



A Tutorial Introduction to Elements

The Free SIMPLIS Demonstration Version

Christophe Basso

Business Development Manager

IEEE Senior Member

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

The screenshot displays the SIMPLIS Elements interface. On the left is the **File View** pane with a tree structure of project files. The main workspace shows a **Schematic capture** of a boost converter circuit. The circuit includes an 8V DC source (V_{in}), a 100mΩ resistor (R21), a 47μH inductor (L1), a MOSFET (S2) driven by a pulse generator (Qdrv), an ideal diode (D1), a 50mΩ resistor (R1), a 220μF capacitor (C13), a 15Ω load resistor (R3), and a feedback network with a 1KΩ resistor (R2) and a 100pF capacitor (C2). The feedback loop is controlled by an op-amp (U4) with a gain of 333m, and a reference is provided by an AC source (U5) and a voltage source (V6).

On the right, the **Waveforms viewer** displays three plots over a 50μs period:

- IL / A**: Inductor current showing a triangular ripple between approximately 1.3A and 1.7A.
- Qdrv / V**: MOSFET gate drive showing a square wave between 0V and 9V.
- VOUT / V**: Output voltage showing a triangular ripple between approximately 13.31V and 13.39V.

At the bottom right, the **Legend** and **Curve label** table are visible:

Label	Legend	Curve label	Name	Value
IL	[Yellow line]			
Qdrv	[Blue line]			
VOUT	[Green line]			
Vsw	[Red line]			

File viewer

Command shell

Schematic capture

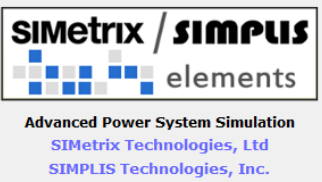
Waveforms viewer

Tabs

Legend

Dock un-dock

- Close
- Close tab group
- Close other tabs
- Close all but this
- Dock to...
- Undock
- Undock all
- Copy full path
- Open containing folder



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 \(~109MB\)](#) 

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

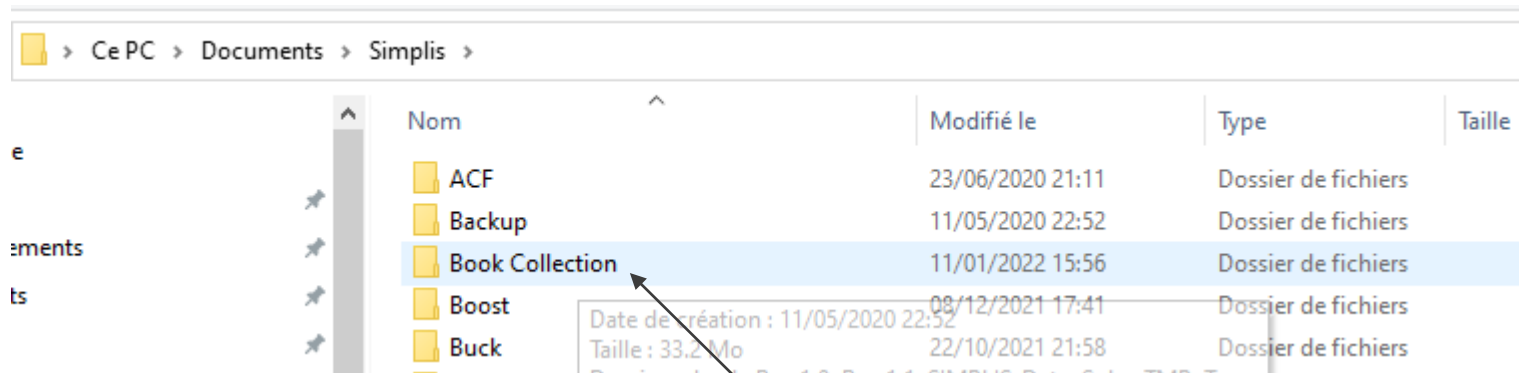
SIMetrix/SIMPLIS Elements requires Windows 10 **64 bit edition** (Home, Pro or Enterprise).

SIMetrix/SIMPLIS Elements Limitations

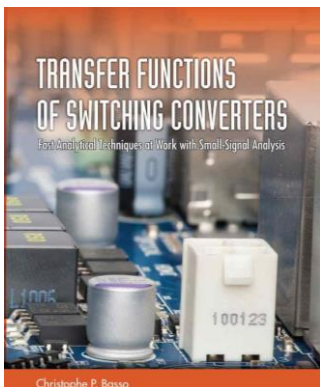
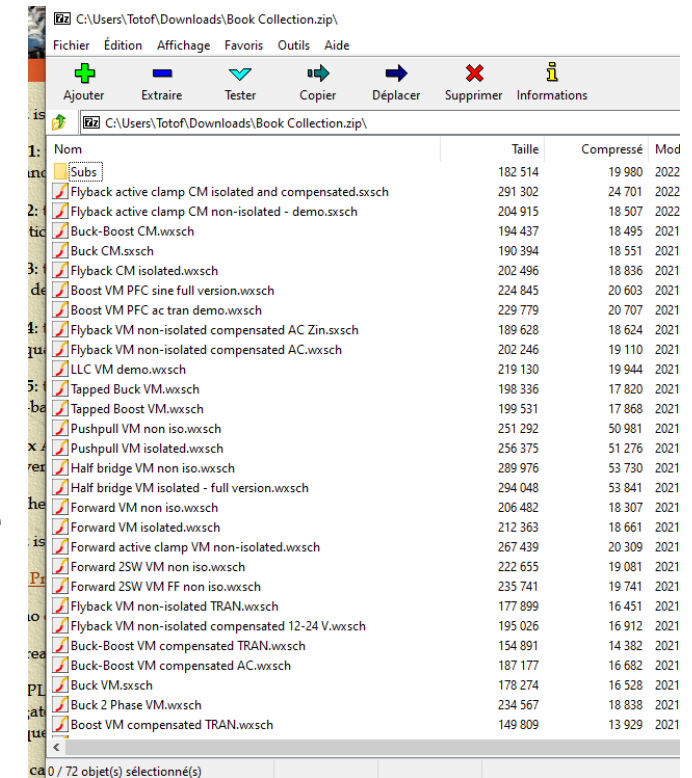
Then go to my webpage and download the ready-made templates:

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

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

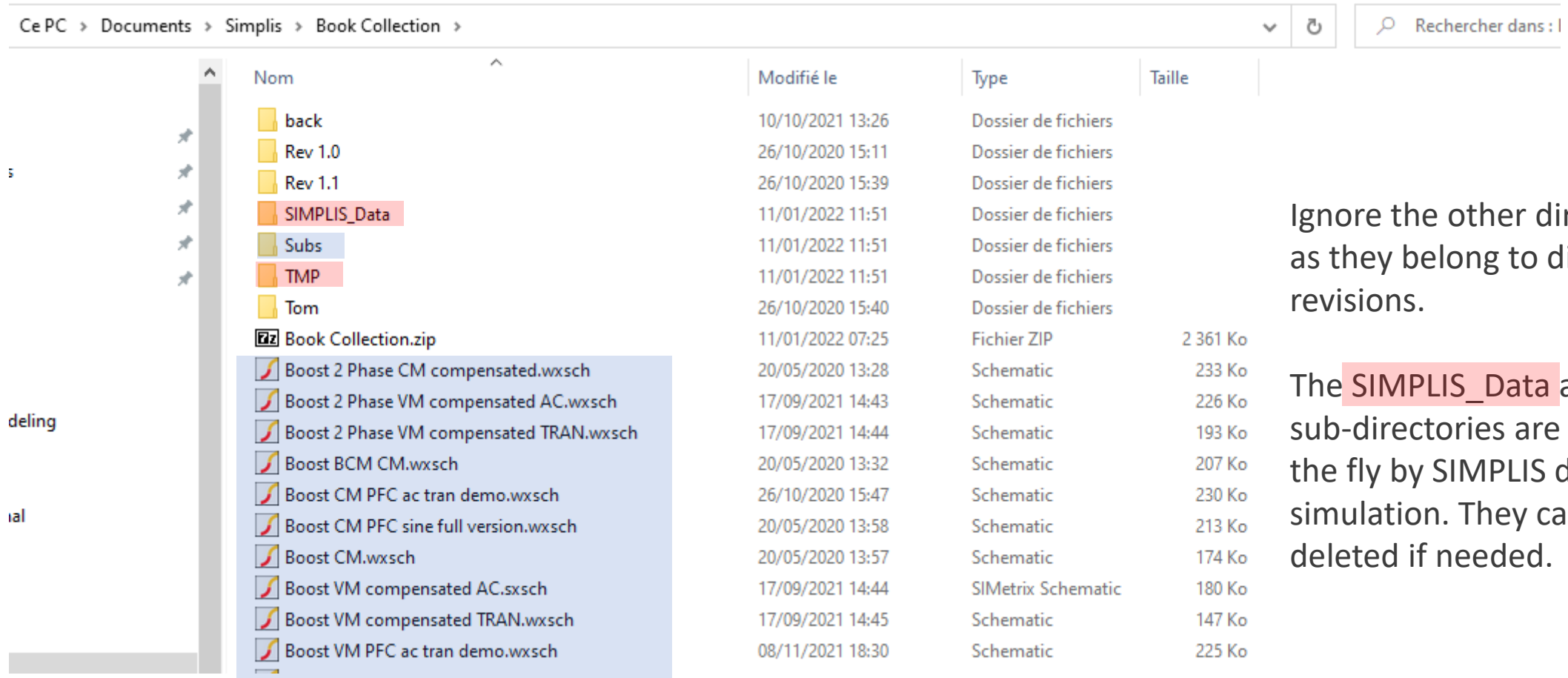


Extract the ZIP here



These files are examples I documented in my last book
Transfer Functions of Switching Converters – [Faraday Press](#)

Once installed, you have this structure:



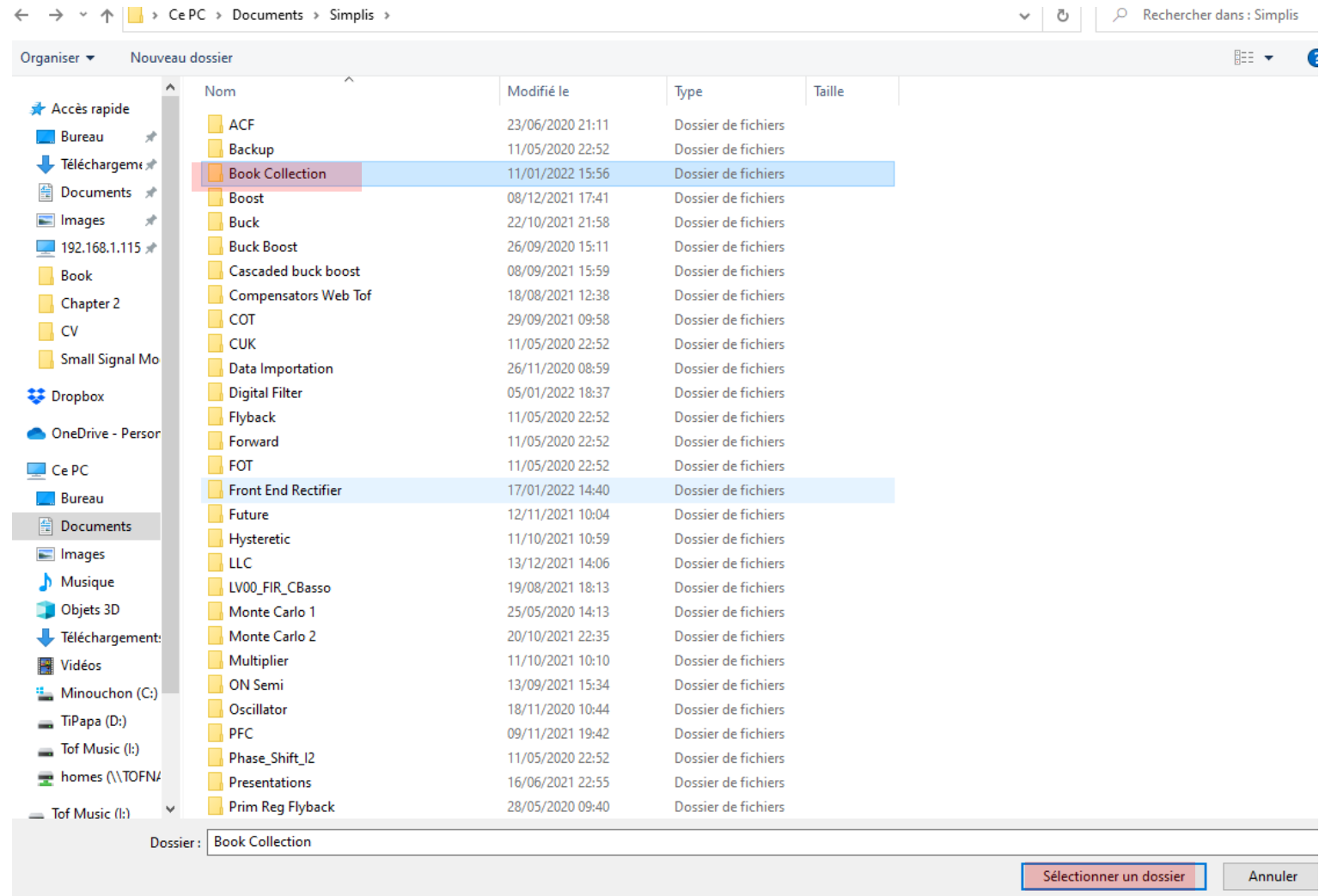
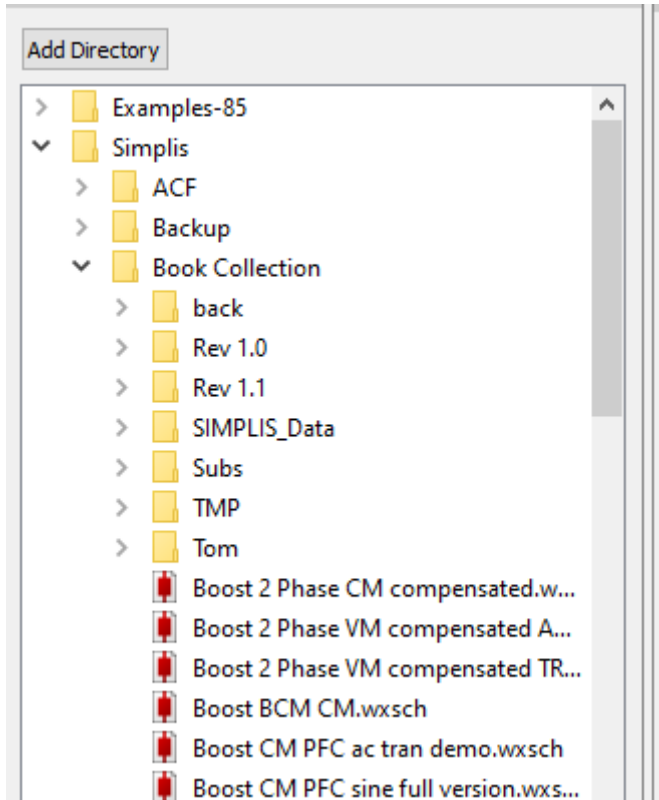
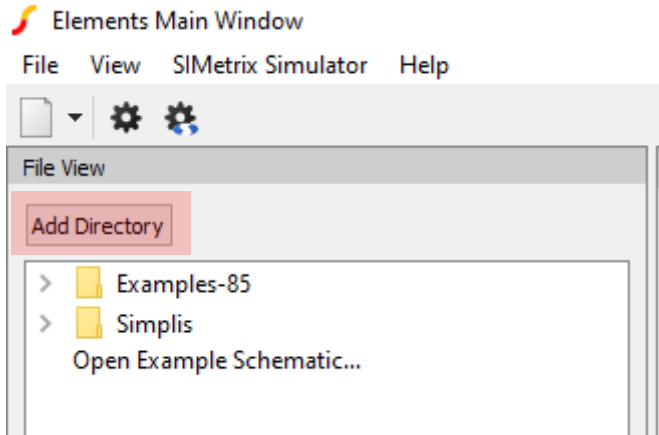
Nom	Modifié le	Type	Taille
back	10/10/2021 13:26	Dossier de fichiers	
Rev 1.0	26/10/2020 15:11	Dossier de fichiers	
Rev 1.1	26/10/2020 15:39	Dossier de fichiers	
SIMPLIS_Data	11/01/2022 11:51	Dossier de fichiers	
Subs	11/01/2022 11:51	Dossier de fichiers	
TMP	11/01/2022 11:51	Dossier de fichiers	
Tom	26/10/2020 15:40	Dossier de fichiers	
Book Collection.zip	11/01/2022 07:25	Fichier ZIP	2 361 Ko
Boost 2 Phase CM compensated.wxsch	20/05/2020 13:28	Schematic	233 Ko
Boost 2 Phase VM compensated AC.wxsch	17/09/2021 14:43	Schematic	226 Ko
Boost 2 Phase VM compensated TRAN.wxsch	17/09/2021 14:44	Schematic	193 Ko
Boost BCM CM.wxsch	20/05/2020 13:32	Schematic	207 Ko
Boost CM PFC ac tran demo.wxsch	26/10/2020 15:47	Schematic	230 Ko
Boost CM PFC sine full version.wxsch	20/05/2020 13:58	Schematic	213 Ko
Boost CM.wxsch	20/05/2020 13:57	Schematic	174 Ko
Boost VM compensated AC.sxsch	17/09/2021 14:44	SIMatrix Schematic	180 Ko
Boost VM compensated TRAN.wxsch	17/09/2021 14:45	Schematic	147 Ko
Boost VM PFC ac tran demo.wxsch	08/11/2021 18:30	Schematic	225 Ko

Ignore the other directories as they belong to different revisions.

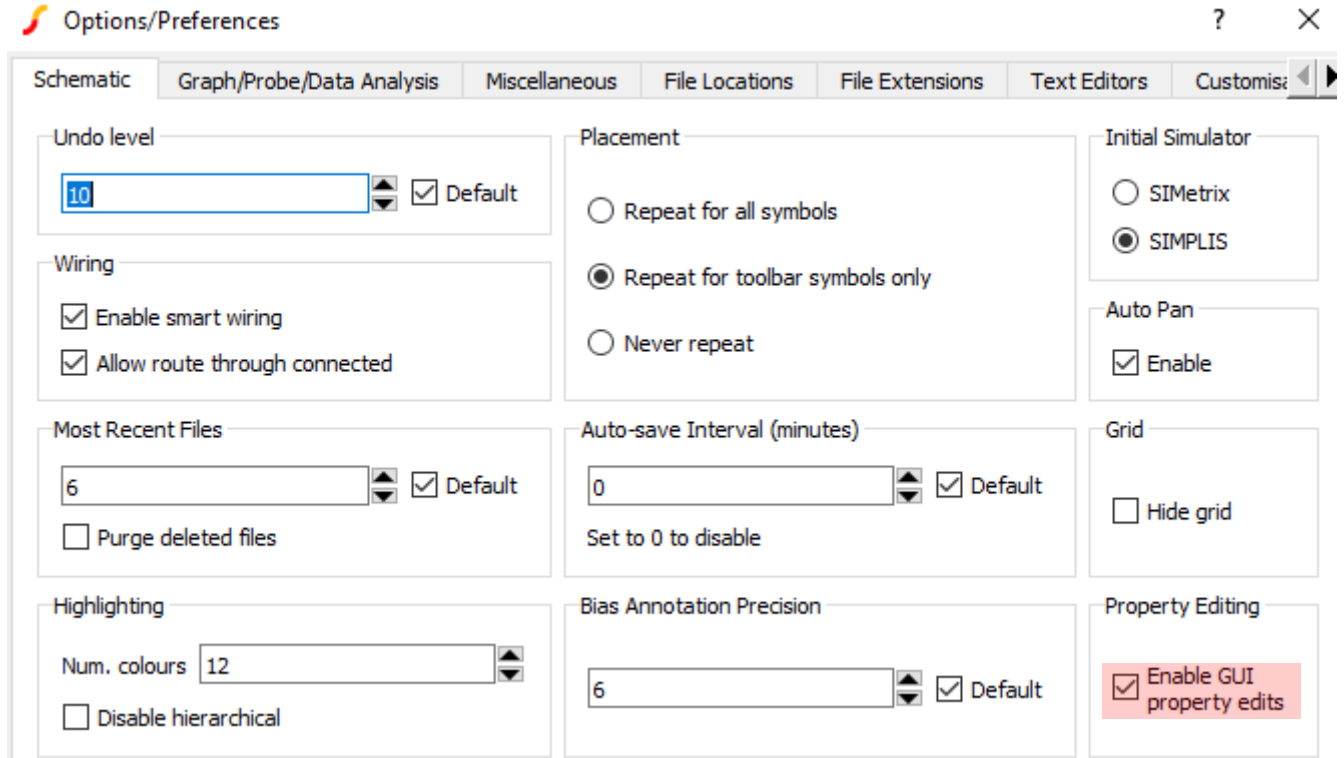
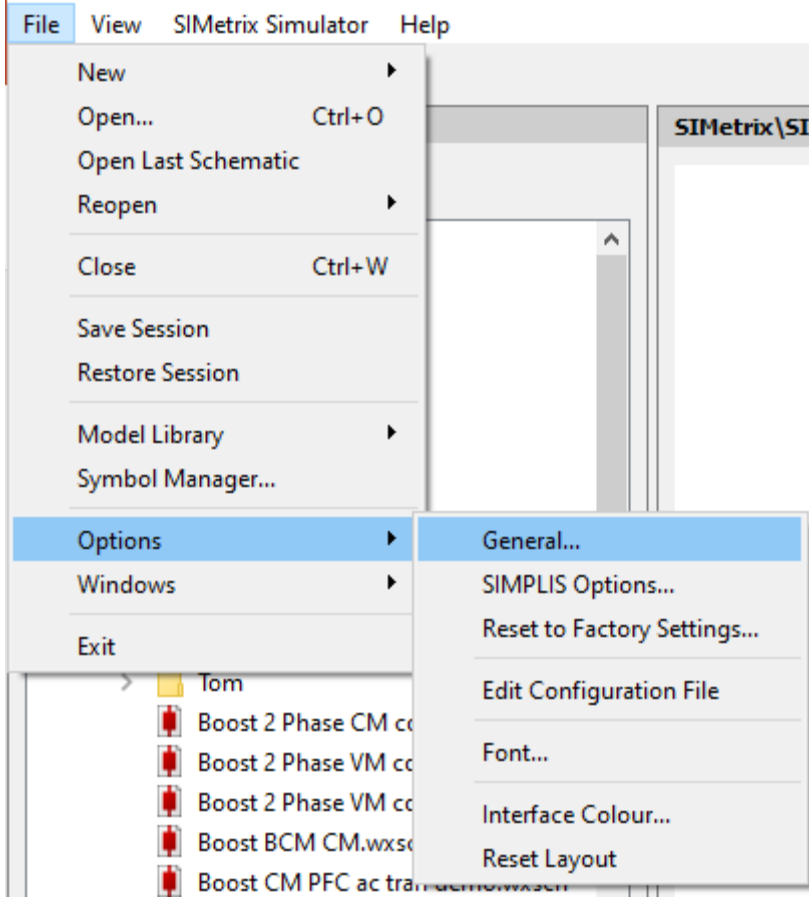
The **SIMPLIS_Data** and **TMP** sub-directories are filled on the fly by SIMPLIS during simulation. They can safely be deleted if needed.

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.

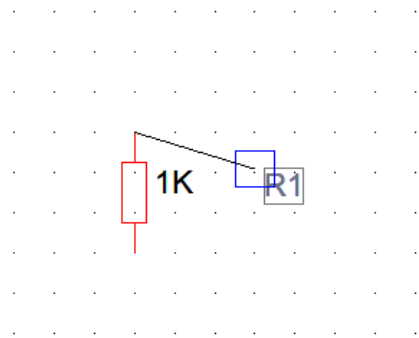
Once SIMPLIS Elements is installed, instruct the file viewer with the directory location:



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

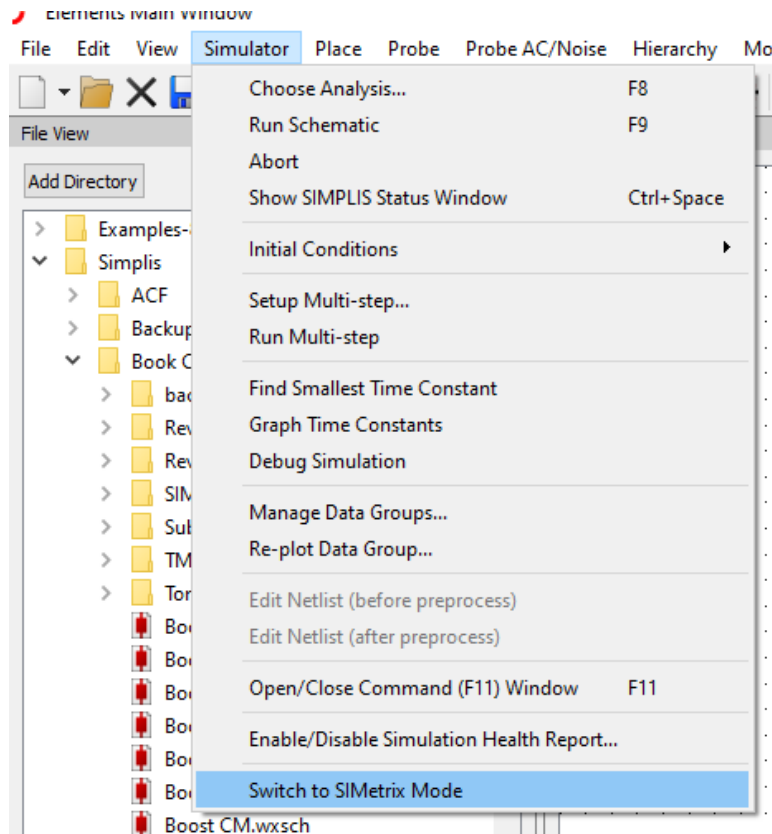


Edit property and
move/align labels



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

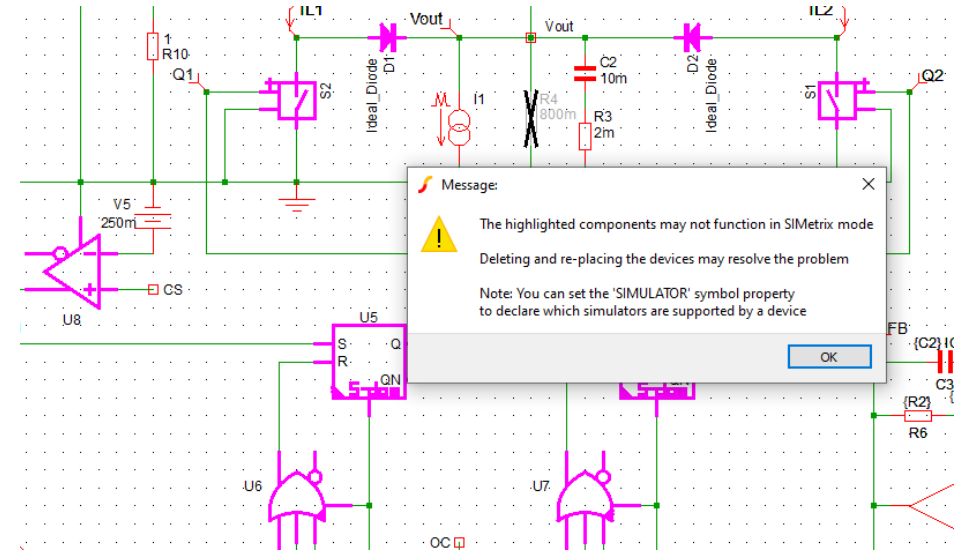
1. SIMetrix is a classical SPICE-based program and is compatible with many SPICE engines. It can perform ac and transient analyses as any SPICE package would do.
2. SIMPLIS is exclusively a time-domain simulator which can also simulate in ac and transient modes. SIMPLIS can extract the ac response from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



You can switch between the two engines at any time



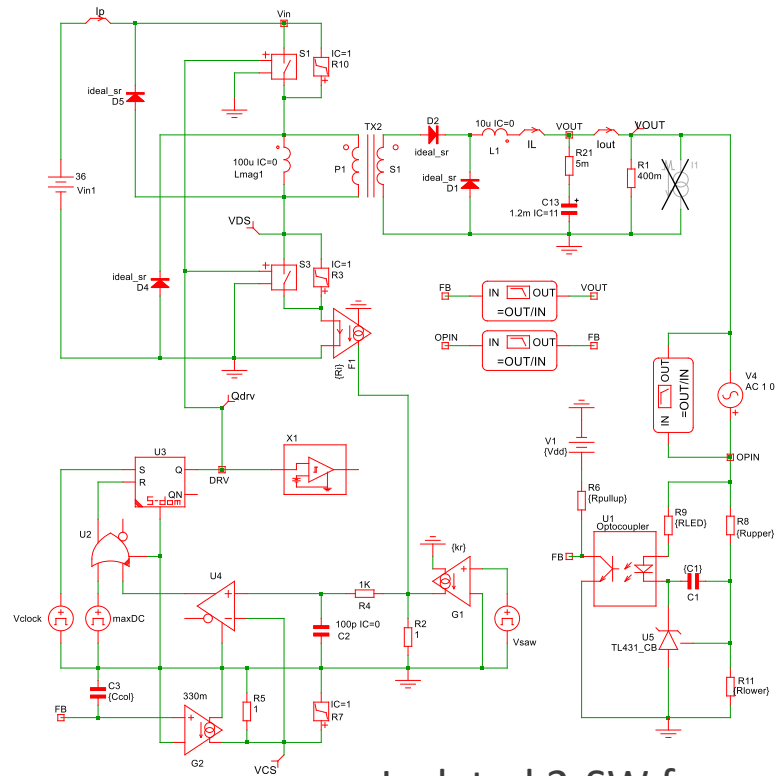
Some components are not compatible with both engines and the program warns you



SIMPLIS Circuit Size Limits

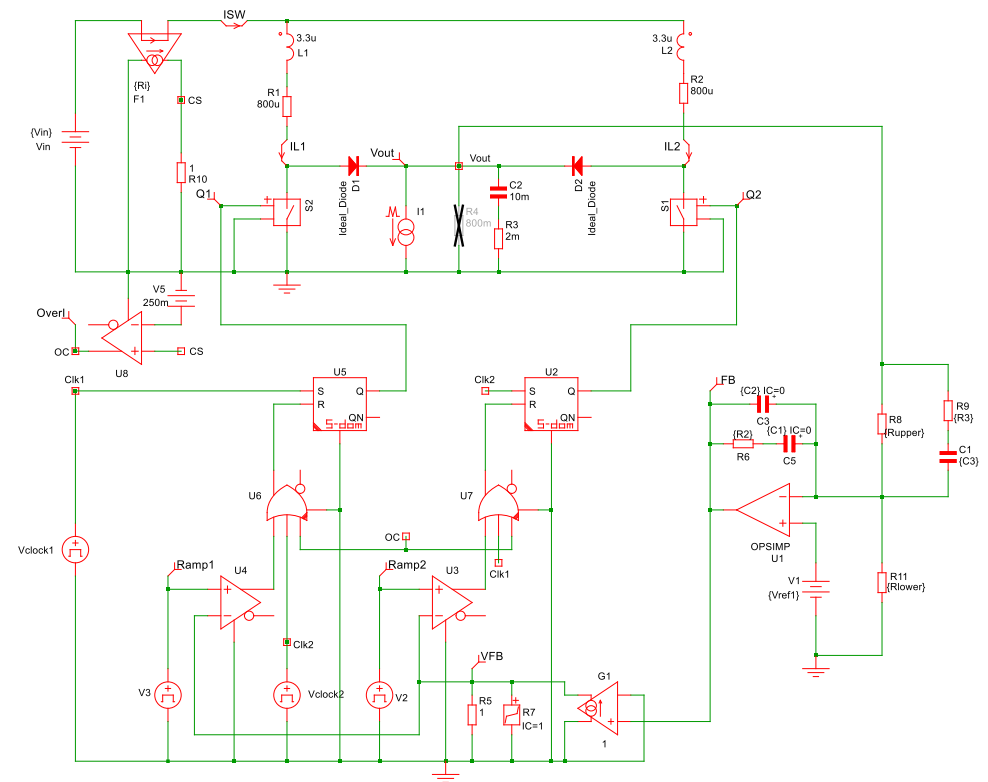
The exact limits for the SIMPLIS simulator are:

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

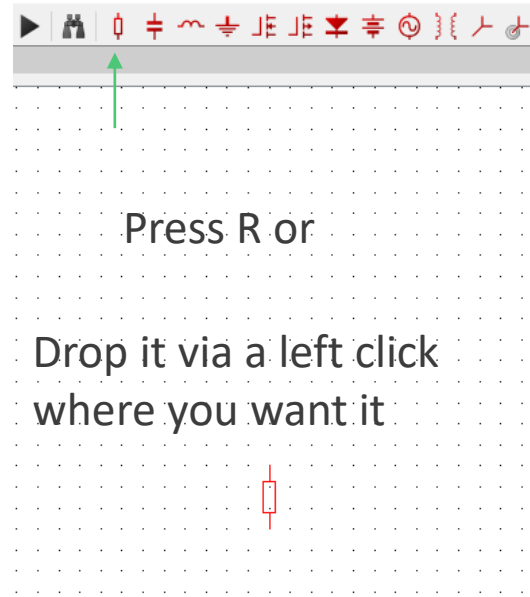
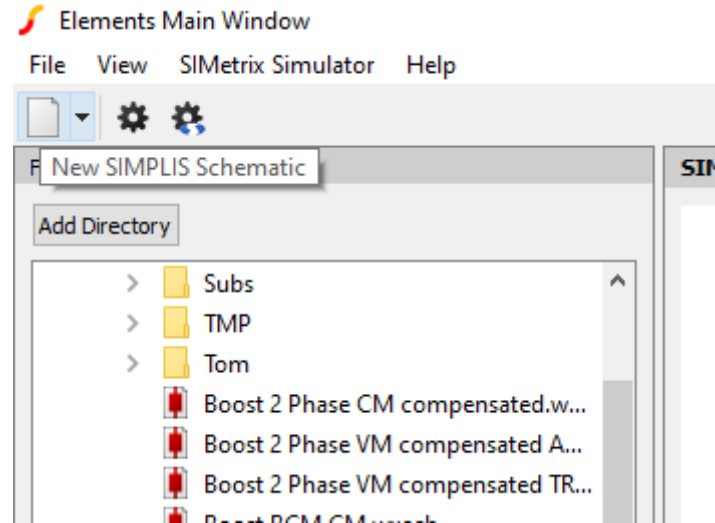


Compensated 2-phase boost

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

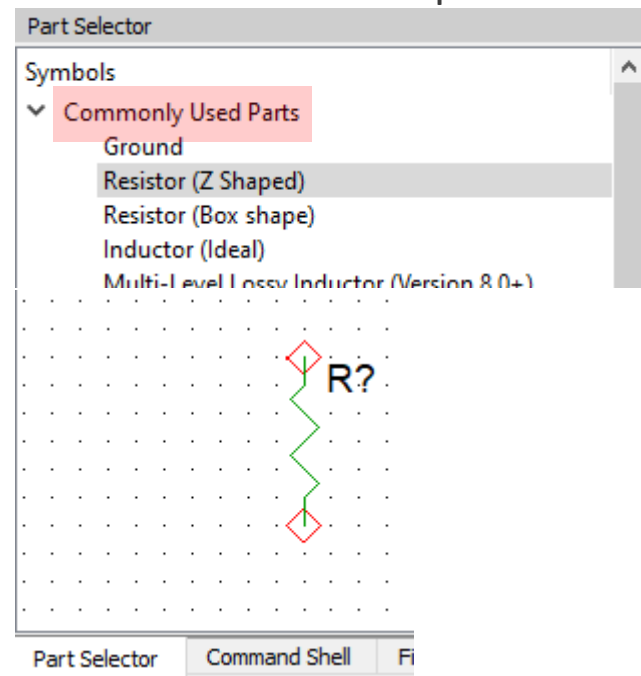
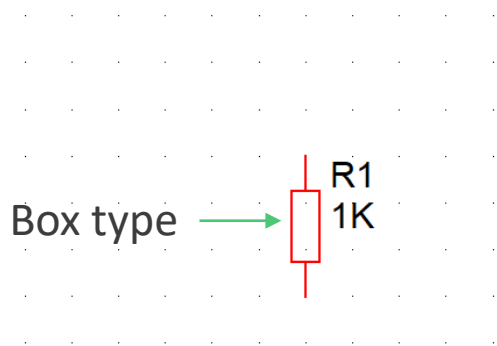
Let's start with a simple simulation and a blank sheet



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

Box or Z-shape



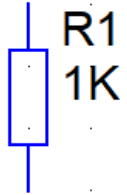
Zoom-in and out by <CTRL> + central mouse wheel

Zoom by <CTRL> + mouse wheel

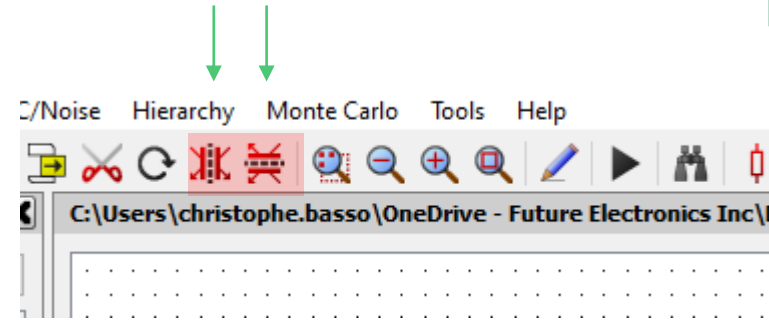
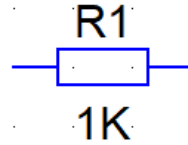
Panning with mouse: right-click and maintaining the pressure, the cursor turns to hand



Select the part
It becomes blue

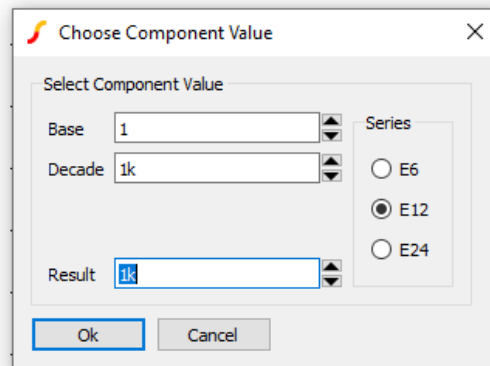
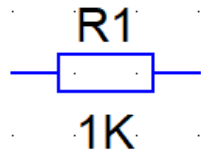


Rotate it with F5
Mirror it with F6

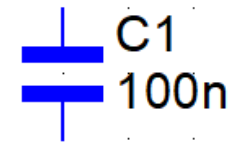
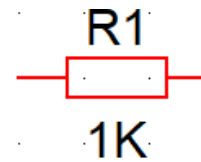


You can also use the toolbar buttons to rotate or mirror the part.

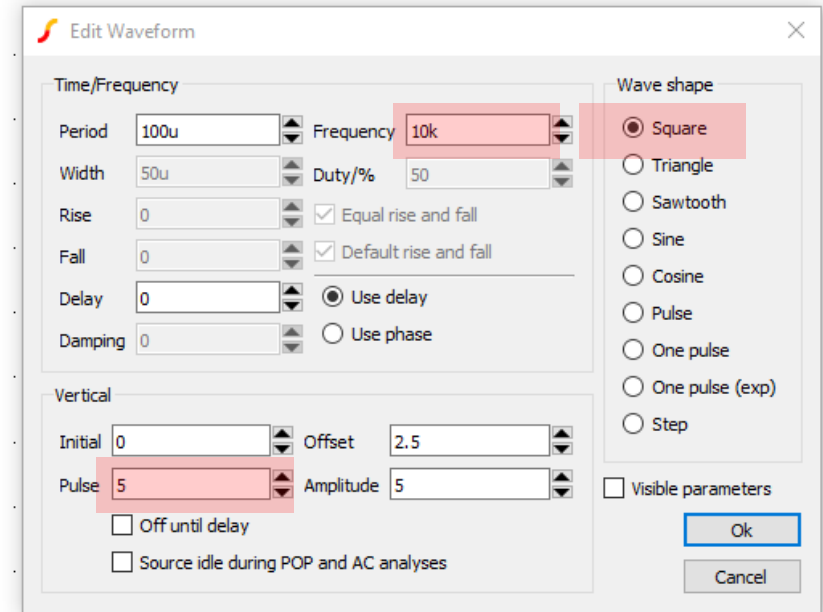
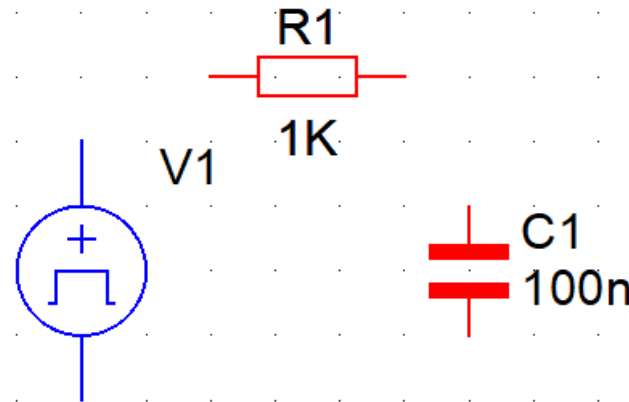
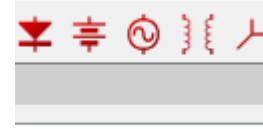
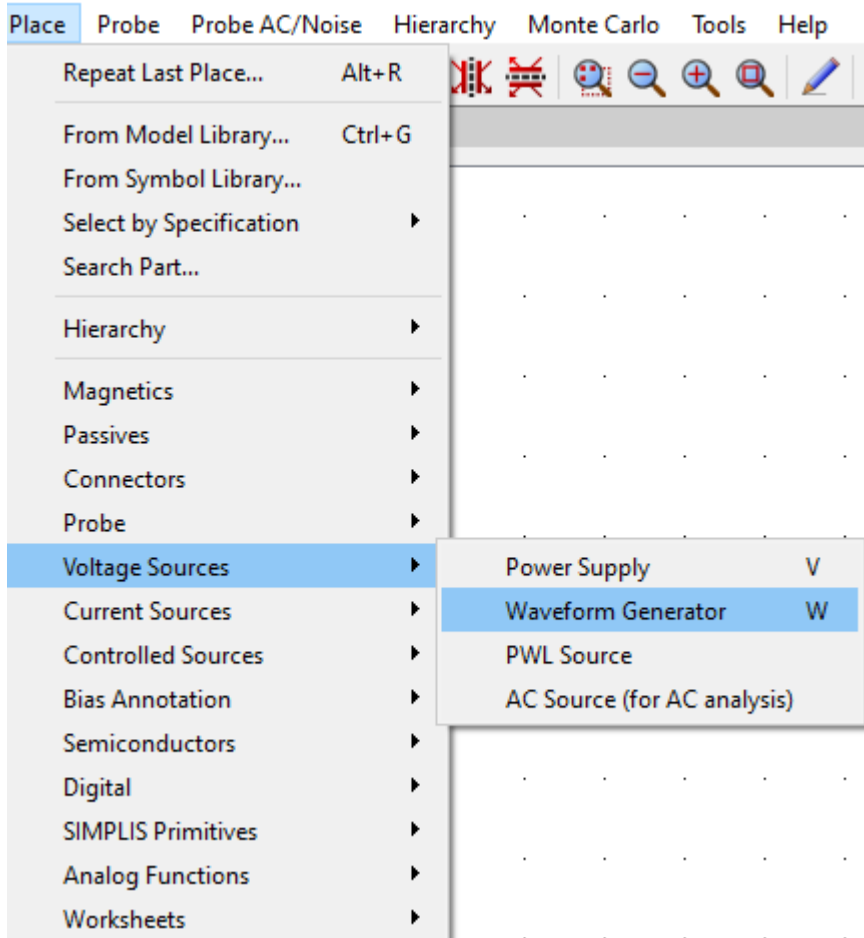
Change the component value
by double clicking



Press C or use toolbar

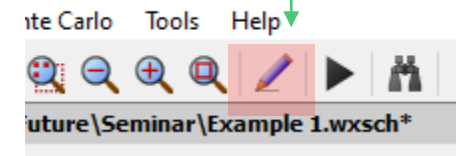
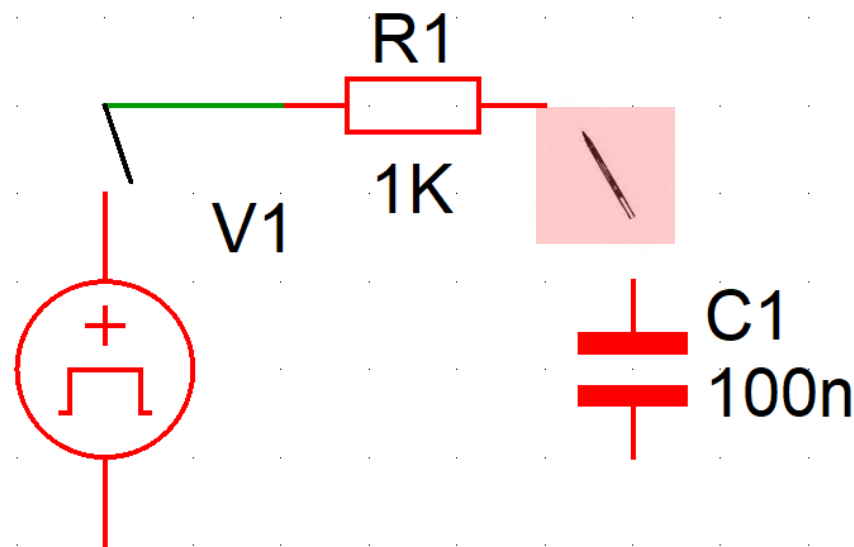
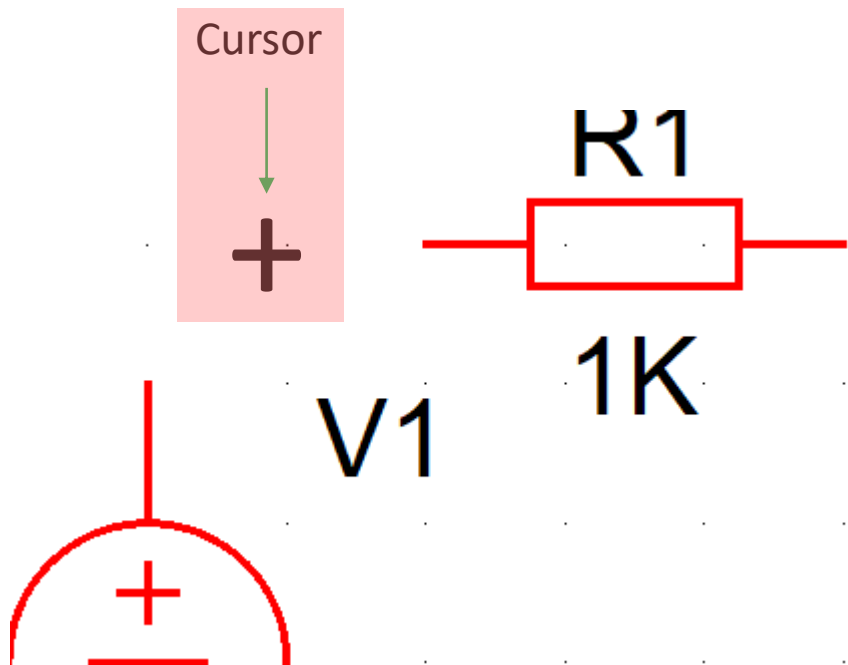


Add a waveform generator – press W or pick-up in the toolbar

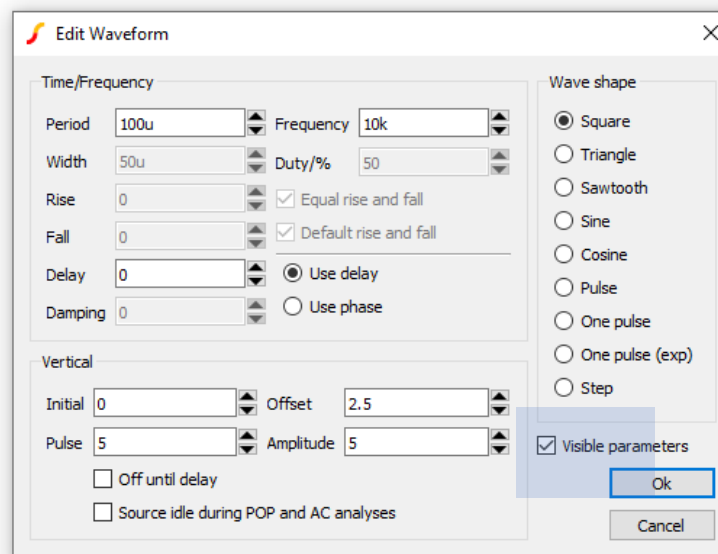
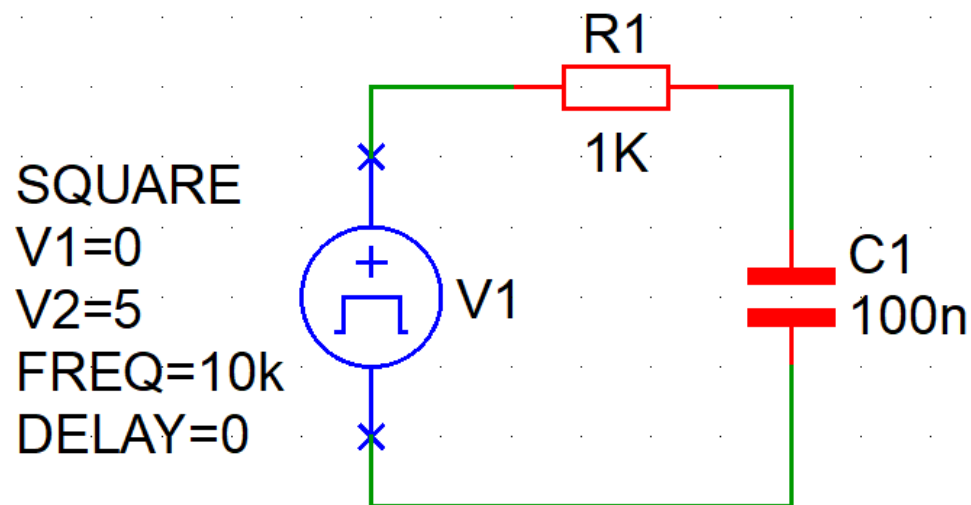


Double click

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

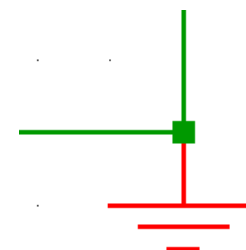


Double-click on the left mouse button also starts the pen

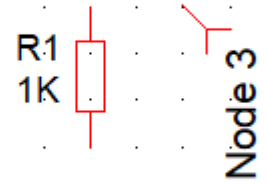


Add a ground by pressing G

Forgetting to place a ground is a classic!



How to modify or rotate labels



- Edit Part...
- Edit/Add Measurement...
- Hide/Show Value
- Descend Hierarchy
- Probe Voltage...
- Probe Current...
- Display Device Bias Info
- Disable Selected
- Disable & Short-circuit Selected
- Enable Selected
- Copy
- Paste
- Edit/Add Properties...**
- Delete Properties...
- Move Text
- Edit Symbol
- Update Symbol
- Restore Properties...

You can hide the value

Edit Properties

Name	Value
*Handle	I3
CLASS	PROBE
EditPropScript	edit_probe_props
guid	{d3c720ae-42b8-4ca4-b3ff-1a7629...}
label	Node 3
MULTISTEP_OPTIONS	advancedstats=false mode=multi ...
PARAMS_MENU	Edit/Add Measurement...
PARAMSSCRIPT	edit_probe_measure
PROBE_DISABLED	false
PROBETYPE	V
REF	Probe1
SIMPLIS_TEMPLATE	.PRINT V(<probe[1]>);GRAPH :<pr...
SIMULATOR	DUAL
StyleNormal	DefaultProbe
StyleSelected	DefaultSelected
TEMPLATE	.GRAPH <node[1]> %VALUE% <gu...
VALUE	axisType="auto" persistence=-1 cu...
VALUE_SIMPLIS	axisType="auto" persistence=-1 cu...
VALUESCRIPT	edit_probe

Double click item to edit Add Property...

Ok Cancel

Double click

Edit Property

Name: label

Value: Node 3

Text Location

Auto

Normal: Left

Rotated: Left

Absolute

Justification: Left Top

Hidden

Vertical

Property Attributes

Font style: Default

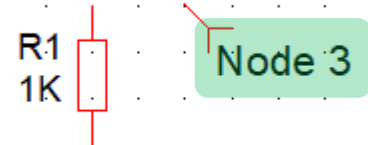
Selectable

Protected

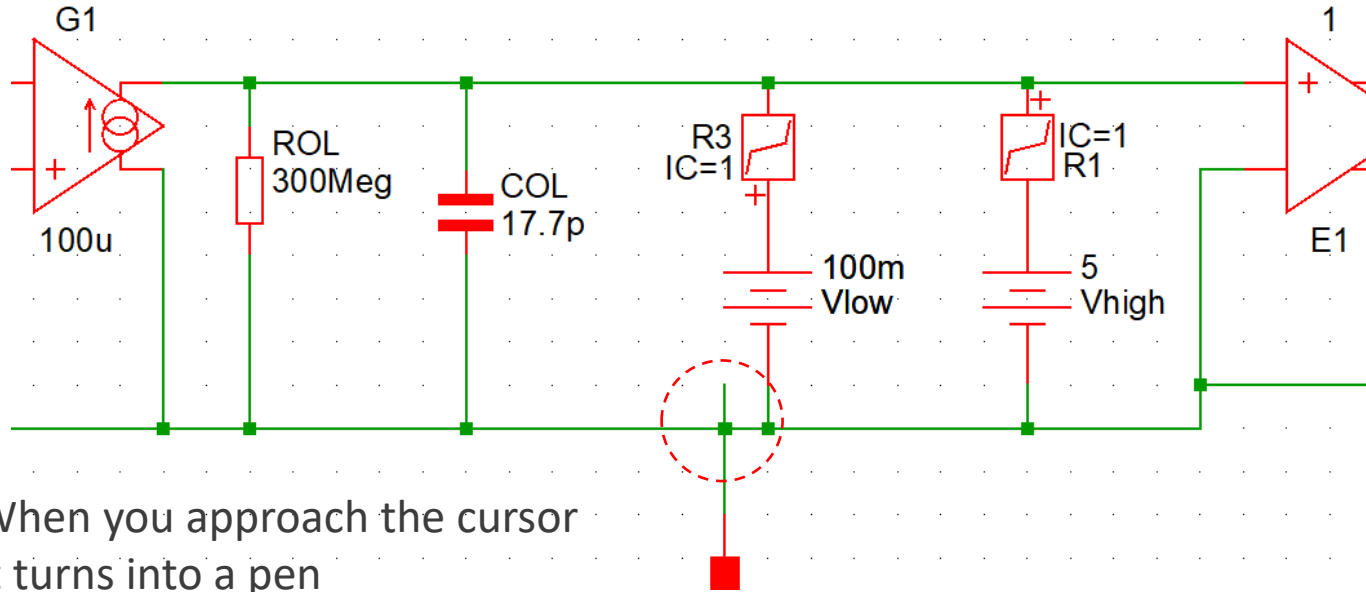
Show name

Resolve symbolic

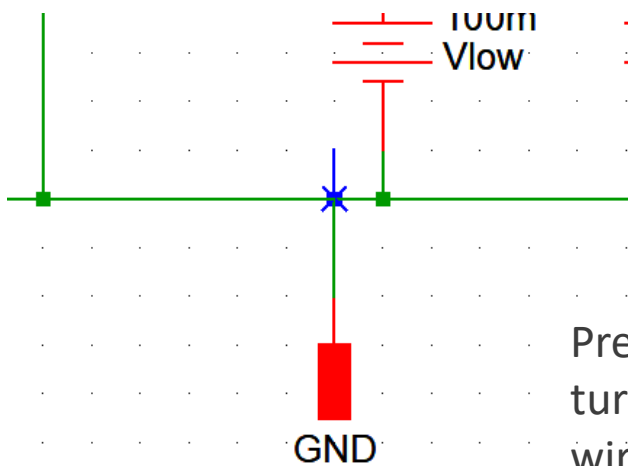
Ok Cancel Help



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

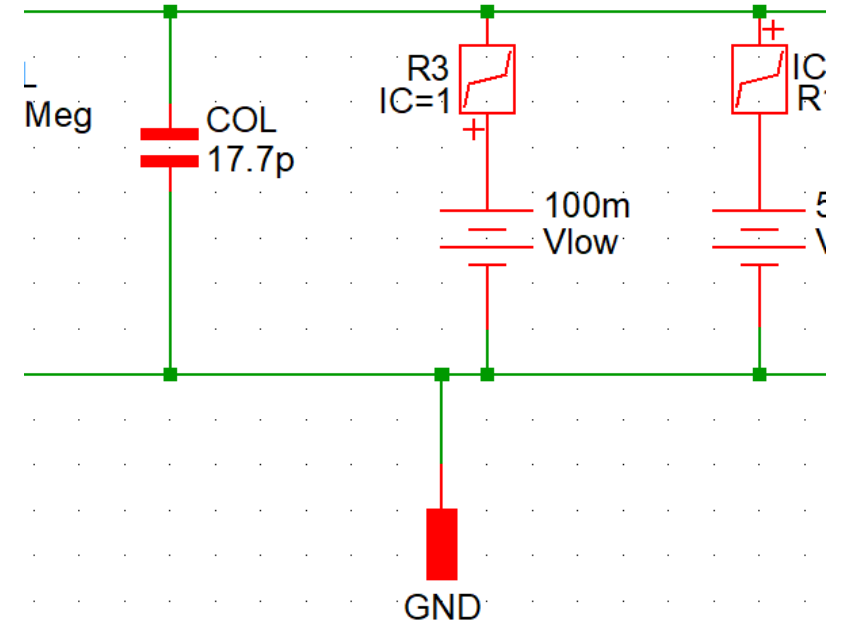


When you approach the cursor it turns into a pen

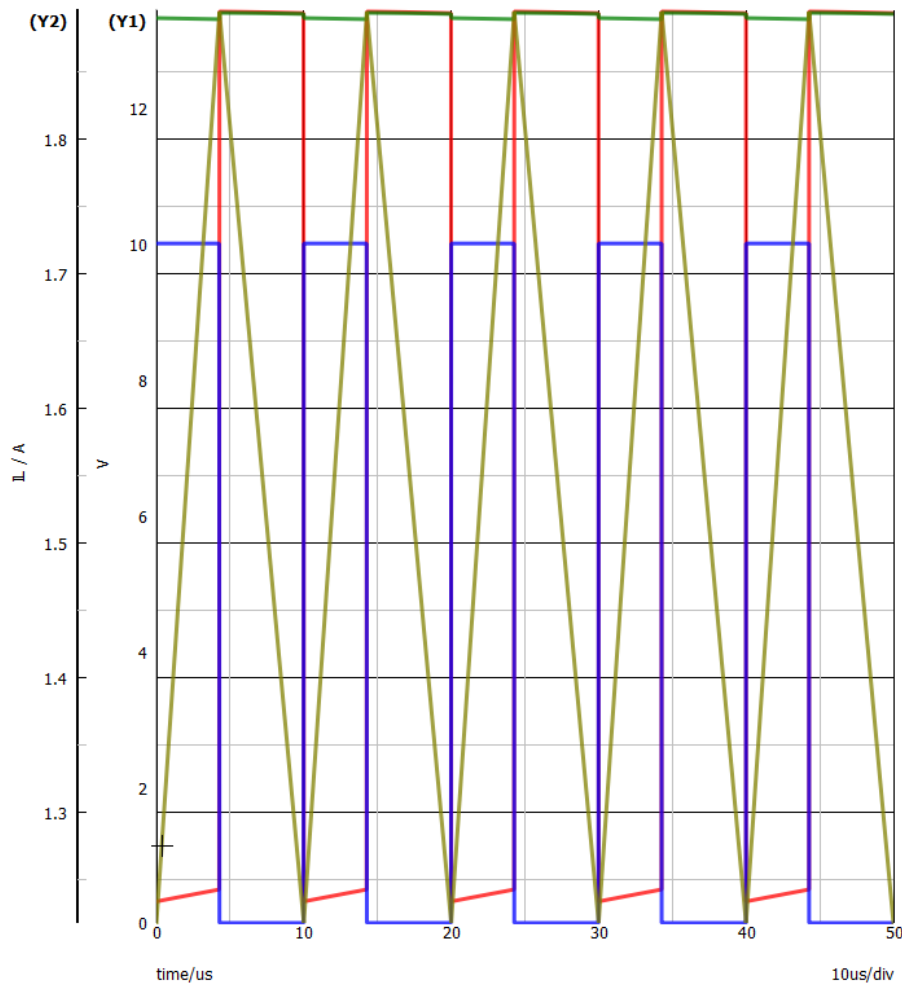


Press SHIFT and the cursor turns into a cross: select the wire which turns blue. Delete it.

Voilà!

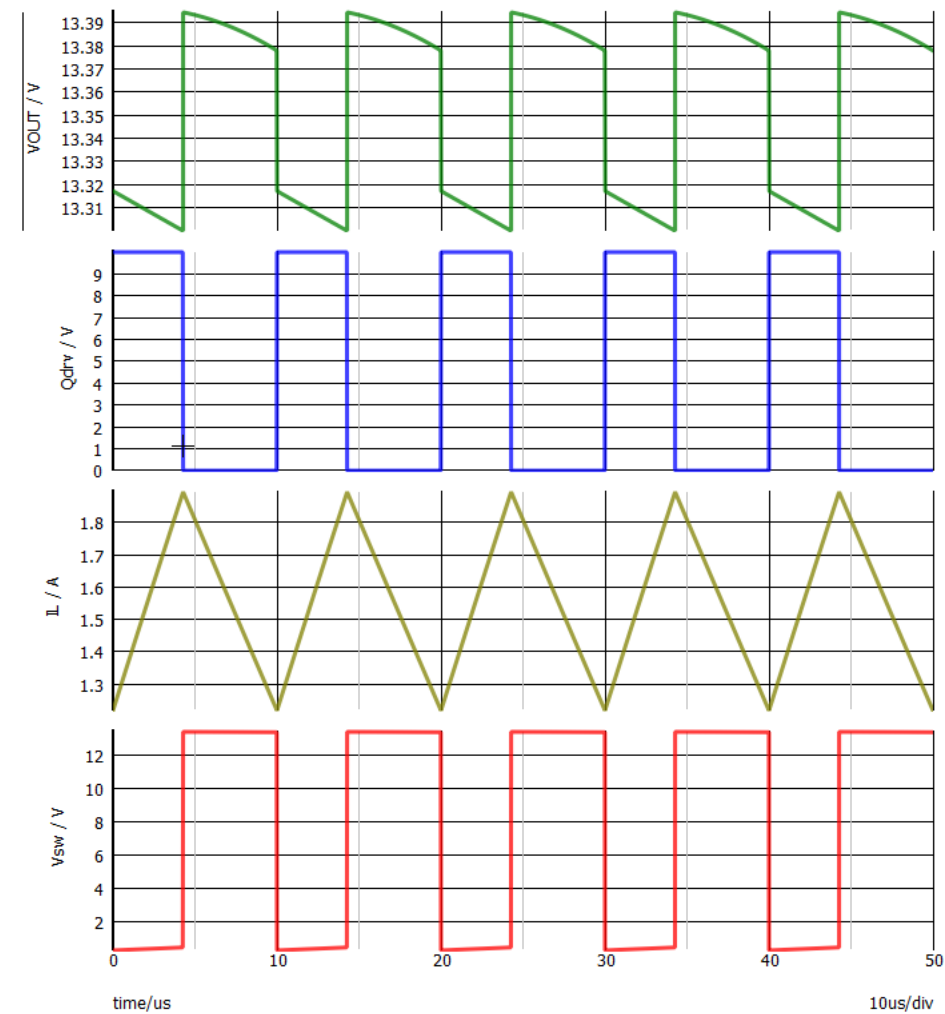


If you place probes without specifying axis information, by default, the viewer stacks up all the curves, ending in an ugly representation as below. To have them go to their own axis – what it should do by default in my opinion – open the Curves menu and choose “stack all curves”:

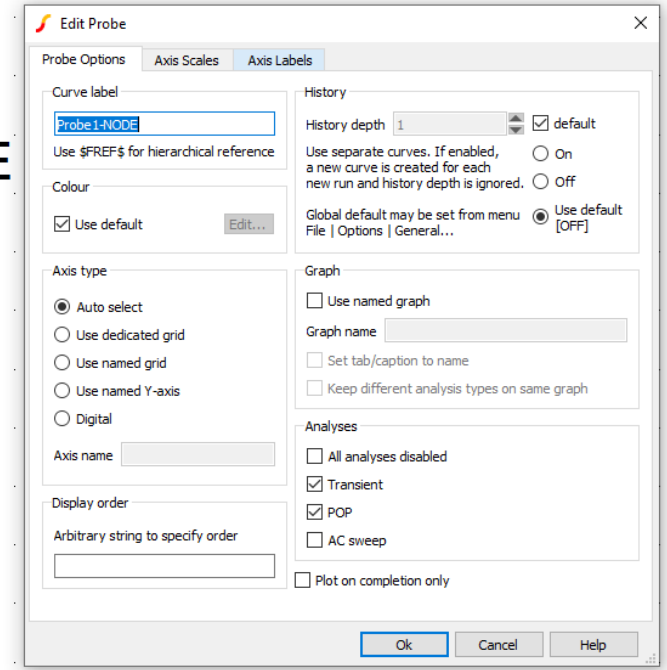
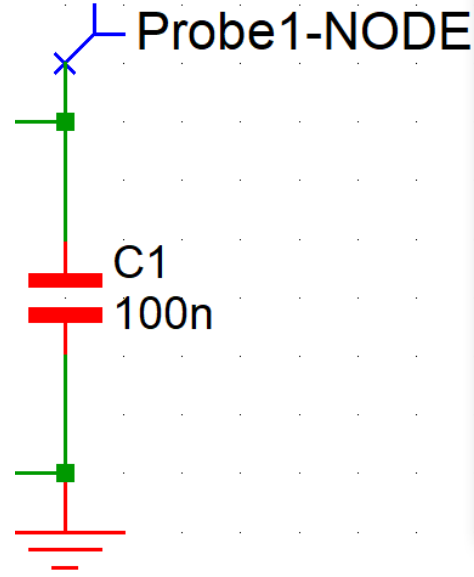
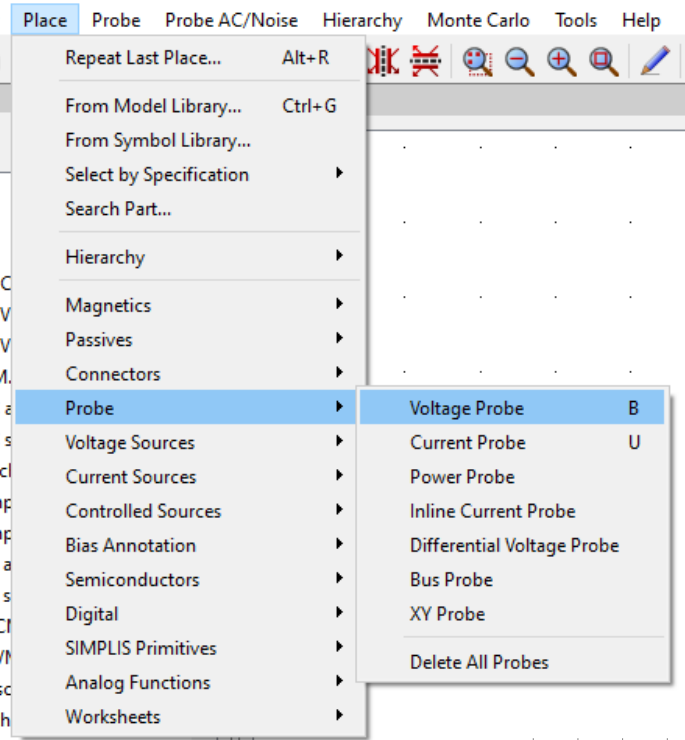


Curves Axes Measure Plot Monte Ca

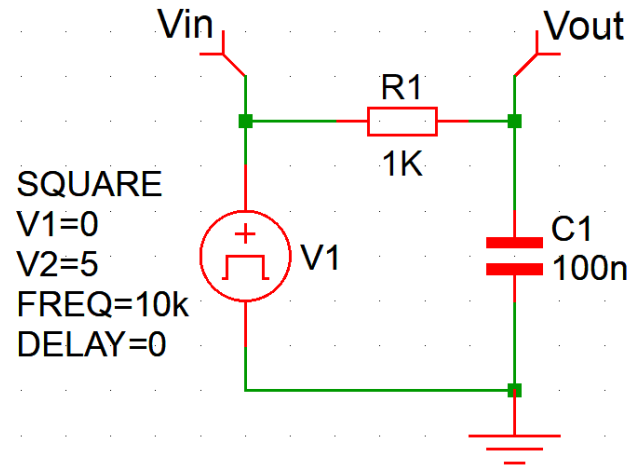
- Edit Curve Labels and Colours...
- Show/Hide Points
- Hide Selected Curves
- Show Selected Curves
- History
- Ungroup Selected Curves
- Delete Selected Curves Del
- Move Selected Curves
- Stack All Curves**
- Stack Selected Curves
- Select All Curves
- Unselect All Curves
- Highlight Selected Curves H
- Unhighlight Selected Curves U
- Unhighlight All Curves



You place probes by pressing B on the keyboard:

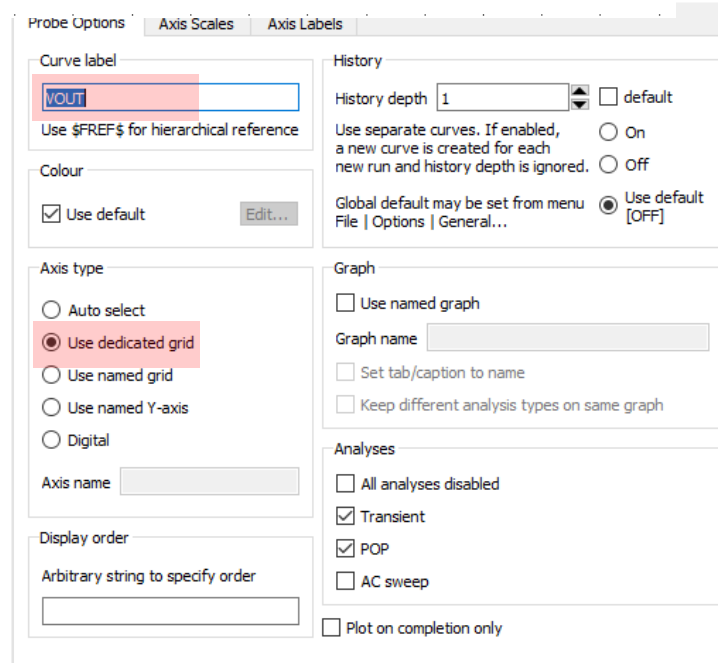


Label the probe



Tick this button

Enter the name



Choose the simulation parameters – press F8

Choose SIMPLIS Analysis

Periodic Operating Point AC **Transient**

Analysis parameters **Time duration**

Stop time 1m s

Start saving data at t = 0 s Default

Plot data output

Number of plot points 10k Default

Advanced...

Select analysis

POP

AC

Transient

Transient analysis

Save options

All

Voltages Only

Probes Only

No Forced Output Data

Force New Analysis

Ok Run Cancel Help

Wanted granularity
Does not affect precision

Once the circuit is saved, go!

SIMPLIS Status

Analysis status

Analysis Transient Data Group simplis_tran2

Run status Finished Step

Run Progress 100%

POP Status	Pass	n/a	Convergence	n/a
Time	1m		Topologies	New topology #1

Elapsed Time Current 0 hr 0 min 1 sec Total 0 hr 0 min 2 sec

CPU time Current 0 hr 0 min 0.04 sec Total 0 hr 0 min 0.07 sec

New topology #1

A starting operating point located.

Elapsed time : 0 hr 0 min 1 sec
CPU time : 0 hr 0 min 0.03 sec
Simulation time: 0.000000000000e+00 sec

TIME-DOMAIN TRANSIENT ANALYSIS

```

01 02 03 04 05 06 07 08 09 10
11 12 13 14 15 16 17 18 19 20
21 22 23 24 25 26 27 28 29 30
31 32 33 34 35 36 37 38 39 40
41 42 43 44 45 46 47 48 49 50
51 52 53 54 55 56 57 58 59 60
61 62 63 64 65 66 67 68 69 70
71 72 73 74 75 76 77 78 79 80
81 82 83 84 85 86 87 88 89 90
91 92 93 94 95 96 97 98 99 100
    
```

Elapsed time : 0 hr 0 min 1 sec
CPU time : 0 hr 0 min 0.04 sec
Simulation time: 1.000000000000e-03 sec

Writing pertinent data files ...
Leaving SIMPLIS.

***** Thank you for trying Elements *****
***** For additional support, please visit www.simplistechnologies.com *****

Abort Clear Messages Close Close on completion

When everything is ok, the complete windows pops-up

Welcome to **SIMetrix/SIMPLIS Elements**.
For help using this application, please
use the help menu located above.

```

*** ERRORS REPORTED BY SIMPLIS ***
On Elements, Version : 8.50e
*****
<<<<<<< Error Message ID: 1077 >>>>>>>

input file C:/Users/christophe.basso/
OneDrive - Future Electronics Inc/
Documents/SIMPLIS/Future/Seminar/
SIMPLIS_Data/Simple buck 3.deck, line 19:

.PRINT V(14)

V(14): 'Node 14' is not defined in
the circuit called 'MAIN CIRCUIT'

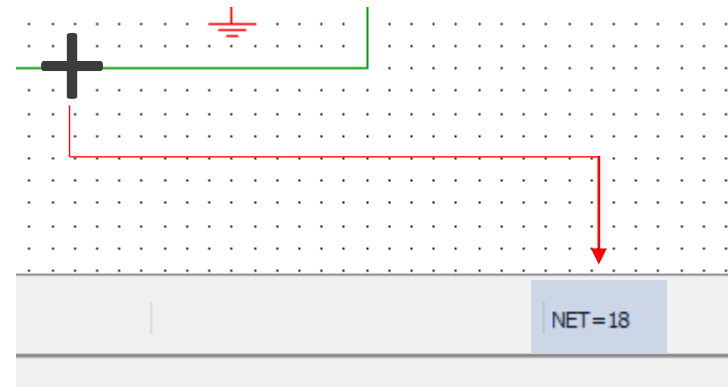
On Elements, Version : 8.50e
*** END SIMPLIS ERROR REPORT ***

```

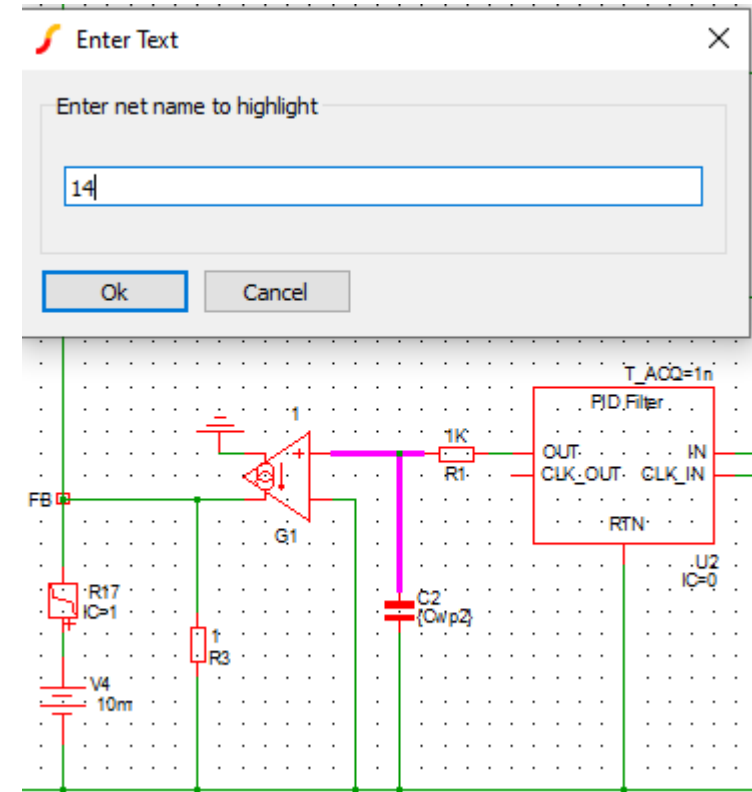
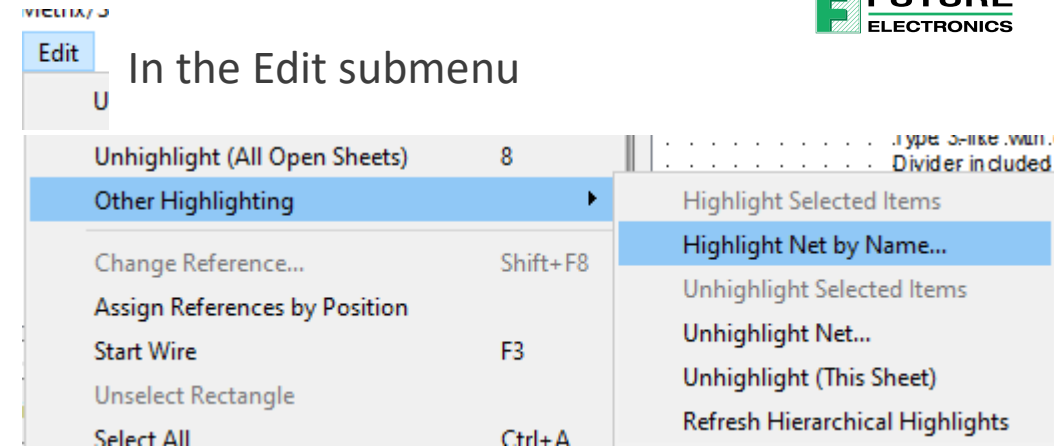
In case simulation issues occur,
look at the Command Shell
window.

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.

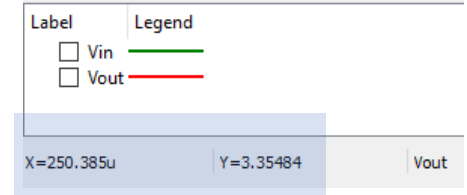
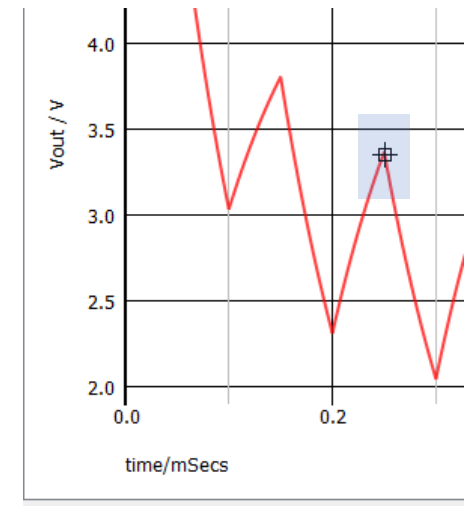
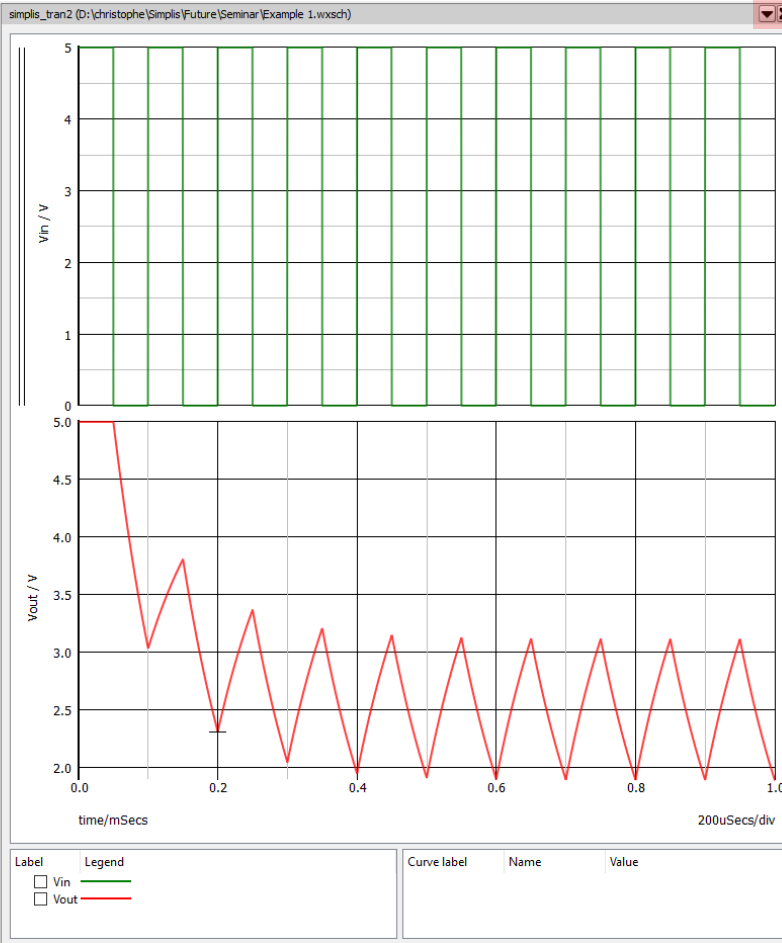
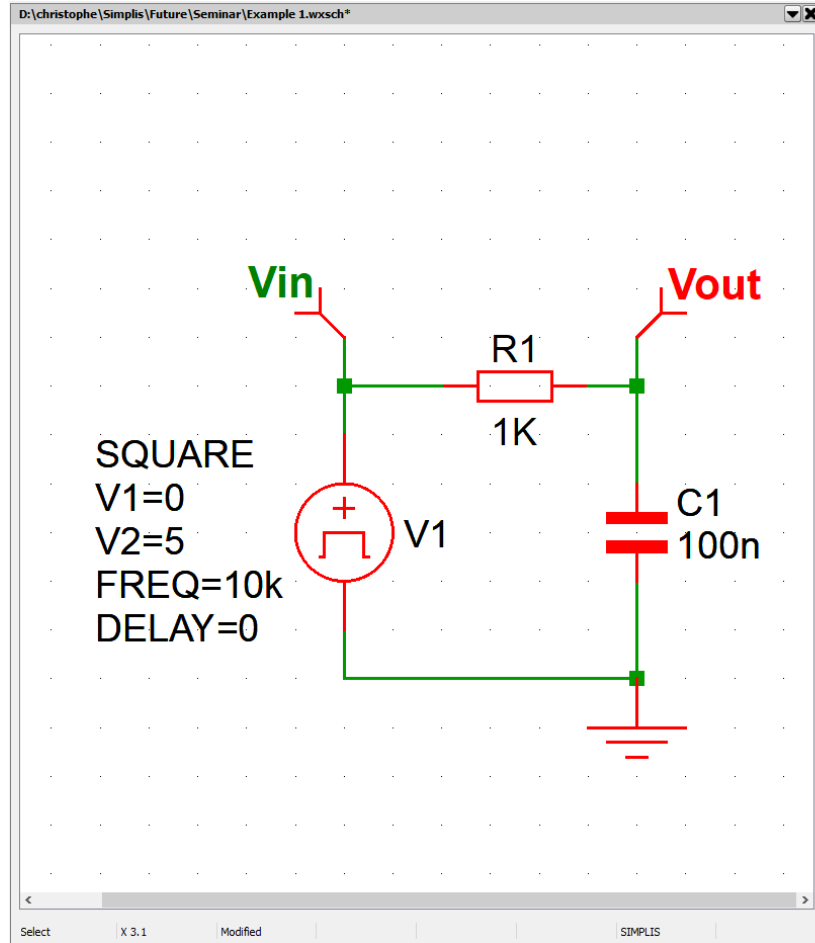
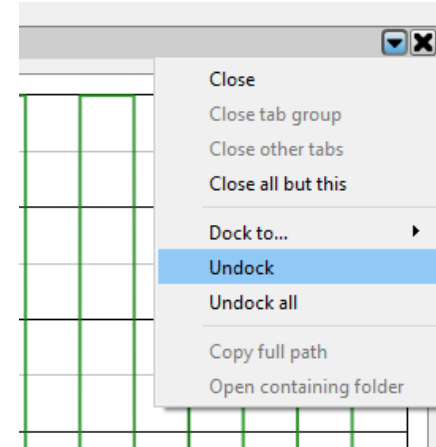


Fly over a (cuckoo's) net and see
the number displayed in the
low-side bar.



To erase the highlight, press 8 on the keypad

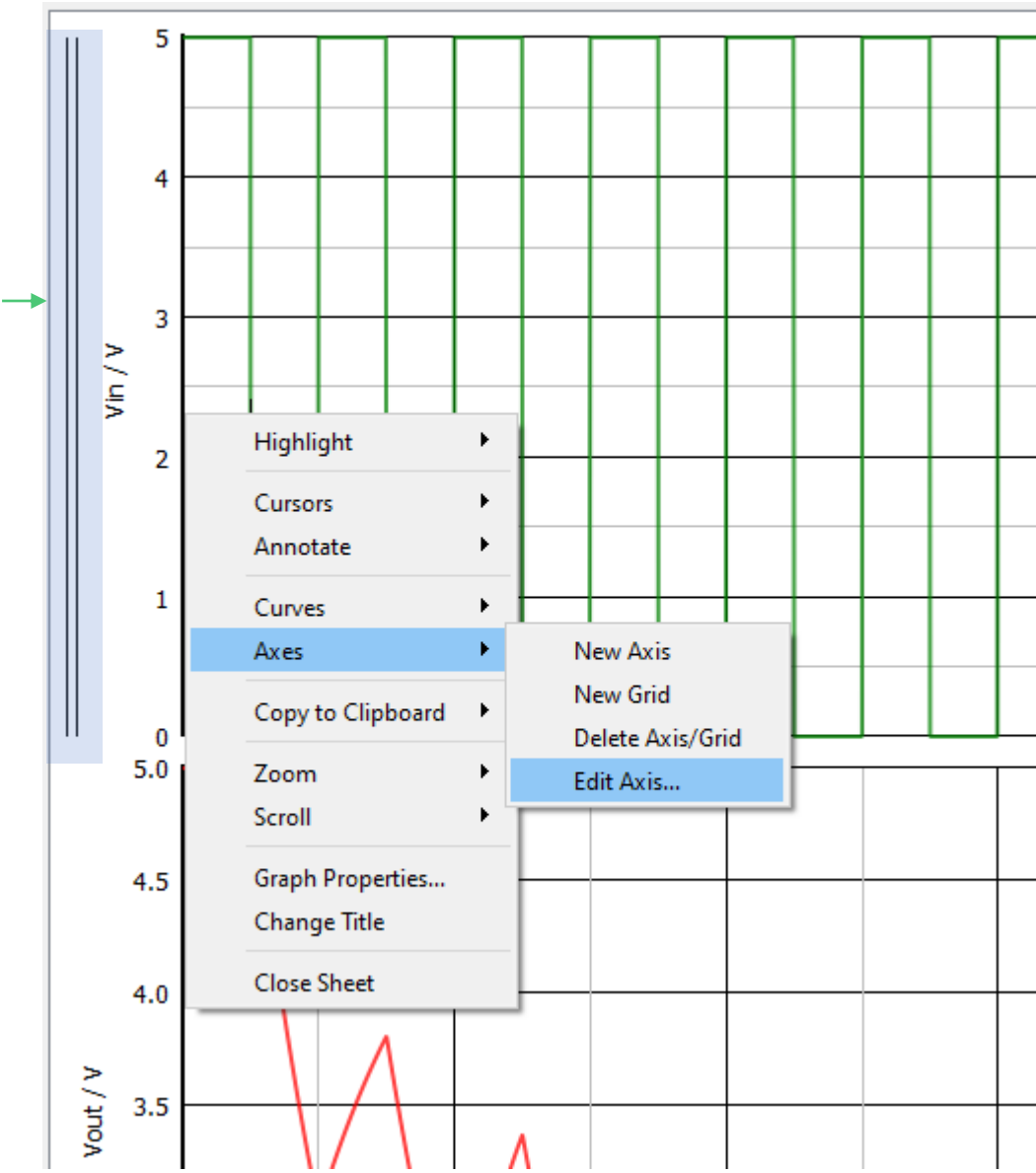
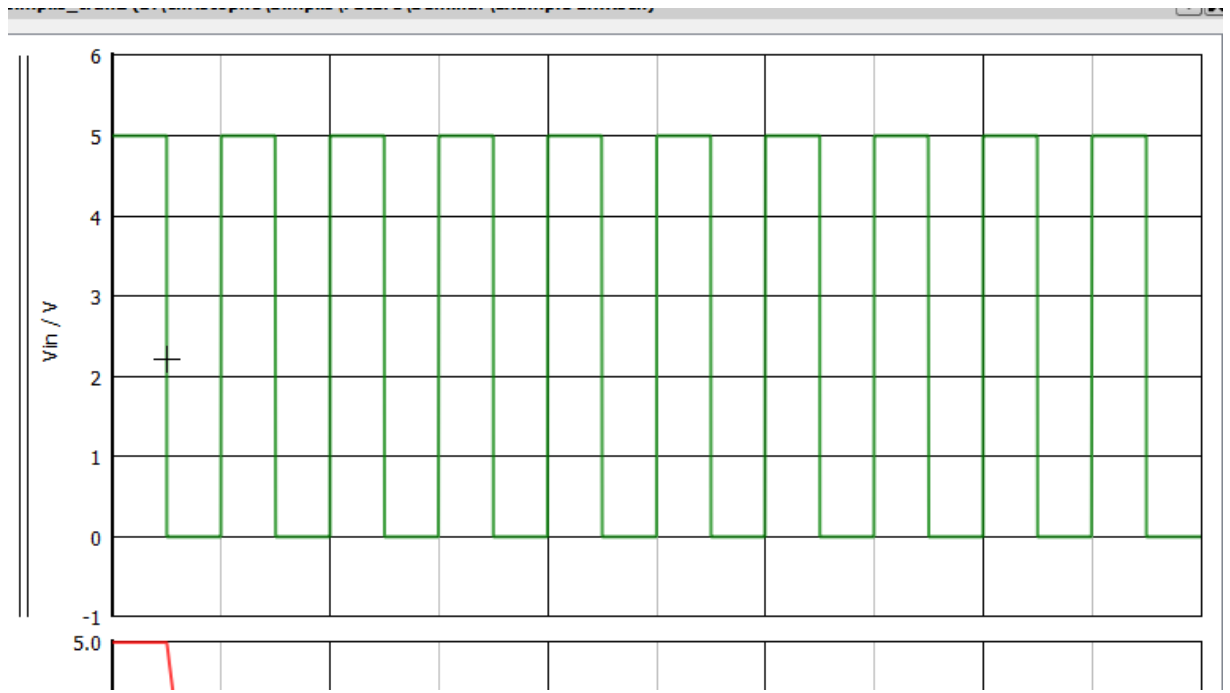
You can un-dock the result window



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

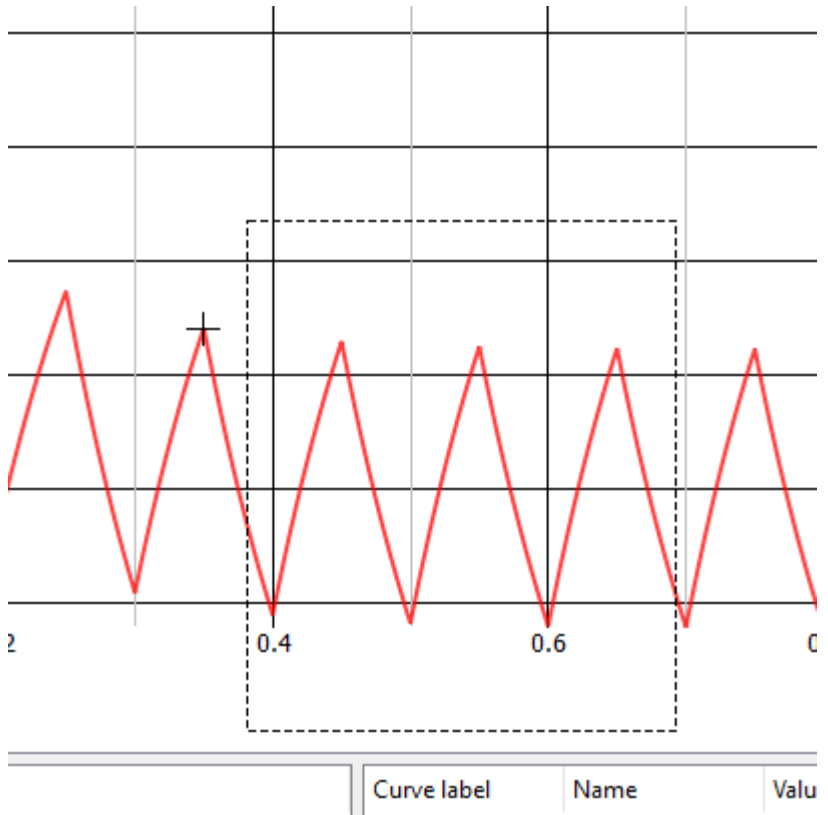
Selected sign

To change the axis

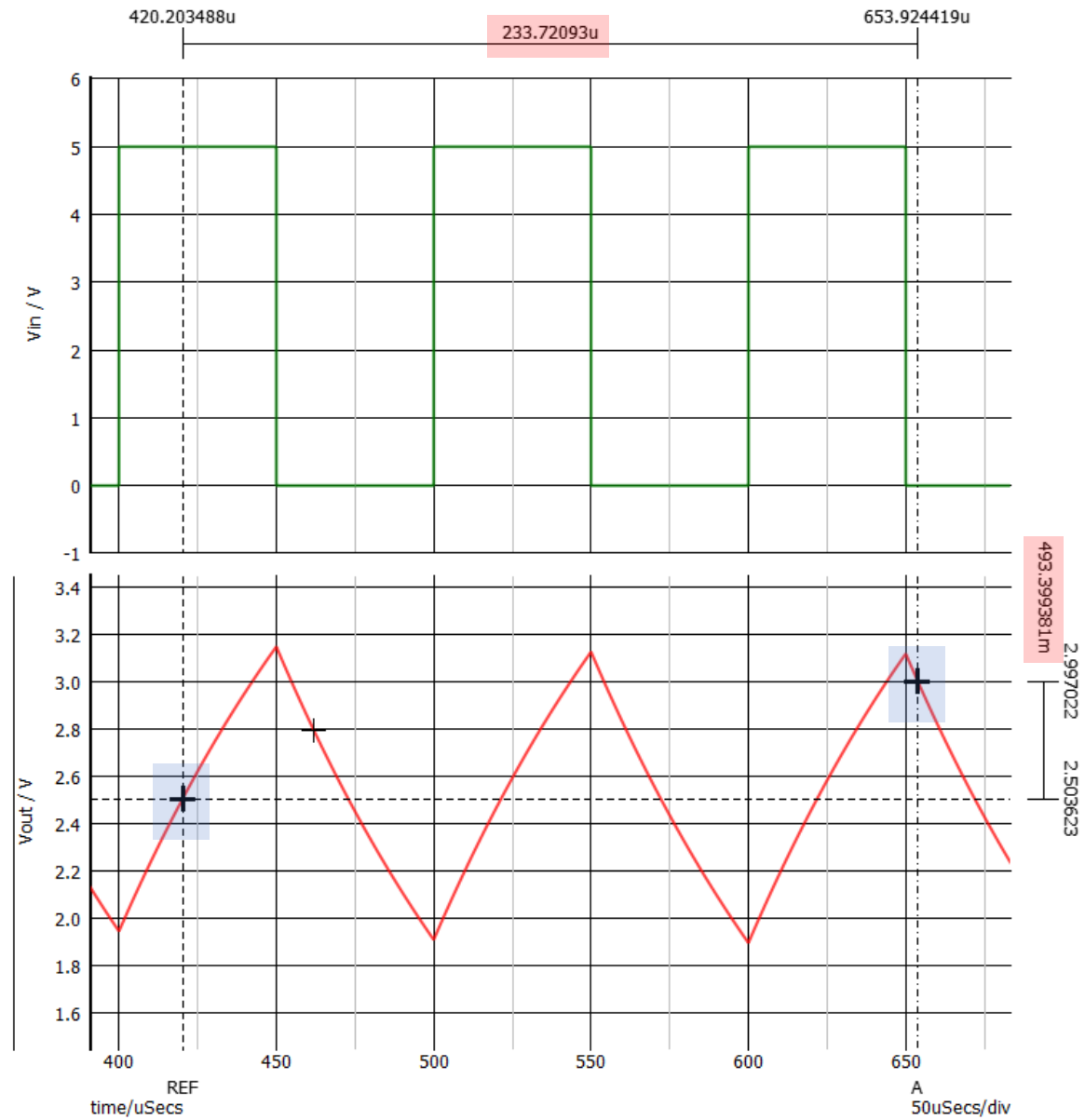



ΔX

Zoom-in by dragging a window

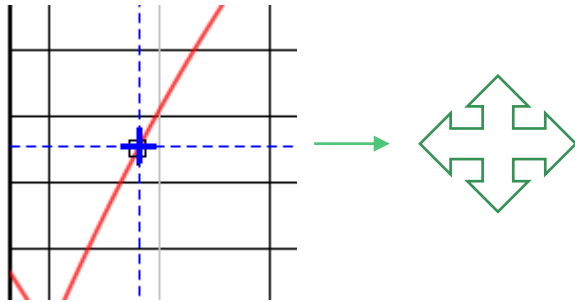


And call the cursors by pressing C



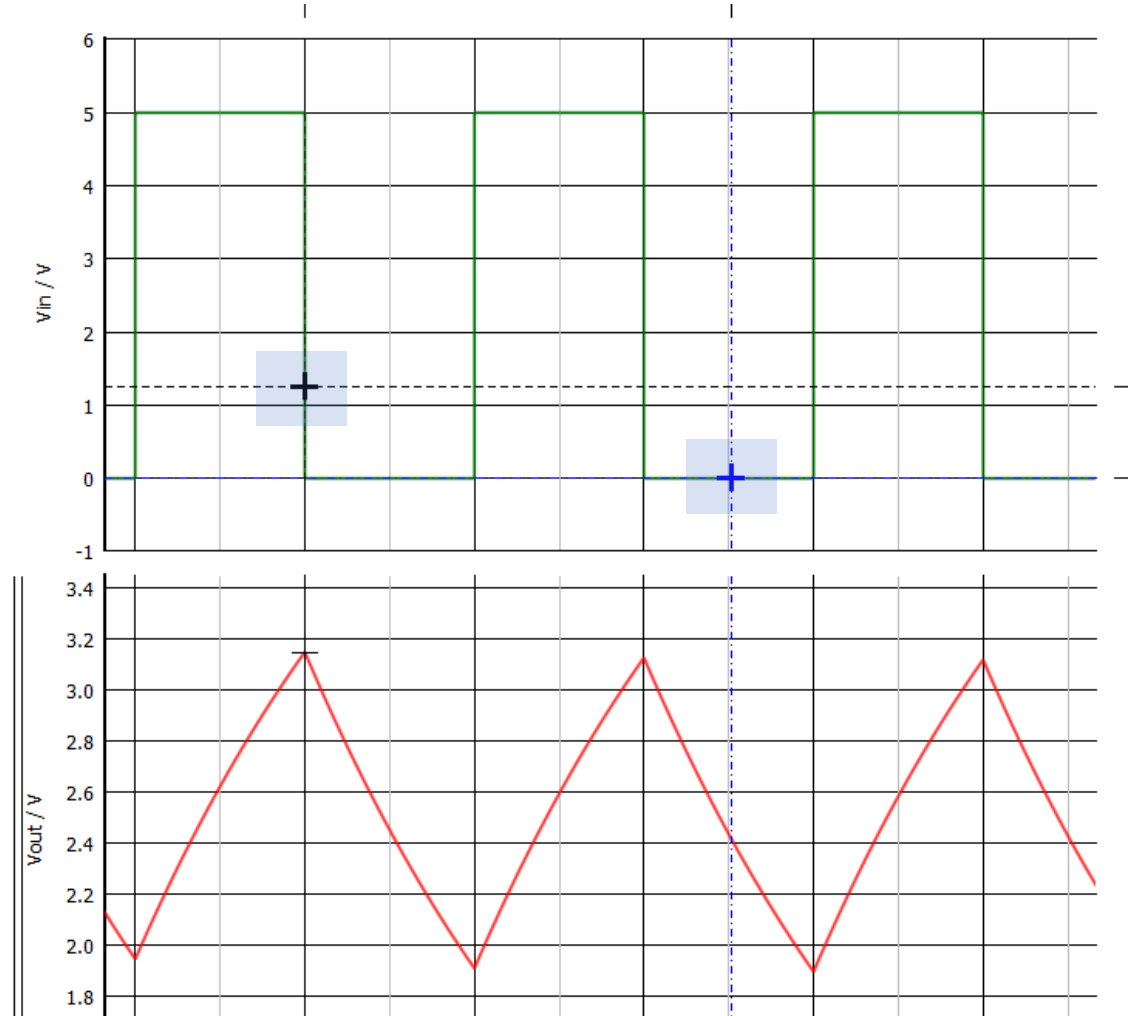
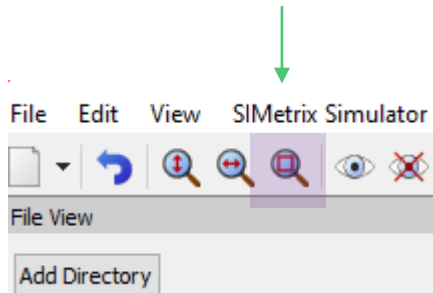
ΔY

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

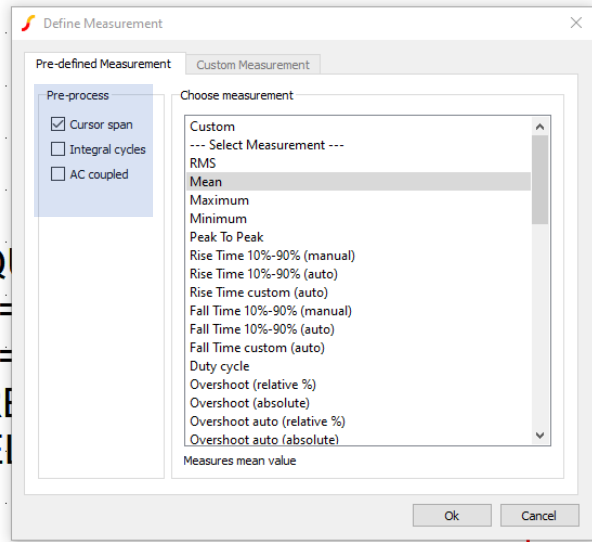


Then drop the cursor
to the other waveform
to explore it

Go back to full span



To run a measurement on the waveforms, press F3

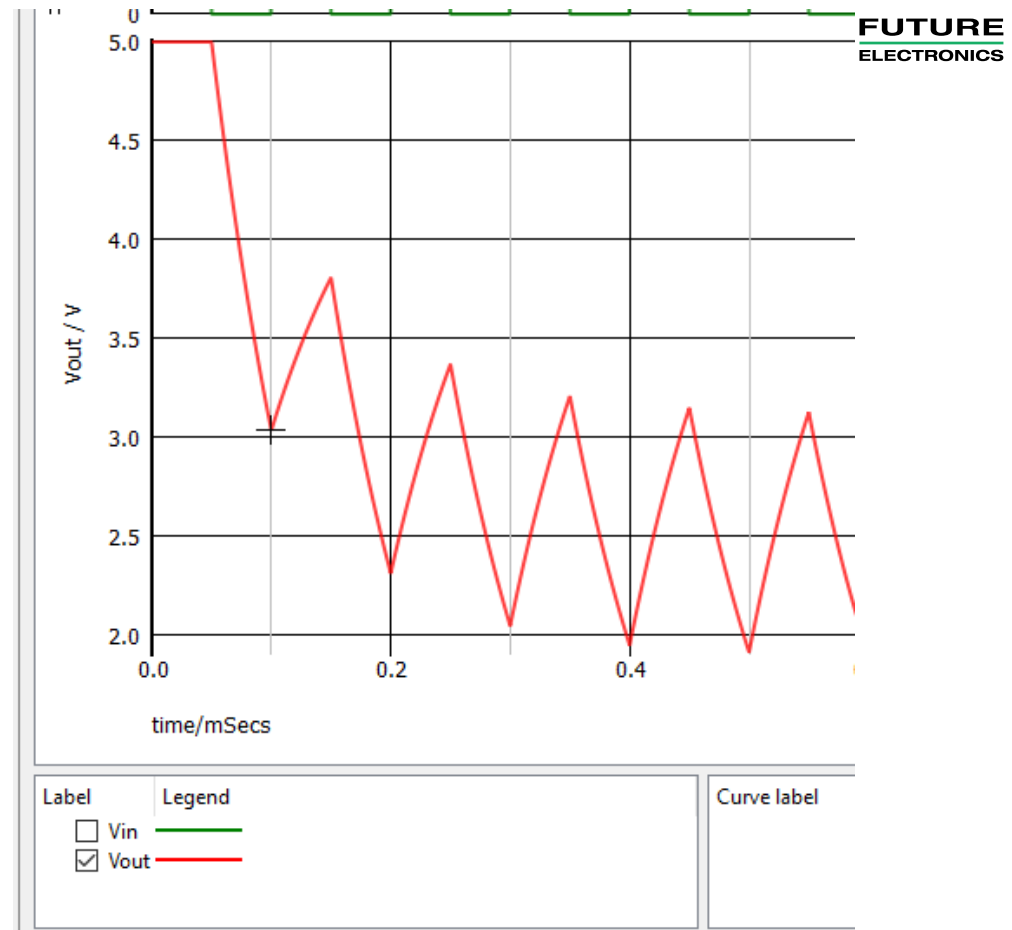
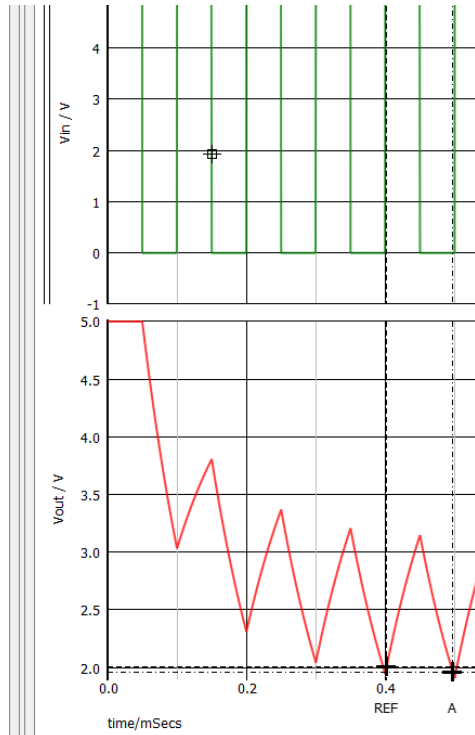


Vout

100n



Curve label	Name	Value
Vout	Mean	2.8112156V



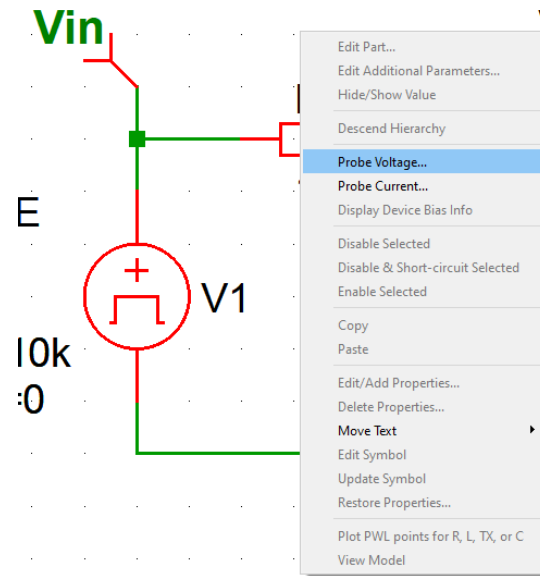
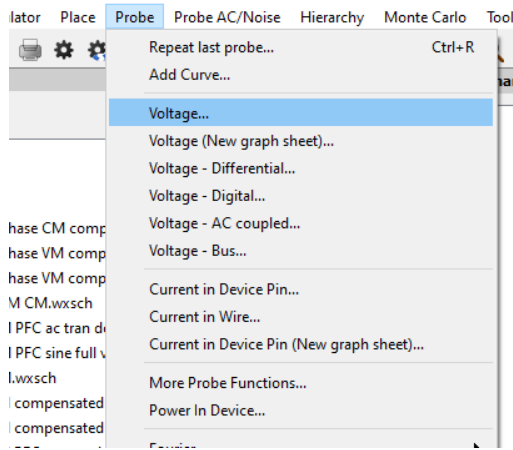
To delete a waveform, select it and press delete

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.

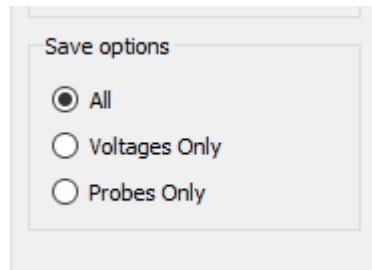
AC coupled Offsets the data by the mean value. This is equivalent to 'ACcoupling' the data.

You can probe on the fly



Or right click in the schematic

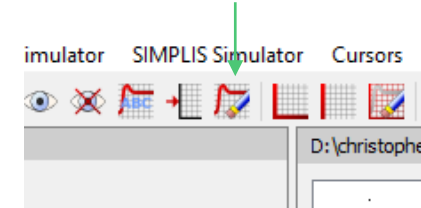
For on-the-fly probing don't forget to activate the All option in the sim window (F8)



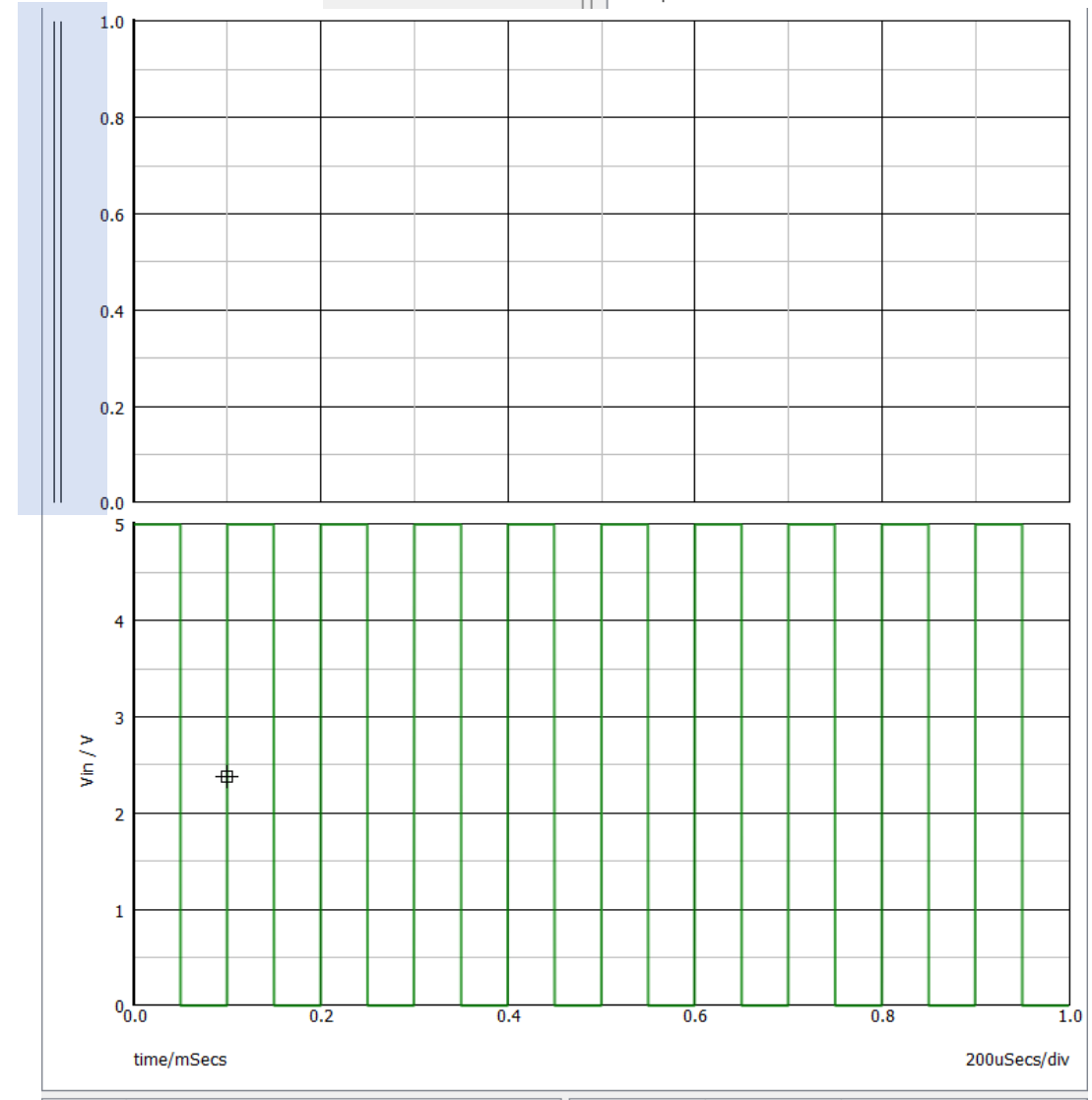
The probed signal will go to the newly-selected grid

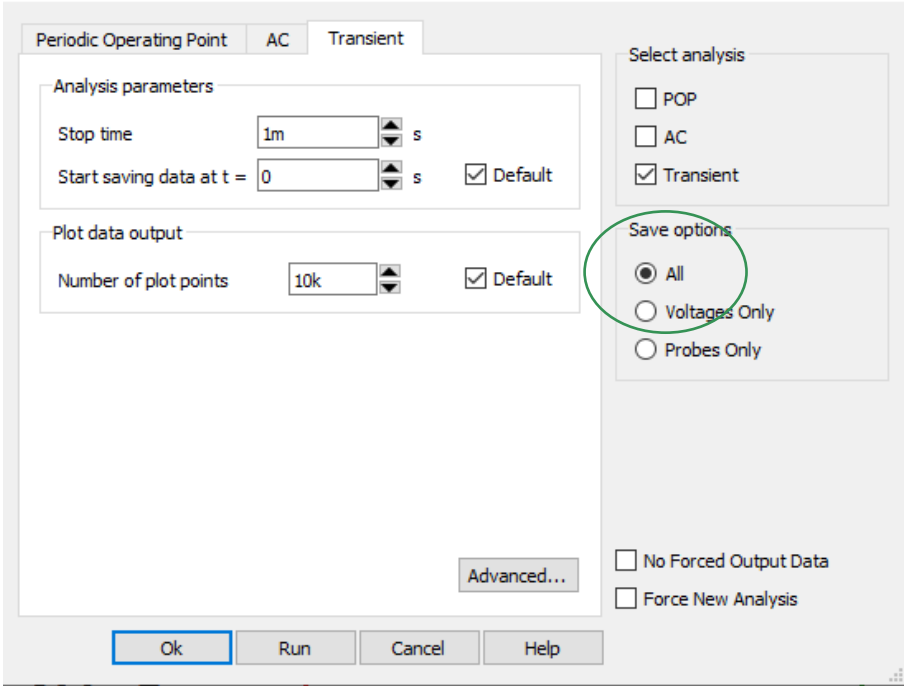


New grid



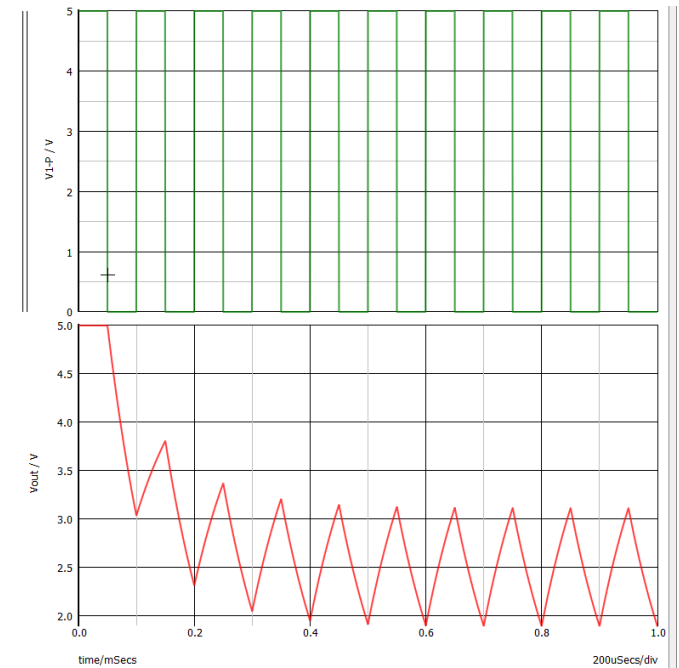
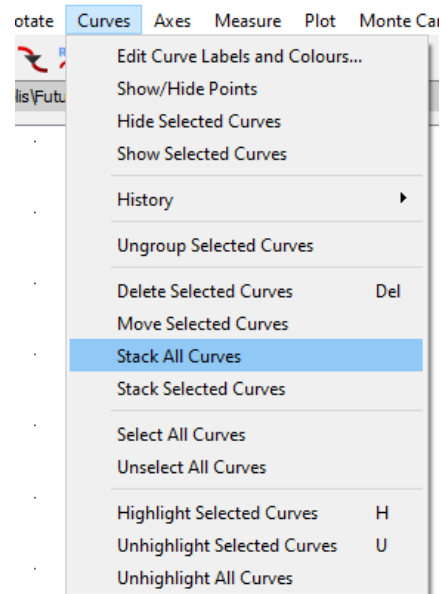
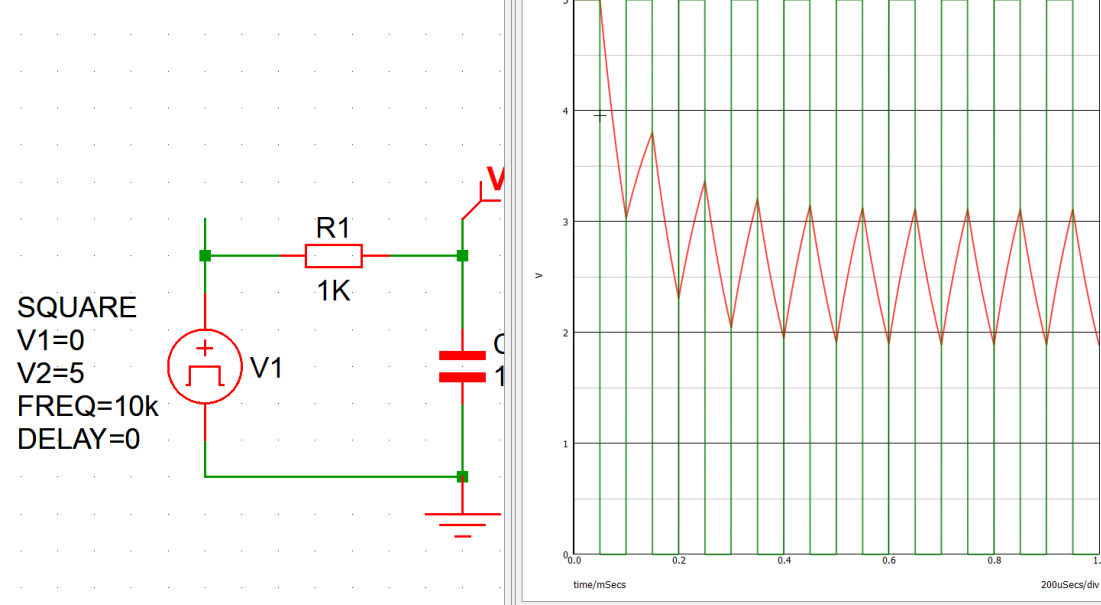
Selected





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.

The waveforms are placed on a common grid



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.

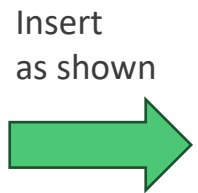
Press F11 while on the main schematic:

```

param Rlower={2:5/lb}
param fc=40

1 .simulator SIMETRIX
2 *.tran 1m
3 .ac dec 100 10m 1k
4 .simulator DEFAULT
5

```



SIMetrix

```

1 .simulator SIMETRIX
2 *.tran 1m
3 .ac dec 100 10m 1k
4 .KEEP **I **V
5 .simulator DEFAULT
6

```

SIMPLIS

```

1 .simulator SIMPLIS
2 .ac DEC 200 10 100k
3 .KEEP **I **V
4 .print
5 + ALL
6 .options

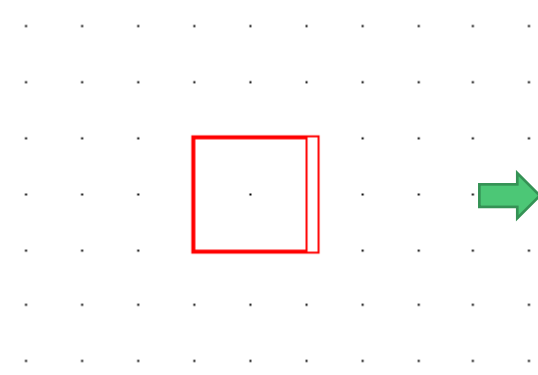
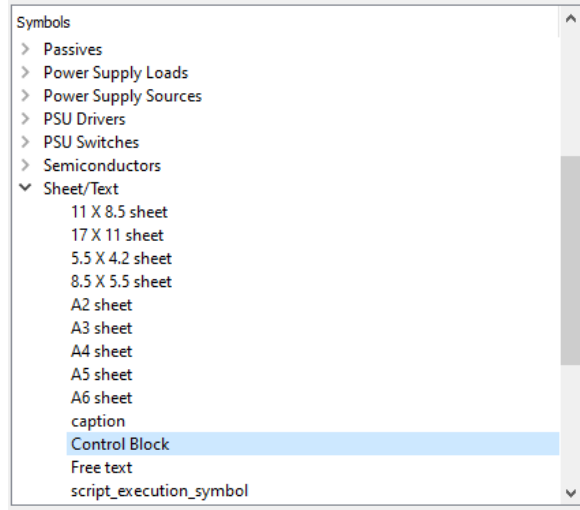
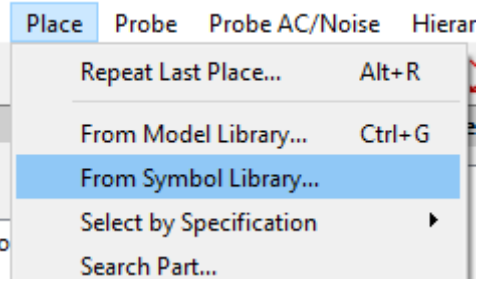
```

Saves all currents and all voltages at all levels

Agenda

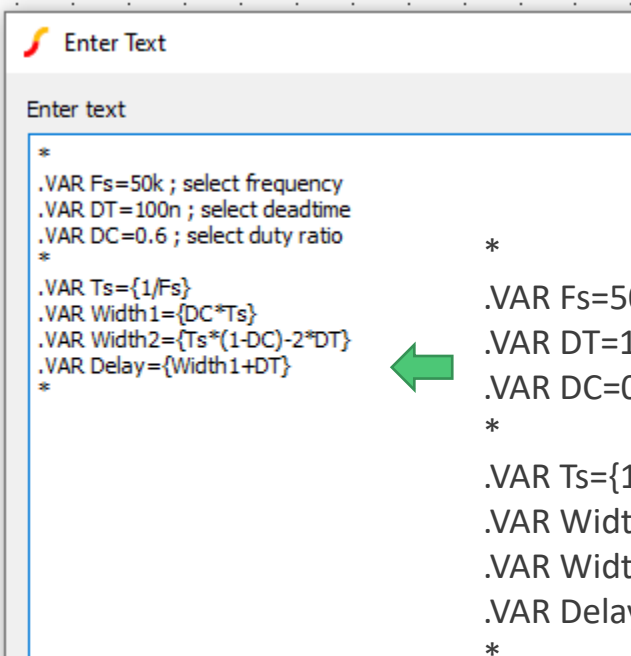
- What is Elements?
- Running a Basic Simulation
- **A Buck Converter**
- Ac Linear Analysis with SIMPLIS
- Importing a SPICE Model
- The Ready-to-Use Template

Passing parameters from the schematic with a control block



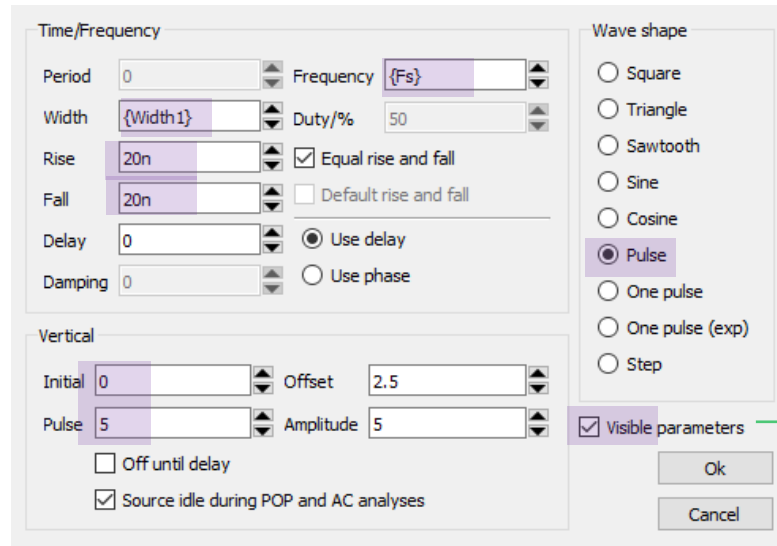
Double click in the zone

Text



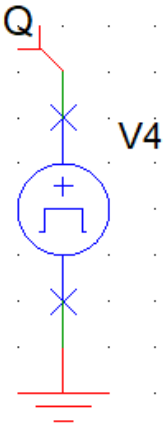
Copy/paste

Now bring a generator by pressing W:



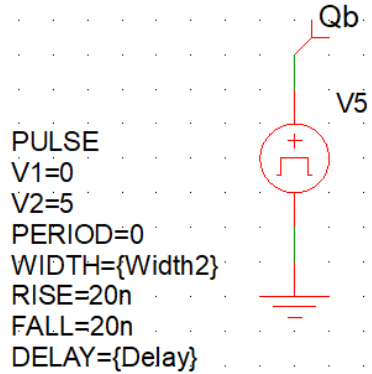
```

PULSE
V1=0
V2=5
PERIOD=0
WIDTH={Width1}
RISE=20n
FALL=20n
DELAY=0
    
```



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

Do the same for a second source



Edit Waveform

Time/Frequency

Period: 0 Frequency: {Fs}

Width: {Width2} Duty/%: 48

Rise: 20n Equal rise and fall

Fall: 20n Default rise and fall

Delay: {Delay} Use delay

Damping: 0 Use phase

Wave shape

Square

Triangle

Sawtooth

Sine

Cosine

Pulse

One pulse

One pulse (exp)

Step

Vertical

Initial: 0 Offset: 2.5

Pulse: 5 Amplitude: 5

Off until delay

Source idle during POP and AC analyses

Visible parameters

Ok Cancel

Press F8

Choose SIMPLIS Analysis

Periodic Operating Point AC Transient

Select analysis

POP

AC

Transient

Save options

All

Voltages Only

Probes Only

Analysis parameters

Stop time: 200u s

Start saving data at t = 0 s Default

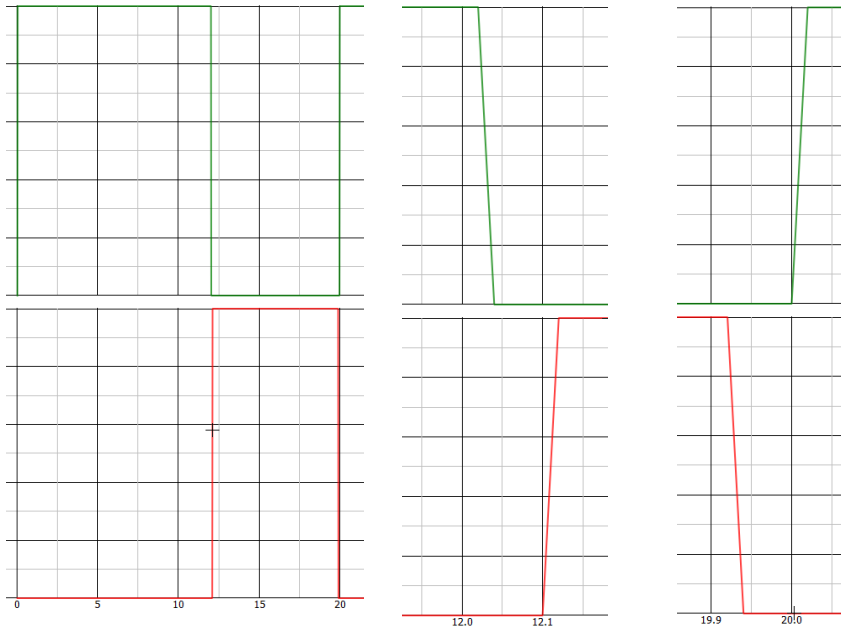
Plot data output

Number of plot points: 10k Default

Advanced...

Ok Run Cancel Help

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

```

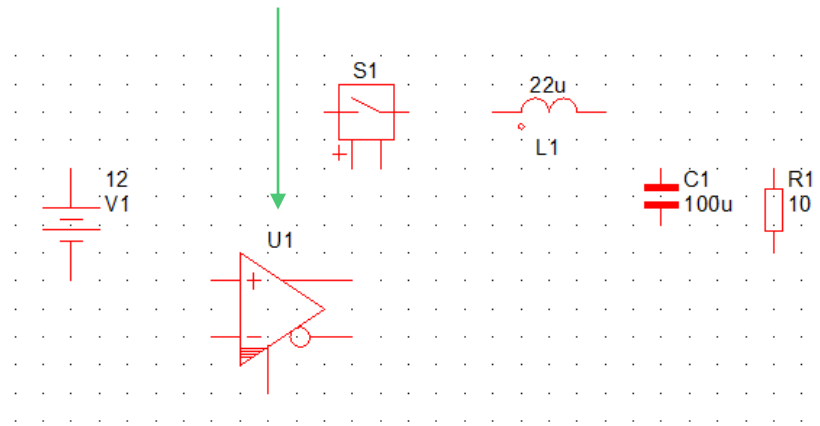
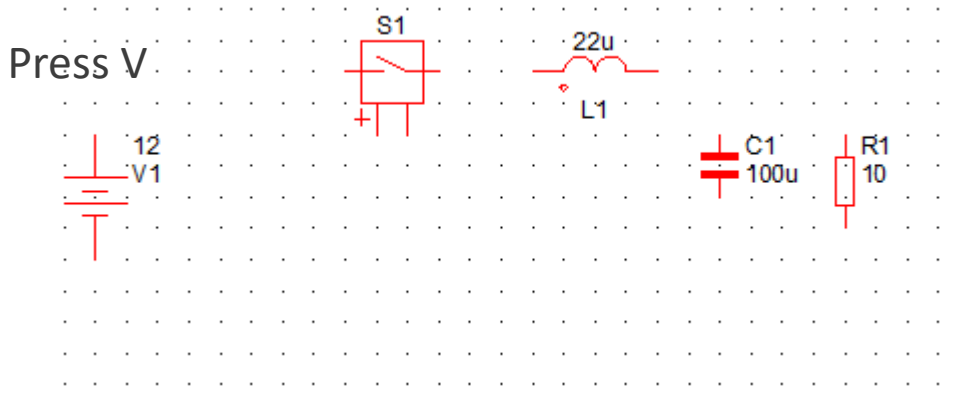
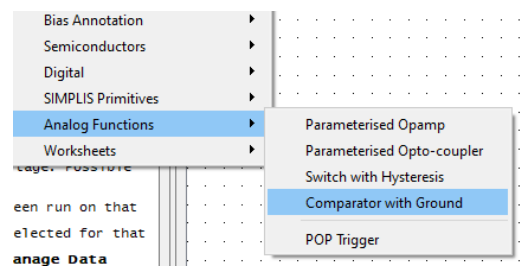
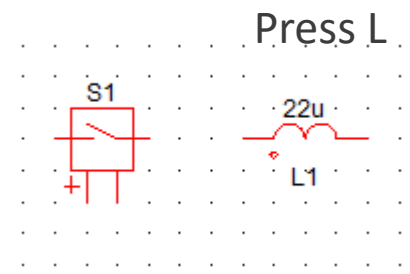
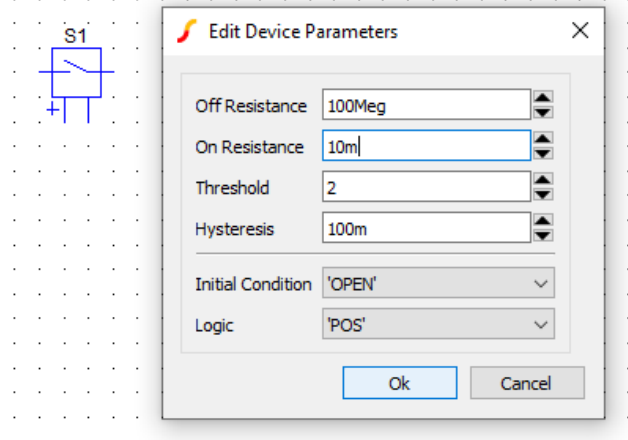
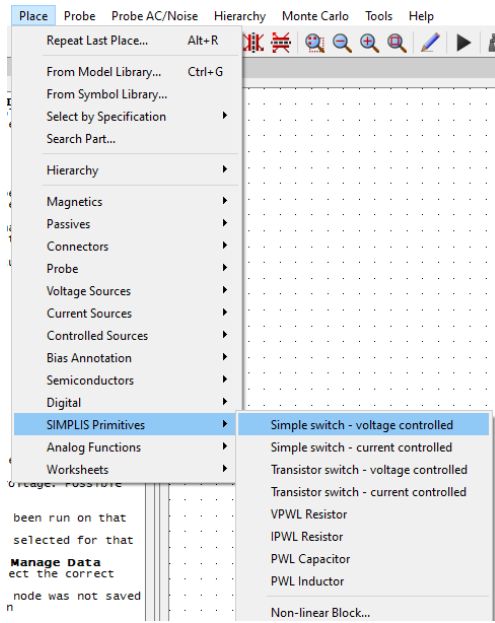
1 .simulator SIMPLIS
2 .options
3 + PSP_NPT=10001
4 + POP_ITRMAX=20
5 + POP_OUTPUT_CYCLES=5
6 + SNAPSHOT_INTVL=0
7 + SNAPSHOT_NPT=11
8 + MIN_AVG_TOPOLOGY_DUR=1a
9 + AVG_TOPOLOGY_DUR_MEASUREMENT_WINDOW=128
10 .tran 200u 0
11
12 *
13 .VAR Fs=50k ; select frequency
14 .VAR DT=100n ; select deadtime
15 .VAR DC=0.6 ; select duty ratio
16 *
17 .VAR Ts={1/Fs}
18 .VAR width1={DC*Ts}
19 .VAR width2={Ts*(1-DC)-2*DT}
20 .VAR Delay={width1+DT}
21 *
22
23 .simulator DEFAULT
24

```

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 1/2-bridge control.

Let' simulate a simple buck converter



Common sources

- Bias Annotation
- Semiconductors
- Digital
- SIMPLIS Primitives**
- Analog Functions
- Worksheets

Simple switch - voltage controlled

Simple switch - current controlled

Transistor switch - voltage controlled

Transistor switch - current controlled

VPWL Resistor

IPWL Resistor

PWL Capacitor

PWL Inductor

Non-linear Block...

been run on that selected for that

Manage Data

ct the correct

node was not saved

	Voltage	Current
1	0	0
2	1	1
3	2	2
4		

Entry mode: Arbitrary Symmetric

Initial segment: 1

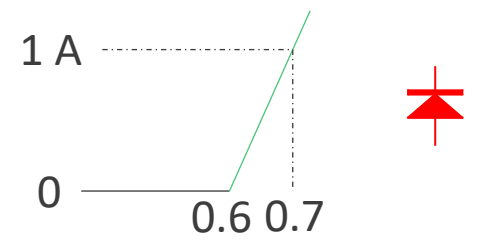
Press the insert key to insert a row or rows. Press the delete key to delete the selected rows.

Ok Paste Cancel Help



Define VPWL Resistor: R2

	Voltage	Current
1	0	0
2	0.6	1p
3	0.7	1
4		



Edit Waveform

Time/Frequency

Period: 10u Frequency: 100k

Width: 0 Duty/%: 100

Rise: 10u

Fall: 0

Delay: 0

Damping: 0

Wave shape

Sawtooth

Square

Triangle

Sine

Cosine

Pulse

One pulse

One pulse (exp)

Step

Vertical

Initial: 0 Offset: 500m

Pulse: 1 Amplitude: 1

Ok Cancel

Choose SIMPLIS Analysis

Periodic Operating Point AC Transient

Analysis parameters

Stop time: 1m s

Start saving data at t = 0 s

Plot data output

Number of plot points: 10k

Select analysis

POP

AC

Transient

Save options

All

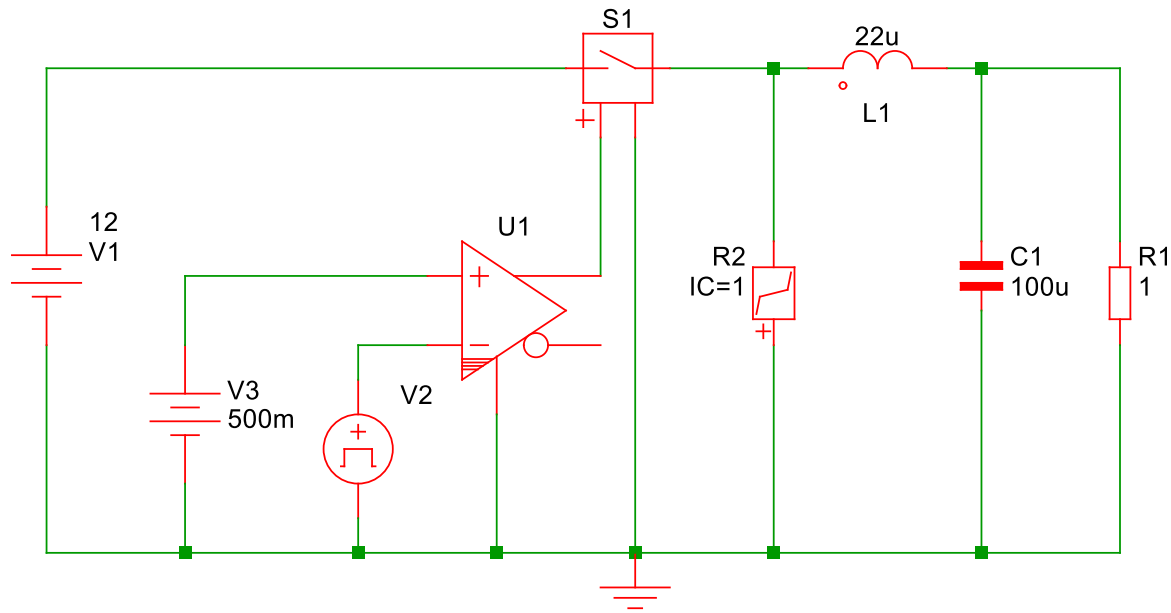
Voltages Only

Probes Only

Tick

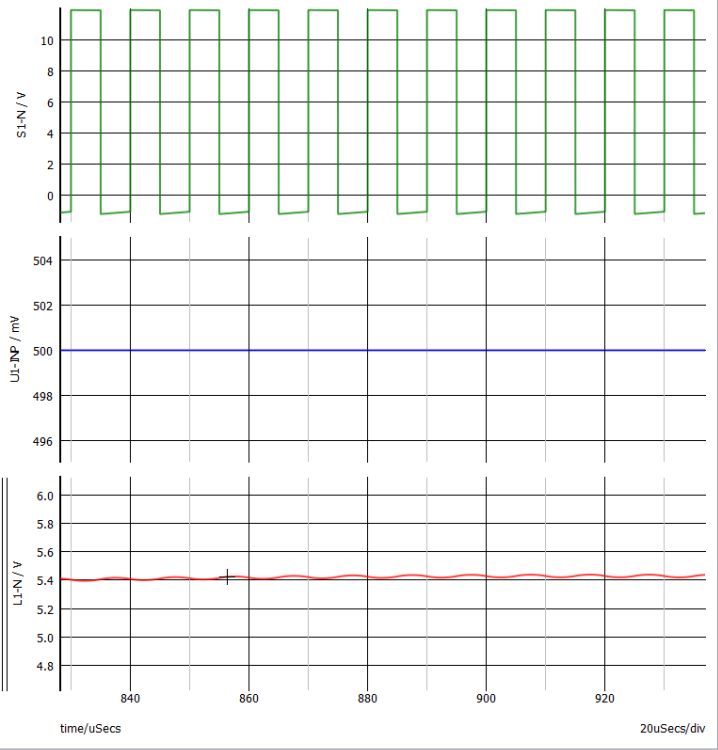
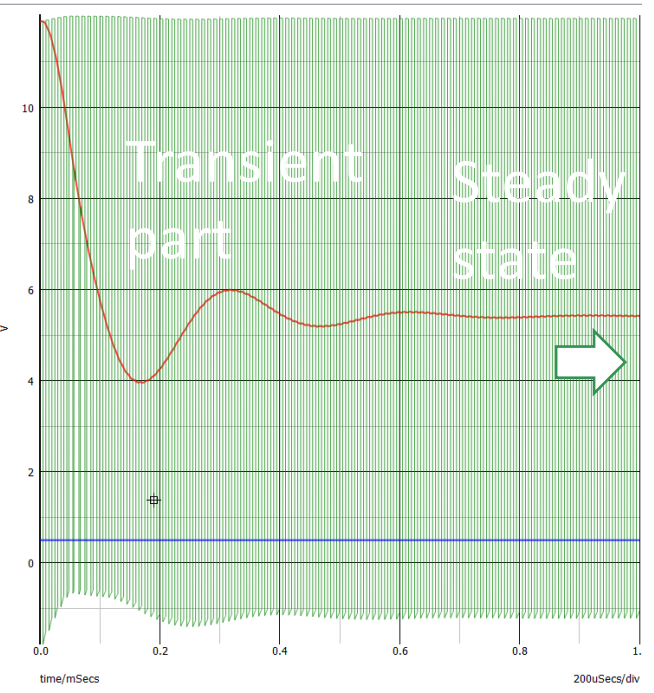
Ok Run Cancel Help

The 1-V sawtooth generator



- Edit Part...
- Edit Additional Parameters...
- Hide/Show Value
- Descend Hierarchy
- Probe Voltage...**
- Probe Current...
- Display Device Bias Info
- Disable Selected
- Disable & Short-circuit Selected
- Enable Selected
- Copy
- Paste
- Edit/Add Properties...
- Delete Properties...
- Move Text
- Edit Symbol
- Update Symbol
- Restore Properties...
- Plot PWL points for R, L, TX, or C
- View Model

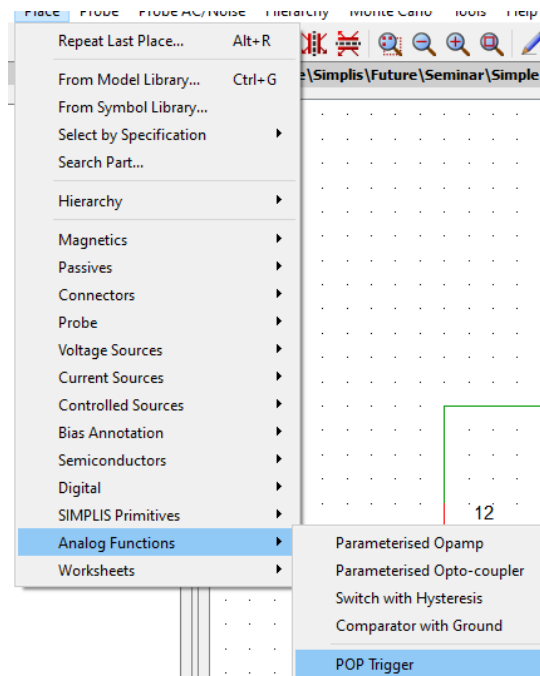
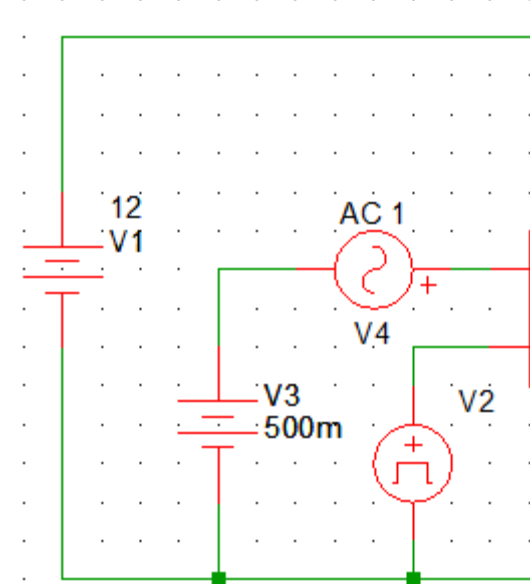
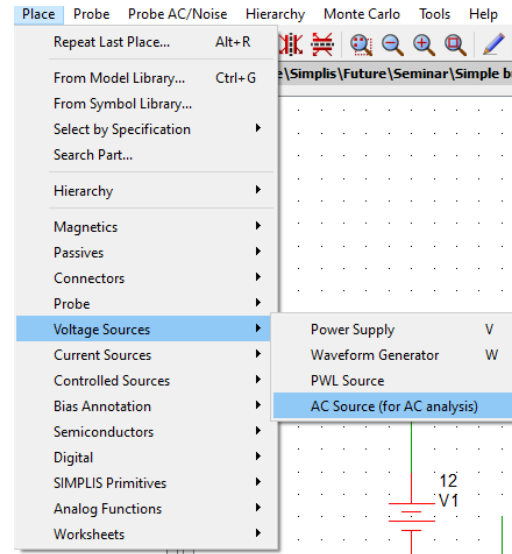
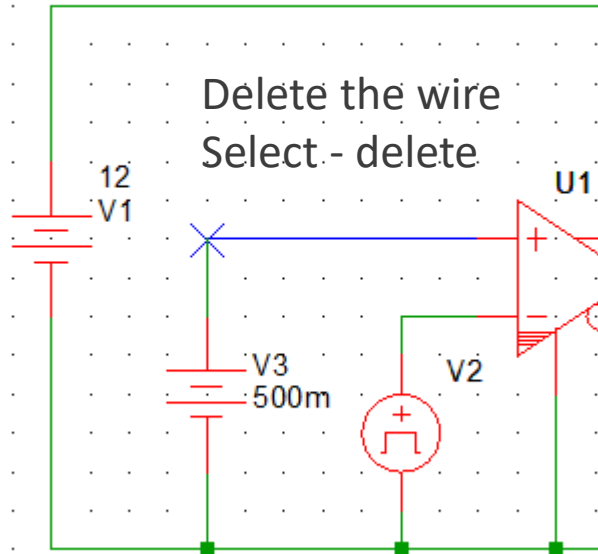
You can probe all nodes as you want and stack curves in the end



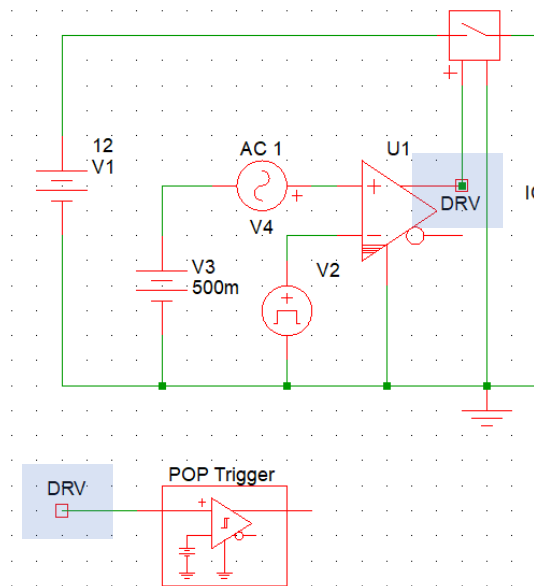
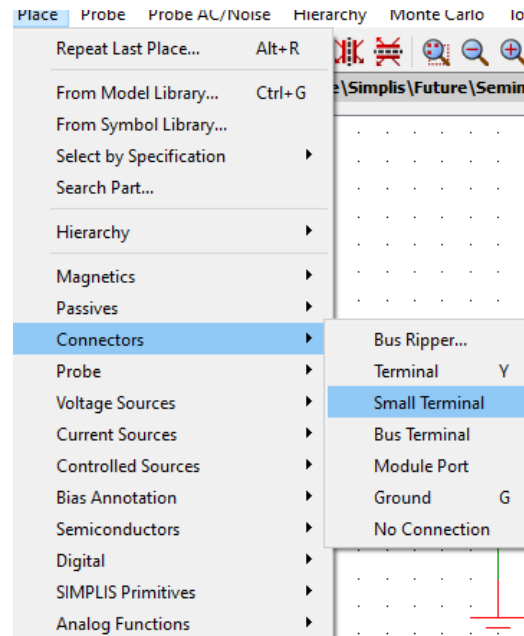
Stack and zoom-in

Label	Legend	Curve label	Name	Value
<input type="checkbox"/> L1-N	—			
<input type="checkbox"/> S1-N	—			
<input type="checkbox"/> U1-INP	—			

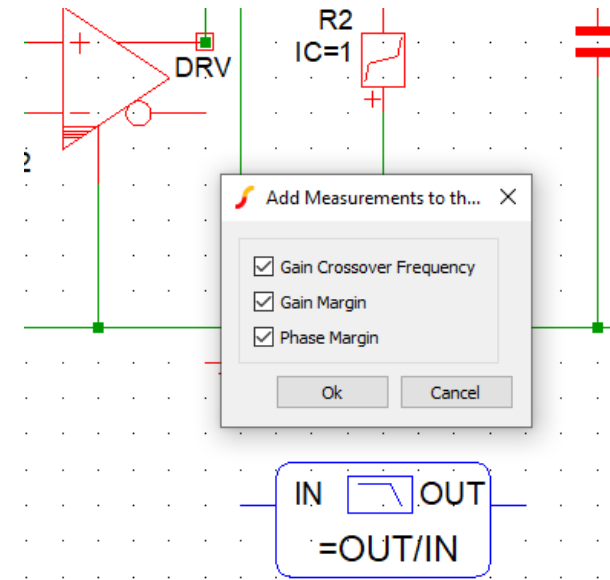
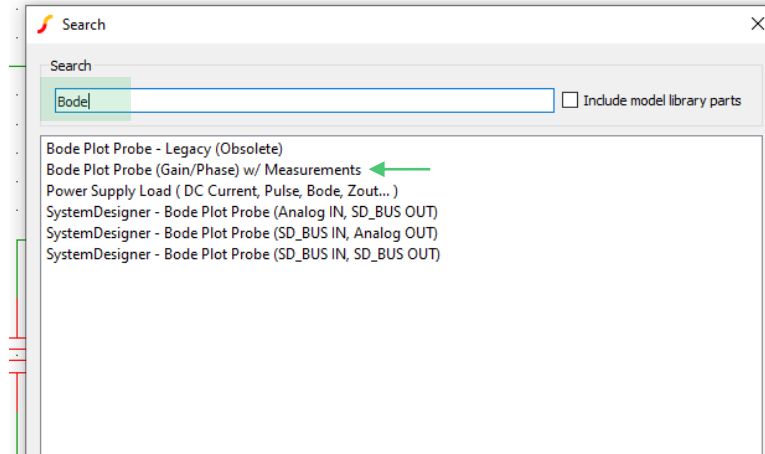
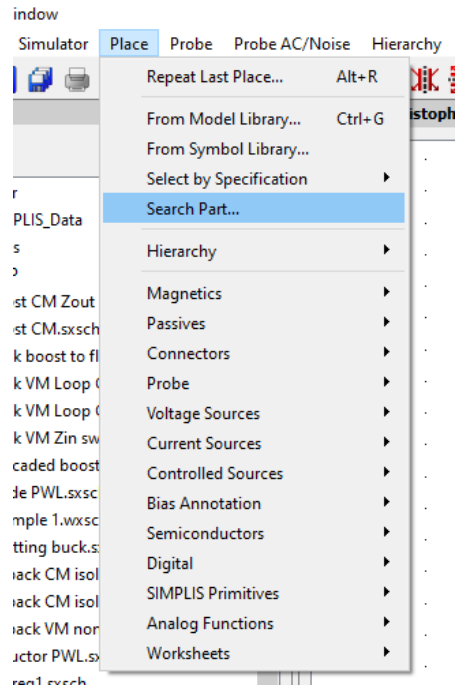
The strength of SIMPLIS is to extract the small-signal response from this switching circuit



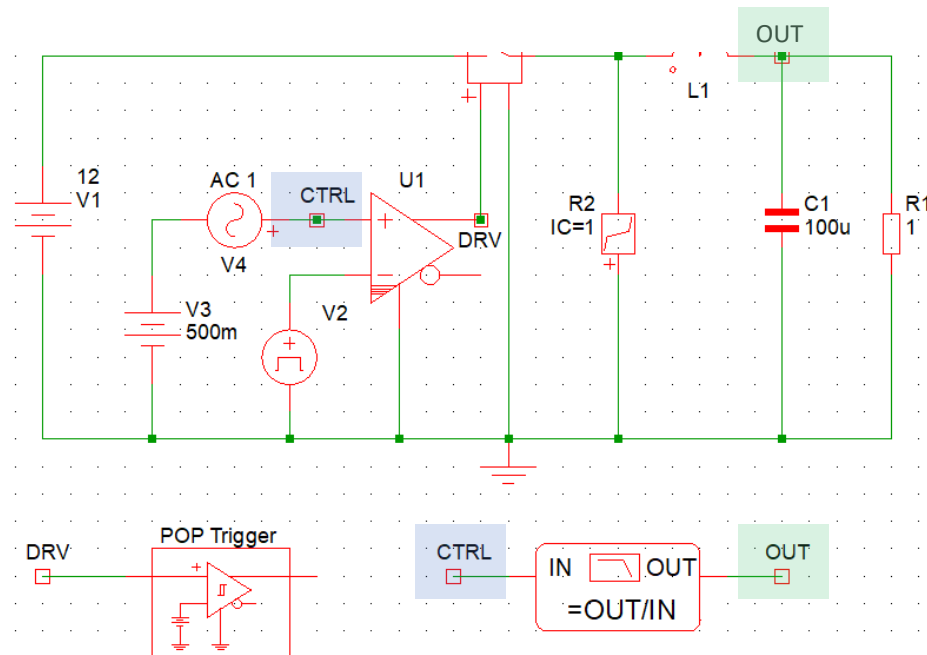
The POP for Periodic Operating Point instructs SIMPLIS on switching event



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

Curves

- Use separate graph
- Graph name: Control-to-Output
- Set tab/caption to name

Gain

- Labels: Curve label: Gain, Y axis label:
- Vertical scale: dB, Linear
- Vertical axis (check to auto calculate): Maximum limit: 40, Minimum limit: -40
- Grid spacing: 20
- Colour: Use default, Edit...

Phase

- Labels: Curve label: Phase, Y axis label:
- Curve: Phase, Phase - 180 degrees
- Vertical axis (check to auto calculate): Maximum limit: 180, Minimum limit: -180
- Grid spacing: 45
- Colour: Use default, Edit...

Use **Phase -180 degrees** to plot the phase curve with 180 degree offset

Display curves on

- Single grid
- Two grids

Vertical order

- Phase above Gain
- Gain above Phase

Example curve output

Black magnitude

Red phase

Configuration

Save Configuration

Save current properties as default Bode Plot Probe configuration. New probes will automatically have these saved properties applied when placed.

Ok Cancel Help

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

Select Color

Basic colors

Pick Screen Color

Custom colors

Add to Custom Colors

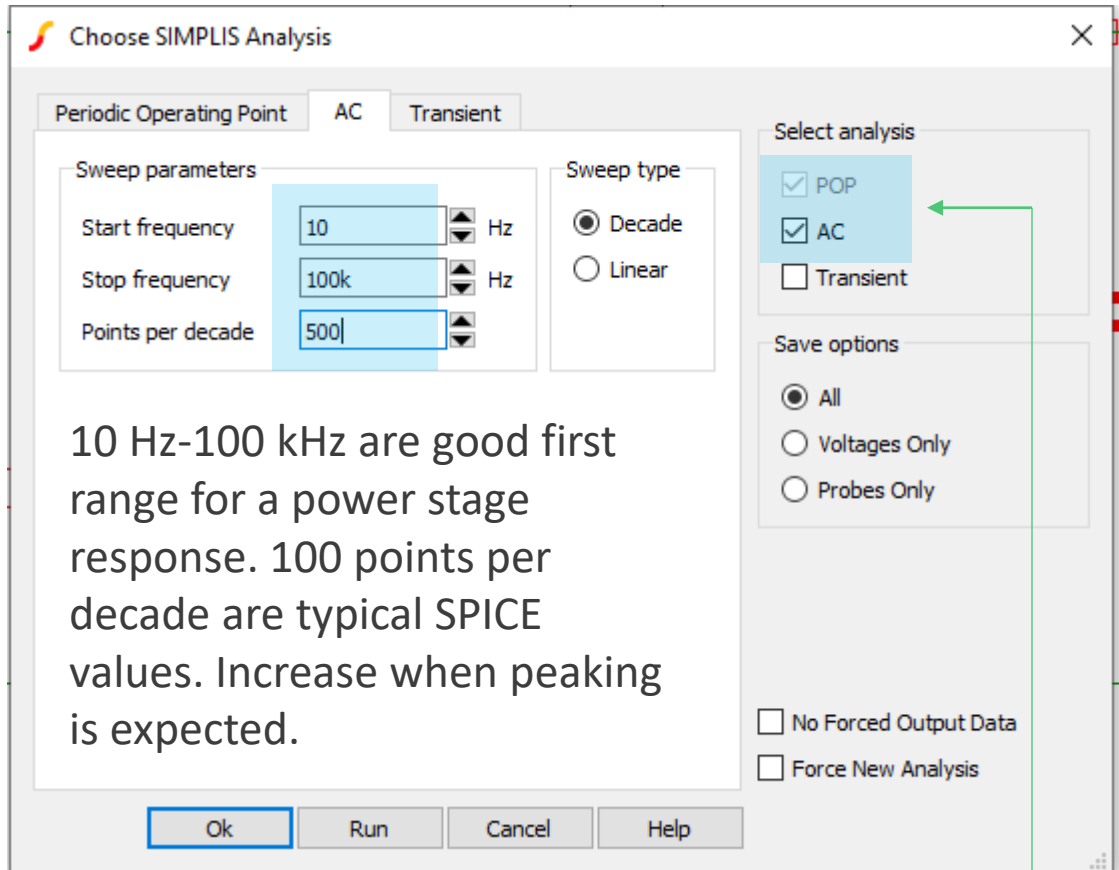
Hue: 0 Red: 0
Sat: 0 Green: 0
Val: 0 Blue: 0

HTML: #000000

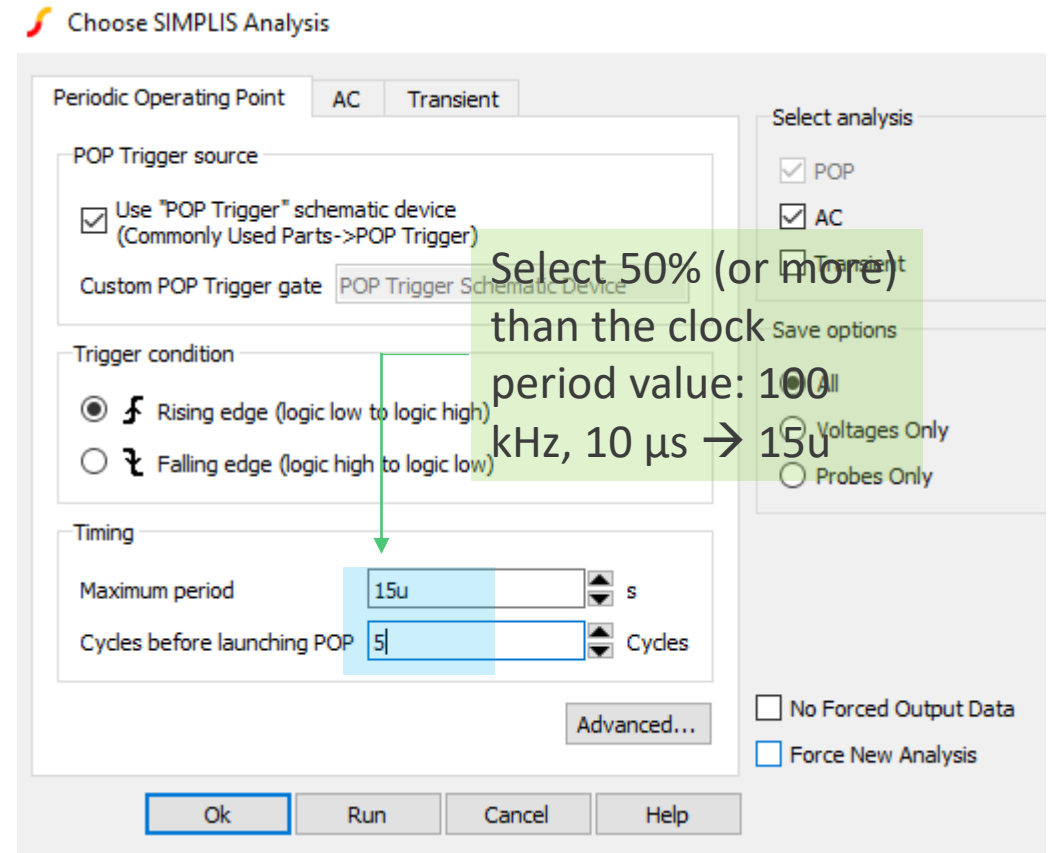
OK Cancel

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

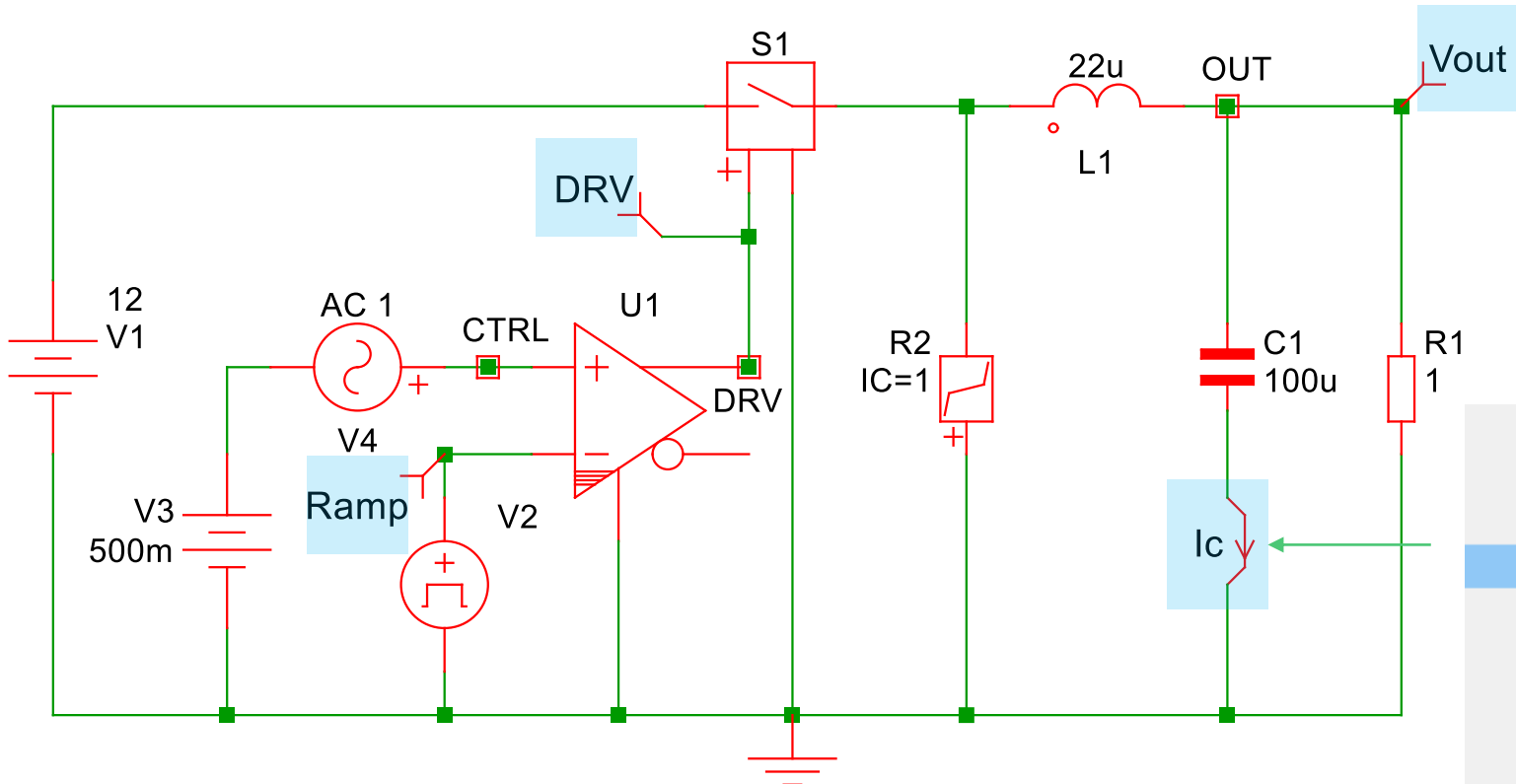
Press F8 to access the simulation parameters



Tick ac analysis, POP will be checked automatically

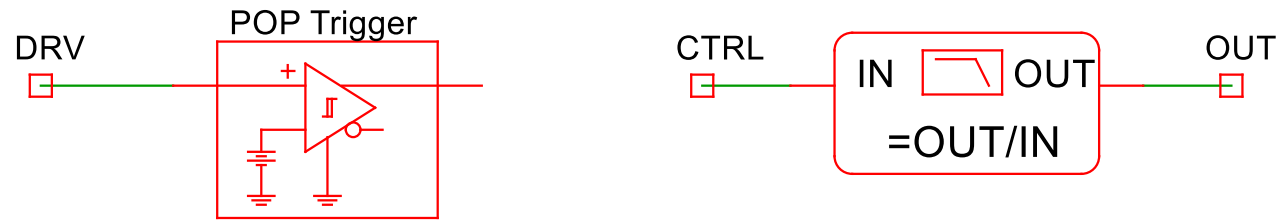


The periodic operating point or POP is a SIMPLIS proprietary algorithm which determines the steady-state operating point (average currents in capacitor and average voltages across inductors are all zero) in a record time. When POP is finished, the program launches the ac analysis.

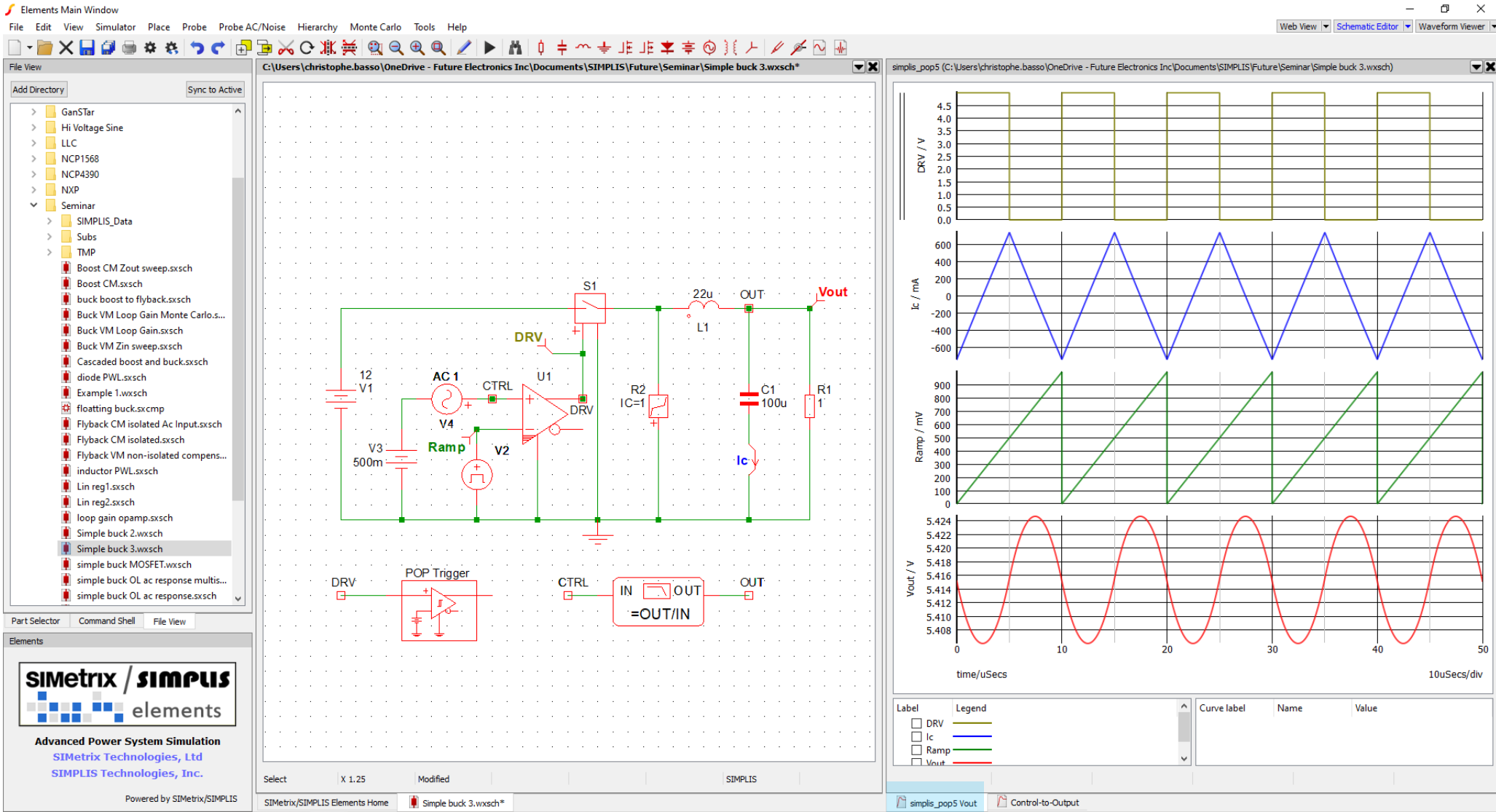


Add a few probes to visualize the waveforms after steady-state operation is done.

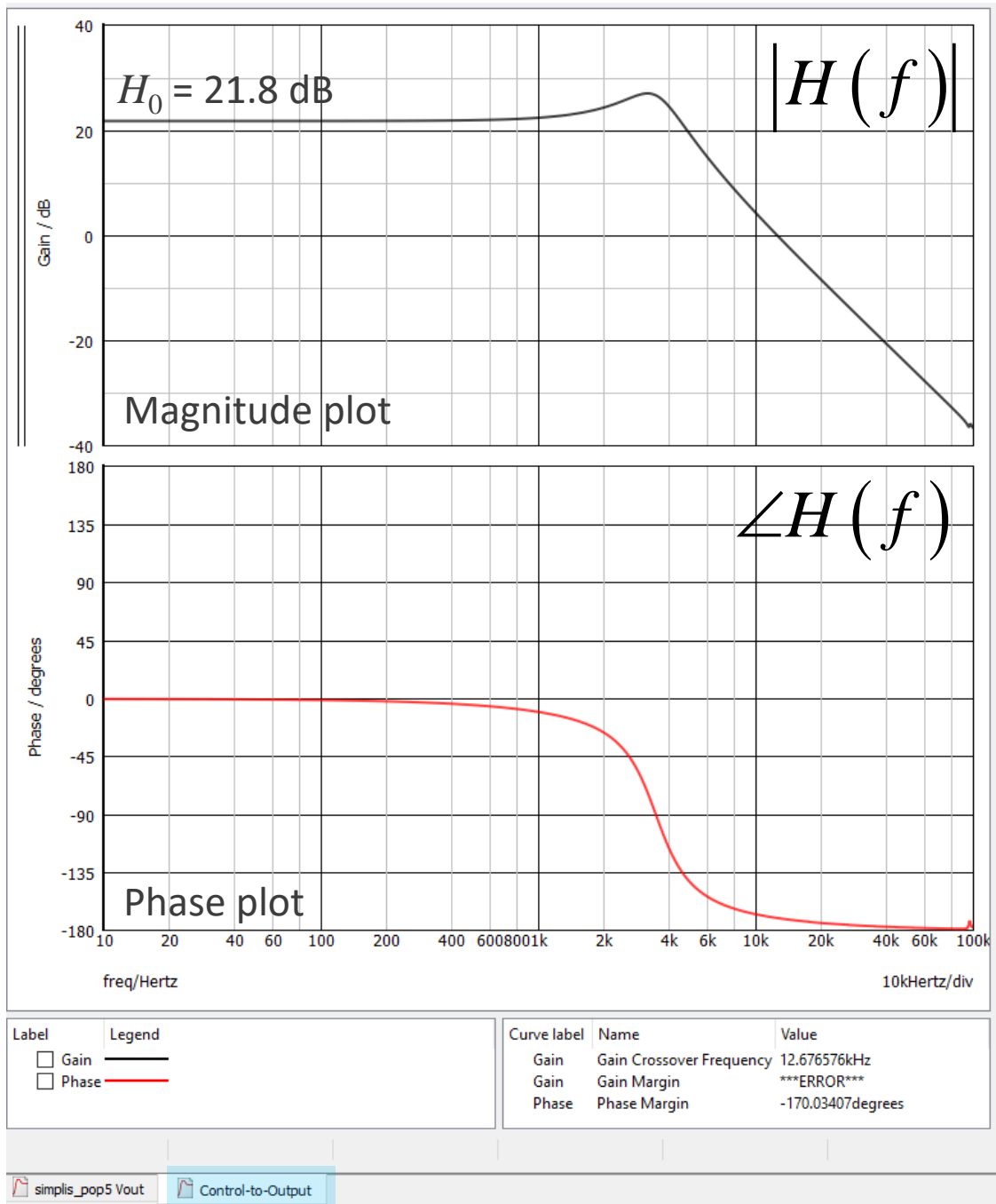
- Magnetics ▶
- Passives ▶
- Connectors ▶
- Probe ▶**
 - Voltage Probe B
 - Current Probe U
 - Power Probe
 - Inline Current Probe**
 - Differential Voltage Probe
 - Bus Probe
 - XY Probe
 - Delete All Probes
- Voltage Sources ▶
- Current Sources ▶
- Controlled Sources ▶
- Bias Annotation ▶
- Semiconductors ▶
- Digital ▶
- SIMPLIS Primitives ▶
- Analog Functions ▶



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:

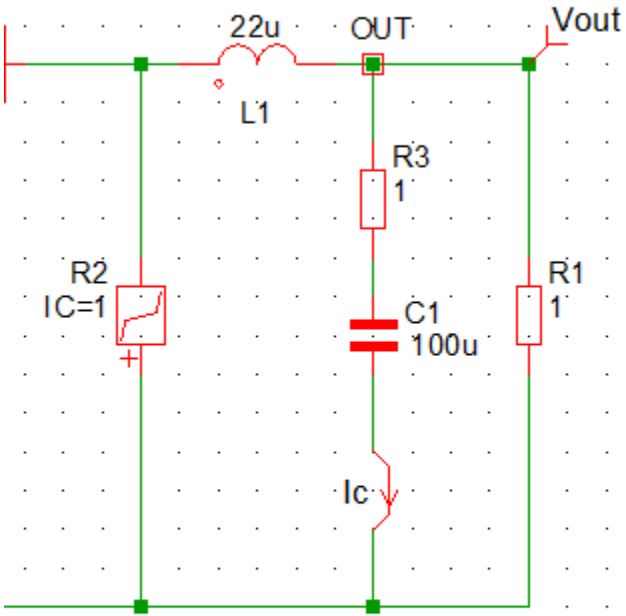
$$H(s) = H_0 \frac{N(s)}{1 + \frac{s}{\omega_0 Q} + \left(\frac{s}{\omega_0}\right)^2}$$

Input voltage, 12 V

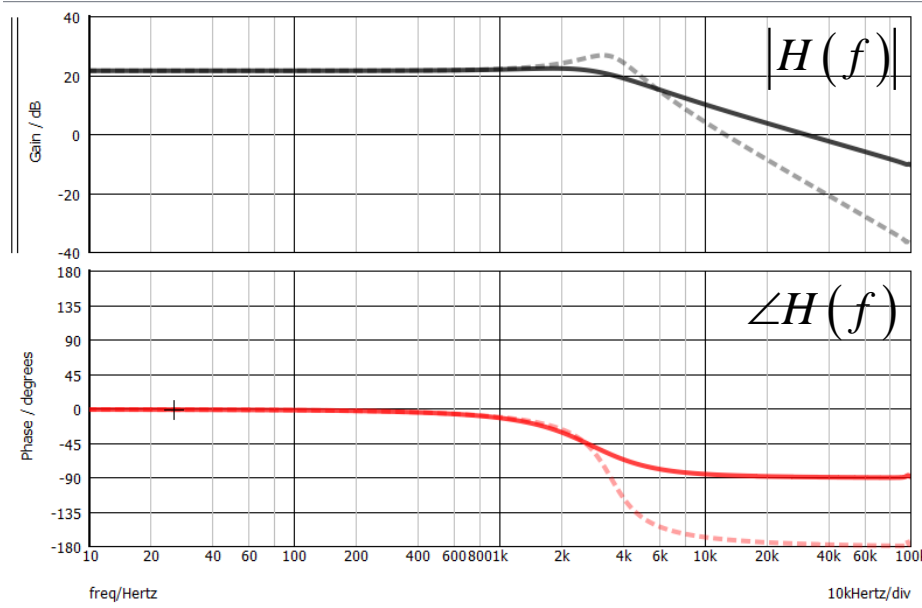
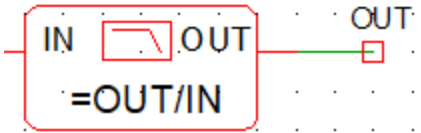
$$H_0 = \frac{V_{in}}{V_p} \frac{R_L}{R_L + r_L} \approx \frac{V_{in}}{V_p} \Rightarrow H_0 = 20 \log \frac{12 \text{ V}}{1 \text{ V}} \approx 21.6 \text{ dB}$$

Peak amplitude of the sawtooth

Select a remanence of 2



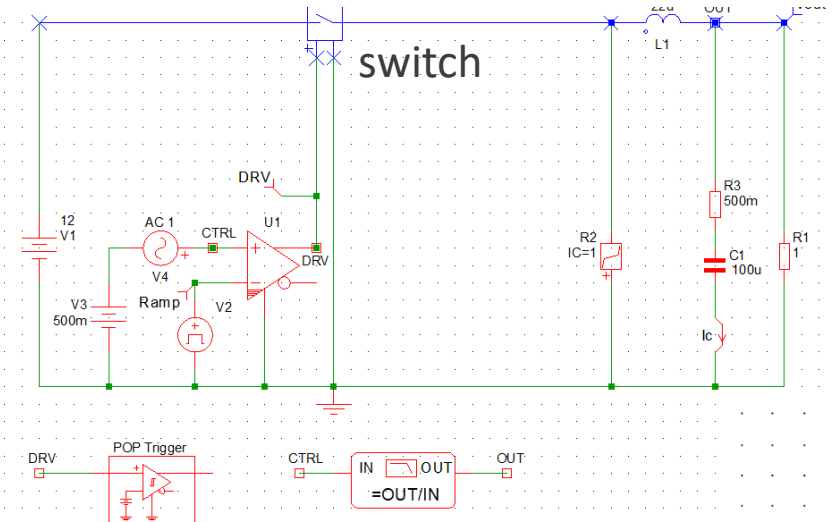
Insert a resistor in the schematic



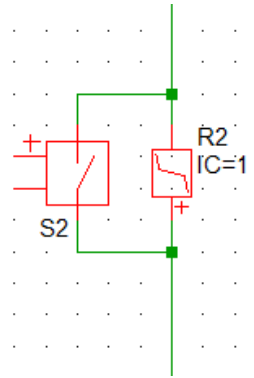
Change the series resistance value and see the different responses. The dashed lines are the first results.

Let's add some synchronous rectification

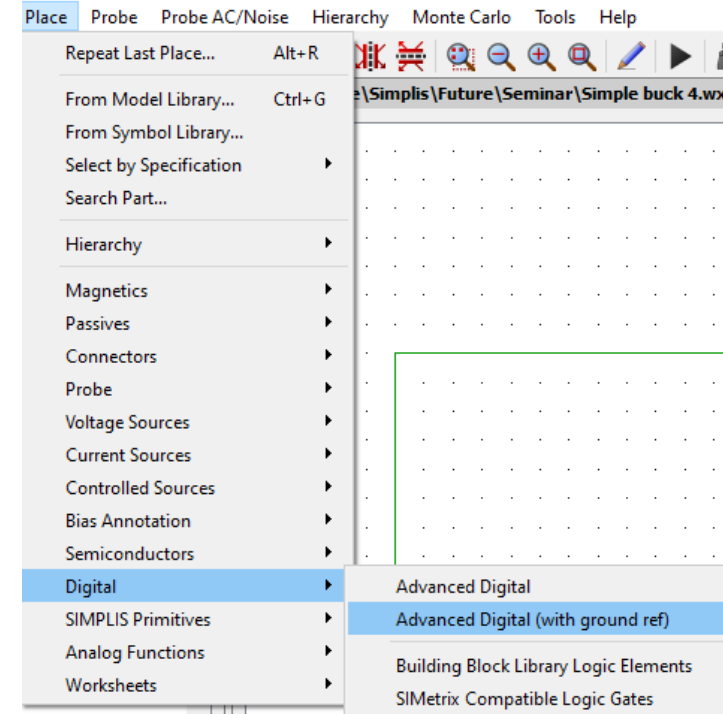
Make some space first



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

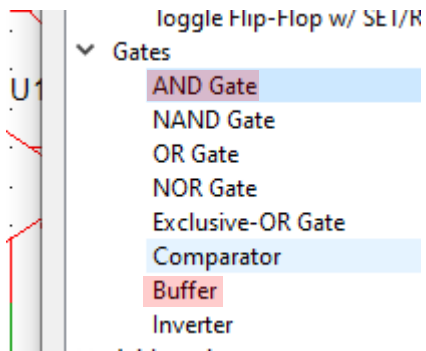
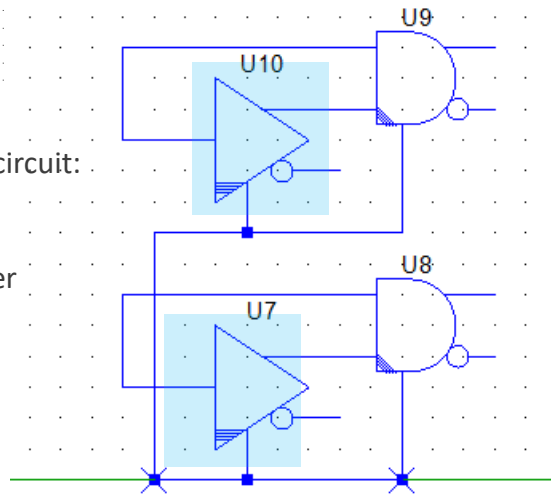


Grab a few gates to implement deadtime

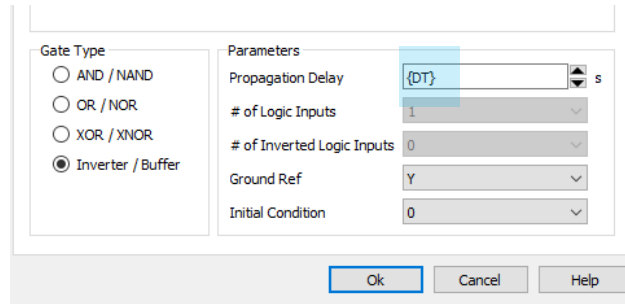
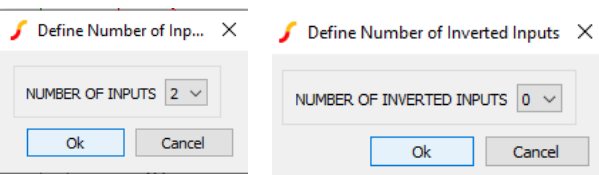
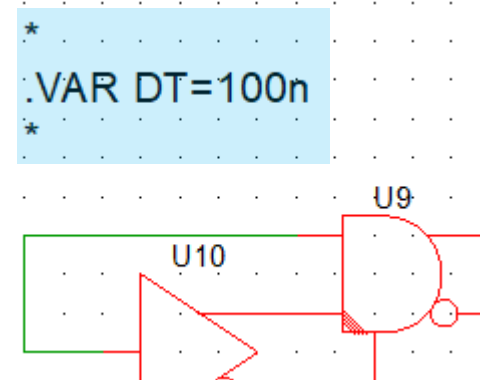


Capture this circuit:

Double-click on each buffer



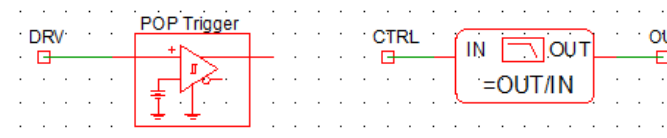
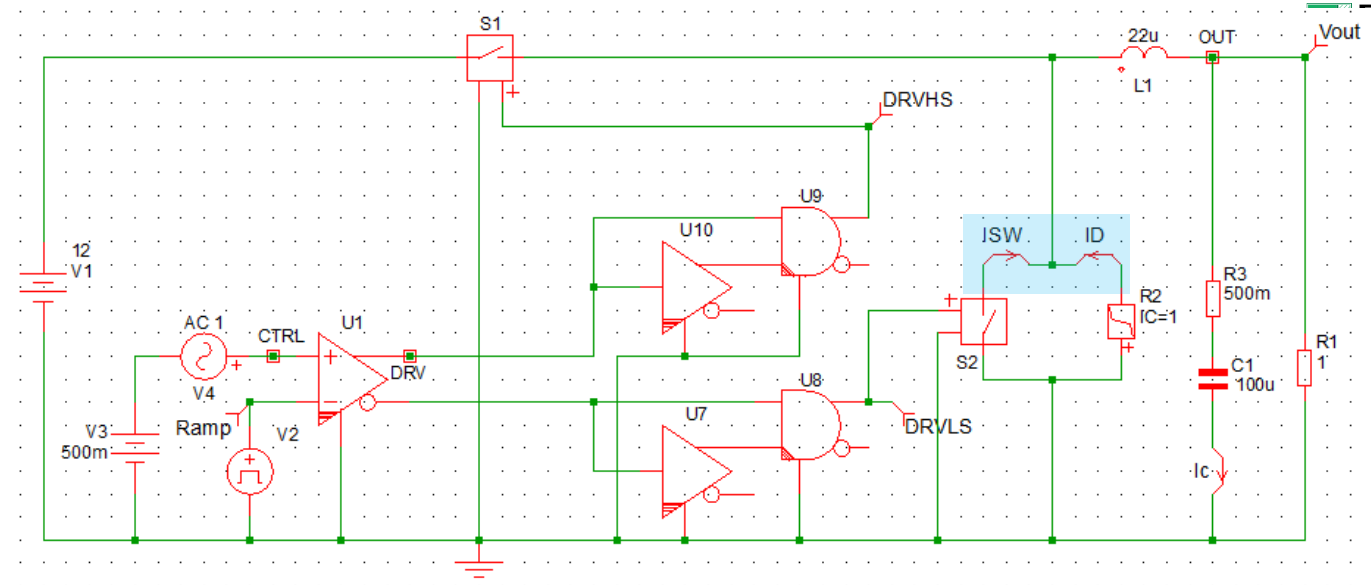
Insert a control block



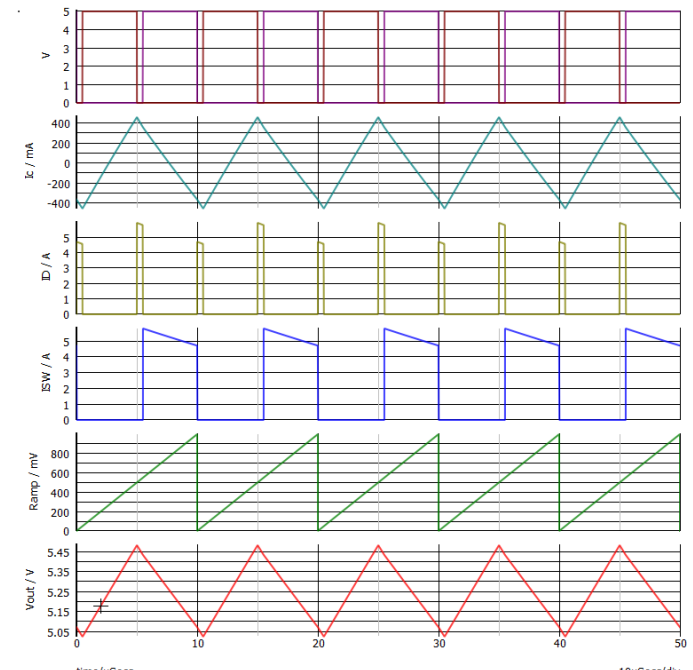
Finish the wiring and insert two current probes

Place Probe Probe AC/Noise Hierarchy Monte Carlo Tools Help

- Repeat Last Place... Alt+R
- From Model Library... Ctrl+G
- From Symbol Library...
- Select by Specification
- Search Part...
- Hierarchy
- Magnetics
- Passives
- Connectors
- Probe**
 - Voltage Probe B
 - Current Probe U
 - Power Probe
 - Inline Current Probe**
 - Differential Voltage Probe
 - Bus Probe
 - XY Probe
 - Delete All Probes
- Voltage Sources
- Current Sources
- Controlled Sources
- Bias Annotation
- Semiconductors
- Digital
- SIMPLIS Primitives
- Analog Functions
- Worksheets



* VAR DT=500n *



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

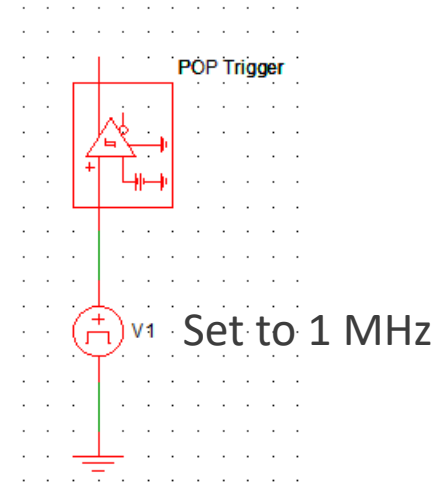
Agenda

- 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

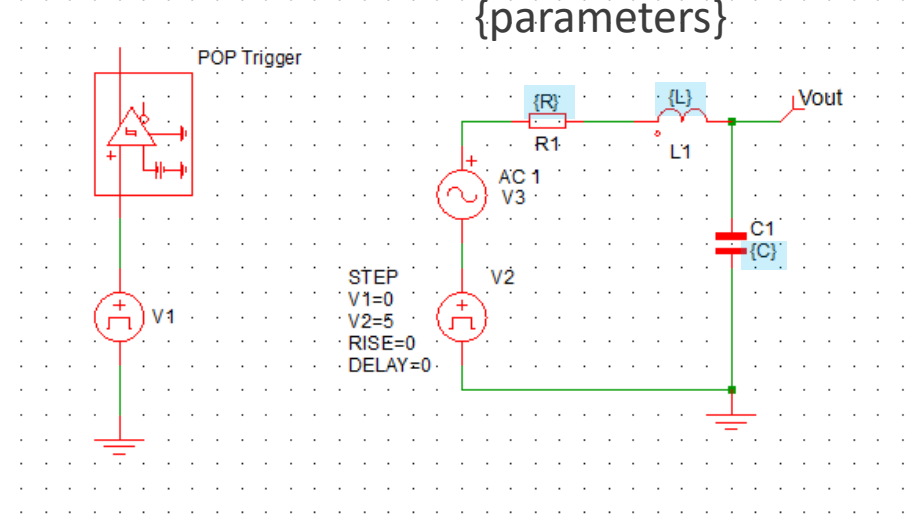
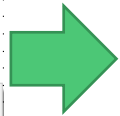
SIMPLIS is a time-domain simulator therefore switching is needed to perform an ac analysis

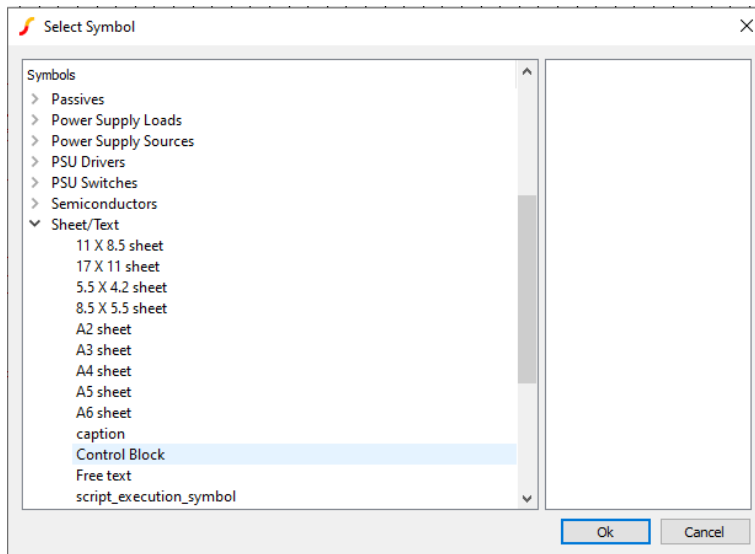
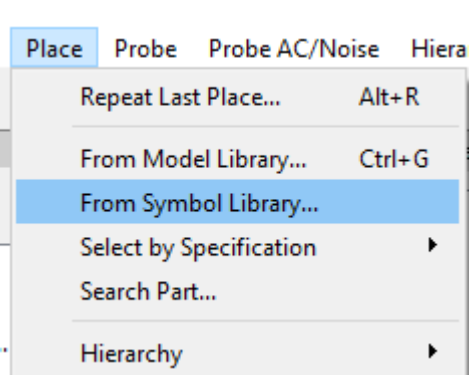
→ “cheat” the simulator with a simple clock circuit

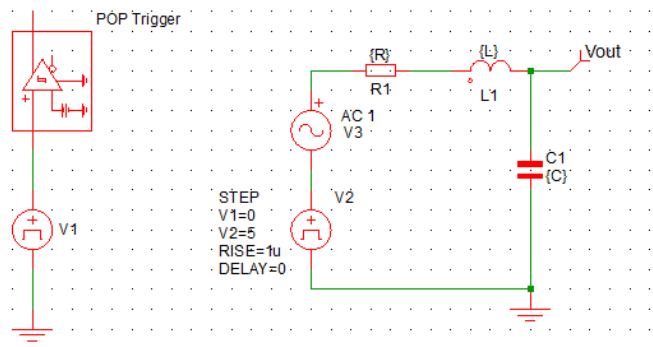
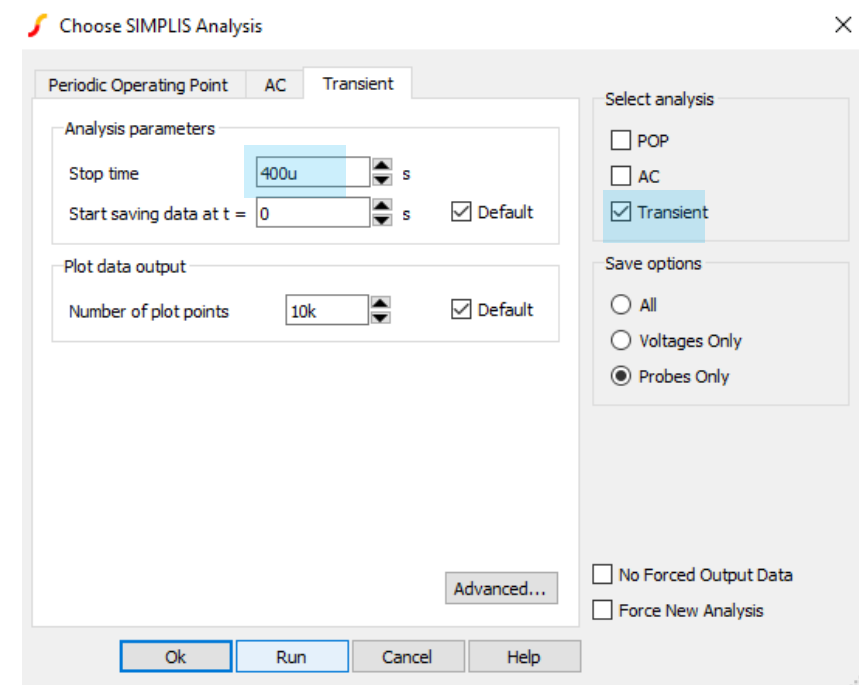
Add R, L and C with {parameters}

Place another waveform gen



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



```
*
.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}
{**}
```

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)}
*
```

Curly braces

```
{**}
{**} L = {L}
{**} R = {R}
{**} C = {C}
{**} Q1 = {Q1}
{**} Dzeta = {Dzeta}
{**}
```

Add these lines to display the calculated values

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

Probe Options Axis Scales Axis Labels

Curve label

 Use \$FREF\$ for hierarchical reference

Colour
 Use default Edit...

Axis type
 Auto select
 Use dedicated grid
 Use named grid
 Use named Y-axis
 Digital
 Axis name

Display order
 Arbitrary string to specify order

History
 History depth default
 Use separate curves. If enabled, a new curve is created for each new run and history depth is ignored.
 On
 Off
 Global default may be set from menu Use default [OFF]
 File | Options | General...

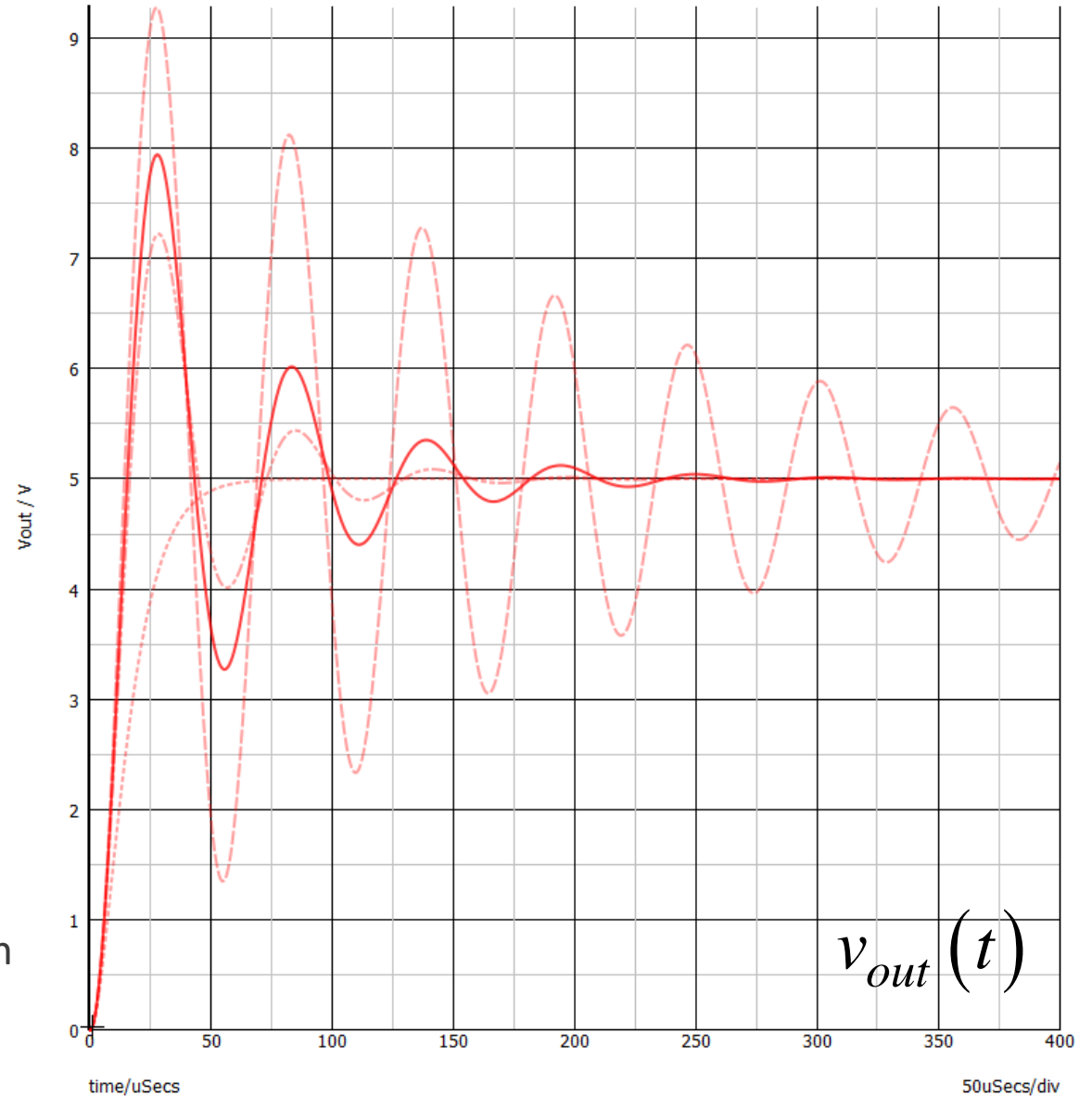
Graph
 Use named graph
 Graph name
 Set tab/caption to name
 Keep different analysis types on same graph

Analyses
 All analyses disabled
 Transient
 POP
 AC sweep
 Plot on completion only

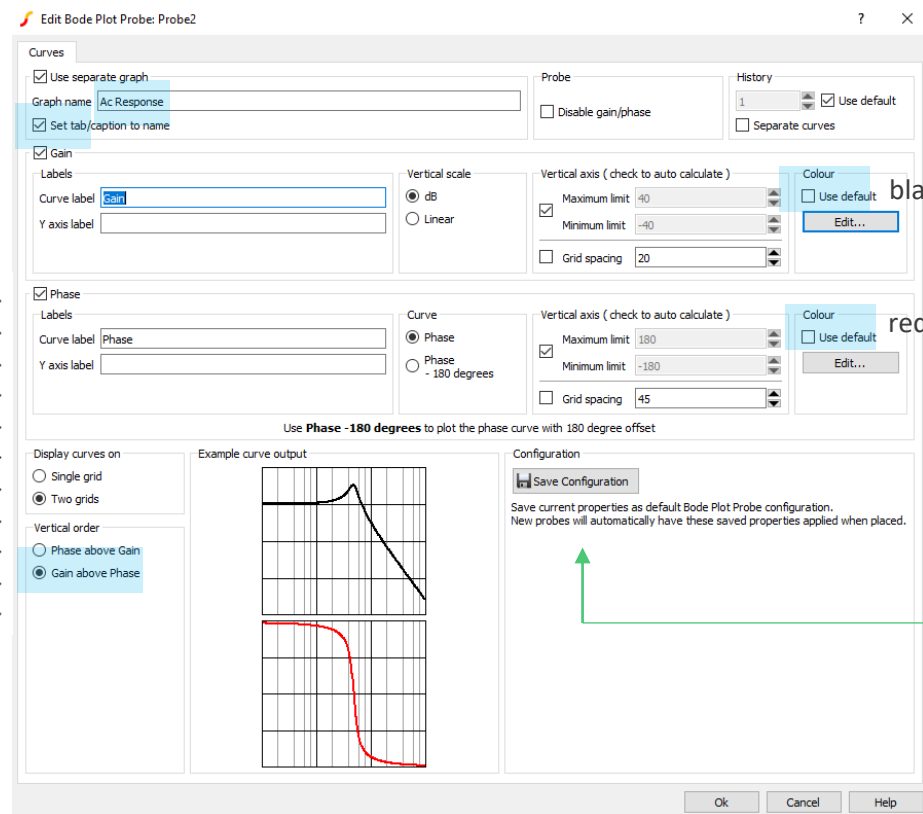
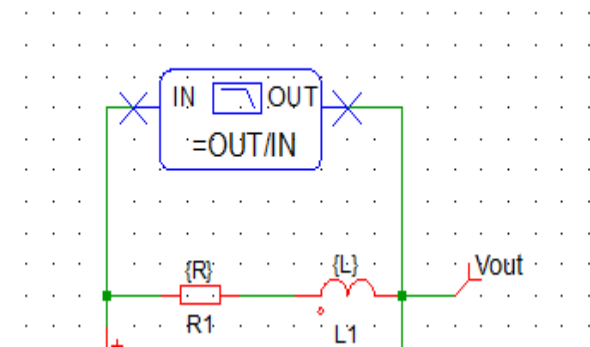
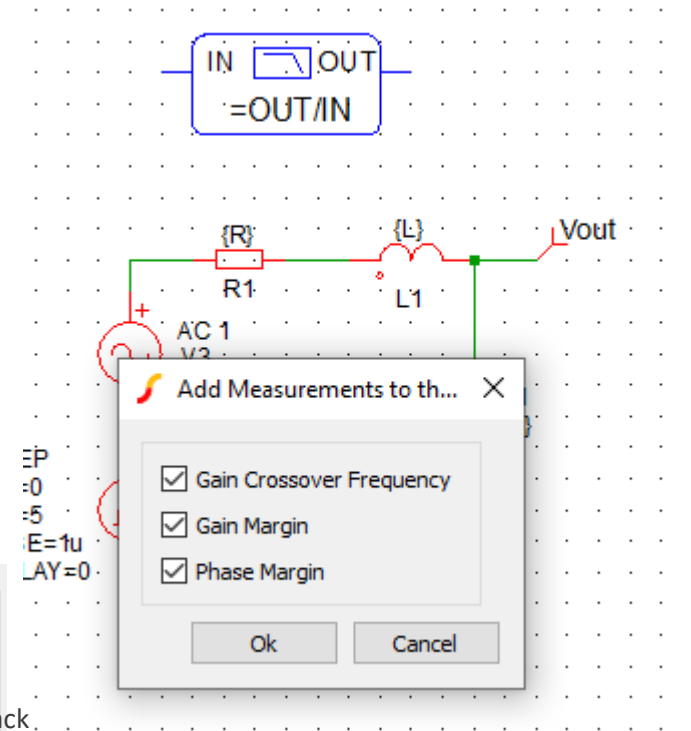
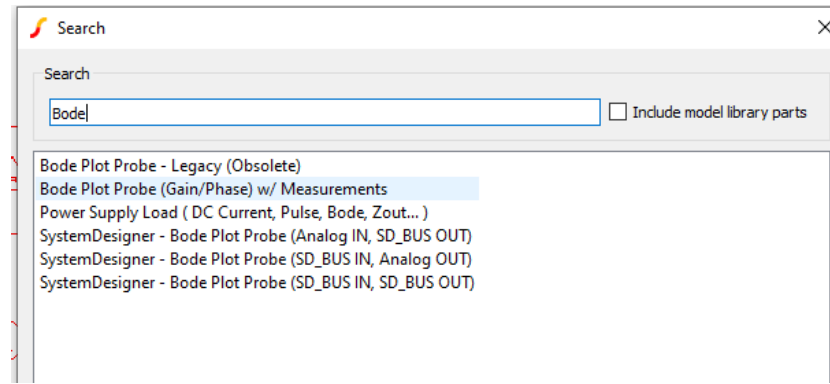
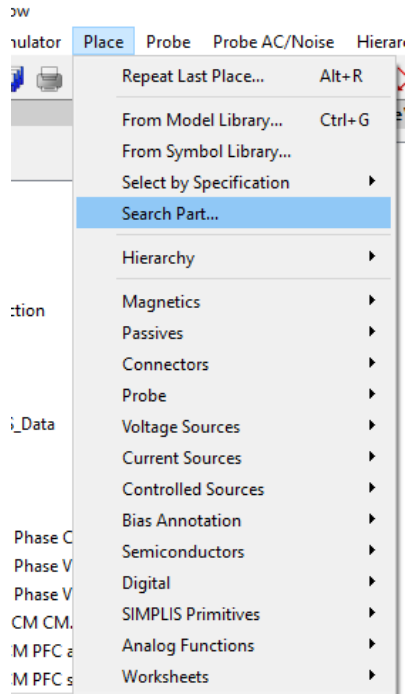
Ok Cancel Help

```
*
.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:



black.

red

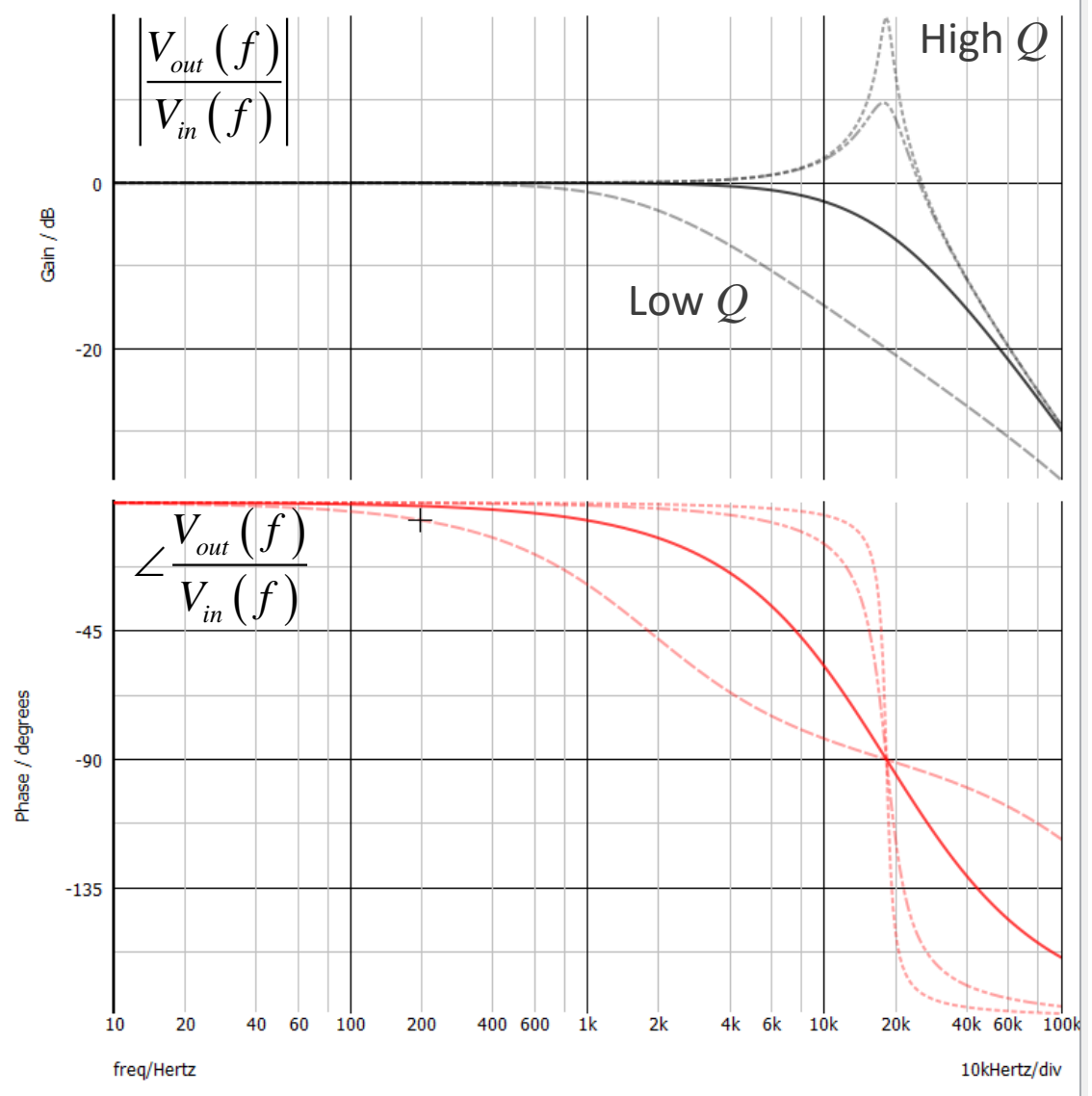
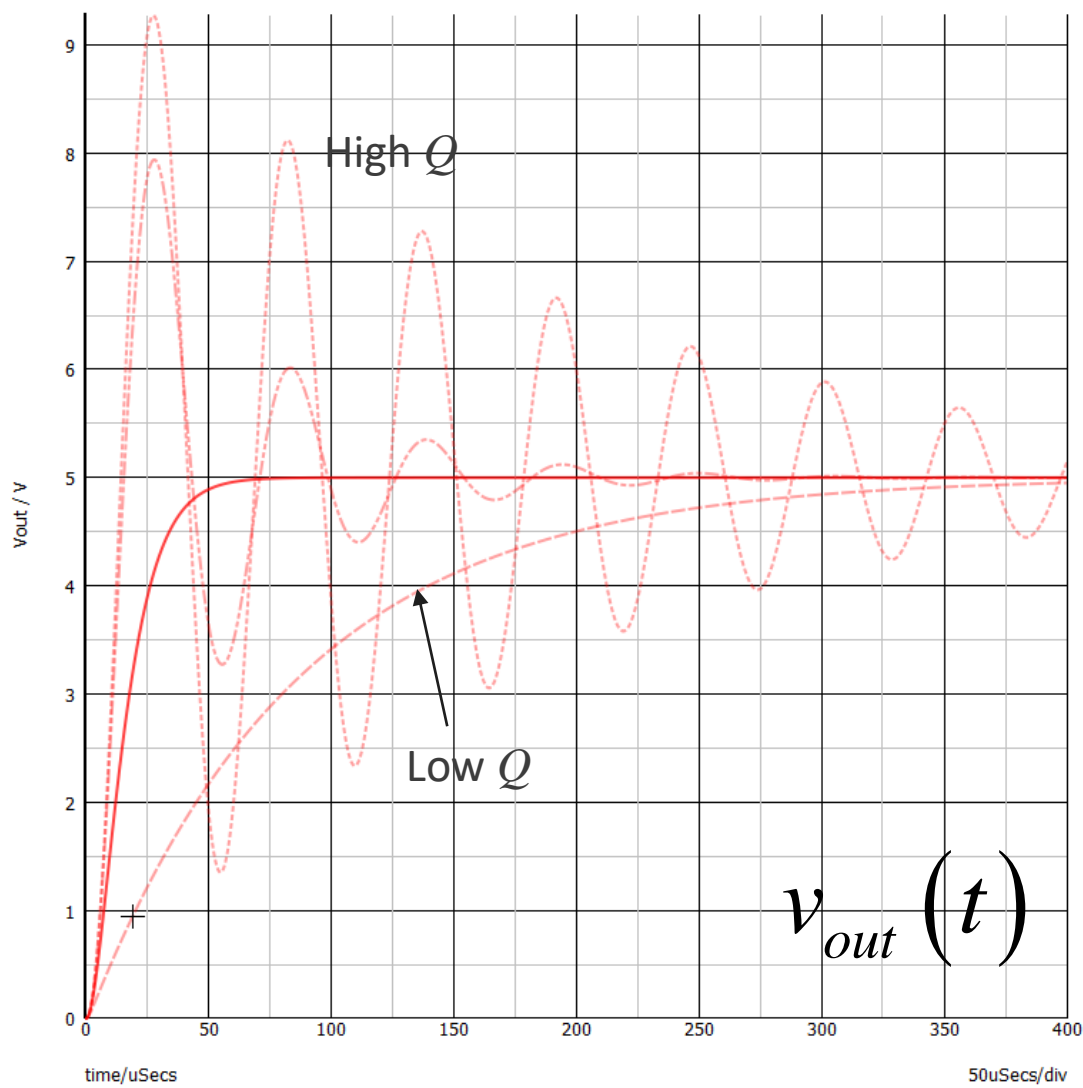
You can save the Bode box configuration for the next studies if you want.

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

History

4 Use default

Separate curves



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:

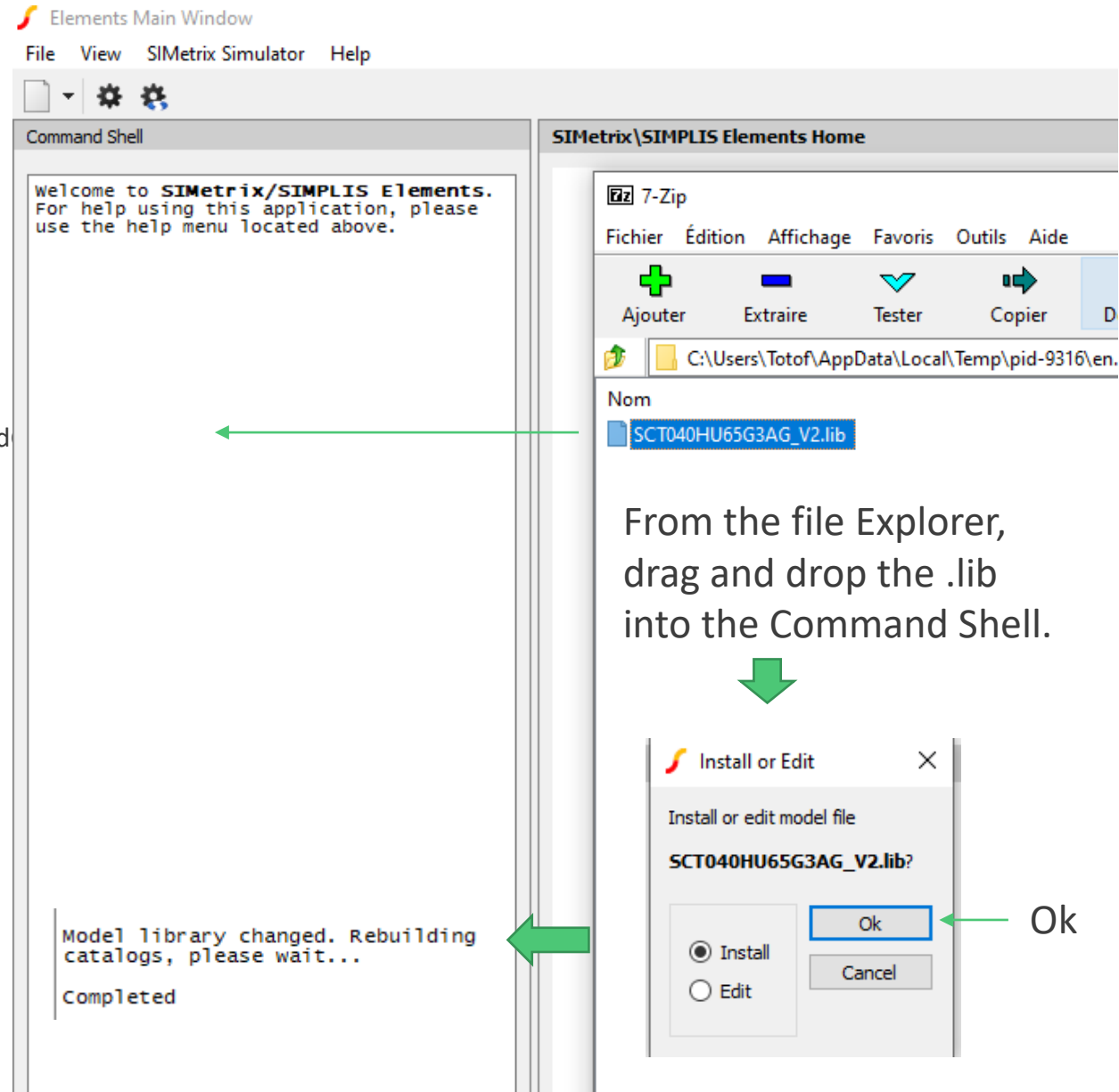
```

*****
***** 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
.....

```

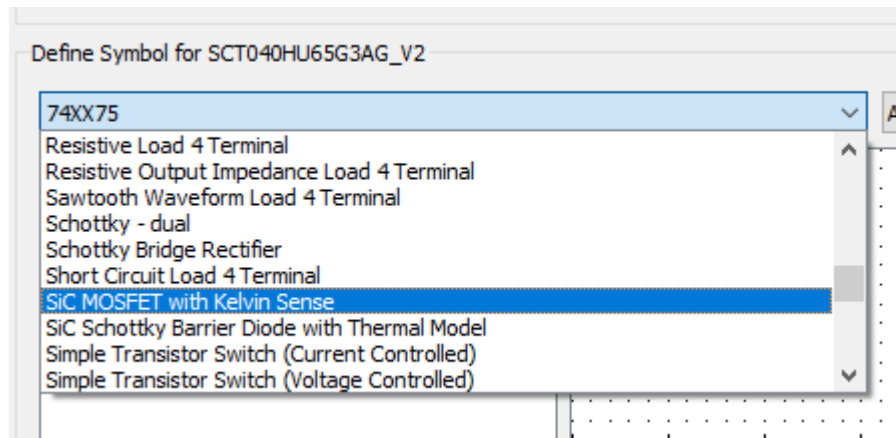
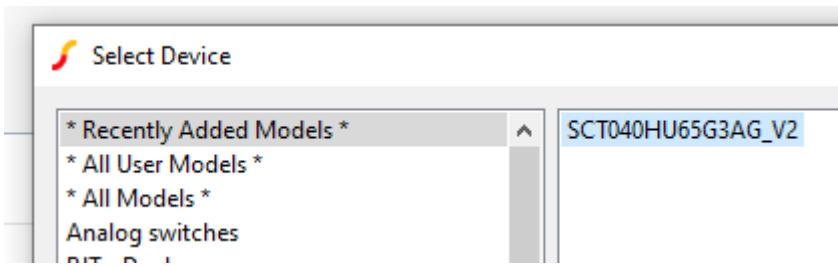
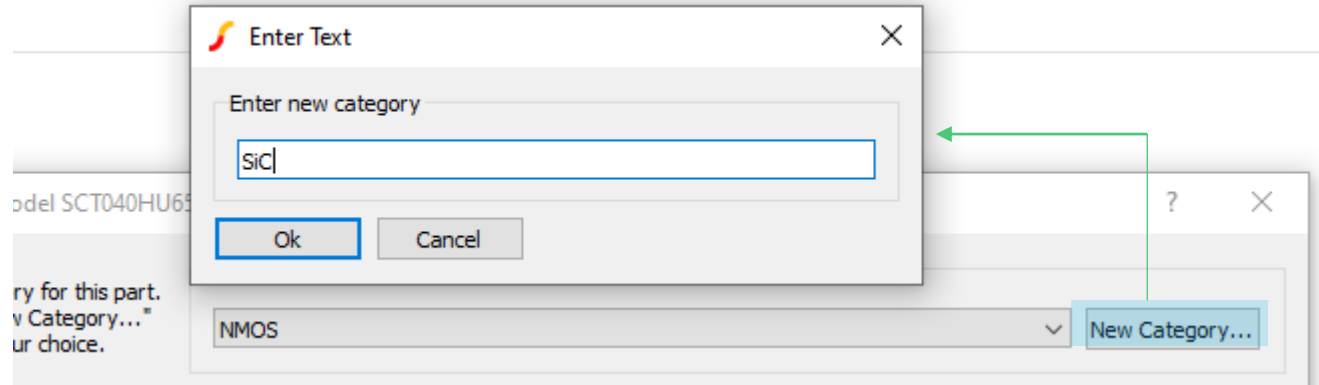
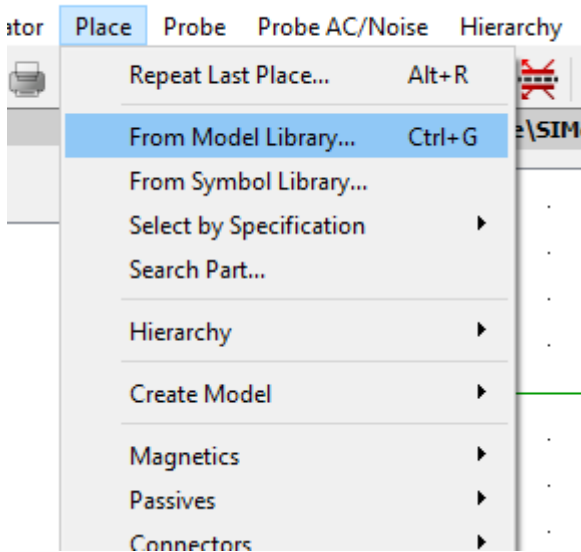


From the file Explorer, drag and drop the .lib into the Command Shell.

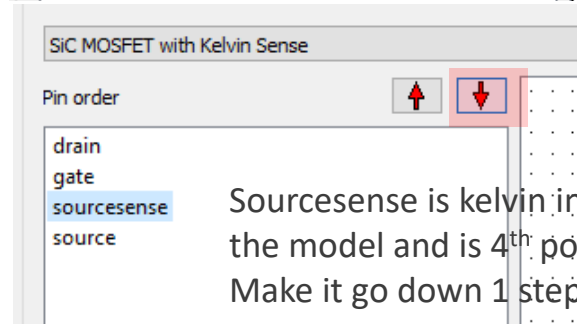
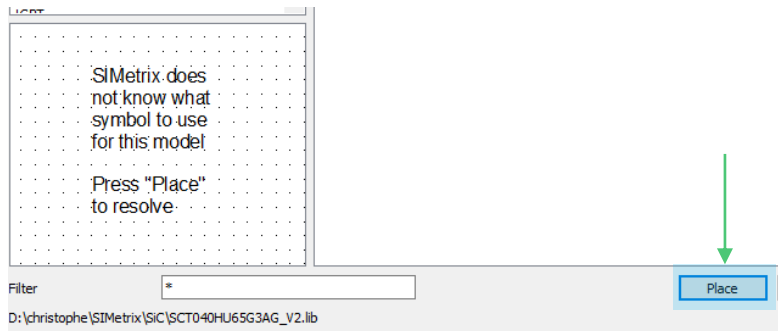


Ok

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

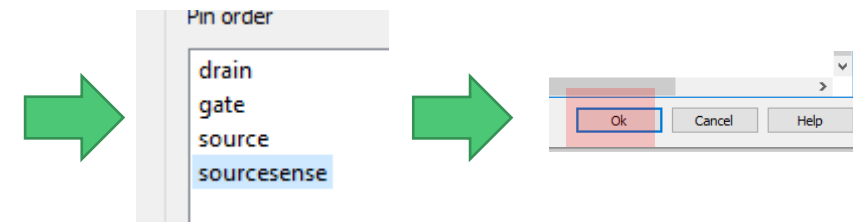


Choose the right symbol

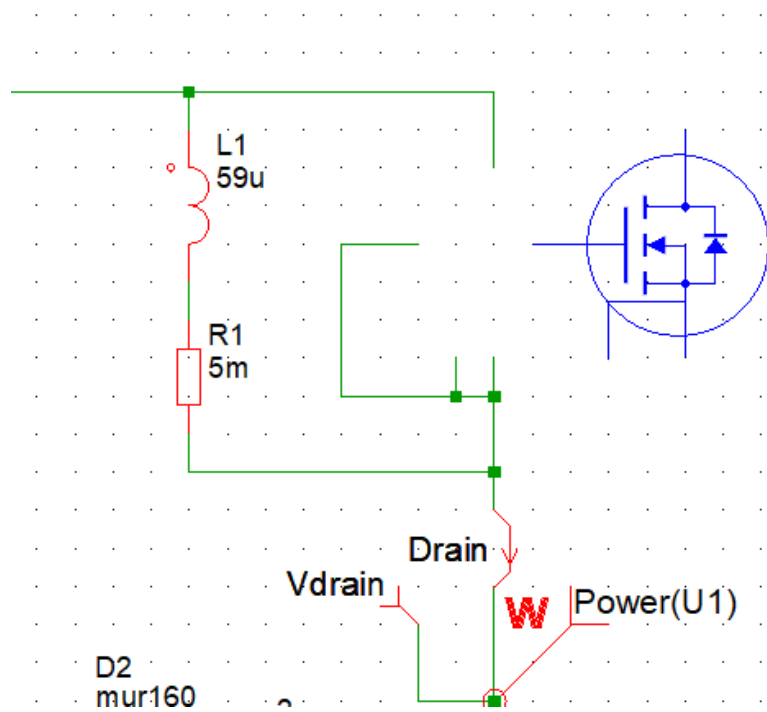


Sourcesense is kelvin in the model and is 4th position. Make it go down 1 step.

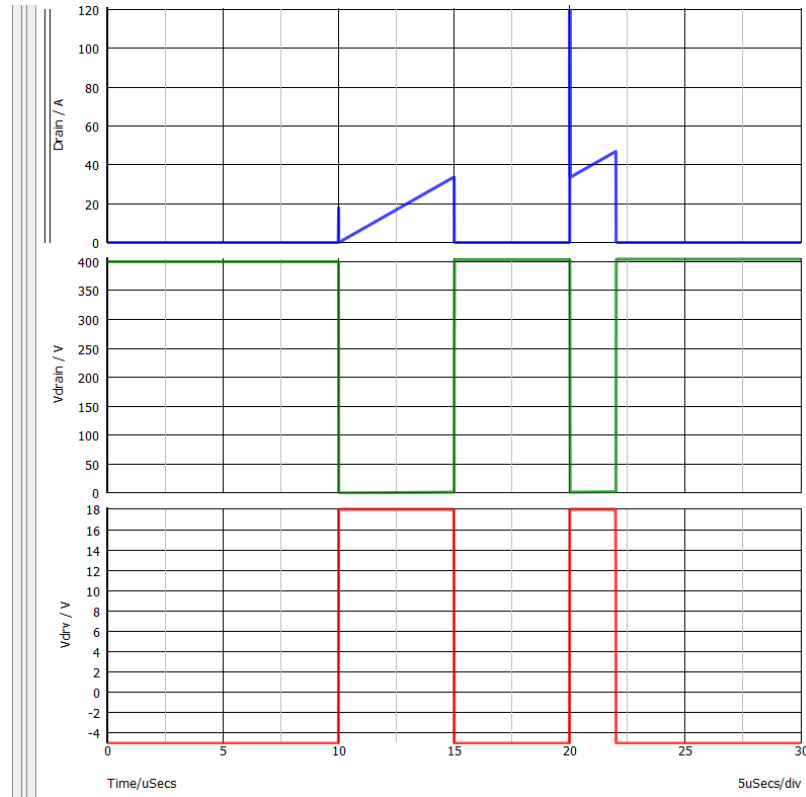
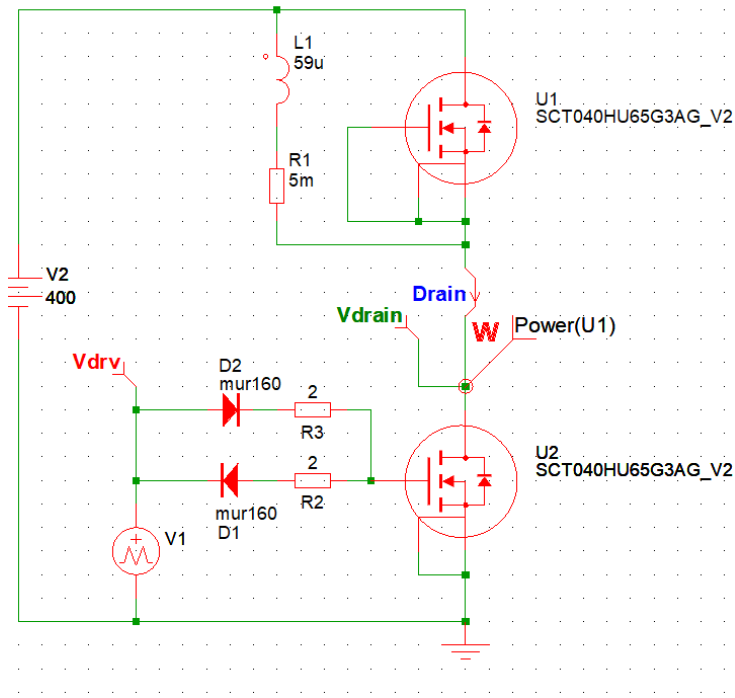
1 2 3 4
 .subckt SCT040HU65G3AG_V2 drain gate source kelvin PARAMS



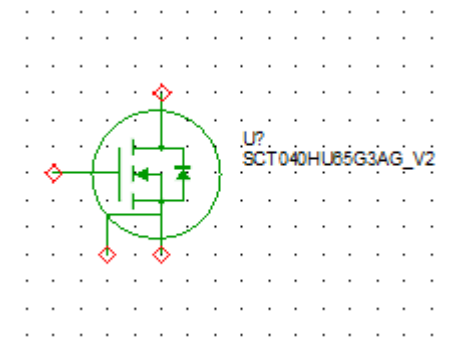
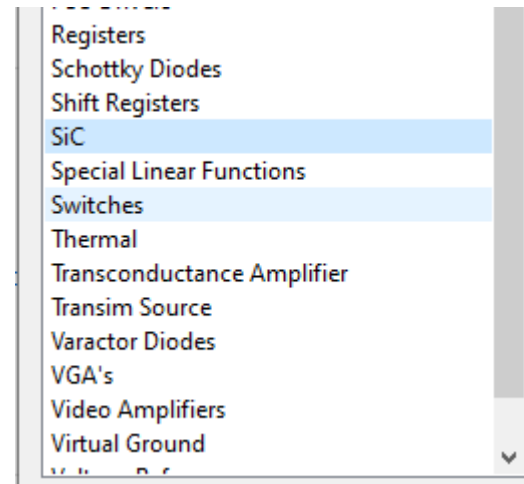
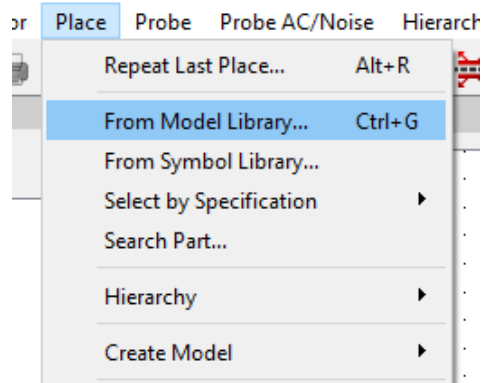
Place the model in the circuit:



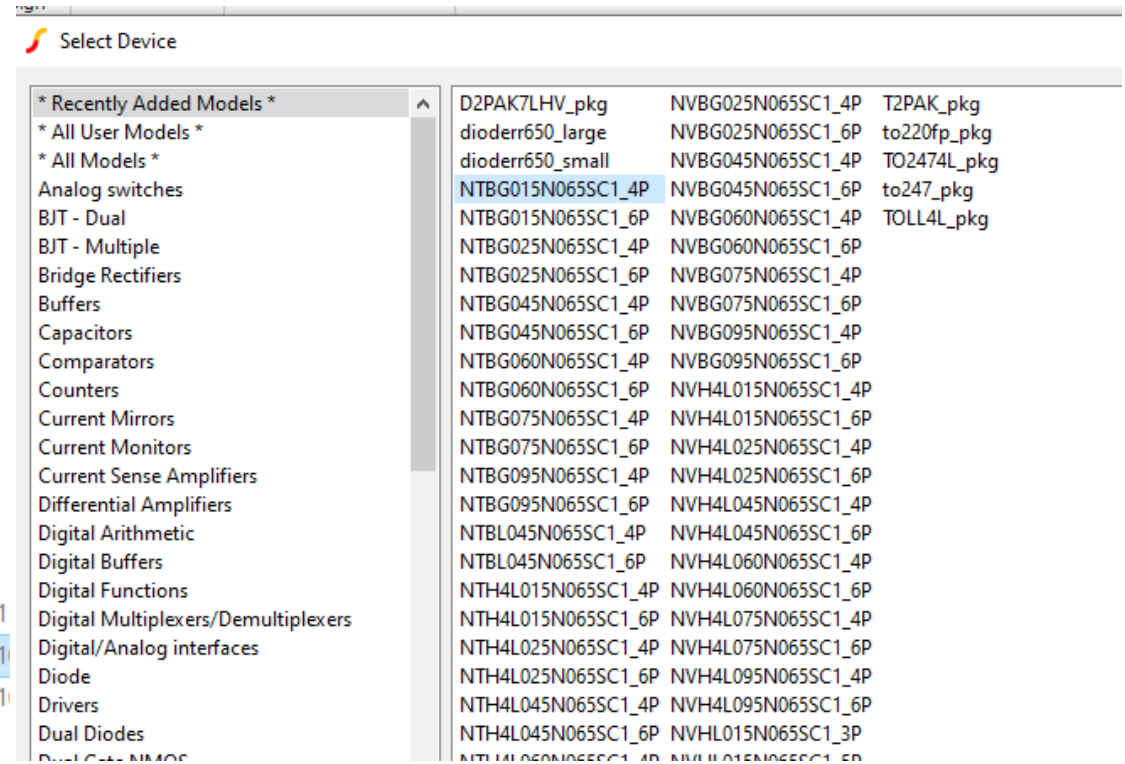
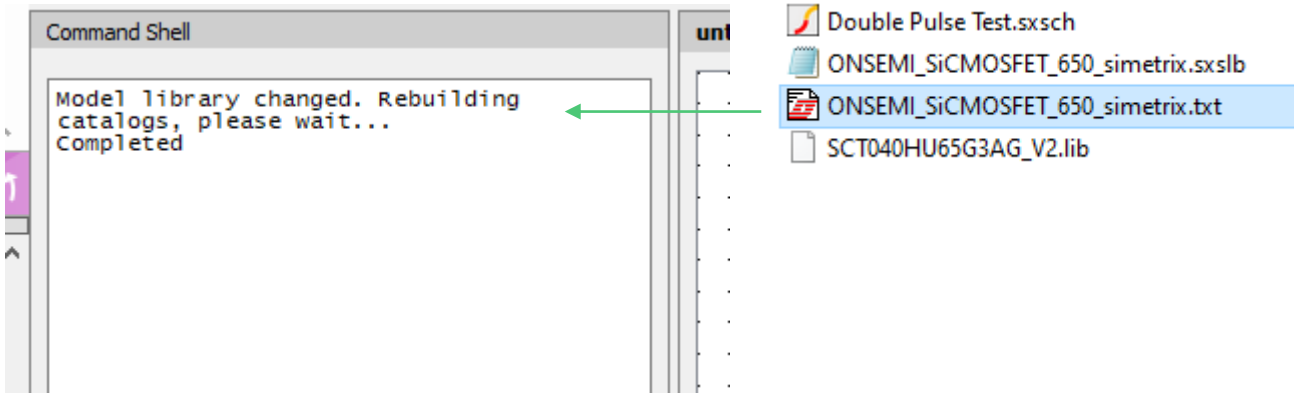
You can now run the simulation



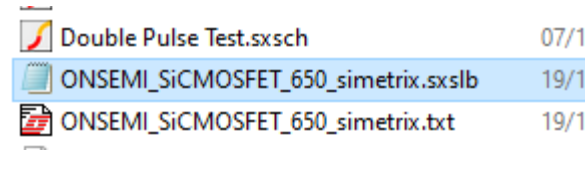
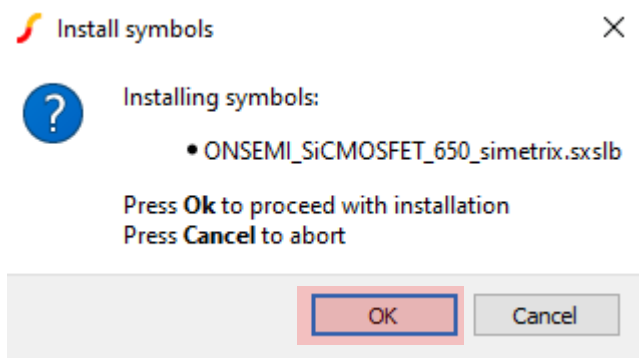
To place a component,
go to Place Symbol



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:

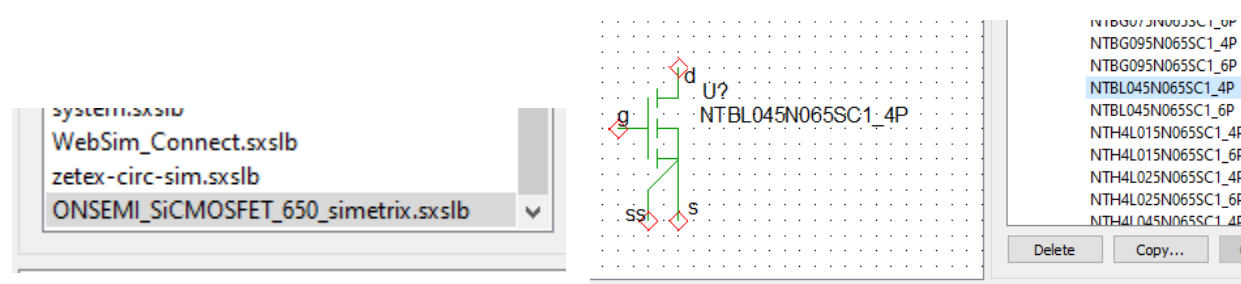
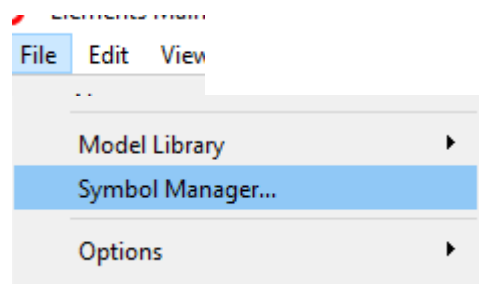


Now, you have to load the associated symbol file:



Then CTRL+ G to access the models

You can see the models are properly imported:



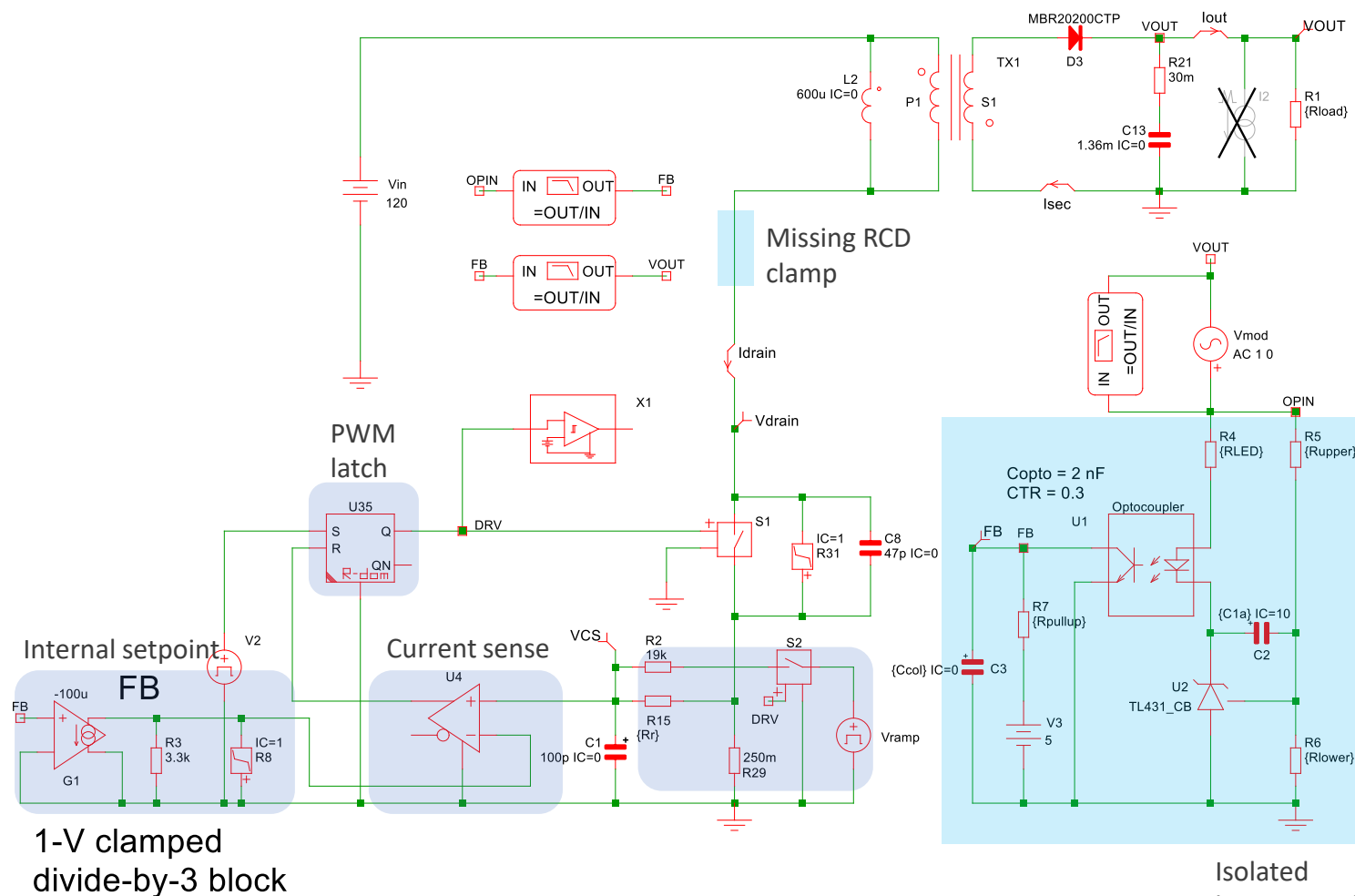
Agenda

- 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

Add Directory Sync to Active

- Book Collection
 - SIMPLIS_Data
 - Subs
 - TMP
 - Boost 2 Phase CM compensated.wxsch
 - Boost 2 Phase VM compensated AC.wxsch
 - Boost 2 Phase VM compensated TRAN.wxsch
 - Boost BCM CM.wxsch
 - Boost CM PFC ac tran demo.wxsch
 - Boost CM PFC ac Zout demo.sxsch
 - Boost CM PFC sine full version.wxsch
 - Boost CM.wxsch
 - Boost VM compensated AC.sxsch
 - Boost VM compensated TRAN.wxsch
 - Boost VM PFC ac tran demo.wxsch
 - Boost VM PFC sine full version.wxsch
 - Buck 2 Phase CM.wxsch
 - Buck 2 Phase VM.wxsch
 - Buck BCM.wxsch
 - Buck CM Synchro.sxsch
 - Buck CM.sxsch
 - Buck COT.wxsch
 - Buck FOT.wxsch
 - Buck VM.sxsch
 - Buck-Boost CM.wxsch
 - Buck-Boost VM compensated AC.wxsch
 - Buck-Boost VM compensated TRAN.wxsch
 - Flyback 2SW CM isolated.wxsch
 - Flyback active clamp CM isolated and compensated.sx...
 - Flyback active clamp CM non-isolated - demo.sxsch
 - Flyback CM isolated ac sine input.wxsch
 - Flyback CM isolated.wxsch



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 *
```

Your targeted
values (2 kHz, 60°)

```
* 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
```

```
* 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
```

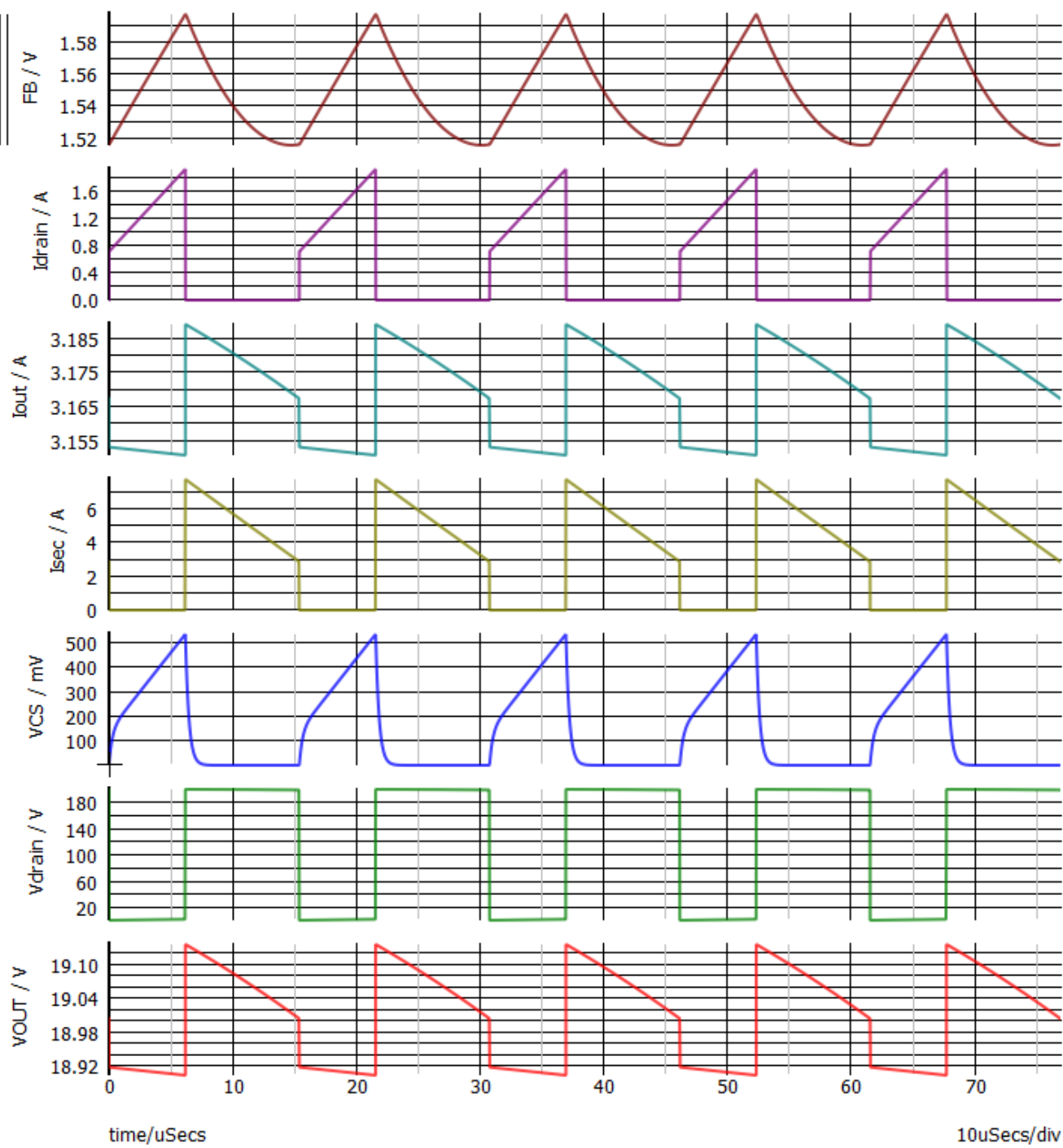
Pole-zero calculation


```
{ '*' }
{ '*' } 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: $V_{out} = 19\text{ V}/3\text{ A}$ ✓



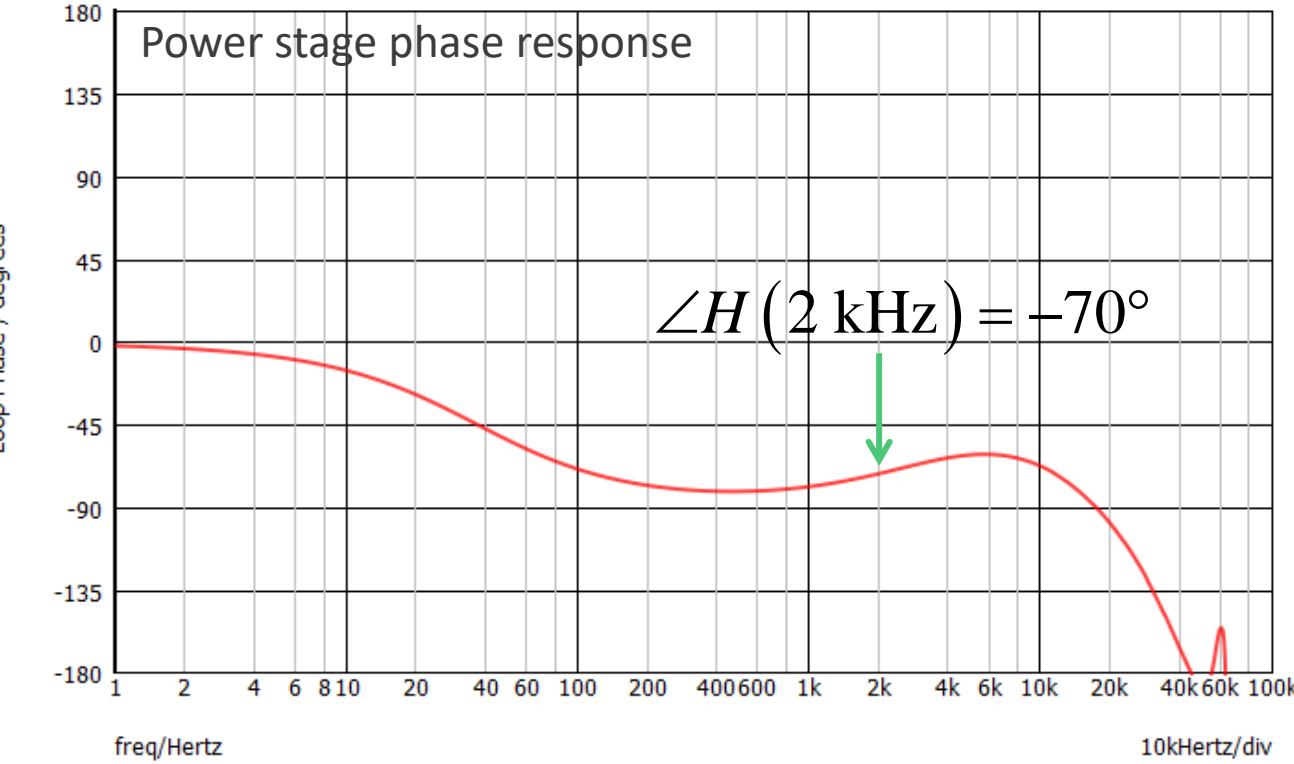
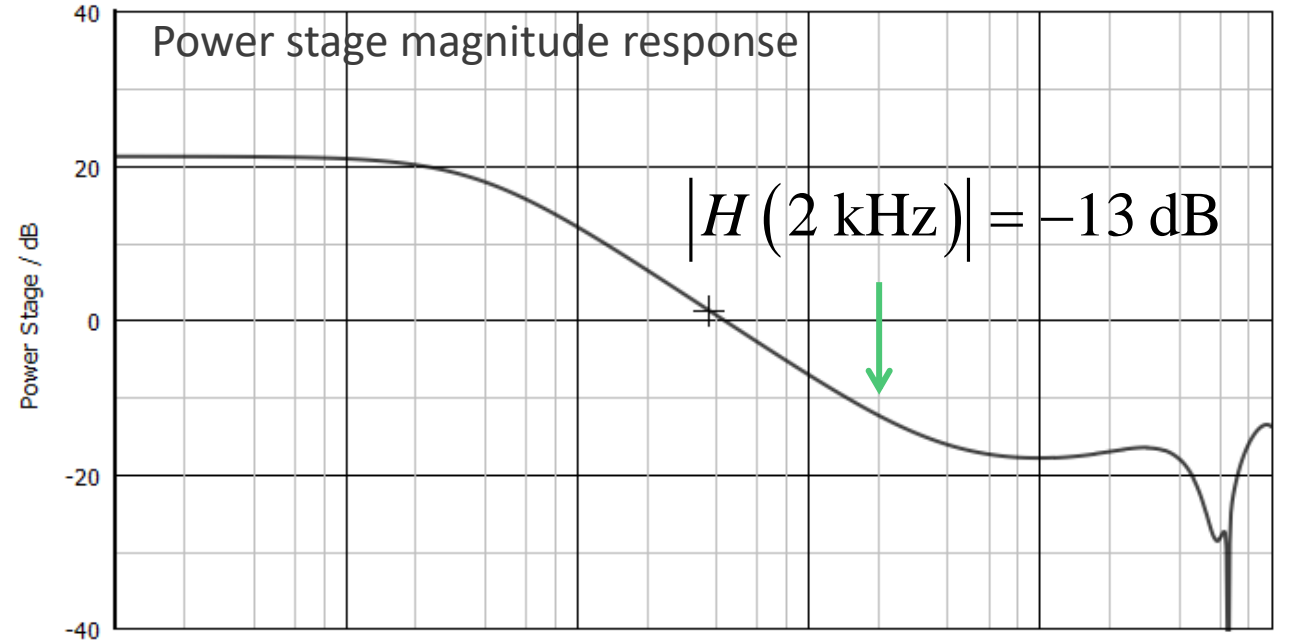
Feedback voltage  within the 3-V dynamic

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:

```

12 *
13 * Rupper = 66000
14 * Rlower = 10000
15 * C2 = 1.44819154804453e-09
16 * C1 = 3.31268645711794e-09
17 * Boost = 50
18 * Fz = 727.940468532405
19 * Fp = 5494.95483890924
20 * Sn = 50000
21 * Se = 25000
22 * D = 0.387755102040816
23 * Mc = 1.5
24 * Rramp = 2850.001
25 * Rmax = 24995.7446808511
26 * FRHPZ = 24616.8762677045
27 * FcMAX = 7385.06288031136
28 *

```

→ $f_{c,max} \approx 7 \text{ kHz}$



Consider the 6-kHz opto pole

Choose a 2-kHz crossover
 Read the magnitude plot a 2 kHz $\approx -13 \text{ dB}$
 Read the phase plot at 5 kHz $\approx -70^\circ$



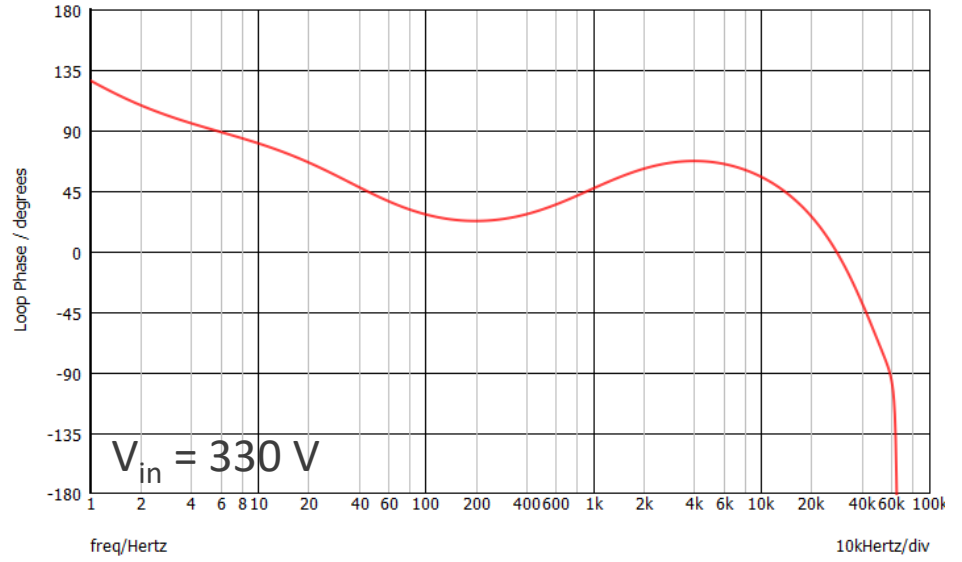
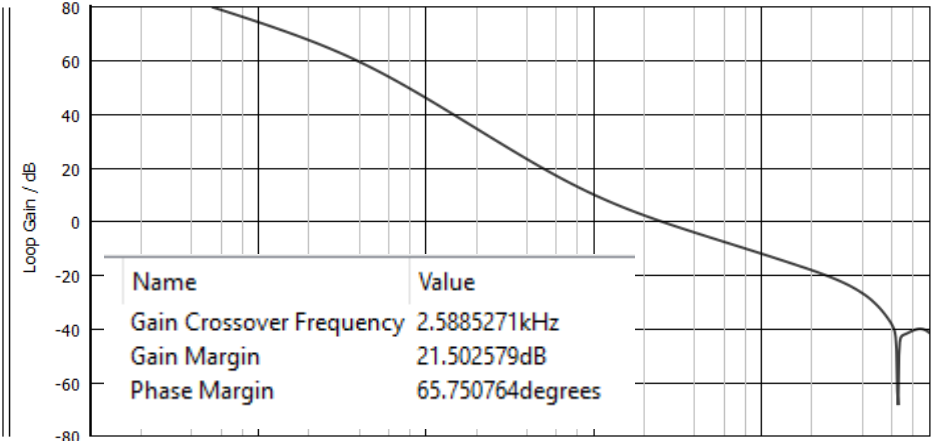
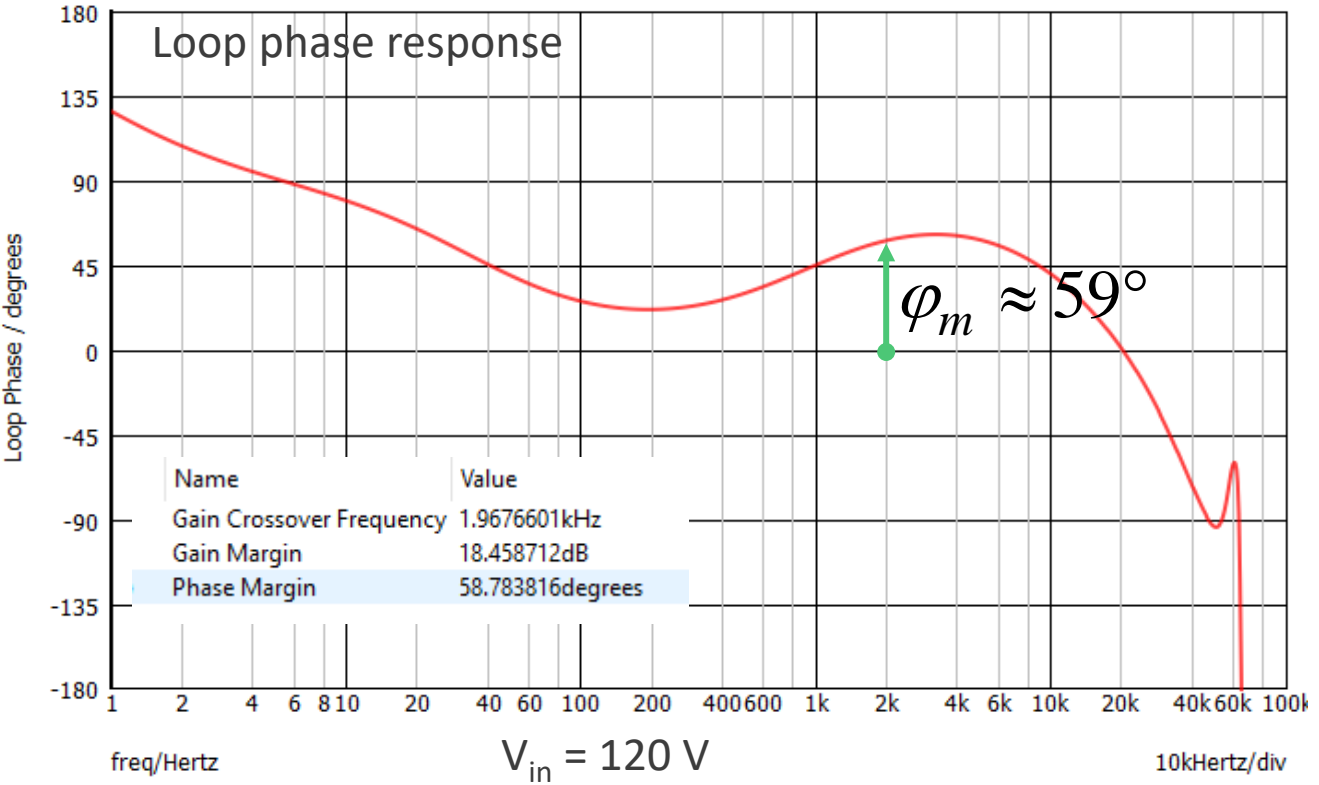
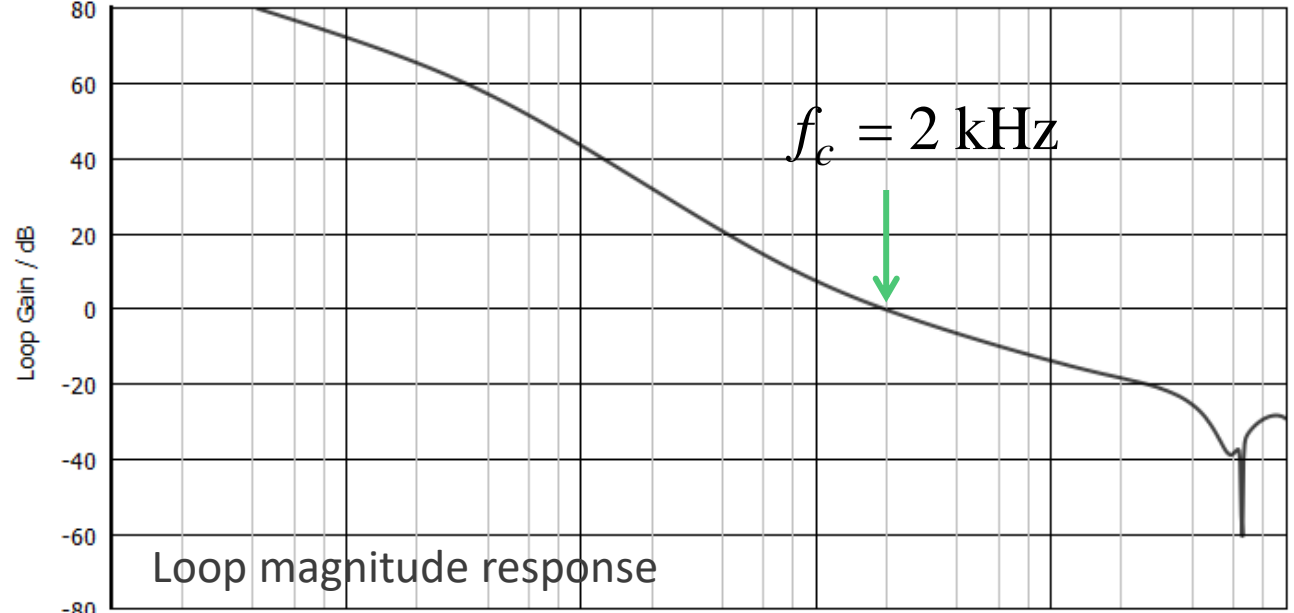
```

* 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 *
*

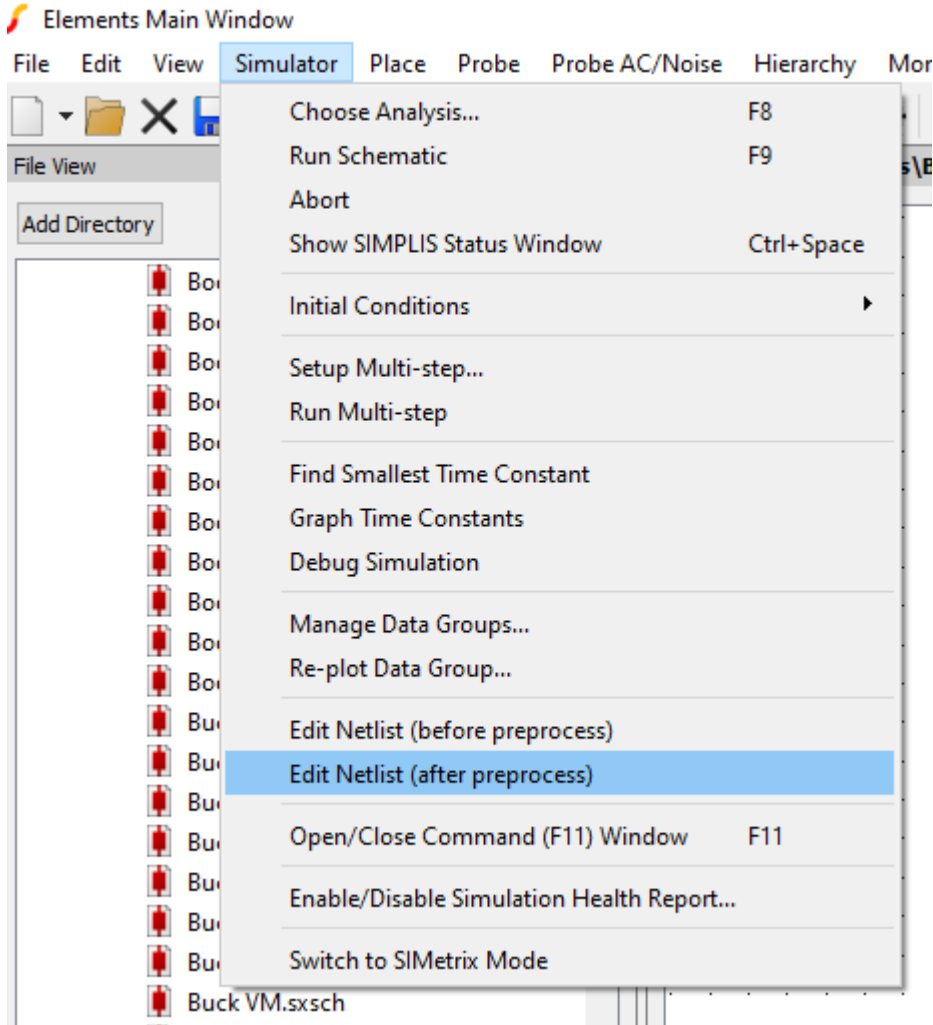
```

Fill out the requirements

The crossover frequency and phase margin are our 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:



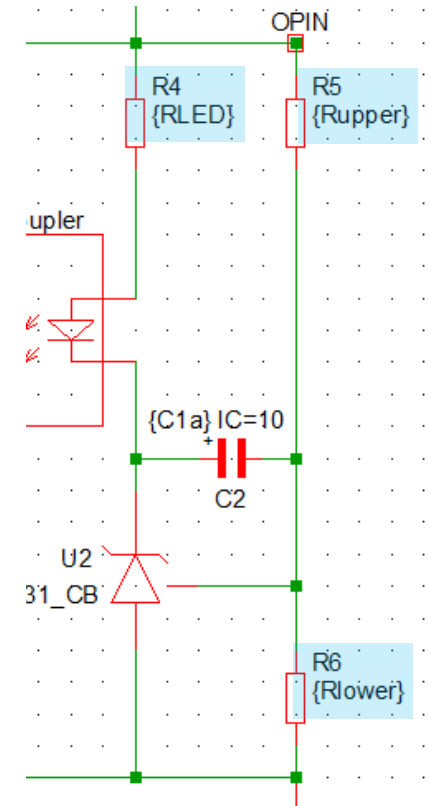
```

7 .print ALL
8 .options PSP_START=0 PSP_NPT=100001 POP_I
9 + MIN_AVG_TOPOLOGY_DUR=1a AVG_TOPOLOGY_DUR
10 .pop TRIG_GATE=X1 !D_CYCLE TRIG_COND=0_
11 + TD_RUN_AFTER_POP_FAILS=-1
12 *
13 * Rupper = 66000
14 * Rlower = 10000
15 * C2 = 1.85537921992434e-09
16 * C1 = 2.58567330980208e-09
17 * Boost = 40
18 * Fz = 932.615316309997
19 * Fp = 4289.01384101912
20 * Sn = 50000
21 * Se = 25000
22 * D = 0.387755102040816
23 * Mc = 1.5
24 * Rramp = 2850.001
25 * Rmax = 24995.7446808511
26 * FRHPZ = 24616.8762677045
27 * FcMAX = 7385.06288031136
28 *
29 C1 33 0 100p IC=0
30 C13 20 0 1.36m IC=0
31 C2 32 27 2.58567330980208e-09 IC=10
32 C3 41 0 5.29088027491884e-10 IC=0
33 C8 24 31 47p IC=0
34 X$D3 18 39 DIODE_SPICE_V2$1
35 G1 37 0 41 0 -100u
  
```

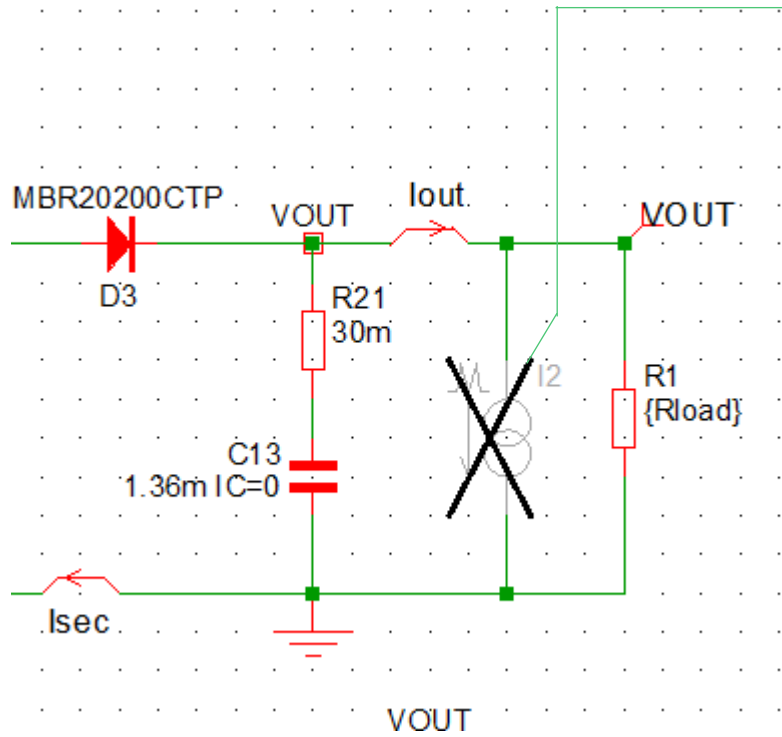
You can add more computed values

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

Value passed to the component

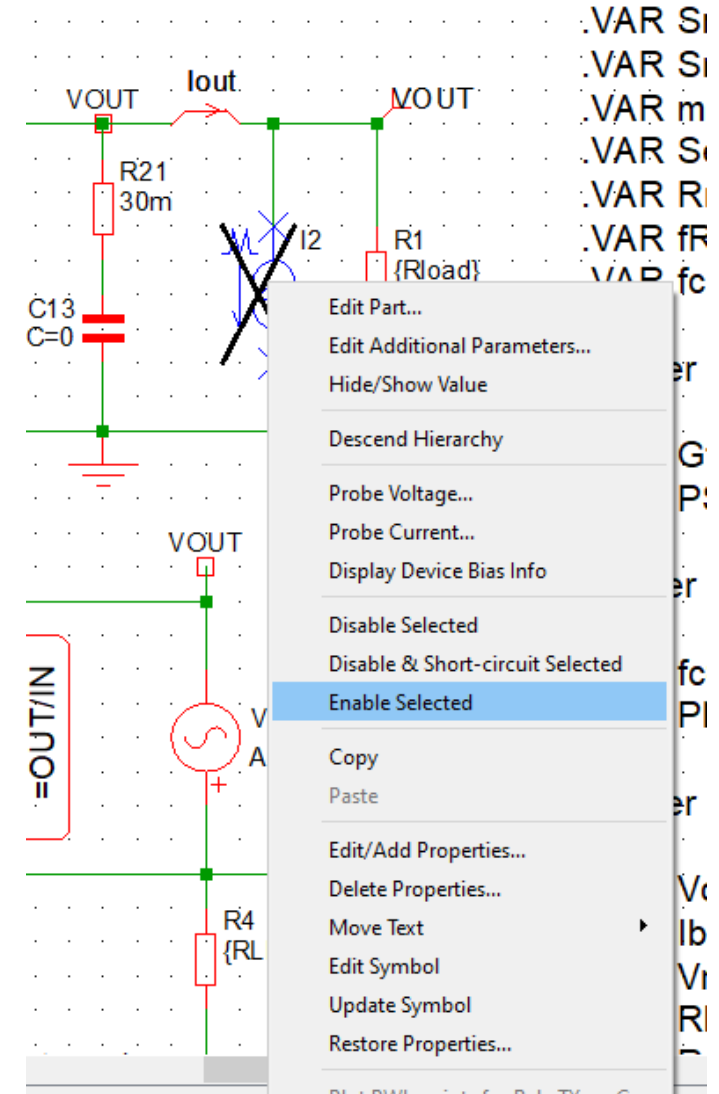


Test the transient response



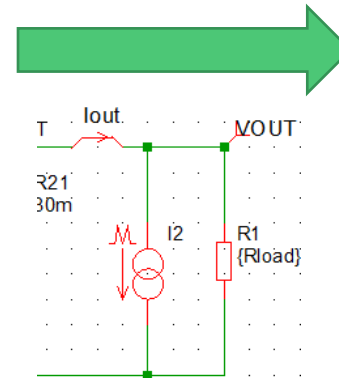
Define PWL Current Source: I2

	Time	Current
1	0	2
2	5m	2
3	5.01m	4
4	5.5m	4
5	5.51m	2
6		

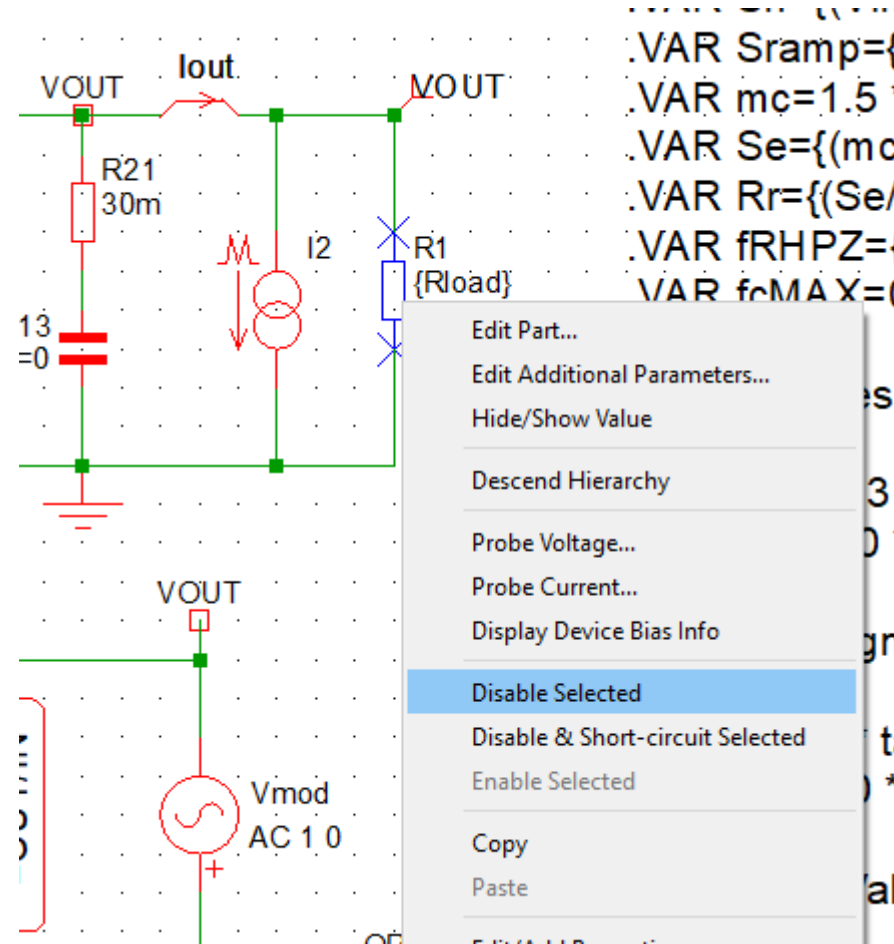


The X means the component is disabled
To activate it, right-click on the part:

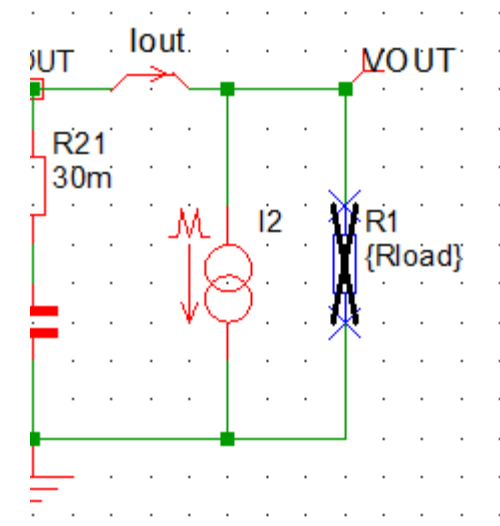
The cross disappears



Now disable the resistor, right-click and disable it



```
.VAR Sramp={
.VAR mc=1.5
.VAR Se={(mc
.VAR Rr={{(Se/
.VAR fRHPZ={
.VAR fcMAX=(
```



Then press F8 to access the simulation setup

The dialog box 'Choose SIMPLIS Analysis' is shown with the 'Transient' tab selected. The 'Analysis parameters' section includes 'Stop time' set to 6m s and 'Start saving data at t = 4.5m s'. The 'Plot data output' section has 'Number of plot points' set to 100k. On the right, under 'Select analysis', 'POP' and 'AC' are unchecked, and 'Transient' is checked. Under 'Save options', 'All' is selected. At the bottom, the 'Run' button is highlighted with a green arrow.

Uncheck
Check

Then run!

Place

Probe

Voltage Sources

Current Sources

Controlled Sources

Bias Annotation

Semiconductors

120

DC Source

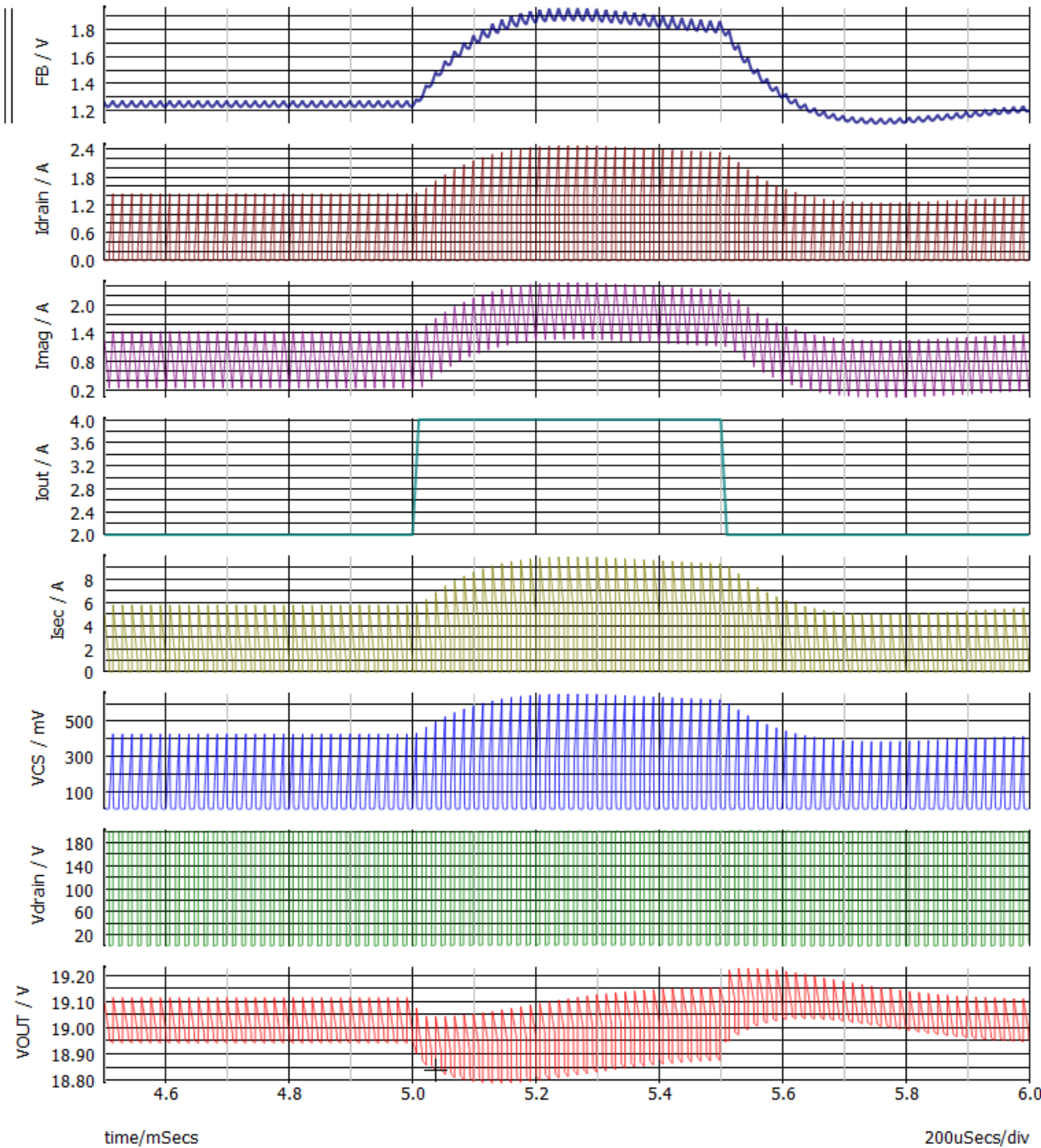
Waveform Generator

PWL Source

AC Source

Define PWL Current Source: I2

	Time	Current
1	0	2
2	5m	2
3	5.01m	4
4	5.5m	4
5	5.51m	2
6		



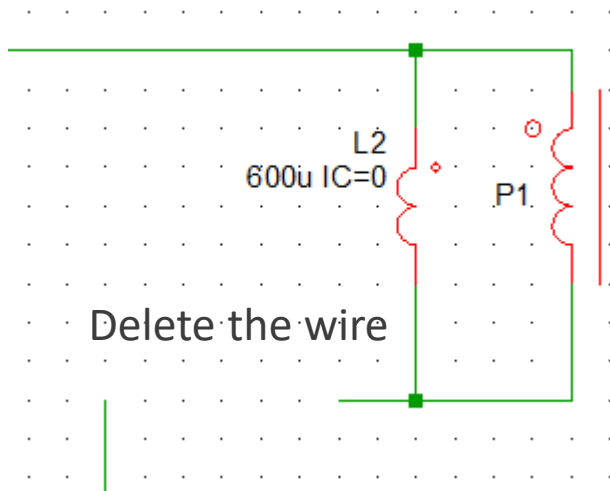
$$V_{out}(t)$$

$V_{in} = 120\text{ V}$

time/mSecs

200uSecs/div

We can now try to add a RCD clamping network



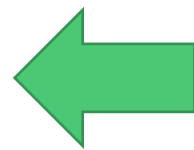
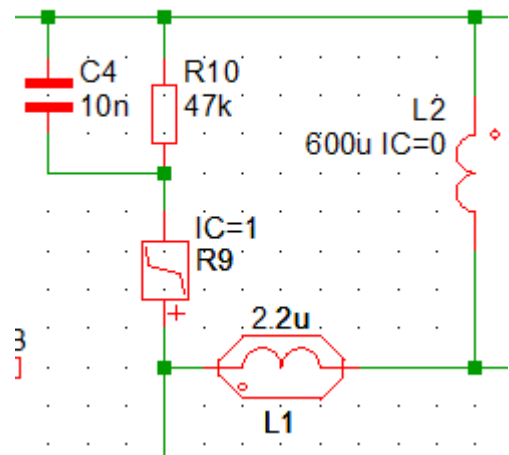
Place Probe Probe AC/Noise Hierarchy Monte Carlo Tools Help

- Repeat Last Place... Alt+R
- From Model Library... Ctrl+G
- From Symbol Library...
- Select by Specification
- Search Part...
- Hierarchy
- Magnetics**
 - Ideal Transformer...
 - Ideal DC Transformer...
 - Ideal Inductor L L
 - Lossy Inductor (Simple)**
 - Lossy Inductor (Multi-level)
- Passives
- Connectors
- Probe
- Voltage Sources
- Current Sources
- Controlled Sources
- Bias Annotation
- Semiconductors
- Digital
- SIMPLIS Primitives
- Analog Functions
- Worksheets

Edit Device Parameters

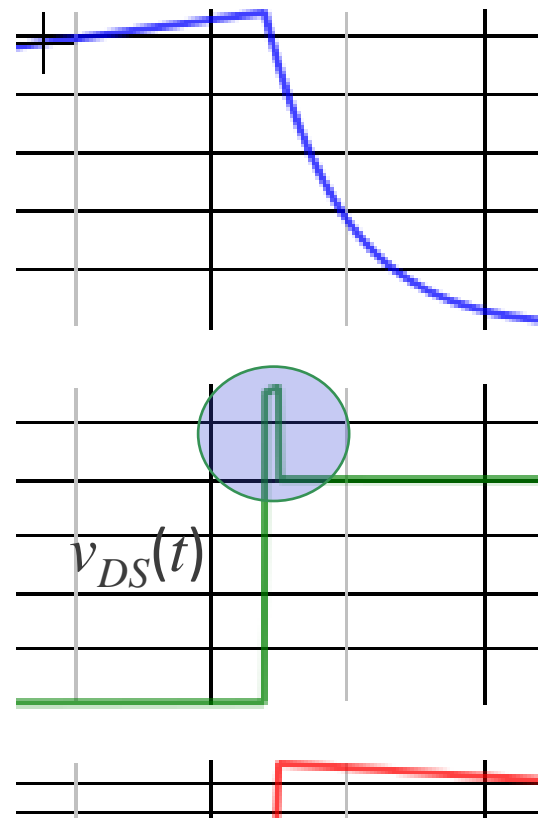
- Inductance: 2.2u
- Shunt Resistance: 1Meg
- Series Resistance: 100m
- Initial Condition: 0
- Use IC

Ok Cancel



The lossy inductor offers more robust convergence by including parasitics

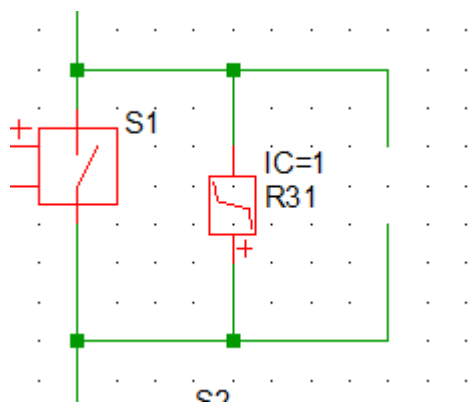
The screenshot shows the SIMPLIS Status window with an error dialog box. The error message states: "The number of components has exceeded the limit for the DEMO/INTRO unit. You should reduce the number of components in the following group: capacitors, inductors, large-signal transient sources, transistors, switches, logic gates, and PWL elements." The background shows a circuit schematic with components like R21 (30m), C8 (47p), and S1.



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

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

Delete the DS capacitance



Support

[Home](#) » [Support](#)

Documentation

[Product Documentation](#)

[Learning SIMPLIS](#)

[Knowledge Base Videos](#)

[Downloads & Updates](#)

[Example Schematics](#)

[MDM Example Schematics](#)

[Books on Power Electronics](#)

Documentation

We offer the following documentation both online and bundled with the SIMetrix/SIMPLIS product.

Tutorials & Training

SIMPLIS Tutorial - The SIMPLIS tutorial is intended to help new users get started with the SIMPLIS simulator and to serve as a general reference for SIMPLIS. The tutorial follows the progression of a buck converter design from first constructs to a final, parametrized, hierarchical design.

Learning SIMPLIS - A structured experience in which a capable and willing engineer can come up to speed with SIMPLIS in four (full-time) weeks. At the conclusion of the experience, the engineer will be confident that they will be able to competently apply SIMPLIS to appropriate switching power supply modeling objectives.

DVM Tutorial - The DVM Tutorial guides a user through configuring a working schematic to run in the Design Verification Module, run built-in test plans and customize them. A host of other, more advanced, topics is covered as well, including schematic and component modification and test configuration.

MDM Tutorial - The MDM Tutorial introduces a user to the basics of the SIMPLIS Magnetics Design Module through the design of an inductor for a DC-to-DC Buck converter and a transformer for an isolated AC line self-oscillating flyback converter.

Advanced SIMPLIS Training Materials - This section of the documentation comes directly from the training course that SIMPLIS Technologies conducts several times per year in different locations. The course material is intended for those with some experience using SIMPLIS and covers a wide range of topics from the POP analysis to Parameterization to measuring Switching Losses and Efficiency.



[SIMPLIS TUTORIAL](#) [WHAT IS SIMPLIS?](#) [ADVANCED SIMPLIS TRAINING](#) [SIMPLIS PARTS](#) [SIMPLIS ANALYSIS MODES - OVERVIEW](#)

[DVM - DESIGN VERIFICATION MODULE](#) [SYSTEMDESIGNER](#) [USER MANUAL](#) [SIMPLIS REFERENCE](#) [SIMETRIX SCRIPT MANUAL](#) [SIMPLIS MDM BETA TUTORIAL](#)

[SIMPLIS VERILOG HDL](#)

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