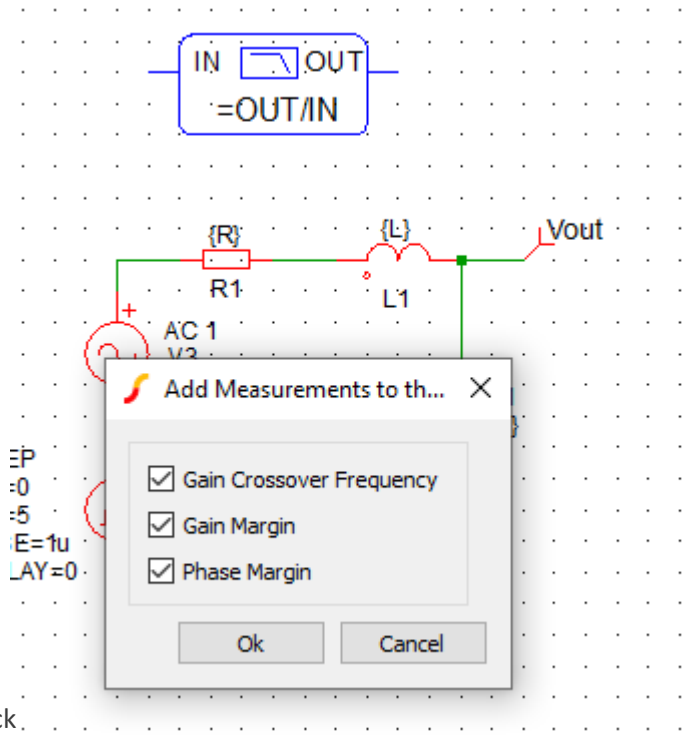
Change the value of Q in the control block and press F9. The plain curve is the last run.

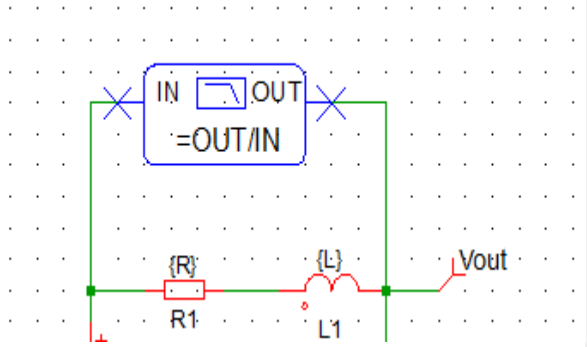


For the ac analysis, add a Bode box:

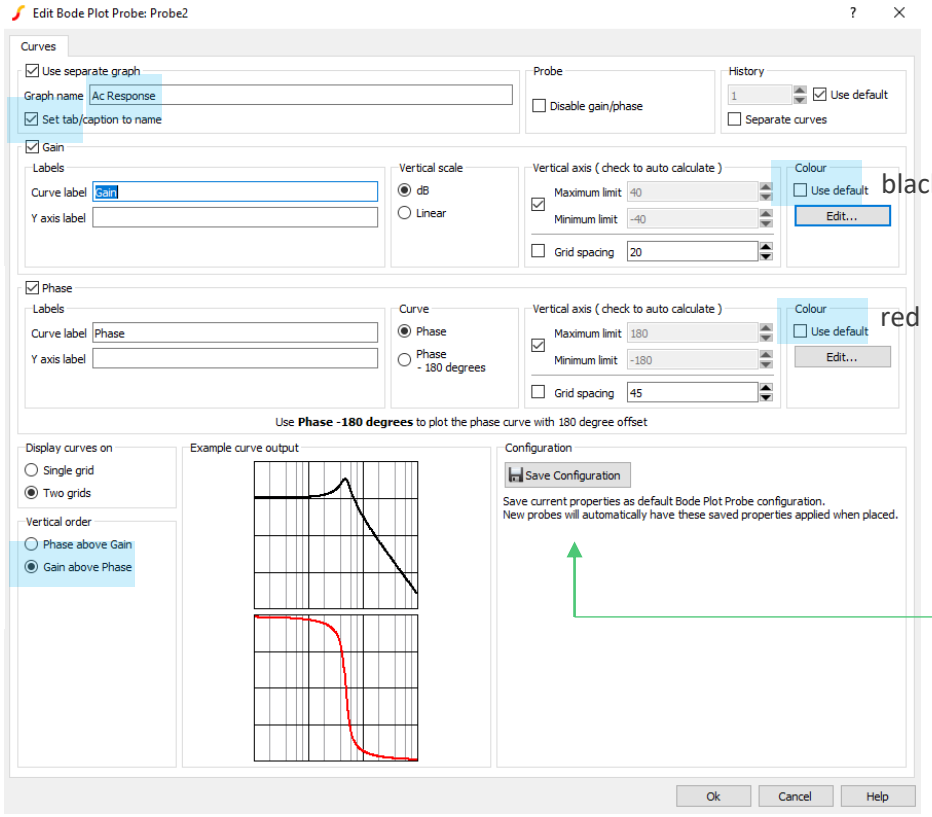






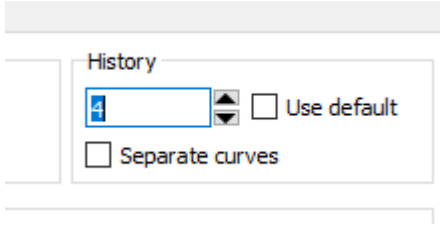


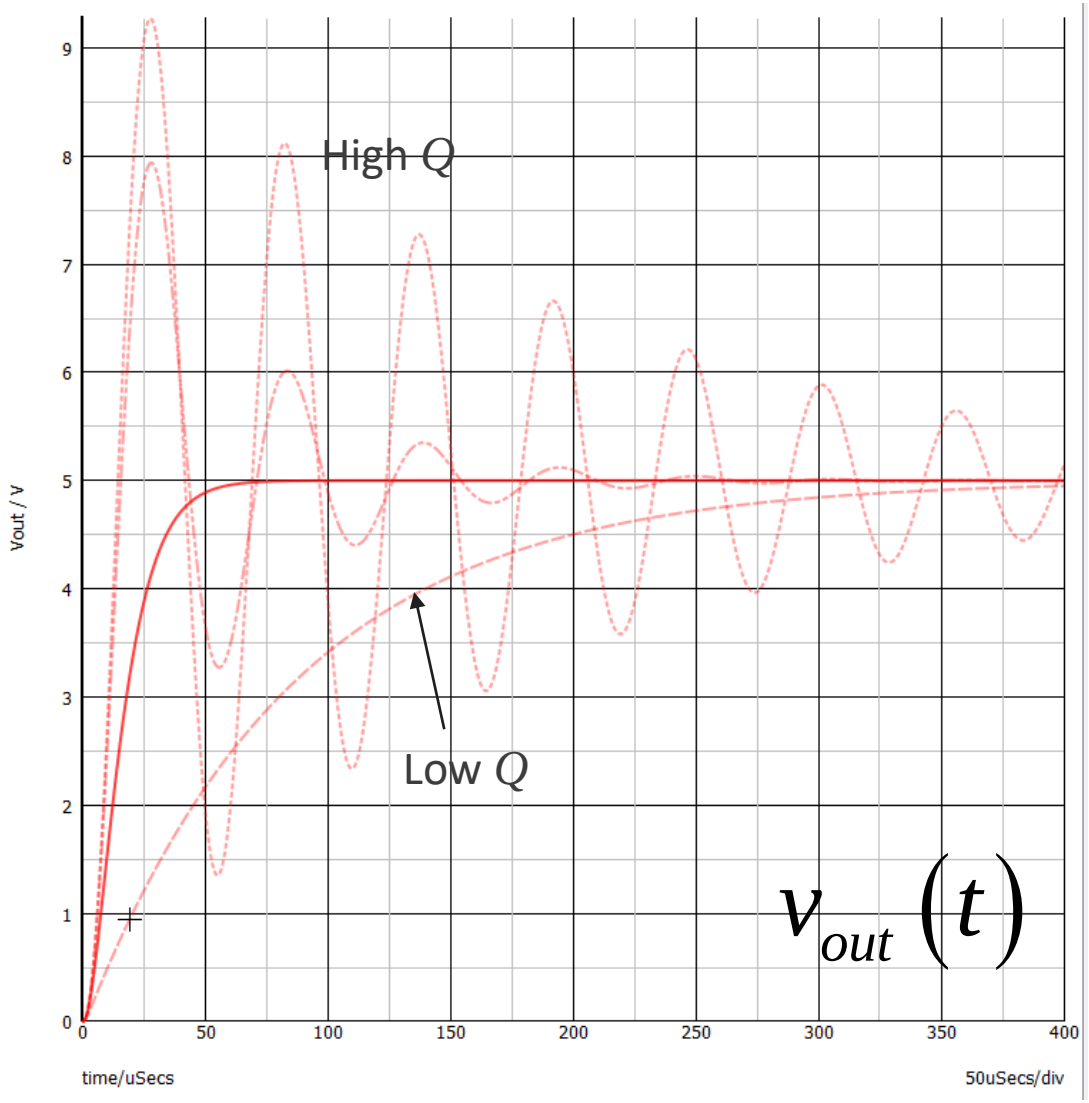
Connect in/out to the correct nodes

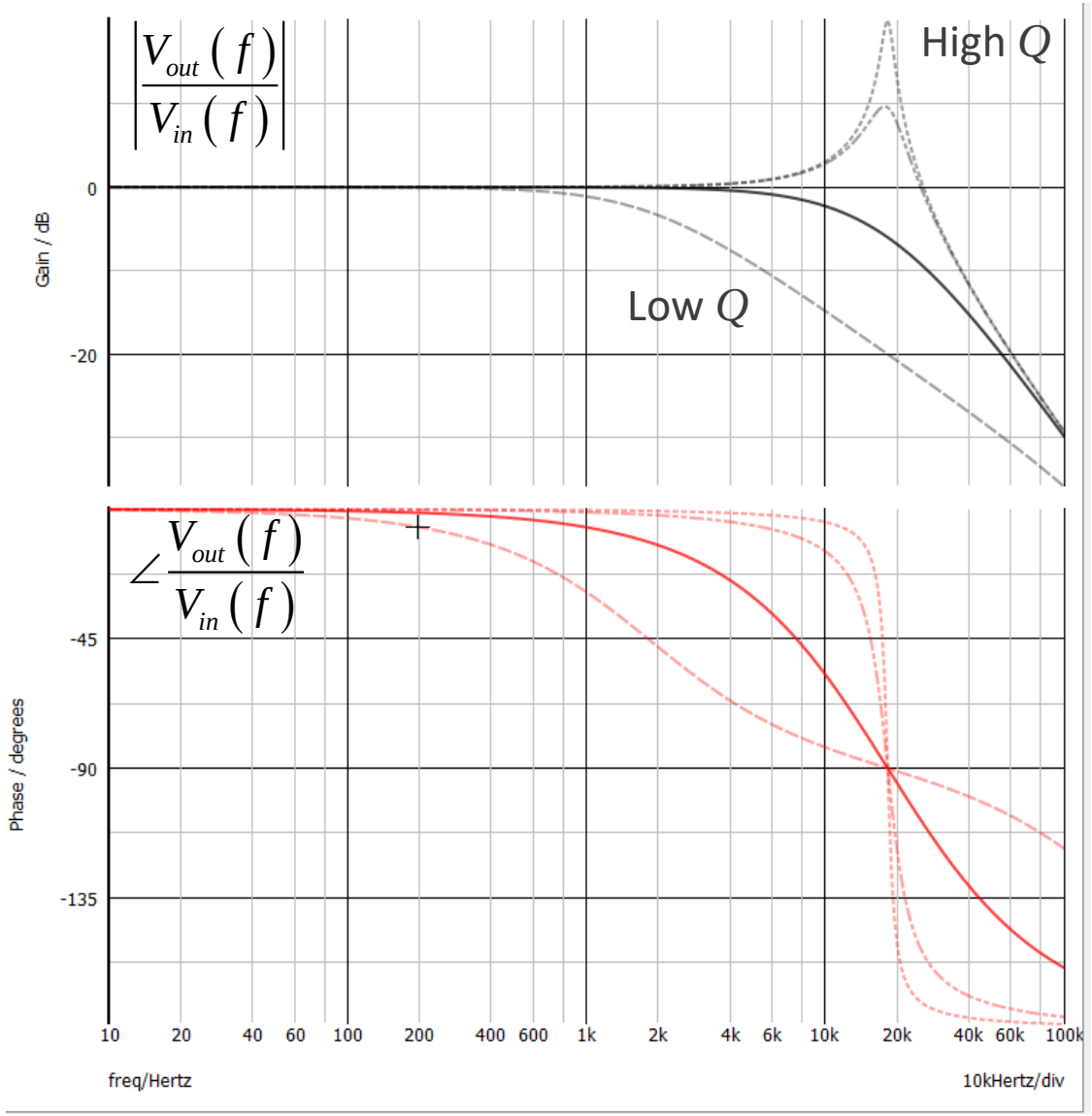


You can save the Bode box configuration for the next studies if you want.

If you set history or persistence to 4 and change Q, you should see





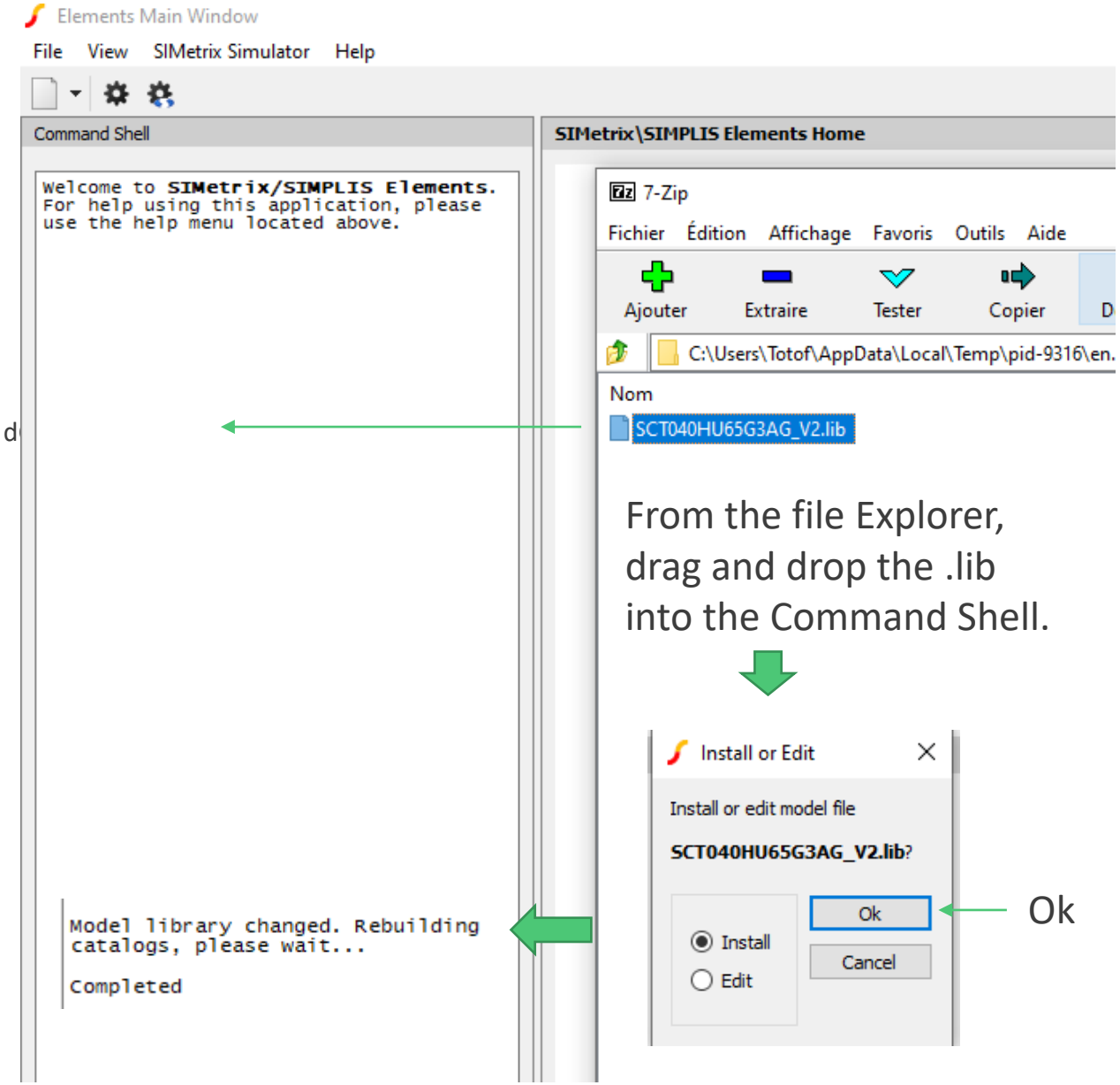


Agenda

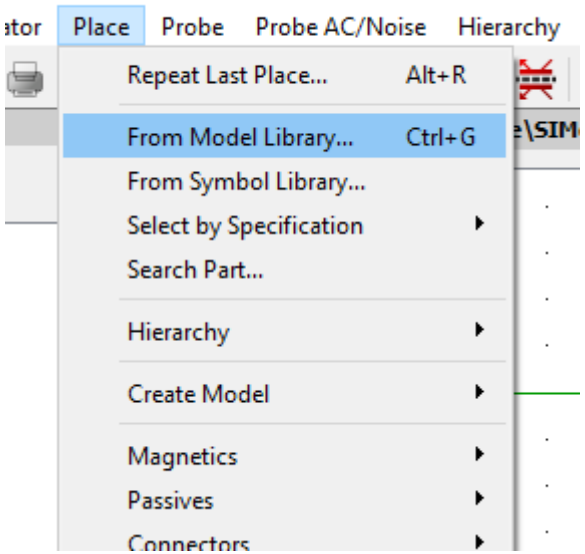
- What is Elements?
- Running a Basic Simulation
- A Buck Converter
- Ac Linear Analysis with SIMPLIS
- Importing a SPICE Model
- The Ready-to-Use Template

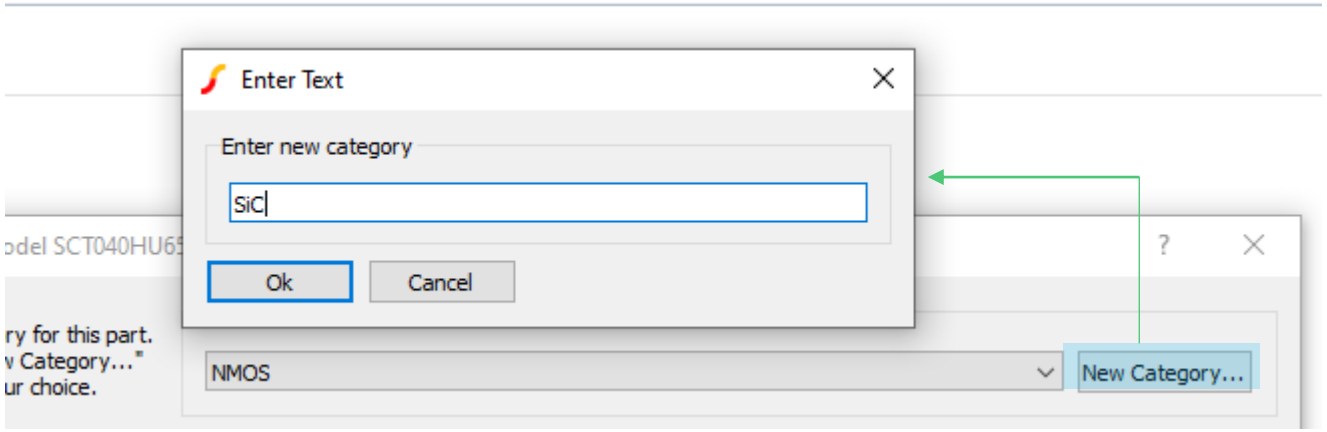
Assume you have downloaded a SPICE model coming from a manufacturer, ST in this example:

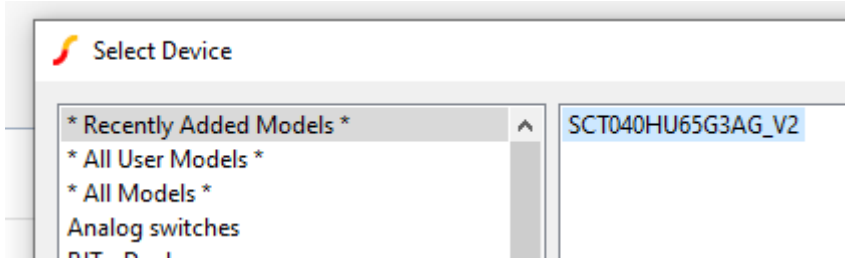
```
**********************************************************************
****** STMicroelectronics MOSFET, IGBT and Bipolar Library ******
**********************************************************************
*                                                                    *
* STMicroelectronics therefore does not assume any                   *
* responsibility arising from their use.                             *
* STMicroelectronics reserves the right to change models             *
* without prior notice.                                              *
*                                                                    *
* v.19.3.0 - FEB 2021                                                *
**********************************************************************
.subckt SCT040HU65G3AG_V2 drain gate source kelvin  PARAMS:  dR=0 dVth=0 dVsd=0 dCi=0 d

E1    Tj     val_T     VALUE={TEMP}
R1   val_T  0      1m
Rkelvin kelvin s2 5m
Ckelvin kelvin s2 1p
VLd    drain   d3x  0
*RLd   drain   d3x    1
VR_dr  d3  d2   0
Rdrain-fissa d3x d3 3m
VLg gate    g2     0
*RLg gate    g2      1
VLs source  s2      0
*RLs source  s2      1

Rg   g2 g1  1.0
……
```

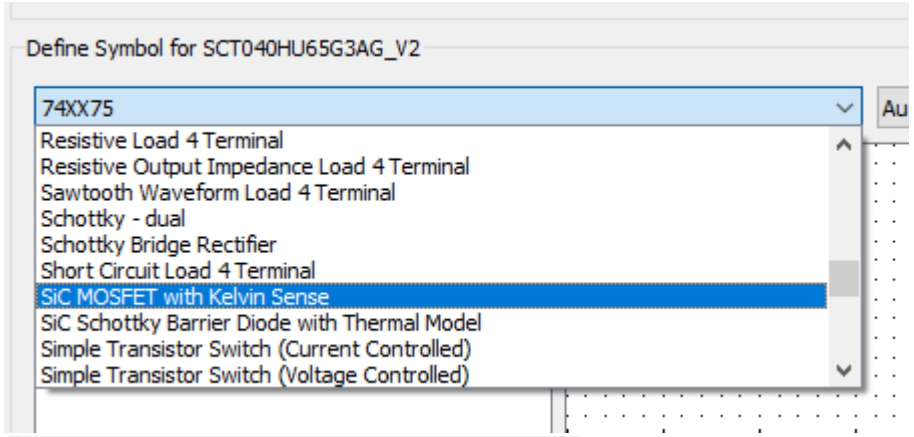


Now that the .LIB has been installed, you have to associate a symbol with it:

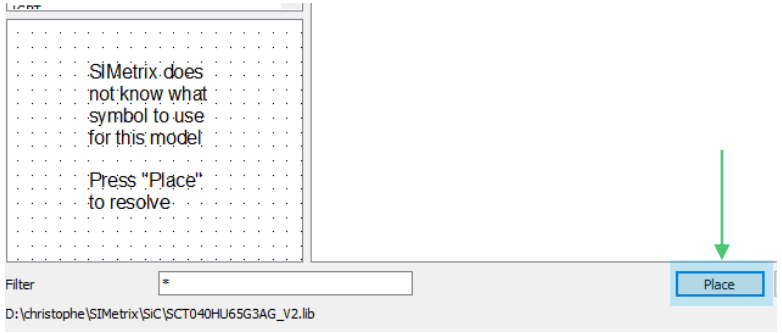


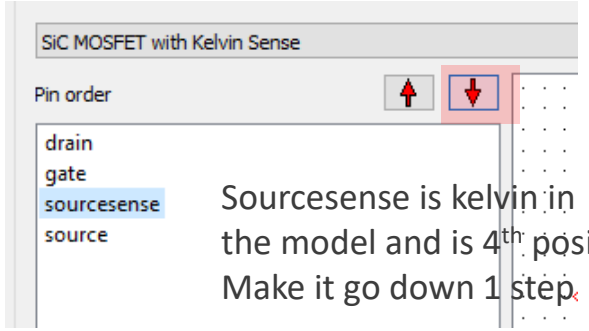






Choose the right symbol

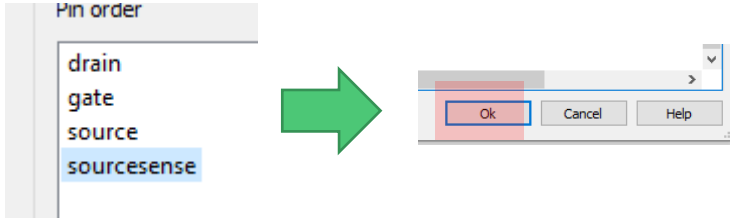




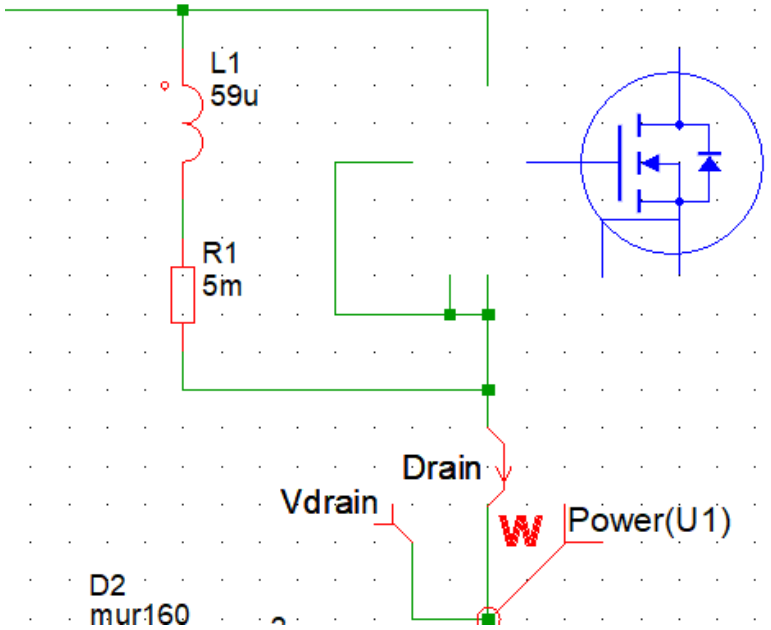
Sourcesense is kelvin in the model and is 4th position. Make it go down 1 step

1 2 3 4

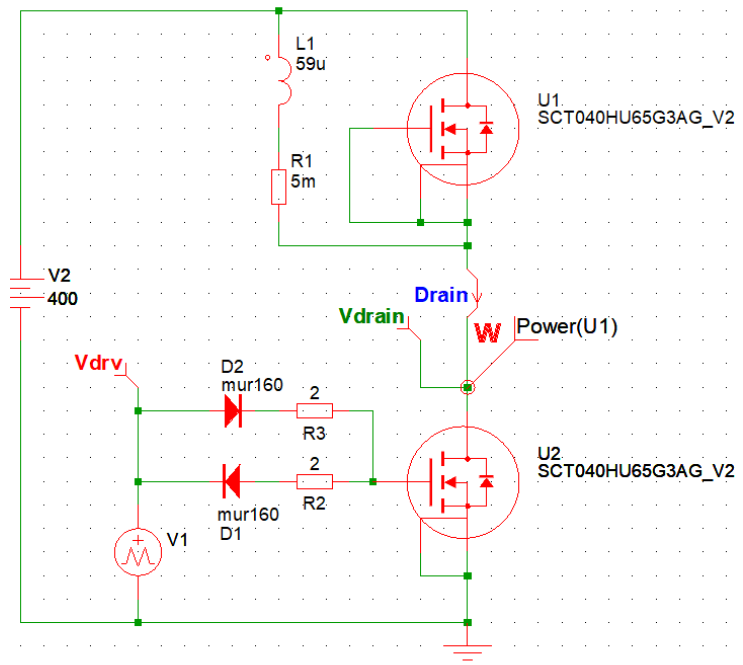
.subckt SCT040HU65G3AG_V2 drain gate source kelvin PARAMS

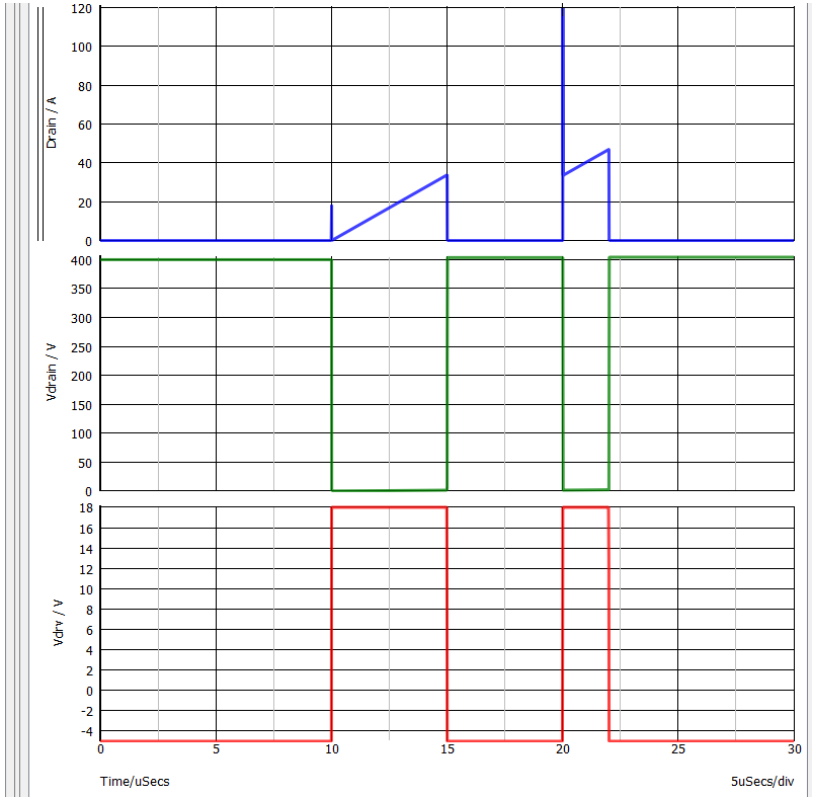


Place the model in the circuit:

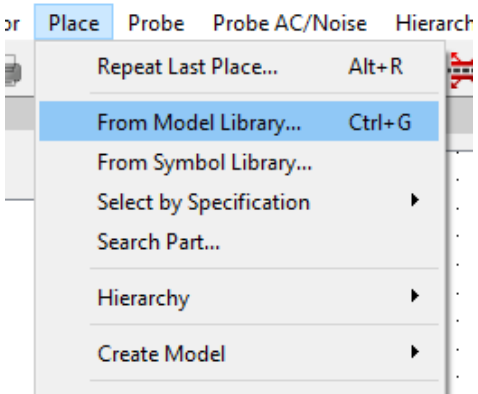


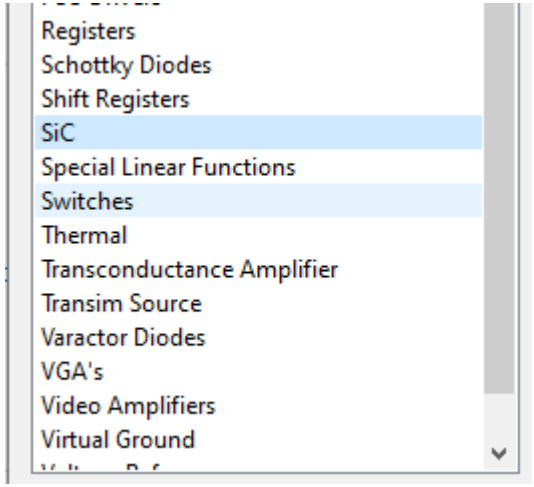
You can now run the simulation

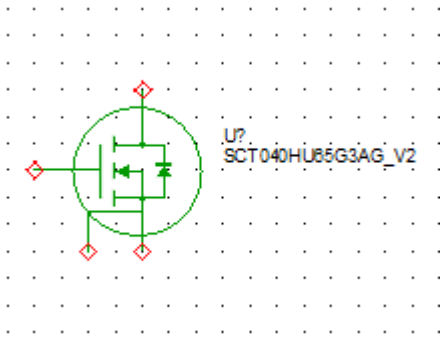




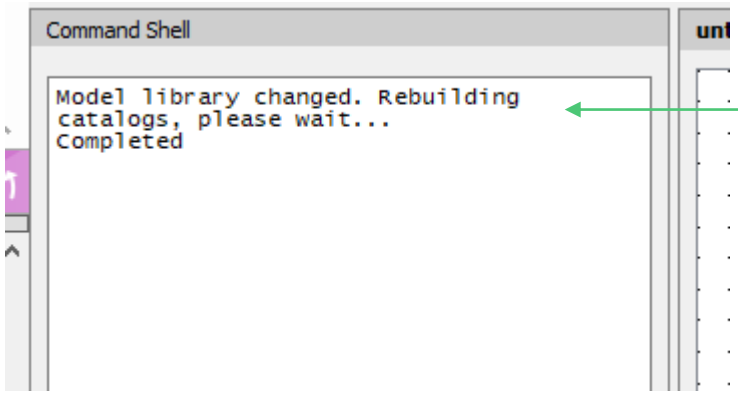
To place a component, go to Place Symbol

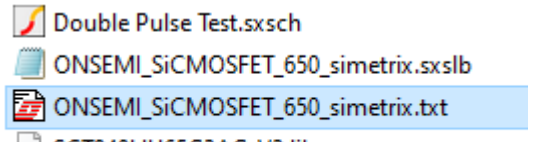


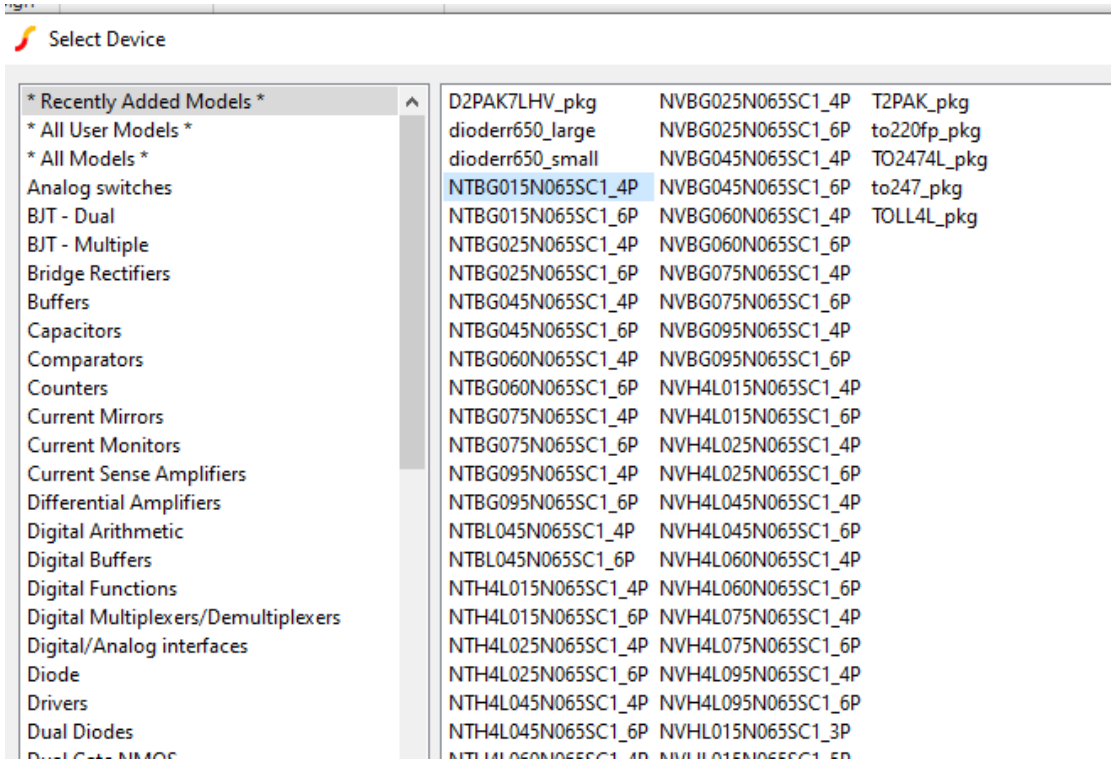




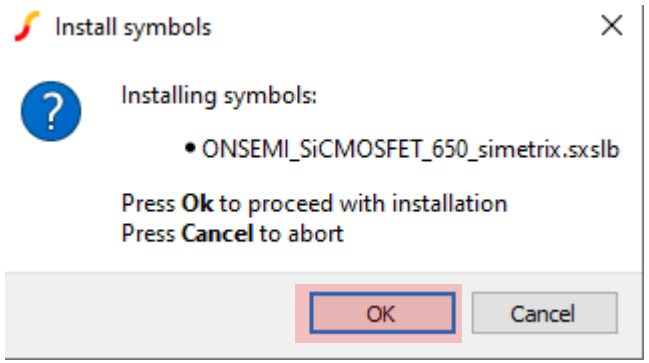
Very often, manufacturers provide models and their associated symbols in case of a complete library. The operation is very similar (you can also see AND9783D). First drop the .LIB containing all the subcircuits in the Command Shell Window:

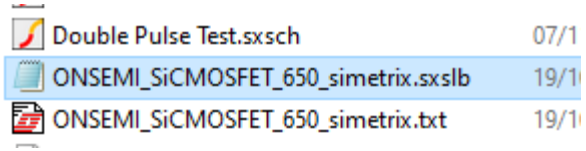






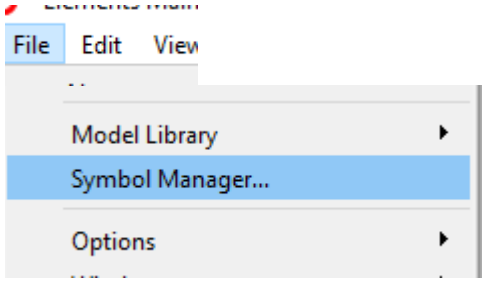
Now, you have to load the associated symbol file:

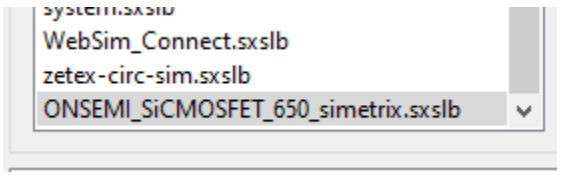


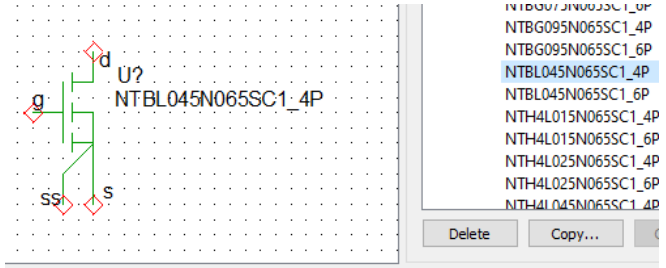


Then CTRL+ G to access the models

You can see the models are properly imported:



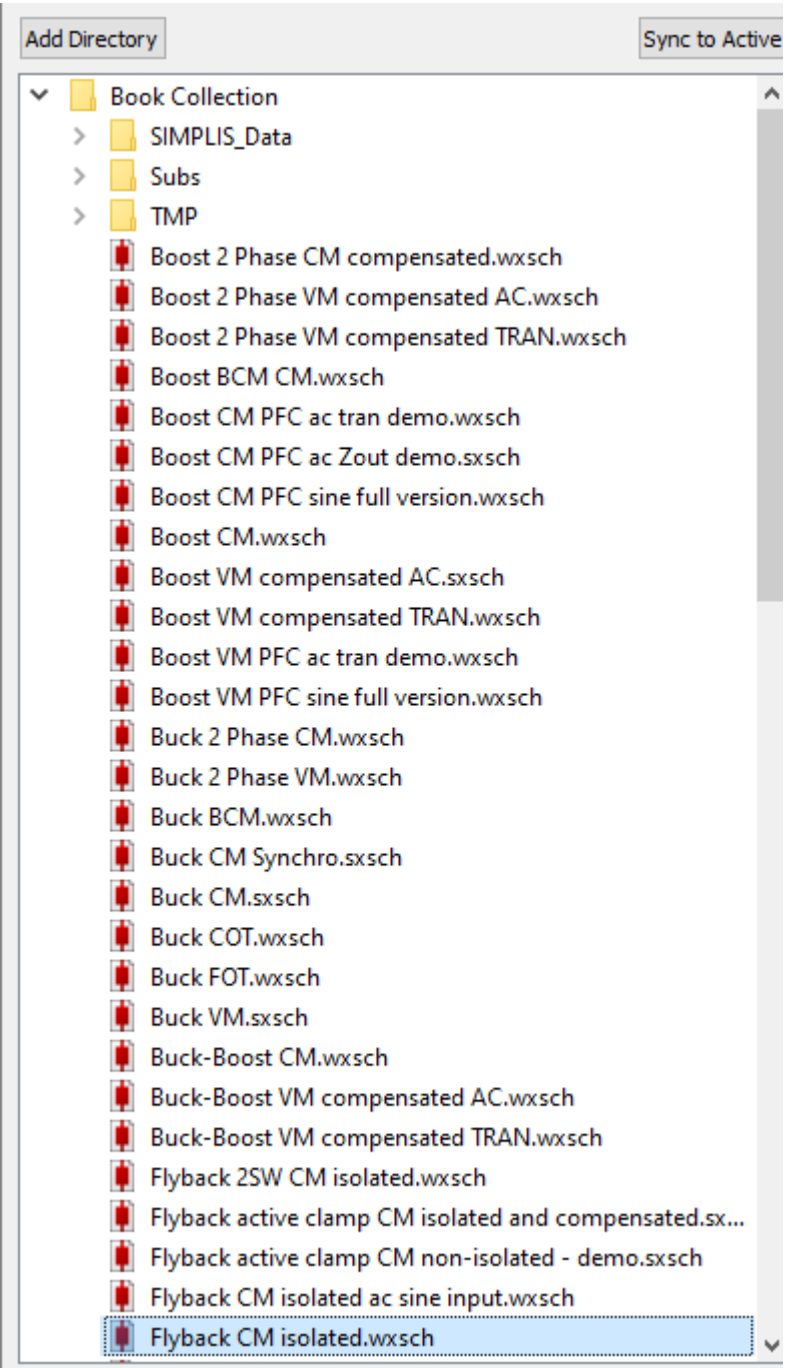


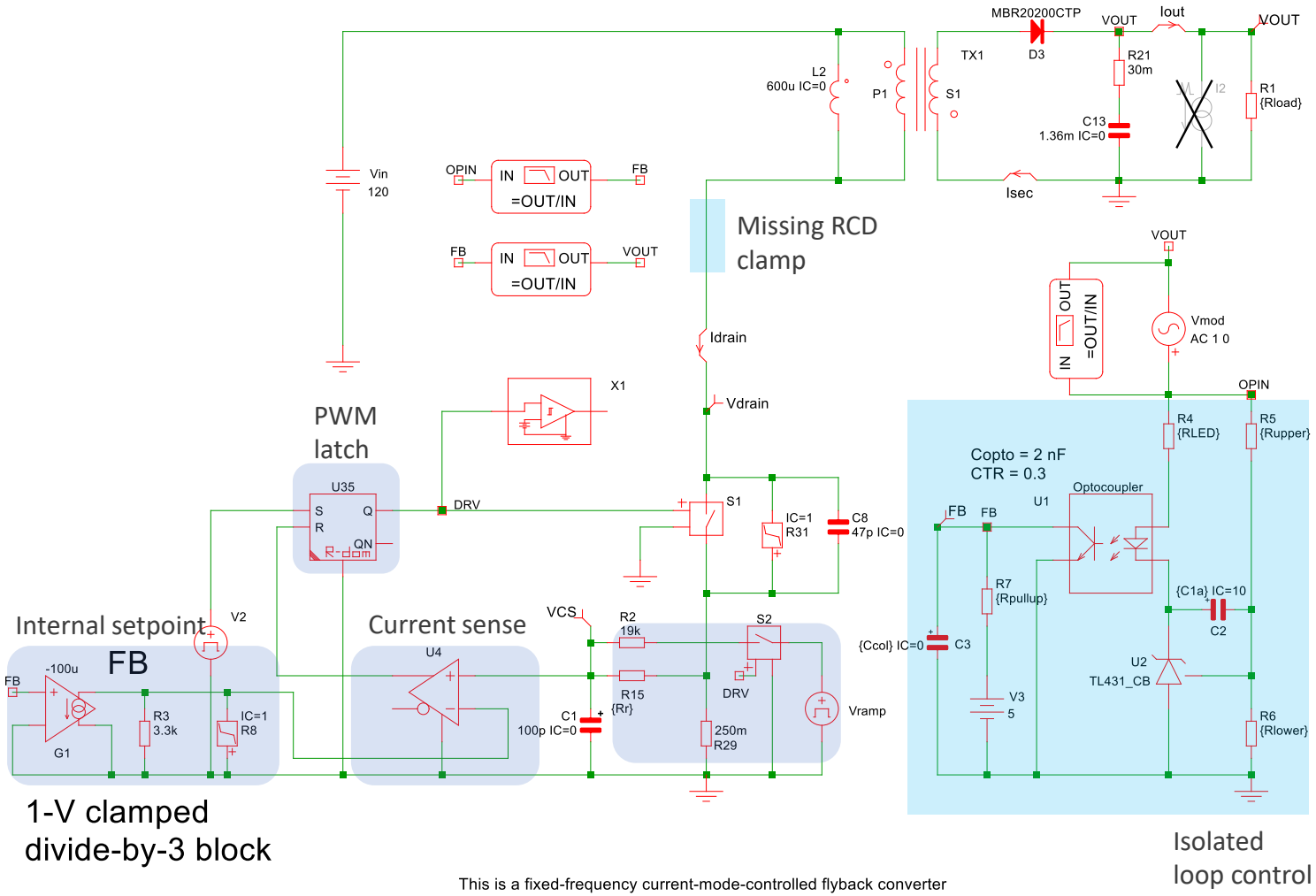


Agenda

- What is Elements?
- Running a Basic Simulation
- A Buck Converter
- Ac Linear Analysis with SIMPLIS
- Importing a SPICE Model
- The Ready-to-Use Template

Open one of the examples – Flyback isolated CM





This is a fixed-frequency current-mode-controlled flyback converter delivering 19 V 3 A from a 120-V source. Enable the 6-ohm load (R1) for ac analysis and disable the PWL source (right-click after selection) to see the transient response. Check Simulator>Edit Netlist (after preprocess) to see the calculated component values.

This is a typical converter for an ac-dc notebook adapter.

- Christophe Basso - Transfer Functions of Switching Converters -

```
*
.VAR Vin=120
.VAR Vout=19
.VAR Lp=600u
.VAR Ri=250m
.VAR N=250m
.VAR Rload=6
.VAR Ts=15u * please update clock and ramp generators *
*
.VAR D=Vout/(Vout+N*Vin) * duty ratio calculation *
.VAR mc=0.818/(1-D) * recommended compensation value for a Q of 1 *
.VAR Sn={(Vin/Lp)*Ri}
.VAR Sramp={2.5/Ts} * 2.5 V over Ts - check your IC specs *
.VAR mc=1.5 * set this value for ramp comp *
.VAR Se={(mc-1)*Sn}
.VAR Rr={(Se/Sramp)*19k+1m}
```

Slope compensation

```
.VAR fRHPZ={((1-D)^2*Rload/(D*Lp*N^2))/(2*pi)}
.VAR fcMAX=0.3*fRHPZ
*
```

RHPZ location for max crossover

```
* Enter values extracted from the plant Bode plot
*
.VAR Gfc=-13 * magnitude at crossover *
.VAR PS=-80 * phase lag at crossover *
*
```

Read the power stage response

```
* Enter Design Goals Information Here *
*
.VAR fc=2k * targeted crossover *
.VAR PM=60 * choose phase margin at crossover *
*
* Enter the Values for Vout and Bridge Bias Current *
*
.VAR Vout=19
.VAR Ibias=250u
.VAR Vref1=2.5
.VAR Rlower=Vref1/Ibias
.VAR Rupper=(Vout-Vref1)/Ibias
```

Your targeted values (2 kHz, 60°)

```
* Optocoupler specifications *
*
.GLOBALVAR Rpullup=20k * check with the selected control chip *
.GLOBALVAR Fopto=6k
.GLOBALVAR Copto=1/(2*pi*Fopto*Rpullup)
.GLOBALVAR CTR=0.33
*
```

Optocoupler characterization

```
.VAR VL=0.2
.VAR VCEsat=0.3
.VAR Vdd=5
.VAR Vf=1
.VAR A=Vout-Vf-VL
.VAR B=Vdd-VCEsat
.VAR Rmax=(A/B)*Rpullup*CTR
*
* Do not edit the below lines *
*
.VAR boost=PM-PS-90
.VAR fp=(tan(boost*pi/180)+sqrt((tan(boost*pi/180))^2+1))*fc
.VAR fz=fc^2/fp
.VAR G=10^(-Gfc/20)
.VAR RLED=CTR*Rpullup/G
.VAR C1a=1/(2*pi*fz*Rupper)
.VAR C2a=1/(2*pi*fp*Rpullup)
.VAR Ccol=C2a-Copto
*
*
```

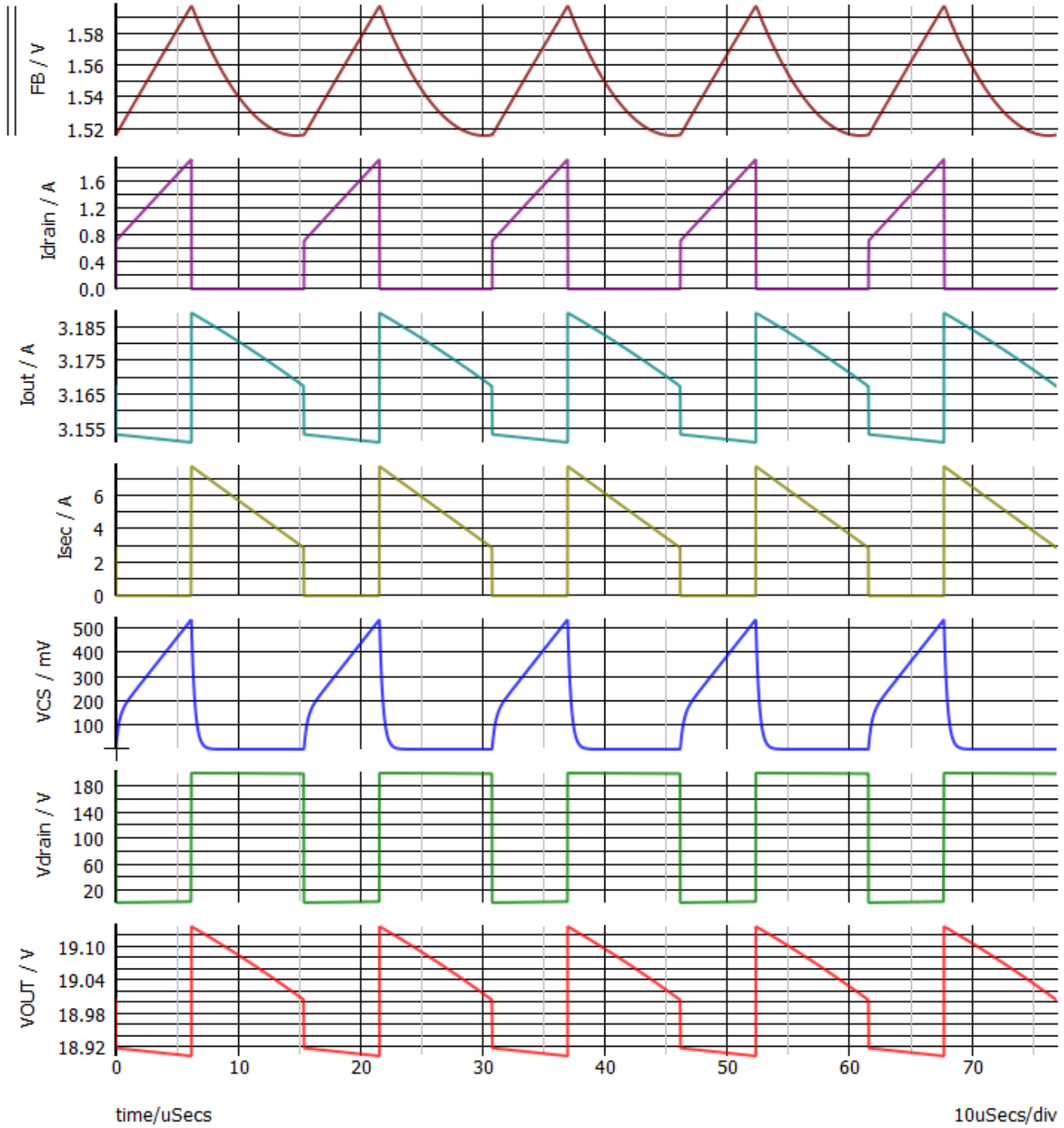
Pole-zero calculation

```
{ '*' }
{ '*' } Rupper = {Rupper}
{ '*' } Rlower = {Rlower}
{ '*' } C2 = {C2a}
{ '*' } C1 = {C1a}
{ '*' } Boost = {boost}
{ '*' } Fz = {Fz}
{ '*' } Fp = {Fp}
{ '*' } Sn = {Sn}
{ '*' } Se = {Se}
{ '*' } D = {D}
{ '*' } Mc = {mc}
{ '*' } Rramp = {Rr}
{ '*' } Rmax = {Rmax}
{ '*' } FRHPZ = {fRHPZ}
{ '*' } FcMAX = {fcMAX}
{ '*' }
```

Print components values

Most of the ready-made templates come with a script compensating the loop

Press F9 to run the simulation and make sure the operating point is correct: V_{out} = 19 V/3 A √



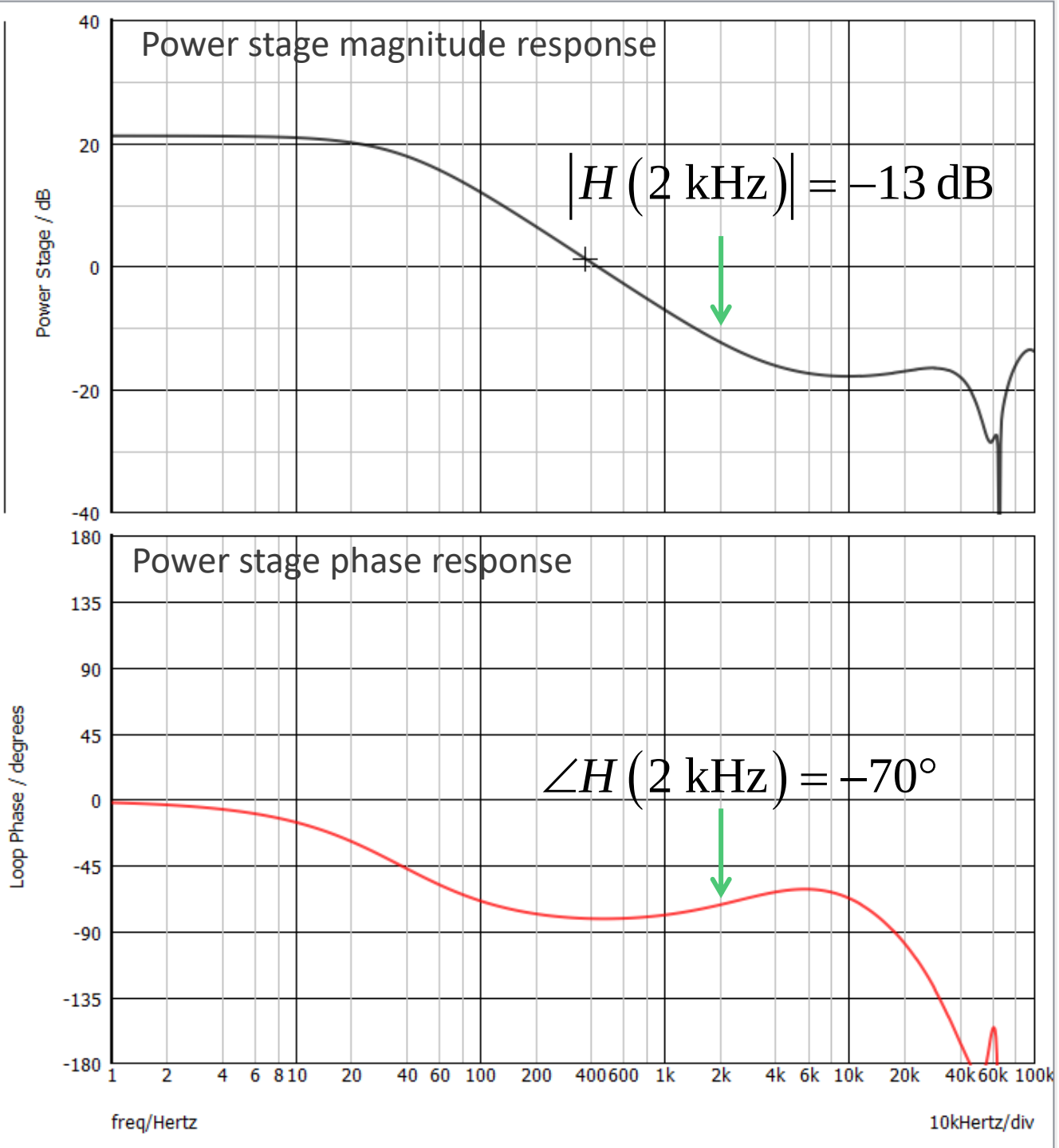
Feedback voltage within the 3-V dynamic

Current in the main switch – check rms current

Dc output current

Secondary-side current – check peaks and rms content

Add leakage inductance and RCD clamp (full version)



Now check the control-to-output transfer function of the power stage. Check the RHPZ value and limit the crossover frequency to 30-20% of this zero position:

```
*
* Rupper = 66000
* Rlower = 10000
* C2 = 1.44819154804453e-09
* C1 = 3.31268645711794e-09
* Boost = 50
* Fz = 727.940468532405
* Fp = 5494.95483890924
* Sn = 50000
* Se = 25000
* D = 0.387755102040816
* Mc = 1.5
* Rramp = 2850.001
* Rmax = 24995.7446808511
* FRHPZ = 24616.8762677045
* FcMAX = 7385.06288031136
*
```

$f_{c,\max} \approx 7\text{ kHz}$

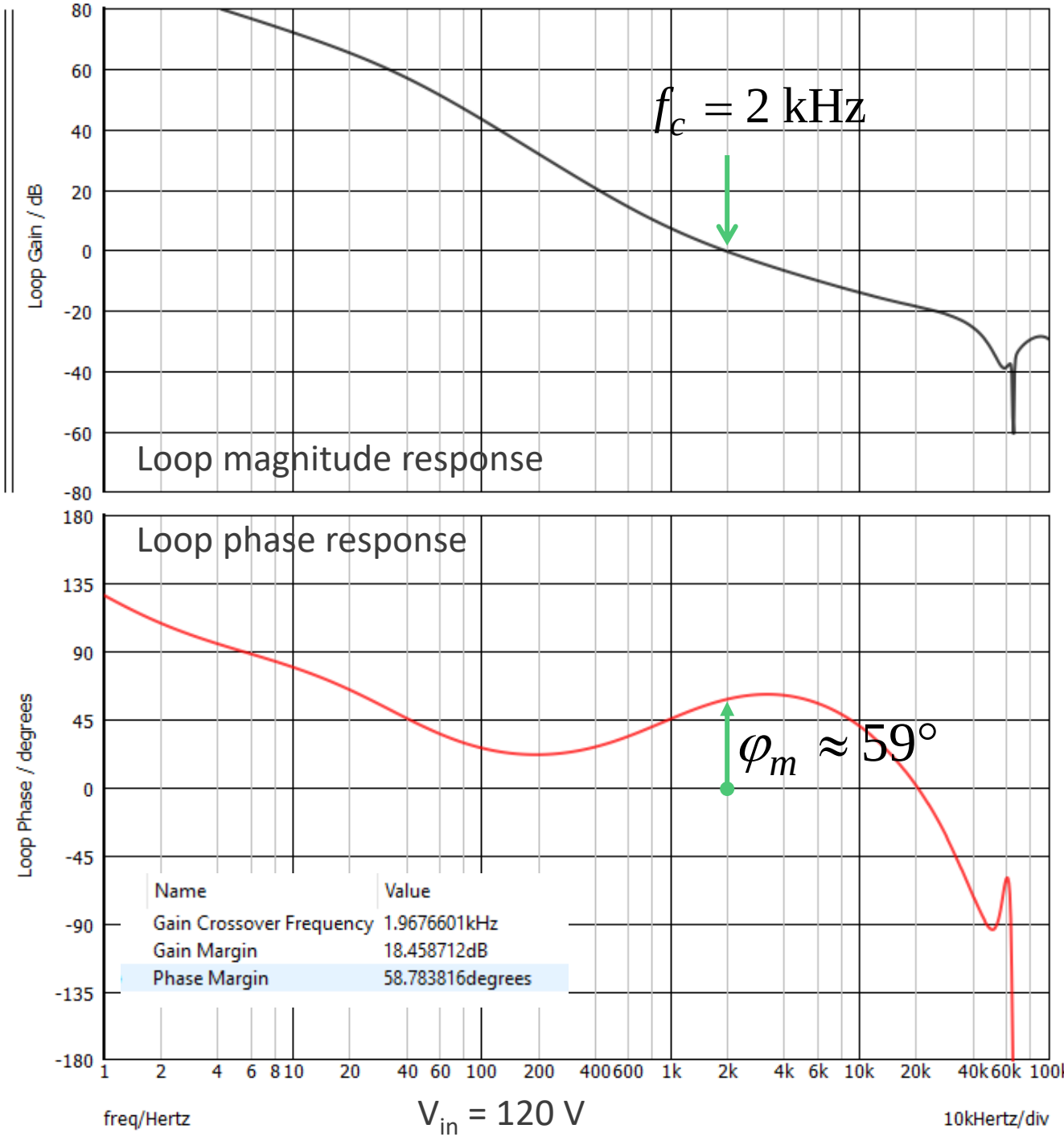
Consider the 6-kHz opto pole

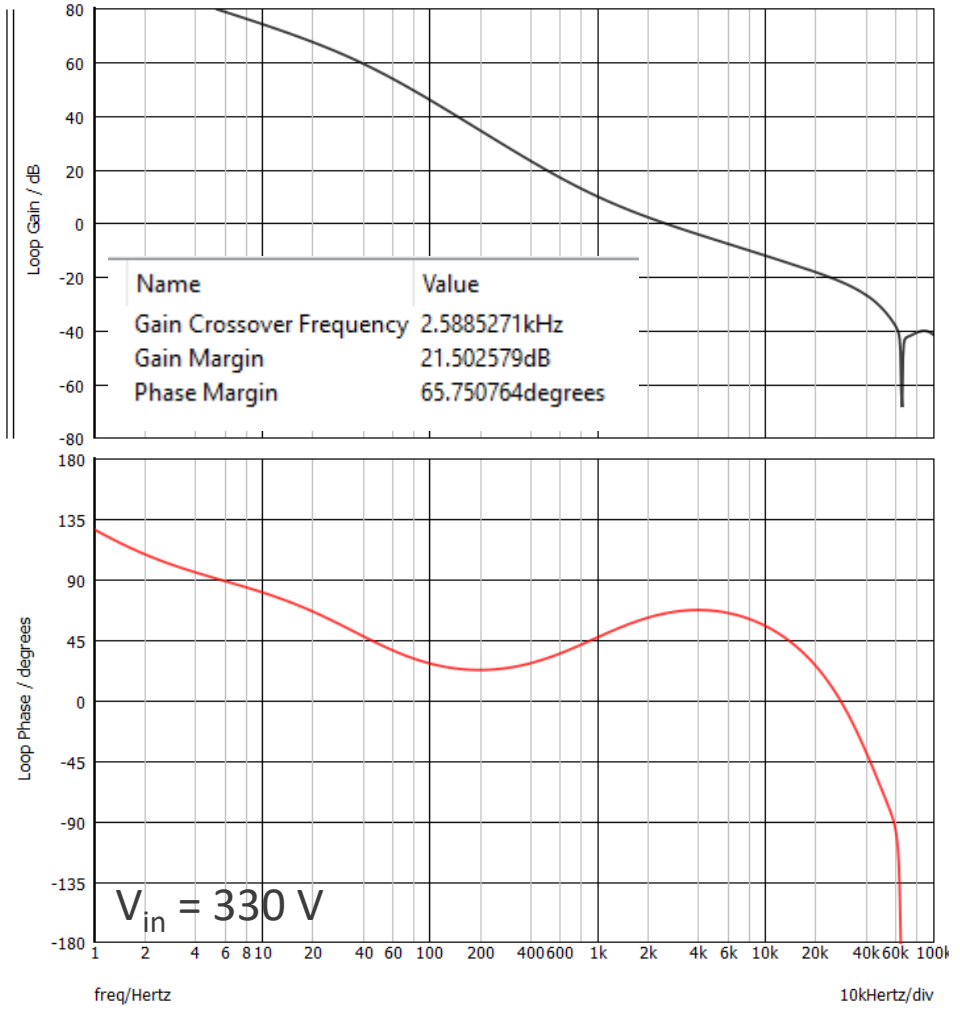
Choose a 2-kHz crossover
Read the magnitude plot a 2 kHz ≈ -13 dB
Read the phase plot at 5 kHz ≈ -70°

```
* Enter values extracted from the plant Bode plot
*
.VAR Gfc=-13 * magnitude at crossover *
.VAR PS=-70 * phase lag at crossover *
*
* Enter Design Goals Information Here *
*
.VAR fc=2k * targetted crossover *
.VAR PM=60 * choose phase margin at crossover *
*
```

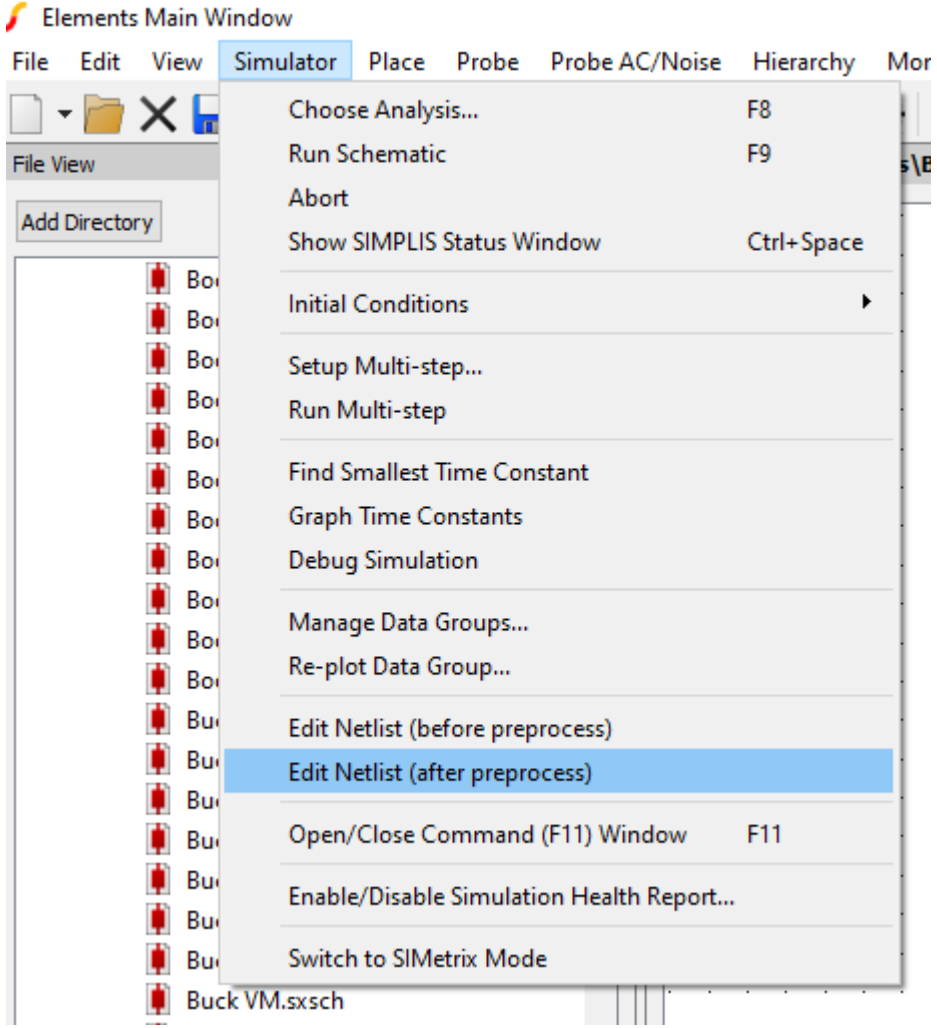
Fill out the requirements

The crossover frequency and phase margin are off target for this low-line operation. You can now increase the input voltage and check how stability evolves at this voltage. Same with the load, make it change and assess the stability at different points.





Once the simulation is run, you can unveil the values computed by SIMPLIS:

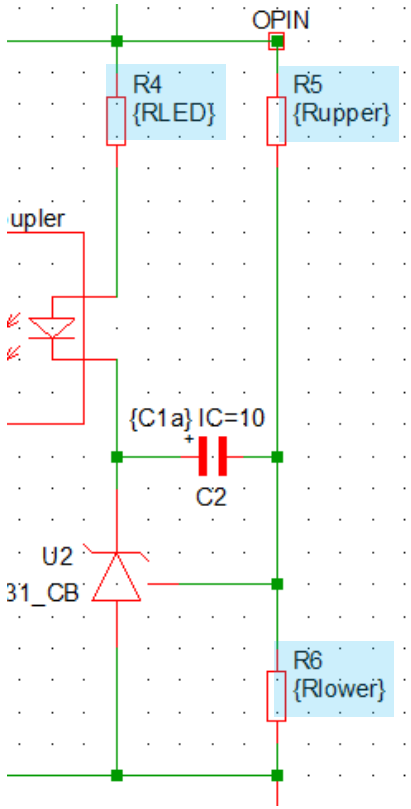



```
.print  ALL
.options PSP_START=0 PSP_NPT=100001 POP_I
+ MIN_AVG_TOPOLOGY_DUR=1a AVG_TOPOLOGY_DU
.pop  TRIG_GATE=X1.!D_CYCLE  TRIG_COND=0_
+  TD_RUN_AFTER_POP_FAILS=-1
*
* Rupper = 66000
* Rlower = 10000
* C2 = 1.85537921992434e-09
* C1 = 2.58567330980208e-09
* Boost = 40
* Fz = 932.615316309997
* Fp = 4289.01384101912
* Sn = 50000
* Se = 25000
* D = 0.387755102040816
* Mc = 1.5
* Rramp = 2850.001
* Rmax = 24995.7446808511
* FRHPZ = 24616.8762677045
* FcMAX = 7385.06288031136
*
C1 33 0 100p IC=0
C13 20 0 1.36m IC=0
C2 32 27 2.58567330980208e-09 IC=10
C3 41 0 5.29088027491884e-10 IC=0
C8 24 31 47p IC=0
X$D3 18 39 DIODE_SPICE_V2$1
G1 37 0 41 0 -100u
```

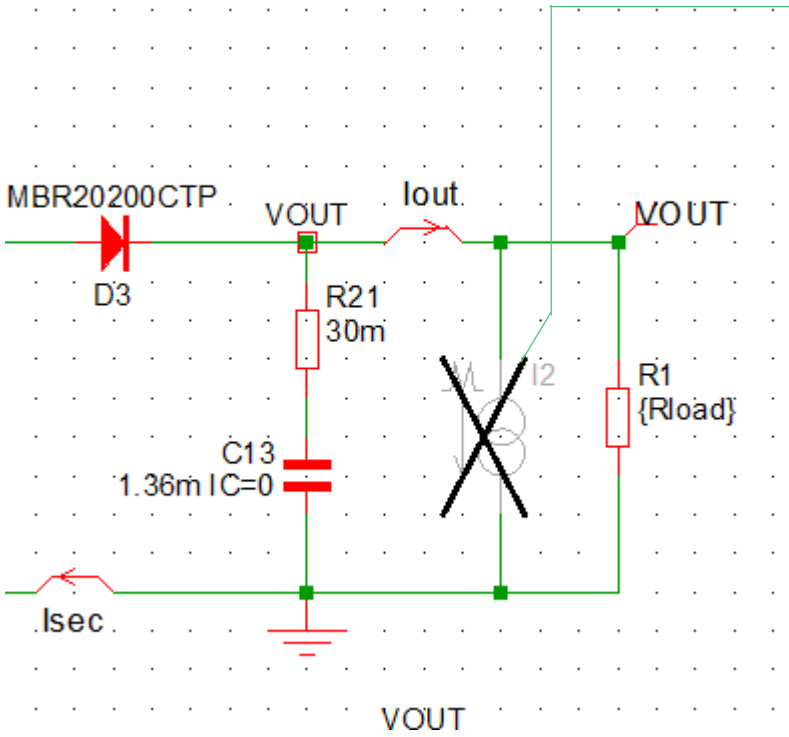
You can add more computed values

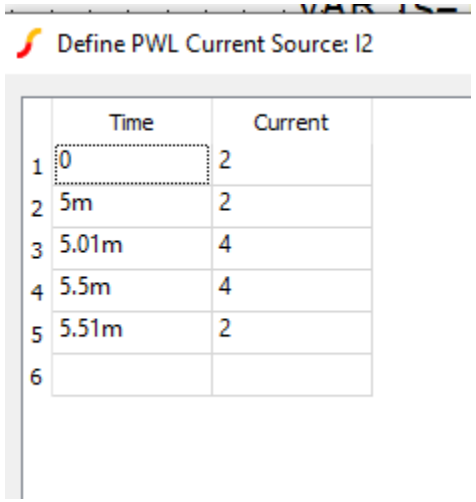
{ '*' } Rupper = {Rupper}
{ '*' } Rlower = {Rlower}
{ '*' } C2 = {C2a}
{ '*' } C1 = {C1a}
{ '*' } Boost = {boost}

Value passed to the component



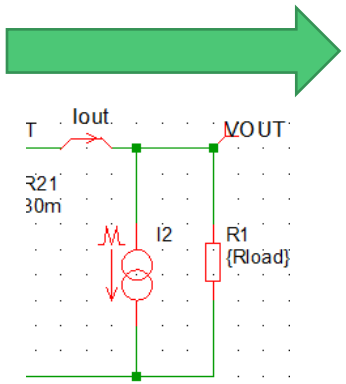
Test the transient response

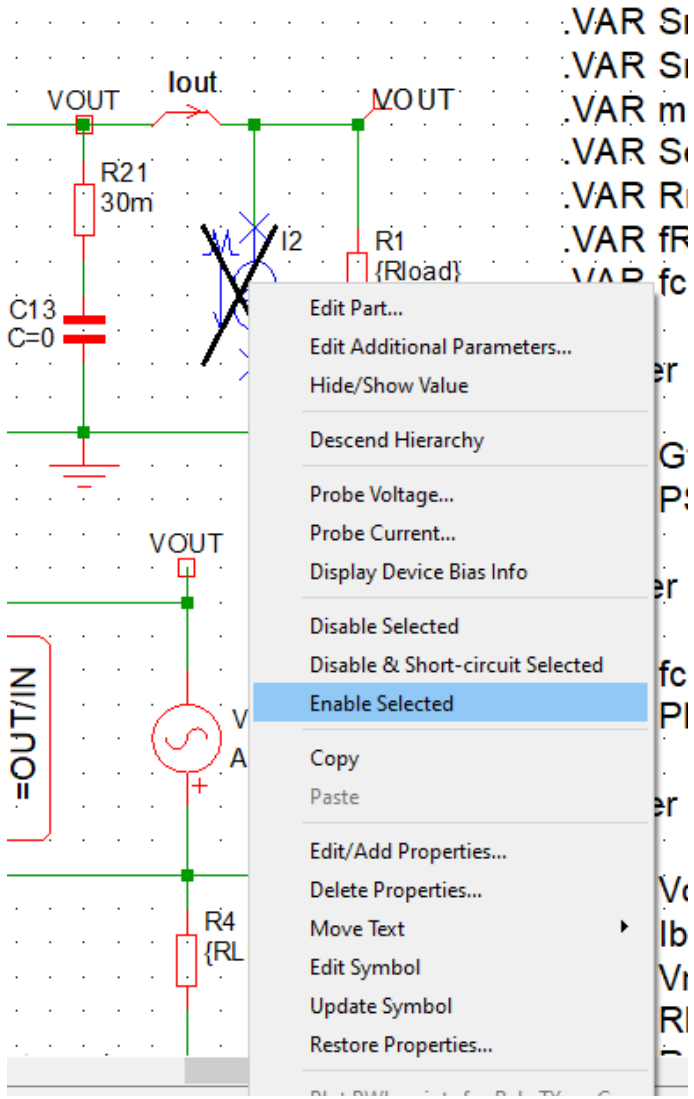




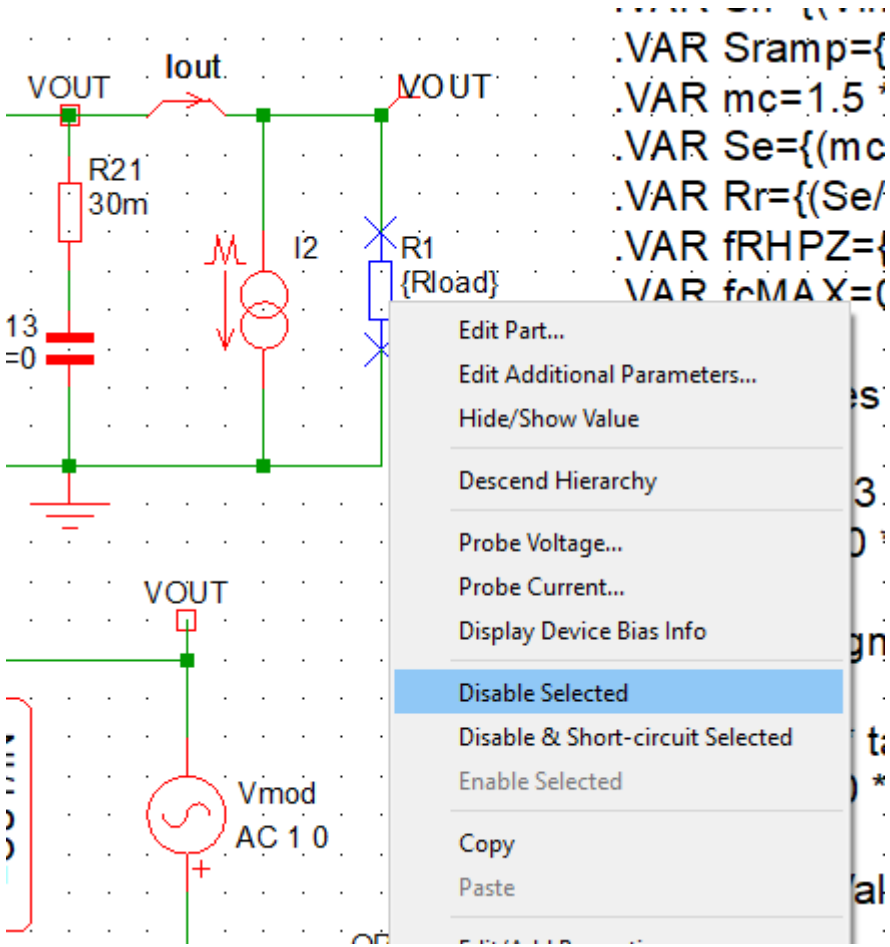
The X means the component is disabled
To activate it, right-click on the part:

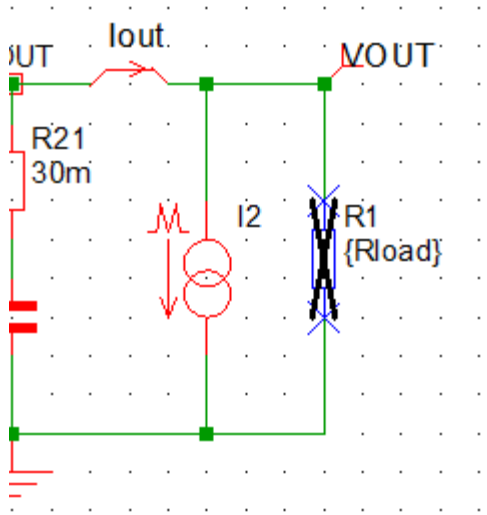
The cross disappears



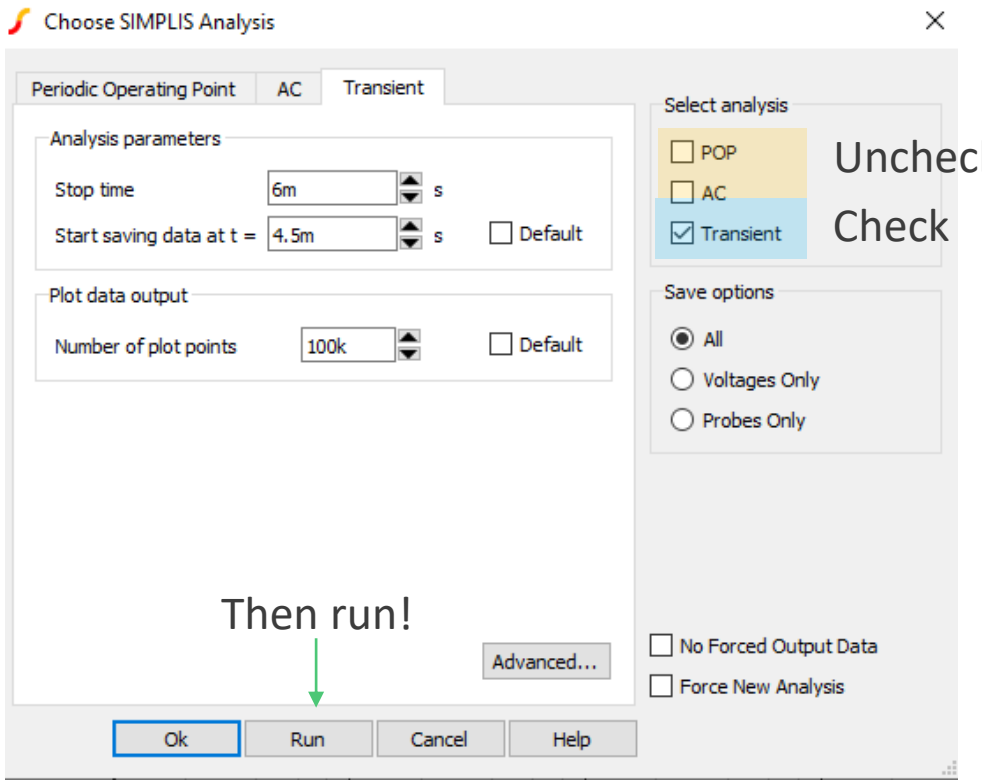


Now disable the resistor, right-click and disable it

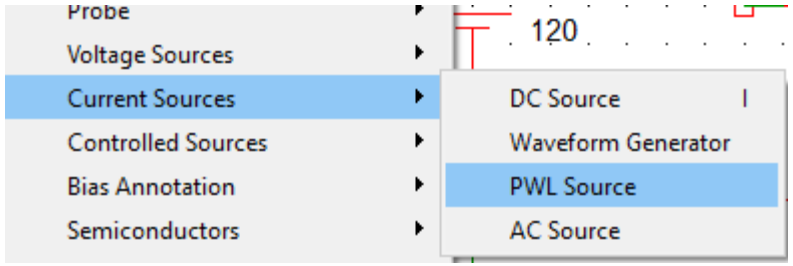


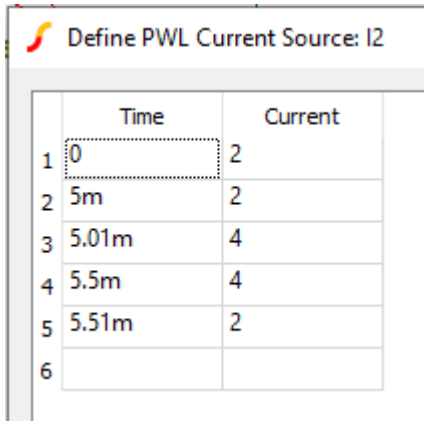


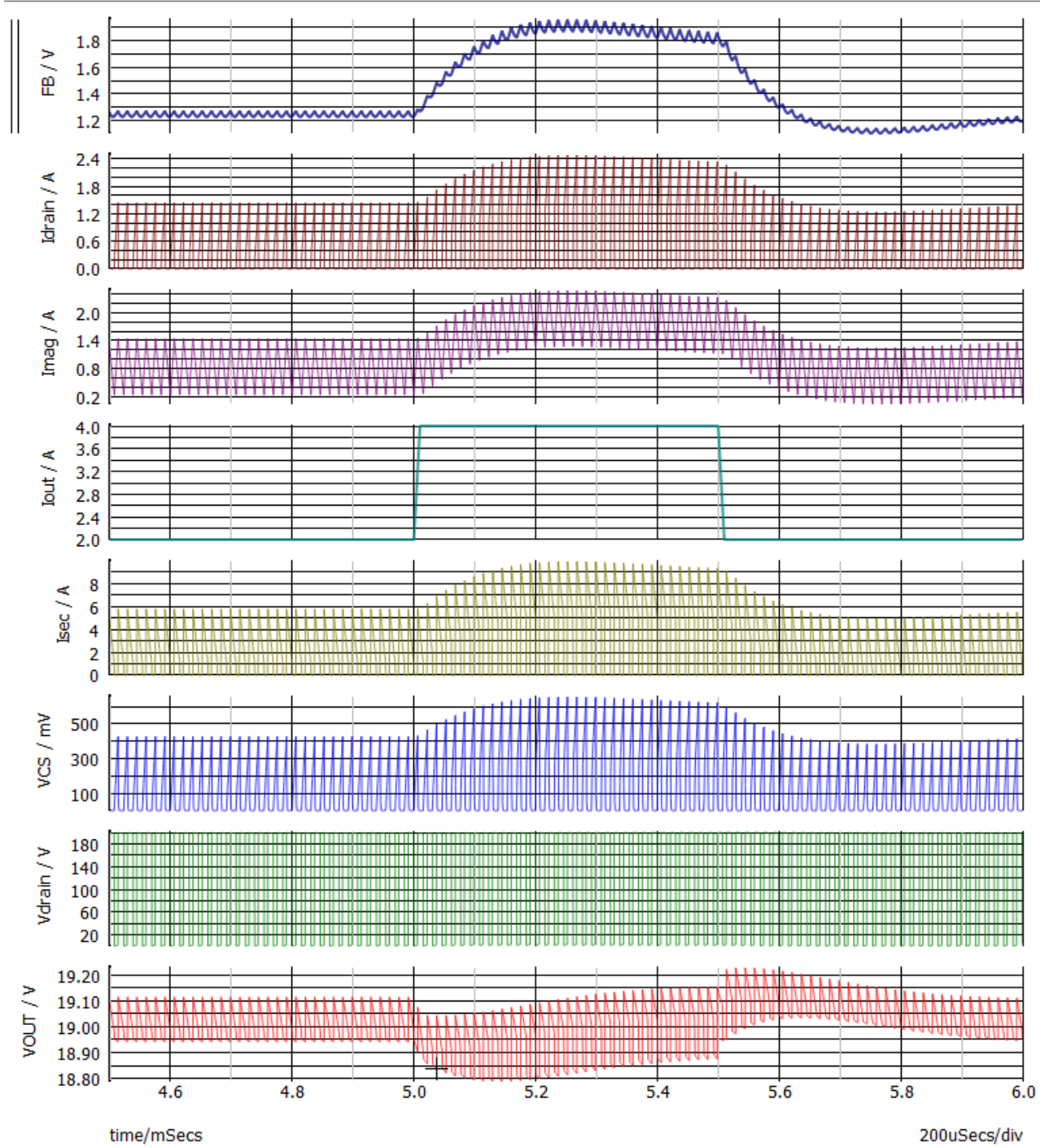
Then press F8 to access the simulation setup



Place



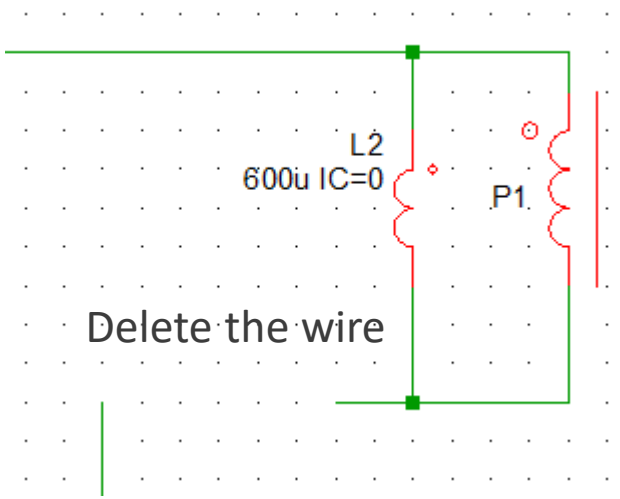


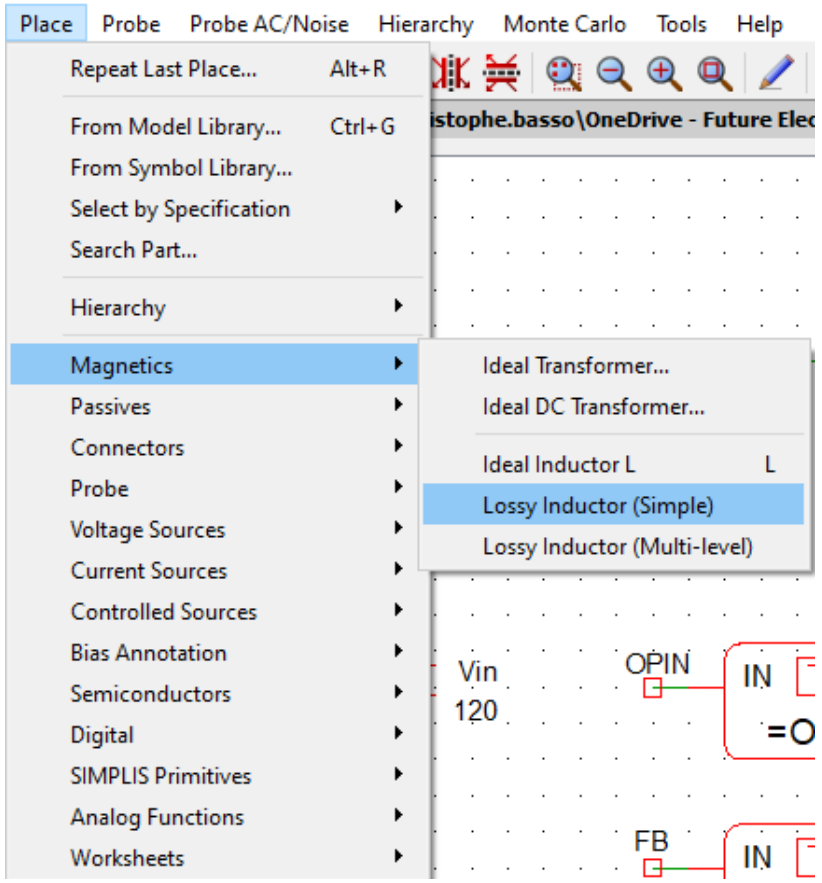


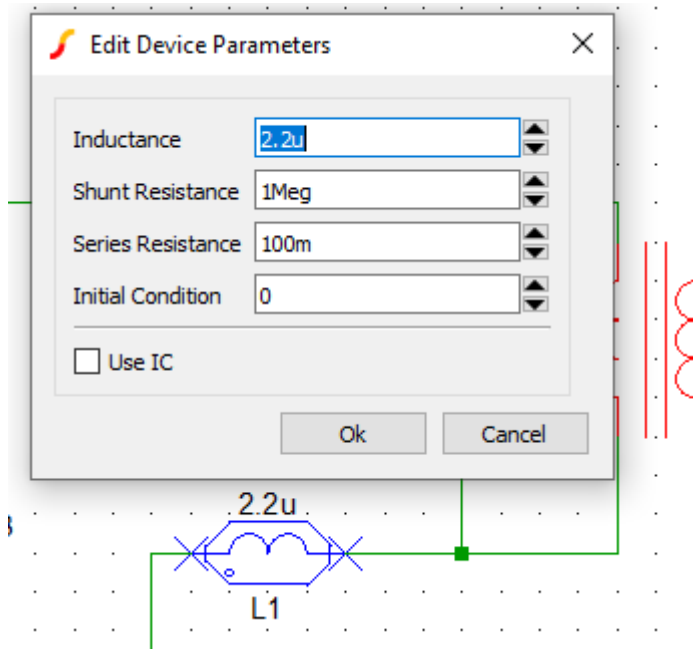
$v_{out}(t)$

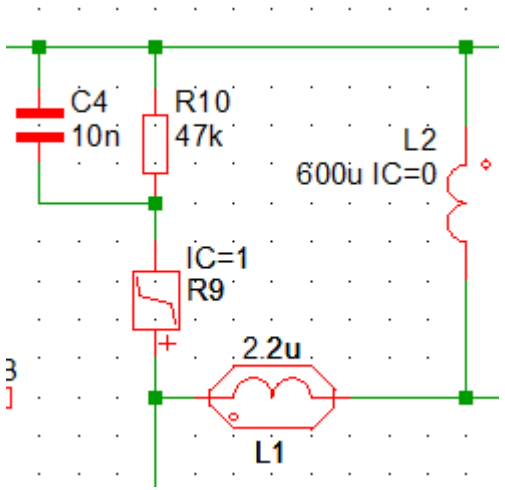
V_{in} = 120 V

We can now try to add a RCD clamping network

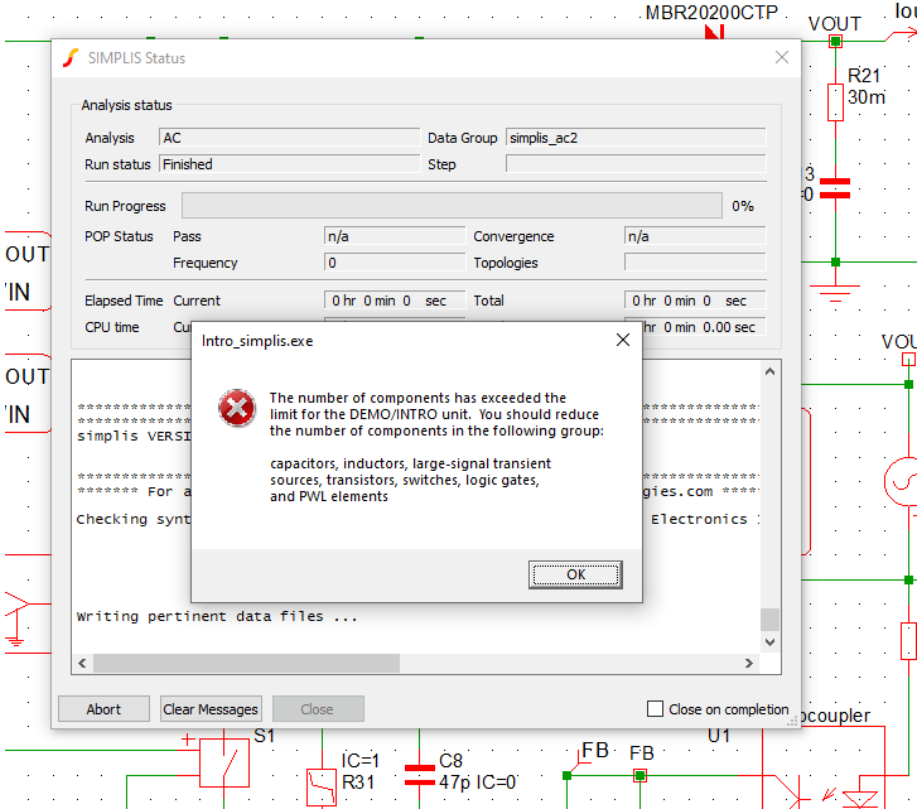




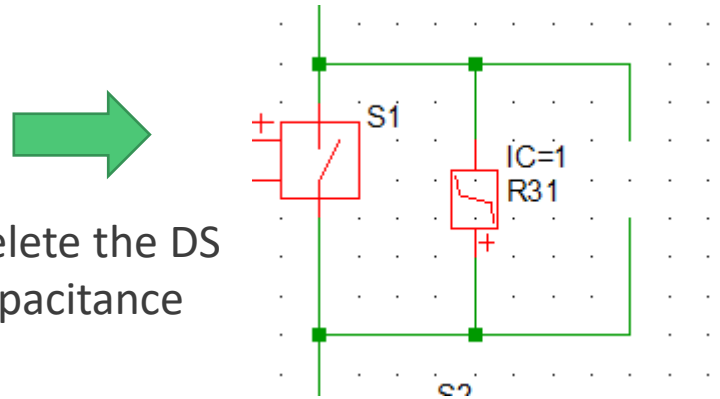




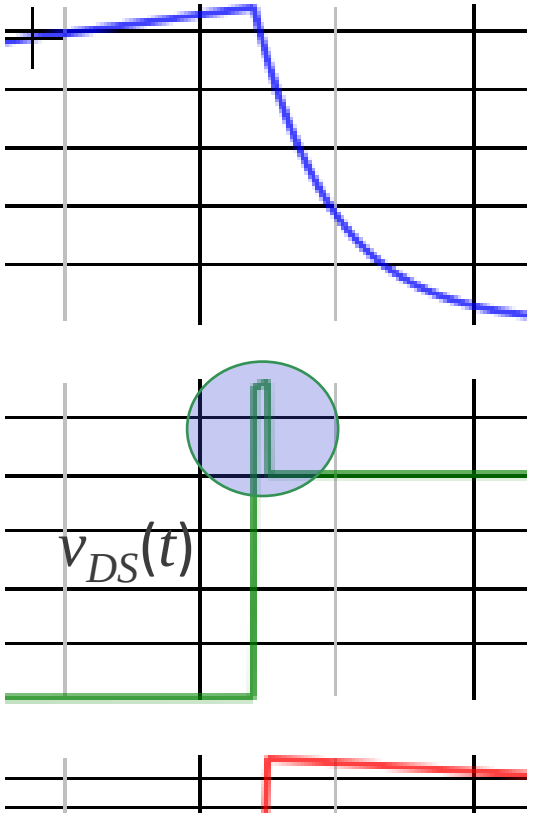
The lossy inductor offers more robust convergence by including parasitics



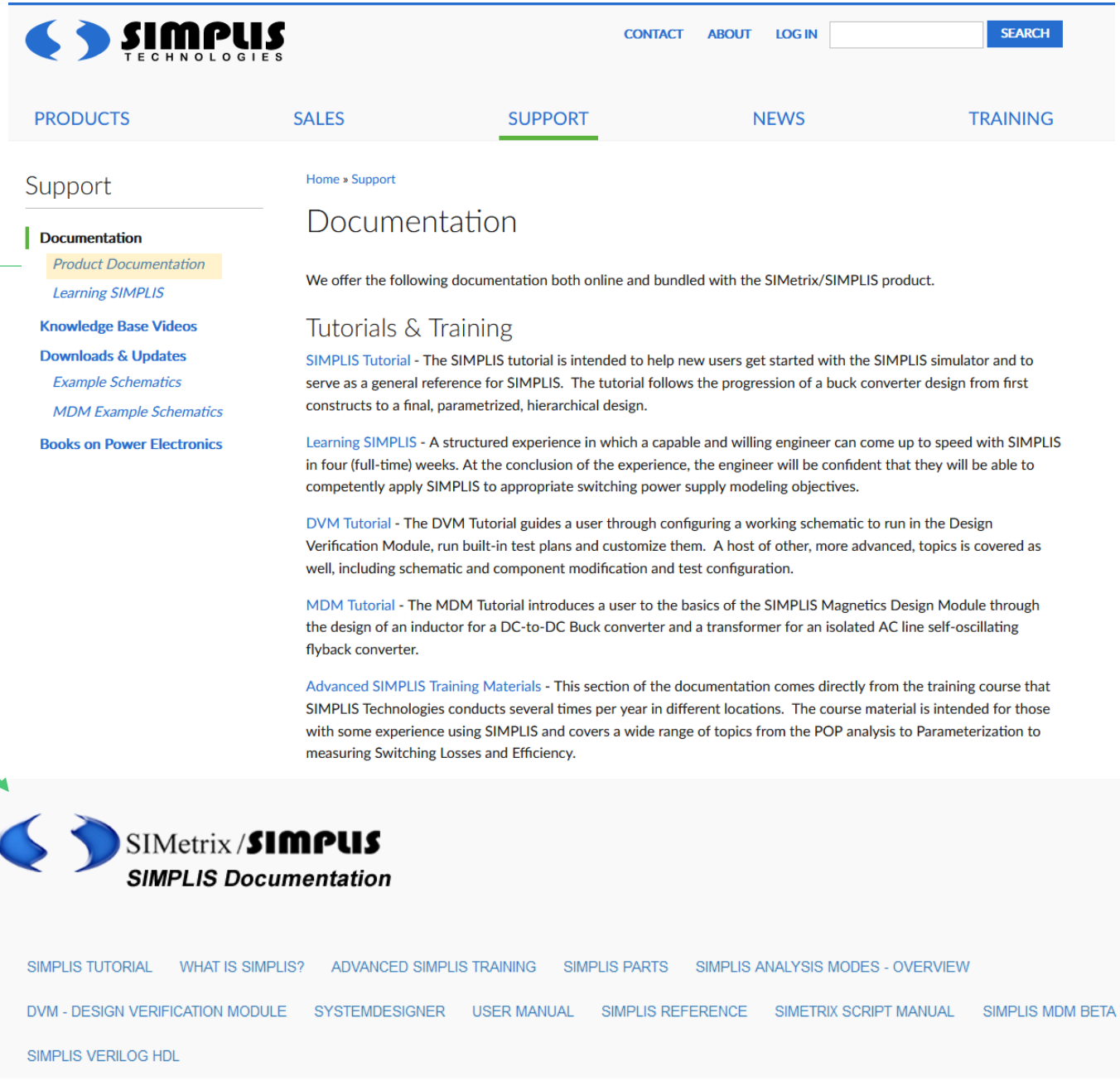
If you run the simulation, you now exceed the demo size limit.



Delete the DS capacitance



The drain-source voltage now shows the spike linked to the leakage inductance.



The documentation database of SIMPLIS is a truly comprehensive source of information on the program with many examples, tutorials and videos.

https://www.simplistechnologies.com/support/documentation