# Qspice - Entry User Guide by KSKelvin

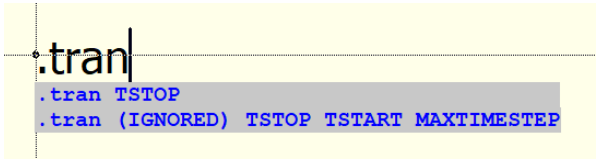
KSKelvin Kelvin Leung

Created on 9-15-2023
Last Update on 5-16-2025

# Qspice

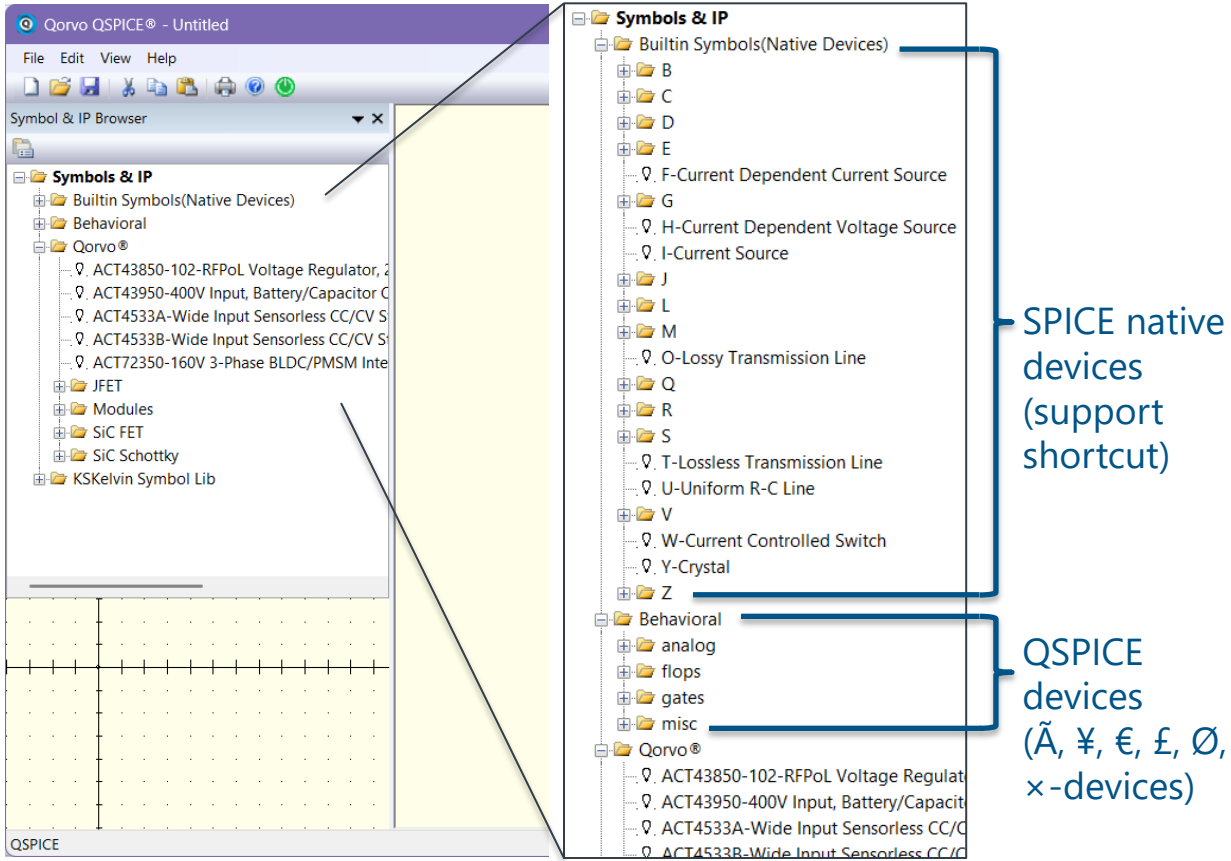
- Qspice
  - Author : Mike Engelhardt
  - Website : https://www.qorvo.com/design-hub/design-tools/interactive/qspice
  - Direct Qspice Installer Download Link : https://getqspice.com/InstallQSPICE.exe

- GUI (Graphical User Interface)
  - Most inputs require keyboard shortcuts
    - For example, pressing R gives a resistor, pressing R again cycles through different symbols, Ctrl-R rotates, W draws a wire, etc.
  - GUI provides hints for the syntax underneath your typing, which eliminate the need for diagnose/toolbox
    - Some users may not appreciate this initially, but in my experience, this GUI is more convenient
    - Example of hint syntax underneath



# Qspice – Symbol & IP Browser

- Devices
  - View > Symbol & IP Browser
    - Shortcut F2



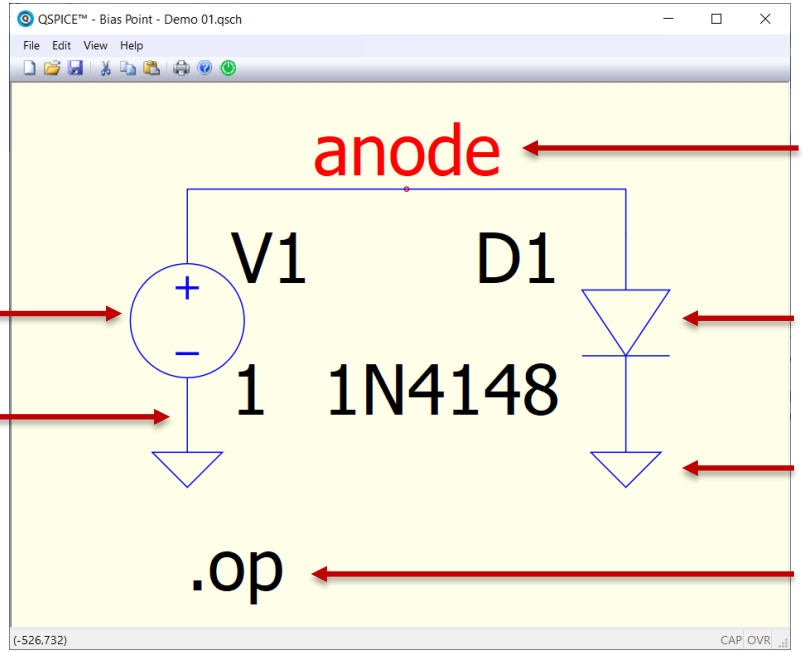
# Part 01
# Introduction to Simulation

# Qspice Command

- Analysis Directive
  - **.op** : Bias Point Analysis / Operation Point Analysis
    - DC operation point analysis used to calculate the DC steady-state voltage and current
  - **.dc** : DC Sweep
    - DC sweep analysis that is similar to .op but can sweep the source voltage/current values during the analysis
  - .**ac** : AC Analysis
    - AC analysis that utilizes phasors for calculations. Before running .ac, it automatically performs a .op for the DC operation point, and .ac is simulated based on this DC bias condition
  - **.tran** : Non-Linear Transient Analysis
    - Transient analysis, which by default runs a .op before .tran. The .tran is executed based on this bias point condition at t=0s. Users can skip the .op by adding UIC in the .tran directive
  - **.bode** : Frequency Response Analysis [topic not cover in this report]
- This section includes
  - **.param** : User-Defined Parameter
  - **.step** : Step User-Defined Parameter
  - **.plot** : Plot Suggestion

# Draw your first schematic – step by step procedure

Follow the sequence of step [1] to step [6]




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

- Keep pressing [V] to cycle throught symbols

[4] Press [W] to place wire

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

[2] Press [D] to place a diode, (Ctrl-R to rotate, Ctrl-M to horizontal flip)
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]

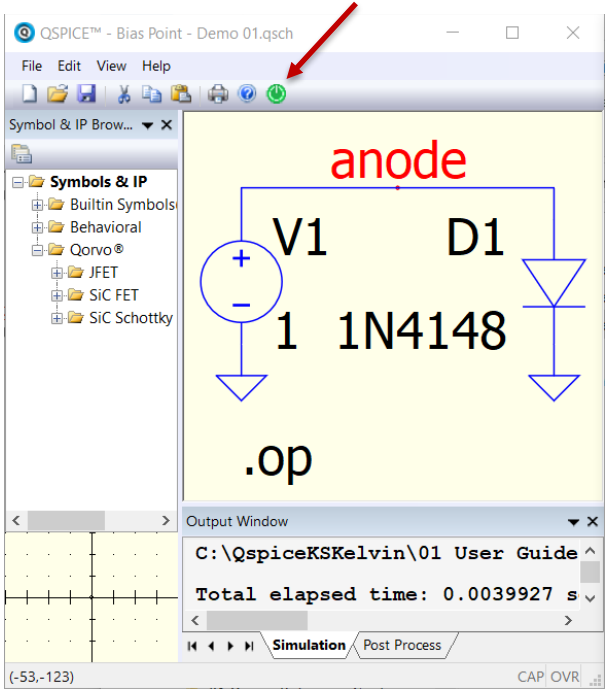
** another method is to Press [T] to start input text, and type **.op**

# Bias Point Analysis (.op) : DC Operation Point

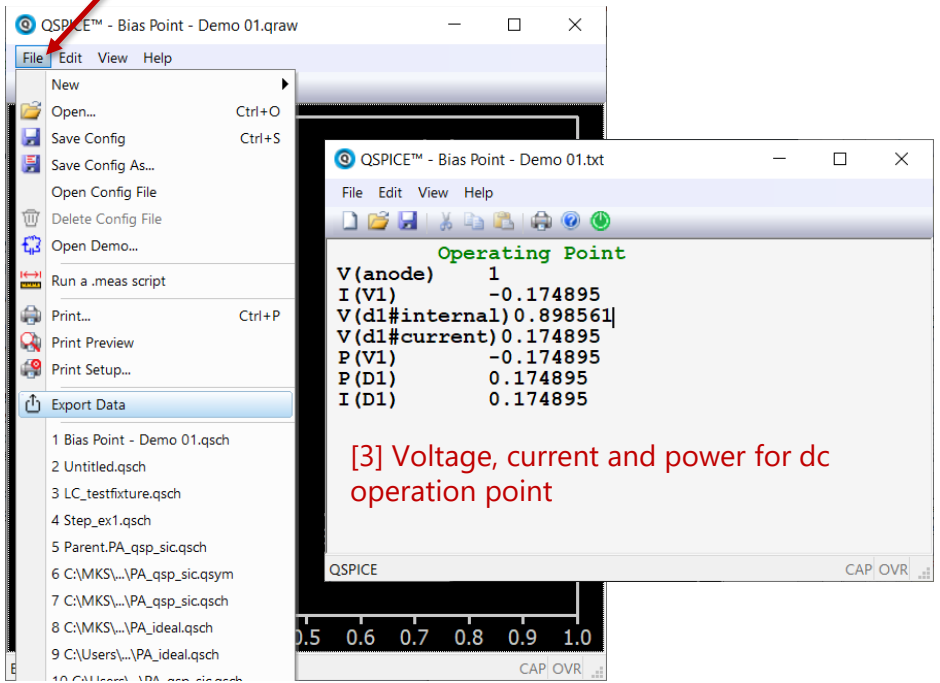
## Qspice : Bias Point – Demo 01.qsch

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

[1] Run to simulate the circuit

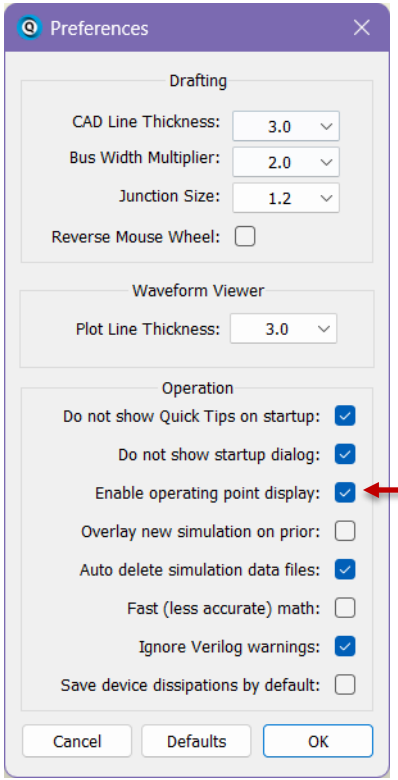



[2] Result of **.op** can be obtained by File > Export Data

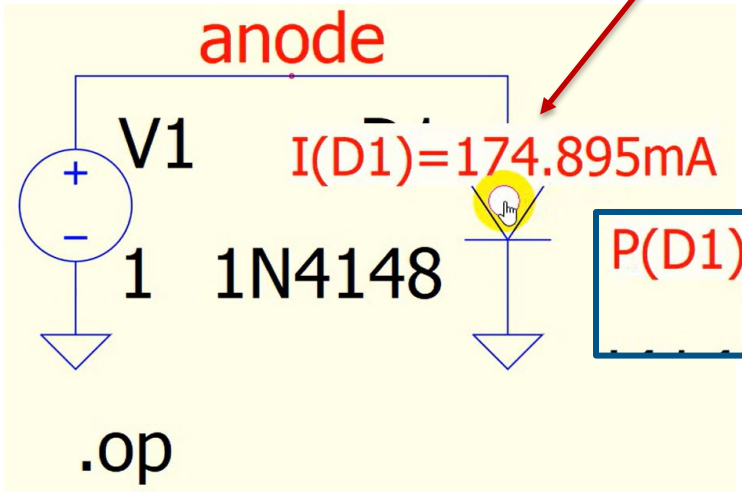



[3] Voltage, current and power for dc operation point

# Feature : Operating Point Display in Schematic

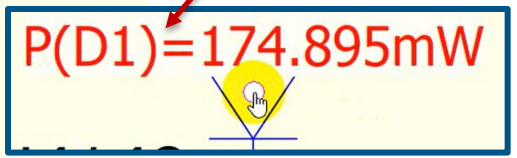



[1] Edit > Preferences
Select Enable operating point display




[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

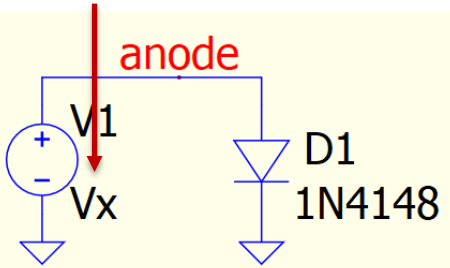
# Bias Point Analysis (.op) with Step User-Defined Parameter (.step)

## Qspice : Bias Point - Demo 02.qsch

[0] This example demonstrates using **.op** to plot I-V characteristic of diode D1 1N4148

- The concept is to sweep the anode voltage from 0.5V to 1V and plot the diode current

[1] Change V1 value to Vx
Vx is a variable parameter. ** In Qspice, it accepts parameter without curly bracket {}




| Pin Nets | |
|---|---|
| Pin Names | Net |
| A | anode |
| K | GND |

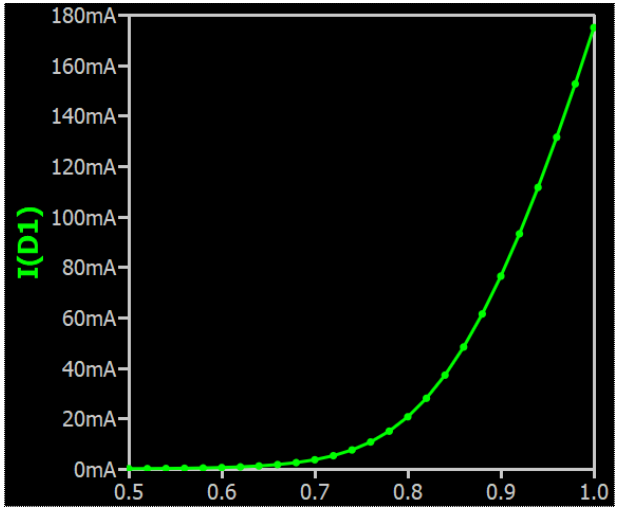
** Current direction is defined by pin order
+I(D1) represent flow from A to K

```
.op
.param Vx=1
.step param Vx 0.5 1 0.02
.plot I(D1)
```

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

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

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

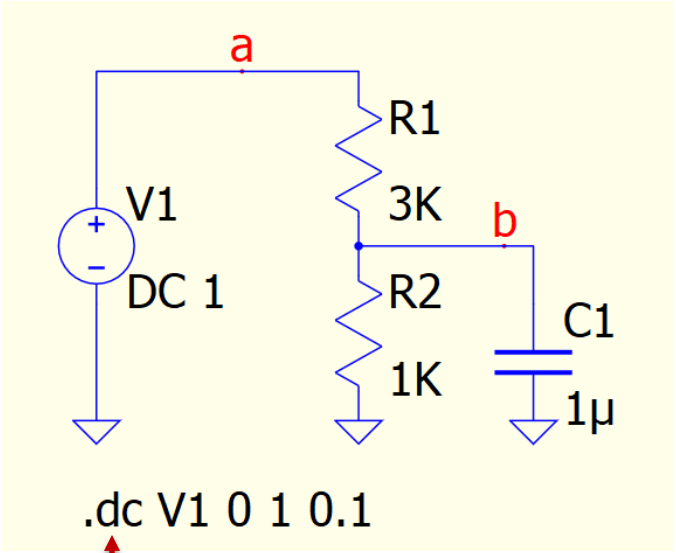



X-axis of this plot is Vx based on .step

# DC Sweep (.dc) and Probing Signal Waveform

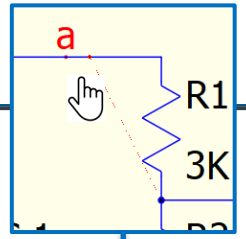
## Qspice : DC Sweep - Demo 01.qsch

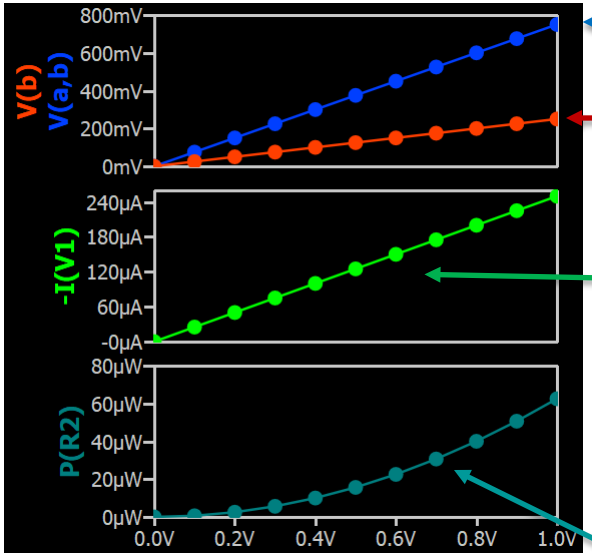
[0] DC Sweep (**.dc**) can be utilized to analyze the steady state voltage under sweeps of current sources, voltage sources or temperature



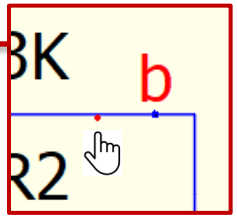

[1] **.dc** to sweep V1 from 0V to 1V with 0.1 per step
** In DC analysis, capacitor is OPEN circuit and inductor is SHORT circuit

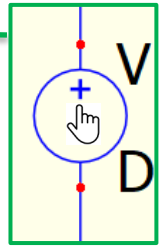
Differential voltage probe : V(a,b)
Hold [ALT] and click on node a, pull to node b



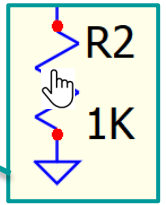

Voltage probe : V(b)
click node b




Current probe : I(V1)
Hover on device

Power probe : P(R2)
Press Ctrl and Hover on device

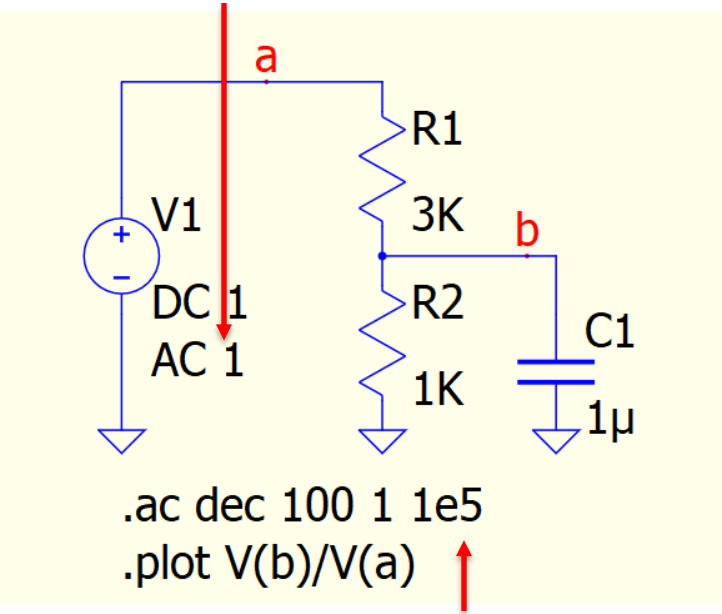


# AC Analysis (.ac)

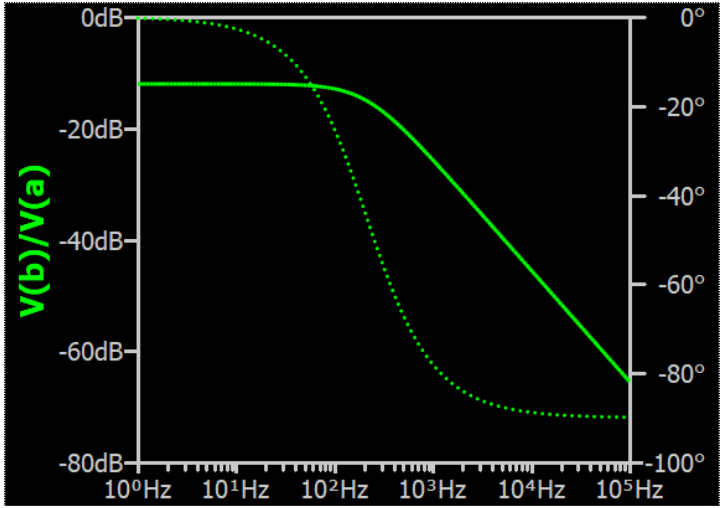
## Qspice : AC Analysis - Demo 01.qsch

[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 compared to AC source voltage.

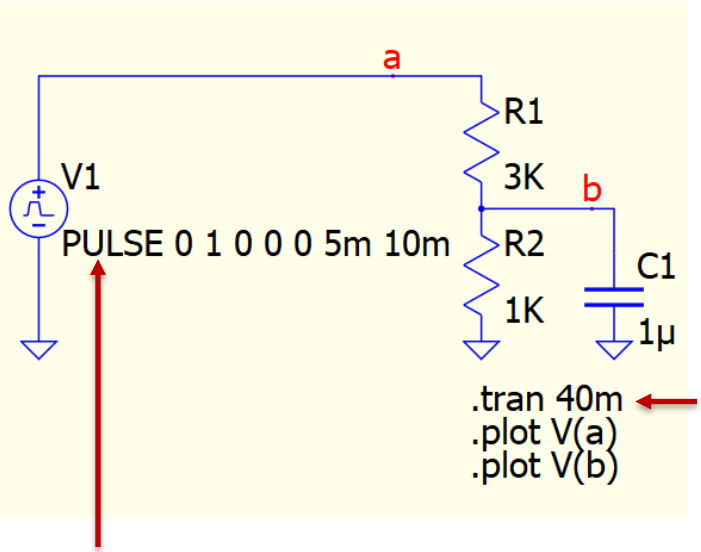



[2] this is to sweep AC source frequency from 1Hz to 1e5Hz (100kHz) with 100 points per decade

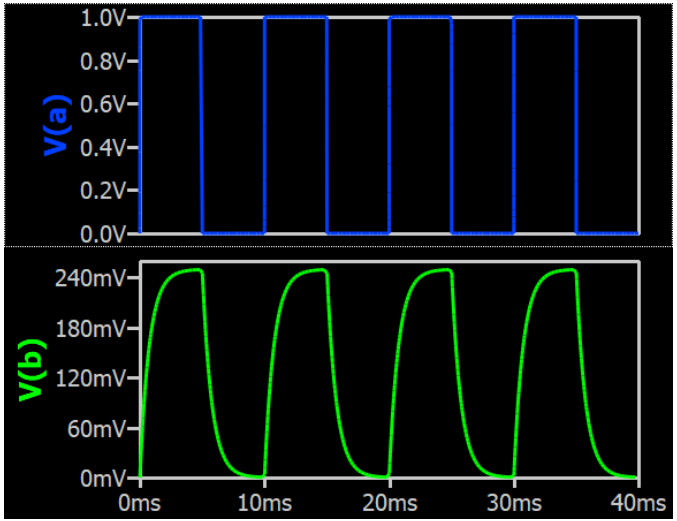
# Non-Linear Transient Analysis (.tran)

## 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

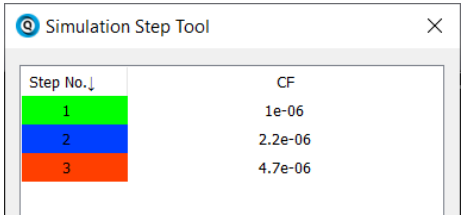



[2] Specify a time domain source (commonly are DC, PULSE, SINE)
This is a pulse source with format : PULSE Voff Von Tdelay Trise Tfall Ton Tperiod Ncycles

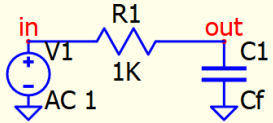
# Step User-Defined Parameter (.step)

## Qspice : Step - filter (.ac).qsch | Step - filter (.tran).qsch

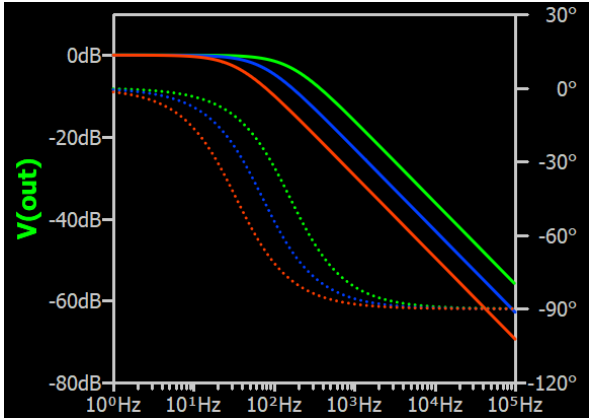
- .step
  - Step User-Defined Parameter (.step) is used to run a simulation multiple times changing one or more parameters
  - In waveform windows, press F6 to show Simulation Step Tool and can review color and parameter relationship
    - Or right click > Simulation Step Tool



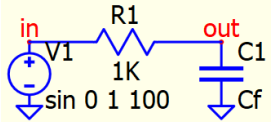

### Example of .step in .ac analysis



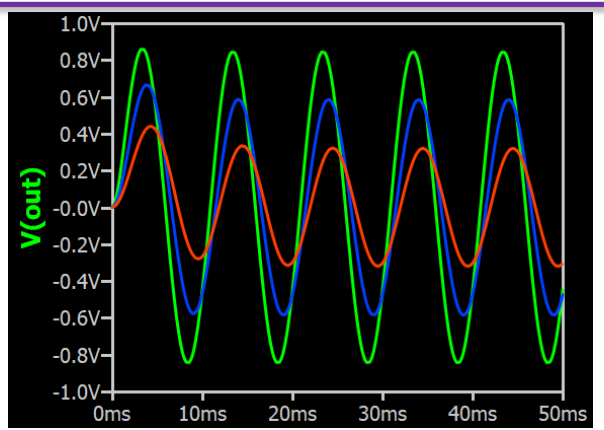

```
.param Cf=1µ ; only need this if .step is comment
.step param Cf list 1µ 2.2µ 4.7µ

.ac dec 100 1 100K
.plot V(out)
```




### Example of .step in .tran analysis



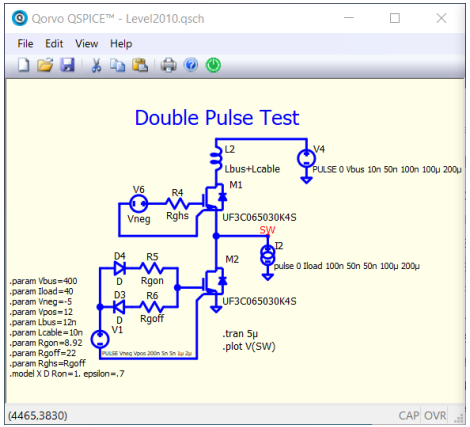

```
.param Cf=1µ ; only need this if .step is comment
.step param Cf list 1µ 2.2µ 4.7µ

.tran 5/100
.plot V(out)
```

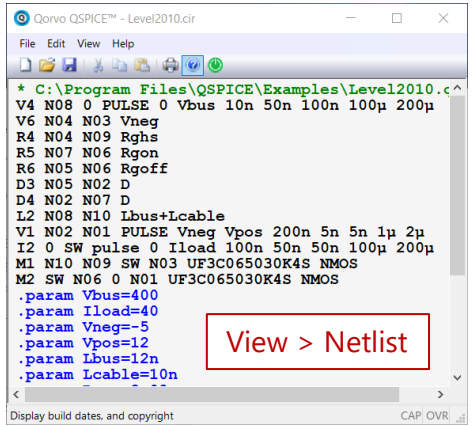


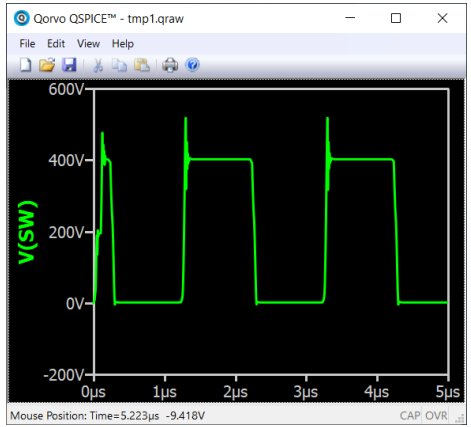
# Part 01
# Supplementary

# SPICE Simulation Workflow

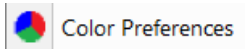
- SPICE Simulation Workflow
  - It is important to understand the concept of SPICE simulation, here is workflow of Qspice
  - QSPICE schematic drawing window is from QUX.exe, after you finish schematic drawing and hit “Run Simulation”, this is what happen
    - QUX.exe convert schematic (.qsch) into 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 be aware that there is a conversion of .cir as this process runs silently in the 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

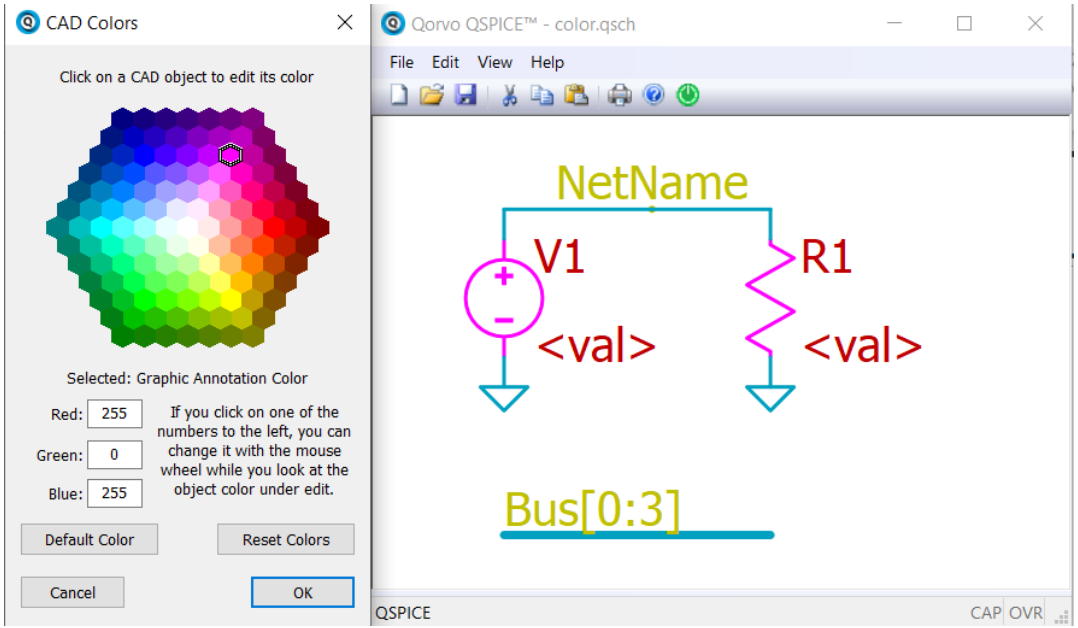


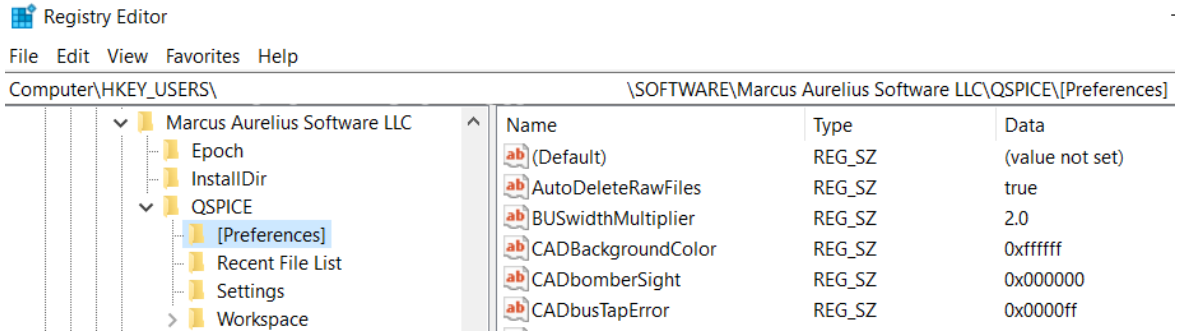
# Schematic Window – Color Scheme

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



# Qspice Preferences

- Qspice preferences are stored in regedit



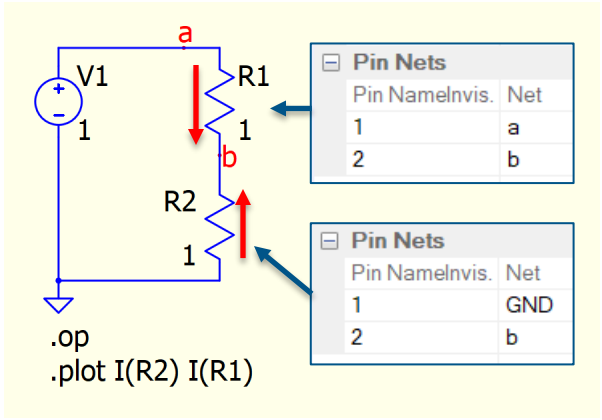

- Procedure to find regedit location
  - Run Registry Editor
  - Select HKEY_USERS
  - Edit > Find (Ctrl-F)
    - Find what: qspice
    - Look at: Keys only
  - Registry location:
    - Computer\HKEY_USERS\%%%%%\SOFTWARE\Marcus Aurelius Software LLC\QSPICE

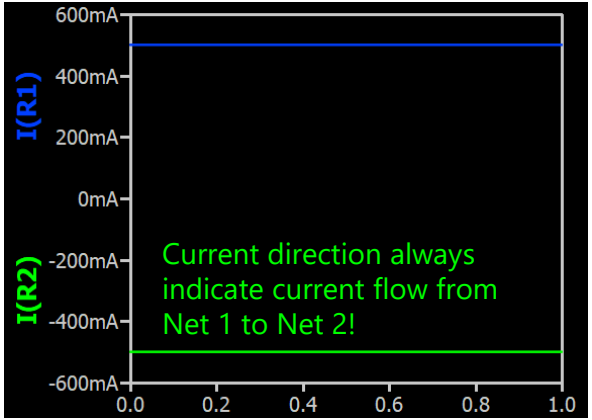
# Part 02
# Fundamentals of SPICE

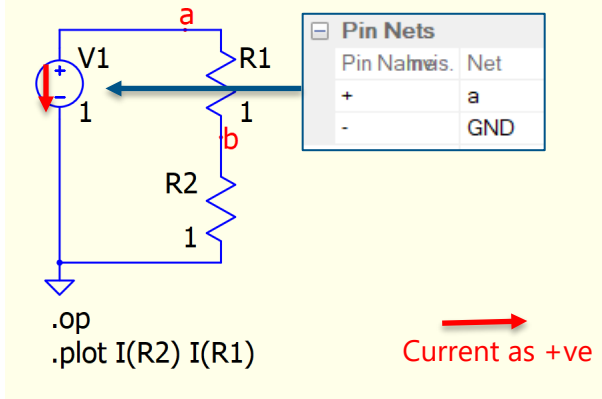
# Current Representation in Spice

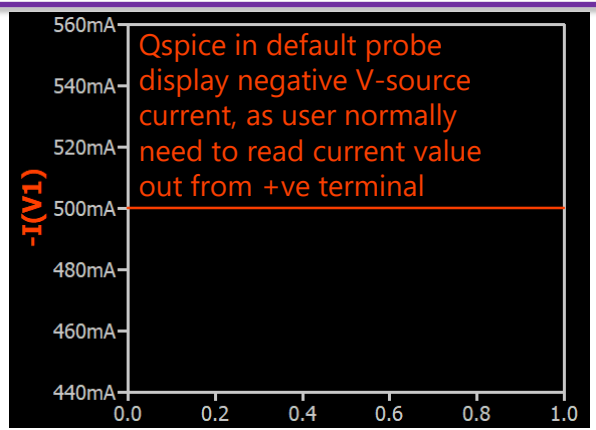
## Qspice : Current Representation - Explain.qsch

- 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)



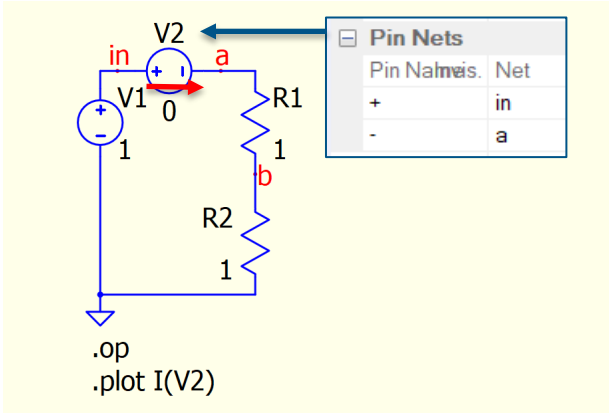



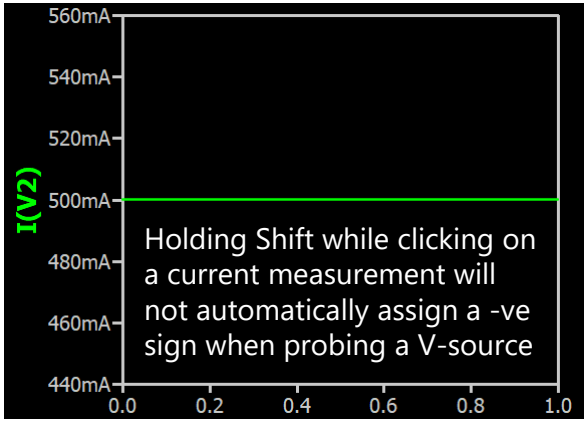



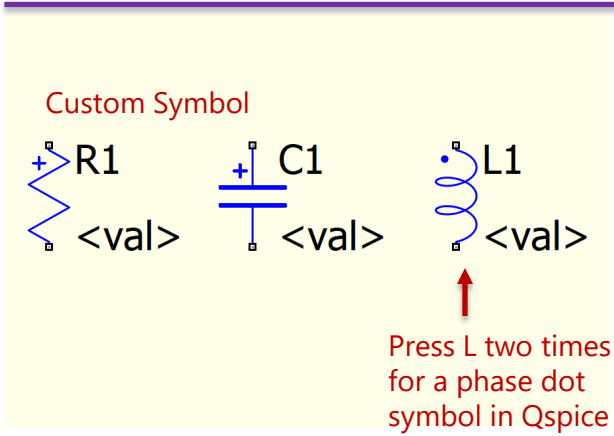
# Current Representation in Spice

## 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/](https://github.com/KSKelvin-Github/Qspice/)



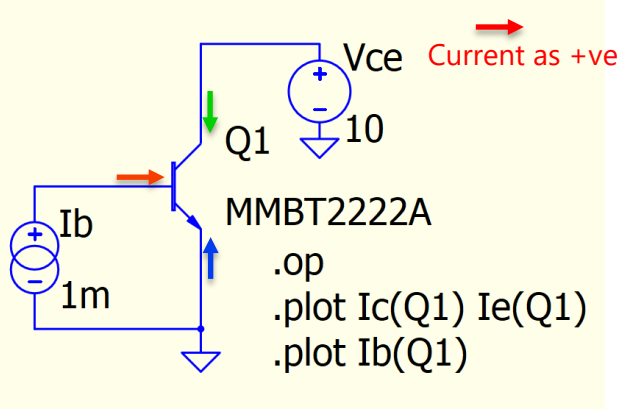



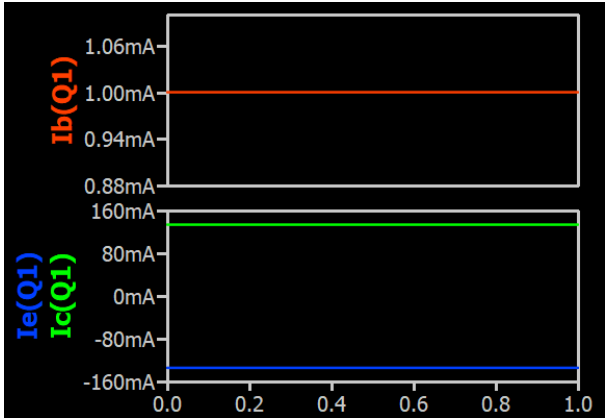



# Current Representation in Spice

## 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, Ib(Q1) and Ic(Q1) both return positive values as the simulated current flows into Q1. However, Ie(Q1) is negative as the simulated current flows out from Q1, in a reverse direction compares to its definition



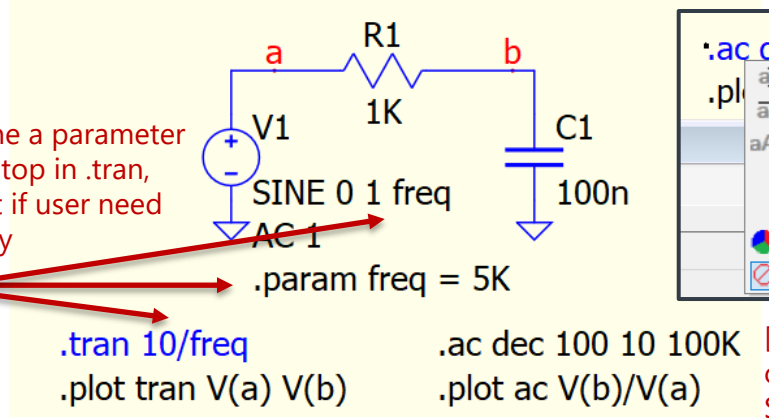



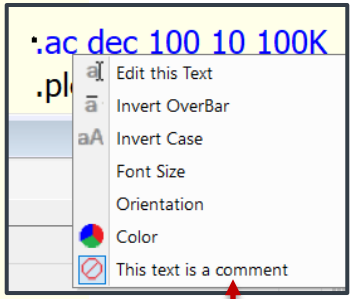
# Part 03
# Useful Techniques

# Parameter and Comment for Analysis Directive

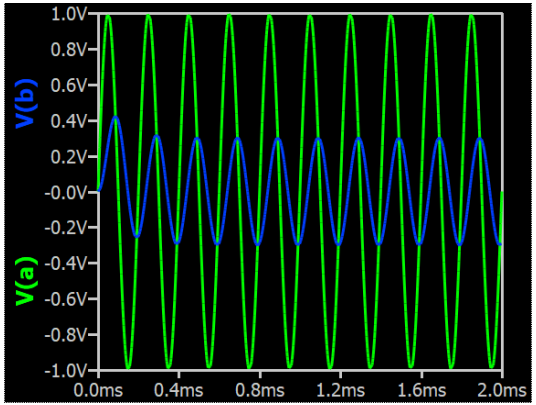
## Qspice : Comment and Params.qsch

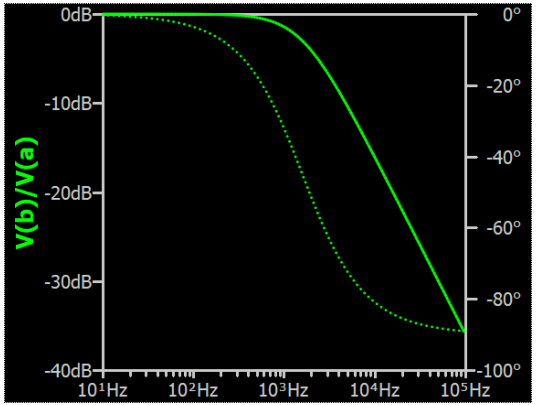
[2] For example in transient, by define a parameter can control source frequency and Tstop in .tran, which help simulation to auto adjust if user need to study circuit at different frequency




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

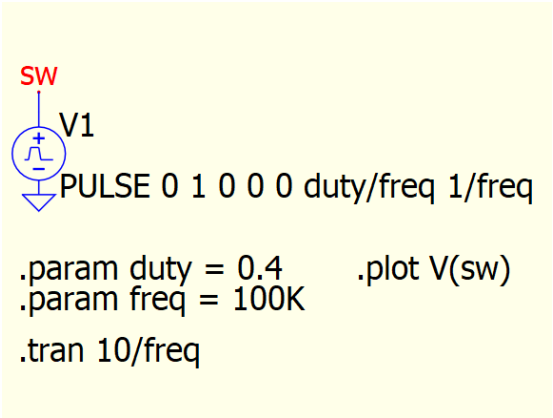


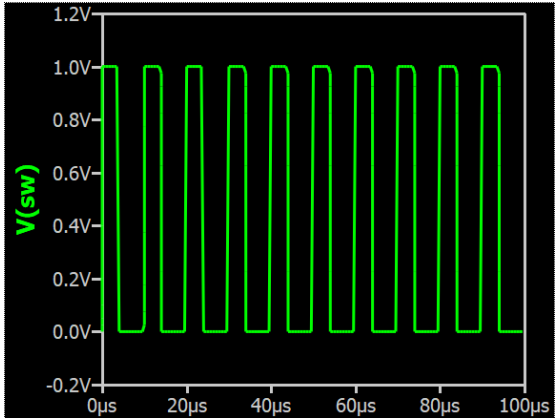



# Parameter for Pulse Source / Transient Convergence

## Qspice : Params.qsch

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



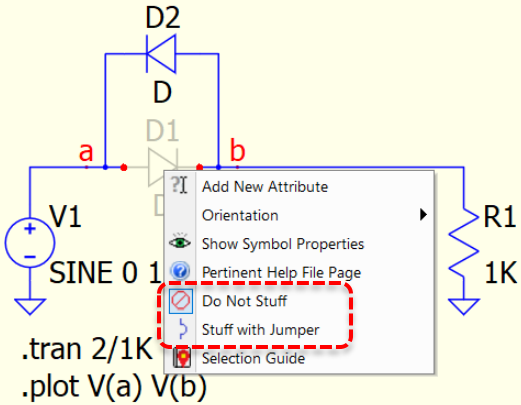



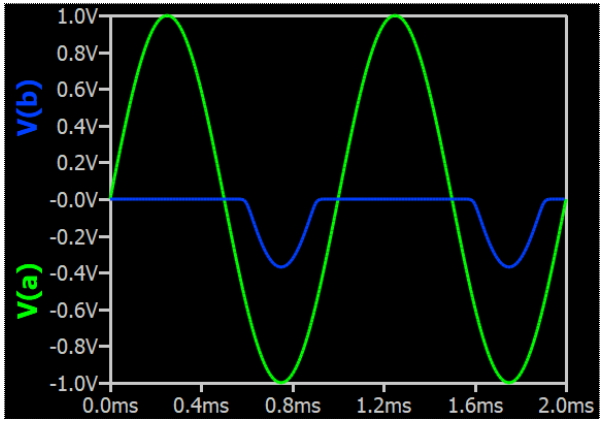

- In transient analysis, if simulation results is not convergence, try following options
  - Add **.option MAXSTEP=x**, where x is maximum step size for transient analysis
    - Sometimes it requires to limit step size especially circuit consist of pulse and logic
  - Goto Edit > Preferences, disable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

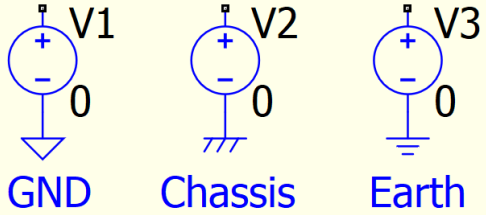
## Qspice : Open Short Devices.qsch | GND CHASSIS EARTH.qsch

- 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



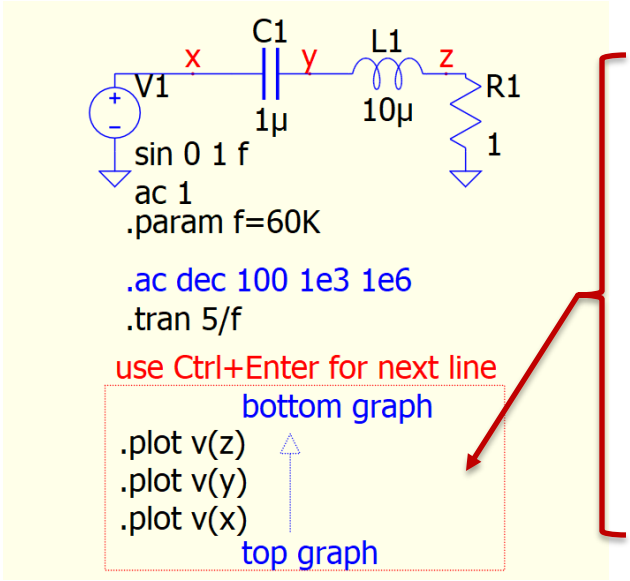

- View > Netlist

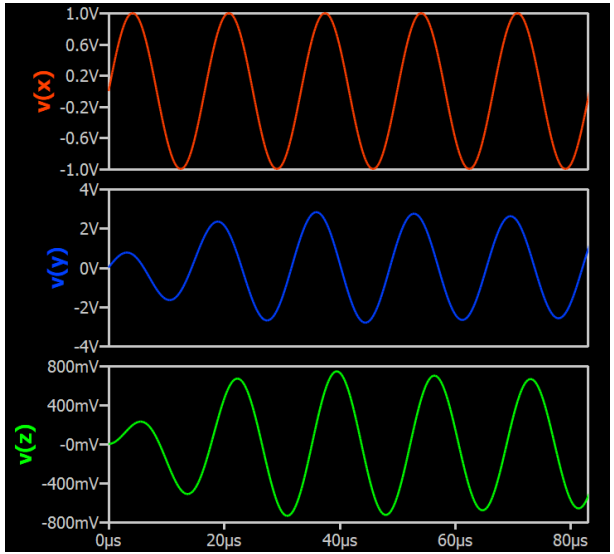
```
V1 ¥0 0 0
V2 ¥1 0 0
V3 ¥2 0 0
.end
```

# Deterministic .plot method to define plots sequence in waveform viewer

## Qspice : plot Sequence.qsch

- Use .plot <expression>
  - .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 follows a particular sequence, 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 to be effective
    - Close waveform viewer before Run simulation
    - No plot configuration file is present (i.e. [qschname].pfg is deleted in schematic directory)



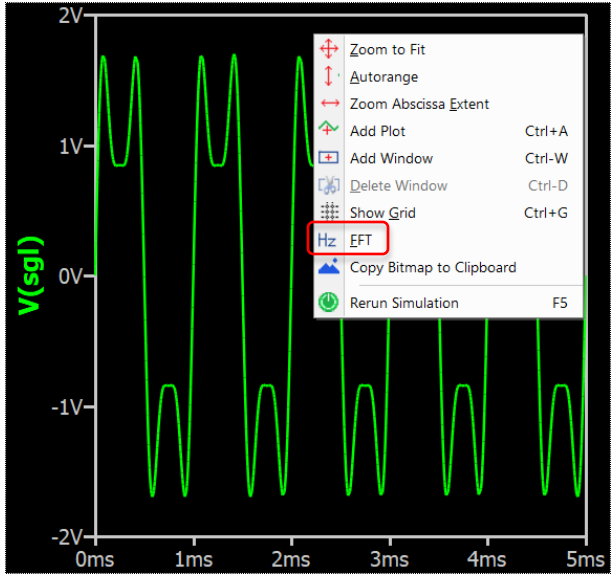
# FFT in Waveform Viewer

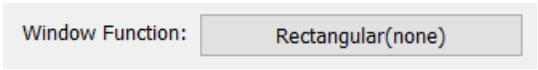
## Qspice : FFT waveform viewer.qsch

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

sgl
V1
sin 0 rms2pk(1) 1K
V2
sin 0 rms2pk(0.6) 3K
V3
sin 0 rms2pk(0.2) 5K

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

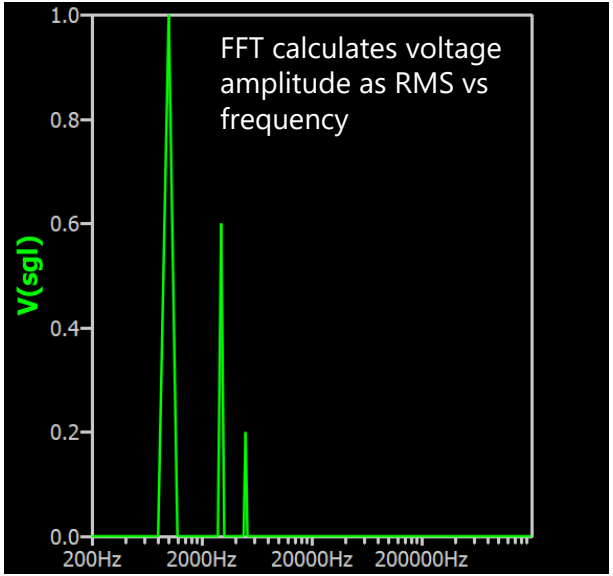
Use .func for rms to peak conversion

[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)



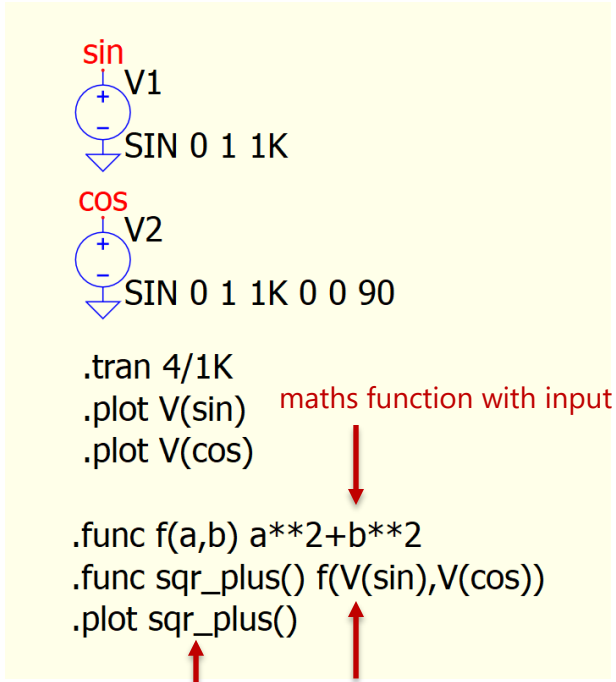
# Part 04
# User-Defined Functions and Parameters

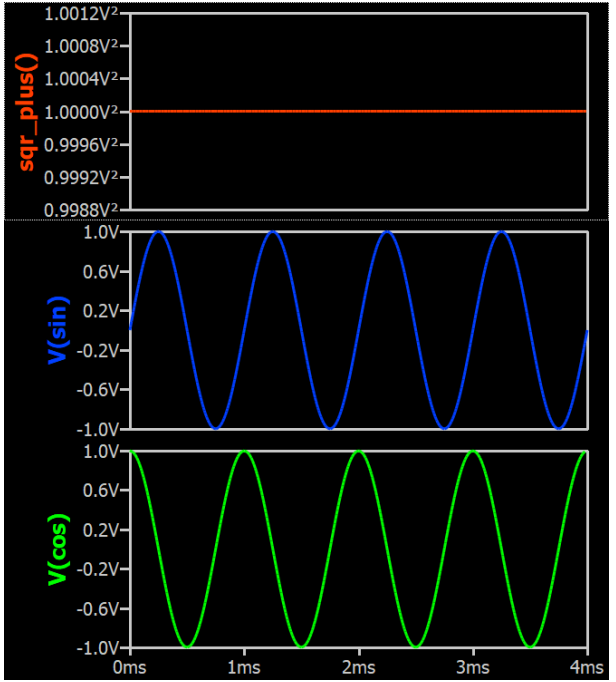
# User-Defined Function .func

## 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 B-source with function to overcrowding the schematic
  - Function is hard-wired to exactly the thing you want to plot and doesn't need an argument

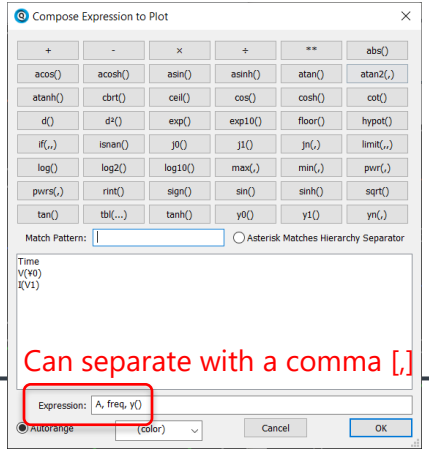


# User-Defined Parameters (.param) and Functions (.func)

## 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

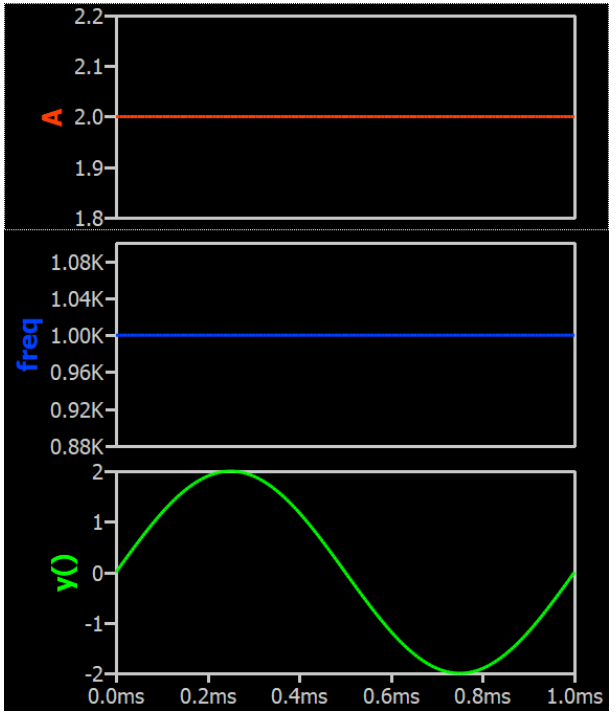
```
.tran 1/freq
.plot A
.plot freq
.plot y()
```

.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)}
```



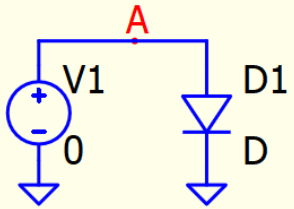
# Part 05
# Simulation Techniques

# Simulate Ideal Diode and Switch : Misconception

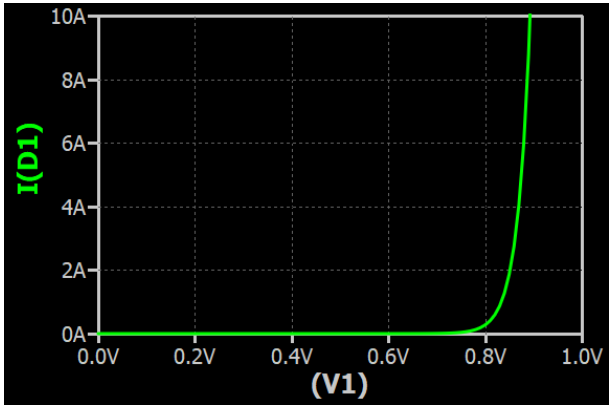
## Qspice : Misconception - D as Default.qsch | Misconception - M as Default.qsch

- Misconception for D-Device and M-device
  - It is common that new user misunderstand diode and mosfet with default parameters (i.e no .model statement) can be used as Ideal switching components
  - A diode with default parameters with knee voltage ~ 0.85V
  - A nmos with default parameters with Rds,on ~ 5kΩ when fully turned on
  - Ideal devices doesn't exist in SPICE simulation (i.e. Ron=0 and Roff=Inf), but next two slides explain technique to model near ideal diode and switch

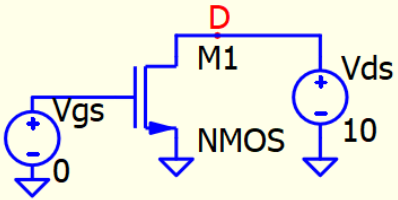
Diode (Default model parameters)



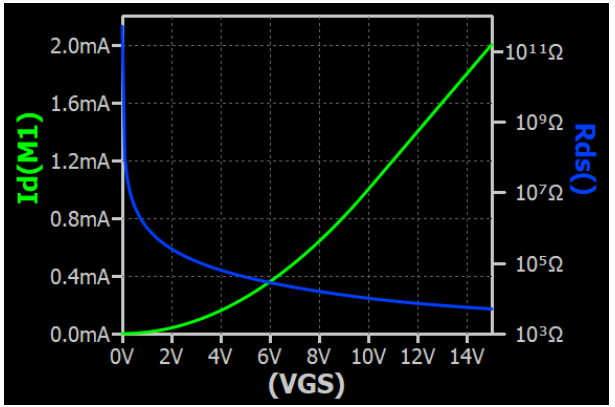

```
.dc V1 0 1 .01
.plot I(D1)
```




NMOS (Default model parameters)



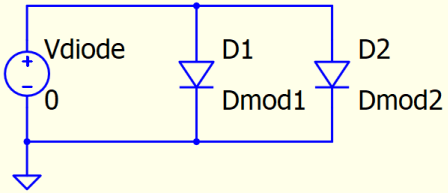

```
.dc Vgs 0 15 .1
.func Rds() V(D)/Id(M1)
.plot Id(M1) Rds()
```



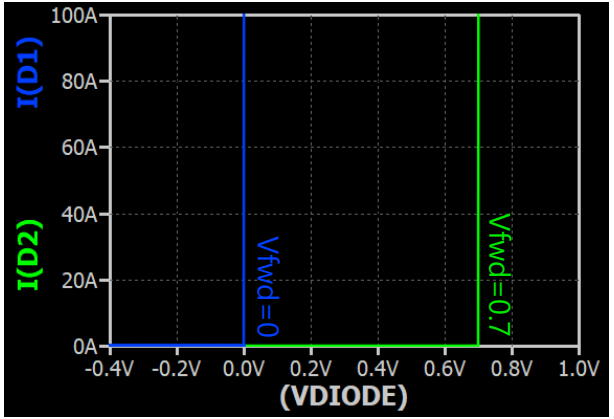
# Simulate Ideal Diode : Diode (D)

## Qspice : D - Ideal Diode.qsch

- D Diode
  - Ideal diodes are typically simulated using the behavioral diode model method
  - Avoid setting Ron=0 as it will force the use of the conventional SPICE semiconductor diode equation instead of the behavioral diode model
  - Prevent setting Ron to an extremely small value, as this may lead to convergence problems




```
.model Dmod1 D Ron=1µ Roff=10Meg Vfwd=0
.model Dmod2 D Ron=1µ Roff=10Meg Vfwd=0.7
.dc Vdiode -.4 1 0.01
.plot I(D2) I(D1)
```

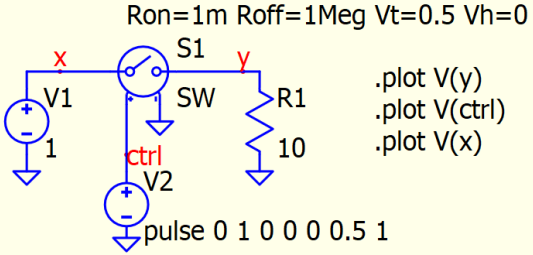


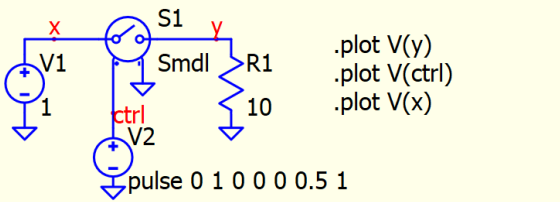
# Simulate Ideal Switch : Voltage Controlled Switch (S)

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

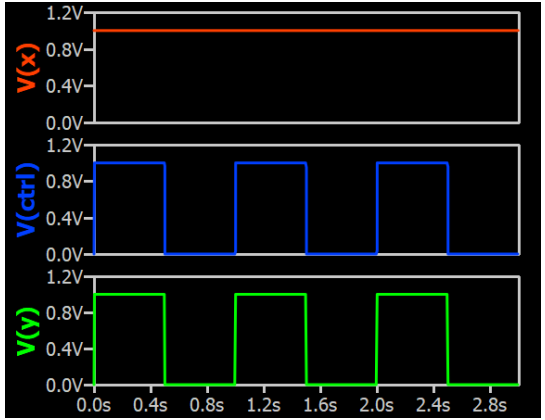
- S Switch
  - Switch is normally used as Ideal Switching device
  - S is voltage controlled switch
    - Voltage between control nodes can switch impedance between switch terminals
  - Switch can be configured with instance parameters or .model
  - Simulation results of these two examples are identical

Add Instance parameter
Right click on S1, Add New Attribute




Use .model model (shortcut T to place text)
Smdl is model name
SW is Switch
Vt is threshold, Vh is hysteresis

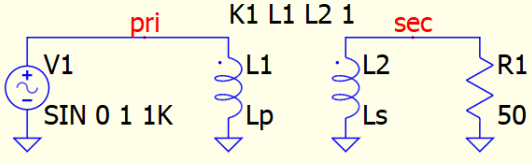


# Simulate Transformer : Transformer with Coupled Inductor (L)

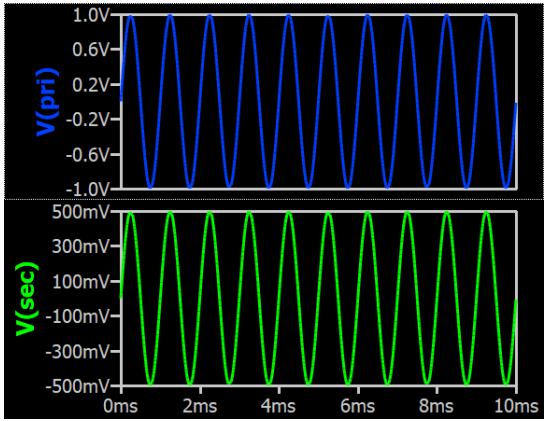
## Qspice : L - Transformer with 2 Windings.qsch | L - Transformer with 3 Windings.qsch

- Transformer with L
  - Two or more coupled inductors are required
    - Press L two times to get an inductor symbol with a dot notation (Not necessary but recommend to indicate direction)
  - $\frac{L_p}{N_p^2} = \frac{L_s}{N_s^2}$ and $n = \frac{N_p}{N_s}$
  - $L_p = n^2 L_s$ or $L_s = \frac{1}{n^2} L_p$
    - In general, we can measure primary inductance (Lp) and turn ratio (n) is given in assembly
  - K is Mutual Inductance defines mutual coupling coefficient of coupled inductors
    - Ideal coupling is K=1

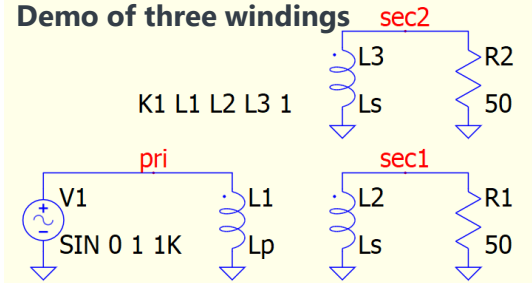
**Demo of two windings**



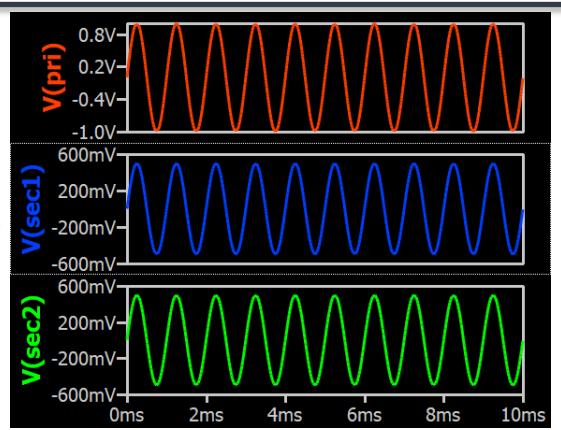

```
.tran 10/1K
.plot V(pri)
.plot V(sec)
.param n=2
.param Lp=100µ
.param Ls=Lp/n**2
```




**Demo of three windings**



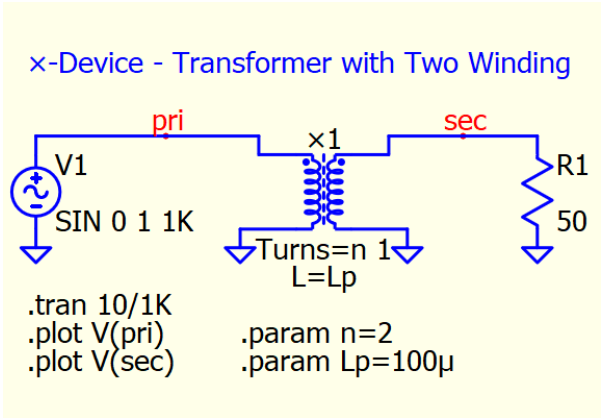

```
.tran 10/1K
.plot V(pri)
.plot V(sec1)
.plot V(sec2)
.param n=2
.param Lp=100µ
.param Ls=Lp/n**2
```

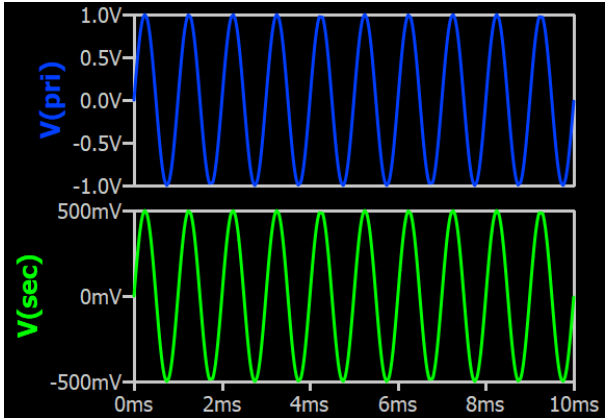


# Simulate Transformer : Transformer with ×-Device

## Qspice : ×-Device - Transformer with Two Winding.qsch

- ×-Device
  - ×-device is transformer in Qspice, it defaults as an ideal transformer if its instance parameter L is not set



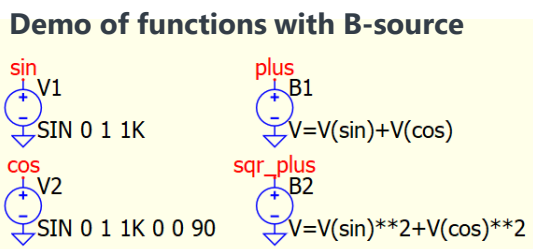



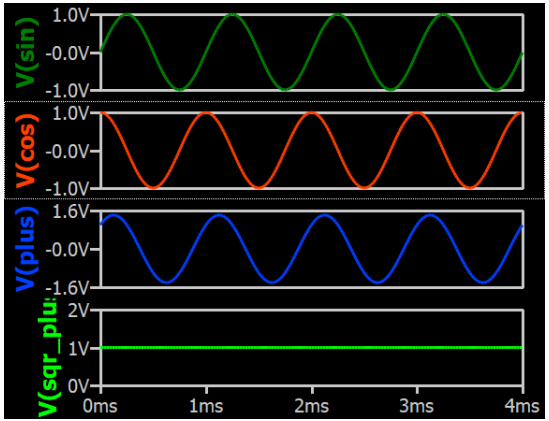
# Simulate Functions : Arbitrary Behavioral Source [B]

## Qspice : B - Functions.qsch | E - Laplace.qsch

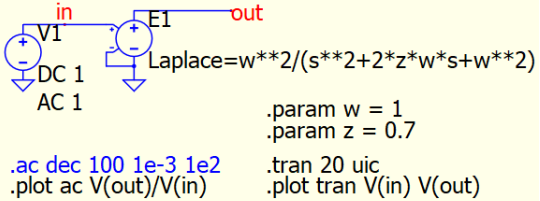
- Arbitrary Behavioral Source [B]
  - Mathematic functions and logical operators can be used
  - This is useful for mathematic calculation during simulation
  - It also support Laplace transfer function
- Remark
  - E, F, G, H source has similar application properties as B source
  - Recommend go to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

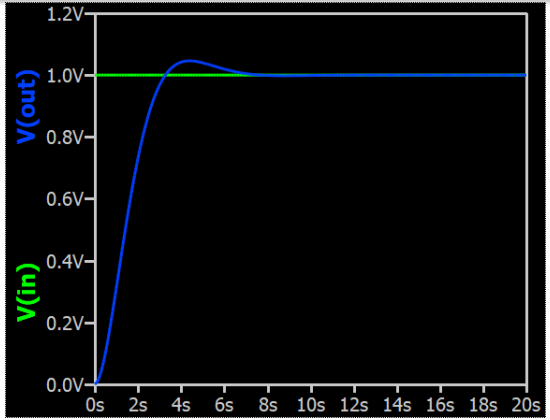
**Demo of functions with B-source**




**Demo of Laplace with E-source**



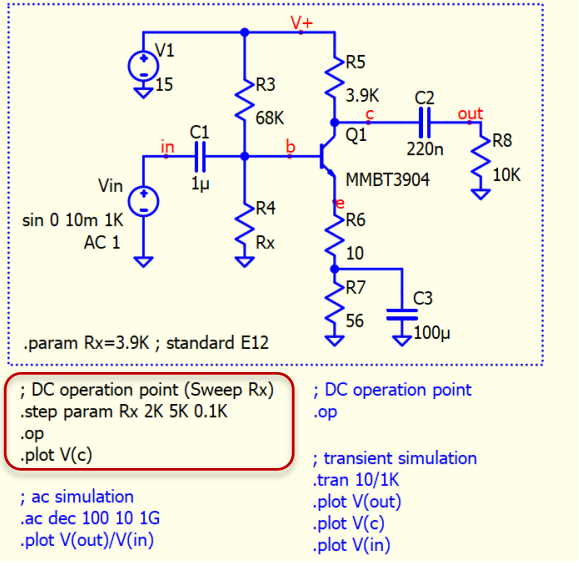



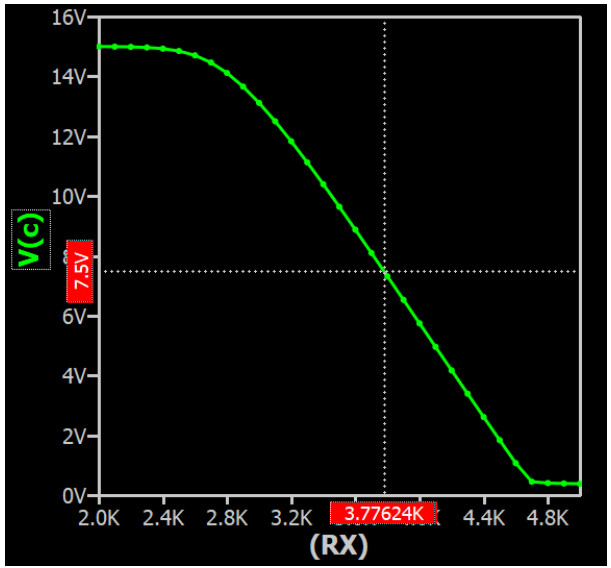
# Part 06
# BJT Amplifier Circuit Tutorial

# Tutorial – BJT Amplifier Circuit [Part 1 DC Sweep Analysis]

## Qspice : BJT-Circuit-Tutorial.qsch

- DC Sweep (.op + .step)
  - This is a tutorial on using Qspice to design and analyze the response of a BJT amplifier
  - The first step is to determine resistance of R4 (Rx), with the goal of setting the collector voltage at half of V+ to achieve the largest possible voltage swing
  - Using .op with .step Rx and plotting V(c) can help in determining the optimized DC operating point

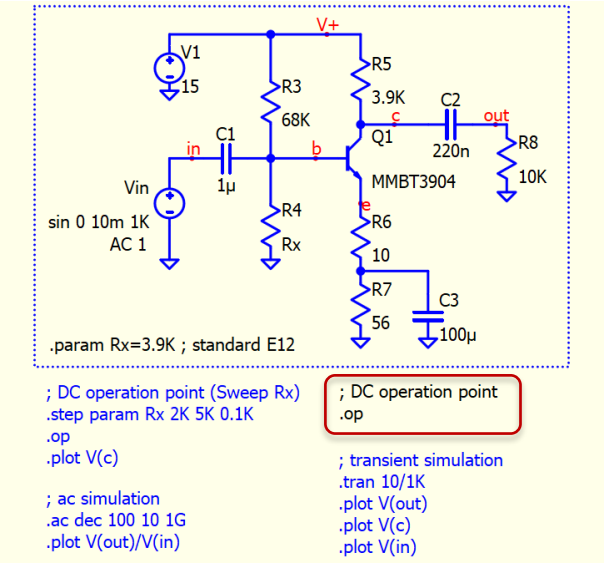


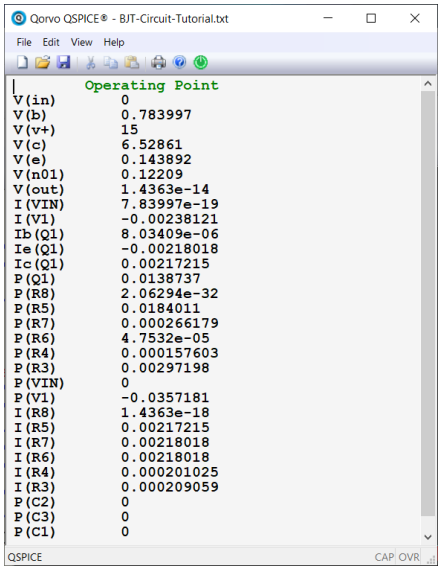



# Tutorial – BJT Amplifier Circuit [Part 2 DC Operating Point Analysis]

## Qspice : BJT-Circuit-Tutorial.qsch

- DC operating point (.op)
  - Rx is set using .param Rx=3.9k, and a .op directive is executed for the DC operating point
  - In the waveform window, by navigating to File > Export Data, you can retrieve the Operating Point results in text format

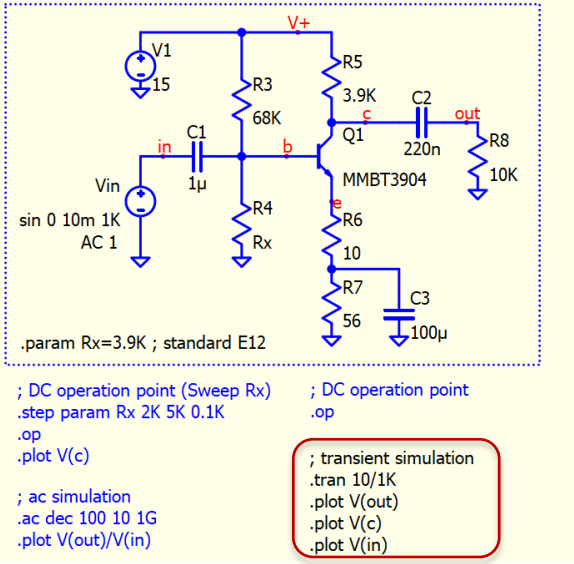


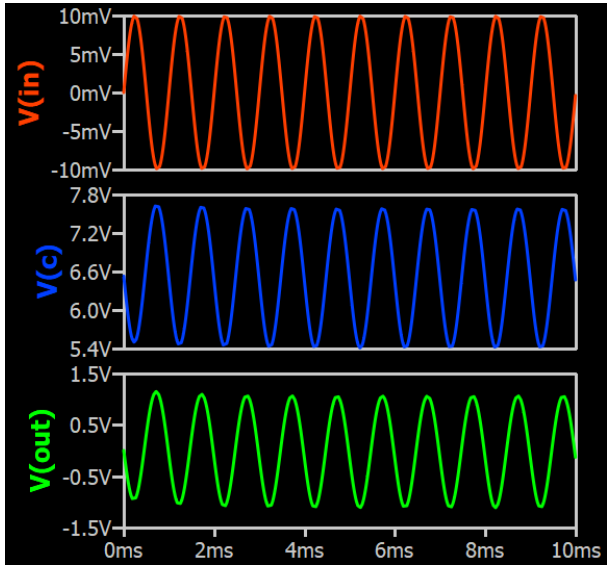



# Tutorial – BJT Amplifier Circuit [Part 3 Transient Analysis]

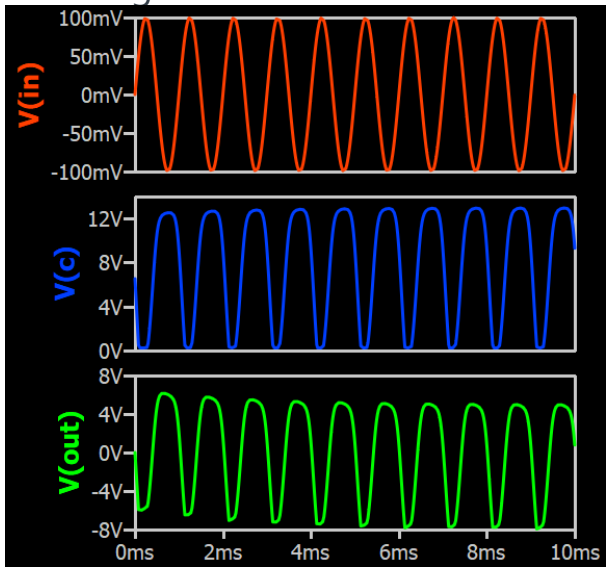
## Qspice : BJT-Circuit-Tutorial.qsch

- Transient Analysis (.tran)
  - Transient analysis is set up using .tran, where the input voltage source Vin has two attributes: sin 0 10m 1K and AC 1. The .tran command utilizes sin 0 10m 1k to generate a sine wave at V(in) for simulation



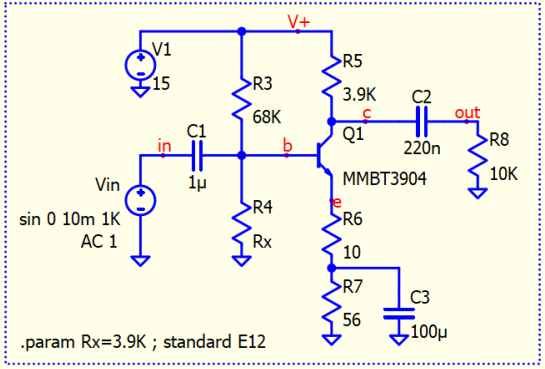

Changed Vin to **sin 0 100m 1K**



# Tutorial – BJT Amplifier Circuit [Part 2 AC Analysis]

## Qspice : BJT-Circuit-Tutorial.qsch

- AC Analysis (.ac)
  - Vin is an AC source with a magnitude of 1V and a phase of 0 degrees
  - The circuit is linearized in .ac, and it never saturates in regardless of the AC magnitude
  - .ac directive here is set to sweep frequencies from 10Hz to 1GHz, with 100 points per decade.




```
; DC operation point (Sweep Rx)
.step param Rx 2K 5K 0.1K
.op
.plot V(c)

; ac simulation
.ac dec 100 10 1G
.plot V(out)/V(in)
```

```
; DC operation point
.op

; transient simulation
.tran 10/1K
.plot V(out)
.plot V(c)
.plot V(in)
```