
Qspice - Entry User Guide

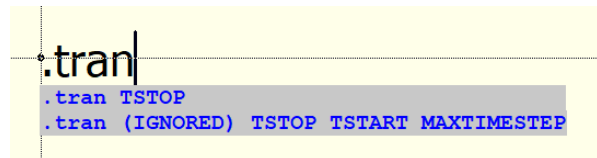
KSKelvin Kelvin Leung

Created on 9-15-2023

Last Update on 8-12-2024

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



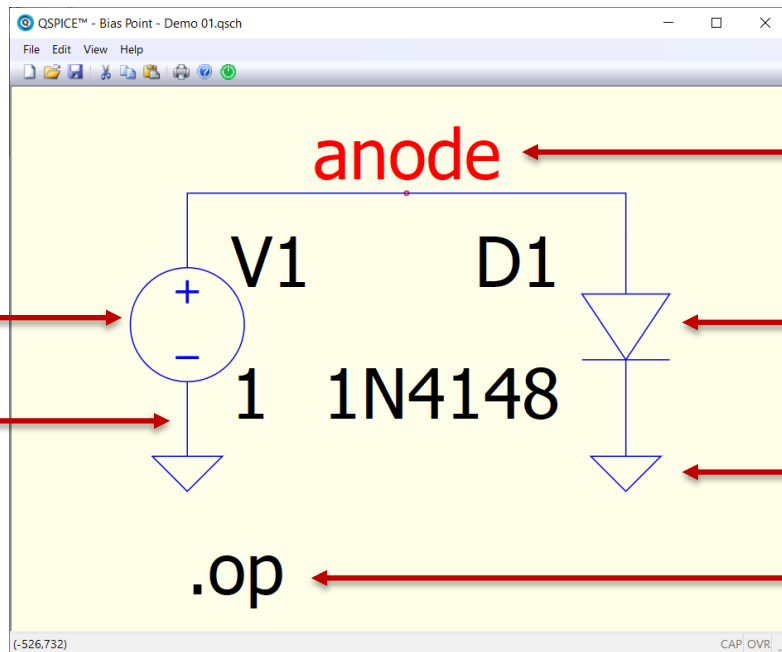
Part 01

Start 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



[1] Press [V] to place a voltage source, Type [1] in <val>

- Keep pressing [V] can cycle symbol

[4] Press [W] to place wire

anode

[5] Press [N] for a net name on top wire

[2] Press [D] to place a diode, (Ctrl-R to rotate)
Right click on diode > selection guide
Select 1N4148 diode

[3] Press [G] to place ground

[6] Press [.] to start typing for directive, input
.op [hint syntax underneath during typing]

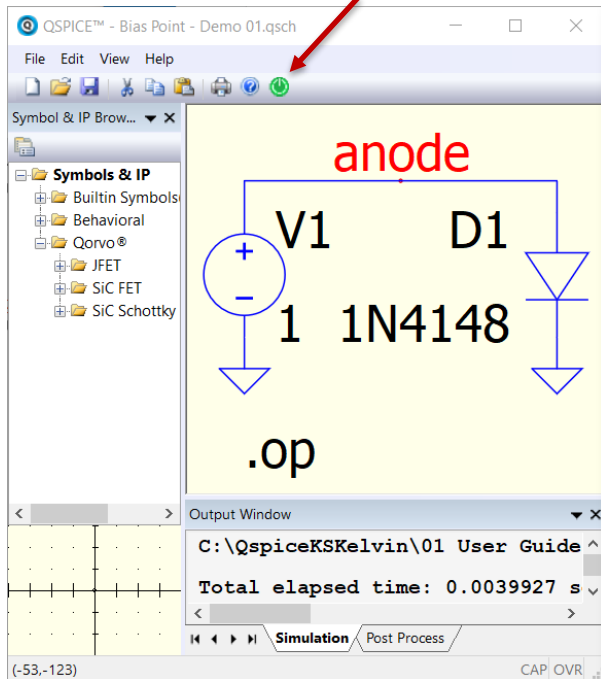
** another method is to Press [T] to start input text, and type **.op**

Bias Point Analysis (.op) : DC Operation Point

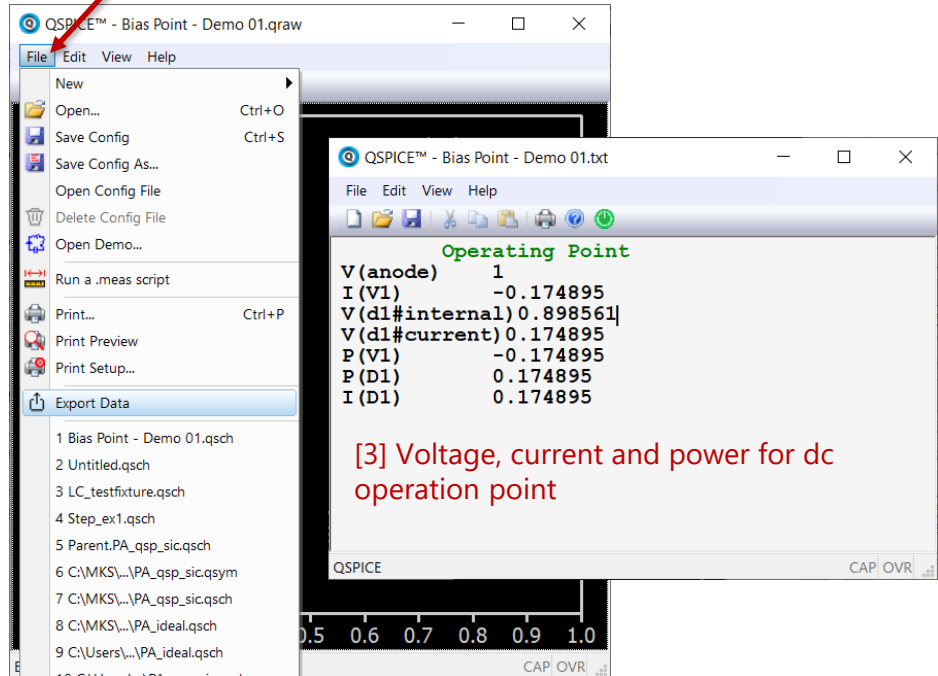
Qspice : Bias Point – Demo 01.qsch

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

[1] Run to simulate the circuit

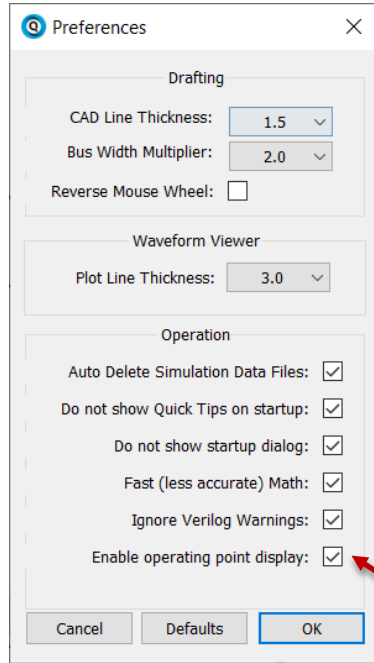


[2] Result of .op can be obtained by
File > Export Data

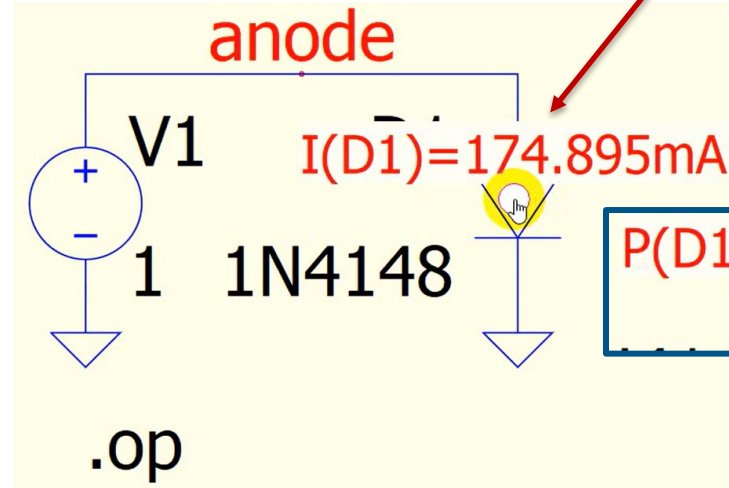


[3] Voltage, current and power for dc
operation point

Feature : Operating Point Display in Schematic

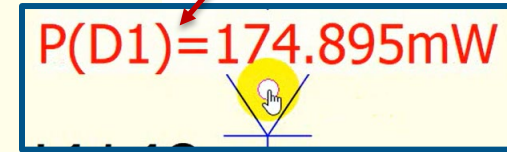


[1] Edit > Preferences
Select Enable operating point display



[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)

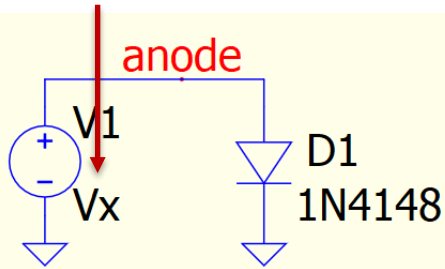
Qspice : Bias Point - Demo 02.qsch

[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

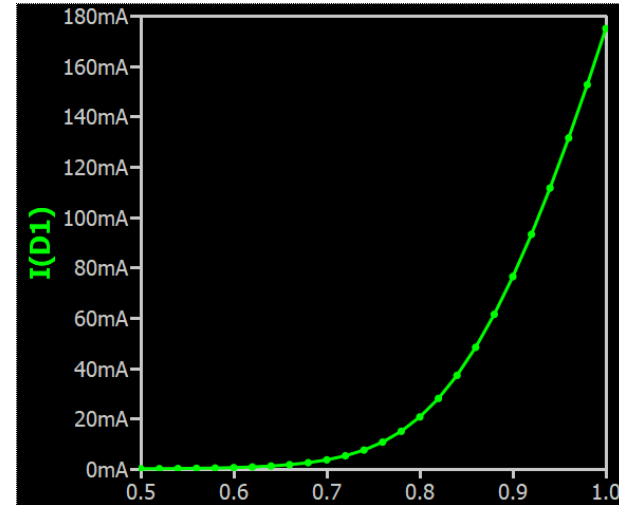
[2] Add **.param** to define Vx
value in this simulation

.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

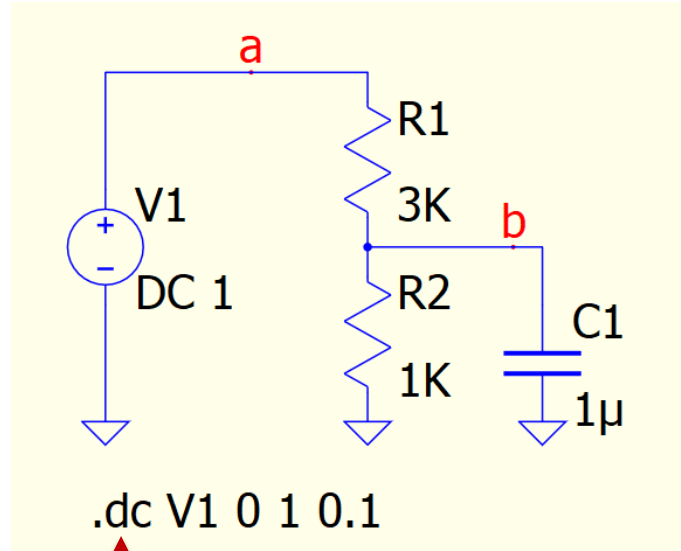


X-axis of this plot is Vx based on .step

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

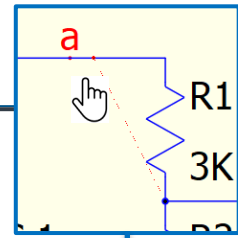


[1] **.dc** to sweep V1 from 0V to 1V with 0.1 per step

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

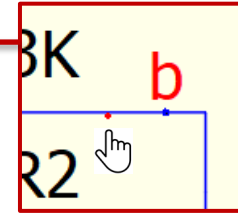
Differential voltage probe : V(a,b)

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

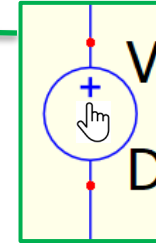


Voltage probe : V(b)

click node b

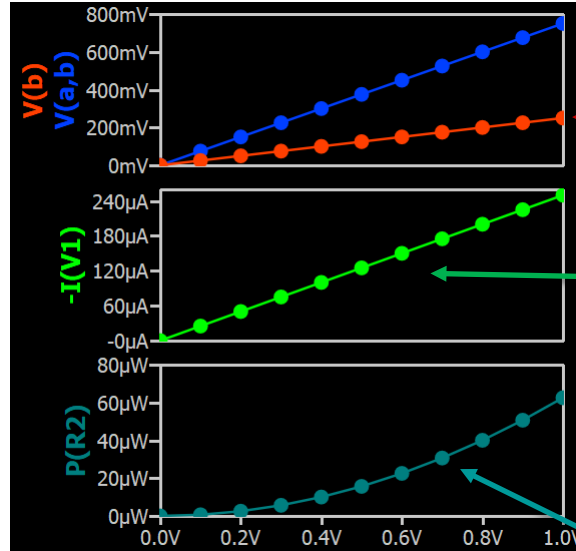
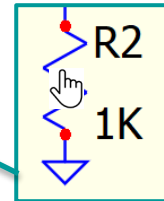


Current probe : I(V1)
Hover on device



Power probe : P(R2)

Press Ctrl and Hover on device

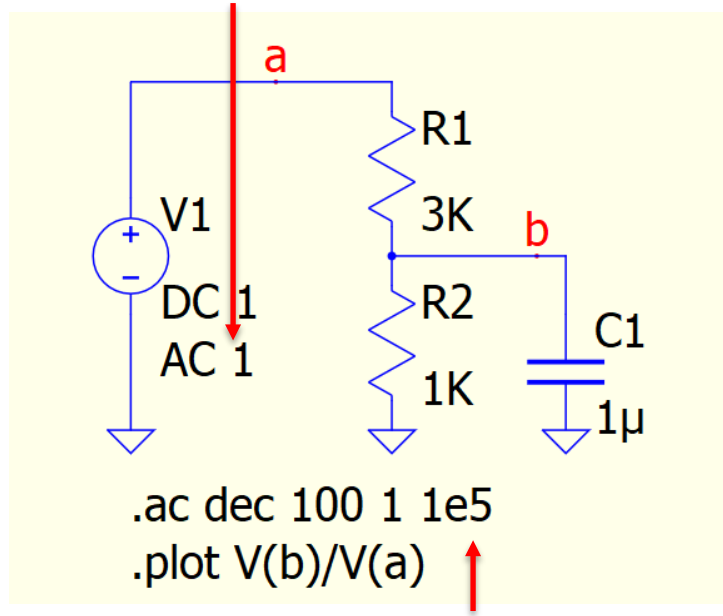


AC Analysis (.ac)

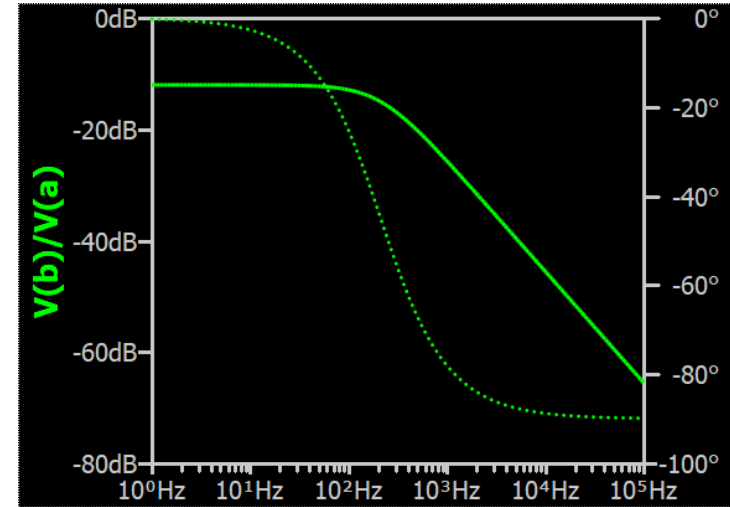
Qspice : AC Analysis - Demo 01.qsch

[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.



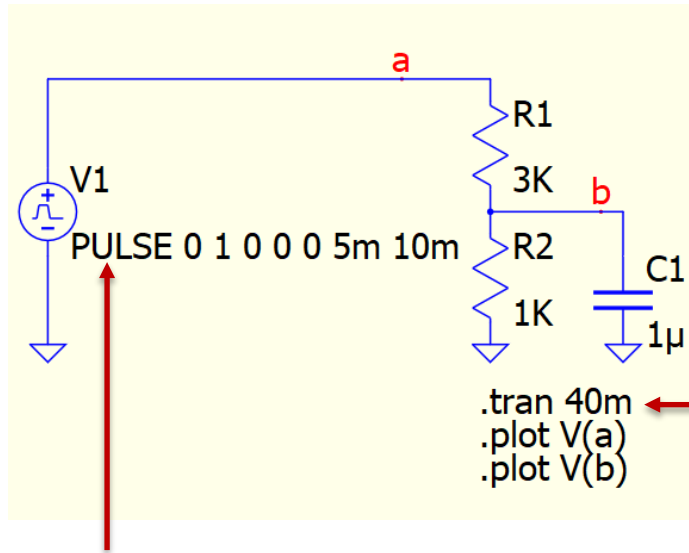
[2] this is to sweep AC source frequency from 1Hz to 1e5Hz (100kHz) with 100 points per decade

Non-Linear Transient Analysis (.tran)

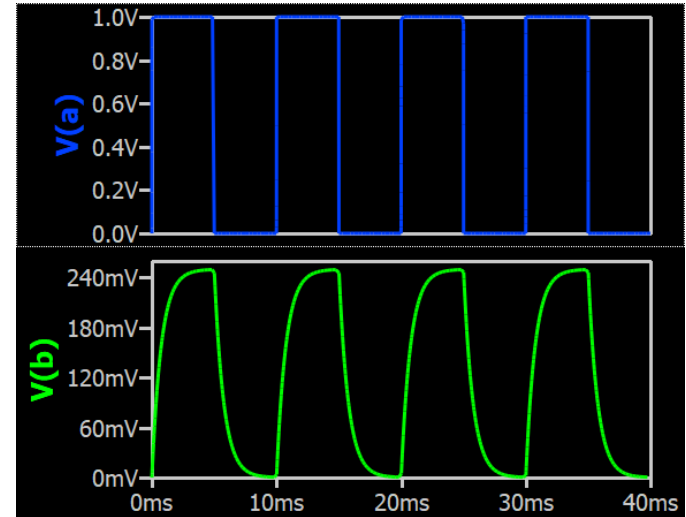
Qspice : Transient Analysis - Demo 01.qsch

[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

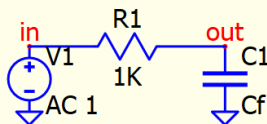
Step User-Defined Parameter (.step)

Qspice : Step - filer (.ac).qsch | Step - filer (.tran).qsch

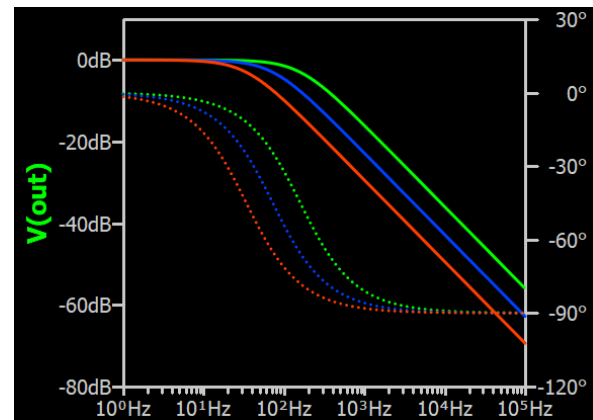
- .step
 - Step User-Defined Parameter (.step) is used to run a simulation multiple times changing one or more parameters
 - In waveform windows, press F6 to show Simulation Step Tool and can review color and parameter relationship
 - Or right click > Simulation Step Tool

Simulation Step Tool	
Step No.↓	CF
1	1e-06
2	2.2e-06
3	4.7e-06

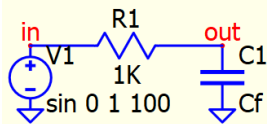
Example of .step in .ac analysis



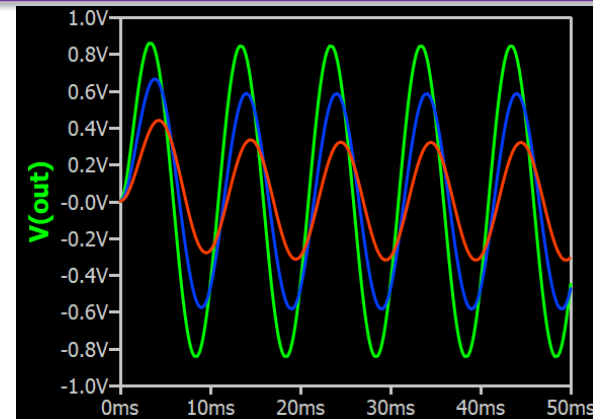
```
.param Cf=1μ ; only need this if .step is comment  
.step param Cf list 1μ 2.2μ 4.7μ  
  
.ac dec 100 1 100K  
.plot V(out)
```



Example of .step in .tran analysis



```
.param Cf=1μ ; only need this if .step is comment  
.step param Cf list 1μ 2.2μ 4.7μ  
  
.tran 5/100  
.plot V(out)
```



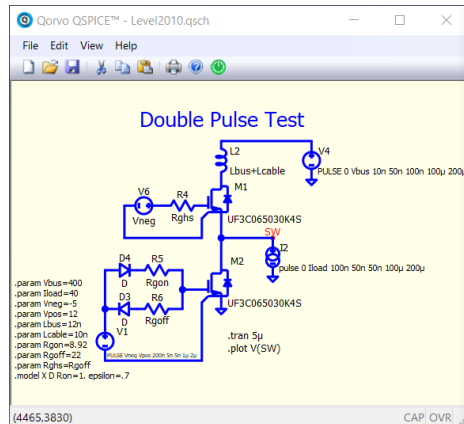
Part 01

Supplementary

SPICE Simulation Workflow

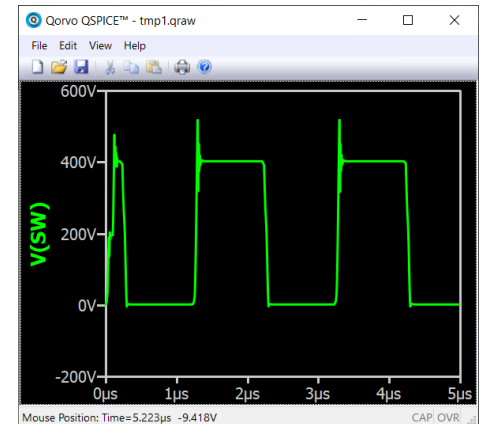
- SPICE Simulation Workflow

- It is important to understand the concept of SPICE simulation, here is workflow of Qspice
- QSPICE schematic drawing window is from QUX.exe, after you finish schematic drawing and hit "Run Simulation", this is what happen
 - QUX.exe convert schematic (.qsch) into a text based netlist (.cir)
 - QSPICE64.exe or QSPICE80.exe run netlist (.cir) and output results into data file (.qraw)
 - As a user, you may not aware there is a conversion of .cir as this process is run silent in background
 - QUX.exe run a waveform window to plot data file (.qraw)
 - QPOST.exe run .meas and .four directive in netlist (.cir) for data file (.qraw) and return result in output window
- By understanding this workflow, you may aware that you can troubleshoot problem from netlist (.cir). SPICE run a netlist which is text based, QUX.exe is only to convert graphical schematic into a netlist for simulation, other spice program may accept this netlist directly or with a bit of modification




```
* C:\Program Files\QSPICE\Examples\Level2010.cir
V4 N08 0 PULSE 0 Vbus 10n 50n 100n 100u 200u
V6 N04 N03 Vneg
R4 N04 N09 Rghs
R5 N07 N06 Rgon
R6 N05 N06 Rgoff
D3 N05 N02 D
D4 N02 N07 D
L2 N08 N10 Lbus+Lcable
V1 N02 N01 PULSE Vneg Vpos 200n 5n 5n 1u 2u
I2 0 SW pulse 0 Iload 100n 50n 50n 100u 200u
M1 N10 N09 SW N03 UF3C065030K4S NMOS
M2 SW N06 0 N01 UF3C065030K4S NMOS
.param Vbus=400
.param Iload=40
.param Vneg=-5
.param Vpos=12
.param Lbus=12n
.param Lcable=10n
.tran 5u
.plot V(SW)
```

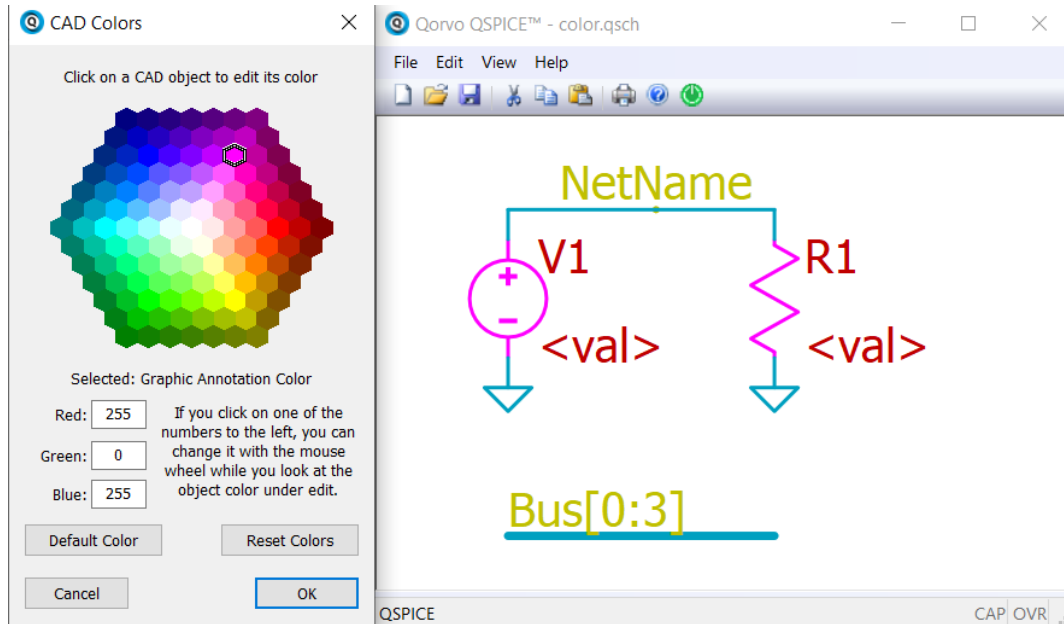
View > Netlist



Schematic Window – Color Scheme

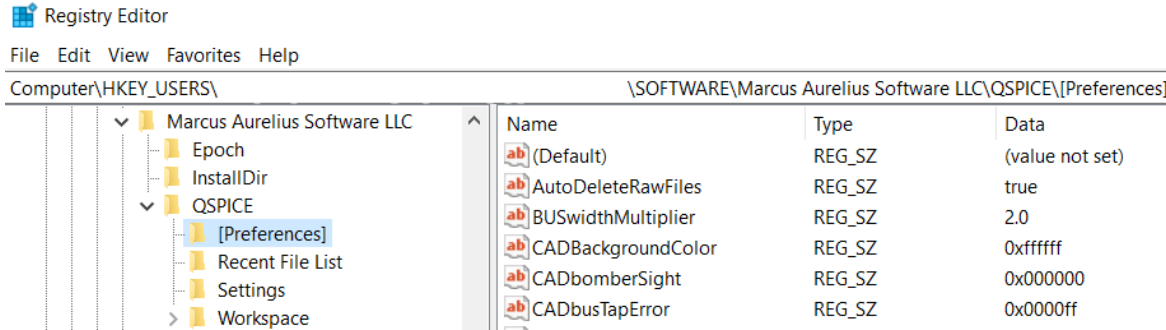
- Change Color Scheme

- Edit > Color Preferences  Color Preferences
- Keep open **CAD Colors** window, select items in schematic and assign color you want



Qspice Preferences

- Qspice preferences are stored in regedit



- Procedure to find regedit location
 - Run Registry Editor
 - Select HKEY_USERS
 - Edit > Find (Ctrl-F)
 - Find what: qspice
 - Look at: Keys only
 - Registry location:
 - Computer\HKEY_USERS\%\%\%\%\SOFTWARE\Marcus Aurelius Software LLC\QSPICE

Part 02

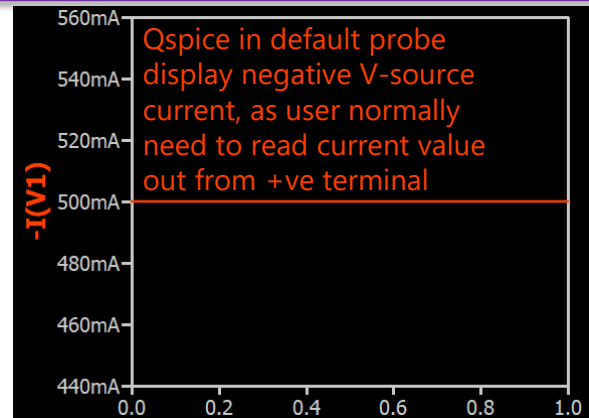
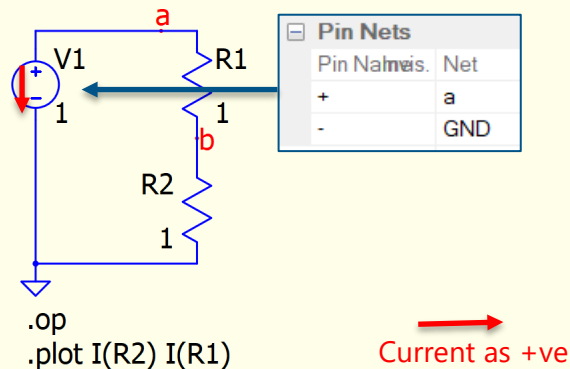
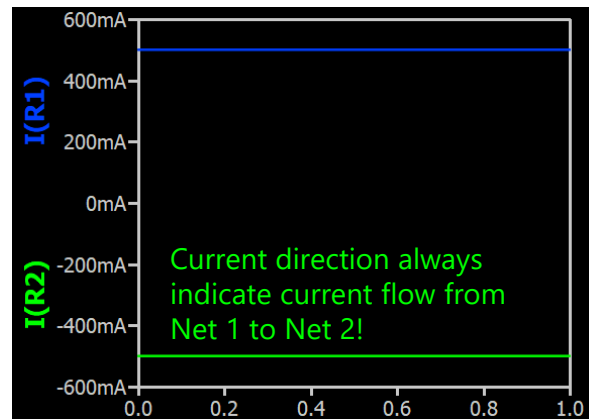
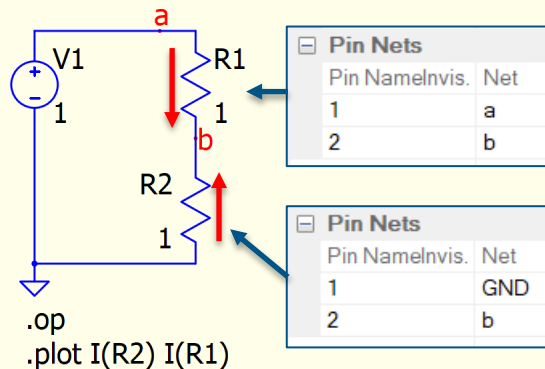
Useful Technique

Current Representation in Spice

Qspice : Current Representation - Explain.qsch

- Current Representation

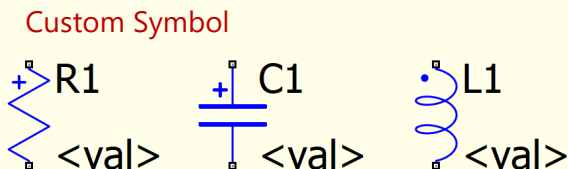
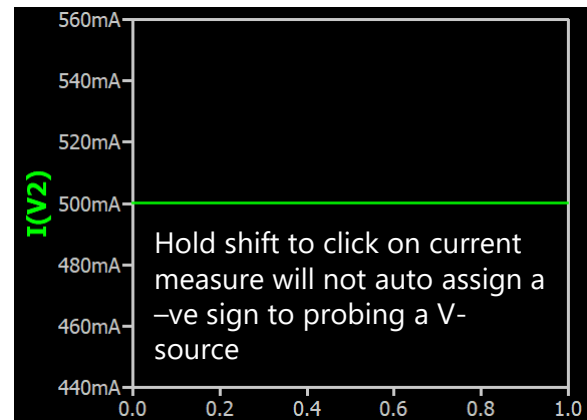
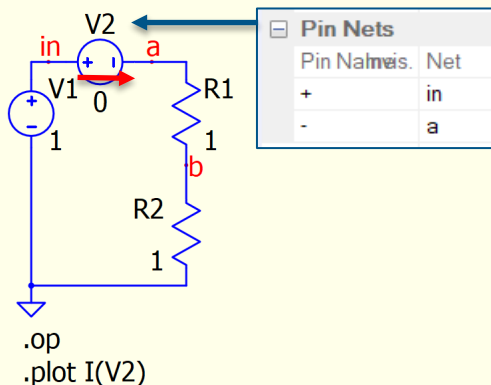
- It is important to note that rotating a device in Spice GUI may appear to reverse the current direction, and here is the reason why
- For device likes R, L, C etc., positive current is defined as flowing from device Pin 1 to Pin 2
- Right click on a device, select *Show Symbol Properties*, it can display pin and net relationships
- In this example
 - $I(R2)$ shows negative sign in simulation because positive is defined as current flow from Net GND to Net b
 - By definition, $-I(V1)$ is current flowing out from its +ve terminal. Therefore, simulation gives a positive value in $-I(V1)$



Current Representation in Spice

Qspice : Current Representation - 0V Source.qsch | Current Representation - Custom.qsch

- Current Representation
 - A common technique to prevent confusion in current direction is to use a 0V voltage source for current measurement, as a 0V voltage source has +ve and -ve pin names in its symbol
 - User can create custom symbol for standard devices by adding a positive pin indicator to devices
 - KSKelvin Github includes Symbol library with Qspice alternative custom symbol
 - <https://github.com/KSKelvin-Github/Qspice/>

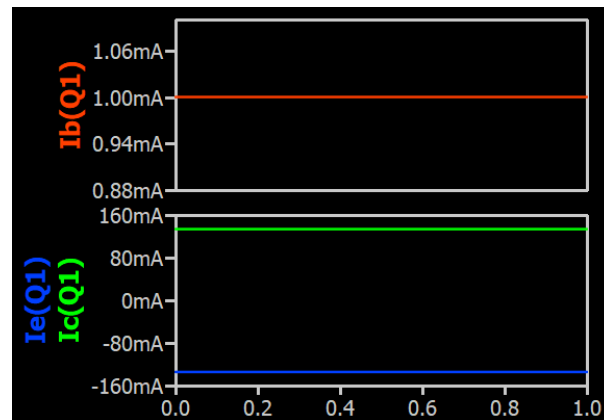
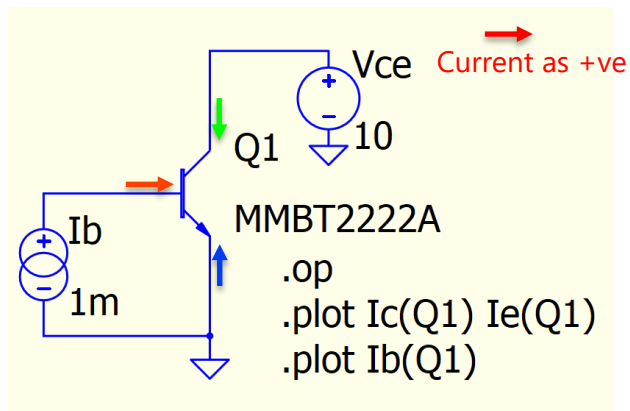


Press L two times for a phase dot symbol in Qspice

Current Representation in Spice

Qspice : Current Representation - Multi-terminals.qsch

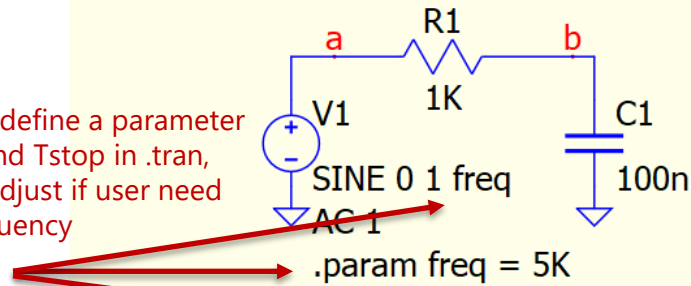
- Multi-terminals Device
 - Current in multi-terminal devices like transistors and MOSFETs has current probes from a node
 - Positive (+ve) node current represents current flowing INTO the device
 - In this NPN transistor example, $I_b(Q1)$ and $I_c(Q1)$ both return positive values as the simulated current flows into Q1. However, $I_e(Q1)$ is negative as the simulated current flows out from Q1, in a reverse direction compares to its definition



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

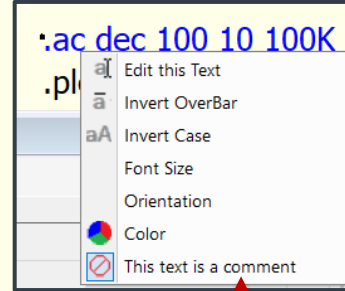


`.tran 10/freq`

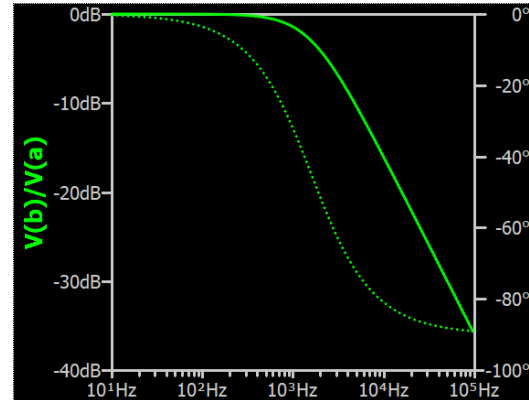
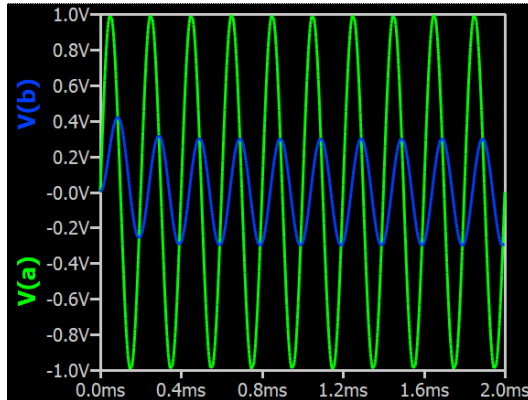
`.plot tran V(a) V(b)`

`.ac dec 100 10 100K`

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



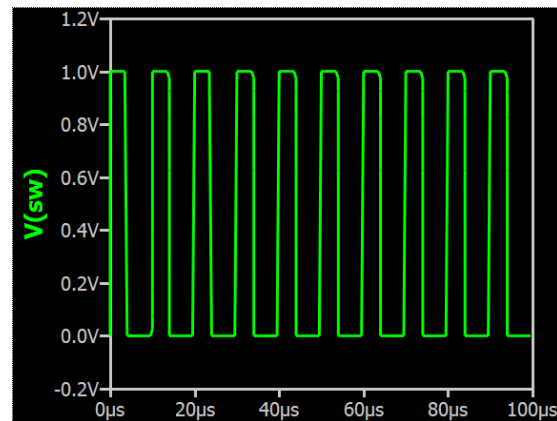
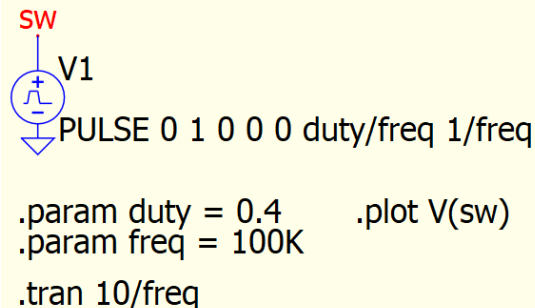
[1] For a single file with multiple analysis directive, comment can be use to control which one is active
Shortcut is [:]



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

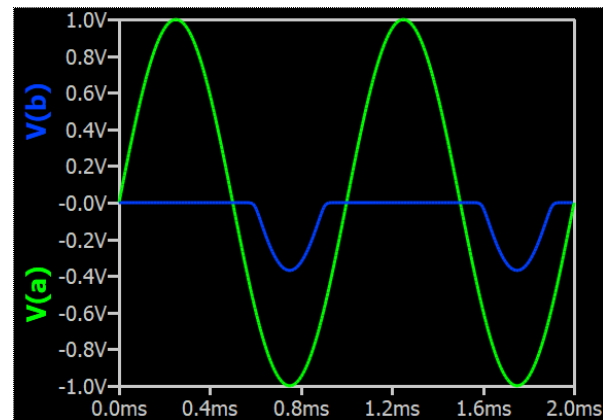
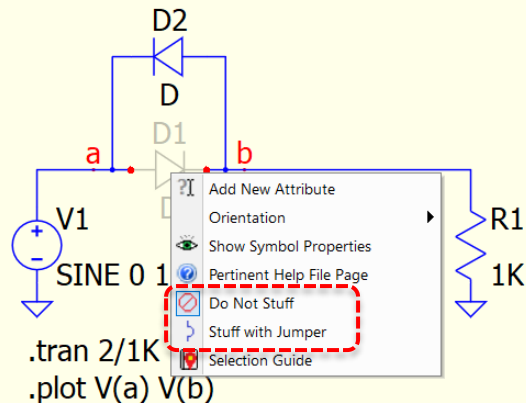


- In transient analysis, if simulation results is not convergence, try following options
 - Add **.option MAXSTEP=x**, where x is maximum step size for transient analysis
 - Sometimes it requires to limit step size especially circuit consist of pulse and logic
 - Goto Edit > Preferences, disable Fast (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)

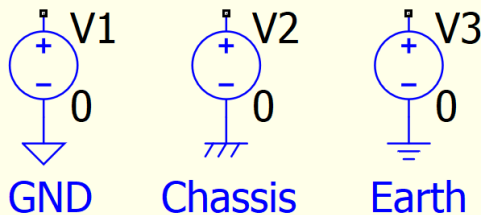
Open/Short Devices and 0 node synonyms

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



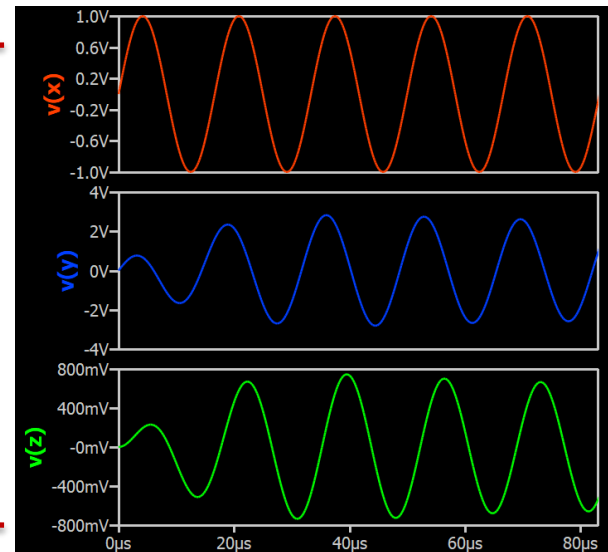
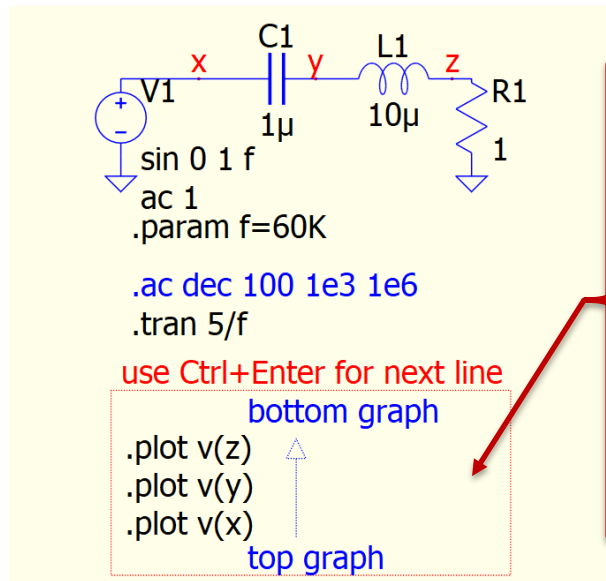
- View > Netlist

```
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)

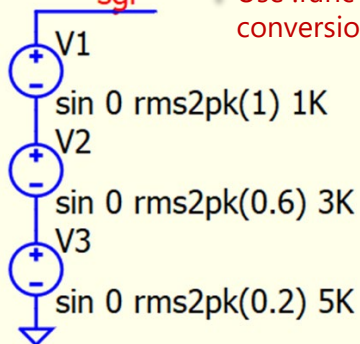


FFT in Waveform Viewer

Qspice : FFT waveform viewer.qsch

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

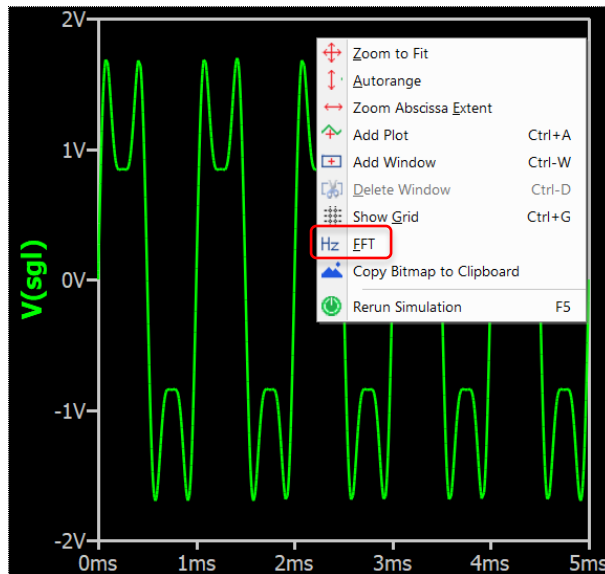
Use .func for rms to peak conversion



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

[1] Right Click and select FFT

[2] In FFT Setup, user can select Window Function

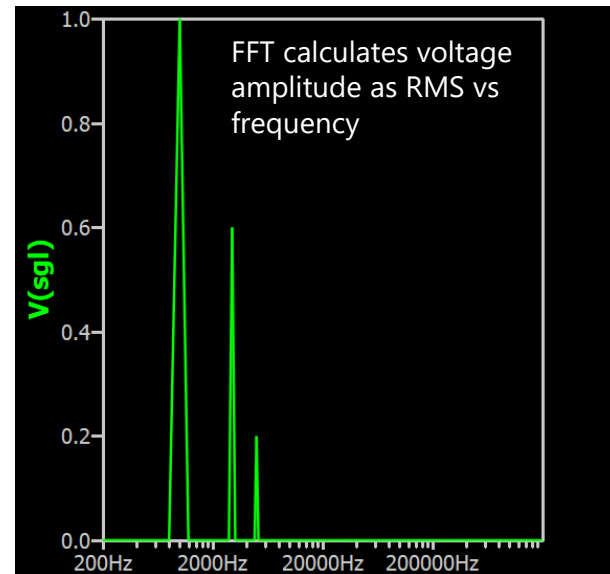


Window Function:

Rectangular(none)

[3] In FFT, right click y scale

[4] In Axis setting, deselect (dB) can change to linear magnitude (no selection)



Part 03

User Defined Function and Parameter

User-Defined Function .func

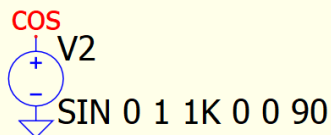
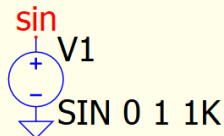
Qspice : Functions - func.qsch

- User-Defined Function

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

- Purpose of function

- It performs 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



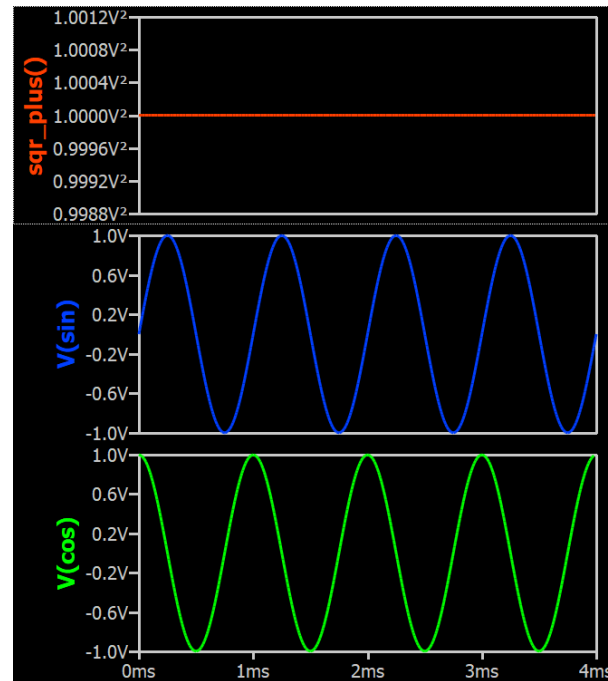
```
.tran 4/1K  
.plot V(sin)  
.plot V(cos)
```

↑ **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**

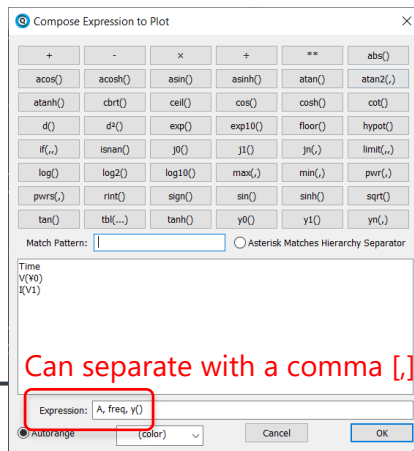
↑ **Use .plot to call calculated results of .func to be plotted**



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
```

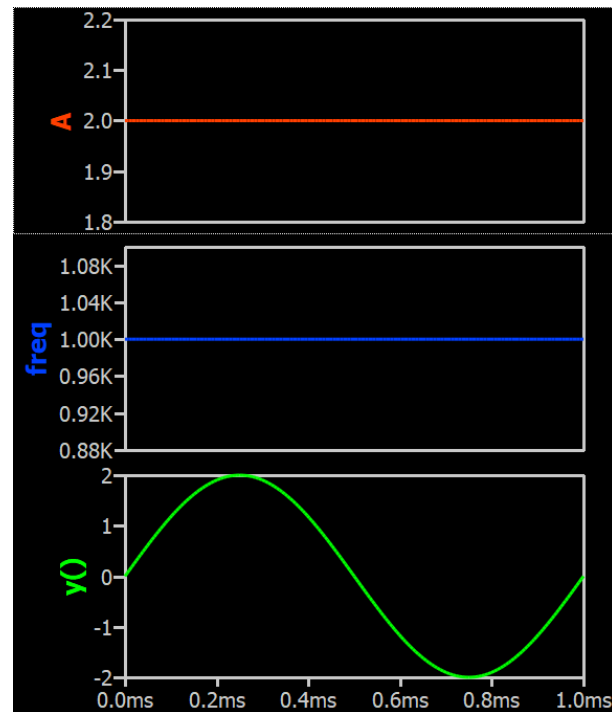
```
.plot y()
```

.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) }
```



Part 04

Simulation Technique

Transformer with Coupled Inductor (L)

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

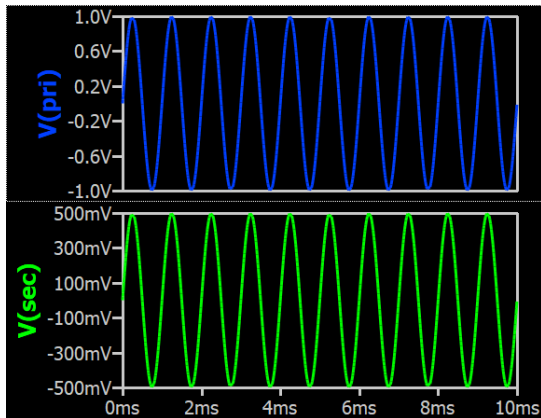
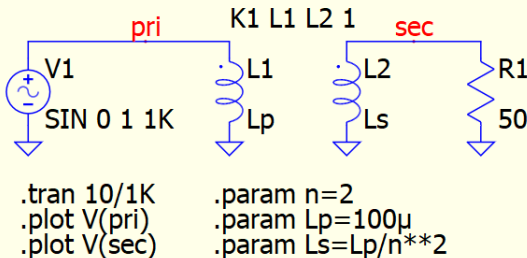
- L as Transformer

- $\frac{L_p}{N_p^2} = \frac{L_s}{N_s^2}$ 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)

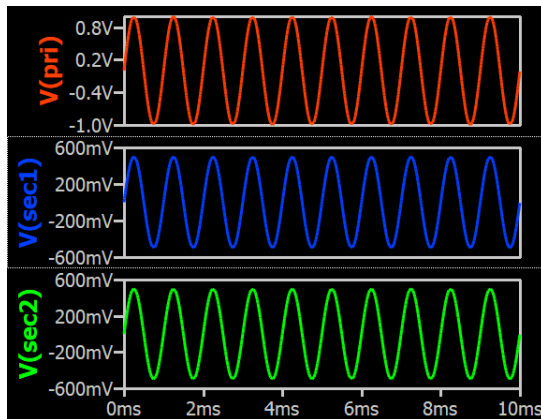
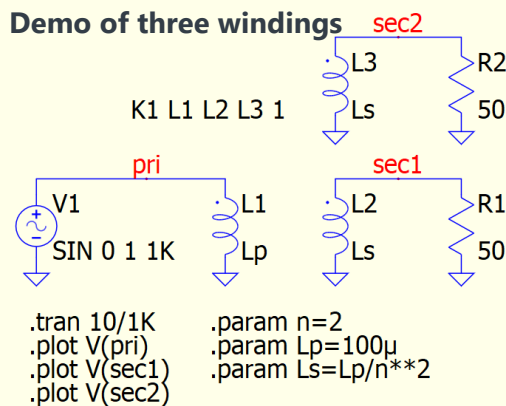
- Model

- 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

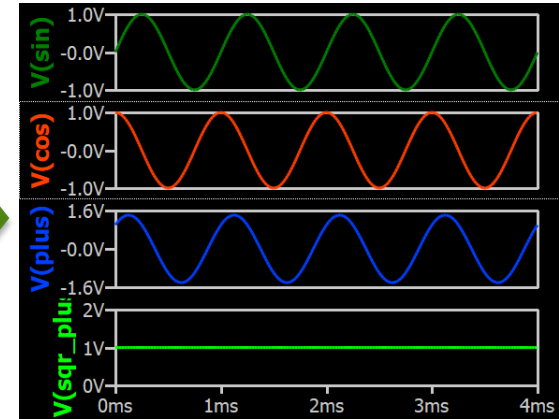
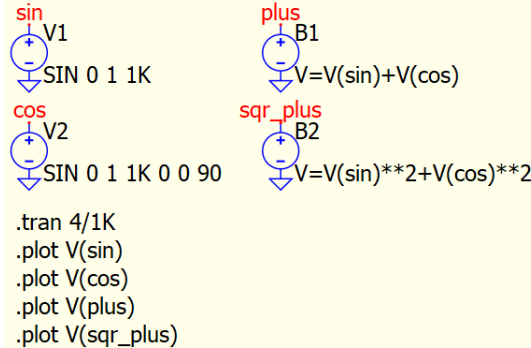


Arbitrary Behavioral Source [B]

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

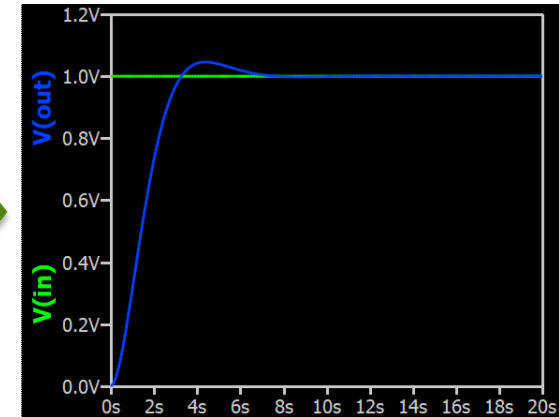
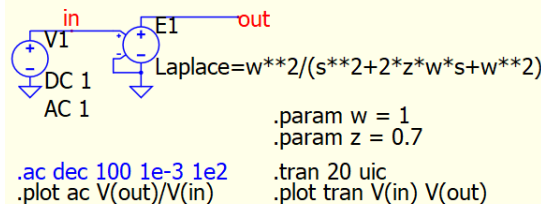
- Arbitrary Behavioral Source [B]
 - Mathematic functions and logical operators can be used
 - This is useful for mathematic calculation during simulation
 - It also support Laplace transfer function
- Remark
 - E, F, G, H source has similar application properties as B source
 - Recommend go to Qspice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

