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
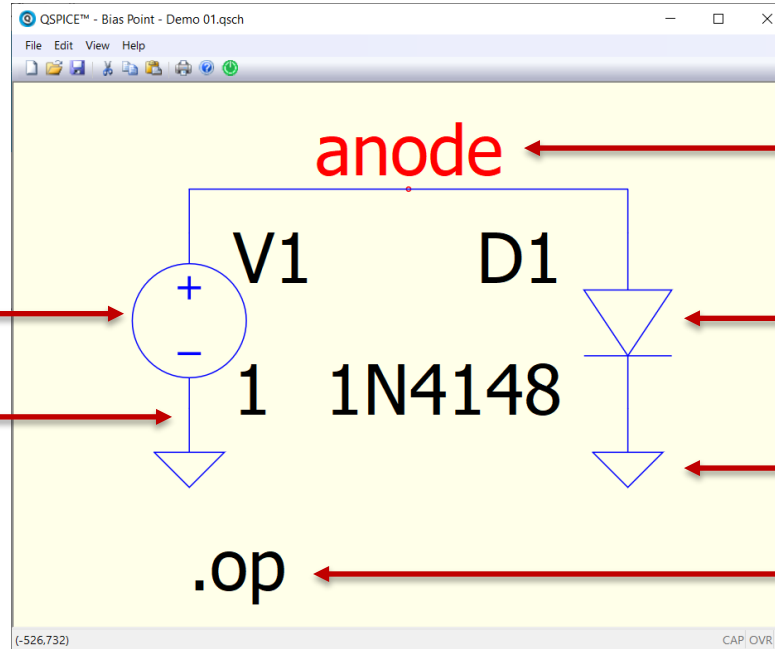   - Example of hint syntax underneath

```
.tran
.tran TSTOP
.tran (IGNORED) TSTOP TSTART MAXTIMESTEP
```
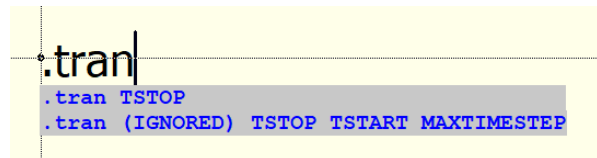
Part 01
Start Simulation

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 – step by step procedure

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

[1] Press [V] to place a voltage source, Type [1] in <val>
• Keep pressing [V] can cycle symbol

[4] Press [W] to place 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

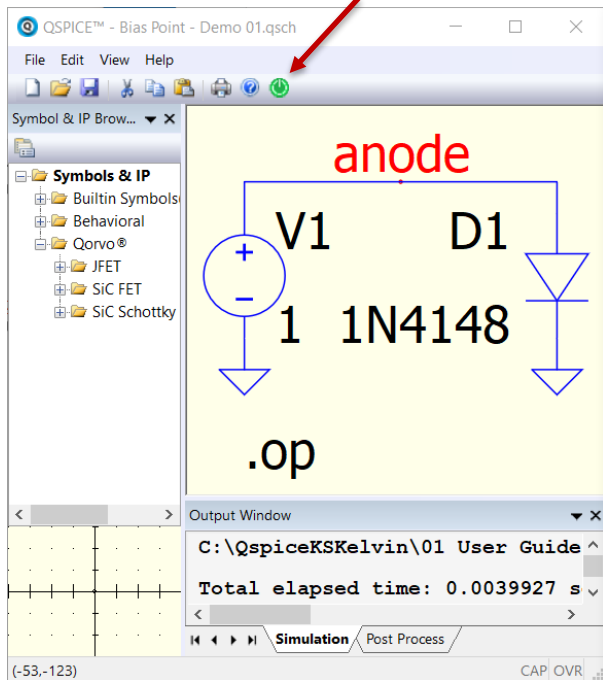[6] Press [.] to start typing for directive, input **.op** [hint syntax underneath during typing]

** another method is to Press [T] to start input text, and type **.op**
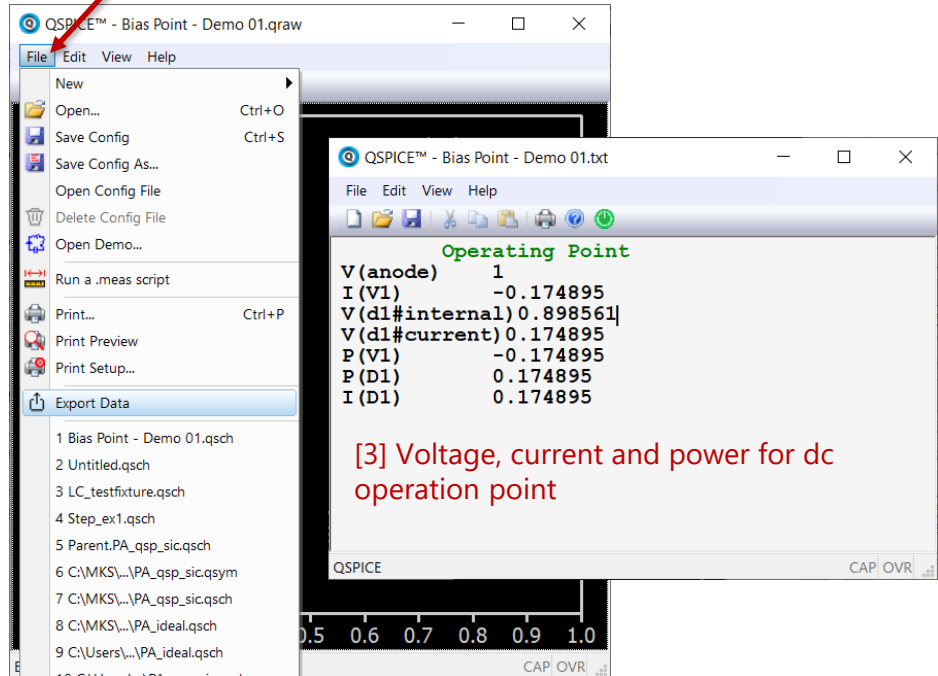
Bias Point Analysis (.op) : DC Operation Point
Qspice : Bias Point – Demo 01.qsch

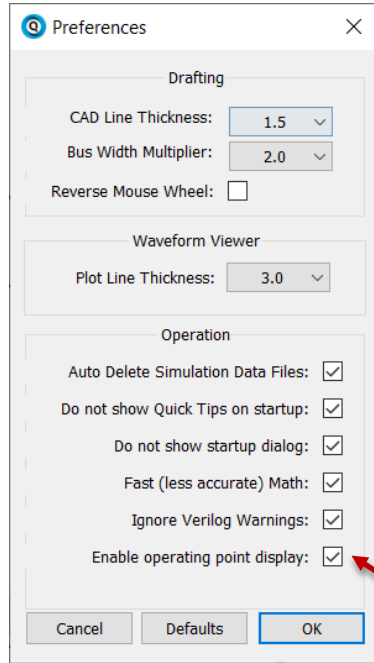[0] Bias Point Analysis (**.op**) computes DC operation point
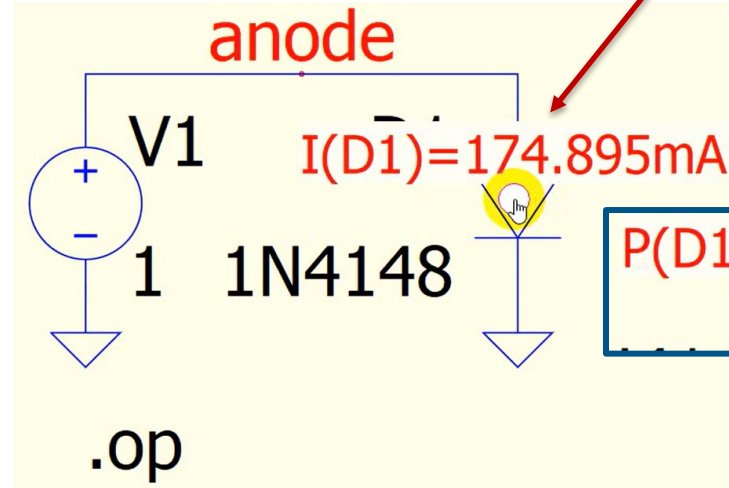
[1] Run to simulate the circuit

[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

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

anode

V1

I(D1)=174.895mA

[3] Hold [Ctrl] key to display power

P(D1)=174.895mW

1 1N4148

.op

[1] Edit > Preferences
Select Enable operating point display
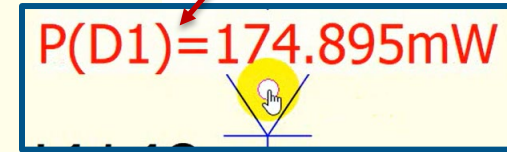
Hint : If you press [F6], the **.op** is recomputed without redoing the whole simulation
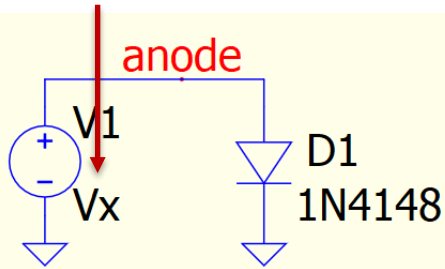
[0] This example is to use **.op** to plot V-I curve of diode D1 1N4148
- Idea is to sweep anode voltage from 0.5V to 1V and plot diode current

[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

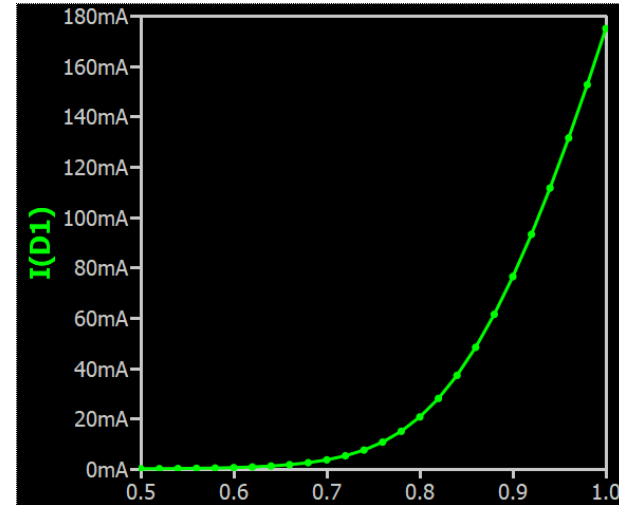| Pin Names.	Inv.	Net
A		anode
K		GND

anode

V1

+

–

Vx

D1
1N4148

.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 swep from 0.5 to 1 with 0.02 per step
.plot I(D1)   ←
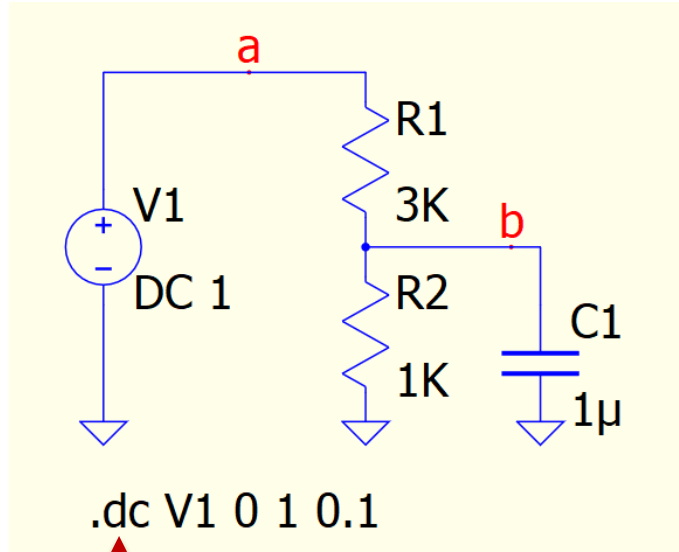[4] **.plot** tell the waveform viewer what to plot

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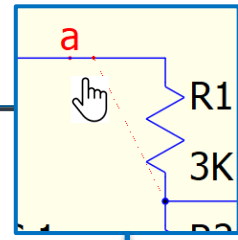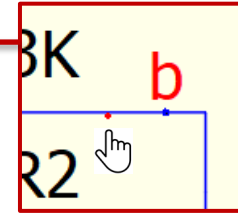
Differential voltage probe : V(a,b)
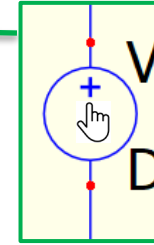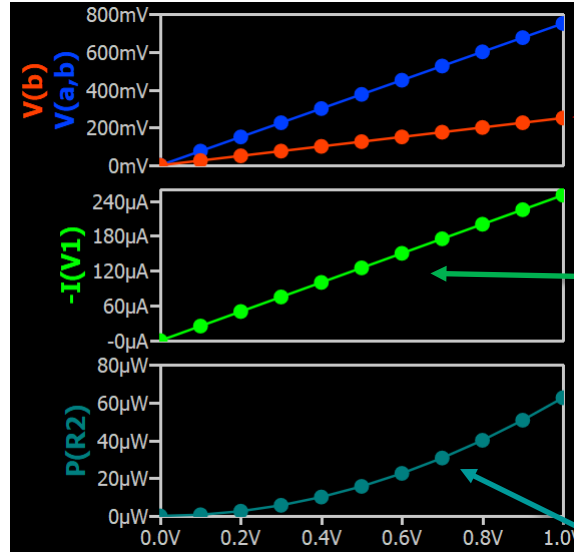Hold [ALT] and click on node a, pull to node b

Voltage probe : V(b)
click node b

Current probe : I(V1)
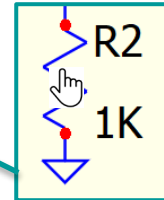Hover on device

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

.dc V1 0 1 0.1

[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

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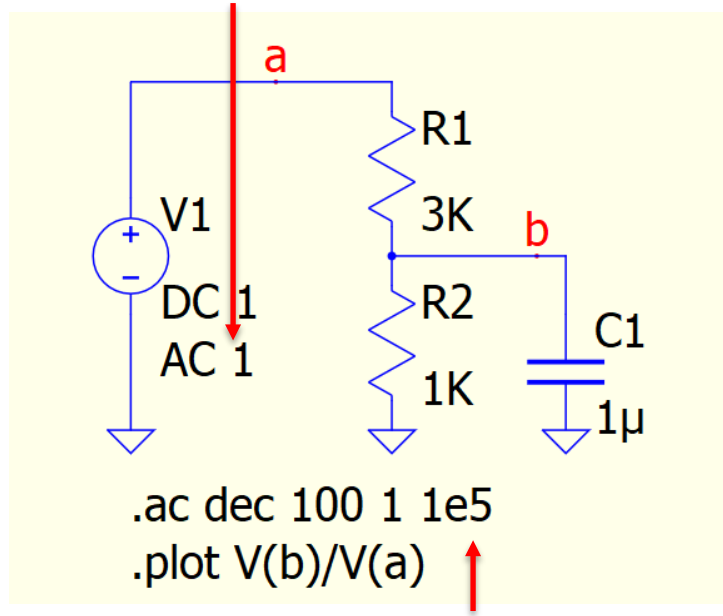
[3] AC analysis can plot bode (magnitude and phase relationship) of two probe positions. In this example, it is V(b) and V(a). If denominator is not specified, probe voltage is compare to AC source voltage.


.ac dec 100 1 1e5
.plot V(b)/V(a)
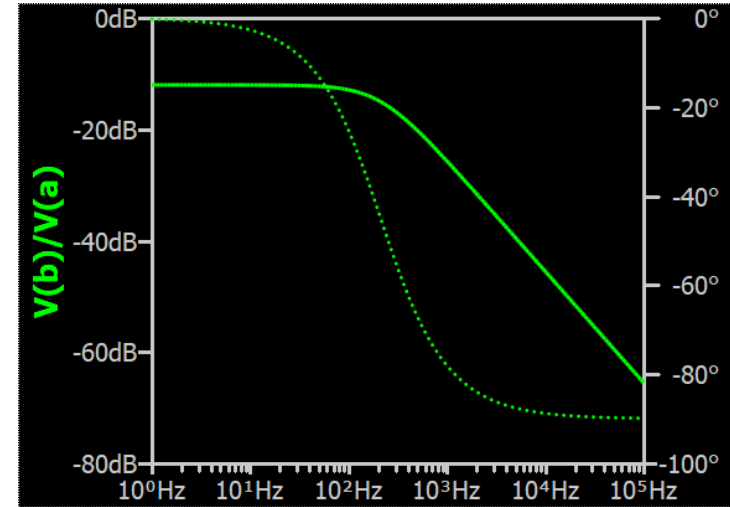
[2] this is to sweep AC source frequency from 1Hz to 1e5Hz (100kHz) with 100 points per decade
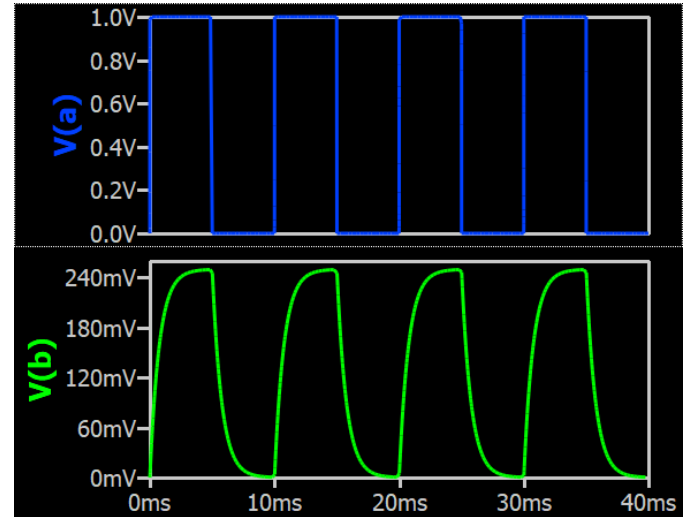
Non-Linear Transient Analysis (.tran)
Qspice : Transient Analysis - Demo 01.qsch

[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

a

V1

PULSE 0 1 0 0 0 5m 10m

R1

3K

b

R2

1K

C1

1μ
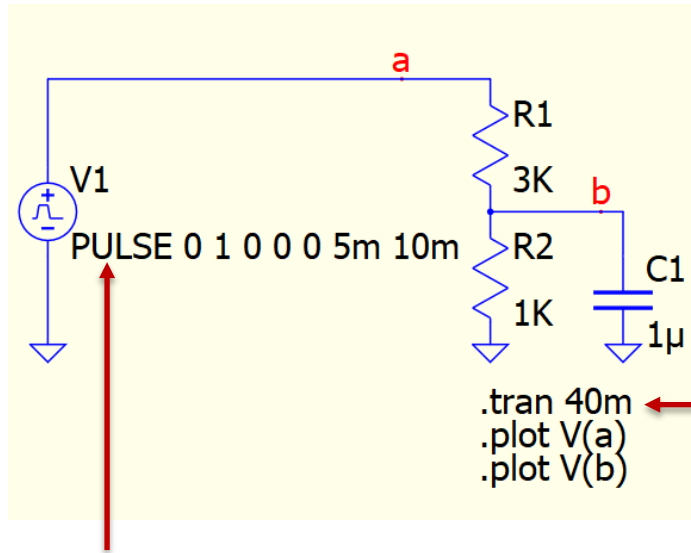
.tran 40m ← [1] transient analysis set
.plot V(a)   Tstop at 40ms
.plot V(b)

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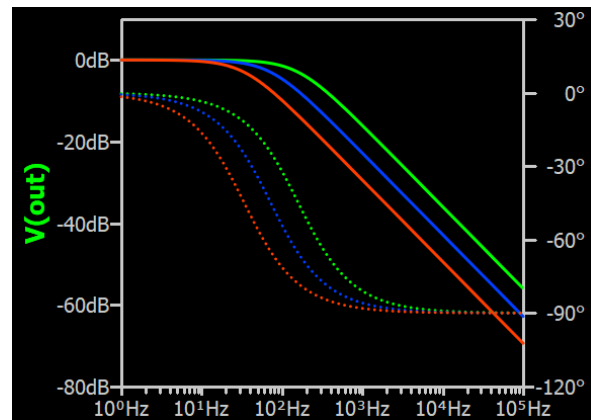
- .step
  - Step User-Defined Parameter (.step) is used to run a simulation multiple times changing one or more parameters
  - In waveform windows, press F6 to show Simulation Step Tool and can review color and parameter relationship
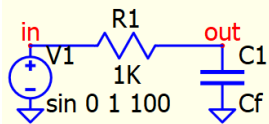   - Or right click > Simulation Step Tool
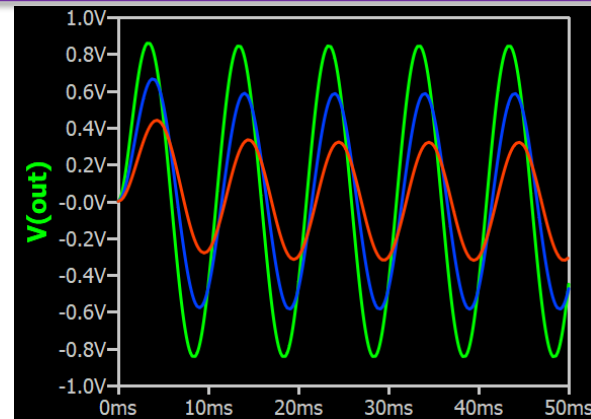

Example of .step in .ac analysis
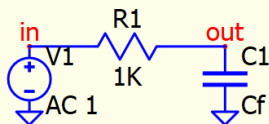
.param Cf=1μ ; only need this if .step is comment
.step param Cf list 1μ 2.2μ 4.7μ

.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
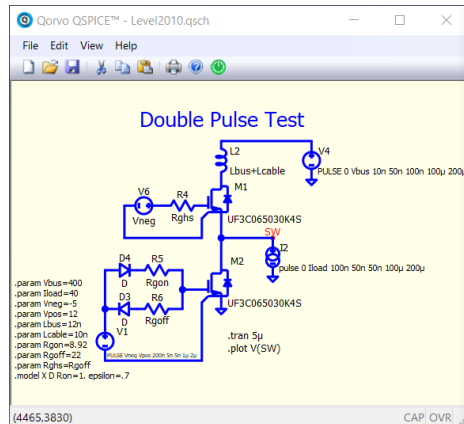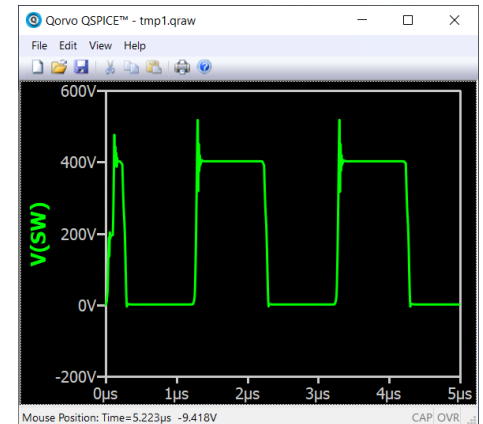
Schematic Window – Color Scheme

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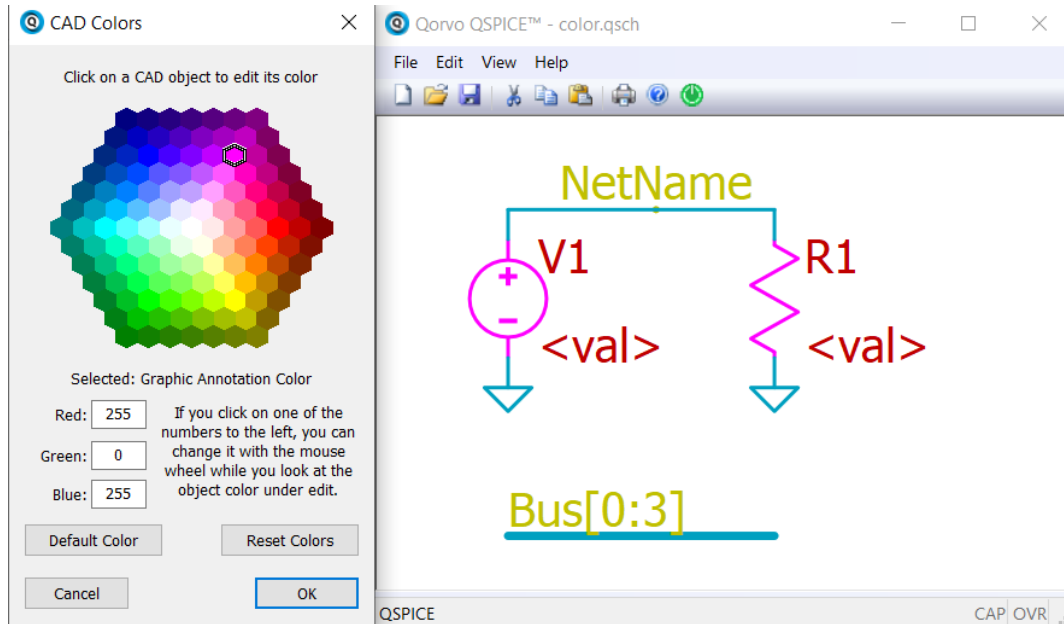
Qspice Preferences

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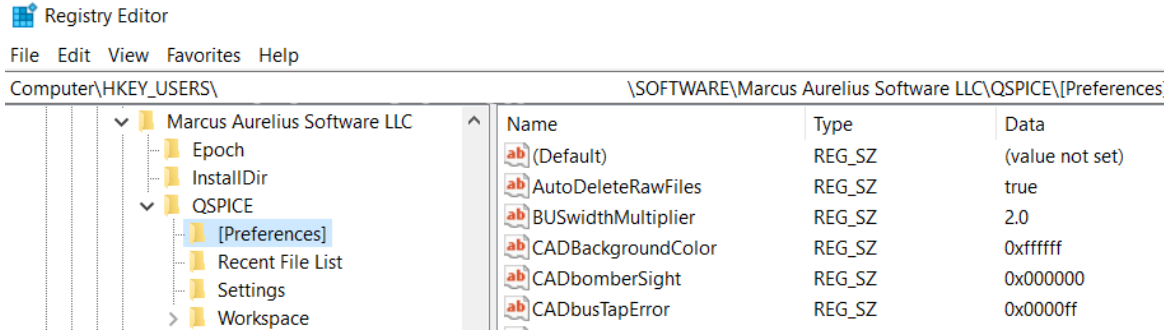
- Procedure to find regedit location
  - Run Registry Editor
  - Select HKEY_USERS
  - Edit > Find (Ctrl-F)
    - Find what: qspice
    - Look at: Keys only
  - Registry location:
    - Computer\HKEY_USERS\%%%%%\SOFTWARE\Marcus Aurelius Software LLC\QSPICE

Part 02
Fundamental in 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)
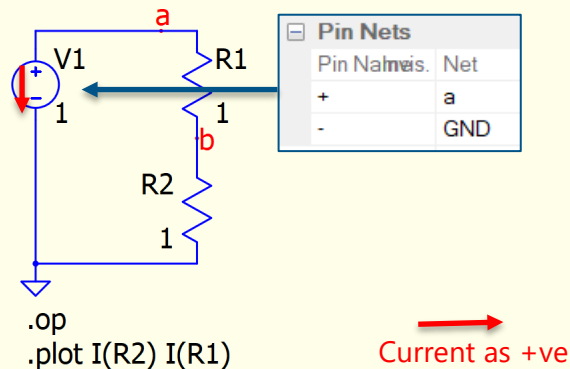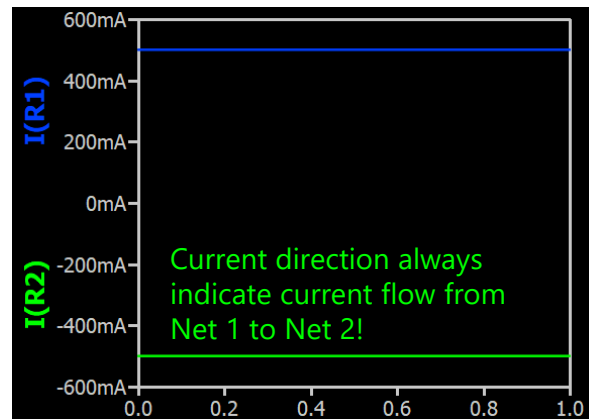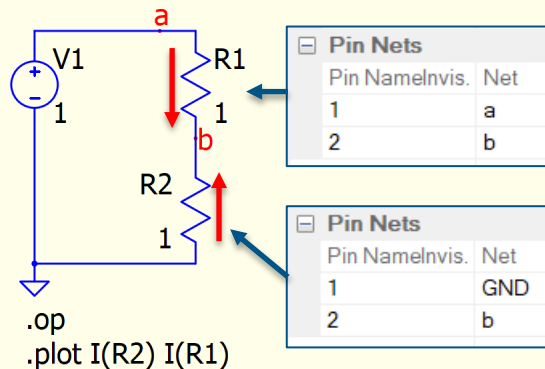
Current Representation in Spice

- 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/

Holding Shift while clicking on a current measurement will not automatically assign a -ve sign when probing a V-source
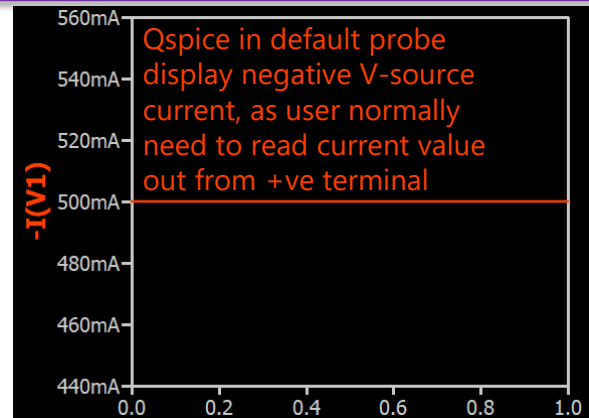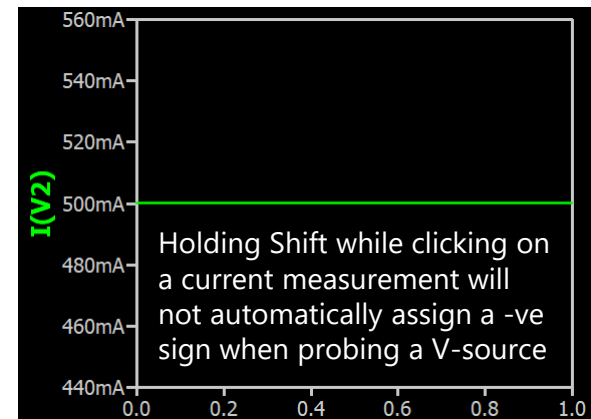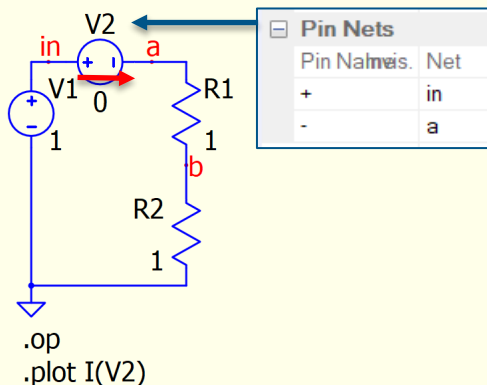
Custom Symbol
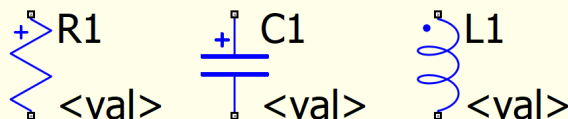
Press L two times for a phase dot symbol in Qspice

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, 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
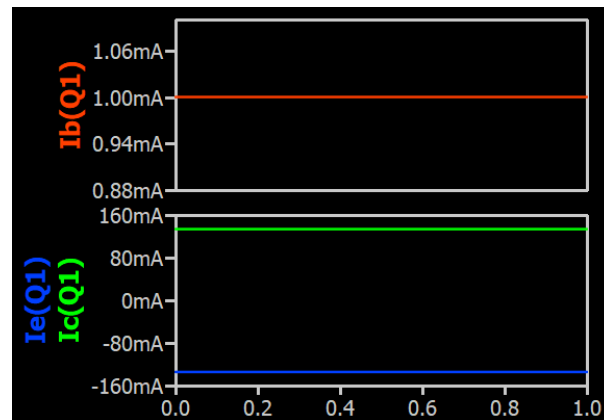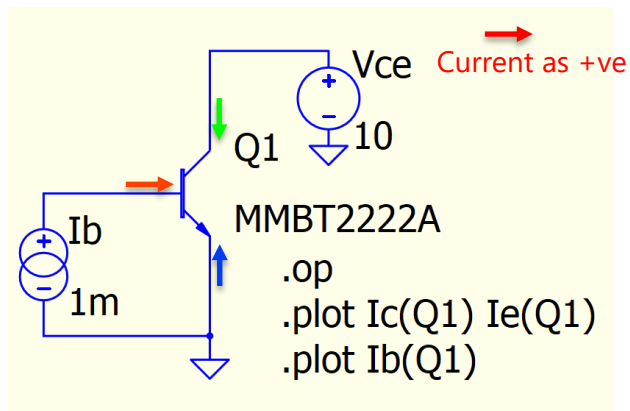
Part 03
Useful Technique

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

a   R1   b
1K

V1
SINE 0 1 freq
AC 1

C1
100n

.param freq = 5K
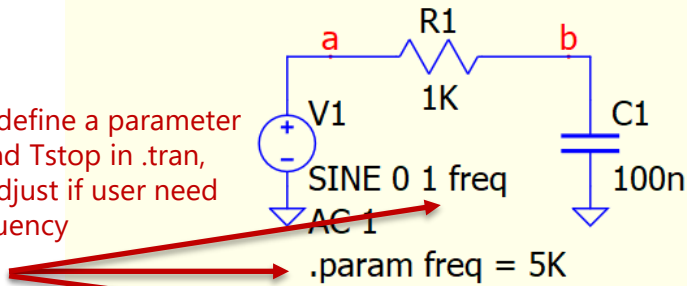
.tran 10/freq
.plot tran V(a) V(b)

.ac dec 100 10 100K
.plot ac V(b)/V(a)

:.ac dec 100 10 100K
.pl

- Edit this Text
- Invert OverBar
- Invert Case
- Font Size
- Orientation
- Color
- This text is a comment
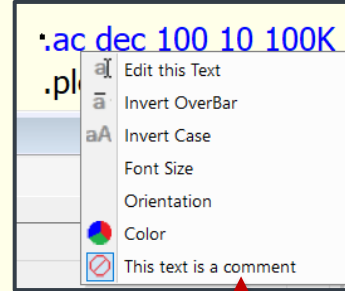
[1] For a single file with multiple analysis directive, comment can be use to control which one is active
Shortcut is [;]
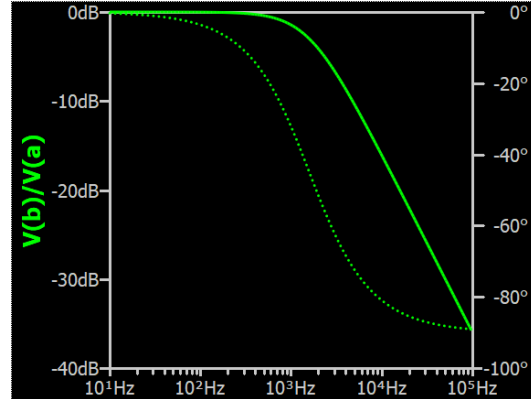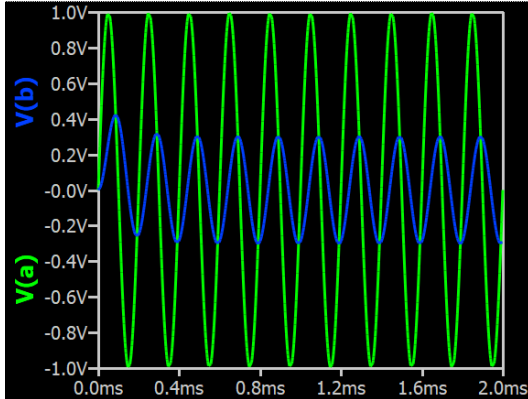
Parameter for Pulse Source / Transient Convergence
Qspice : Params.qsch

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

SW

V1

PULSE 0 1 0 0 0 duty/freq 1/freq

.param duty = 0.4   .plot V(sw)
.param freq = 100K

.tran 10/freq
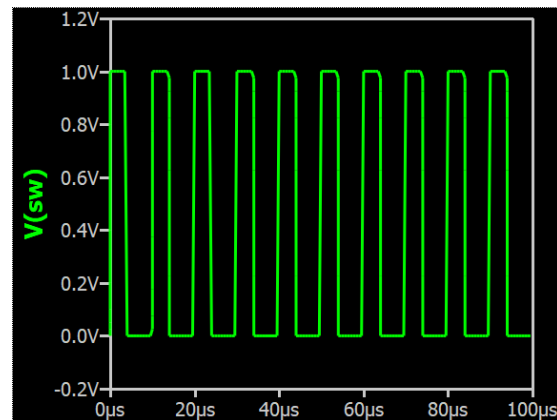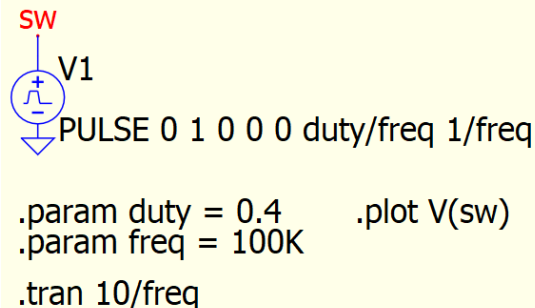
- In transient analysis, if simulation results is not convergence, try following options
  - Add **.option MAXSTEP=x**, where x is maximum step size for transient analysis
    - Sometimes it requires to limit step size especially circuit consist of pulse and logic
  - Goto Edit > Preferences, disable 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
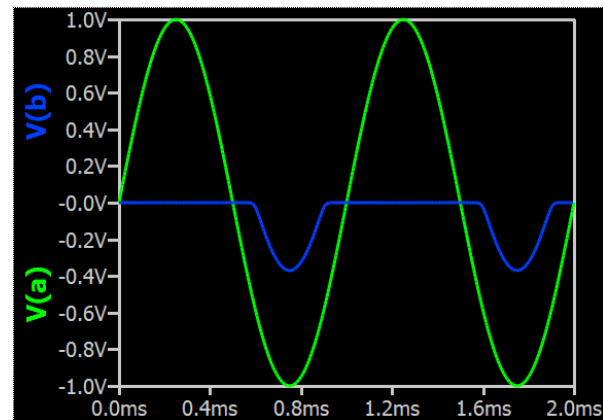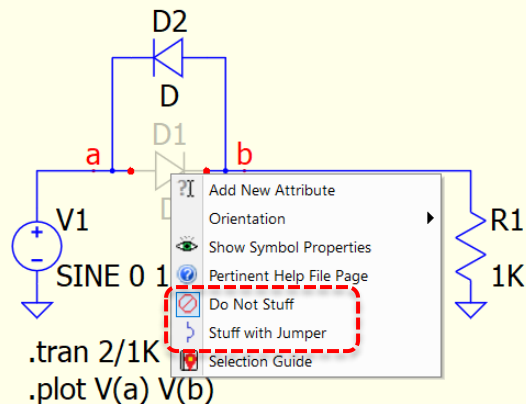
- GND, Chassis, Earth
  - These net names are synonyms to node "0"
    - ** synonyms only apply when schematic to netlist. You can use these names in .cir as normal net name in simulation
  - If you type a net with these names in schematic, they will auto convert into ground symbol. These symbols in netlist are all named as 0
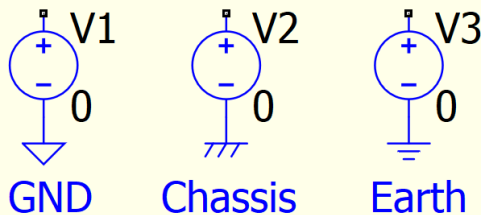
- View > Netlist

  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)

C1
x   y
V1
L1
z
R1
+
−
1µ   10µ
1
sin 0 1 f
ac 1
.param f=60K

.ac dec 100 1e3 1e6
.tran 5/f

use Ctrl+Enter for next line

bottom graph
.plot v(z)
.plot v(y)
.plot v(x)
top graph
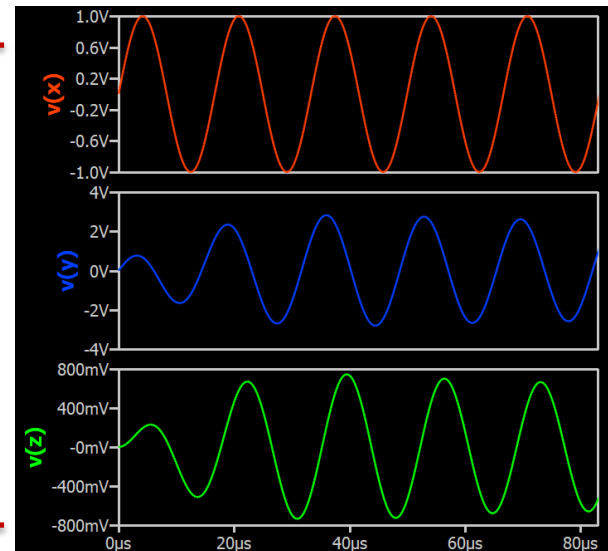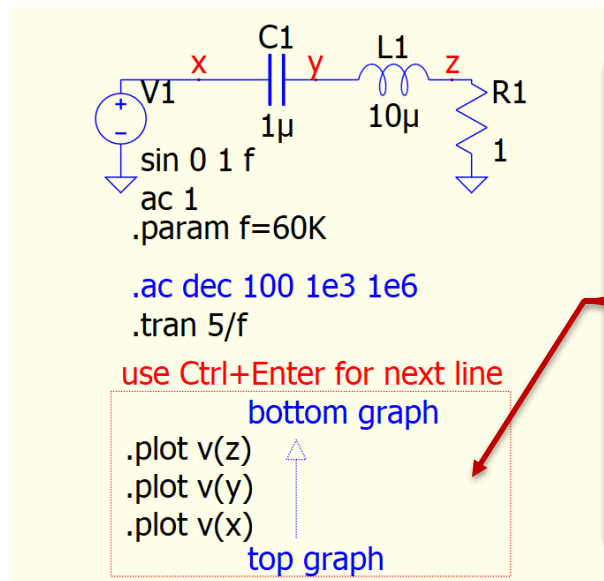
FFT in Waveform Viewer
Qspice : FFT waveform viewer.qsch

[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)

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

Use .func for rms to peak conversion

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)

Zoom to Fit	
Autorange	
Zoom Abscissa Extent	
Add Plot	Ctrl+A
Add Window	Ctrl-W
Delete Window	Ctrl-D
Show Grid	Ctrl+G
Hz FFT	
Copy Bitmap to Clipboard	
Rerun Simulation	F5

FFT calculates voltage amplitude as RMS vs frequency

Window Function: Rectangular(none)

Part 04
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 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

sin
V1
SIN 0 1 1K

cos
V2
SIN 0 1 1K 0 0 90
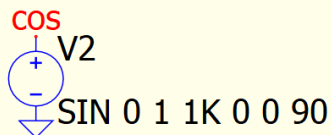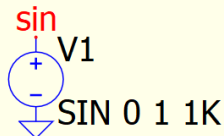
.tran 4/1K
.plot V(sin)
.plot V(cos)

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

maths function with input

.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
V1
0

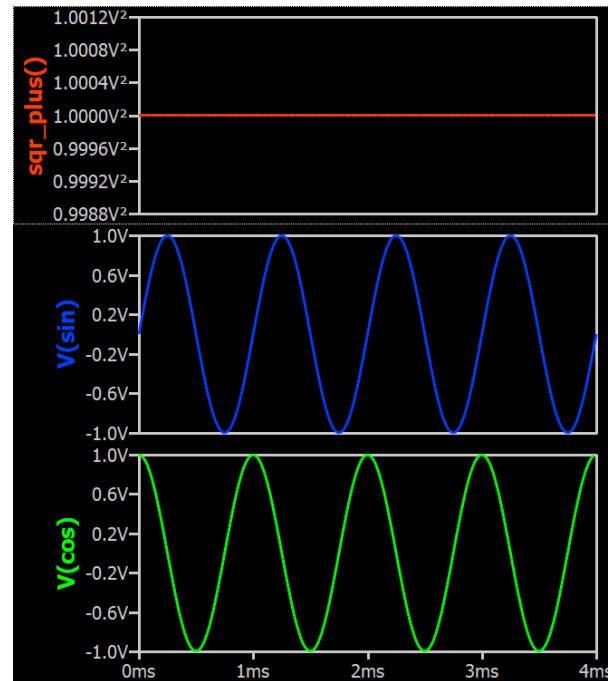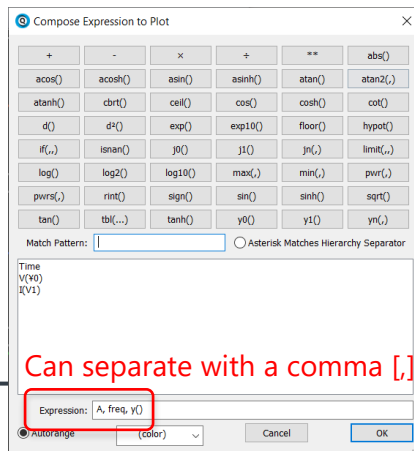Define Amplitude and Freq (in rad/s)
.param A = 2
.param omega = 2*pi*1K

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)}
```

Compose Expression to Plot

+	-	×	÷	**	abs()
acos()	acosh()	asin()	asinh()	atan()	atan2(,)
atanh()	cbrt()	ceil()	cos()	cosh()	cot()
d()	d²()	exp()	exp10()	floor()	hypot()
if(,,)	isnan()	j0()	j1()	jn(,)	limit(,,)
log()	log2()	log10()	max(,)	min(,)	pwr(,)
pwrs(,)	rint()	sign()	sin()	sinh()	sqrt()
tan()	tbl(...)	tanh()	y0()	y1()	yn(,)

Match Pattern:
○ Asterisk Matches Hierarchy Separator

Time
V(¥0)
I(V1)

Can separate with a comma [,]

Expression: A, freq, y()
○ Autorange   (color)   Cancel   OK
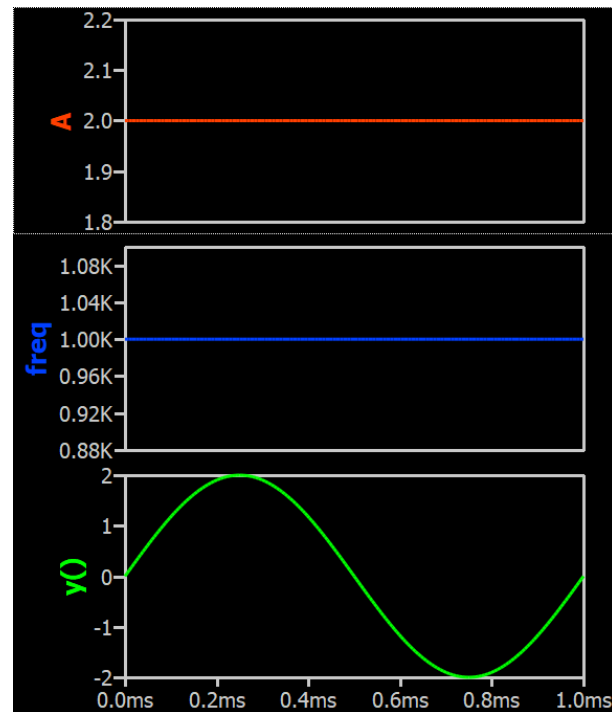
Part 05
Simulation Technique

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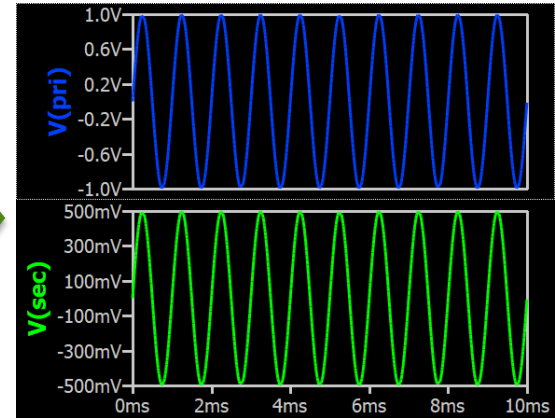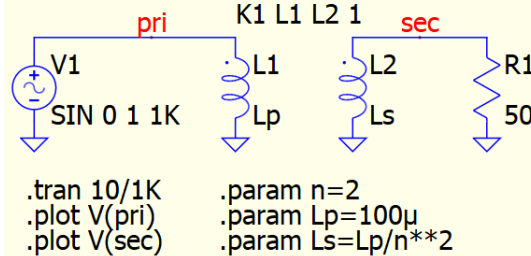 - $L_p = n^2 L_s$ or $L_s = \frac{1}{n^2} L_p$
  - In general practice, we measure primary inductance of transformer (Lp) and know turn ratio (n)
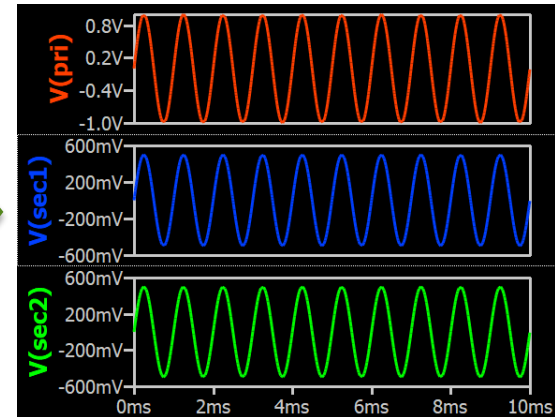
- Model
  - Two or more coupled inductors are required
    - Not necessary but recommend press L two times to get an inductor symbol with a dot notation
  - K is Mutual Inductance defines mutual coupling coefficient of coupled inductors
    - Ideal coupling : 1

Demo of two windings

pri   K1 L1 L2 1   sec

V1
SIN 0 1 1K   L1   L2   R1
              Lp   Ls   50

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

Demo of three windings   sec2

                              L3   R2
K1 L1 L2 L3 1   Ls   50

pri   sec1

V1
SIN 0 1 1K   L1   L2   R1
              Lp   Ls   50

.tran 10/1K   .param n=2
.plot V(pri)   .param Lp=100μ
.plot V(sec1) .param Ls=Lp/n**2
.plot V(sec2)
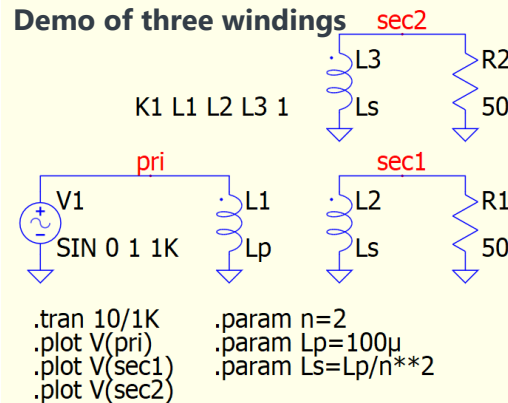
Simulate Functions : Arbitrary Behavioral Source [B]
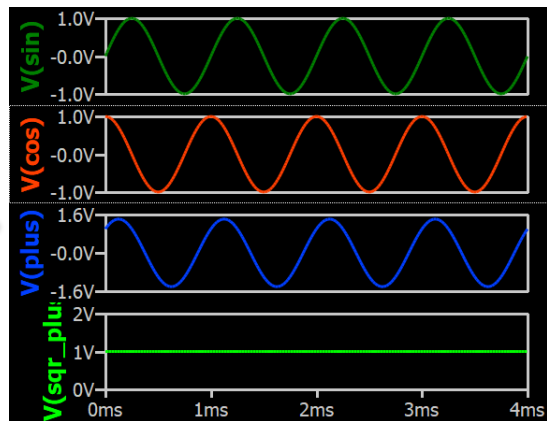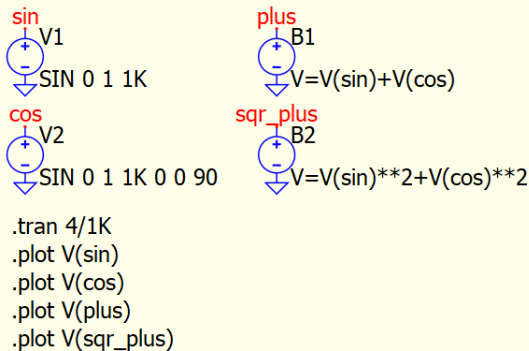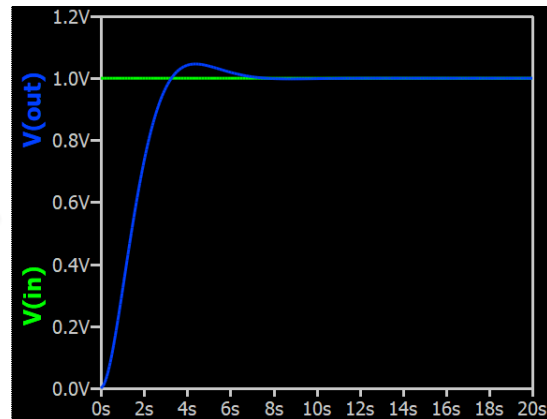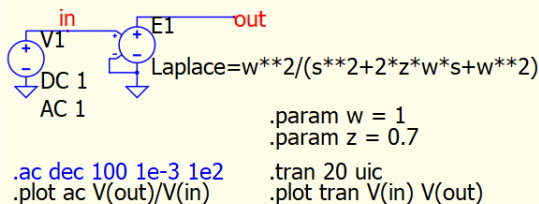## Qspice : B - Functions.qsch ; E - Laplace.qsch

- 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

sin
V1
SIN 0 1 1K

cos
V2
SIN 0 1 1K 0 0 90

plus
B1
V=V(sin)+V(cos)

sqr_plus
B2
V=V(sin)**2+V(cos)**2

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

Demo of Laplace with E-source

Second-Order Transfer Function

in
V1
DC 1
AC 1

E1
out
Laplace=w**2/(s**2+2*z*w*s+w**2)

.param w = 1
.param z = 0.7

.ac dec 100 1e-3 1e2
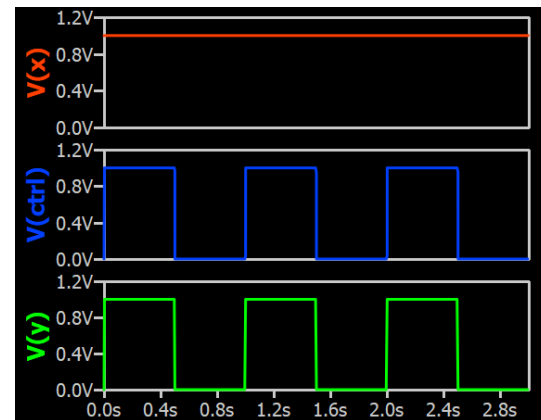.plot ac V(out)/V(in)
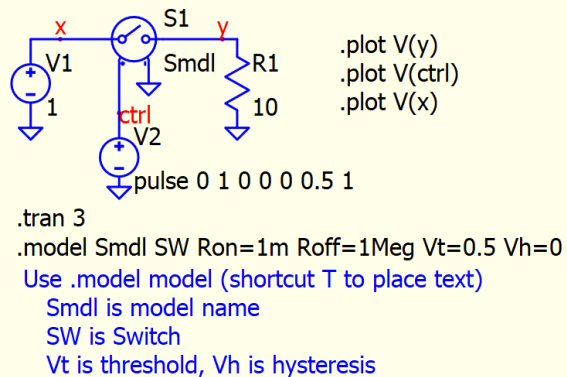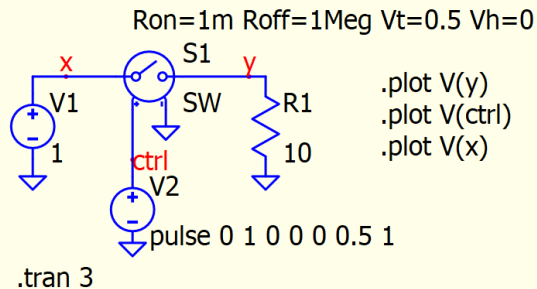
.tran 20 uic
.plot tran V(in) V(out)

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

Ron=1m Roff=1Meg Vt=0.5 Vh=0

S1

x   y

V1   SW   R1

1   ctrl   10

V2

pulse 0 1 0 0 0 0.5 1

.tran 3

.plot V(y)
.plot V(ctrl)
.plot V(x)

S1

x   y

V1   Smdl   R1

1   ctrl   10

V2

pulse 0 1 0 0 0 0.5 1

.tran 3
.model Smdl SW Ron=1m Roff=1Meg Vt=0.5 Vh=0
Use .model model (shortcut T to place text)
   Smdl is model name
   SW is Switch
   Vt is threshold, Vh is hysteresis

.plot V(y)
.plot V(ctrl)
.plot V(x)

Simulate Ideal Diode : Diode (D)

- 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
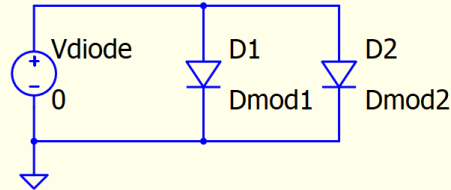
.model Dmod1 D Ron=1µ Roff=10Meg Vfwd=0
.model Dmod2 D Ron=1µ Roff=10Meg Vfwd=0.7

.dc Vdiode -.4 1 0.01
.plot I(D2) I(D1)