
Qspice - Entry User Guide

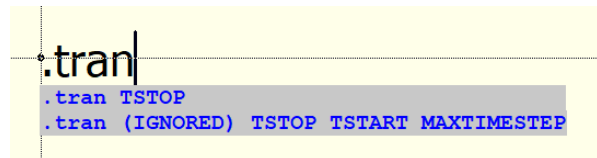
KS Kelvin Kelvin Leung

Created on 9-15-2023

Last Update on 12-5-2023

Qspice

- Qspice
 - Author : Mike Engelhardt
 - Website : <https://www.qorvo.com/design-hub/design-tools/interactive/qspice>
- GUI (Graphical User Interface)
 - Most input requires keyboard shortcuts
 - For example, R gives a resistor, press R again to cycle different symbols, Ctrl-R to rotate, W to draw a wire etc...
 - GUI gives hint for the syntax underneath your typing which eliminate diagnose/toolbox
 - Some user may not like this at beginning but from my usage experience this is a more convenience GUI
 - Example of hint syntax underneath

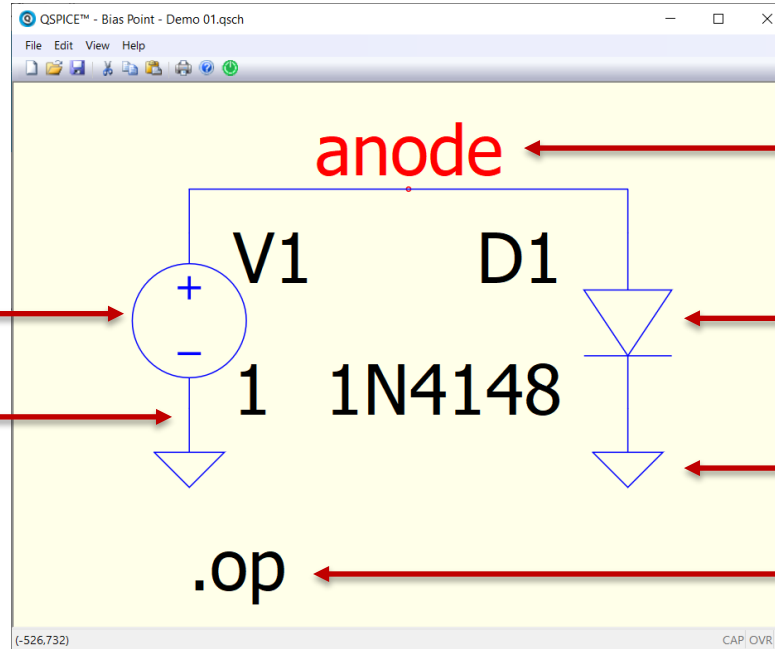


Basic Simulation

Qspice Command

- Analysis Directive
 - **.op** : Bias Point Analysis / Operation Point Analysis
 - dc operation point analysis, to calculate DC steady state voltage and current
 - **.dc** : DC Sweep
 - dc sweep analysis, it likes .op but can change source voltage/current value during analysis
 - **.ac** : AC Analysis
 - ac analysis, same as in circuit theory using phasor for calculation. Before .ac is run, it automatically runs .op for dc operation point and .ac is simulated on this dc bias condition
 - **.tran** : Non-Linear Transient Analysis
 - transient analysis, in default, a .op is run before .tran, and .tran is run based on this bias point condition at t=0s. User can skip .op by adding UIC in .tran directive
 - **.bode** : Frequency Response Analysis [topic not cover in this report]
- Include in this section
 - **.param** : User-Defined Parameter
 - **.step** : Step User-Defined Parameter
 - **.plot** : Plot Suggestion

Draw your first schematic

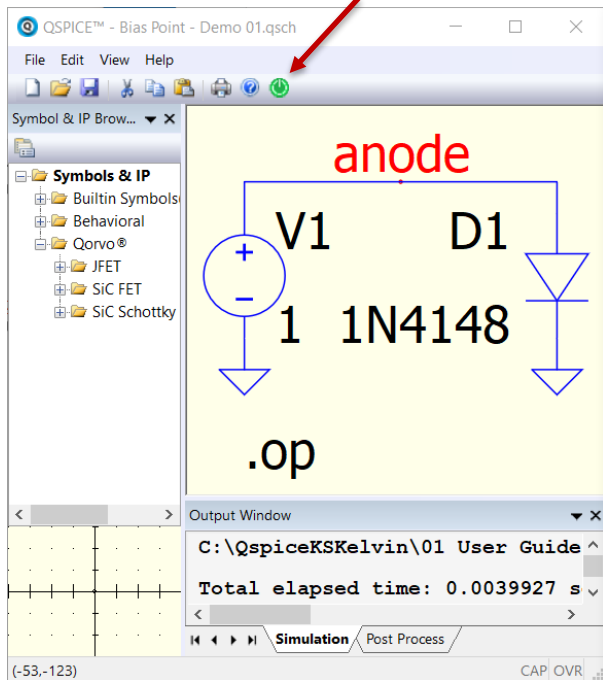


Bias Point Analysis (.op) : DC Operation Point

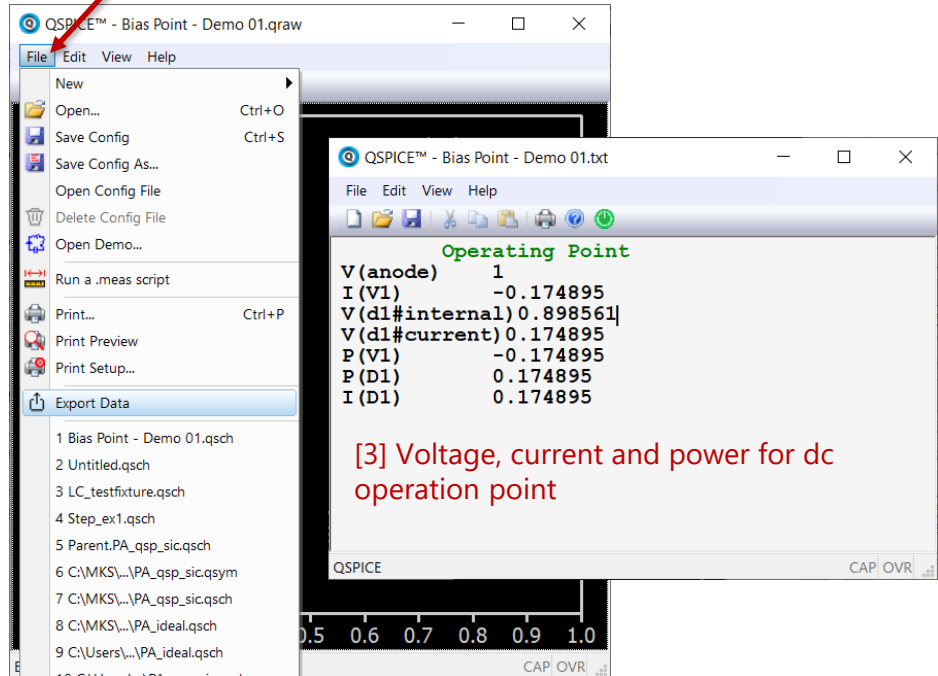
Qspice : Bias Point – Demo 01.qsch

[0] Bias Point Analysis (.op) computes DC operation point

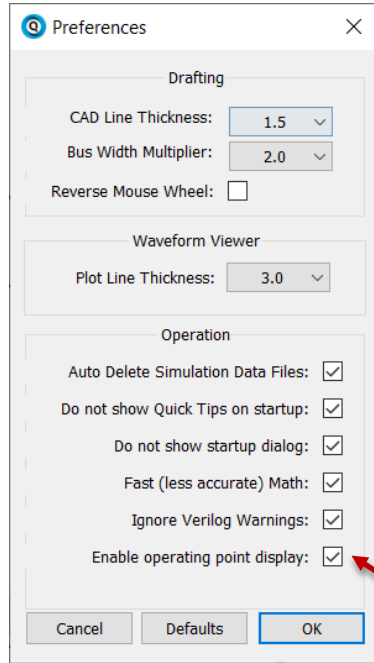
[1] Run to simulate the circuit



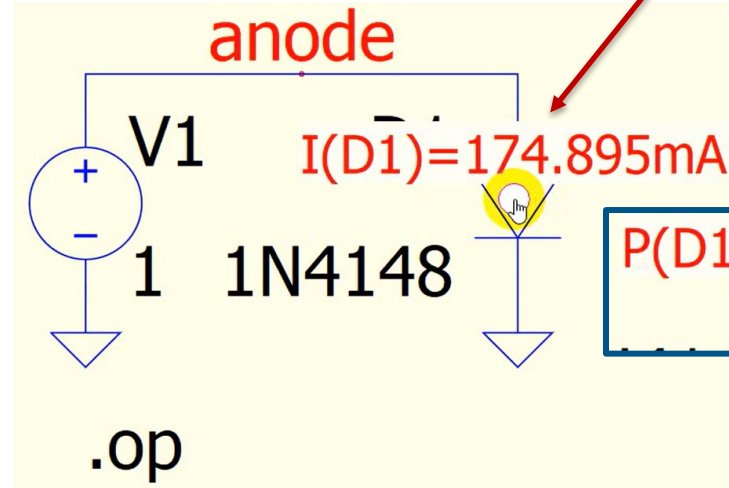
[2] Result of .op can be obtained by File > Export Data



Feature : Operating Point Display in Schematic

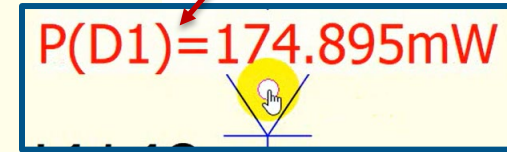


[1] Edit > Preferences
Select Enable operating point display



[2] Run Simulation, mouse cursor hover over component or node to display current or voltage

[3] Hold [Ctrl] key to display power



Hint : If you press [F6], the **.op** is recomputed without redoing the whole simulation

Bias Point Analysis (.op) with Step User-Defined Parameter (.step)

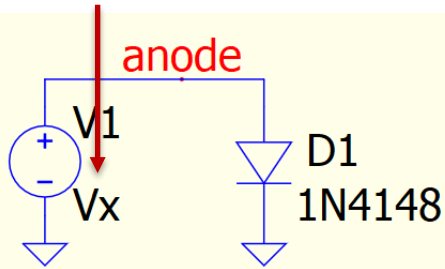
Qspice : Bias Point - Demo 02.qsch

[0] This example is to use **.op** to plot V-I curve of diode D1 1N4148

- Idea is to sweep anode voltage from 0.5V to 1V and plot diode current

[1] Change V1 value to Vx

Vx is a variable. In Qspice, it accept variable without curly bracket {}



** Current direction is defined by pin order
+I(D1) represent flow from A to K

Pin Nets	
Pin Names	Net
A	anode
K	GND

.op

.param Vx=1

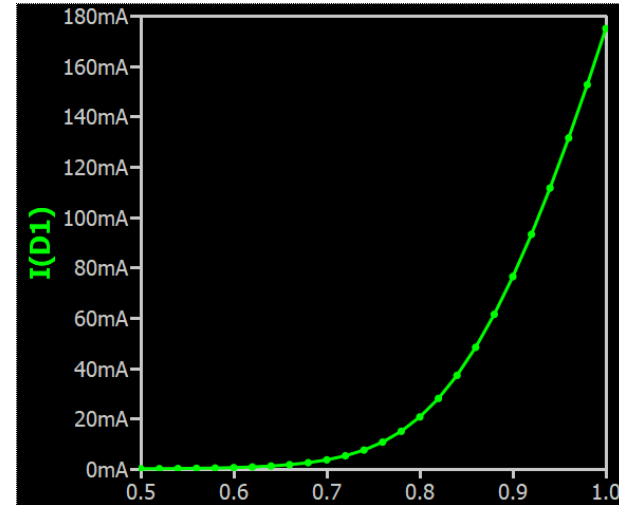
[2] Add **.param** to define Vx
value in this simulation

.step param Vx 0.5 1 0.02

[3] **.step** to sweep Vx, this syntax represent
Vx sweep from 0.5 to 1 with 0.02 per step

.plot I(D1)

[4] **.plot** tell the waveform viewer what to plot

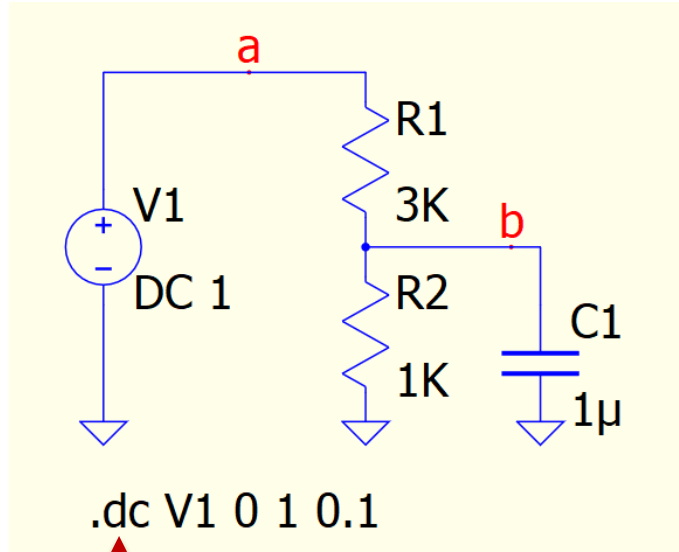


X-axis of this plot is Vx based on .step

DC Sweep (.dc) and Probing Signal Waveform

Qspice : DC Sweep - Demo 01.qsch

[0] DC Sweep (.dc) can be used to analyze steady state voltage under sweep of current source, voltage source or temperature

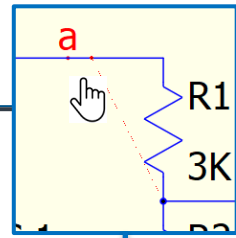


[1] .dc to sweep V1 from 0V to 1V with 0.1 per step

** In DC analysis, capacitor is OPEN circuit and inductor is SHORT circuit

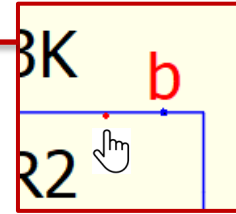
Differential voltage probe : V(a,b)

Hold [ALT] and click on node a, pull to node b

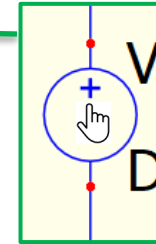


Voltage probe : V(b)

click node b

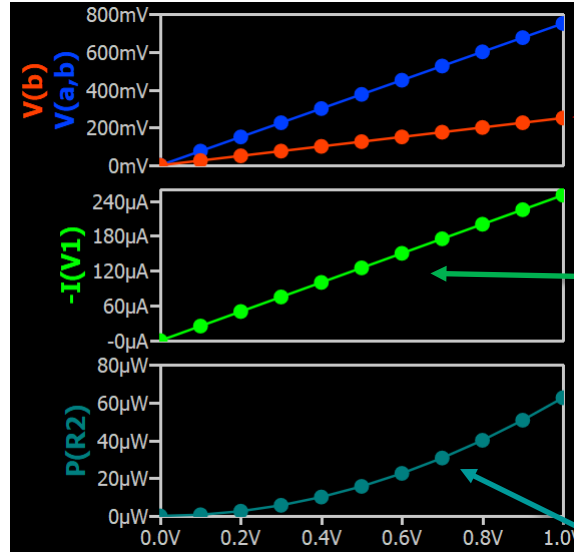
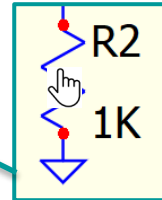


Current probe : I(V1)
Hover on device



Power probe : P(R2)

Press Ctrl and Hover on device

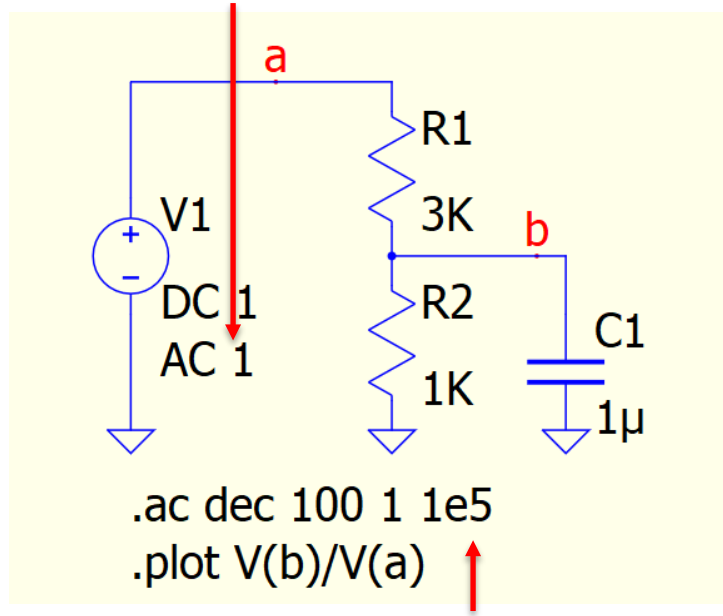


AC Analysis (.ac)

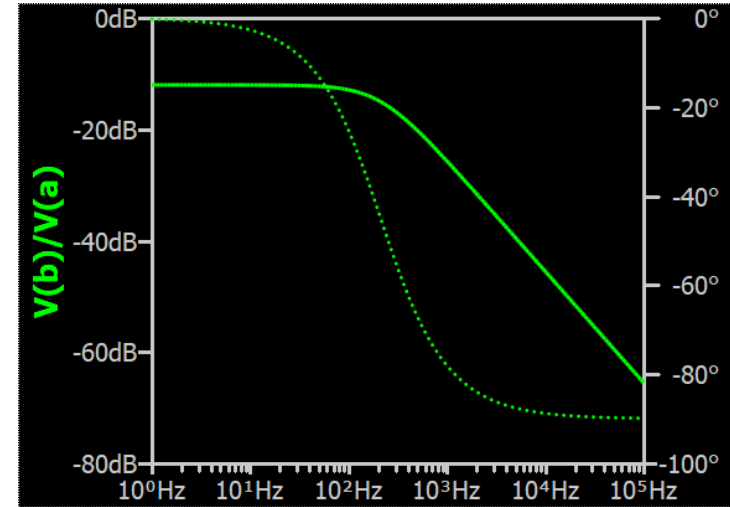
Qspice : AC Analysis - Demo 01.qsch

[0] AC analysis (.ac) computes frequency response of the circuit at its dc operating point (linear region)
.op is automatically run before .ac, .ac is based on this bias point to calculate frequency domain data

[1] Define an AC source, for example, this represent V1 is a 1V AC source.
In this example, a new attribute is used for AC 1 as V1 already defined as DC 1V
To add new attribute, right click on component > Add New Attribute



[3] AC analysis can plot bode (magnitude and phase relationship) of two probe positions. In this example, it is V(b) and V(a). If denominator is not specified, probe voltage is compare to AC source voltage.



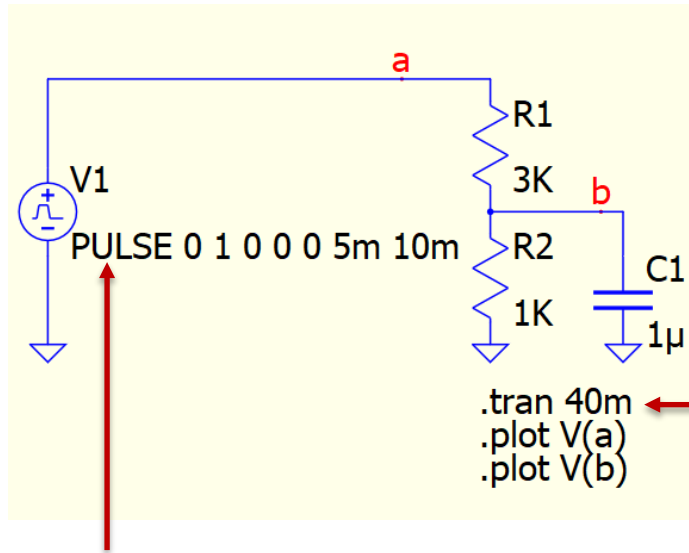
[2] this is to sweep AC source frequency from 1Hz to 1e5Hz (100kHz) with 100 points per decade

Non-Linear Transient Analysis (.tran)

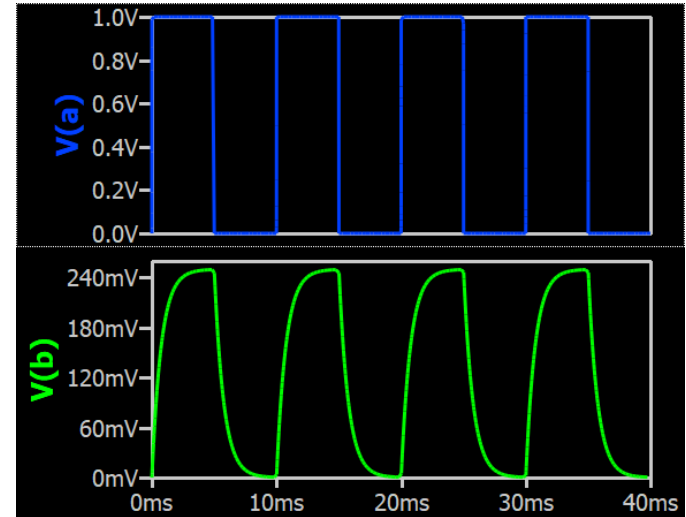
Qspice : Transient Analysis - Demo 01.qsch

[0] Non-Linear Transient Analysis (.tran) is time domain analysis to solving the general non-linear circuit

** .op is run before .tran, .tran will load bias point data to begin its transient analysis. Add UIC in .tran can skip .op before .tran



[1] transient analysis set
Tstop at 40ms



[2] Specify a time domain source (commonly are DC, PULSE, SINE)

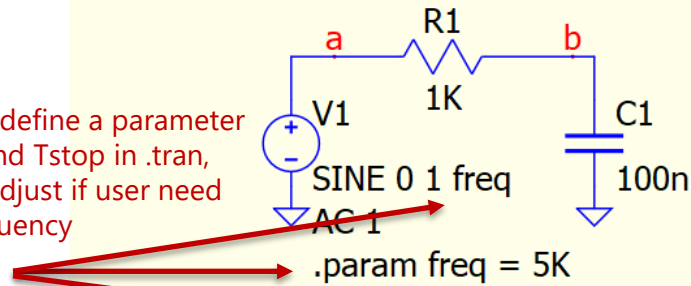
This is a pulse source with format : PULSE Voff Von Tdelay Trise Tfall Ton Tperiod Ncycles

Useful Technique

Parameter and Comment for Analysis Directive

Qspice : Comment and Params.qsch

[2] For example in transient, by define a parameter can control source frequency and Tstop in .tran, which help simulation to auto adjust if user need to study circuit at different frequency

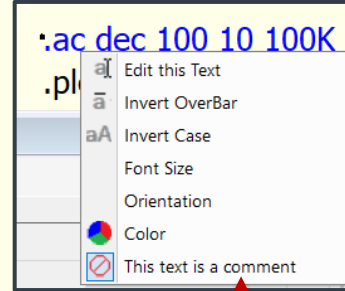


.tran 10/freq

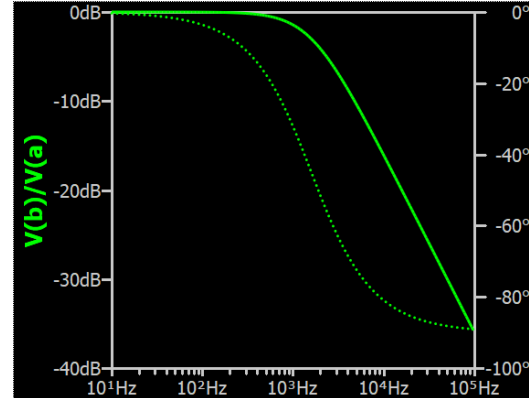
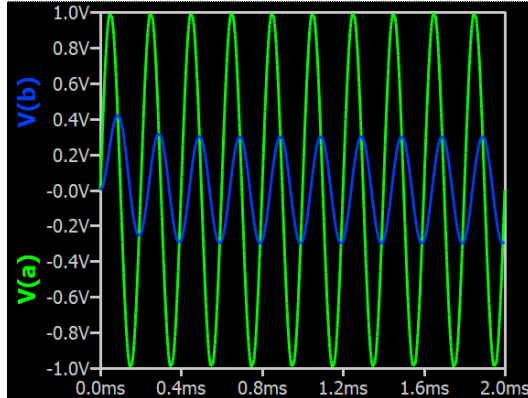
.plot tran V(a) V(b)

.ac dec 100 10 100K

.plot ac V(b)/V(a)



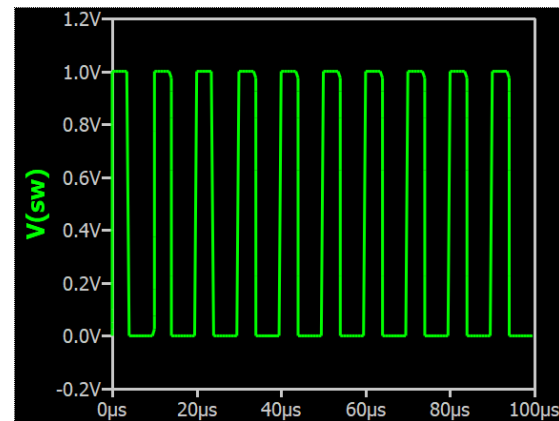
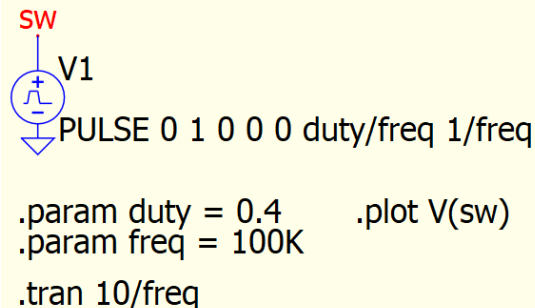
[1] For a single file with multiple analysis directive, comment can be use to control which one is active
Shortcut is [:]



Parameter for Pulse Source / Transient Convergence

Qspice : Params.qsch

- Pulse Source with Param
 - Setup duty and frequency parameters for pulse source can prevent manually calculate Ton and Toff

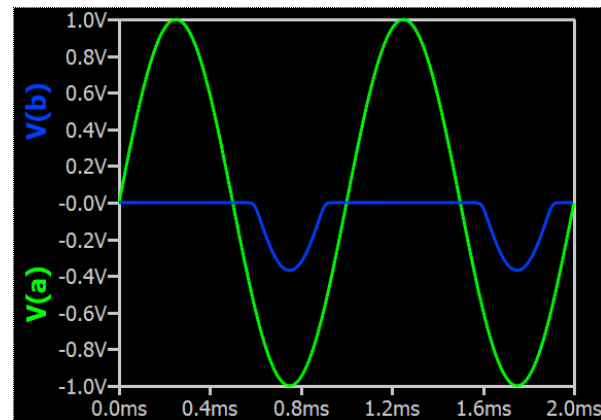
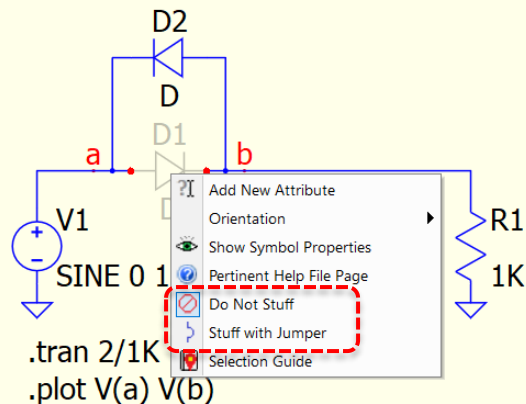


- In transient analysis, if simulation results is not convergence, try following options
 - Add **.option MAXSTEP=x**, where x is maximum step size for transient analysis
 - Sometimes it requires to limit step size especially circuit consist of pulse and logic
 - Goto Edit > Preferences, disable Fast (less accurate) Math
 - Enable Fast (less accurate) Math : QSPICE64.exe (runs faster and use more 64bit double)
 - Disable Fast (less accurate) Math : QSPICE80.exe (runs slower but use more 80bit long double)

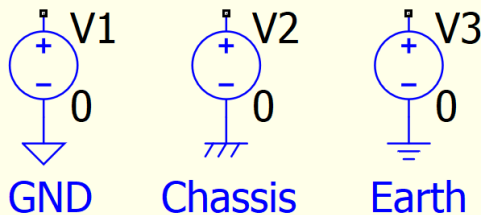
Open/Short Devices and 0 node synonyms

Qspice : Open Short Devices.qsch ; GND CHASSIS EARTH.qsch

- Open/Short Component
 - Right click on component, two options may help
 - [1] Do Not Stuff : Open
 - [2] Stuff with Jumper : Short



- GND, Chassis, Earth
 - These net names are synonyms to node "0"
 - ** synonyms only apply when schematic to netlist. You can use these names in .cir as normal net name in simulation
 - If you type a net with these names in schematic, they will auto convert into ground symbol. These symbols in netlist are all named as 0



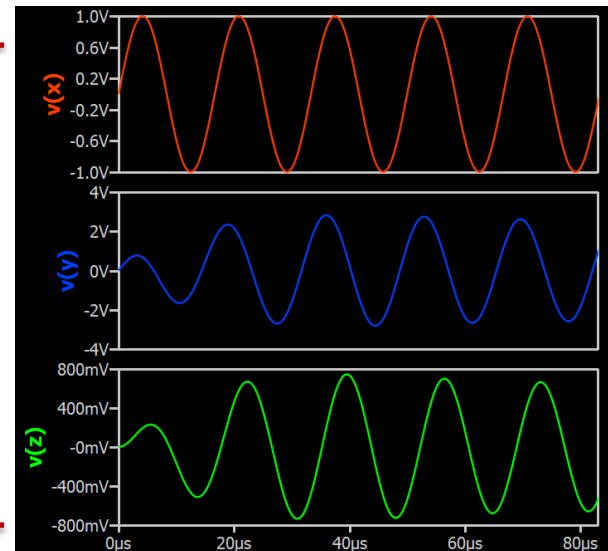
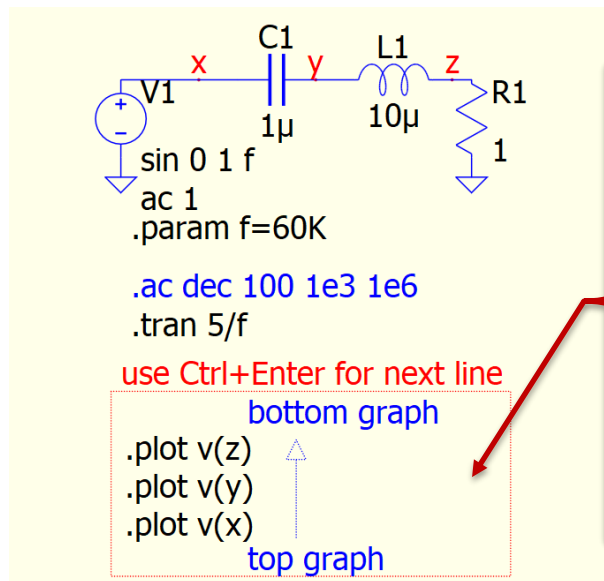
- View > Netlist

```
V1 ¥0 0 0
V2 ¥1 0 0
V3 ¥2 0 0
.end
```

Deterministic .plot method to define plots sequence in waveform viewer

Qspice : plot Sequence.qsch

- .plot can be used to define plot windows in waveform viewer
- However, the order of separated .plot command depends sequence when .plot command is added
- To ensure .plot command sequence in netlist, user can define .plot in a single text box, by using Ctrl+Enter for new line
- The first line will be plot at bottom and last line will be plot at top
- For .plot can tell waveform viewer what to plot
 1. Close waveform viewer before Run simulation
 2. No plot configuration file is present (i.e. [qschname].pfg are deleted in schematic directory)



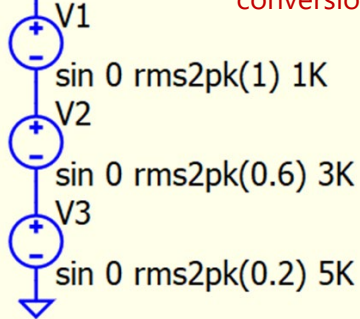
FFT in Waveform Viewer

Qspice : FFT waveform viewer.qsch

```
.func rms2pk(in) in*sqrt(2)
```

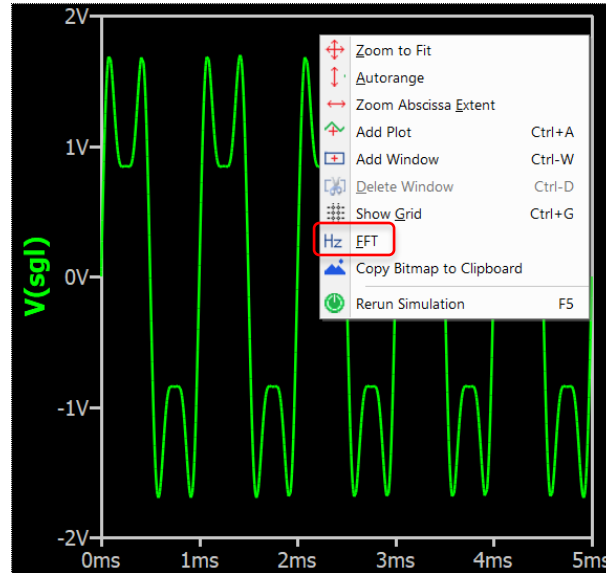
sgl

Use .func for rms to peak conversion

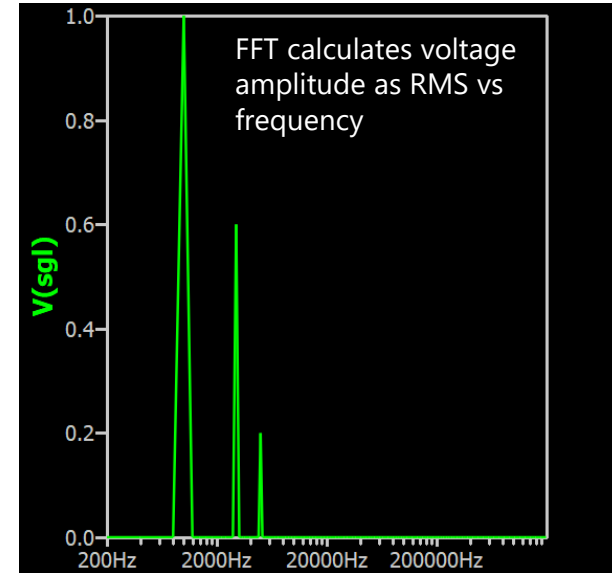


```
.tran 5/1K  
.plot V(sgl)
```

- [1] Right Click and select FFT
- [2] In FFT Setup, user can select Window Function



- [3] In FFT, right click y scale
- [4] In Axis setting, deselect (dB) can change to linear magnitude (no selection)



User Defined Function and Parameter

User-Defined Function .func

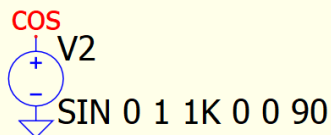
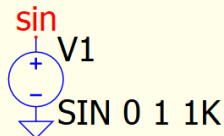
Qspice : Functions - func.qsch

- User-Defined Function

- Syntax
- .func NAME(args) {Expression}
- **** functions name must be with bracket ()**
- **e.g. fsum() instead of fsum**

- Purpose of function

- It preforms similar job as behavioral source, but without the need of math calculation in interest to become a signal source to overcrowding the schematic
- Function is hard-wired to exactly the thing you want to plot and doesn't need an argument



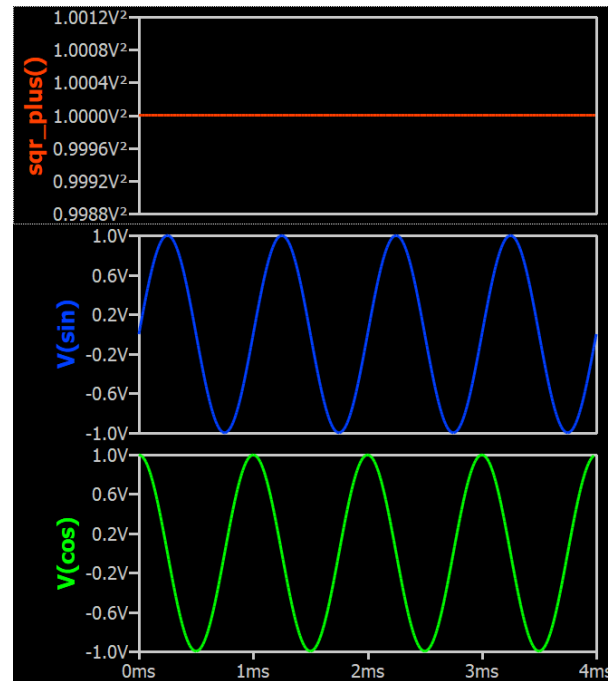
```
.tran 4/1K  
.plot V(sin)  
.plot V(cos)
```

maths function with input

```
.func f(a,b) a**2+b**2  
.func sqr_plus() f(V(sin),V(cos))  
.plot sqr_plus()
```

.func can call another function

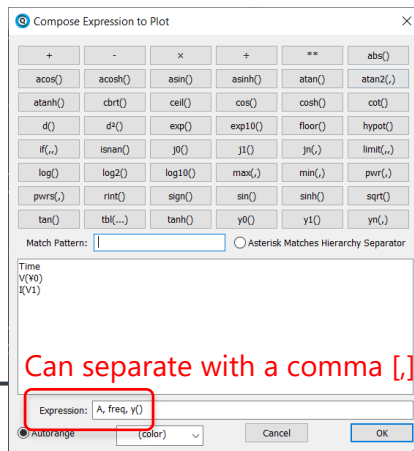
Use .plot to call calculated results of .func to be plotted



User Defined Parameters (.param) and Functions (.func)

Qspice : Func and Param Display.qsch

- .param and .func
 - .param is defined as NAME
 - .func is defined as NAME()
 - Both can be displayed in waveform viewer
 - [1] with .plot command
 - [2] right click > Add Plot > type parameters or function name (with bracket) in expression
 - Parameters and Functions name are not displayed in Add Plot list, but actual value are there



Dummy for Ground



Define Amplitude and Freq (in rad/s)

```
.param A = 2  
.param omega = 2*pi*1K
```

Calculate frequency in Hz

```
.param freq = omega/2/pi
```

Function to calculate a sine wave

```
.func y() A*sin(omega*time)
```

.tran and .plot

```
.tran 1/freq
```

```
.plot A
```

```
.plot freq
```

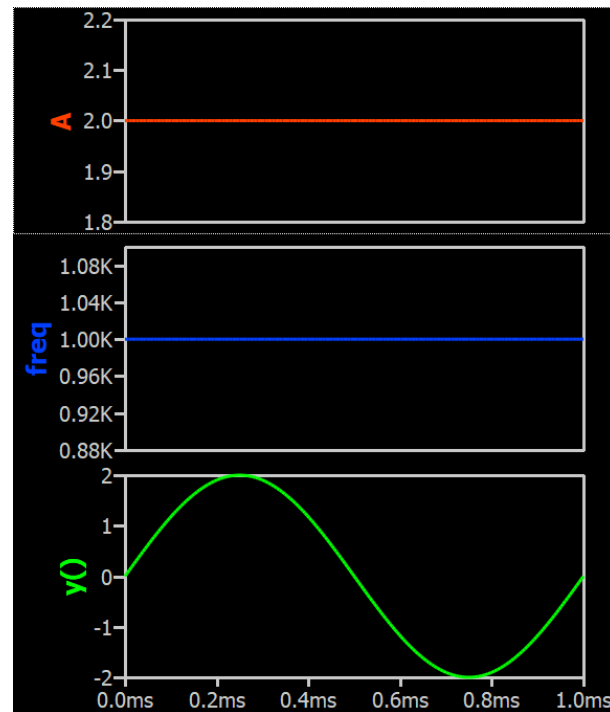
```
.plot y()
```

.options LISTPARAM

This option can print a list of the evaluated parameters

Output Window

```
--- Parameter Evaluations ---  
TEMP      = 27      "CKTTEMP"  
OMEGA     = 6.28319K "2*PI*1K"  
FREQ      = 1K      "OMEGA/2/PI"  
A         = 2       "2"  
--- User Defined Functions ---  
Y ()      {A*SIN(OMEGA*TIME) }
```



Basic Device Usage

Inductor (L) as Transformer

Qspice : L as Transformer - Two Winding.qsch / L as Transformer - Three Winding.qsch

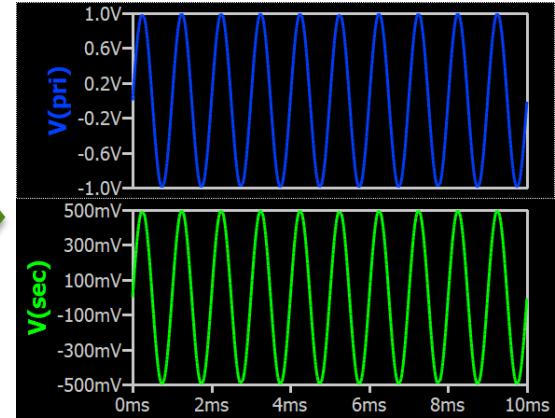
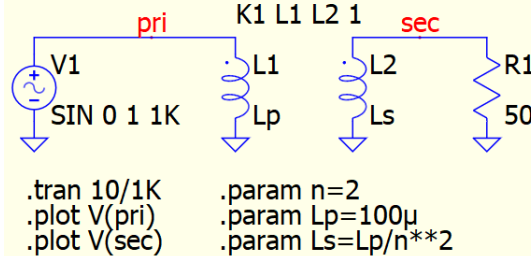
- L as Transformer

- $\frac{L_p}{N_p^2} = \frac{L_s}{N_s^2}$ and $n = \frac{N_p}{N_s}$
- $L_p = n^2 L_s$ or $L_s = \frac{1}{n^2} L_p$
- In general practice, we measure primary inductance of transformer (L_p) and know turn ratio (n)

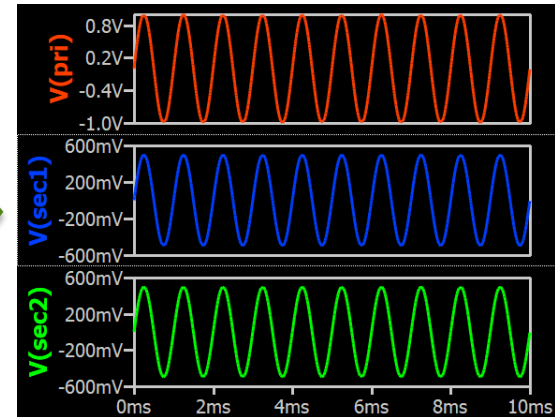
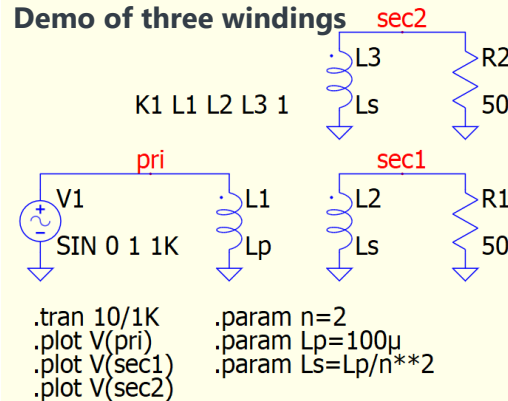
- Model

- Two or more coupled inductors are required
 - Not necessary but recommend press L two times to get an inductor symbol with a dot notation
- K is Mutual Inductance defines mutual coupling coefficient of coupled inductors
 - Ideal coupling : 1

Demo of two windings



Demo of three windings

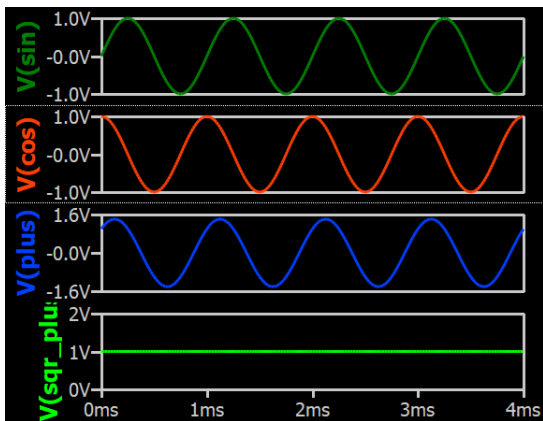
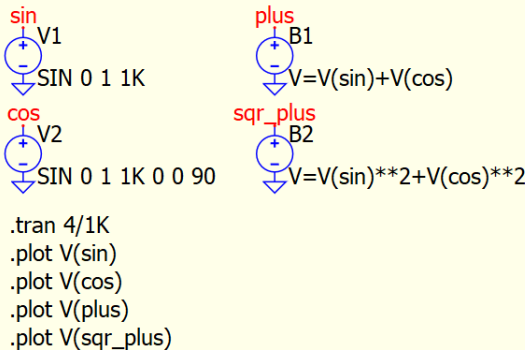


Arbitrary Behavioral Source [B]

Qspice : B - Functions.qsch ; E - Laplace.qsch

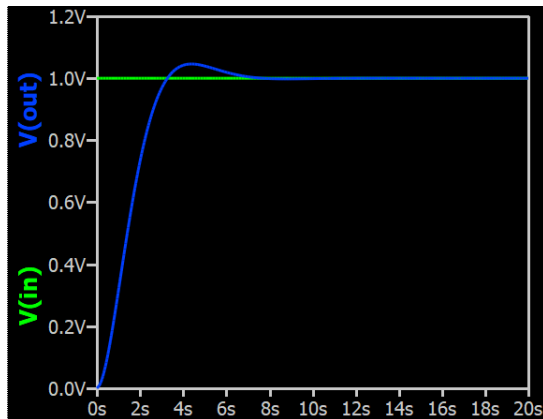
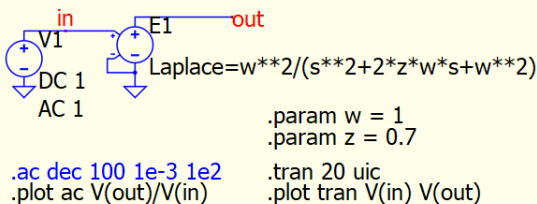
- Arbitrary Behavioral Source [B]
 - Mathematic functions and logical operators can be used
 - This is useful for mathematic calculation during simulation
 - It also support Laplace transfer function
- Remark
 - E, F, G, H source has similar application properties as B source
 - Recommend go to Qspice HELP for more information of functions and operators

Demo of functions with B-source



Demo of Laplace with E-source

Second-Order Transfer Function



Voltage Controlled Switch (S)

Qspice : Switch - instance param.qsch ; Switch - model.qsch

- S Switch

- S is voltage controlled switch
 - Voltage between control nodes can switch impedance between switch terminals
- Switch can be configured with instance parameters or .model
- Simulation results of these two examples are identical

Add Instance parameter
Right click on S1, Add New Attribute

