

# Qspice - Command Reference Guide by KSKelvin

KSKelvin Kelvin Leung

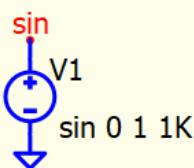
Created on 8-4-2023  
Last update on 2-2-2025

**Comment**  
**(Shortcut ";" )**

# Comment

Qspice : Comment - 3 Type.qsch | Comment - Chinese.qsch

## 3 Type of Comment



Type #1 : double slash //

//Type 1:  
//Text Comment with double slash

Type #2 : Shortcut semicolon ";"

Type 2:  
Text Comment  
\* Right Click Text Box > This is a text comment  
\* OR Shortcut ";"

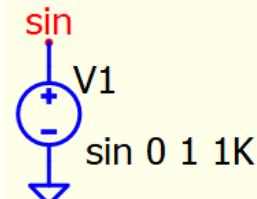
Type #3 : semicolon in directive

.tran 5/1K ;Type 3 : Text Comment with semicolon in directive  
.plot V(sin)

## Comment with Chinese Character

### Qspice New Feature

01/30/2024 Unicode text is now supported for comments placed on schematics.



//繁體中文範例  
//简体中文范例

繁體中文範例 (Traditional Chinese)  
简体中文范例 (Simplified Chinese)

.tran 5/1K ;瞬態響應  
.plot V(sin) ;繪劃波型

# Comment – Common Unicode for Electronics

Qspice : Comment - Common Unicode for Electronics.qsch

[https://en.wikipedia.org/wiki/List\\_of\\_Unicode\\_characters](https://en.wikipedia.org/wiki/List_of_Unicode_characters)

<https://www.compart.com/en/unicode/block>

## Unicode Block "Superscripts and Subscripts" U+2070 to U+209F

Superscripts : 0 1 2 3 4 5 6 7 8 9 + - = ( ) n

Subscripts : 0 1 2 3 4 5 6 7 8 9 + - = ( ) a e o x a b k l m n p s t

Unicode Block "Latin-1 Supplement" U+00A1 to U+00BF, U+00D7, U+00F7

Latin-1 : ¡ ¢ £ ¥ § ª « ¬ ® ° ± ² ³ ´ µ ¶ · ¸ ¹ º » ¼ ½ ¾ ¸

## Math : x ÷

## Unicode Block "Greek and Coptic" U+0391 to U+03A9 and U+03B1 to U+03C9

Greek (Capital) : Α Β Γ Δ Ε Ζ Η Θ Ι Κ Λ Μ Ν Ξ Ο Π Ρ Σ Τ Υ Φ Χ Ψ Ω

Greek (Small Letter) : α β γ δ ε ζ η θ ι κ λ μ υ ξ ο π ρ σ τ υ φ χ ψ ω

## Unicode Block "Number Forms" between U+2150 to U+218F

Number Forms :  $\frac{1}{2}$   $\frac{1}{9}$   $\frac{1}{10}$   $\frac{1}{3}$   $\frac{2}{3}$   $\frac{1}{5}$   $\frac{2}{5}$   $\frac{3}{5}$   $\frac{4}{5}$   $\frac{1}{6}$   $\frac{5}{6}$   $\frac{1}{8}$   $\frac{3}{8}$   $\frac{5}{8}$   $\frac{7}{8}$   $\frac{1}{0/3}$

Unicode Block "Mathematical Operators" between U+2200 to U+22FF

Math : A  $\nabla$   $\Sigma$  -  $\mp$   $\circ$   $\sqrt{3}$ / $\sqrt[3]{4}$

Math :  $\infty \infty$

Math ; ; ; < >

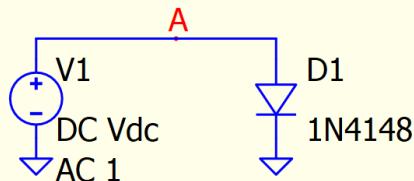
.ac

## Small-Signal AC Analysis

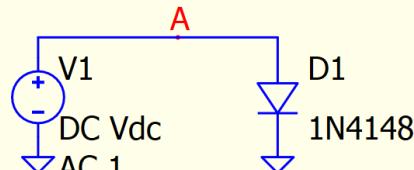
# .ac – Small-Signal AC Analysis

Qspice : .ac - basic.qsch

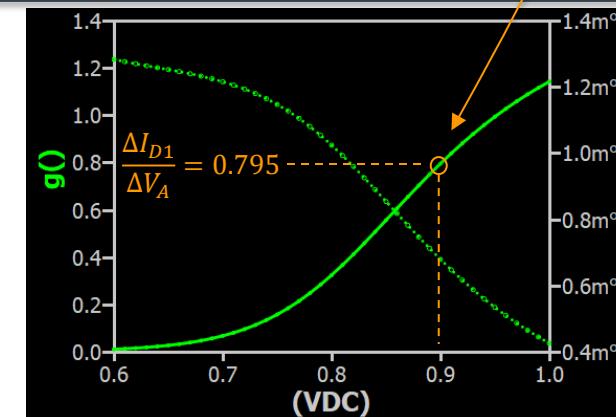
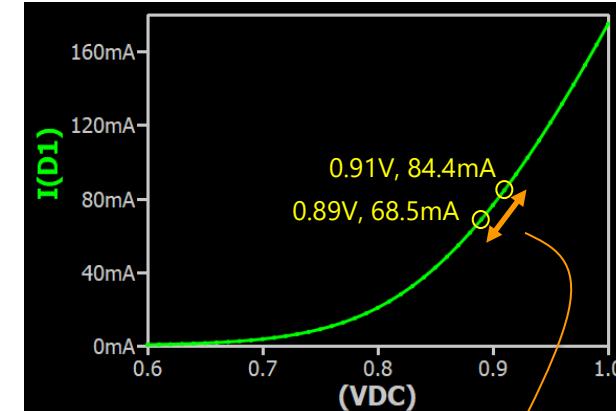
- Small-Signal AC Analysis
  - .ac performs small-signal AC analysis by computing the frequency domain response at the circuit's DC operating point
    - .op is executed to calculate the DC operating point before running .ac
  - This technique is effective for continuous-time non-switching circuits. For small-signal response analysis of switching circuits, the .bode directive
  - In this example, .step is utilized to sweep through different DC operating points and obtain .ac results at a single frequency point



```
.step param Vdc 0.6 1 0.01  
.op  
.plot I(D1) .ac list 1K  
.func g() I(D1)/V(a)  
.plot g()
```



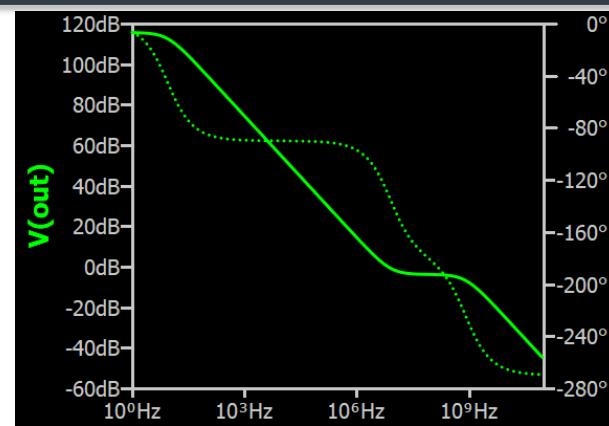
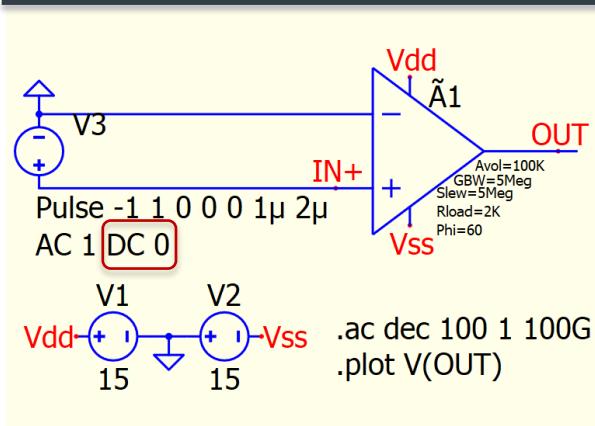
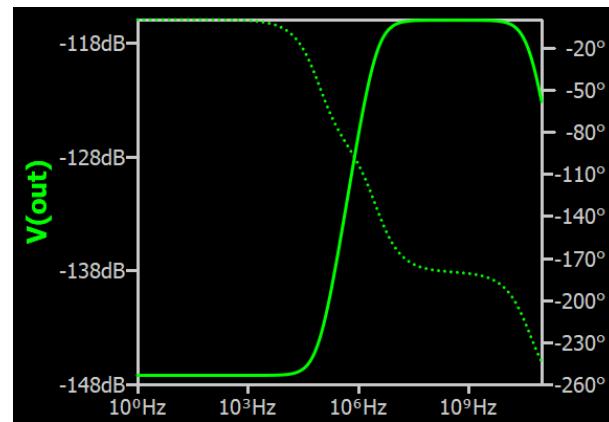
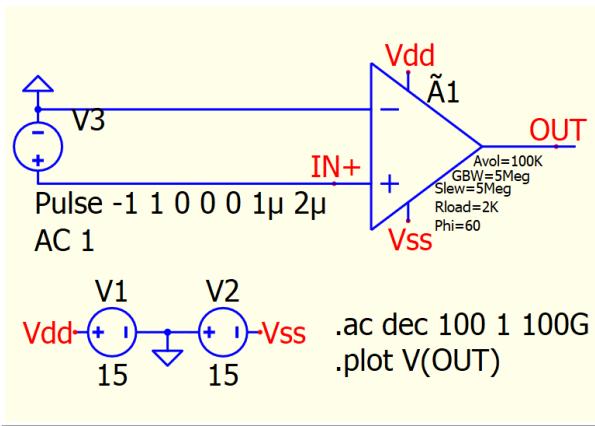
```
.step param Vdc 0.6 1 0.01  
.op  
.plot I(D1) .ac list 1K  
.func g() I(D1)/V(a)  
.plot g()
```



# .ac – DC operating point in .ac from V-Source

Qspice : .ac - DC in V-source.qsch

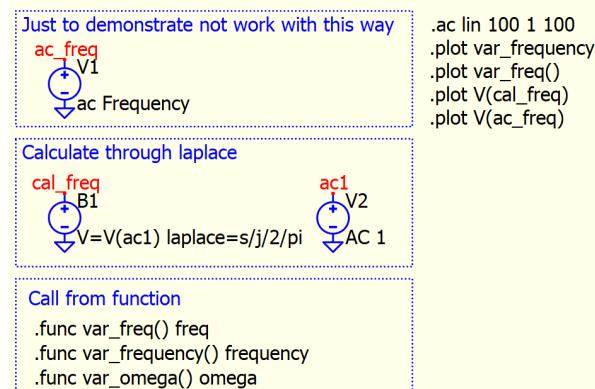
- DC operating point
  - There is special attention in .ac with source that support transient analysis
  - This example measures opamp open loop gain, but with a pulse statement included in V3, open loop profile is affected
  - The reason is that, in default, .op is run with source voltage at t=0 if DC is not defined. This pulse statement set -1V at t=0, which saturated this opamp by .op with  $V(IN+)= -1V$
  - Solution is to add DC into V-source to force a DC voltage for .op before .ac is run
  - More information can be found in ".op" section



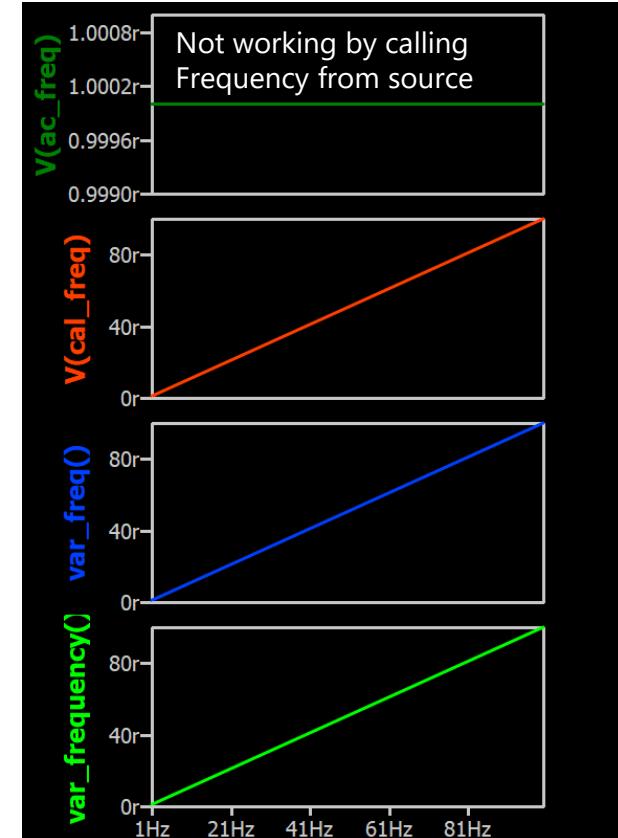
# .ac – keyword : freq, frequency and omega

Qspice : .ac - keyword freq and frequency.qsch

- Keyword
  - Freq/Frequency/Omega are keywords used in .ac analysis, it is advisable not to define .param with these keywords, as it may result in errors
  - These keywords can be accessed in post-processing (thus, .func can use freq/frequency/omega), but they are not available during the simulation run
  - It can calculate frequency with Laplace function ( $\frac{s}{j2\pi}$ ) and an AC 1V source



```
.ac - keyword freq and frequency.qsch
1 Title: * C:\KSKelvinQspice\01 U
2 Date: Mon May 13 21:10:35 2024
3 Plotname: AC Analysis
4 Plot Suggestion(s): «var_frequ
5 Flags: complex
6 Abscissa: 1.0000000000000000
7 No. Variables: 7
8 No. Points: 100
9 Command: QSPICE80, Build May 11
10 .param temp=27
11 .func VAR_FREQ() FREQ
12 .func VAR_FREQUENCY() FREQUENCY
13 .alias Freq Frequency
14 .alias Omega 2*pi*Frequency
15 Variables:
16     0    Frequency   frequency
17     1    V(ac_freq)  voltage
18     2    V(ac1)      voltage
19     3    V(cal_freq) voltage
```



# .ac – use frequency in .ac

Qspice : .ac - freq from keyword.qsch | .ac - frq with list.qsch

- Use frequency in .ac

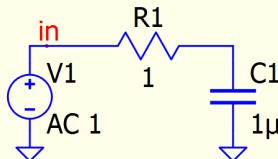
- Examples to demonstrate if simulation requires frequency information

- Example #1

- Use keyword (e.g. freq) in post-processing calculation
- Advantage is faster simulation, but only available for post-processing calculations such as .func

- Example #2

- Define .param frq and step a single point frequency analysis .ac list
- Advantage is that frq is defined as a .param and can be used during simulation. However, simulation is slower because each step requires recalculating .op
- This method is most reliable way for .ac with behavioral devices

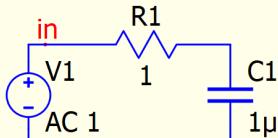


.plot Cs()  
.plot Rs()  
.plot Z()

.ac dec 10 1K 100K

.func Z() V(in)/-I(V1)  
.func Rs() re(Z())  
.func Cs() -1/im(Z())/2/pi/freq

Use keyword freq

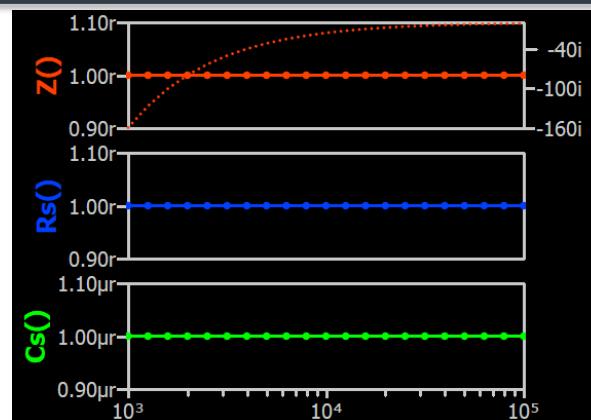
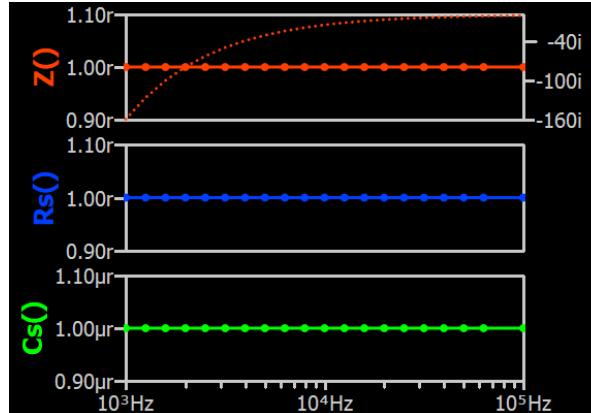


.plot Cs()  
.plot Rs()  
.plot Z()

.param frq=1K  
.step dec param frq 1K 100K 10  
.ac list frq

.func Z() V(in)/-I(V1)  
.func Rs() re(Z())  
.func Cs() -1/im(Z())/2/pi/frq

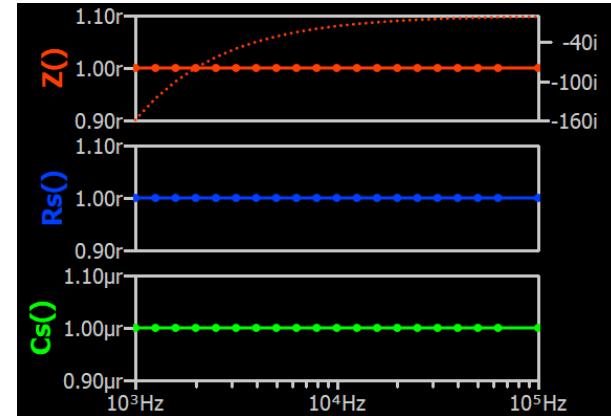
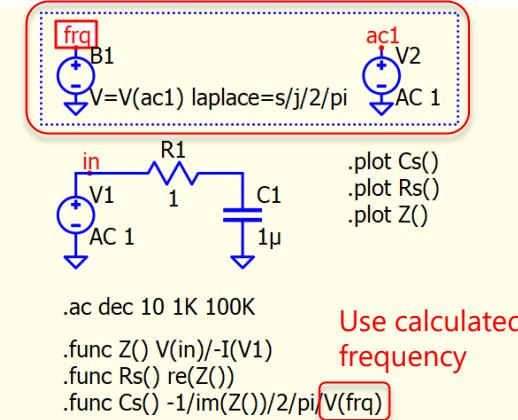
Define param frq  
Use .ac list



# .ac – use frequency in .ac

Qspice : .ac - freq with symbol.qsch

- Use frequency in .ac
  - Example #3 (Not Recommend)
    - Use frequency calculation behavioral source and utilize its result for calculation
    - Unlike previous case, this frequency is captured from a voltage node and expressed as V(freq)
    - Be caution that don't V(freq) can be used to define a behavioral devices as it may not work! Recommend to use 2<sup>nd</sup> example method



.four

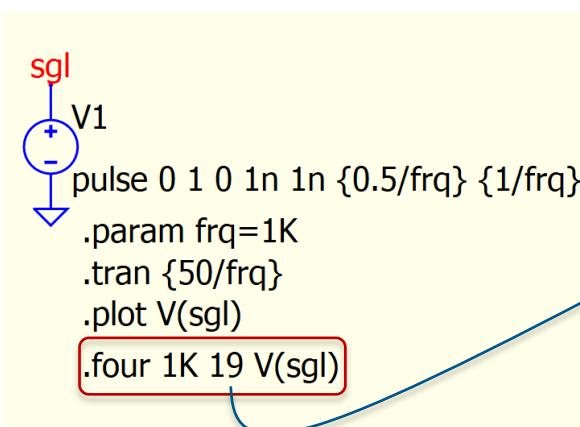
THD Computation

# .four THD Computation

Qspice : .four - harmonics.qsch

- Syntax: .four FREQ [HARMONICS] [PERIODS] expr1 [expr2 [expr3 [...]]]
  - FREQ : fundamental frequency
  - HARMONICS : number of harmonics to compute (Default HARMONICS=9)
  - PERIODS : number of period to be used to compute THD
  - expr1 : expression (e.g. V(out), I(V1) etc...)

- [HARMONICS]
  - Number of harmonics to compute



Harmonic	Frequency	Magnitude	Phase
1	1.000e+03	1.000e+00	0.00°
2	2.000e+03	5.890e-06	-90.00°
3	3.000e+03	3.333e-01	-0.00°
4	4.000e+03	5.890e-06	-90.00°
5	5.000e+03	2.000e-01	-0.01°
6	6.000e+03	5.890e-06	-89.99°
7	7.000e+03	1.429e-01	-0.01°
8	8.000e+03	5.890e-06	-89.99°
9	9.000e+03	1.111e-01	-0.01°
10	1.000e+04	5.890e-06	-89.99°
11	1.100e+04	9.091e-02	-0.02°
12	1.200e+04	5.890e-06	-89.99°
13	1.300e+04	7.692e-02	-0.02°
14	1.400e+04	5.890e-06	-89.99°
15	1.500e+04	6.667e-02	-0.02°
16	1.600e+04	5.890e-06	-89.99°
17	1.700e+04	5.882e-02	-0.02°
18	1.800e+04	5.890e-06	-89.99°
19	1.900e+04	5.263e-02	-0.03°

THD = 45.686% (48.3398%)

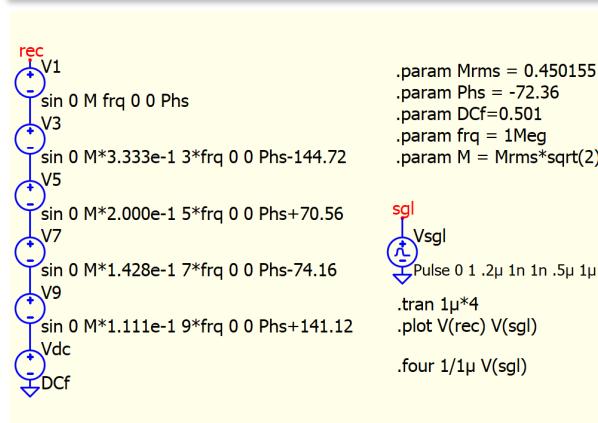
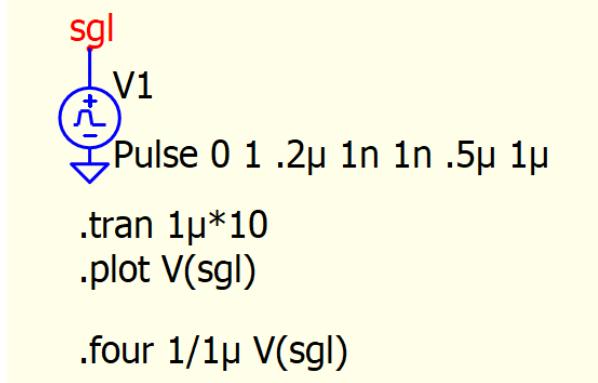
# .four meaning and reconstruction

Qspice : .four - explain.qsch | .four - reconstruction

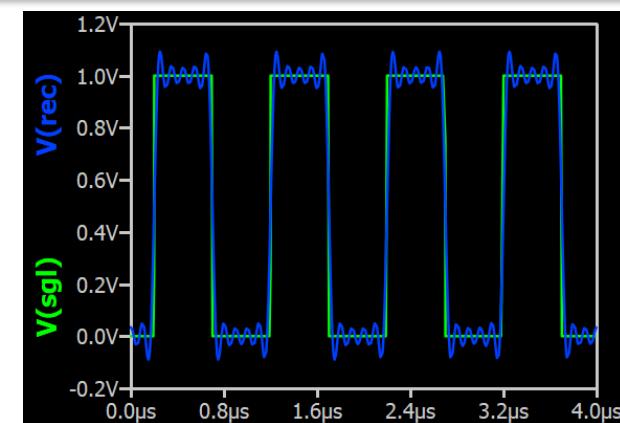
- Meaning from .four
  - [1] Fundamental Return
    - RMS Magnitude ( $Mag_{rms}$ )
    - Phase ( $Phs$ )
    - DC ( $DC$ )
  - [2] Harmonic Return
    - Harmonic order ( $n$ )
    - Frequency ( $f_n$ )
    - Magnitude ( $Mag_{nom,n}$ )
    - Phase ( $Phs_{nom,n}$ )
- Reconstruction
  - .four uses SINE function for reconstruction (unlike .meas four)

Signal

$$= \sum_n \sqrt{2} Mag_{rms} Mag_{nom,n} \sin(2\pi f_n T + Phs + Phs_{nom,n})$$



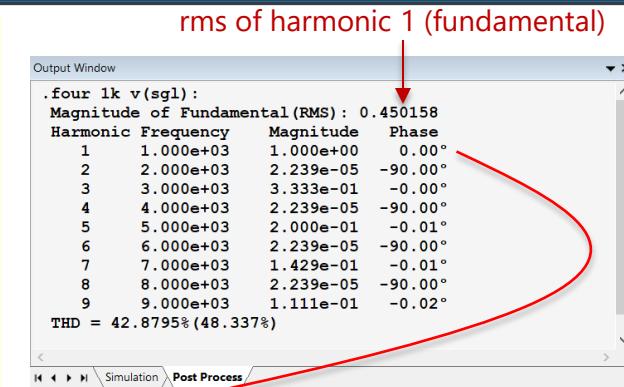
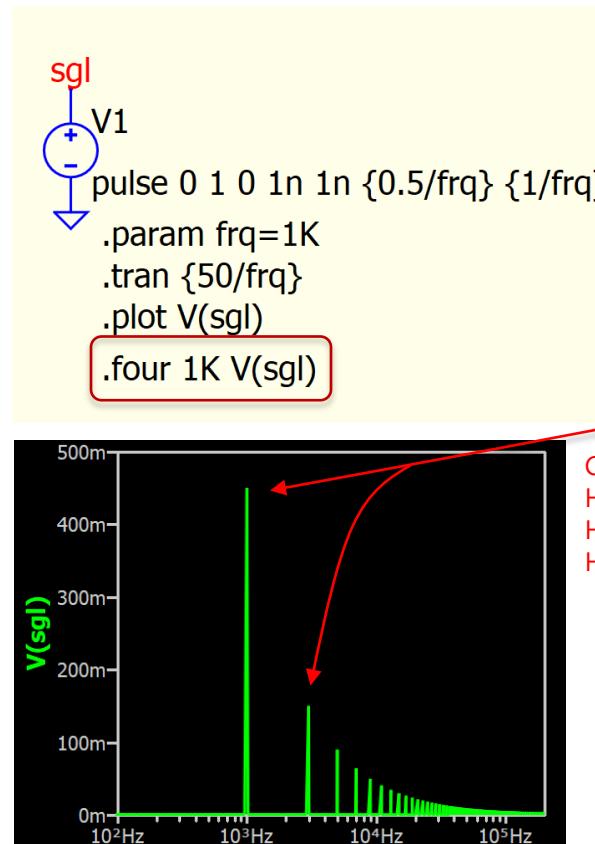
```
.four 1/1μ v(sgl);
Fundamental: RMS Magnitude=0.450155 Phase=-72.36° DC=0.501
Harmonic   Frequency      Magnitude    Phase
1          1.000e+06     1.000e+00     0.00°
2          2.000e+06     3.142e-03    17.64°
3          3.000e+06     3.333e-01   -144.72°
4          4.000e+06     3.141e-03   -127.08°
5          5.000e+06     2.000e-01    70.56°
6          6.000e+06     3.141e-03    88.20°
7          7.000e+06     1.428e-01   -74.16°
8          8.000e+06     3.141e-03   -56.52°
9          9.000e+06     1.111e-01   141.12°
THD = 42.8782% (48.1732%)
```



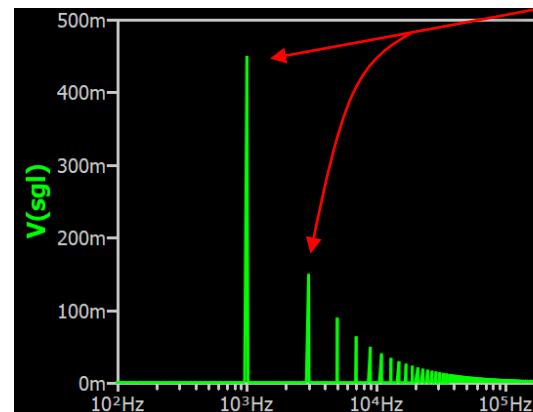
# .four THD Computation

Qspice : .four - basic.qsch

- .four THD
  - [HARMONICS] and [PERIODS] is not necessary input parameters
  - Magnitude and phase are normalized component equal 1 and 0 degree
  - To compare FFT, FFT is to plot magnitude in RMS
    - Therefore, it requires to calculate with Magnitude \* Magnitude of Fundamental (RMS) in .four to match calculation in FFT
    - In waveform viewer, right click, select FFT



Calculate  $V(sgl)$ ,rms from .four into FFT chart  
Harmonic 1 : Magnitude\*RMS =  $1.000e0 * 0.45 = 0.45$   
Harmonic 2 : Magnitude\*RMS =  $2.239e-5 * 0.45 = 0.00001$   
Harmonic 3 : Magnitude\*RMS =  $3.333e-1 * 0.45 = 0.15$   
:  
:



# Compare Qspice and LTspice .four log

## Qspice

```
.four 1k v(sgl):
Fundamental: RMS Magnitude=0.450158 Phase=-0.00241709° DC=0.5
Harmonic Frequency Magnitude Phase
  1 1.000e+03 1.000e+00 0.00°
  2 2.000e+03 2.239e-05 -90.00°
  3 3.000e+03 3.333e-01 -0.00°
  4 4.000e+03 2.239e-05 -90.00°
  5 5.000e+03 2.000e-01 -0.01°
  6 6.000e+03 2.239e-05 -90.00°
  7 7.000e+03 1.429e-01 -0.01°
  8 8.000e+03 2.239e-05 -90.00°
  9 9.000e+03 1.111e-01 -0.02°
THD = 42.8795% (48.337%)
```

Equivalent  
(for order e-05, error  
between simulator can  
be expected)

## LTspice

Fourier Component in LTspice .four is Peak  
Qspice : Magnitude \* RMS \* sqrt(2) = 1\*0.450158\*sqrt(2) = 0.6366

Fourier components of V(sgl)  
DC component: 0.500001

Harmonic Number	Frequency [Hz]	Fourier Component	Normalized Component	Phase [degree]	Normalized Phase [deg]
1	1.000e+3	6.366e-1	1.000e+0	90.00°	0.00°
2	2.000e+3	2.000e-6	3.142e-6	0.00°	-90.00°
3	3.000e+3	2.122e-1	3.333e-1	90.00°	0.00°
4	4.000e+3	2.000e-6	3.142e-6	0.00°	-90.00°
5	5.000e+3	1.273e-1	2.000e-1	90.00°	0.00°
6	6.000e+3	2.000e-6	3.142e-6	0.00°	-90.00°
7	7.000e+3	9.095e-2	1.429e-1	90.00°	0.00°
8	8.000e+3	2.000e-6	3.142e-6	0.00°	-90.00°
9	9.000e+3	7.074e-2	1.111e-1	90.00°	0.00°

Partial Harmonic Distortion: 42.880402%  
Total Harmonic Distortion: 48.342146%

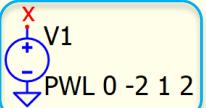
.func  
Function

# .func – User-Defined Function

Qspice : .func - math ex1.qsch | .func - math ex2.qsch

- .func
  - User-Defined Function
  - Mathematic formula can be implemented in multiple ways, here are two examples in .tran and .op
  - Two example to show how to plot formula
    - $y = ax^2 + bx + c$
  - In .tran example, a voltage source is used for input x
  - In .op example, use .step for input x
  - In both examples, .step is used to change coefficient b

Math formula implementation with .tran

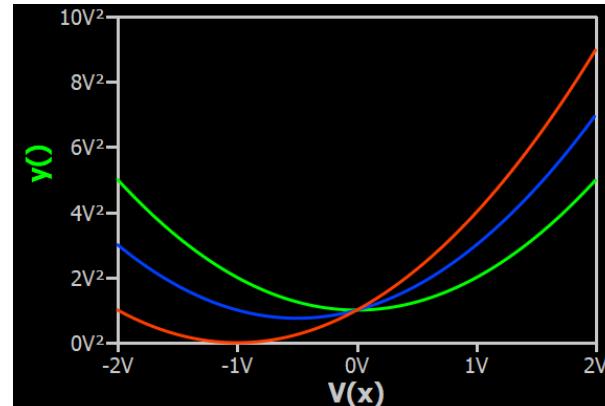
```
.param a=1  
.param b=0  
.param c=1  
  
.step param b list 0 1 2
```

Change coefficient b

```
.func y() a*V(x)**2+b*V(x)+c .plot y()
```

```
.option SaveParam
```

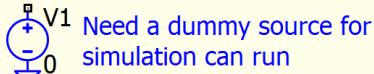
```
.tran 1
```



Math formula implementation with .op

```
.param a=1  
.param b=0  
.param c=1  
.step param x -2 2 0.01 param b list 0 1 2
```

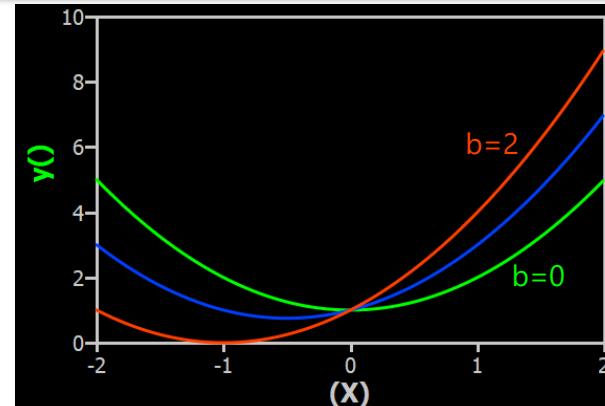
Change coefficient b

 Need a dummy source for simulation can run

```
.func y() a*x**2+b*x+c .plot y()
```

```
.option SaveParam
```

```
.op
```



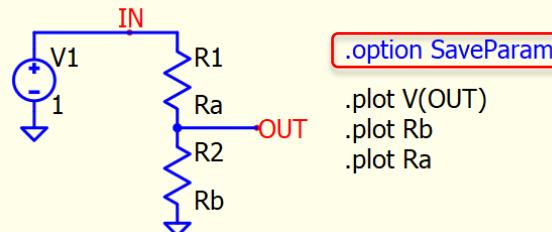
# .func – User-Defined Function (with .option SaveParam)

Qspice : .func with listparam.qsch

- .option SaveParam
  - Example to demonstrate .param with formula
  - In this Monte Carlo example, .param defines Ra and Rb as resistance that randomly change by 5% in each simulation step with the help of dummy .step command
- However, without .option SaveParam, the randomly generated Ra and Rb for each step will not store in the data file as they are only parameters
- By including .option SaveParam, their randomly generated values will be stored in the data file (.qraw) and can be selected to plot

```
.param Ra mc(1K,0.05)
.param Rb mc(1.2K,0.05)

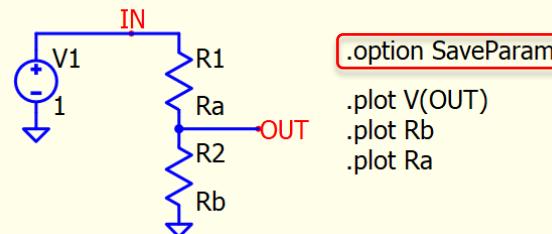
.step param LOOP 1 40 1
.op
```



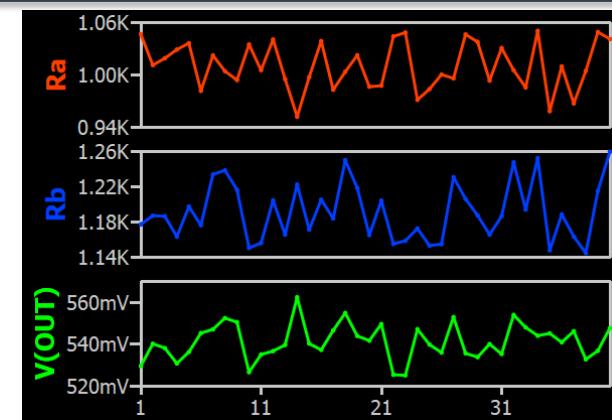
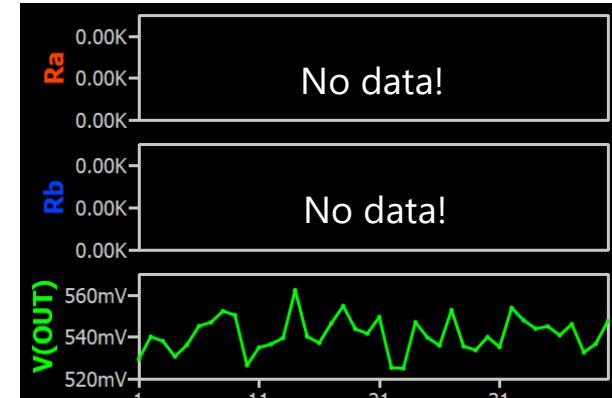
```
.plot V(OUT)
.plot Rb
.plot Ra
```

```
.param Ra mc(1K,0.05)
.param Rb mc(1.2K,0.05)

.step param LOOP 1 40 1
.op
```



```
.option SaveParam
.plot V(OUT)
.plot Rb
.plot Ra
```



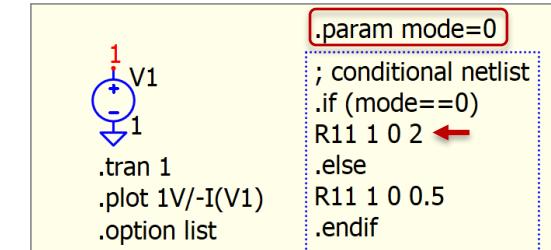
**.if/.else/.endif**

**If conditional netlist  
(Undocumented)**

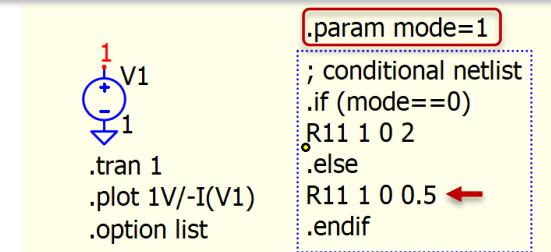
# .if/.else/.endif – If conditional netlist processing

Qspice : .if - netlist device.qsch

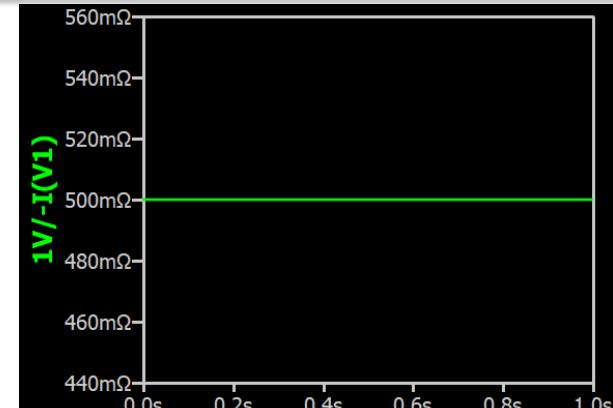
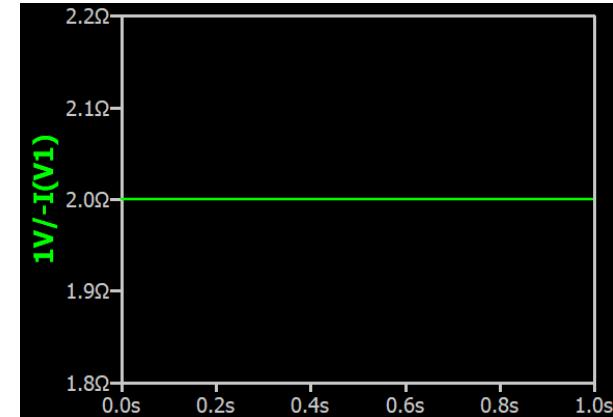
- .if/.else/.endif
  - Conditional netlist processing
  - Limitation
    - .elseif not implemented
    - .step param not support as implementation is from the schematic level
    - Mike explained that this implementation is for in-house Qorvo engineers could replace Spectre with QSPICE for some foundry models



```
* C:\KSKelvinQspice\01 User Guide and Script\03 Command
V1 1 0 1
R11 1 0 2
.TRAN 1
.PLOT 1V/-I(V1)
.PARAM MODE=0
.OPTION LIST
.END
```



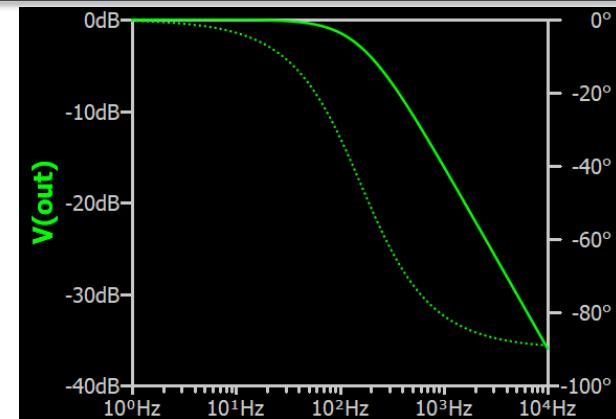
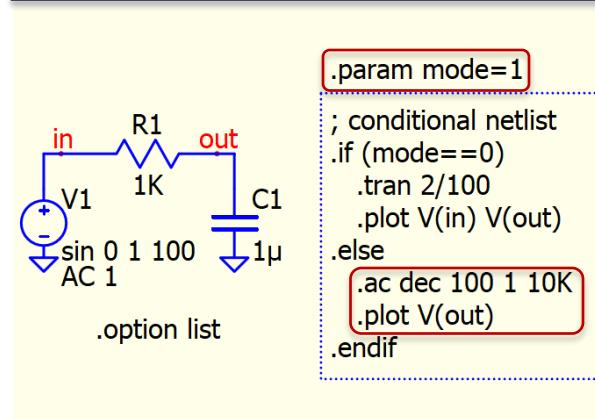
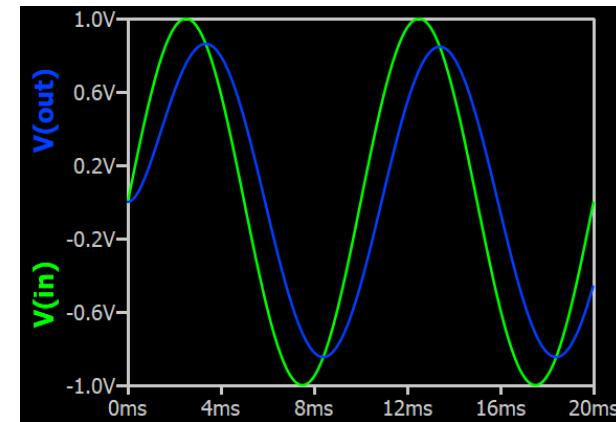
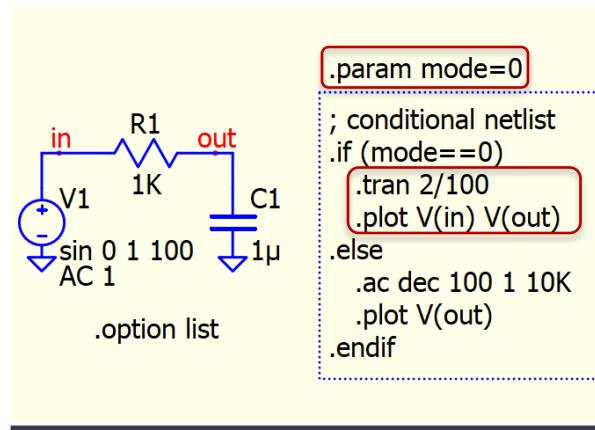
```
* C:\KSKelvinQspice\01 User Guide and Script\03 Command Re
V1 1 0 1
R11 1 0 0.5
.TRAN 1
.PLOT 1V/-I(V1)
.PARAM MODE=1
.OPTION LIST
.END
```



# .if/.else/.endif – If conditional netlist processing

Qspice : .if - simulation directive.qsch

- .if/.else/.endif
  - An example application in controlling simulation directive
  - Another usage is to select .model to be used



.inc  
**Include File**

# Include File (.inc) : HELP > Schematic Capture > Simulator > Include File (.inc)

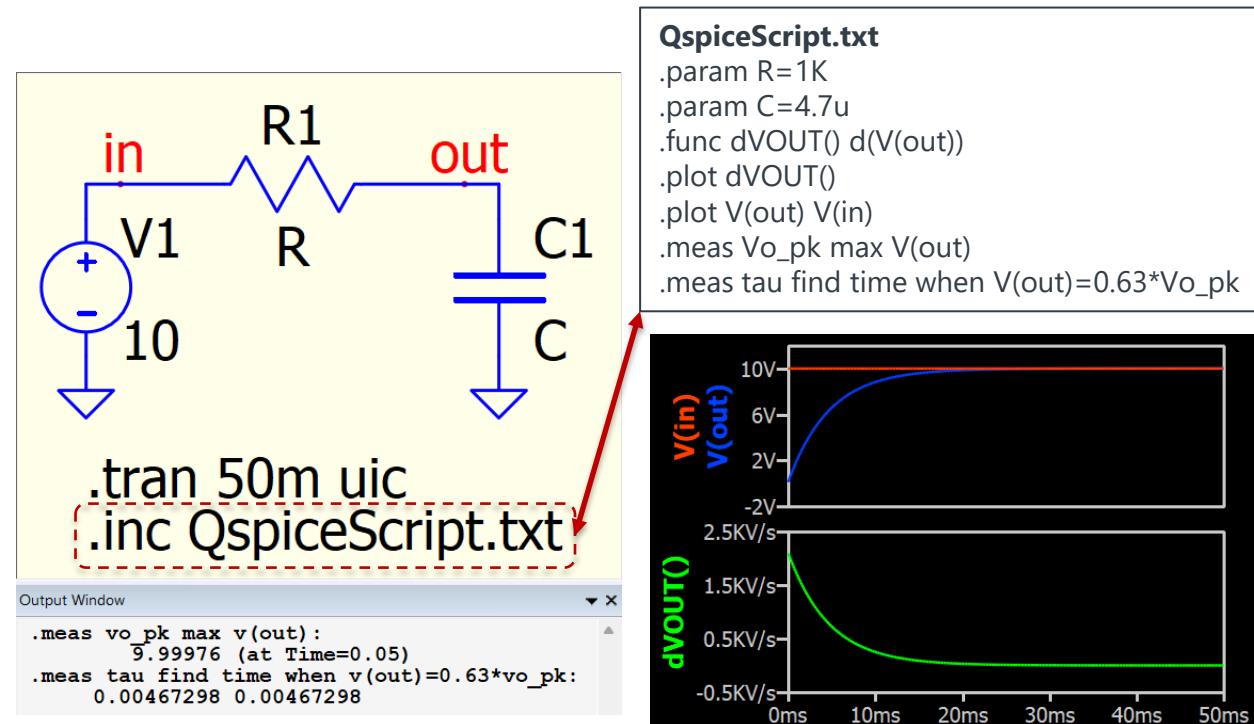
Qspice : .inc - QspiceScript.qsch

- Include File

- To include a file to execute by simulator
- This allow to simplify schematic for directive or reuse purpose

- Example

- This example use .inc to include a file called QspiceScript.txt
- This script can
  - Input .param
  - Calculate .func
  - Define .plot
  - Calculate .meas

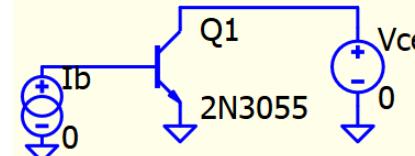


**.libpath**

**Include a Directory  
into the Library File  
Search Path**

# .libpath (Include a Directory into the Library File Search Path)

- **.libpath**
  - Syntax : .libpath <directory>
  - Search path for library .lib and include .inc directive
  - **Search path priority**
    1. Absolute path in .lib and .include
    2. Current working directory
    3. .libpath directories
    4. Qspice installation directory
      - Normal Install Path : C:\Program Files\QSPICE
  - **If .libpath is used, .lib or .inc must after .libpath in netlist**
    - Therefore, recommend to use Ctrl-Enter method after .libpath to add .lib / .inc to ensure this sequence



.dc Vce 0 10 0.01 Ib list 0.01 0.1 0.4 1 2

.libpath C:\QspiceKSKelvin\Qspice Common Library  
.lib "KSKelvin\_standard (from Itwiki).bjt"

.plot Ic(Q1)

C:\QspiceKSKelvin\Qspice Common Library	
Q	KSKelvin_standard (from LTspice).bjt
D	KSKelvin_standard (from LTspice).dio
J	KSKelvin_standard (from LTspice).jft
M	KSKelvin_standard (from LTspice).mos
Q	KSKelvin_standard (from Itwiki).bjt
D	KSKelvin_standard (from Itwiki).dio
J	KSKelvin_standard (from Itwiki).jft
M	KSKelvin_standard (from Itwiki).mos

.meas  
Measure Statements

# Available Syntax for .meas

HELP > Simulator > Command Reference > Measure(.meas)

- Syntax: .meas NAME find EXPRESSION1 at EXPRESSION2
- Syntax: .meas NAME find EXPRESSION1 when EXPRESSION2=EXPRESSION3
- Syntax: .meas NAME find EXPRESSION1 when EXPRESSION2=EXPRESSION3 td=5n cross=10
- Syntax: .meas NAME find EXPRESSION1 when EXPRESSION2=EXPRESSION3 cross=last
- Syntax: .meas NAME deriv EXPRESSION1 at EXPRESSION2
- Syntax: .meas NAME trig EXPRESSION1=EXPRESSION2
- Syntax: .meas NAME targ EXPRESSION1=EXPRESSION2
- Syntax: .meas NAME trig EXPRESSION1=EXPRESSION2 targ EXPRESSION3=EXPRESSION4
- Syntax: .meas NAME trig EXPRESSION1=EXPRESSION2 rise=1 targ EXPRESSION1=EXPRESSION2 rise=11
- Syntax: .meas NAME avg|max|min|pp|rms|integ EXPRESSION1
- Syntax: .meas NAME avg|max|min|pp|rms|integ EXPRESSION1 from EXPRESSION2 to EXPRESSION3
- Syntax: .meas NAME avg|max|min|pp|rms|integ EXPRESSION1 trig EXPRESSION2=EXPRESSION3 targ EXPRESSION4=EXPRESSION5
- Syntax: .meas NAME four FREQ EXPRESSION [...]
- Syntax: .meas NAME fra FREQ INPUT OUTPUT [...]

# .meas with param

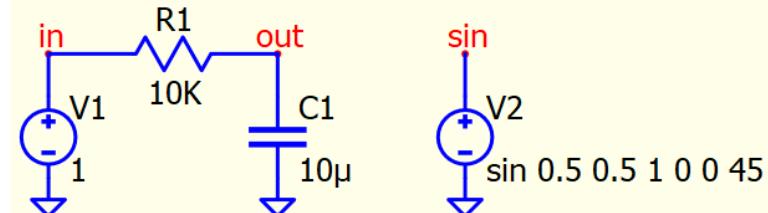
Qspice : meas - param.qsch

- .meas with param

Incomplete syntax: Return result at max simulation time

Syntax with param: Return result at max simulation time

Return result at measurement time



```
.ic V(out)=0
```

```
.tran 1
```

```
.meas Vnothing V(out)
```

```
.meas Vout param V(out)
```

```
.meas Vsine param V(sin)
```

```
.meas Vout@0.15s find V(out) at 0.15
```

Output Window

```
.meas vnothing v(out) : 0.999955  
.meas vout param v(out) : 0.999955  
.meas vsin param v(sin) : 0.853553  
.meas vout@0.15s find v(out) at 0.15: 0.776868 0.15
```

measurement time

measurement time

# .meas in .ac directive – example

Qspice : meas ac demo 01.qsch

Diagram of a circuit with an AC voltage source V1, a dependent voltage source E1, and a load resistor R1. The output voltage is measured across R1.

```
.param fn = 1K
.param wn = 2*pi*fn
.param z = 0.2
Laplace=wn^2/(s^2+2*z*wn*s+wn^2)

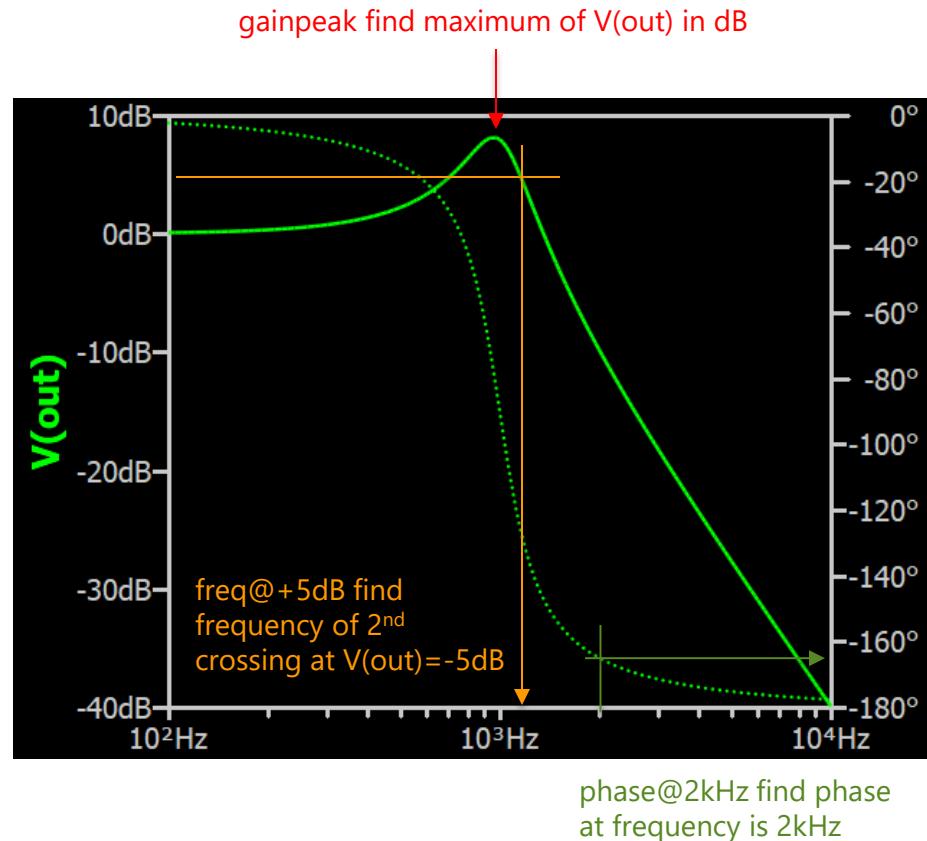
.ac dec 100 100 1e4
.plot V(out)

.meas gainpeak max dB(V(out))
.meas phase@2kHz find phase(V(out)) WHEN frequency=2K
.meas freq@+5dB find frequency WHEN dB(V(out))=5 cross=2

.meas gainpeak max db(v(out)):
( 8.13428, -0.350754) (at Frequency=954.993)
.meas phase@2khz find phase(v(out)) when frequency=2k:
( -165.064, 0)
.meas freq@+5db find frequency when db(v(out))=5 cross=2:
( 1150.32, 0)
```

**Support 6 types of measure**

- avg/ave : average
- max : maximum
- min : minimum
- pp : peak to peak
- rms : root mean square
- integ : integral



# .meas in .ac directive – complex format results

Qspice : meas ac representation.qsch

- .meas results in .ac directive
  - Result is complex number : (real, imag)

$$V(out) = 0.0247045 - j0.155223$$

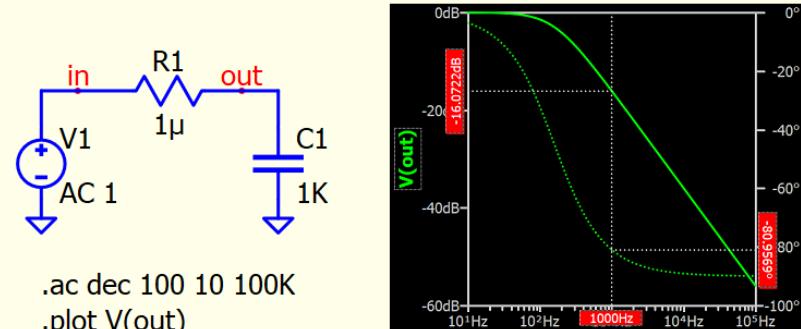
$$20 \log_{10} V(out) = -16.0722 - j12.2729$$

\*\* it log a complex number!

$$|V(out)| = 0.157177 \text{ (in Volt)}$$

$$20 \log_{10} |V(out)| = -16.0722 \text{ (in dB)}$$

$$\angle V(out) = -80.9596^\circ$$



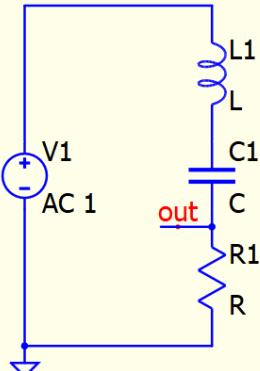
```
.meas v_complex find V(out) when frequency=1000
.meas v_dB_complex find dB(V(out)) when frequency=1000
.meas v_mag find abs(V(out)) when frequency=1000
.meas v_mag_dB dB(abs(V(out))) when frequency=1000
.meas v_phase phase(V(out)) when frequency=1000
```

Output Window

```
.meas v_complex find v(out) when frequency=1000:
( 0.0247045, -0.155223)
.meas v_db_complex find db(v(out)) when frequency=1000:
( -16.0722, -12.2729)
.meas v_mag find abs(v(out)) when frequency=1000:
( 0.157177, 0)
.meas v_mag_db db(abs(v(out))) when frequency=1000:
( -16.0722, 0)
.meas v_phase phase(v(out)) when frequency=1000:
( -80.9569, 0)
```

# .meas in .ac directive – Q-factor example

Qspice : meas - Q of LCR Resonant.qsch



```
.param L=10μ  
.param C=1μ  
.param R=0.2
```

```
.ac dec 1000 1 1G  
.options LISTPARAM  
.plot V(out)
```

Series RLC:

$$Q = \frac{1}{R} \sqrt{\frac{L}{C}}$$

Q formula of Series RLC

```
.param Qcal 1/R*(L/C)**0.5
```

Q calculation from Bandwidth (BW) and Center Frequency (fo) :  $Q = fo/BW$

```
.meas Vmax max mag(V(out))
```

```
.meas fo FIND frequency WHEN mag(V(out))=Vmax
```

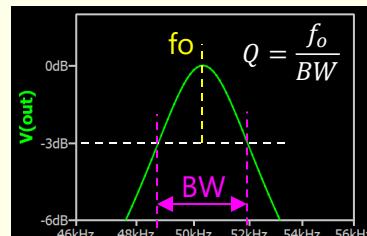
```
.meas fL FIND frequency WHEN mag(V(out))=Vmax/sqrt(2) rise=1
```

```
.meas fH FIND frequency WHEN mag(V(out))=Vmax/sqrt(2) fall=last
```

```
.meas BW fH-fL
```

```
.meas Q fo/BW
```

[1] Add this option to display parameter evaluations result in output window



Output Window

--- Parameter Evaluations ---

TEMP = 27 "CKTTEMP"

L = 10μ "10μ"

C = 1μ "1μ"

R = 200M "0.2"

QCAL = 15.8114 "1/R\*(L/C)\*\*0.5"

C:\Qspice\KSkelvin\01 User Guide and Script\01 Qspice Refer

Total elapsed time: 0.129028 seconds.

In simulation, it has .param calculation results

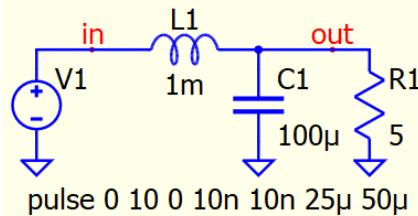
Output Window

```
.meas vmax max mag(v(out)):  
    ( 0.999835, 0) (at Frequency=50350.1)  
.meas fo find frequency WHEN mag(v(out))=vmax:  
    ( 50350.1, 0)  
.meas fl find frequency WHEN mag(v(out))=vmax/sqrt(2) rise=1:  
    ( 48762.3, 0)  
.meas fh find frequency WHEN mag(v(out))=vmax/sqrt(2) fall=last:  
    ( 51946.9, 0)  
.meas bw fh-fl:  
    ( 3184.61, 0)  
.meas q fo/bw:  
    ( 15.8104, 0)
```

In post process, it has .meas calculation results

# .meas in .tran directive – example

Qspice : meas tran demo 01.qsch



```
.tran 20m  
.options MAXSTEP=0.1 $\mu$ 
```

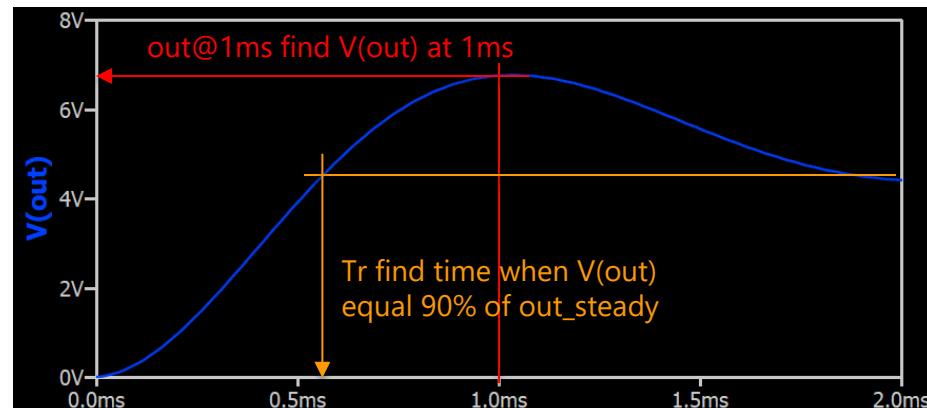
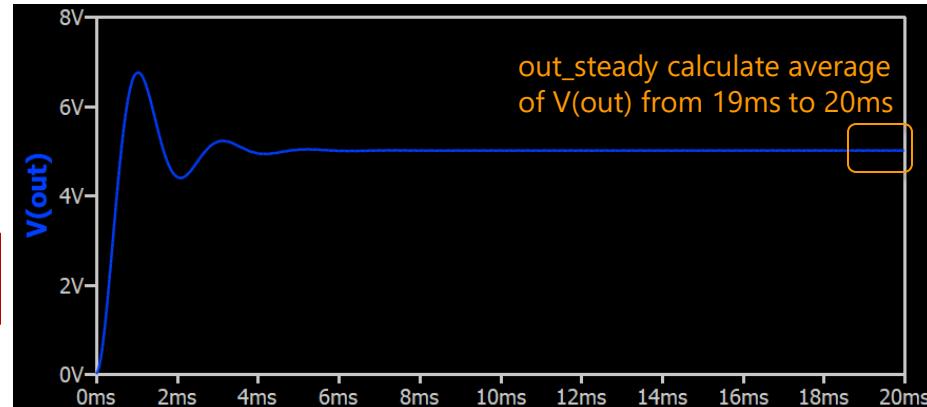
```
.mease out@1ms V(out) at=1m
```

```
.mease out_steady avg V(out) from 19m to 20m  
.mease Tr find Time when V(out)=0.9*out_steady
```

```
.meas Pin avg V(in)*I(V1) from 19m to 20m  
.meas Pout avg V(out)*I(R1) from 19m to 20m  
.meas %error (Pin-Pout)/Pout*100
```

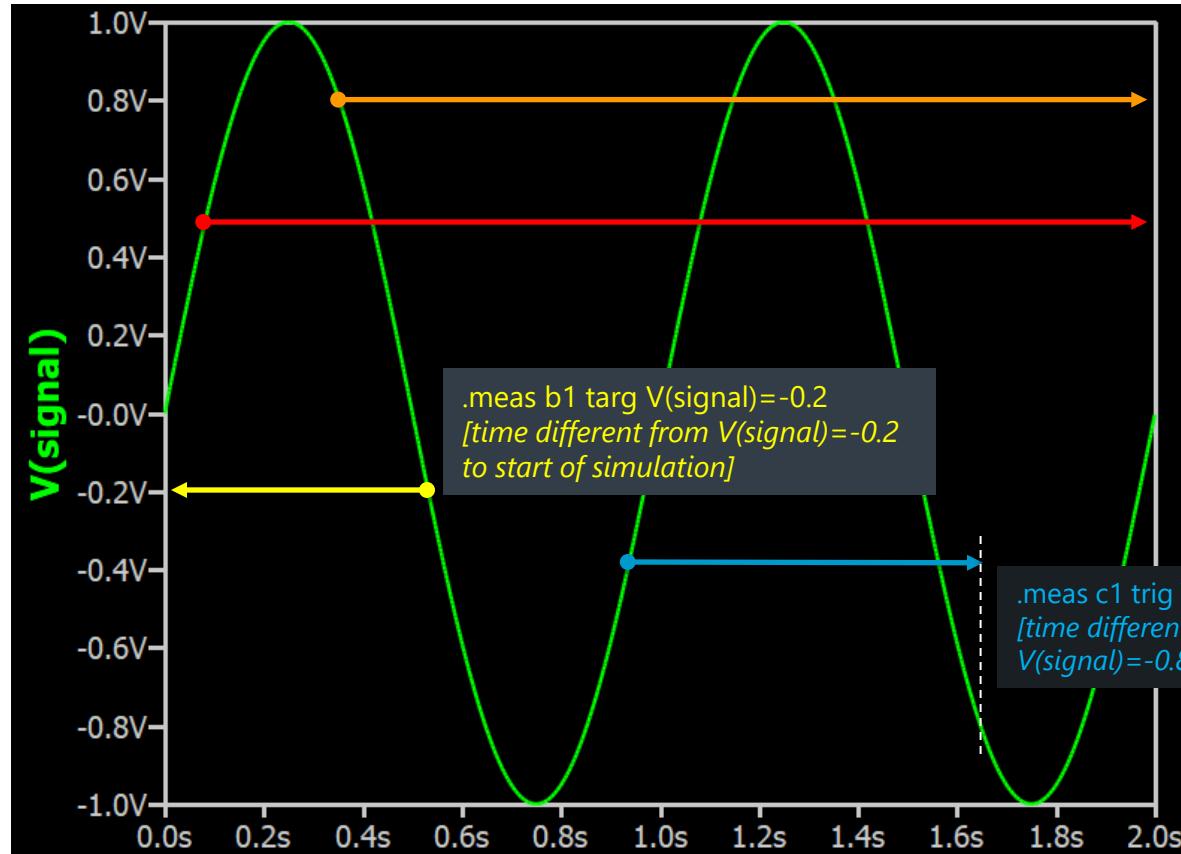
```
.mease out@1ms v(out) at=1m:  
6.74785 0.001  
.mease out steady avg v(out) from 19m to 20m:  
5.00264  
.mease tr find time when v(out)=0.9*out_steady:  
0.000562296 0.000562296  
.meas pin avg v(in)*i(v1) from 19m to 20m:  
5.00497  
.meas pout avg v(out)*i(r1) from 19m to 20m:  
5.00528  
.meas %error (pin-pout)/pout*100:  
-0.0061396 0.02
```

Trick : If at=1m is removed, it return measurement at Tstop



# .meas in .tran directive – trig and targ for time different calculation

Qspice : meas tran demo 02.qsch



`.meas a2 trig V(signal)=0.8 cross=2  
[time different from 2nd crossing  
 $V(\text{signal})=0.8$  to end of simulation]`

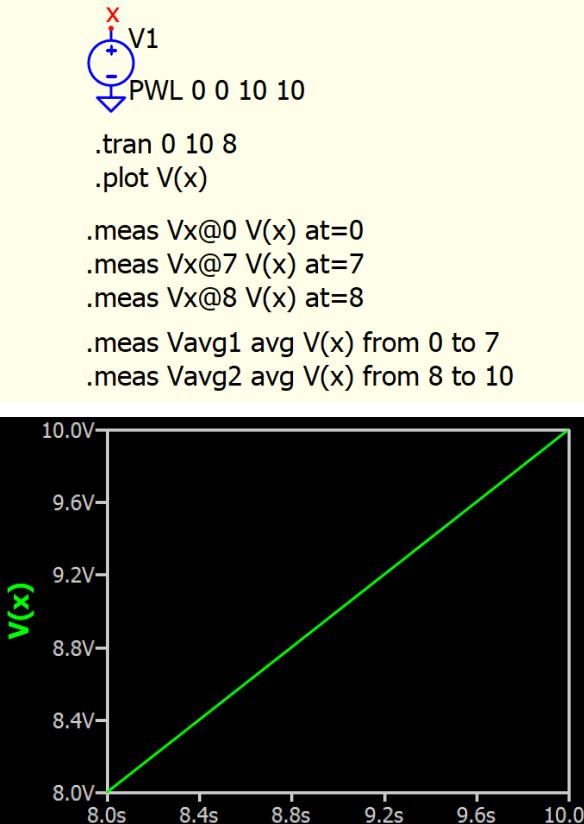
`.meas a1 trig V(signal)=0.5  
[time different from  $V(\text{signal})=0.5$  to end of  
simulation]`

`.meas b1 targ V(signal)=-0.2  
[time different from  $V(\text{signal})=-0.2$  to start of simulation]`

# .meas AT FROM...TO in .tran with Tstart

Qspice : .meas - at from (.tran Tstart).qsch

- .meas with Tstart
  - If .tran with Tstart, data storage will only begin from Tstart
  - Time in .meas is absolute time
    - In this example, .tran only stores data from 8s to 10s
    - .meas at 0s, 7s will give "nan"



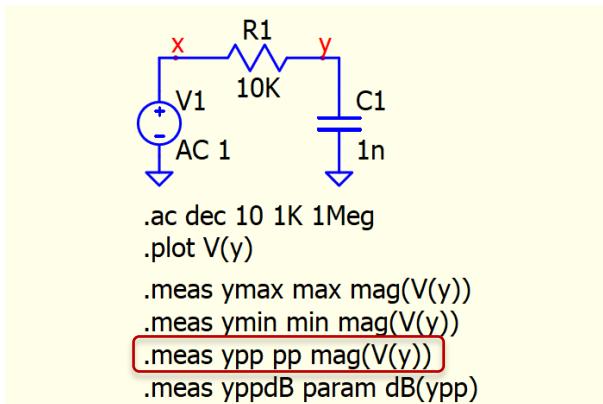
Output Window

```
.meas vx@0 v(x) at=0:
(          nan,      nan)
.meas vx@7 v(x) at=7:
(          nan,      nan)
.meas vx@8 v(x) at=8:
8          8
.meas vavg1 avg v(x) from 0 to 7:
8.00063
.meas vavg2 avg v(x) from 8 to 10:
9
```

# .meas PP in .ac (ac analysis) – Ratio

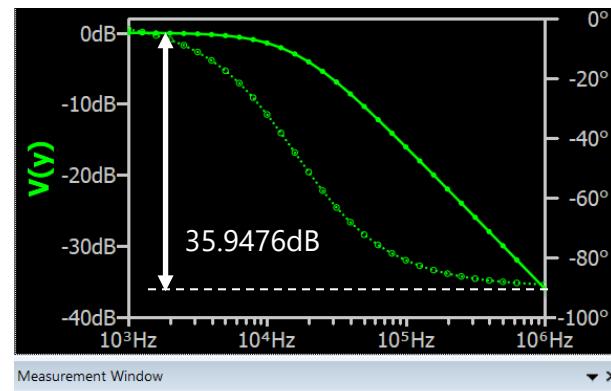
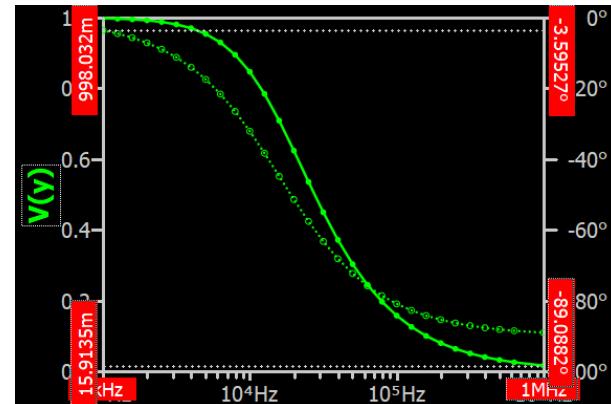
Qspice : .meas - pp (.ac).qsch

- .meas : pp in .ac
  - PP measures
    - In .tran : Peak-to-Peak
    - In .ac : Ratio
  - In this example
    - $V(y)$  max and min magnitude are 0.998032 and 0.0159135
    - .meas PP gives ratio of the magnitude =  $\frac{0.998032}{0.0159135} = 62.7161 = 35.9476\text{dB}$
    - This is peak-to-peak gain in ratio OR gain difference in dB



Output Window

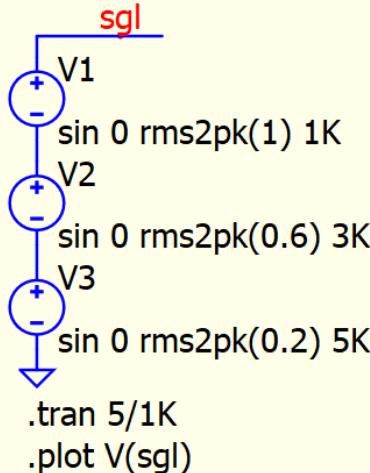
```
.meas ymax max mag(v(y)) :
( 0.998032,
 0)
.meas ymin min mag(v(y)) :
( 0.0159135,
 0)
.meas ypp pp mag(v(y)) :
( 62.7161,
 0)
.meas yppdb param db(ypp) :
( 35.9476,
 0)
```



# .meas in .tran directive – four (fourier component)

Qspice : meas fourier demo 01.qsch

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



THD Total Harmonic Distortion

```
.four 1K V(sgl)
```

Fourier component with .meas

```
.meas xx four 1K V(sgl)
```

```
.meas |xx| abs(xx)
```

THD (.four)

.meas with four

```
.four 1k v(sgl) :|  
Magnitude of Fundamental (RMS) : 0.999922  
Harmonic Frequency Magnitude Phase  
1 1.000e+03 1.000e+00 0.00°  
2 2.000e+03 3.498e-08 105.41°  
3 3.000e+03 5.996e-01 0.00°  
4 4.000e+03 1.713e-08 64.78°  
5 5.000e+03 1.996e-01 -0.00°  
6 6.000e+03 1.341e-08 35.02°  
7 7.000e+03 1.292e-06 -8.50°  
8 8.000e+03 2.028e-08 -30.72°  
9 9.000e+03 2.366e-07 -5.73°
```

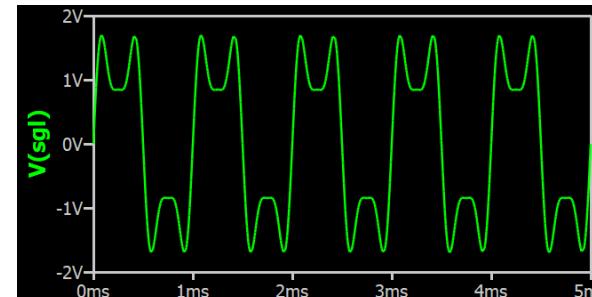
THD = 63.1981% (63.1981%)

```
.meas xx four 1k v(sgl) :  
(1.54886e-07, -0.999999)
```

```
.meas |xx| abs(xx) :  
0.999999 0.005
```

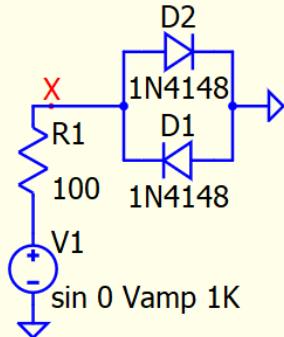
Fourier component is a complex number (re+j\*im)

Magnitude (rms) can be calculated with abs()



# .meas in .tran directive – four (fourier component) [with .step]

Qspice : meas fourier demo 02.qsch



```
.step dec param Vamp 100m 10 2  
.tran 2m  
.plot V(X)
```

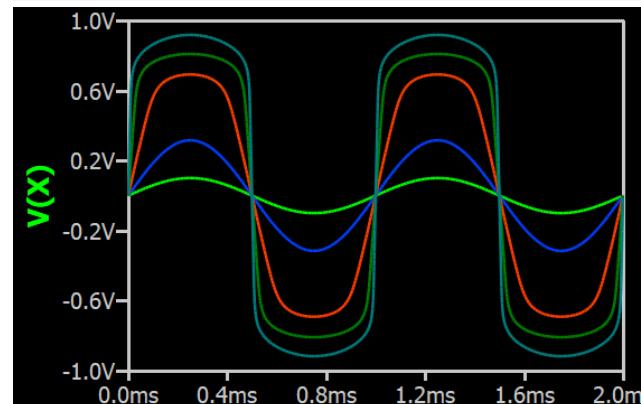
THD Total Harmonic Distortion

.four 1K V(X)

Fourier component with .meas

.meas xx four 1K V(x) format : complex number  
.meas |xx| abs(xx) convert complex to magnitude

```
.meas xx four 1k v(x) :  
0 (-7.73763e-08,-0.0707086)  
1 (-2.48947e-07, -0.223511)  
2 (-3.44697e-06, -0.551818)  
3 (-7.78738e-06, -0.698485)  
4 ( 2.254e-06, -0.797766)  
.meas |xx| abs(xx) :  
0 0.0707086 0.002  
1 0.223511 0.002  
2 0.551818 0.002  
3 0.698485 0.002  
4 0.797766 0.002
```

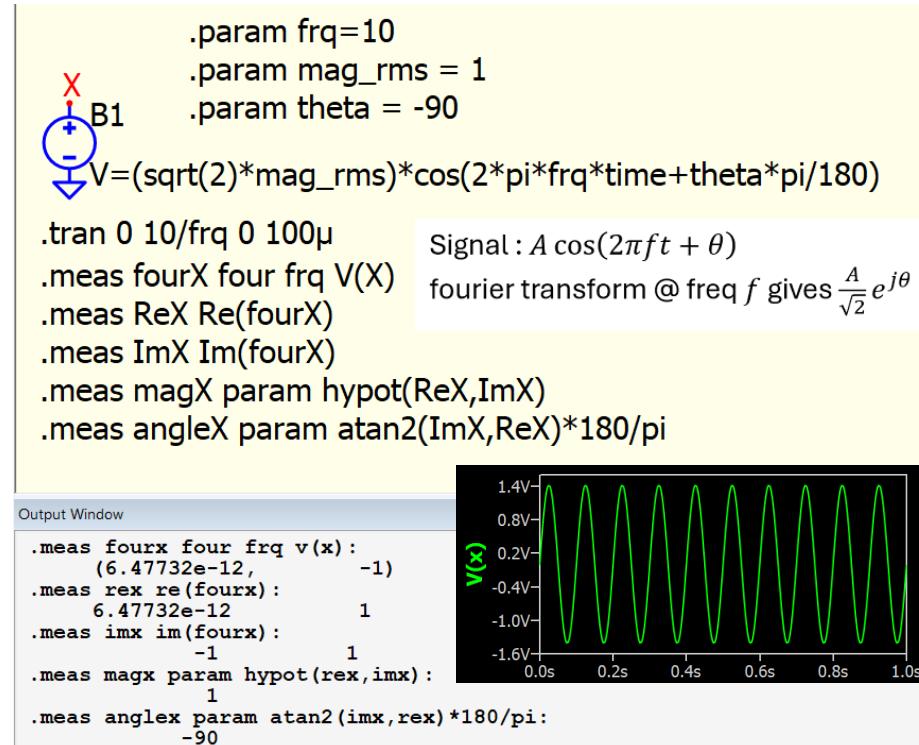


# .meas in .tran directive – four : Mathematic Explanation

Qspice : meas four - math.qsch

## • Mathematic Explanation

- In Qspice, fourier transform calculate signal magnitude (in rms) and phase and output a complex number  $R+jX$  (Re,Im) format
- Fourier transform in Qspice
  - Signal :  $A \cos(2\pi ft + \theta)$
  - Fourier transform @ frequency f gives  $\frac{A}{\sqrt{2}} e^{j\theta}$
  - Therefore, fourier transform can give signal magnitude and phase at tested frequency, and to use consine function to reconstruct the waveform in time domain
- Calculate magnitude and phase from (Re,Im) format
  - Magnitude (rms) :  $\sqrt{R^2 + X^2}$  : hypot(Re,Im)
  - Phase :  $\tan^{-1} \frac{X}{R}$  : atan2(Im,Re)
    - 4 quadrant arc tangent is required to calculate phase in correct quadrant
- In this example,  $A_{rms} = 1$  and  $\theta = -90^\circ$ , which is a sine wave with 0 degree at t=0
  - Therefore, a sine source with 0 degree will give fourier transform with phase -90 degree! As fourier uses consine as reconstruction



# Plot .meas data in Waveform Viewer

Qspice : meas waveform viewer.qsch

Method #1 :

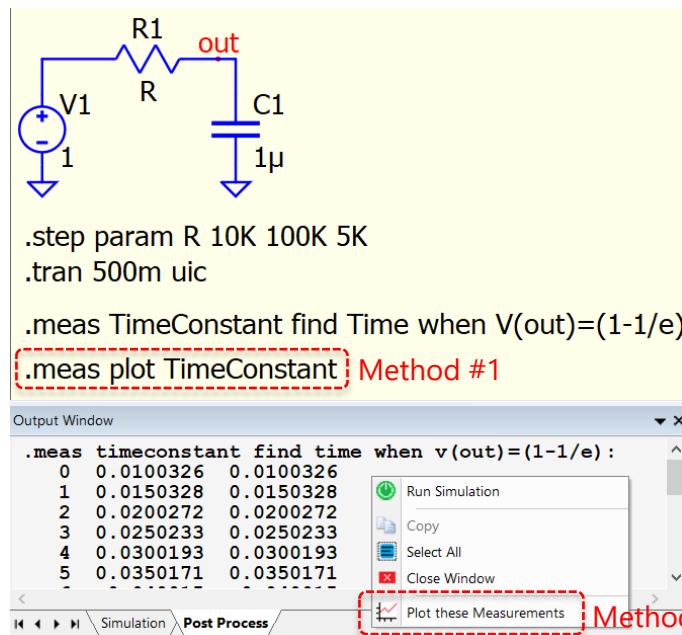
- [1] Add ".meas plot [Name]"

Method #2 :

- [1] Run Simulation

- [2] In Output Window

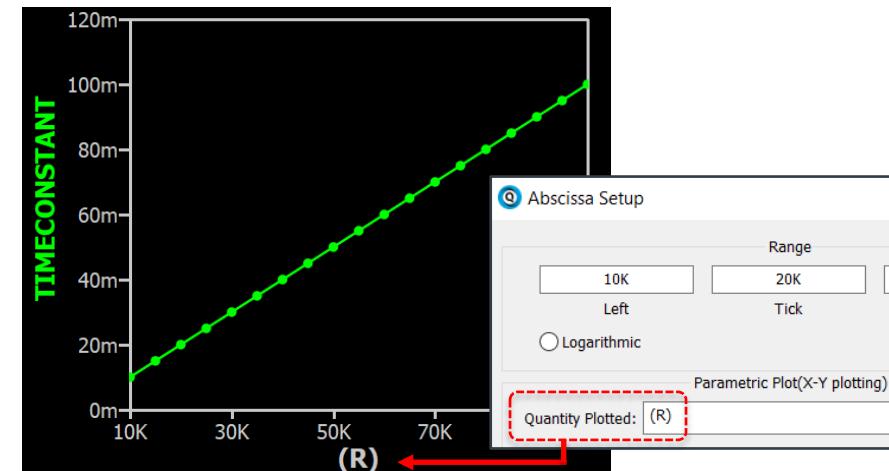
Right click in Post Process > Plot these Measurements



[3] X-axis default is .step parameter

[4] If you want to display X-axis parameter name

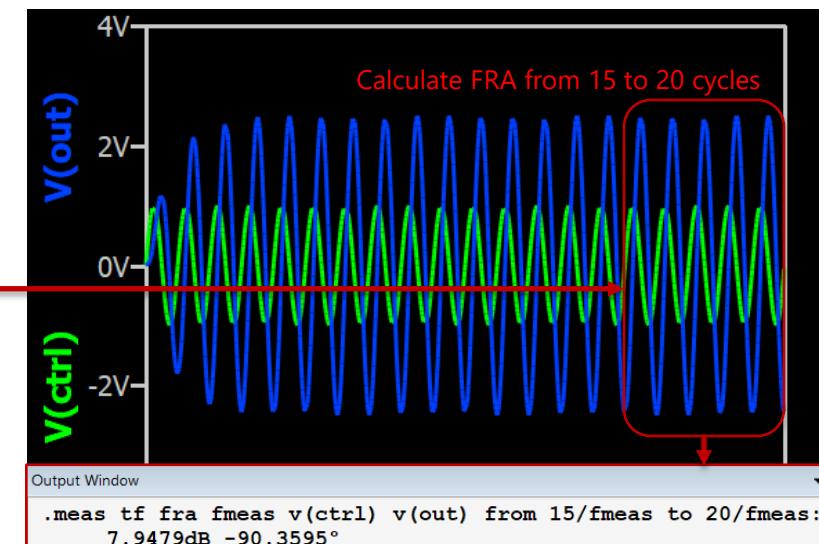
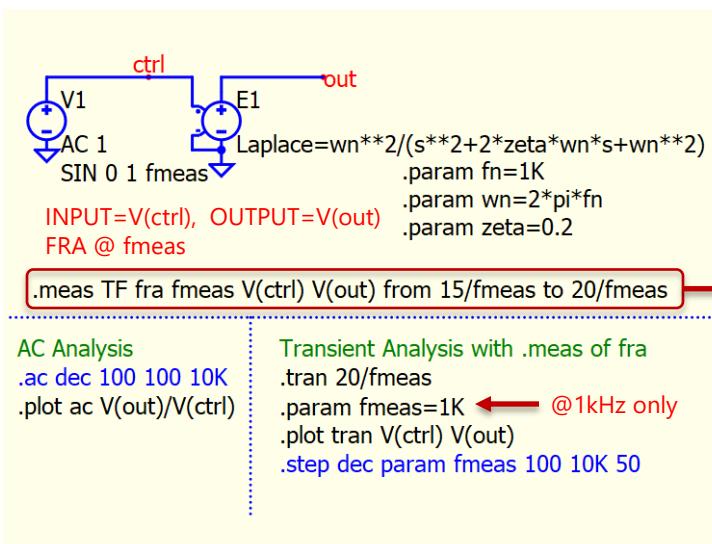
Right click x-axis > add bracket (or curly bracket) to parameter



# .meas – FRA : fourier component between OUTPUT and INPUT

Qspice : meas - fra demo 01.qsch

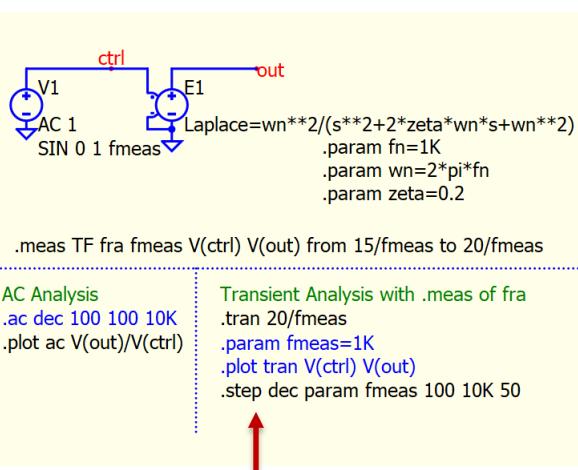
- Syntax : .meas NAME **fra** **FREQ** **INPUT** **OUTPUT** [... range limits ...]
  - Time domain frequency response analysis
  - FRA** : Fourier component of **OUTUT** at **FREQ** divided by the Fourier component of **INPUT** at **FREQ**
  - Range limits can be set with from/to or trig/targ syntax
  - Normalization is to the time domain RMS



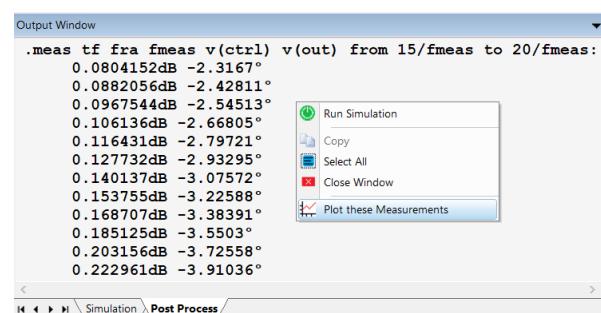
# .meas – FRA : fourier component between OUTPUT and INPUT

Qspice : meas - fra demo 02.qsch

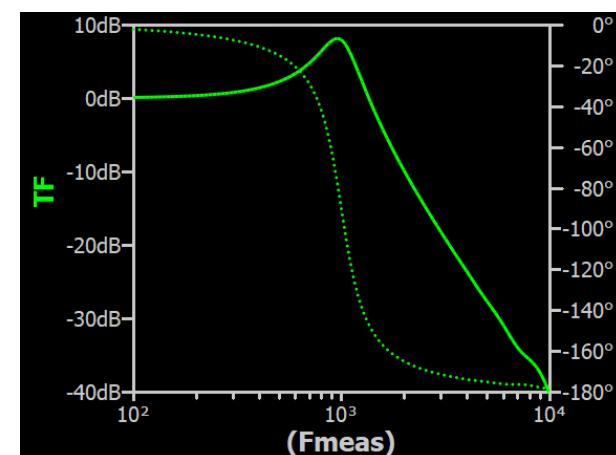
- Frequency response (bode plot) from time domain using FRA
  - In this example, .step is used to sweep FRA frequency
  - Time domain simulation is performed at each FRA frequency. Unlike .ac analysis, which linearizes the circuit around its DC operating point to compute signal magnitude and phase, FRA is performed in the time domain, making it useful for finding the small-signal response of SMPS
  - Qspice includes a demo in File > Open Demo... named FRA SMPS.qsch



fmeas sweep from 100Hz to 10K with 50 points per decade



After .meas is ready in output window  
Right click → Plot these Measurements  
OR add ".meas plot TF" in schematic to get  
this plot automatically after simulation



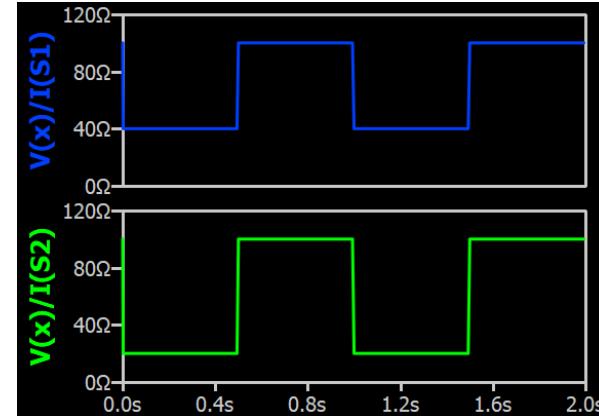
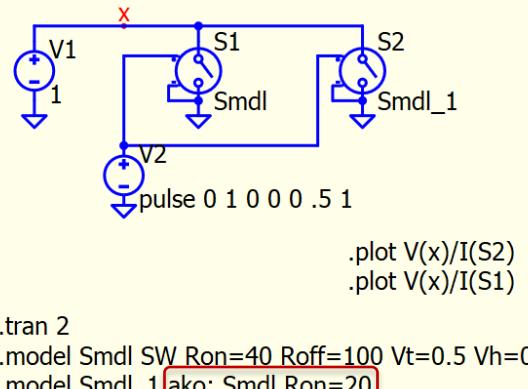
**.model**

**Define Model**

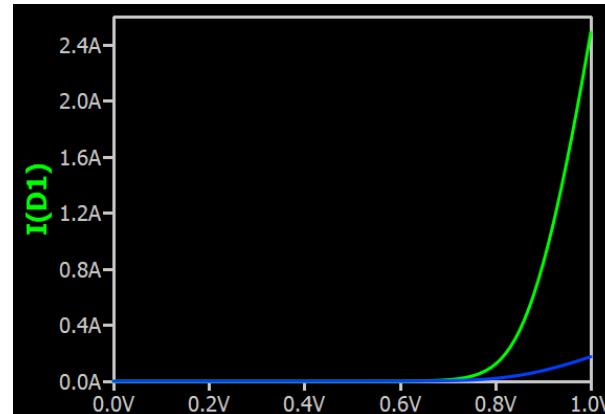
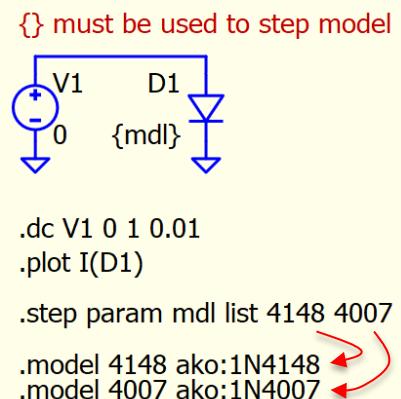
# .model – aka Aliases (A Kind Of)

Qspice : model - aka.qsch

- **ako** (undocumented)
  - Aliases (**A Kind Of**)
  - Syntax : **ako:**
  - Modify parameters of an existing model
- Example
  - Smdl\_1 aliases model Smdl, but only changed Ron from 40 to 20



- Step model with **ako!**
  - With **ako**, it is possible to step a model in simulation
  - **.step** only accept numerical value
  - **.model** use numerical value for **ako** model name
  - Model name of device must be in curly bracket {}

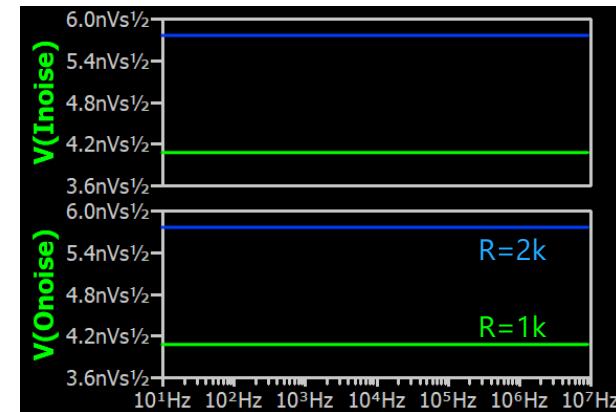
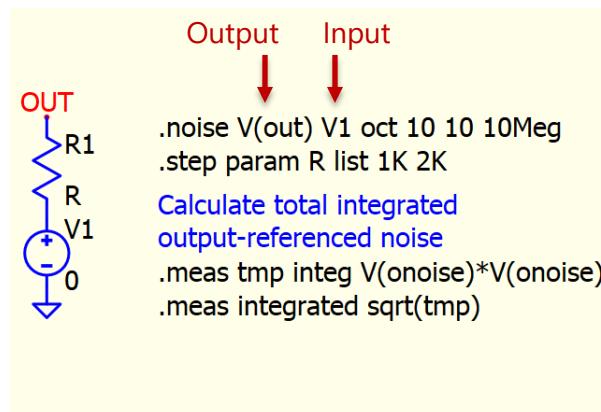


.noise  
**Stochastic Noise**  
Analysis

# .noise Stochastic Noise Analysis

Qspice : noise - basic.qsch

- .noise
  - Stochastic Noise Analysis
  - .noise calculated results
    - Noise spectrum density per unit square root bandwidth ( $V/\sqrt{Hz}$ )
    - Input : Inoise
    - Output : Onoise
  - Notes
    - Noise analysis is performed without the needs of input voltage
    - In output window, it calculates total integrated output and input-referenced noise in rms
    - $\sqrt{\int(V_{noise})^2 df}$



Output Window (Simulation)

```
1 of 2 steps: .step r=1000
Total integrated output-referenced noise: 12.8748 $\mu$ V rms
Total integrated input-referenced noise: 12.8748 $\mu$ V rms
2 of 2 steps: .step r=2000
Total integrated output-referenced noise: 18.2077 $\mu$ V rms
Total integrated input-referenced noise: 18.2077 $\mu$ V rms
```

Output Window (Post-Processing)

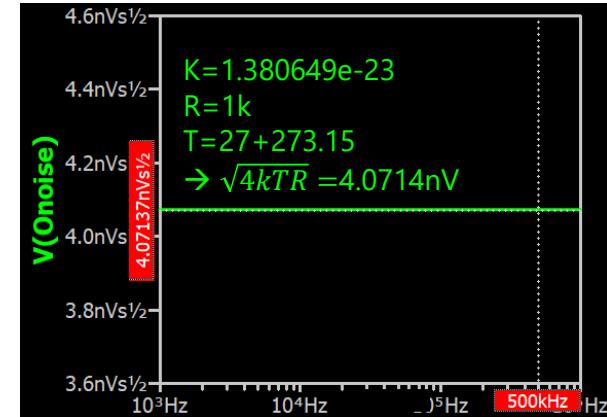
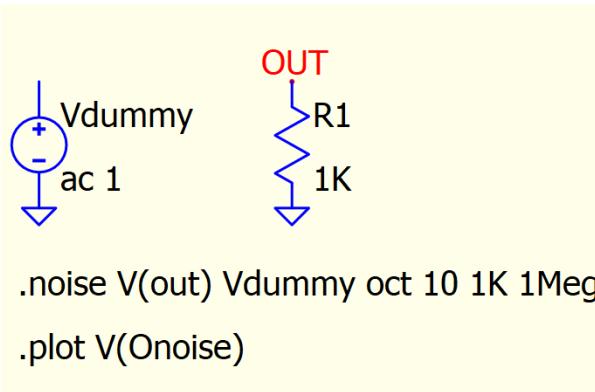
```
.meas tmp integ v(onoise)*v(onoise):
0 1.65761e-10
1 3.31521e-10
.meas integrated sqrt(tmp):
0 1.28748e-05 1e+07
1 1.82077e-05 1e+07
```

# Thermal Noise Formula of Resistor

Qspice : .noise - resistor.qsch

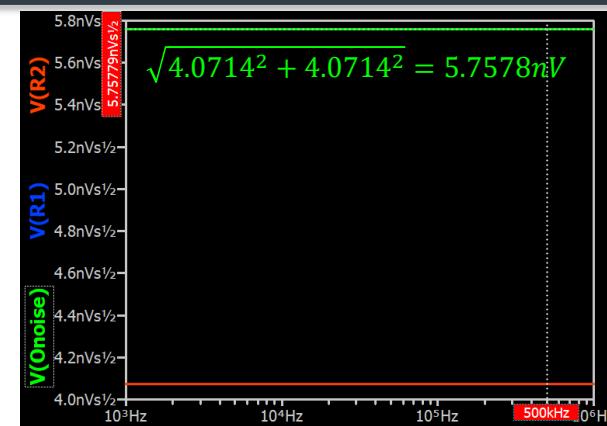
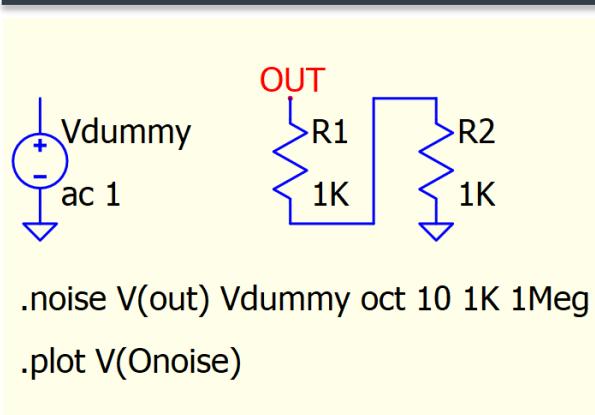
- Thermal noise of R

- $V_{noise} = \sqrt{4kTRB}$ 
  - $V_{noise}$  = Thermal noise in voltage V
  - $k$  = Boltzmann's constant  $1.380649 \times 10^{-23} \text{ J/K}$
  - R = Resistance in Ohms
  - T = Temperature in Kelvin  $0\text{K} = -273.15^\circ\text{C}$
  - $B (\Delta f)$  = Bandwidth of system in Hz
- $\frac{V_{noise}}{\sqrt{1\text{Hz}}} = \sqrt{4kTR}$ 
  - Unit is  $V/\sqrt{\text{Hz}}$  or  $Vs^{1/2}$



- Two Resistors in series

- $V_{n,Total} = \sqrt{V_{n1}^2 + V_{n2}^2}$



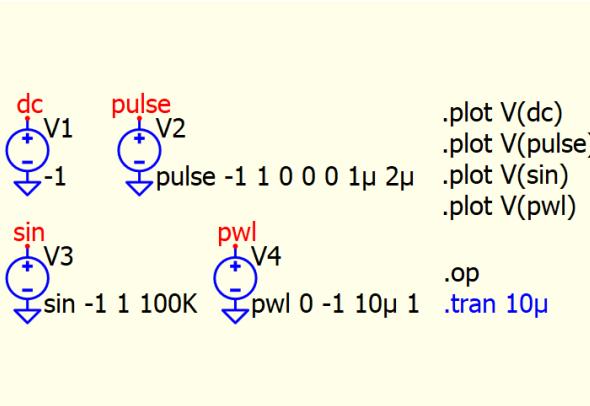
.op

## Bias Point Analysis

# .op – DC Solution in Source

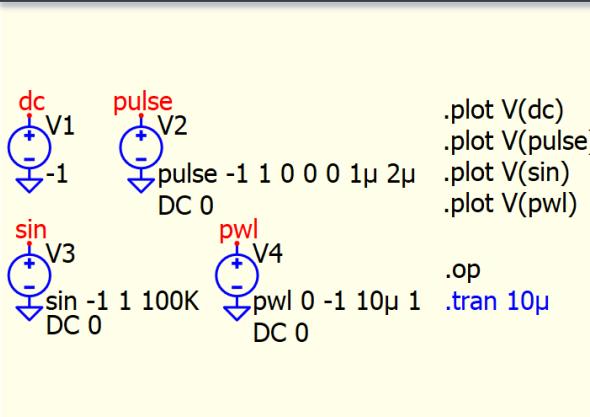
Qspice : .op - source without DC.qsch | .op - source with DC.qsch

- DC Solution in Source
  - The DC solution for source will be its voltage at t=0 unless a DC value is specified



The screenshot shows the Qorvo QSPICE software interface. The operating point table is highlighted with a red box. The table lists various variables and their values: V(dc) = -1, V(pulse) = -1, V(sin) = -1, V(pwl) = -1, I(V4) = 0, I(V3) = 0, I(V2) = 0, I(V1) = 0, P(V4) = -0, P(V3) = -0, P(V2) = -0, P(V1) = -0.

Operating Point	
V(dc)	-1
V(pulse)	-1
V(sin)	-1
V(pwl)	-1
I(V4)	0
I(V3)	0
I(V2)	0
I(V1)	0
P(V4)	-0
P(V3)	-0
P(V2)	-0
P(V1)	-0



The screenshot shows the Qorvo QSPICE software interface. The operating point table is highlighted with a red bracket. The table lists various variables and their values: V(dc) = -1, V(pulse) = 0, V(sin) = 0, V(pwl) = 0, I(V4) = 0, I(V3) = 0, I(V2) = 0, I(V1) = 0, P(V4) = 0, P(V3) = 0, P(V2) = 0, P(V1) = -0.

Operating Point	
V(dc)	-1
V(pulse)	0
V(sin)	0
V(pwl)	0
I(V4)	0
I(V3)	0
I(V2)	0
I(V1)	0
P(V4)	0
P(V3)	0
P(V2)	0
P(V1)	-0

**.option / .options**  
Set Simulator Options

# Set Simulator Options

## Set Simulator Options

Syntax: .option NAME1=VALUE1 [NAME2=VALUE2 [...]]

### Recognized Options

Name	Description	Default
ABSTOL	Absolute error tolerance	1e-12A
ACCT	Print accounting information	(not set)
ASCII	ASCII .qraw file	(not set)
BINARY	Override command line switch to use ASCII .qraw file	(not set)
BODEAMPFREQ	Frequency with the minimum perturbation amplitude. Set to 0. for constant amplitude.	(not set)
BODEHIPOW	Controls perturbation amplitude for above BODEAMPFREQ by pow (freq/BODEAMPFREQ, BODEHIPOW)	1.
BODEINPUT <sup>1</sup>	Override input node for transfer function computation(aka BODEIN)	auto
BODELOPOW	Controls perturbation amplitude for below BODEAMPFREQ by pow (freq/BODEAMPFREQ, BODELOPOW)	1.
BODEPERIODS	Maximum number of periods to include in deconvolution	20
BODEREF	Reference node to use for Frequency Response Analysis	Node 0 (global ground)
BODEOUTPUT <sup>1</sup>	Override output node for transfer function computation(aka BODEOUT)	auto
BODETOL	A Frequency Response Analysis relative tolerance	10.
CAPOP	0: Use model value 1: Use Meyer, >1 Use BSIM1	0
CHGTOL	Charge error tolerance	1e-14C
CSHUNT	Capacitance added from every node to ground(aka CMIN)	0F
DEFAD	Default MOSFET area of drain	0m <sup>2</sup>
DEFAS	Default MOSFET area of source	0m <sup>2</sup>
DEFL	Default MOSFET length	10μm
DEFW	Default MOSFET width	10μm
FEATHER	Trap integration damping factor	0
GMIN	Minimum conductance	1e-12Ω
GMINSTEPS <sup>2</sup>	Number of Gmin steps	10
GSHUNT	Conductance added from every node to ground	0Ω
ITL1	DC iteration limit	100
ITL2	DC transfer curve iteration limit	50
ITL4	Transient analysis iteration limit	10
KEEPINFO	Record operating point for small-signal analysis	(not set)
LAUNCHQUX <sup>3</sup>	Open the .qraw file in the waveform viewer after the simulation	(not set)

Bode

LAUNCHQUX	Open the .qraw file in the waveform viewer after the simulation	(not set)
LIST <sup>4</sup>	Print an expanded netlist	(not set)
LISTPARAM	Print a list of the evaluated parameters(also sets SAVEPARAM)	(not set)
LFT	Frequency at which default inductor damping has parallel resistance equal to reactance.	(not set)
MAXORD	Maximum integration order	2
MAXSTEP	Maximum timestep size for .bode and .tran	infinite
MAX1STSTEP	Maximum timestep size the very first timestep for a .tran	100ns
METHOD	Integration method(trap or Gear)	trapezoidal
MINBREAK <sup>5</sup>	Minimum time between breakpoints	0s
NOOPTITER	Go directly to Gmin stepping	(not set)
NUMDGT	Number of significant digits in an ASCII .qraw file	15
PIVREL	Minimum relative matrix pivot	1e-3
PIVTOL	Minimum absolute matrix pivot	1e-13
RELTOL	Relative error tolerance	0.1%
RIC <sup>6</sup>	Impedance of source asserting initial conditions	1mΩ
SAVEPARAM	Include evaluated user-defined parameters as waveform data	(not set)
SAVEPOWERS <sup>7</sup>	Compute and save the dissipation of components	(not set)
SEED <sup>8</sup>	Initialize the random number generator used in .param statements	
SEEDCLOCK	Initialize the random number generator with a 10MHz clock and the process ID number(aka SEEDCLK).	(not set)
SRCSTEPS <sup>2</sup>	Number of source steps(aka ITL6)	10
TEMP	Operating temperature	27°C
TNOM	Nominal temperature(aka TREF)	27°C
TRTOL	Truncation error overestimation factor	2.5
TRTOL2	Another dimensionless truncation error guidance	1e-8
TRYTOCOMPACT	Try compaction for LTRA lines	(not set)
VNTOL	Voltage error tolerance	1μV

<sup>1]</sup> If a resistor is used to indicate where to insert the perturbation, the resistive divider's contribution is excluded.

<sup>2]</sup> Since an adaptive step size algorithms are used, the value of GMINSTEPS or SRCSTEPS is irrelevant unless set to zero, which means don't try the stepping algorithm.

<sup>3]</sup> Useful when running simulations from the command line. Don't use it if QSPICE64.exe or QSPICE80.exe are launched from the GUI.

<sup>4]</sup> Solely for internal diagnostic purposes. Probably not what you're looking for.

<sup>5]</sup> MINBREAK is automatically computed if left zero.

<sup>6]</sup> Inductor currents are asserted with the compliance of 1e9 \* RIC.

<sup>7]</sup> Computes the true power dissipation while ignoring displacement currents. Implemented for BJTs, Capacitors, Diodes, Inductors, JFETs, MOSFET level 1, MOSFET level 2010 and VDMOS.

<sup>8]</sup> Used in .param functions Random() and Gauss(double sigma).

# One Page Summary for Convergence Related Option

- Convergence ( Voltage / Current )
  - ABSTOL : Absolute Current Tolerance [1pA]
  - VNTOL : Absolute Voltage Tolerance [1uV]
  - RELTOL : Relative Error Tolerance [0.1% = 0.001]
  - CHGTOL : Charge Error Tolerance [1e-14C]
- Convergence ( Impedance Insertion )
  - Gshunt : Conductance (G) added between each node and ground [0Ω]
  - Gmin : Conductance (G) added between PN junction [1e-12Ω]
  - Cshunt (Cmin) : Capacitance (C) added between each node and ground [0F]
- DC Analysis ONLY
  - NoOpiter : Skip Direct Newton Iteration [not set]
  - GminSteps : Enable/Disable Adaptive Gmin Stepping [10]
  - SrcSteps (ITL6) : Enable/Disable Adaptive Source Stepping [10]
  - ITL1 : DC iteration limit [100]
  - ITL2 : DC transfer curve iteration limit [50]
- Transient Analysis ONLY
  - ITL4 : Transient analysis iteration limit [10]
  - TRTOL : Truncation error overestimation factor – affect timestep accuracy [2.5]
  - Maxstep : Maximum timestep size [not set]
  - Method : Integration method ( Trapezoidal / Gear ) [Trap]
  - Feather : Damping Factor for Trapezoidal Integration [0]
  - MaxOrd : Integration method ( 1 : Backward Euler , 2 : Trapezoidal / Gear ) [2]
- Math Precision
  - Fastmath : Preferred 64 bits or 80 bits solver ( 1 : QSPICE64.exe or 0 : QSPICE80.exe ) [none]

# .option – Reltol, Abstol, Vntol (Convergence Criteria Parameters)

\*\* this slide may be incorrect in describing Qspice, use as reference only

- Solving Nonlinear Equation in SPICE : Newton-Raphson Method  
Newton-Raphson algorithm (aka Newton's method)
  - Initial guess on the solution  $v^{<0>}$
  - Linearizes around the initial guess with Jacobian  $J(v^{<0>}) = \frac{d}{dv} f(v^{<0>})$
  - Solves the resulting system of linear equations  $v^{<k>}$
  - Re-linearize around the new point  $J(v^{<k>}) = \frac{d}{dv} f(v^{<k>})$
  - Repeats until the solution **converges (Residue and Update Criterion)**
- Residue Criterion (1<sup>st</sup> Convergence criteria)
  - KCL should be satisfied to a certain degree (ideally, the sum of currents at each node equal zero)
  - In practice :  $\sum I(node_i) < \text{reitol} \times |I_{max}| + \text{abstol}$ 
    - $I_{max}$  is the maximum current in this circuit simulation
  - **Relative error tolerance** : default **reitol** = 0.1% = 0.001
  - **Absolute current error tolerance** : default **abstol** = 1pA (eliminate the need to absolute zero)
- Update Criterion (2<sup>nd</sup> Convergence criteria)
  - The difference in error between the results of the last two iterations is small
  - In practice :  $|v^{<k>} - v^{<k-1>}| < \text{reitol} \times \max(v^{<k>}, v^{<k-1>}) + \text{vntol}$
  - **Absolute voltage error tolerance** : default **vntol** = 1uA

# .option – NoOpIter, GminSteps, SrcSteps (DC Solution Strategy)

- NoOpIter, GminSteps, SrcSteps
  - NoOpIter : Go directly to Gmin Stepping (**Default NoOpIter=(not set)**)
  - GminSteps : Number of Gmin steps (**Default GminSteps=10**)
  - SrcSteps : Number of Source steps (**Default SrcSteps=10**)
  - These options are related to finding the DC operation point
- .OP – Find the DC Operation Point
  - DC operation point (.op) is usually performed as part of another analysis (e.g. .ac, .tran [without UIC]) in order to find the operating point of the circuit
  - But there is no guarantee that the operating point of a general nonlinear circuit can be found with successive linear approximations as is done in Newton-Raphson iteration
  - Method to find an operating point is in this sequence

Method	Directive to Disable
1. Direct Newton Iteration	.option NoOpIter
2. Adaptive Gmin Stepping	.option GminSteps=0
3. Adaptive Source Stepping	.option SrcSteps=0
4. Pseudo Transient	

# .option – NoOPIter, GminSteps, SrcSteps (DC Solution Strategy)

- Example in Output Window
  - If without any warning message, represent Direct Newton Iteration can successfully find operation point

Output Window

```
C:\KSKelvinQspice\01 User Guide and Script\03 Command Re
Starting Gmin stepping. ← This represent Direct Newton Iteration fails and
Warning: Gmin stepping failed. ← change to Gmin stepping
Starting source stepping. ← This represent Gmin stepping fails and change
Warning: Source stepping failed at 0.106199(1.10543e-15) ← to Source stepping
Starting pseudo transient analysis. ← This represent Source stepping fails and change
Accepting Pseudo Transient analysis solution. ← to Pseudo Transient analysis
```

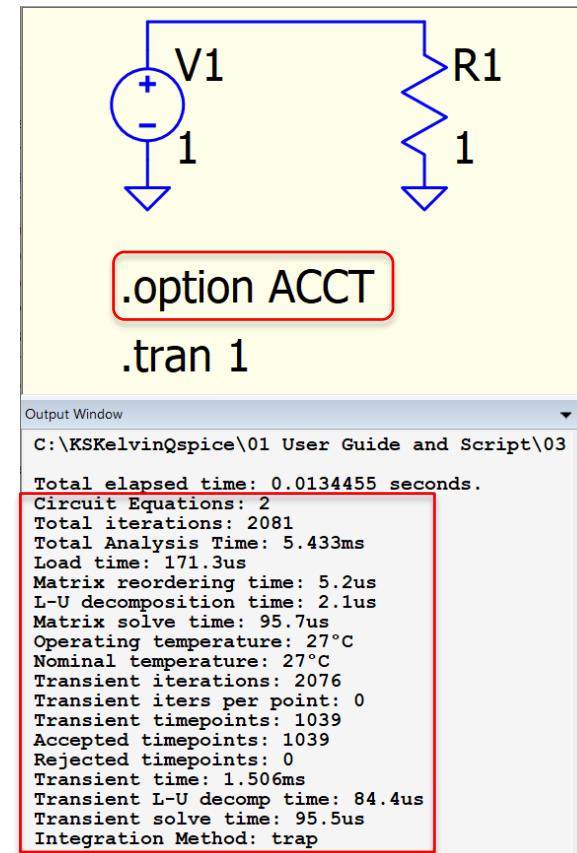
# .option – NoOPIter, GminSteps, SrcSteps (DC Solution Strategy)

- Quote from Mike Engelhardt about Gmin Stepping and Source Stepping
  - **Gmin Stepping**
    - Gmin stepping is a DC solution strategy and is poorly named, as it has nothing to do with Gmin (minimum conductivity)
    - If the solver can't find the solution of the original circuit with Direct Newton Iteration, start with a related circuit that it can find the solution to. The circuit is the original but with a conductivity from every node to ground. Like an Ohm(or even less). Odds are, the solver can find the solution of the circuit if every node is shorted to ground. Give the solution for the circuit with every node shorted to ground, use that solution as a starting point for the Newton iteration when instead of 1 Ohm to ground but, say 1.1 Ohm to ground. If that succeeds, keep decreasing the conductivity until it's some very small number, like GMIN. Then remove the conductivity entirely and see if the solution can be found with Newton iteration using the prior solution as a starting point.
    - Refinements to the algorithm entail (i) first increasing the initial conductively until a first solution can be found and (ii) adapting the change in conductively between steps to ensure that a solution is kept as the conductivity is reduced
    - GMIN stepping is extremely effective for ICs or most any physical circuit
  - **Source Stepping**
    - GMIN stepping is not so good for non-physical macromodels like the TI OpAmp models, Source stepping is often better
    - Source stepping starts with the circuit turned off and then gradually increases the supply voltages
    - There are three types of source stepping
      - DC source stepping - does not take any hints from the reactance in the solution
      - Pseudo transient - take hints from the reactance in the solution
      - .tran with the sources written as PWLs starting a 0

# .option – ACCT (Print accounting information)

Qspice : option - ACCT.qsch

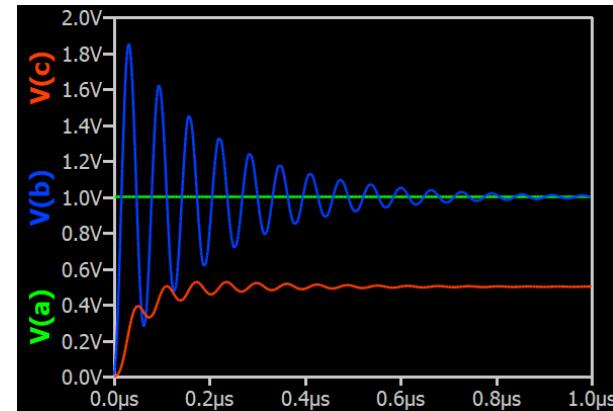
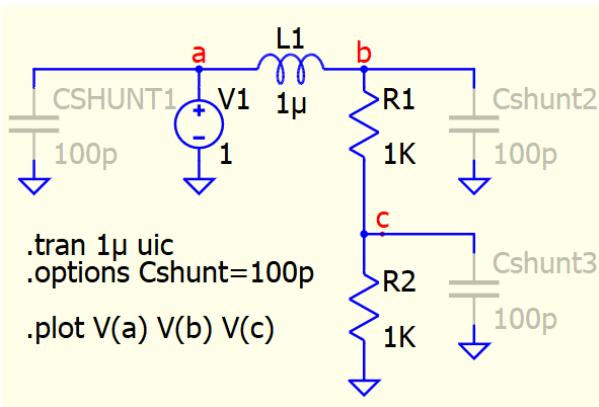
- ACCT
  - Print accounting information
  - **Default ACCT : Not Set**



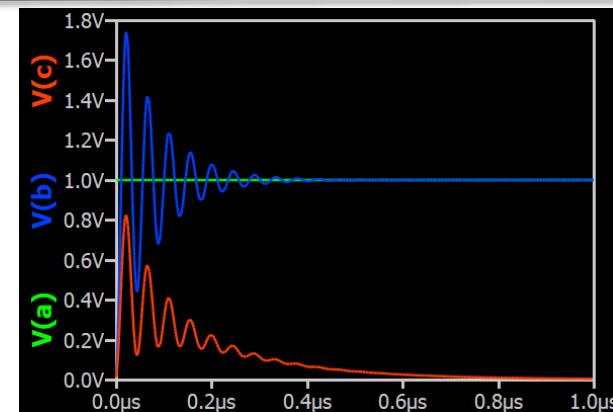
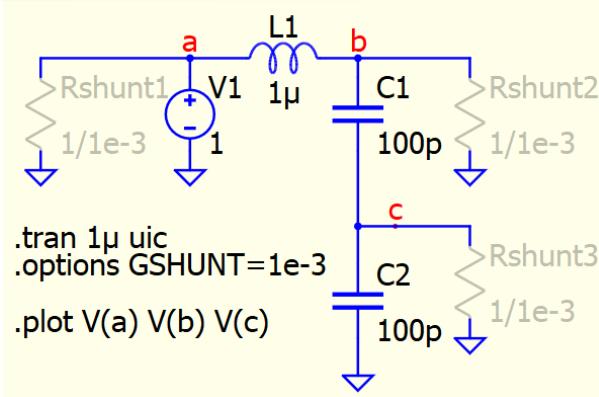
# .option – Cshunt and Gshunt (Capacitance and Conductance Shunt)

Qspice : option - CSHUNT.qsch ; option - GSHUNT.qsch

- Cshunt
  - Capacitance added from every node to ground (aka CMIN)
  - **Default CSHUNT=0F**
  - Example to explain
    - Cshunt is equivalent to add Cshunt1/2/3 in node a/b/c



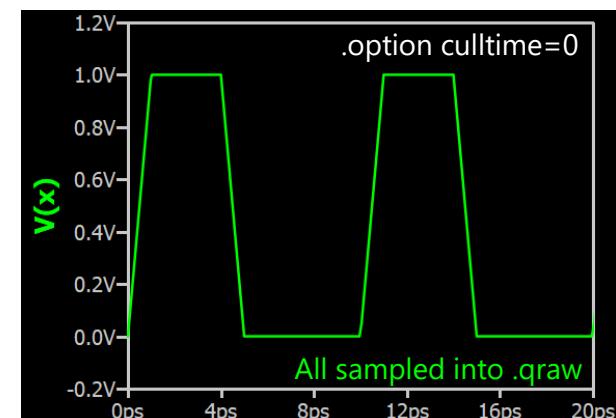
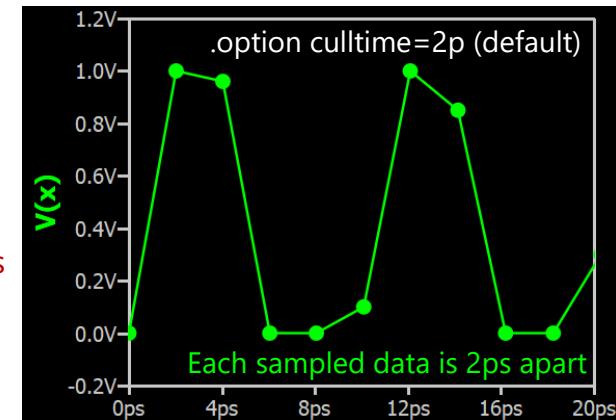
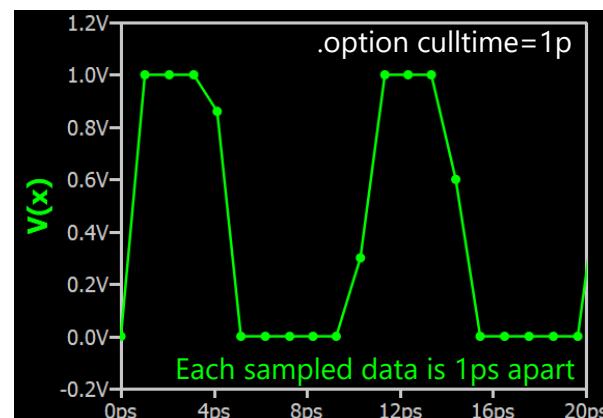
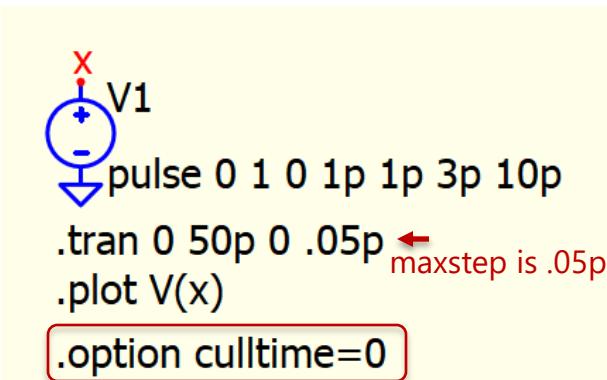
- Gshunt
  - Conductance added from every node to ground
  - **Default GSHUNT=0Ω**
  - Example to explain
    - Gshunt is equivalent to add Rshunt1/2/3 =  $\frac{1}{GSHUNT}$  in node a/b/c



# .option – CullTime (Minimum time to elapse for data saving)

Qspice : option - CullTime.qsch

- CullTime
  - Minimum time to elapse before dumping transient analysis data to disk
  - **Default CULLTIME=2ps**
  - In default, Qspice only updates the .qraw file if at least 2ps have passed since the last update
  - This option is useful for simulating picosecond-level applications
  - \*\* CullTime doesn't affect actual simulation step size, but only the minimum sampling time between two samples that are written into the .qraw data file

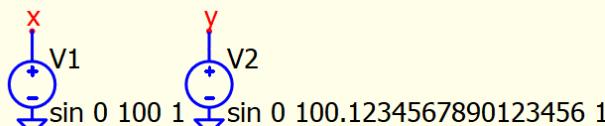


# .option – MEASDGT (number of significant figure in .meas)

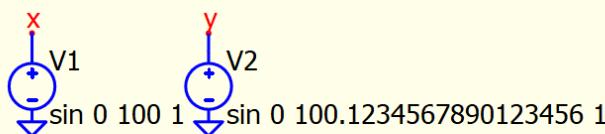
Qspice : .option - measdgt.qsch

- MEASDGT

- [Undocumented]
- Number of Significant figure (digits) in .meas output
- **Default measdgt=6**
- measdgt range from 3 to 15



```
.tran 0 1 0 1/100e4
.meas Xpeak min V(x)
.meas Ypeak min V(y)
.meas Ydiff param Ypeak-Xpeak
.option measdgt=3 ; min digits
.option measdgt=16 ; max digits
```



```
.tran 0 1 0 1/100e4
.meas Xpeak min V(x)
.meas Ypeak min V(y)
.meas Ydiff param Ypeak-Xpeak
.option measdgt=3 ; min digits
.option measdgt=15 ; max digits
```

Output Window

```
.meas xpeak min v(x) :
-100 (at Time=0.75)
.meas ypeak min v(y) :
-100 (at Time=0.75)
.meas ydiff param ypeak-xpeak:
-0.123
```

Simulation Post Process

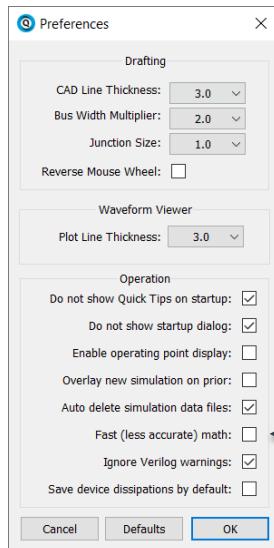
Output Window

```
.meas xpeak min v(x) :
-99.999999999989 (at Time=0.75000002400)
.meas ypeak min v(y) :
-100.123456789011 (at Time=0.75000002400)
.meas ydiff param ypeak-xpeak:
-0.123456789012351
```

Simulation Post Process

# .option – FastMath

- FastMath
  - Annotates the simulation with the preferred solver. Set to true for false
  - **.option fastmath=1 / .option fastmath=true** : use **Qspice64.exe** as solver
  - **.option fastmath=0 / .option fastmath=false** : use **Qspice80.exe** as solver
  - User can check which solver is used by reviewing .qraw line#8 with a text editor



This option can be found in Preferences  
(If .option fastmath is not in schematic netlist, simulator follows Preferences selection)

### **Fast (less accurate) math :**

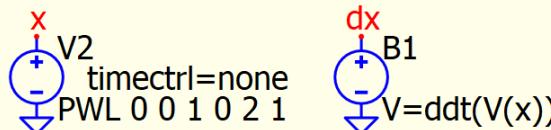
- Clicked : Use Qspice64.exe
- Unclicked : Use Qspice 80.exe

# .option – Feather (Trap Integration Damping Factor)

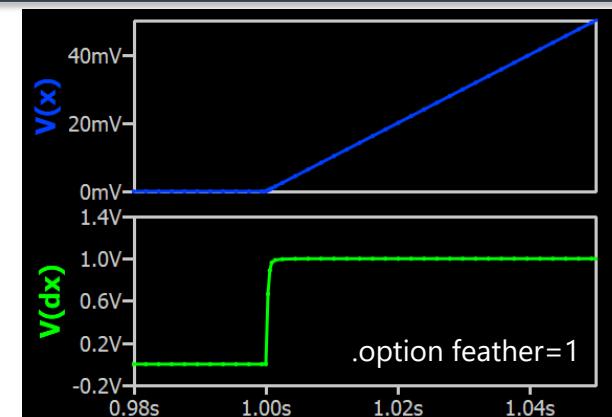
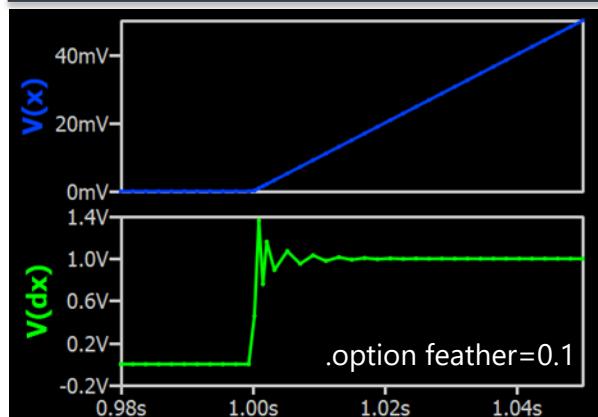
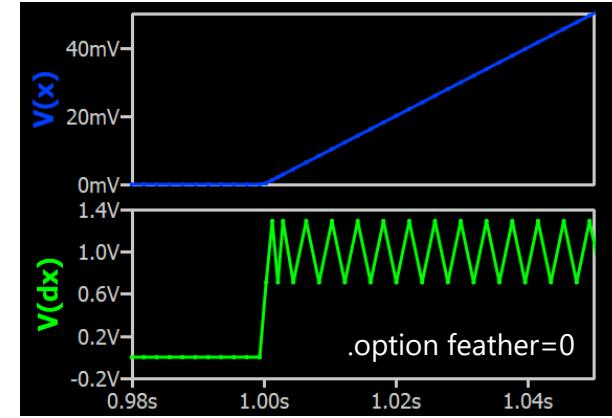
Qspice : option - Feather.qsch

- Feather
  - Trap Integration Damping Factor
  - **Default Feather=0**
  - If trapezoidal ringing is observed when using trapezoidal integration, the damping factor can be employed to reduce it
  - For more information about trapezoidal ringing, refer to section **.option – Method : Trapezoidal Ringing**

## Trap Ringing Simulation Setup



```
.tran 0 2  
.plot V(dx)  
.plot V(x)  
.option method=trap feather=0
```



# .option – Feather (Trap Integration Damping Factor)

- Caution about .option Feather



Engelhardt

Jul 2023

FEATHER is an experimental parameter. It can be used to duplicate the de-tuned trap integration of HSPICE. Its use is not recommended.

As far as trap ringing, I realize it's disconcerting, but in a sense it's giving the right answer: The area under each trapezoid is correct, so one knows that the differential equations are correctly integrated. It can be reduced by either (i) using ".options method=Gear" (ii) stipulating a lower trtol, or (iii) stipulating a smaller maximum timestep. Gear is not recommended because it adds a substantial artificial damping to the circuit. I know of two cases where the use of Gear integration let an IC designer believe his circuit was stable until silicon said different and a turning Gear off confirmed.

Unlike some prior art, I don't smooth trap ringing out it out at all because one needs to know what the gates and flop truly see.

# .option – Gmin (Minimum conductance)

Qspice : option - Gmin.qsch

- Gmin
  - Minimum conductance
    - Default Gmin=1e-12 $\Omega$
    - Minimum conductivity that is added in parallel to every PN junction (diode, JFET, bipolar, MOSFET substrate diodes)
  - Reason : As double precision math isn't accurate enough to find the bias point of two diode in series when reversed bias
- Example
  - In 1<sup>st</sup> example, without Gmin, Gmin stepping and Source stepping both failed to calculate solution of .op

```
.option gmin=0 ← No gmin is set to 0
.option noopter gminsteps=0 ; disable Gmin stepping
.option srcteps=0 ; disable Source stepping

Output Window
C:\KSKelvinQspice\01 User Guide and Script\03 Command Re
Starting Gmin stepping.
Warning: Gmin stepping failed. → Gmin stepping fails to have solution
Starting source stepping.
Warning: Source stepping failed at 0.106199(1.10543e-15). → Source stepping fails to have solution
Accepting Pseudo Transient analysis solution. → Pseudo transient pass
```

```
.option gmin=1e-12 ← With default Gmin=1e-12
.option noopter gminsteps=0 ; disable Gmin stepping
.option srcteps=0 ; disable Source stepping

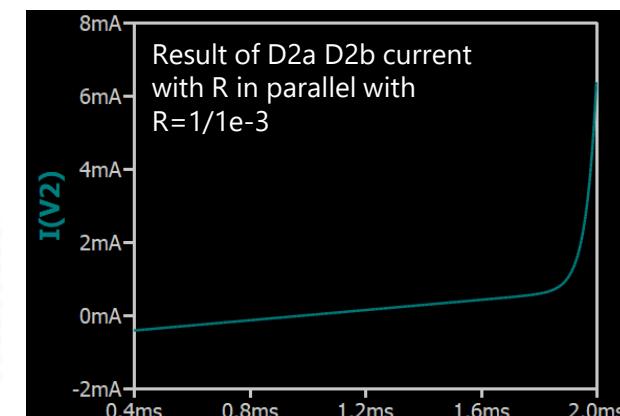
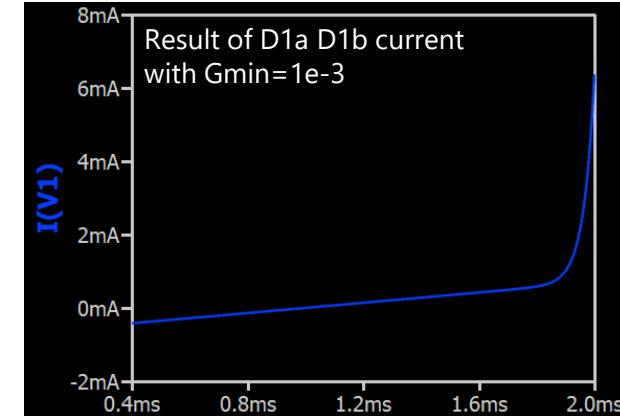
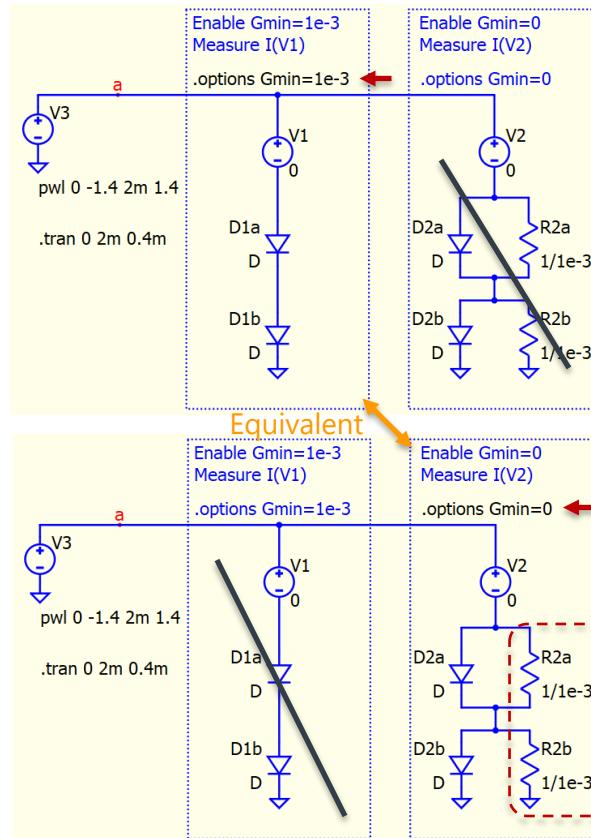
Output Window
C:\KSKelvinQspice\01 User Guide and Script\03 Command Re
Total elapsed time: 0.0126863 seconds.
```



# .option – Gmin (Minimum conductance)

Qspice : option - Gmin Diode.qsch

- Effect of Gmin
  - Examples to demonstrate Gmin is equivalent to add shunt conductance for every PNjunction
  - First example uses Gmin=1e-3 and measure I(V1) profile of D1a/D2a
  - Second example force Gmin=0 (no effect of Gmin) and added R2a = R2b =  $\frac{1}{1e-3}$ , and measure I(V2) profile of D2a and D2b

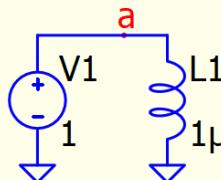


# .option – Gmin (Minimum conductance)

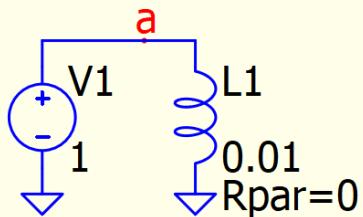
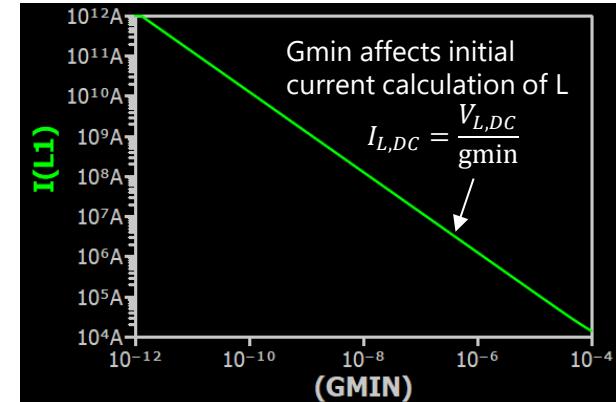
Qspice : option - Gmin L (.op).qsch ; option - Gmin L (.tran).qsch

- Gmin in Inductor

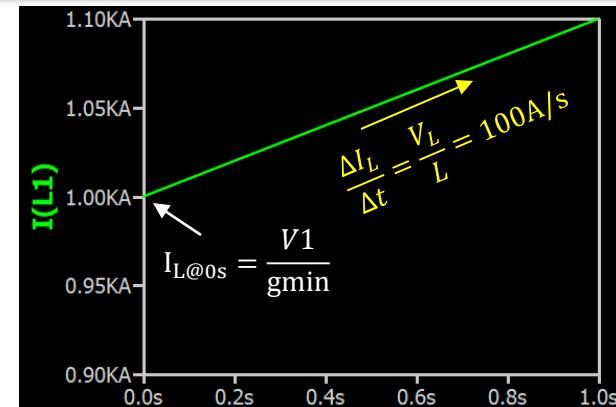
- Gmin is also applied to the inductor in .op and .tran for initial inductor current calculation
- However, gmin, despite its definition as a conductance, functions as a series resistance value utilized for calculating the initial inductor current
- Unlike PN junctions, gmin is solely employed in the initial current calculation and is removed during transient analysis



```
.step dec param gmin 1e-12 1e-4 10
.options GMIN=gmin
.plot I(L1)
.op ← DC Operation Point Analysis
```



```
.options gmin=1e-3
.plot I(L1)
.tran 1
```



## .option – ITL1, ITL2 and ITL4 (Iteration Limit for Simulation Directive)

---

- ITL1 : DC iteration limit – used in .op
  - Default ITL1=100
- ITL2 : DC transfer curve iteration limit – used in .dc
  - Default ITL2 = 50
- ITL4 : Transient analysis iteration limit – used in .tran
  - Default ITL4 = 10

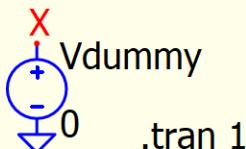
# .option – ListParam (Print a list of evaluated parameters)

Qspice : .option - ListParam.qsch

- ListParam
  - Print a list of the evaluated parameters (also sets SAVEPARAM)
  - This option can return a list of parameter values in the console output
  - It also sets the .option SaveParam, where all parameter values are saved into the output data file .qraw, which can be plotted in the waveform viewer for post-processing

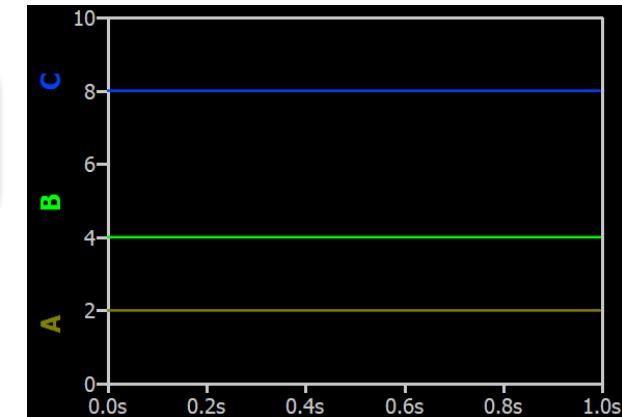
```
.param a=2  
.param b=4  
.param c=a^2+b
```

```
.option ListParam
```



Output Window

```
--- Parameter Evaluations ---
TEMP      = 27      "CKTTEMP"
A         = 2        "2"
B         = 4        "4"
C         = 8        "A^2+B"
```



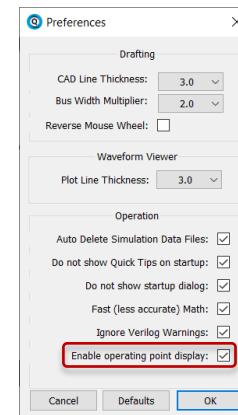
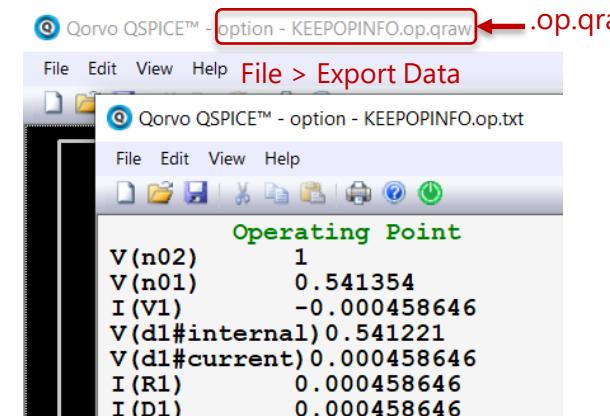
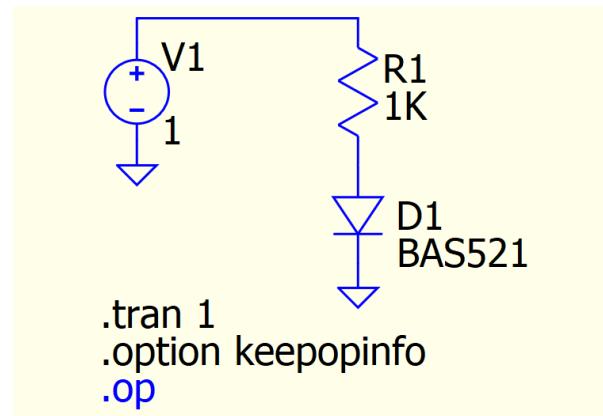
# .option – MAXORD (Maximum Integration Order)

- Maxord
  - Maximum integration order
  - **Default MAXORD=2**

.option MAXORD	.option METHOD	Integration Method	Comment
1	Gear   Trap	Backward Euler	Less accurate than Gear
2 (Default)	Gear	Gear	It dampens all ringing, not just numerical ringing but even physical ringing
	Trap (Default)	Trapezoidal	Trap integration is recommended but can give rise to trap ringing and cause the user to be suspicious of the correctness of the simulator even though each trapezoid contains the correct integrated area

# .option – KEEPOPINFO (Record .OP Info)

- KeepOpInfo
  - Record operation point into .qraw for .ac and .tran
  - A file with extension .op.qraw is forced to create
    - This .op.qraw file contains DC operating point data
    - Open .op.qraw file with waveform viewer, a list of operating point is generated
  - .option keepopinfo is equivalent to use
    - Edit > Preferences
    - Click "Enable operating point display"
    - Hover device to display data only work with this enable



# .option – Method (Integration Method : Trapezoidal or Gear)

---

- Method
  - Integration method : Trapezoidal or Gear
  - **Default METHOD = Trapezoidal**
  - Trapezoidal : Area under every trapezoid is correct which can yield accurate results
    - One disadvantage of using the trapezoidal method is that it can lead to trapezoidal ringing when working with unrealistic circuit elements
    - In LTspice, it smooth trap ringing is smoothed out (method : modified trap), but in Qspice, one needs to know what the gates and flop truly perceive in order to address it
  - Gear : Adds damping that isn't present in the circuit and does not provide as exact of a result. Generally, it is slower and less accurate compared to the trapezoidal method.
    - This is the default method in Pspice
  - Reference article : SPICE Differentiation by Mike Engelhardt in Jun 19 2015
    - <https://www.analog.com/en/resources/technical-articles/spice-differentiation.html>

# .option – Method (Integration Method : Trapezoidal or Gear)

## Reference by Mike Engelhardt in Qspice forum

- <https://forum.qorvo.com/t/need-guidance-on-qspice-integration-method-and-this-feather-parameter/14393>
- <https://forum.qorvo.com/t/apparent-kirchhoff-s-law-violation/15048/2>
- <https://forum.qorvo.com/t/gear-vs-trap-what-are-those-and-how-to-optimize/16431>
- [https://ltwiki.org/files/LTspiceHelp.chm/html/integration\\_method\\_issues.htm](https://ltwiki.org/files/LTspiceHelp.chm/html/integration_method_issues.htm)



Engelhardt

26m

Aside from a few exceptions, Gear is both slower and less accurate than trap. The main problem with trap is a ringing artifact that bothers people, even though the area under each trapezoid is correct.

You've discovered the main point to offering both in the same simulator: if you get the same answer using both methods, you know that the solution is not affected by integration artifacts.

—Mike



Engelhardt

Jul '23

FEATHER is an experimental parameter. It can be used to duplicate the de-tuned trap integration of HSPICE. Its use is not recommended.

As far as trap ringing, I realize it's disconcerting, but in a sense it's giving the right answer: The area under each trapezoid is correct, so one knows that the differential equations are correctly integrated. It can be reduced by either (i) using ".options method=Gear" (ii) stipulating a lower trtol, or (iii) stipulating a smaller maximum timestep. Gear is not recommended because it adds a substantial artificial damping to the circuit. I know of two cases where the use of Gear integration let an IC designer believe his circuit was stable until silicon said different and a turning Gear off confirmed.

Unlike some prior art, I don't smooth trap ringing out it out at all because one needs to know what the gates and flop truly see.



Engelhardt

Aug '23

That looks like trapezoidal ringing - a situation where the numerically-integrated solution oscillates about the true solution timestep to timestep. A side effect is that current monitoring can look off[1].

In a sense, it's giving the correct answer, in that the area under each trapezoid has the correct area, but it is certainly disconcerting.

You can reduce or eliminate it by some of a combination of these methods:

1. Reduce trtol(.options trtol=1)
2. Reduce the maximum allow time step size(4th number on the .tran command)
3. Change to Gear integration(.options method=Gear)

The last option is the most empathetic way of getting rid of it, but it also introduces the greatest error. A simulation done with Gear integration can look stable when in fact, it is not. I've seen IC designers go to silicon only to learn they made the mistake of using Gear integration to check the IC's operation.

—Mike

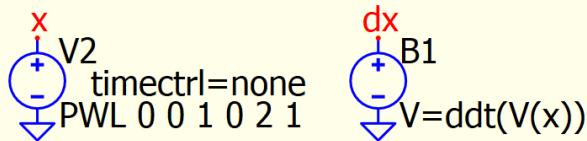
[1] Current monitoring is not really part of the solution of the circuit. It's the node voltages that are solved for, except for the voltage sources where the currents are part of the solution. Current monitoring is done as a forensic analysis of the circuit. Current reporting can be in error even if the rest of the solution is correct. I'll fix errors in report as they come up, e.g., today I fixed an issue in capacitor current reporting.

# .option – Method : Trapezoidal Ringing

Qspice : option - Method (trap ringing).qsch

- **Trapezoidal Ringing**

- Trapezoidal method accurately calculates the area under each trapezoid
- However, trapezoidal ringing can occur when dealing with unrealistic circuit elements or when the function being integrated has sudden changes or contains high-frequency components
- This example demonstrates how trapezoidal ringing occurs in a simulation when using a ramp source and a derivative function
- The first example illustrates the occurrence of trapezoidal ringing, while the second example forces the simulation step to the exact break point without any trapezoidal ringing



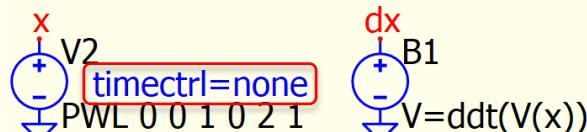
.option maxstep=1.94e-3

.tran 0 2

.plot V(dx)

.plot V(x)

In this setup, if you set different maxstep, you can get different magnitude of trap ringing



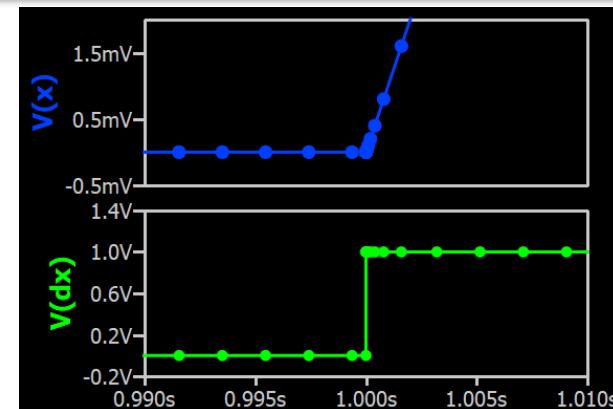
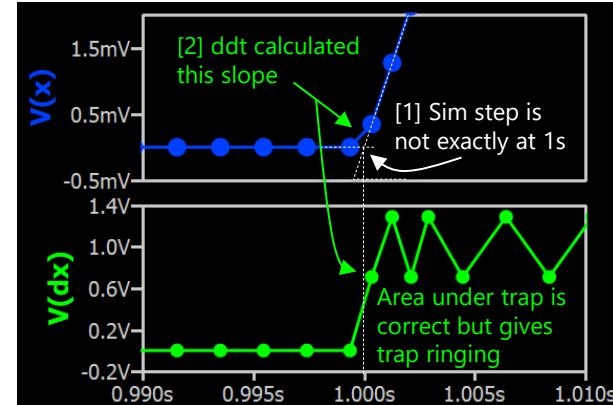
.option maxstep=1.94e-3

.tran 0 2

.plot V(dx)

.plot V(x)

Enable timectrl for Vsource  
Qspice simulation step can happen at exact break point (i.e. at 1s)

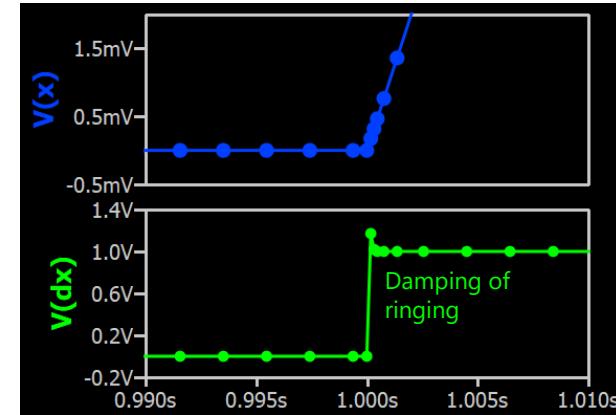
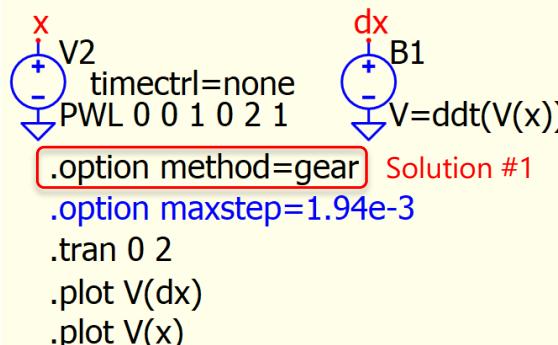


# .option – Method : Trapezoidal Ringing

Qspice : option - Method (trap ringing).qsch

- Trapezoidal Ringing (Solution)

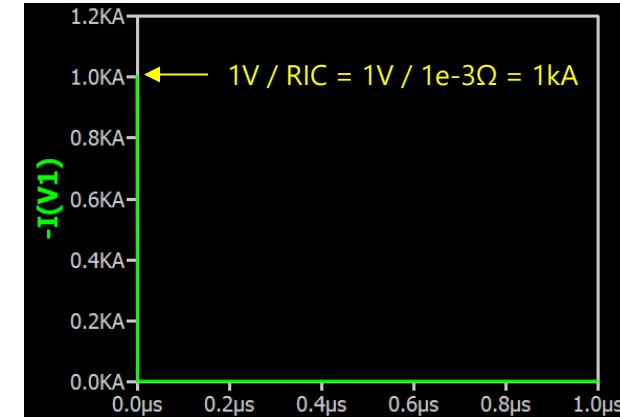
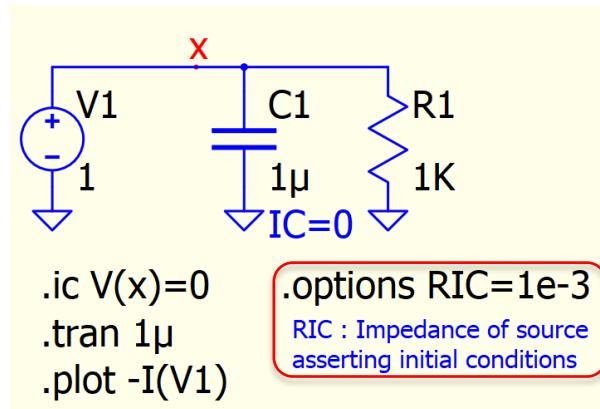
- #1 : Gear integration (.option method=gear) introduces artificial damping that is not present in the circuit, effectively reducing ringing during integration
  - However, this approach is not generally recommended as it does not provide exact results
- #2 : Reduce trtol (.option trtol)
- #3 : Reduce maximum timestep (.option maxstep) or use a device with TTOL to reduce temporal timestep when approach trap ringing
- #4 : Damping factor can be added when trap is used (but not recommended), refer to section **.option – Feather**



# .option – RIC (Impedance of source asserting initial conditions)

Qspice : option - RIC C.qsch

- RIC
  - Impedance of source asserting initial conditions
  - Default RIC=1mΩ
  - In this example, initial voltage of V(x) is defined at 0V. A 0V capacitor connected in parallel to a voltage source will force an infinite current in first time step. RIC is source impedance inserted in source for initial condition calculation purpose. In this case, source current at t=0 becomes  $I_{V1} = \frac{V_{V1}}{RIC}$



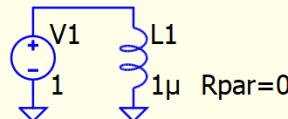
# .option – RIC (Impedance of source asserting initial conditions)

Qspice : option - RIC L.qsch

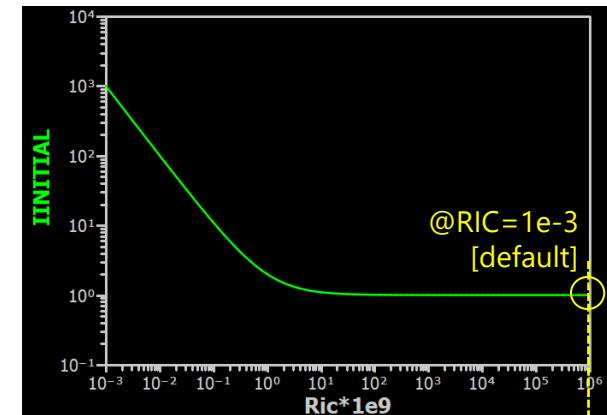
- RIC
  - Impedance of source asserting initial conditions
  - Inductor currents are asserted with the compliance of  $1e9 * RIC$
  - Default RIC=1mΩ

- Important note
  - RIC only affect inductor current if .ic is used to define inductor initial current
  - In this simulation example, initial inductor current is plotted with Gmin=1e-12 and Gmin=1e3 with .ic I(L1)=1
    - When RIC=1e-3 (default), initial current is always equal .ic defined value

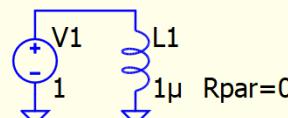
```
.options Gmin=1e-12 ←  
.options RIC=RIC  
.step dec param RIC 1e-12 1e-3 10
```



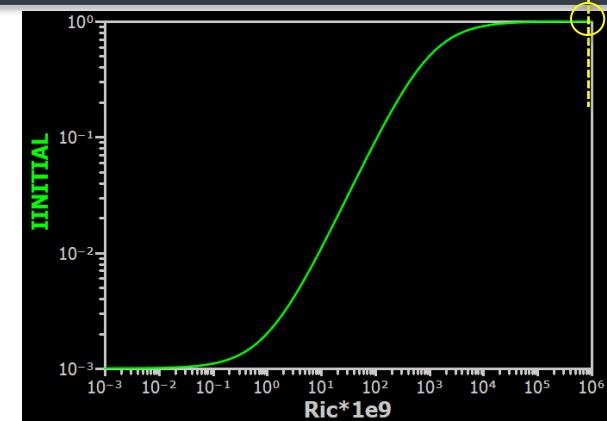
```
.ic I(L1)=1  
.plot I(L1)  
.tran 1n  
.meas Iinitial find I(L1) at 0
```



```
.options Gmin=1e3 ←  
.options RIC=RIC  
.step dec param RIC 1e-12 1e-3 10
```



```
.ic I(L1)=1  
.plot I(L1)  
.tran 1n  
.meas Iinitial find I(L1) at 0
```

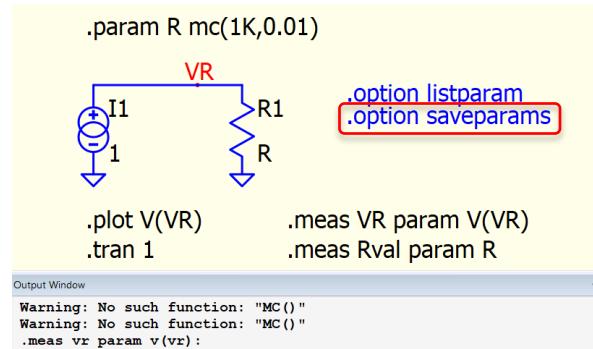


# .option – SaveParam (Save .param into waveform data)

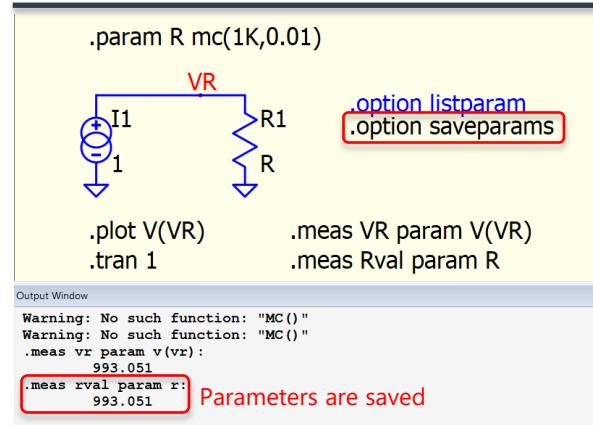
Qspice : option - SaveParam.qsch

- **SaveParam**

- Include evaluated user-defined parameters as waveform data
- In default, Qspice does NOT store user-defined parameters (.param) into output data file (.qraw)
- If user-defined parameters are needed in .plot or .meas, enabling this option is necessary
- Two ways to enable this
  - .option SaveParam
  - .option ListParam
    - This automatically sets SaveParam option, but as param is listed in Output Window, it will slow the simulation as display requires msleep()



```
Title: * C:\KSKelvinQspice\01 User Guide and S
Guide\.option\option - SaveParam.qsch
Date: Wed May 1 09:29:35 2024
Plotname: Transient Analysis
Plot Suggestion(s): «V(VR)»
Flags: real
Abscissa: 0.000000000000000e+00
1.000000000000000e+00
No. Variables: 3
No. Points: 1039
Command: QSPICE80, Build Apr 30 2024 17:13:22
.param R MC(1K,0.01)
.param temp=27
.alias I(R1) (0.00100699774249268mho*V(vr,0))
Variables:
0 Time time
1 V(vr) voltage
2 I(I1) current
```

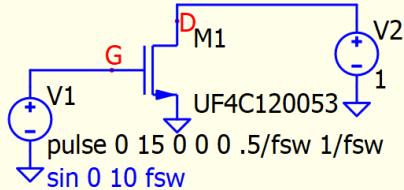


```
Title: * C:\KSKelvinQspice\01 User Guide and S
Guide\.option\option - SaveParam.qsch
Date: Wed May 1 09:28:48 2024
Plotname: Transient Analysis
Plot Suggestion(s): «V(VR)»
Flags: real
Abscissa: 0.000000000000000e+00
1.000000000000000e+00
No. Variables: 5
No. Points: 1039
Command: QSPICE80, Build Apr 30 2024 17:13:22
.param R MC(1K,0.01)
.alias I(R1) (0.00100699774249268mho*V(vr,0))
Variables:
0 Time time
1 V(vr) voltage
2 I(I1) current
3 TEMP parameter
4 R parameter ] Parameters are saved
```

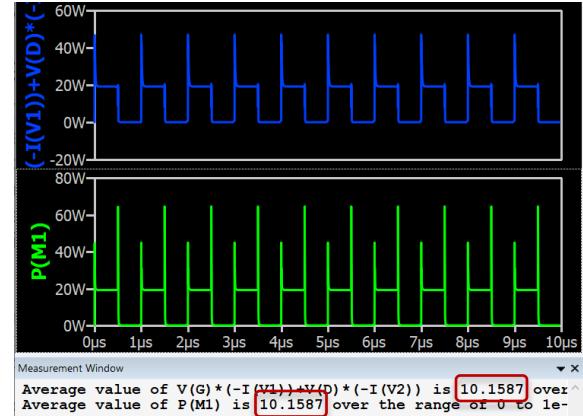
# .option – SavePowers (Compute and save dissipation power)

Qspice : option - SavePower (nmos).qsch

- SavePowers
  - Compute and save the dissipation of components
  - Computes the true power dissipation while ignoring displacement currents
    - Implemented for BJTs, Capacitors, Diodes, Inductors, JFETs, MOSFET level 1, MOSFET level 2010 and VDMOS
  - P() will be available in plot expression, and P() represent dissipation power of entire component
    - e.g. both gate and drain loss for nmos



```
pulse 0 15 0 0 0 0 .5/fsw 1/fsw  
sin 0 10 fsw  
.param fsw=1Meg  
.option maxstep=1/fsw/1e5  
.tran 10/fsw  
.plot P(M1)  
.plot V(G)*(-I(V1))+V(D)*(-I(V2))  
.option savepowers
```



- NMOS example
  - In this example, the NMOS is the only dissipation device, and its power is equal to the input power from V1 and V2
  - Input Power =  $V(G) * -I(V1) + V(D) * -I(V2)$
  - Qspice calculated NMOS power =  $P(M1)$
  - Average power of Input Power and  $P(M1)$  are equal (but Instantaneous power profile are different!!)
    - It can conclude that  $P(M1)$  include gate and drain power dissipation!

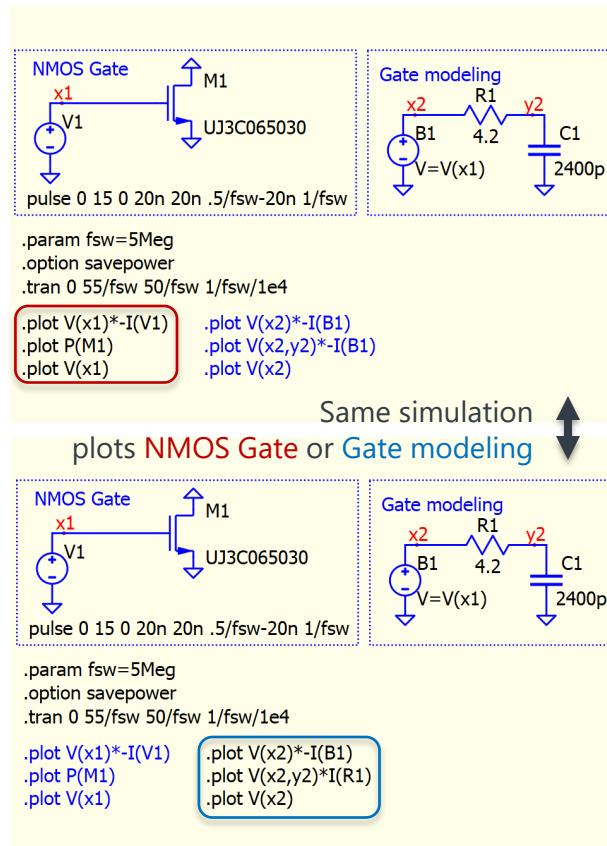
# .option – SavePowers (Compute and save dissipation power)

Qspice : option - SavePower (nmos-gate).qsch

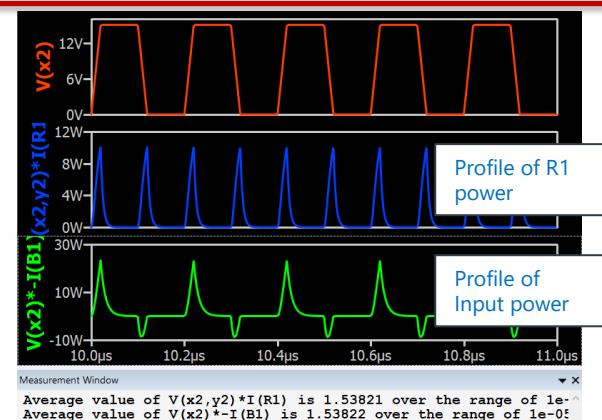
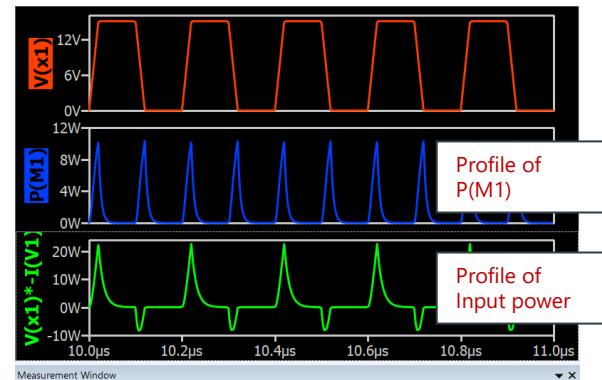
- Instantaneous Power
  - Example of gate power consumption.

Instantaneous input power from V1 and dissipation power  $P(M1)$  is different, but their average power is the same

- Gate model circuit  $R1$  and  $C1$  explains why instantaneous power profiles are different, as gate is not purely resistive but with reactive power in capacitance
- Therefore,  $P(M1)$  in savepower option calculate instantaneous real power in device



Same simulation  
plots NMOS Gate or Gate modeling



# .option – Scale (Geometric element parameters scaling factor)

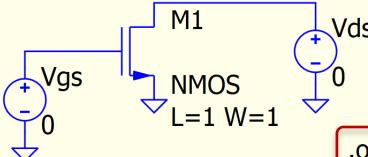
Qspice : option - Scale.qsch

- **Scale**
  - This scaling factor applies to geometric element parameters and has a default unit of meters
  - **Default Scale=1**
  - An external reference in the Ngspice User's Manual on page 306, as quoted below

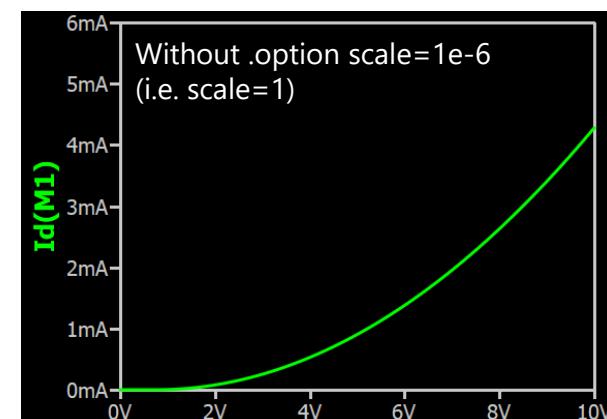
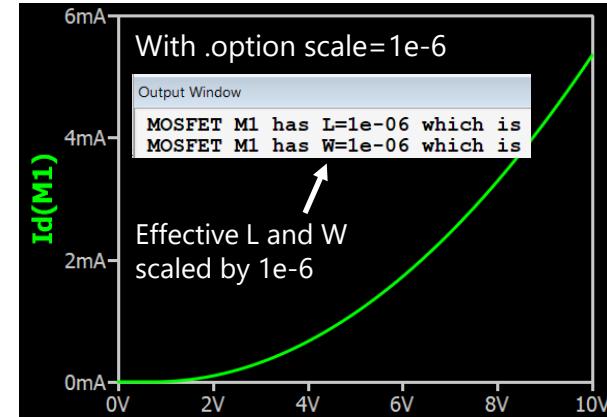
SCALE=x set the element scaling factor for geometric element parameters whose default unit is meters. As an example: scale=1u and a MOSFET instance parameter W=10 will result in a width of  $10\mu m$  for this device. An area parameter AD=20 will result in  $20e-12 m^2$ . Following instance parameters are scaled:

- Resistors and Capacitors: W, L
- Diodes: W, L, Area
- JFET, MESFET: W, L, Area
- MOSFET: W, L, AS, AD, PS, PD, SA, SB, SC, SD

NGspice User's Manual : page 306 (.option scale)



```
.option scale=1e-6  
.model nmos nmos (kp=100u vto=0.75 LD=1e-7)  
.dc Vgs 0 10 0.1 Vds list 10  
.plot Id(M1)
```

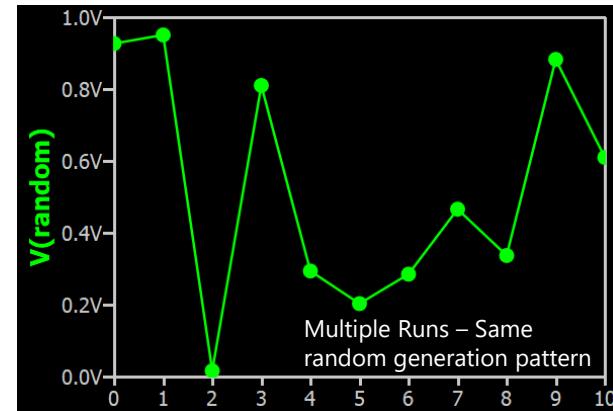


# .option – Seed and Seedclock (Seed of random number generation)

Qspice : option - Seed Seedclock.qsch

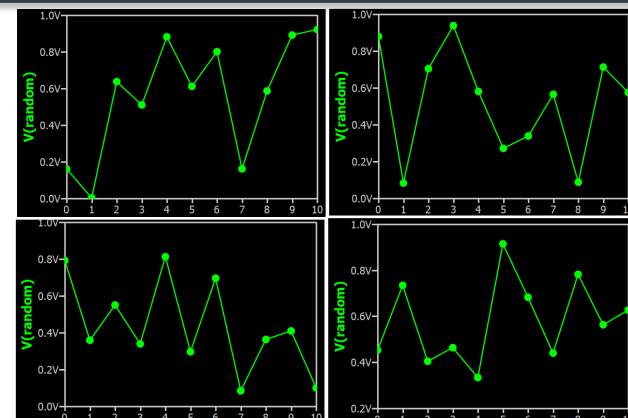
- Seed
  - Initialize the random number generator used in .param statements
  - Same random pattern is generated between Simulation Run

random .option seedclock  
B1 .option seed=719749  
V=random()  
  
.op  
.step param n 0 10 1  
.plot V(random)



- Seedclock (aka Seedclk)
  - Initialize the random number generator with a 10Mhz clock and the process ID number(aka SEEDCLK)
  - Different random pattern is generated between Simulation Run

random .option seedclock  
B1 .option seed=719749  
V=random()  
  
.op  
.step param n 0 10 1  
.plot V(random)



# .option – TRTOL (Truncation error overestimation factor)

---

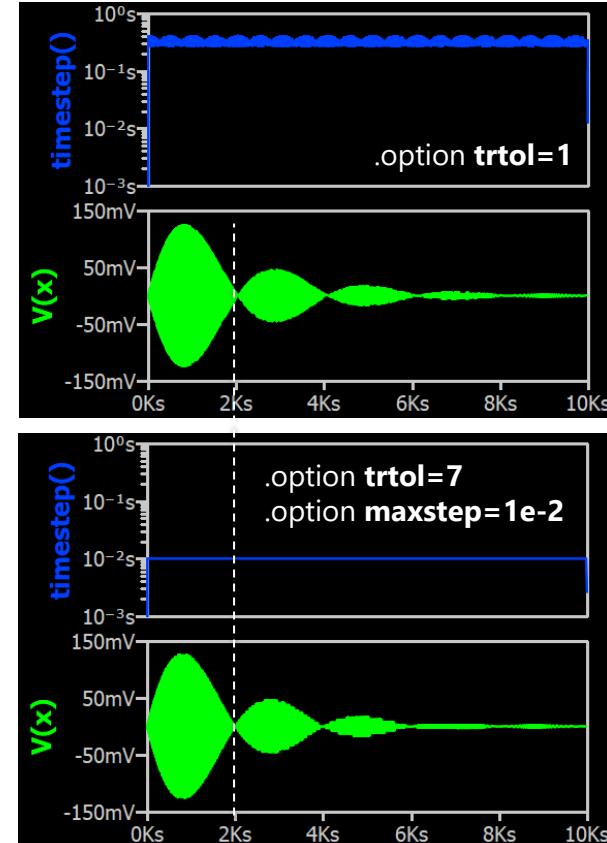
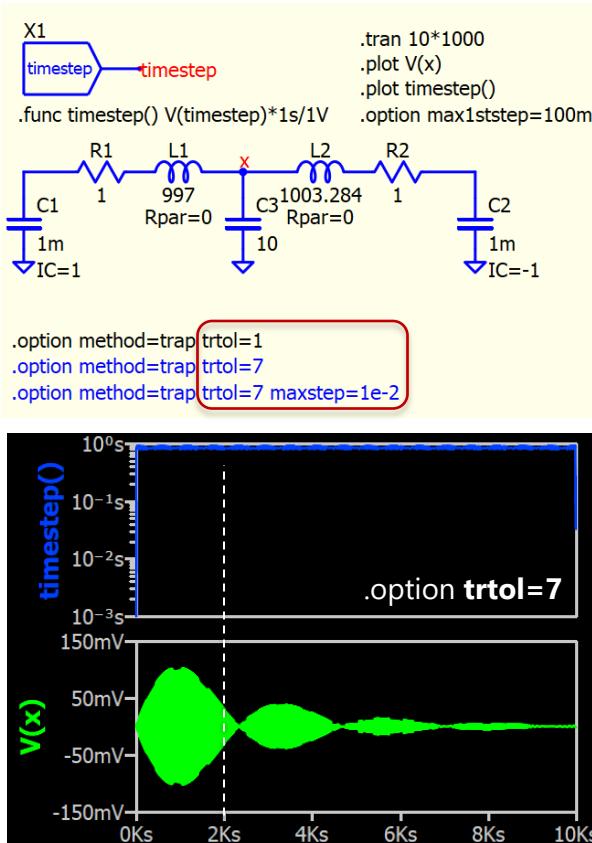
- TRTOL
  - Truncation error overestimation factor (Default Value in Spice Simulator)
    - Qspice : TRTOL = 2.5, Method=Trap
    - LTspice : TRTOL = 2, Method=Trap
    - TI-TINA : TRTOL = 7, Method=Gear
    - TI-PSpice : TRTOL = 7, Method=Gear
    - *\*\* For Qspice to run with Pspice model, may have to consider to change TRTOL=7 and Method=Gear*
  - Trtol affects the timestep strategy more than directly affecting the accuracy of the simulation
  - For transistor-level simulations, a value larger than 1 is usually a better overall solution. You might find that you get a speed of 2x if you increase trtol without adversely affecting simulation accuracy

# .option – TRTOL (Truncation error overestimation factor)

Qspice : option - TRTOL - PrecisionTestSimulations.qsch

- TRTOL

- The accumulation of truncation errors over the present and past time step is managed by Truncation error factor (TRTOL)
- This precision test circuit intermodulates two frequencies separated by 1/2000 Hz, resulting in a beat frequency at 1/2000 Hz in the envelope
- In this example, TRTOL=7 produces an imprecise solution (error in beat frequency). However, limiting the maxstep can yield more accurate results



.param  
User-Defined  
Parameter

# .param – User-defined Parameters

---

- .option relates to .param
  - .option ListParam : Print a list of the evaluated parameters(also sets SAVEPARAM)
  - .option SaveParam : Include evaluated user-defined parameters as waveform data
  - .option Seed : Initialize the random number generator used in .param statements

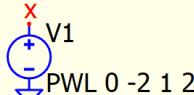
# .param – User-defined Parameters

Qspice : .param from param - SaveParam.qsch

- **.param**
  - If a **.param** argument is derived from another **.param** argument or from a formula, it is necessary to set **.option SaveParam** in order to save the parameters in the **.qraw** file
  - This is because the **.qraw** file only stores the values of **.param** arguments if they are direct values (e.g., **param a=1**)
  - If the value is derived indirectly, the **.qraw** file will not save it in the pre-headers lines. By using the **.option SaveParam**, parameters can be written as data channels

Math formula implementation with **.tran**

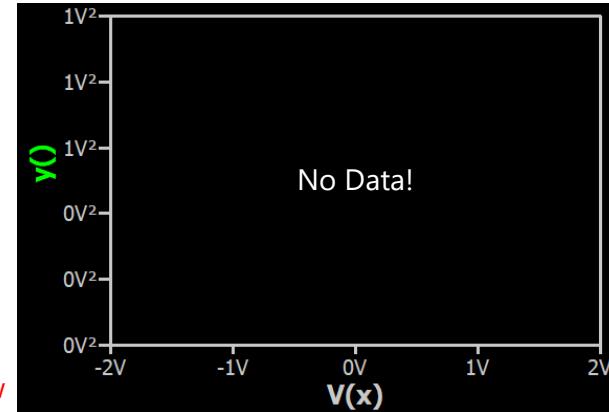
```
.param a=1  
.param b=var  
.param c=1  
.step param var list 0 1 2
```



```
.func y() a*V(x)**2+b*V(x)+c .plot y()
```

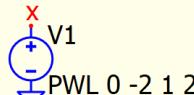
```
.option SaveParam
```

```
.tran 1 Without parameters save into .qraw
```



Math formula implementation with **.tran**

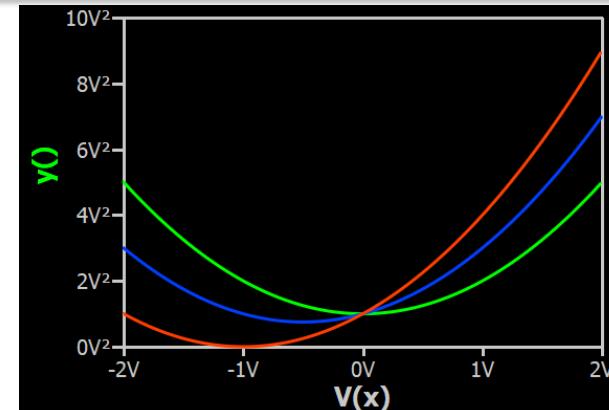
```
.param a=1  
.param b=var  
.param c=1  
.step param var list 0 1 2
```



```
.func y() a*V(x)**2+b*V(x)+c .plot y()
```

```
.option SaveParam
```

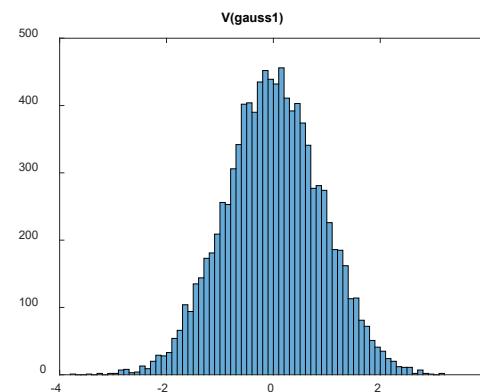
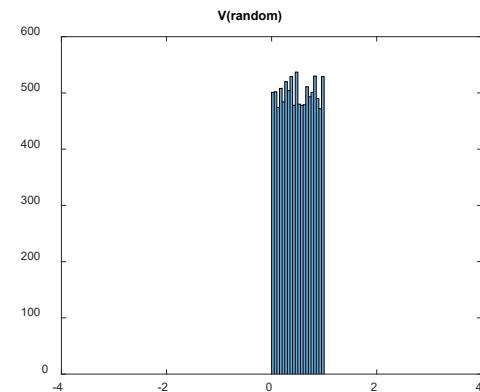
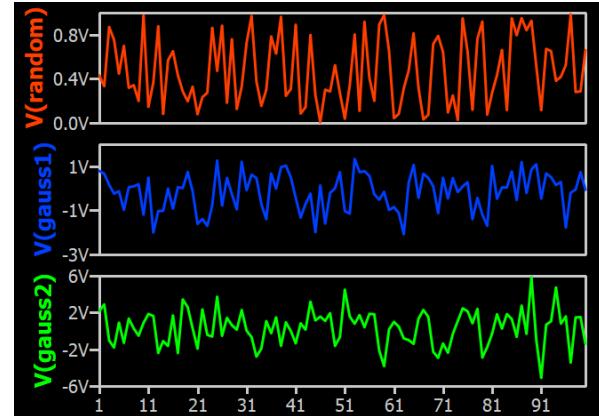
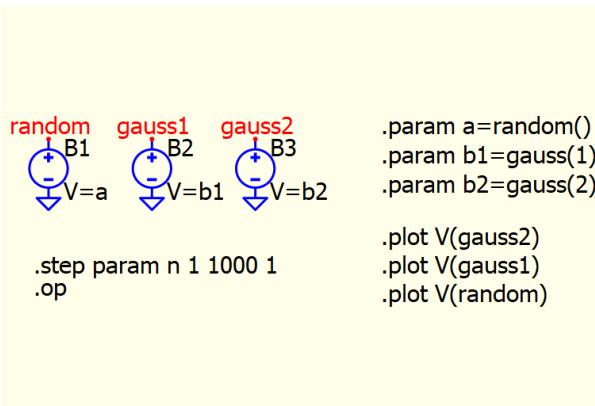
```
.tran 1 With parameters save into .qraw
```



# .param functions – random() and gauss(double sigma)

Qspice : param - random and gauss.qsch

- .param functions
  - Preparing evaluation functions for .param
  - Random()
    - Random value between 0 and 1, uniform distribution
  - Gauss( $\sigma$ ) [Undocumented]
    - Random number from Gaussian/Normal distribution with standard derivation  $\sigma$  and mean value at 0
  - Flat( $x$ ) [Undocumented]
    - Random number between  $-x$  and  $x$  with uniform distribution
  - Mc( $x,y$ ) [Undocumented]
    - Random number between  $x^*(1+y)$  and  $x^*(1-y)$  with uniform distribution



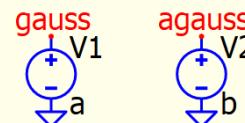
.param functions – Gauss(nom,rvar,sigma) and Agauss(nom,avar,sigma)

# Qspice : param - gauss and agauss.qsch

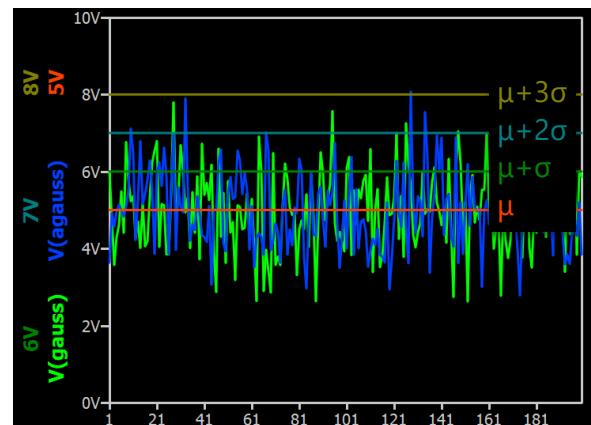
- **Gauss(nom,rvar,sigma)** and **Agauss(nom,avar,sigma)**
    - Both are gaussian distribution (i.e. normal distribution), and different is how standard deviation is defined
    - **GAUSS(Nominal Value, Relative variation, Sigma)** [Undocumented]
      - Mean ( $\mu$ ) = nom = Nominal Value
      - Standard Deviation ( $\sigma$ ) =  $\frac{\text{nom} \times \text{rvar}}{\text{sigma}}$
      - \*\* If mean (nom) is zero, standard deviation is also zero
    - **AGAUSS(Nominal Value, Absolute variation, Sigma)** [Undocumented]
      - Mean ( $\mu$ ) = nom = Nominal Value
      - Standard Deviation ( $\sigma$ ) =  $\frac{\text{avar}}{\text{sigma}}$

```
.param a=gauss(5,.2,1)  
.param b=a $\cdot$ gauss(5,1,1)
```

.step param LOOP 1 200 1



```
.op  
.plot V(gauss),V(agauss),5V,6V,7V,8V
```



.step  
Step User-Defined  
Parameter

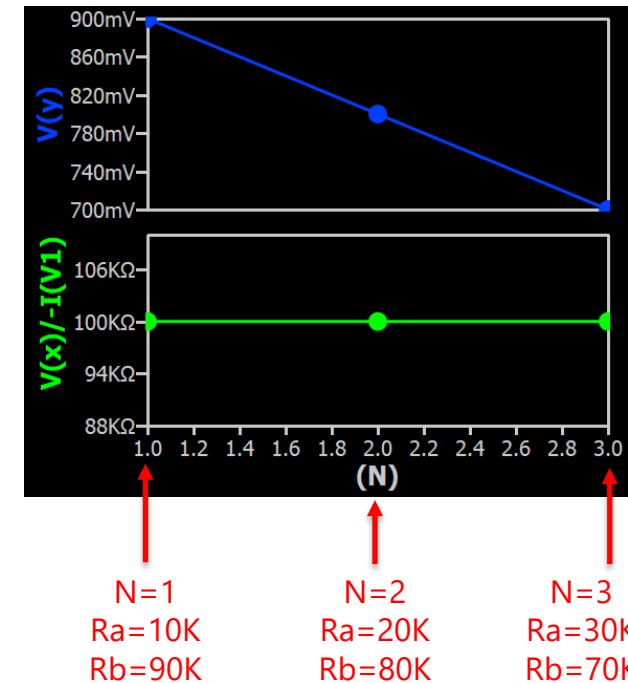
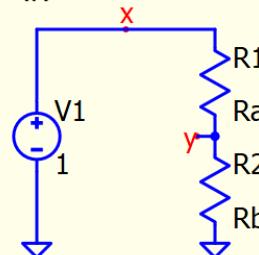
# .step with table : Batch Simulation

Qspice : Step with Table.qsch

- .step with table

- By using table function, user can step integer N from 1, 2, 3, ... and assign value to different parameters according to table
- This is an example to sweep upper and lower resistor network with different resistance combination

```
.op  
.step param N list 1 2 3  
.param Ra table(N, 1,10K, 2,20K, 3,30K)  
.param Rb table(N, 1,90K, 2,80K, 3,70K)  
.plot V(x)/-I(V1)  
.plot V(y)
```

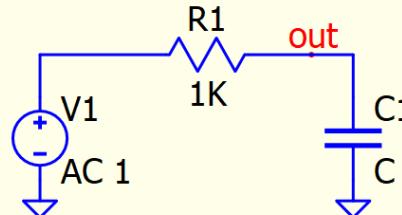


# .step in Single Point Simulation – Explanation (page 1)

Qspice : Step in Single Point - 01 Explain.qsch

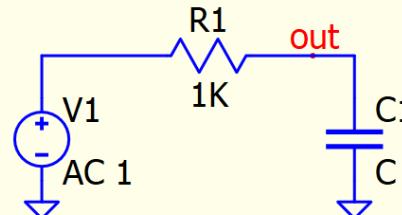
- Single Point Simulation
  - The .step command is treated differently in the output data file depending on whether it is a single point or multiple points simulation directive
  - Single Point Simulation
    - Includes .op and .ac list
    - In .qraw, flags doesn't have "stepped", therefore, .step is not treated as stepped in single point simulation
    - In variables, x-axis variable "0" is C
  - Multiple Point Simulation
    - If simulation directive is multiple points, data file will set stepped flag and with x-axis as Frequency / Voltage / Current depends on it is .ac list or .op

```
.ac list 100 Single point simulation
.param C = 1μ
.step param C 1μ 10μ 1μ
```



```
Step in Single Point Simulation.qraw x
1 Title: * C:\KSKelvinQspice\01 User Guide and S
2 Date: Sun May 5 01:01:07 2024
3 Plotname: AC Analysis
4 Flags: complex ←
5 Abscissa: 1.000000000000000e-06 9.9999
6 No. Variables: 6
7 No. Points: 10
8 Command: QSPICE80, Build Apr 30 2024 17:13:22
9 .param C=1μ
10 .param temp=27
11 .alias Freq Frequency
12 .alias Omega 2*pi*Frequency
13 Variables:
14 0 C parameter x-axis ←
15 1 V(n01) voltage
16 2 V(out) voltage
17 3 I(V1) current
18 4 I(R1) current
19 5 I(C1) current
20 6 C parameter
```

```
.ac list 100 1K Multiple points simulation
.param C = 1μ
.step param C 1μ 10μ 1μ
```

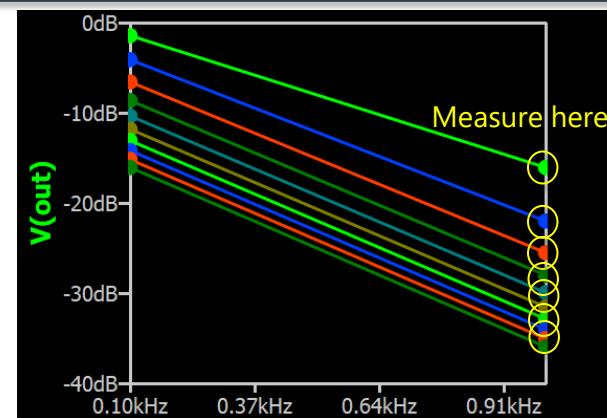
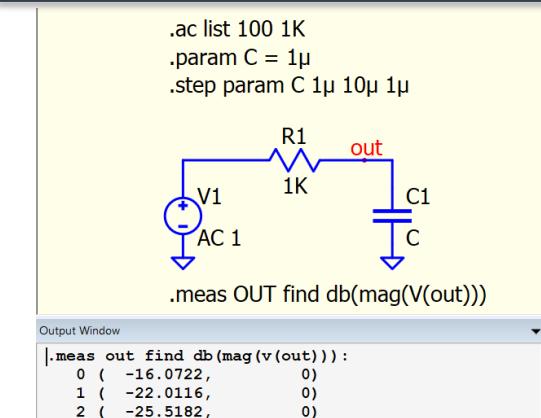
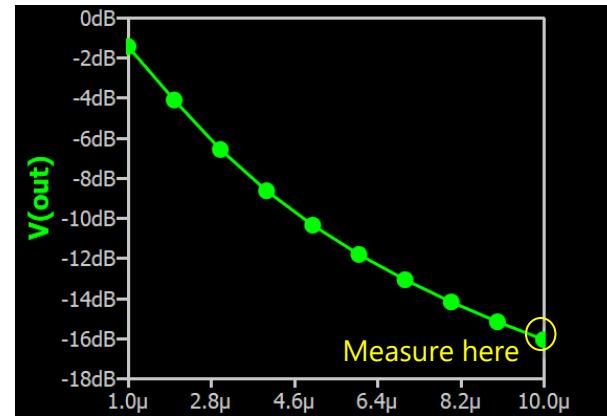
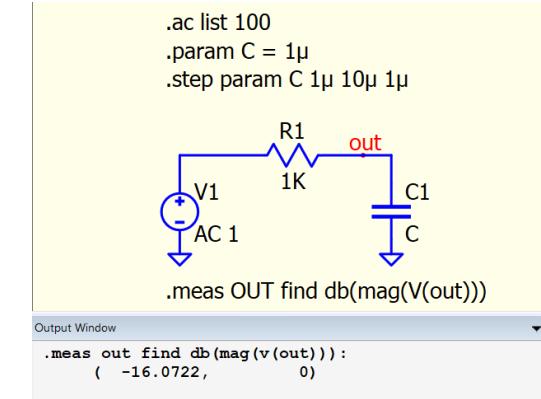


```
Step in Single Point Simulation.qraw x
1 Title: * C:\KSKelvinQspice\01 User Guide and S
2 Date: Sun May 5 01:02:59 2024
3 Plotname: AC Analysis
4 Flags: complex stepped ←
5 Abscissa: 1.000000000000000e+02 1.0000
6 No. Variables: 7
7 No. Points: 20
8 Command: QSPICE80, Build Apr 30 2024 17:13:22
9 .param C=1μ
10 .param temp=27
11 .alias Freq Frequency
12 .alias Omega 2*pi*Frequency
13 Variables:
14 0 Frequency frequency x-axis ←
15 1 V(n01) voltage
16 2 V(out) voltage
17 3 I(V1) current
18 4 I(R1) current
19 5 I(C1) current
20 6 C parameter
```

# .step in Single Point Simulation – Impact (page 2)

Qspice : Step in Single Point - 02 Impact.qsch

- **Impact**
  - In the .plot command, for a single point simulation with .step, the stepped parameters are plotted on the x-axis, resulting in only one curve. However, in a multiple point simulation with .step, each stepped parameter generates a separate curve
  - In the .meas command, for a single point simulation, as the data set is not treated as stepped data, only one measurement result is output. However, in a multiple point simulation, the .meas command returns results equal to the total number of steps



**.system**  
**Execute System**  
**Commands**

# .system – Environment Strings

- Environment Strings

- %DECK%
  - (depends on to run a .qsch, or [View] > [Netlist] for a .cir)
  - [userpath]\[filename].qsch OR
  - [userpath]\[filename].cir
- %RAWFILE%
  - [userpath]\[filename].qraw
- %QUX%
  - [QspiceInstallPath]\QUX.exe
- %QPOST%
  - [QspiceInstallPath]\QPOST.exe
- %QSPICE64%
  - [QspiceInstallPath]\QSPICE64.exe
- %QSPICE80%
  - [QspiceInstallPath]\QSPICE80.exe
- %QSPICE%
  - (depends on which simulator is selected in Qspice preference or .option fastmath)
  - [QspiceInstallPath]\QSPICE64.exe OR
  - [QspiceInstallPath]\QSPICE80.exe

## Declared Environmental Strings

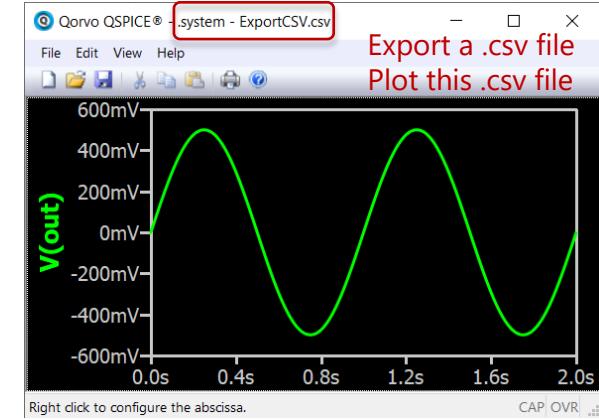
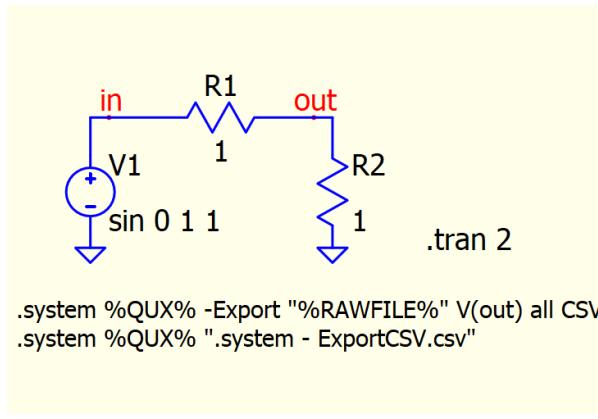
Name	Description
DECK	Path of the input netlist
RAWFILE	Path of the output waveform file
QUX	Path to QUX.exe, the waveform viewer
QPOST	Path to QPOST.exe, the post processor for .meas and .four
QSPICE64	Path to QSPICE64.exe, the simulator using fast math
QSPICE80	Path to QSPICE80.exe, the simulator
QSPICE	Path to the current simulator in use

- Environmental parameters can return entire path and filename for user to quickly access entire filepath for system command

# .system – Execute a Command, Export CSV example

Qspice : .system - ExportCSV.qsch

- .system – export CSV
  - To execute commands once the simulation completes
  - Command will be executed as if you had typed them at the DOS C:\> prompt
- Environment Strings
  - %DECK%
    - [userpath]\[filename].qsch OR
    - [userpath]\[filename].cir
  - %RAWFILE%
    - [userpath]\[filename].qraw
  - %QUX%
    - [QspiceInstallPath]\QUX.exe

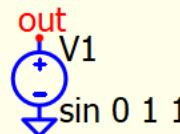


- Explanation of example
  - First .system is to export \*.qraw into csv file with QUX.exe for V(out)
  - Second .system is to plot this exported csv file
  - "" is require if file contains space in environment strings

# .system – Execute a Command, Export .meas results

Qspice : .system - ExportQPost.qsch | meas-script.txt

- .system – export .meas
  - Exporting the .meas output file can be a bit tricky
  - In the command line, QPOST.exe is used to post-process the .qraw waveform data through a text file that contains the .meas statement (\*.cir or \*.txt file)
  - However, Qspice runs from the GUI will not save a \*.cir file from .qsch; therefore, the .meas output is not in a text file format
  - Solution is to create a text file for .meas script, and with .system to call QPOST.exe to execute this script file for .qraw waveform data and generate post-processing result



.tran 10  
.plot V(out)

command syntax : QPOST <.meas in text> -r <.qraw file> -o <.out file>  
.system %QPOST% "meas-script.txt" -r "%RAWFILE%" -o "QPostResult.out"

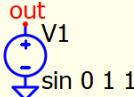
```
meas-script.txt
1 * measure avg, rms, max, min and pp of V(out)
2 .meas Vavg AVG V(out)
3 .meas Vrms RMS V(out)
4 .meas Vmax MAX V(out)
5 .meas Vmin MIN V(out)
6 .meas Vpp PP V(out)
```

```
QPostResult.out
1 .meas vavg avg v(out):
2   8.52067e-09
3 .meas vrms rms v(out):
4   0.707106
5 .meas vmax max v(out):
6   0.999994 (at Time=2.24944)
7 .meas vmin min v(out):
8   -0.999994 (at Time=4.74944)
9 .meas vpp pp v(out):
10  1.99999
```

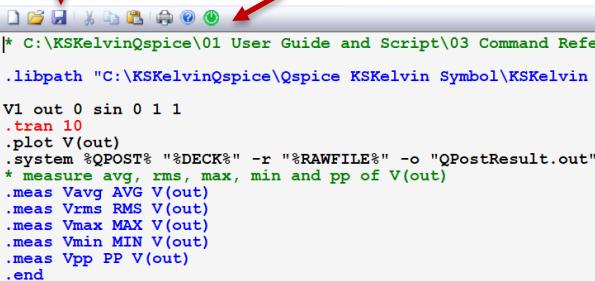
# .system – Execute a Command, Export .meas results

Qspice : .system - ExportQPost-Netlist.qsch

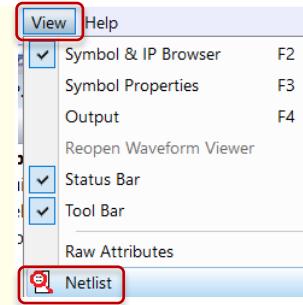
- .system – export .meas
  - If the .meas statement is only in the schematic file and you intend to use %DECK% to load a .cir file in post-processing, the user should run the simulation from the netlist
- Procedure
  - View > Netlist in schematic window
  - In Netlist, File > Save, save file into \*.cir
  - Run the simulation from netlist
  - Qspice will directly load a netlist file (\*.cir) instead of (\*.qsch) when utilizing the environment parameter %DECK%

  
out  
V1 \* measure avg, rms, max, min and pp of V(out)  
.meas Vavg AVG V(out)  
.meas Vrms RMS V(out)  
.meas Vmax MAX V(out)  
.meas Vmin MIN V(out)  
.meas Vpp PP V(out)  
  
.tran 10  
.plot V(out)  
  
command syntax : QPOST <.cir file> -r <.qraw file> -o <.out file>  
.system %QPOST% "%DECK%" -r "%RAWFILE%" -o "QPostResult.out"  
\*\* Procedure for this .system to work (i.e. %DECK% to call a .cir instead of .qsch)  
1. View > Netlist  
2. In Netlist Window, File > Save, save file into .cir  
3. Run simulation from Netlist

Save as a .cir      Run from netlist



```
* C:\KSKelvinQspice\01 User Guide and Script\03 Command Reference
.libpath "C:\KSKelvinQspice\Qspice KSKelvin Symbol\KSKelvin Symbol"
V1 out 0 sin 0 1 1
.tran 10
.plot V(out)
.system %QPOST% "%DECK%" -r "%RAWFILE%" -o "QPostResult.out"
* measure avg, rms, max, min and pp of V(out)
.meas Vavg AVG V(out)
.meas Vrms RMS V(out)
.meas Vmax MAX V(out)
.meas Vmin MIN V(out)
.meas Vpp PP V(out)
.end
```



.temp

Temperature

# .temp – Circuit Temperature in DC analysis

Qspice : .temp - dc directive.qsch

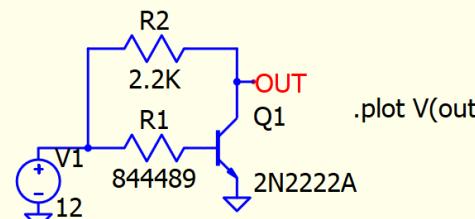
- .temp in DC analysis

- Method 1

- Run DC Sweep (.dc) analysis with sweep parameter as TEMP (circuit temperature)

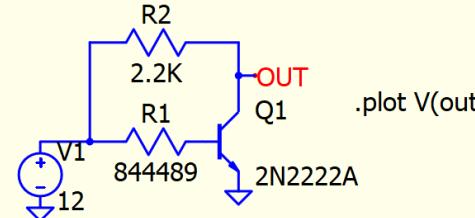
- Method 2

- Run Bias Point Analysis (.op) with step TEMP parameter (.step)



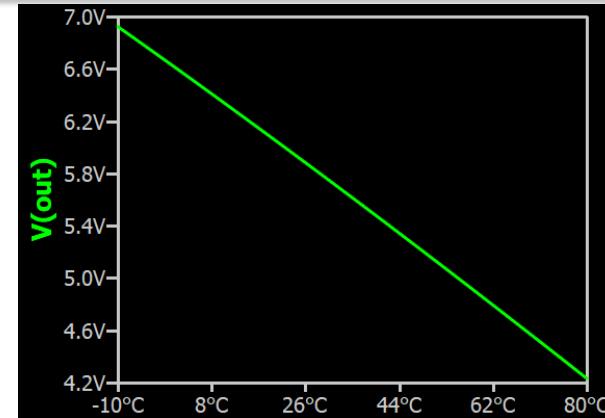
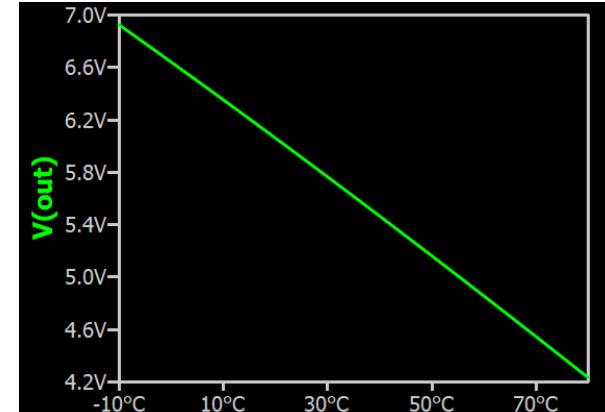
```
.model 2N2222A NPN(IS=1E-14 VAF=100 BF=200 IKF=0.3  
+XTB=1.5 BR=3 CJC=8E-12 CJE=25E-12 TR=100E-9 TF=400E-12  
+ITF=1 VTF=2 XTF=3 RB=10 RC=.3 RE=.2)
```

```
.dc TEMP -10 80 .1      .op  
.step TEMP -10 80 1
```



```
.model 2N2222A NPN(IS=1E-14 VAF=100 BF=200 IKF=0.3  
+XTB=1.5 BR=3 CJC=8E-12 CJE=25E-12 TR=100E-9 TF=400E-12  
+ITF=1 VTF=2 XTF=3 RB=10 RC=.3 RE=.2)
```

```
.dc TEMP -10 80 .1      .op  
.step TEMP -10 80 1
```



# .temp – Circuit Temperature in Transient analysis

Qspice : .temp - tran directive.qsch

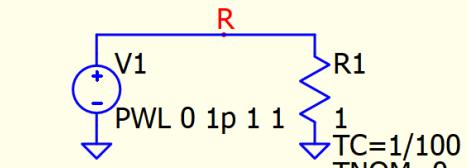
- .temp in Transient

- Method 1

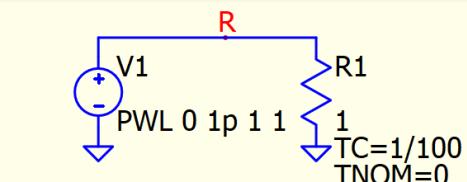
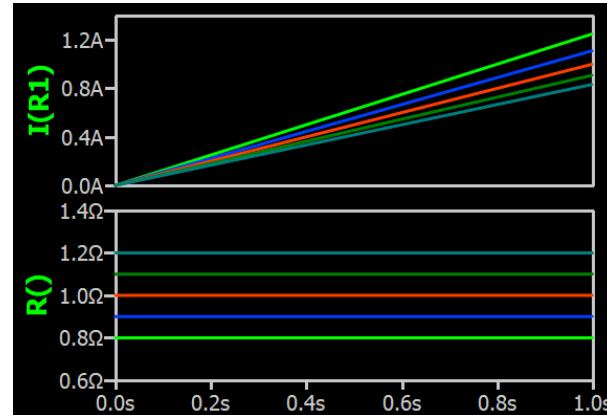
- Run Circuit Temperature (.temp) analysis

- Method 2

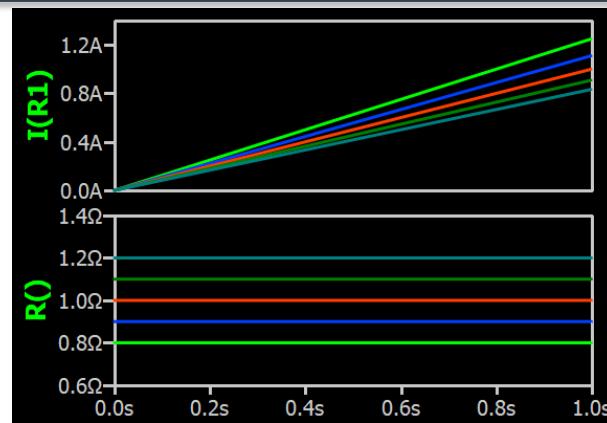
- Run with step TEMP parameter (.step)



```
# .temp syntax  
.temp -20 -10 0 10 20  
.tran 1  
.func R() V(R)/I(R1)  
.plot R()  
.plot I(R1)  
  
# .step syntax  
.step param TEMP -20 20 10  
.tran 1
```



```
# .temp syntax  
.temp -20 -10 0 10 20  
.tran 1  
.func R() V(R)/I(R1)  
.plot R()  
.plot I(R1)  
  
# .step syntax  
.step param TEMP -20 20 10  
.tran 1
```



.tran  
**Non-Linear Transient  
Analysis**

# Syntax of .tran (Non-Linear Transient Analysis)

- Two syntax of .tran Non-Linear Transient Analysis

- .tran TSTOP [UIC]
  - If MAXSTEP is required in this syntax, use *.options MAXSTEP* instead
- .tran IGNORED TSTOP [TSTART [MAXSTEP]] [UIC]
  - Recommend to fill 0 at IGNORED. This syntax allows to set start recording time and maxstep
  - If need to limit .qraw file size, consider to specify Tstart and limit data to disk .qraw file size

## 1. Specify only the stop time

Syntax: .tran TSTOP [UIC]

Name	Description	Units
TSTOP	Total amount of time to simulate	s
UIC	Use initial conditions instead of solving for the initial bias point(SKIPBP)	

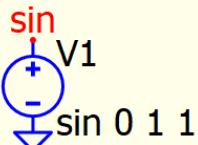
## 2. Traditional Berkeley Syntax

Syntax: .tran IGNORED TSTOP [TSTART [MAXSTEP]] [UIC]

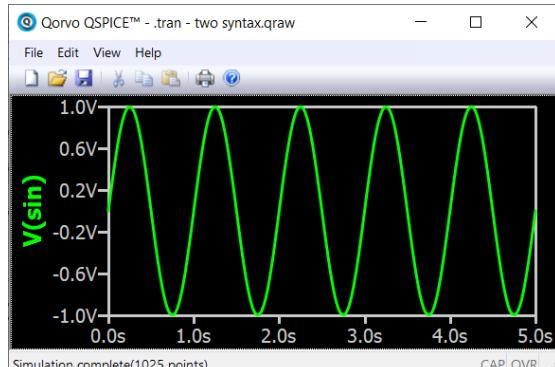
Name	Description	Units
IGNORED <sup>2</sup>	An ignored value	s
TSTOP	Total amount of time to simulate	s
TSTART	Time to start recording waveform data to disk	s
MAXSTEP	Maximum time step size to allow	s
UIC	Use initial conditions instead of solving for the initial bias point(SKIPBP)	

# .tran (Non-Linear Transient Analysis) : Syntax Examples

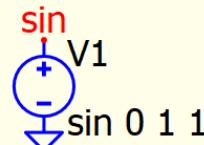
## Only Stop Time Syntax



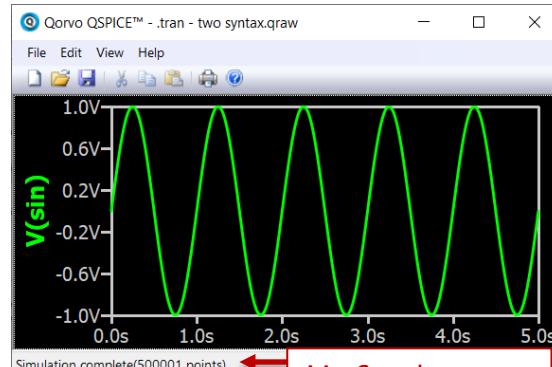
```
sin  
V1  
+  
- sin 0 1 1  
.tran 5 .options MAXSTEP=10μ  
.tran 0 5 2 10μ  
.plot V(sin)
```



## Only Stop Time Syntax with .option MAXSTEP

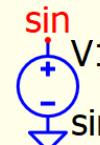


```
sin  
V1  
+  
- sin 0 1 1  
Use .option to set MaxStep  
.tran 5 .options MAXSTEP=10μ  
.tran 0 5 2 10μ  
.plot V(sin)
```

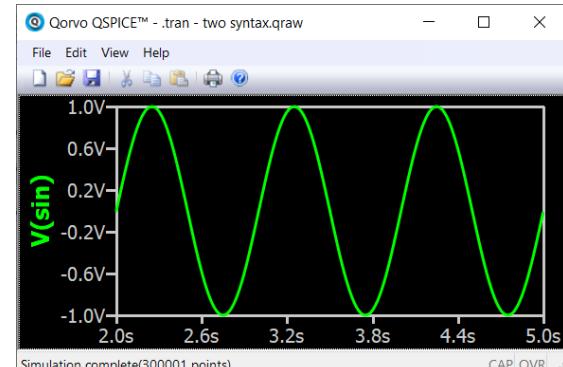


MaxStep increases  
simulation points

## Traditional Berkeley Syntax



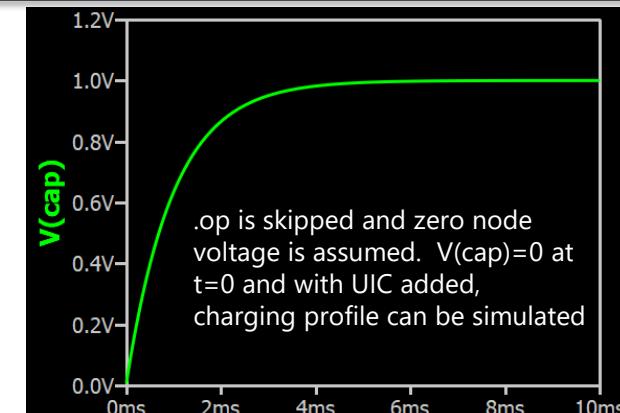
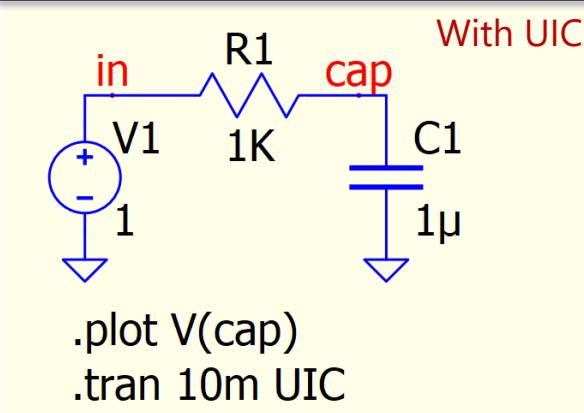
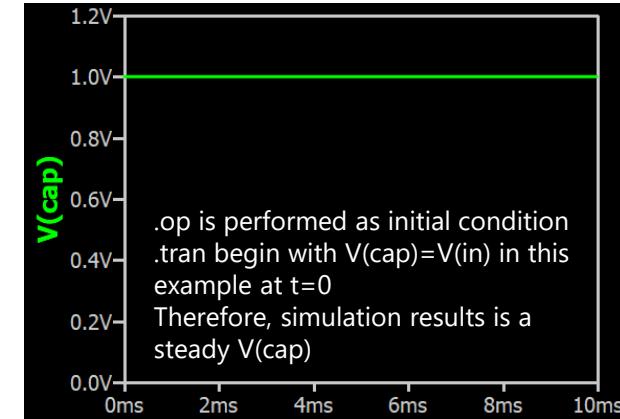
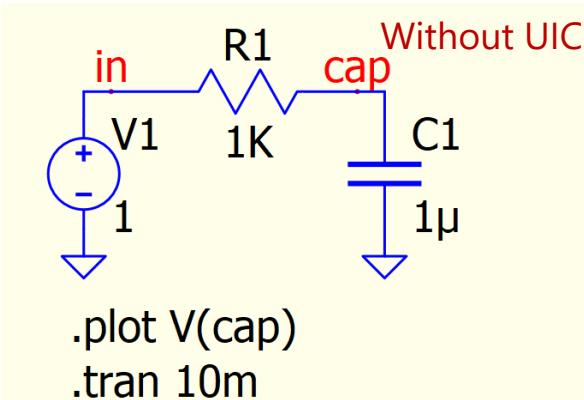
```
sin  
V1  
+  
- sin 0 1 1 .plot V(sin)  
.tran 5 .options MAXSTEP=10μ  
.tran 0 5 2 10μ  
.tran Ignore Tstop Tstart MaxStep
```



# .tran – UIC (Use Initial Condition)

Qspice : .tran - UIC.qsch

- UIC
  - Use Initial Condition
  - A DC operating point analysis (.op) is performed before starting the transient analysis (.tran). This directive suppresses this initialization
  - The node voltage is taken as zero if not specified
  - However, UIC is not a particularly recommended feature of SPICE (refer to UIC Help in LTspice), and the reason is explained in next page
  - An alternative method for this example without UIC is to add initial condition directive  
.IC V(cap)=0



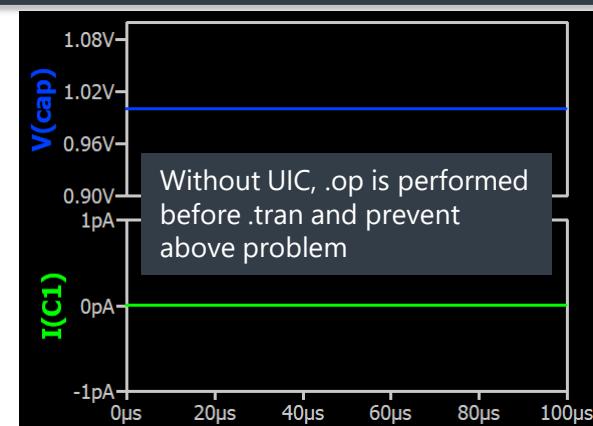
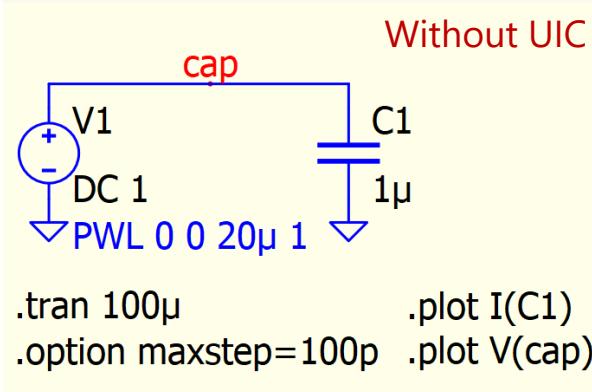
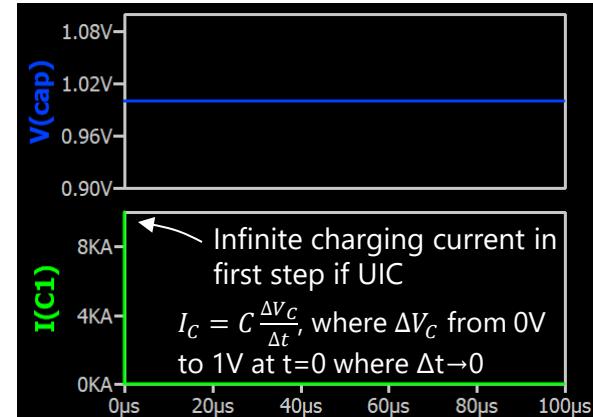
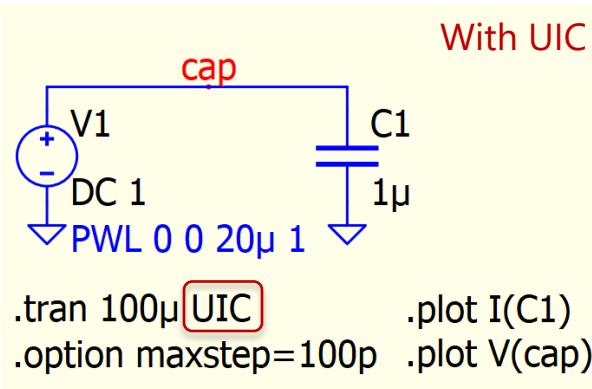
# .tran – UIC (Use Initial Condition) and its Limitation

Qspice : .tran - UIC limitation.qsch

- UIC Limitation

- Skipping the DC operating point analysis leads to nonphysical initial conditions and may introduce difficulty in simulation

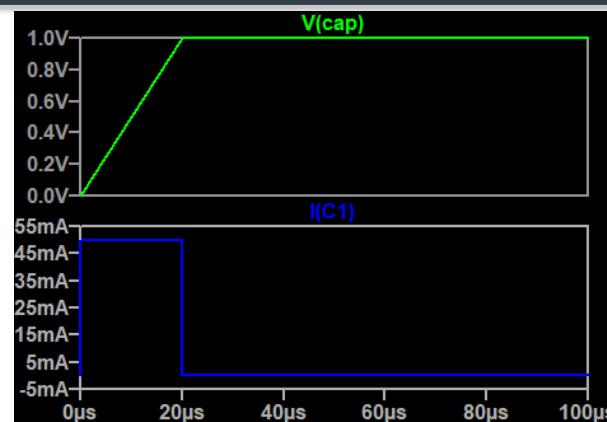
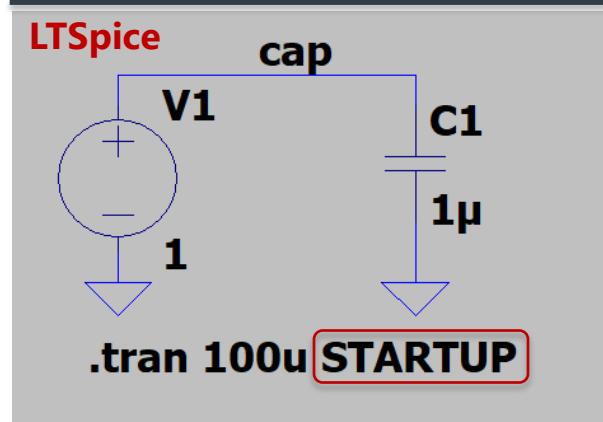
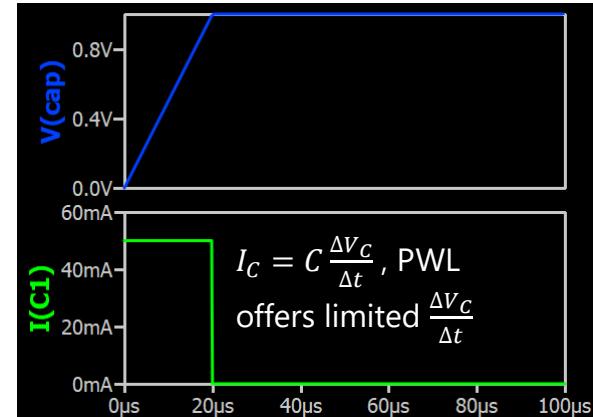
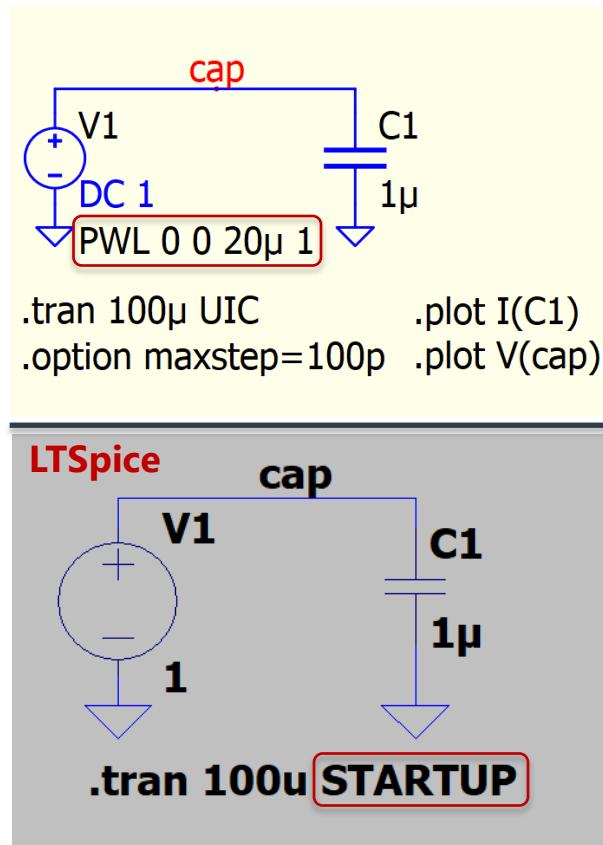
- For example, voltage source in parallel to a capacitor required infinite current to charge in the first time step, which may result in a “time step too small” convergence failure



# .tran – STARTUP (Not in Qspice)

LTspice : .tran - startup.asc

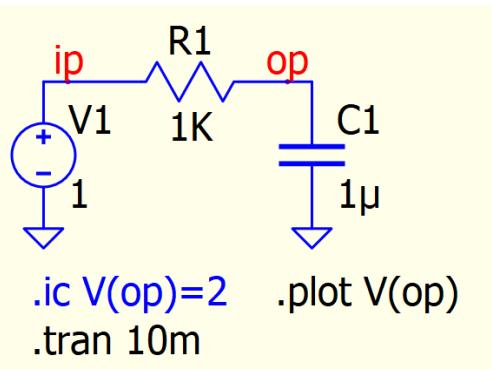
- Startup
  - This is not a modifier directive in Qspice but in LTspice
    - It is needed for many of the switcher models in LTspice according to Mike Engelhardt explanation
  - Startup modifier means independent source should be ramped on during the first 20us of the simulation
    - In my test, only DC source with ramp added, but not PULSE, SINE, PWL etc...
  - An equivalent approach in Qspice is to change voltage source from DC to **PWL 0 0 20u [VDC]**



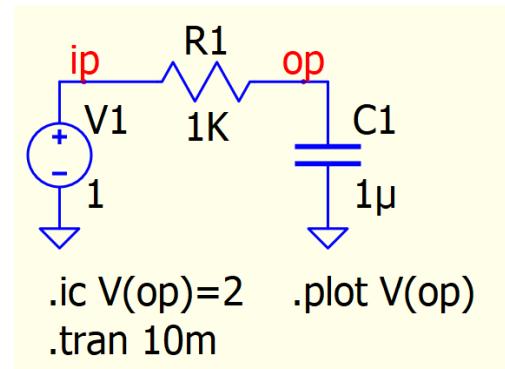
# .tran (Non-Linear Transient Analysis) : .IC/IC Without UIC

Qspice : .tran - UIC and IC.qsch

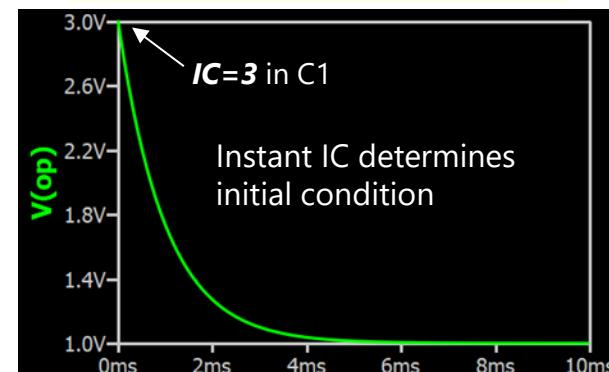
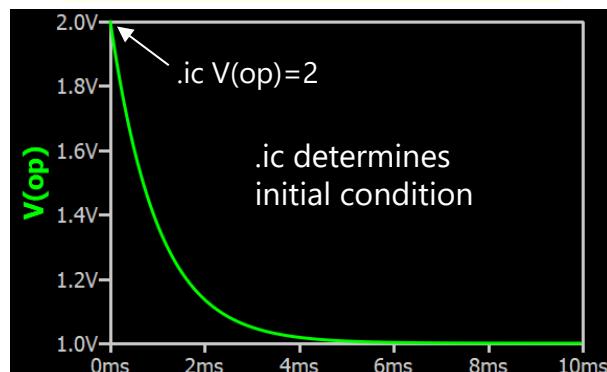
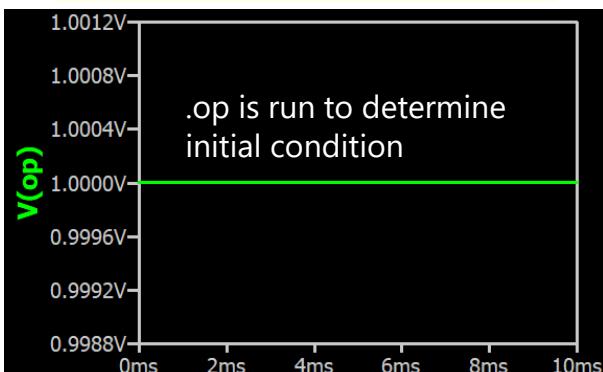
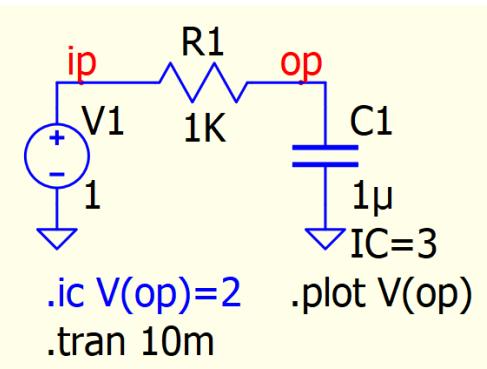
## Without UIC



## .IC Without UIC



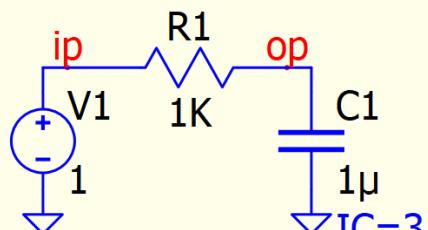
## Instance IC W/o UIC



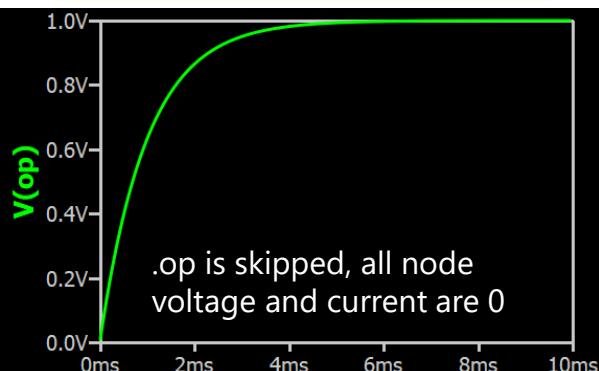
# .tran (Non-Linear Transient Analysis) : .IC/IC With UIC

Qspice : .tran - UIC and IC.qsch

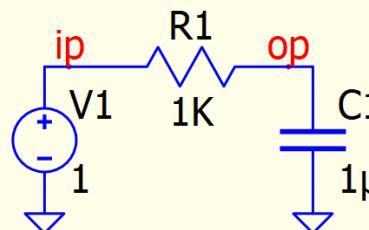
## With UIC



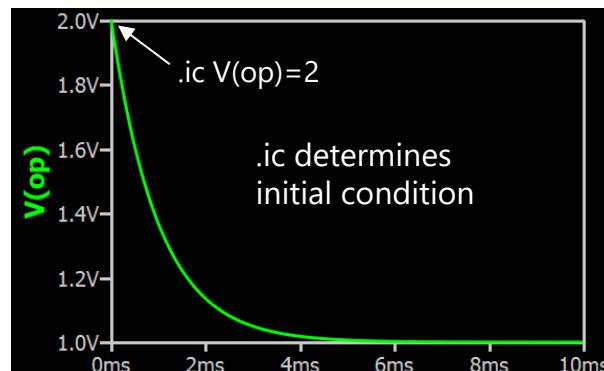
```
.ic V(op)=2  
.plot V(op)  
.tran 10m uic
```



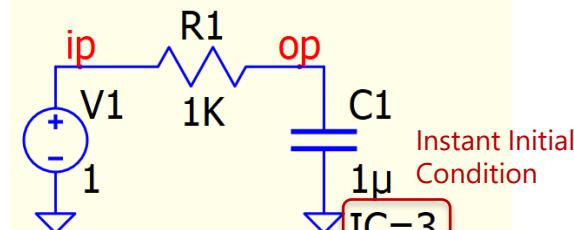
## .IC With UIC



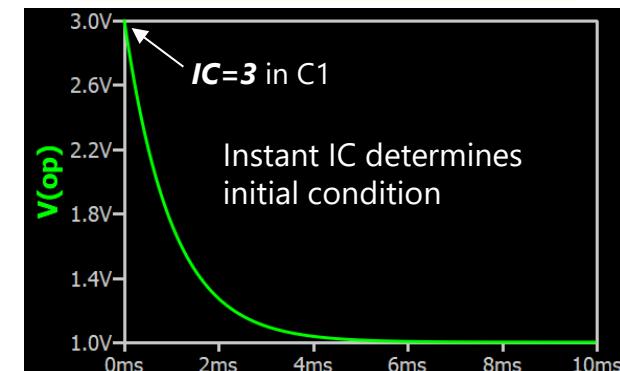
```
.ic V(op)=2  
.plot V(op)  
.tran 10m uic
```



## Instance IC With UIC



```
.ic V(op)=2  
.plot V(op)  
.tran 10m uic
```



# Importance of initial condition for .tran

Qspice : .tran - importance - example1.qsch

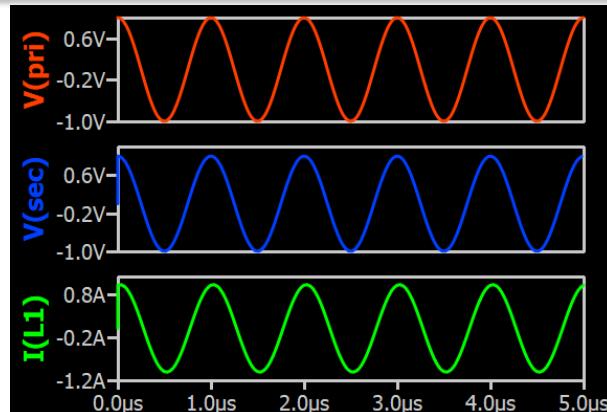
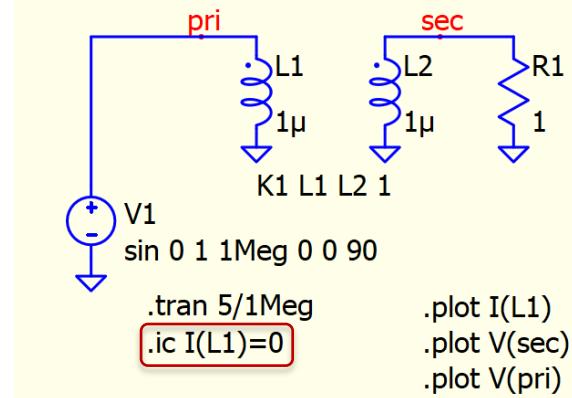
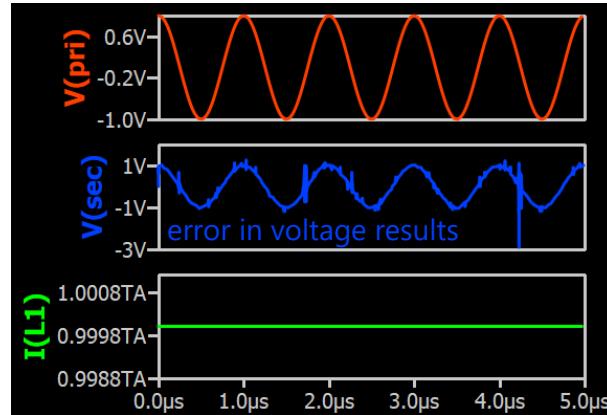
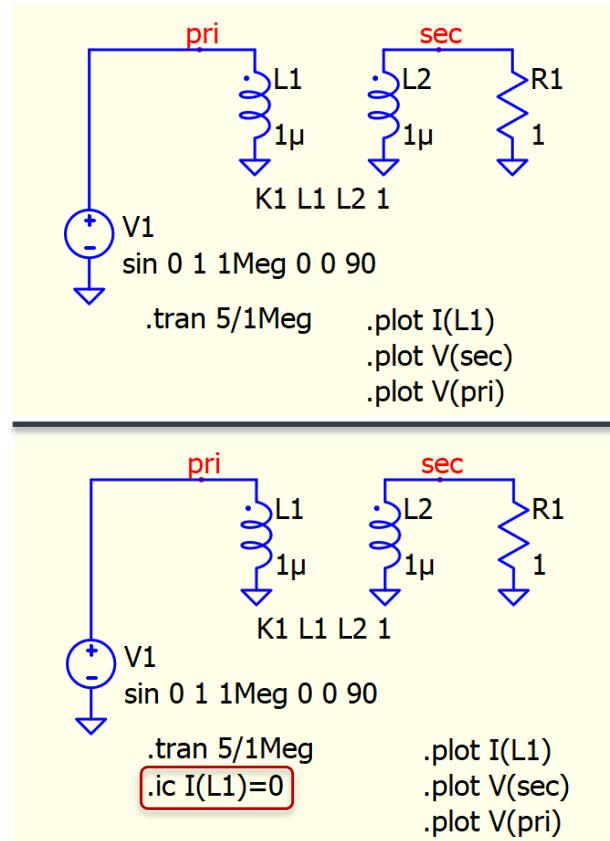
- Importance of Initial Condition for .tran

- Example 1

- In a coupled inductor circuit with a voltage source in parallel with the primary inductor, there may be a significant DC current during .op analysis. This TA level ( $10^{12}$ ) of current can lead to errors in voltage/current results

- Solution #1

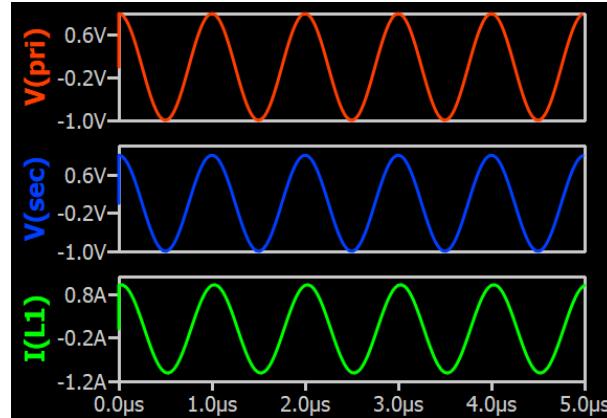
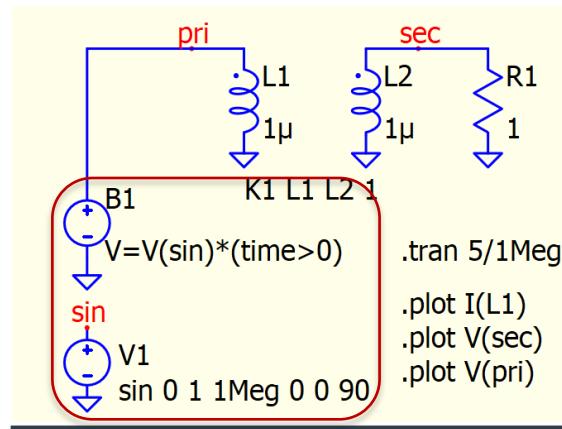
- Set an initial condition using the .ic directive



# Importance of initial condition for .tran

Qspice : .tran - importance – example2.qsch

- Importance of Initial Condition for .tran
  - Solution #2
    - Setup a B-source with  $(time > 0)$  to only activity source when simulation time  $> 0$  (i.e. 0V in DC condition)



**Batch mode**  
**Command**

# Qspice Execution Files

---

- Qspice execution files
  - Directory (default installation) : C:\Program Files\QSPICE
  - Schematic Capture and Waveform Viewer Program (HELP > Waveform Viewer)
    - Execution file : QUX.exe
    - Function #1 : Convert .qsch schematic to .cir
    - Function #2 : Export data from data file .qraw
    - Function #3 : Plot data file .qraw or .csv
  - QSPICE Simulator (HELP > Simulator)
    - Execution file : QSPICE64.exe [Enable Fast (less accurate) Math]
    - Execution file : QSPICE80.exe
    - Function : Run simulation from .cir
  - Post Processor (HELP > Post Processor)
    - Execution file : QPOST.exe
    - Function : Execute .meas and .four from .qraw

# Batch command basic workflow

## Qspice : Qspice\_Batch\_Command\_LPFexample.bat / LPF Circuit.qsch

- Batch command workflow

- Run CMD in Windows, in Command Prompt
- Set path for Qspice program
  - path C:\Program Files\QSPICE\
- Set variable name for working folder
  - set Qname=LPF Circuit
  - set Qpath=C:\QspiceKSKelvin\Qspice Batch
- Goto schematic .qsch directory
  - cd %Qpath%
- Convert .qsch to .cir (netlist)
  - QUX -Netlist "%Qname%.qsch"
- Run Qspice simulation for .qraw
  - QSPICE64 -binary "%Qname%.cir"
  - QSPICE64 -ascii "%Qname%.cir" -r "%Qname%-ascii.qraw"
- Export data from .qraw to .csv
  - QUX -Export "%Qname%.qraw" V(mid),V(out)
- Post Process .meas and .four
  - QPOST "%Qname%.cir" -o "%Qname%.out"

```
C:\ Command Prompt
Microsoft Windows [Version 10.0.19045.3086]
(c) Microsoft Corporation. All rights reserved.

C:\Users\          >path C:\Program Files\QSPICE\
C:\Users\          >set Qname=LPF Circuit
C:\Users\          >set Qpath=C:\QspiceKSKelvin\Qspice Batch
C:\Users\          >cd %Qpath%
C:\QspiceKSKelvin\Qspice Batch>QUX -Netlist "%Qname%.qsch"

C:\QspiceKSKelvin\Qspice Batch>LPF Circuit.cir
C:\QspiceKSKelvin\Qspice Batch>QSPICE64 -binary "%Qname%.cir"
C:\QspiceKSKelvin\Qspice Batch\LPF Circuit.cir

Total elapsed time: 0.0049635 seconds.

C:\QspiceKSKelvin\Qspice Batch>QUX -Export "%Qname%.qraw" V(mid),V(out)

C:\QspiceKSKelvin\Qspice Batch>LPF Circuit.csv
C:\QspiceKSKelvin\Qspice Batch>QPOST "C:\QspiceKSKelvin\Qspice Batch\%Qname%.cir"
.meas fc find frequency when db(mag(v(out)))=-3:
(    242610,      0)    242610

C:\QspiceKSKelvin\Qspice Batch>
```

This PC > OS (C:) > QspiceKSKelvin > Qspice Batch

Name
LPF Circuit.cir
LPF Circuit.csv
LPF Circuit.qraw
LPF Circuit.qsch
Qspice_Batch_Command_LPFexample.bat

# QUX.exe : Netlist a Schematic (.qsch)

- Syntax for QUX buildtimestamp
  - QUX.exe -buildtimestamp

```
C:\Program Files\QSPICE>QUX.exe -buildtimestamp  
C:\Program Files\QSPICE>Build Nov 3 2023 09:11:08
```

- Syntax for –Netlist (\*\* beware, -Netlist is case sensitive)

- QUX.exe -Netlist <schematicfile> [-stdout]

- <schematicfile> : name (+path) of a .qsch schematic, adds " " quotation for filename
- If "-stdout" is not specified, the name of the netlist(.cir) file is computed from the name of the input .qsch file
- [-stdout] : the netlist is printed on the console instead of to a file (not recommended since QSPICE employs a character set that most terminals can't handle)

```
C:\QspiceKSKelvin\Qspice Batch>QUX -Netlist "%Qname%.qsch" -stdout  
C:\QspiceKSKelvin\Qspice Batch>* LPF Circuit.qsch  
L1 in mid 1  
C1 mid 0 1  
R1 out 0 1  
V1 in 0 AC 1  
L2 mid out 1  
.ac dec 100 10K 1Meg  
.plot V(mid) V(out)  
.MEAS fc FIND frequency WHEN db(mag(V(out)))=-3  
.end
```

# QUX.exe : Export Datafile (.qraw)

---

- Syntax for -Export
  - QUX.exe -Export <datafile> <expr1[,expr2[,...]]> [Npoints] [CSV|SPICE|ASCII] [-stdout]
    - <datafile> : name of a .qraw file
    - <expr1[,expr2[,...]]> : expressions of data to extract
      - No space are allowed in the expression
      - Comma-separated expressions
    - [Npoints] : number of equally-spaced data points to extract
      - Default Npoints=1000
      - Npoints=1e308 or Npoints=all : all datapoints are extracted, waveform is not interpolated
    - [CSV|SPICE|ASCII]
      - CSV : Comma-Separated Value file
      - SPICE : .qraw in binary
      - ASCII : .qraw in ASCII
    - [-stdout] : extracted data is printed on the console instead of to a file
    - Example
      - QUX -Export "<filepath filename>" expr,expr2 all ascii ↵ no quotation mark is required for [Npoints] and [CSV|SPICE|ASCII]

# QUX.exe : Plot Datafile (.qraw, .csv)

---

- Syntax for Plot
  - QUX.exe <datafile>
    - <datafile> : name of a .qraw or .csv file
    - Example : to plot a QUX-plot.qraw in C:\KSKelvinQspice\
      - path C:\Program Files\QSPICE\
      - QUX "C:\KSKelvinQspice\QUX-plot.qraw"

# QSPICE64.exe and QSPICE80.exe : QSPICE Simulator

---

- Syntax for output data .qraw name same as netlist .cir name
  - QSPICE64.exe -binary <netlistname> : Binary file format for output data .qraw
  - QSPICE64.exe -ascii <netlistname> : Ascii file format for output data .qraw
  - If 80 bit is used, change QSPICE64 to QSPICE80
- Syntax for specify output data .qraw name
  - QSPICE64.exe -[ascii/binary] <netlistname> -r <path> : specify the name of output data file
  - Example
    - set Qname=LPF Circuit
    - QSPICE64 -ascii "%Qname%.cir" -r "%Qname%-ascii.qraw"
- Syntax to directs the .qraw output to null (not saving a .qraw)
  - QSPICE64.exe <netlistname> -r NUL
    - This special usage is for user who write C++ dll datalogger and not prefer a .qraw to generate when simulating with batch mode

# .qraw Binary Data format

```
Binary-binary.qraw x
1 Title: * Binary.qsch
2 Date: Sun Nov 5 21:41:33 2023
3 Plotname: DC Transfer Characteristic
4 Flags: real
5 Abscissa: 1.000000000000000e+00 5.000000000000000e+00 lin
6 No. Variables: 4
7 No. Points: 3
8 Command: QSPICE64, Build Nov 3 2023 09:29:29
9 .param temp=27
10 Variables:
11   0 V1 voltage
12   1 V(a) voltage
13   2 I(V1) current
14   3 P(V1) power
15 Binary:
16 NUL NUL NUL NUL NUL NUL 83 NUL NUL NUL NUL NUL NUL NUL NUL
```

```
Binary-binary.qraw x Binary-ascii.qraw x
1 Title: * Binary.qsch
2 Date: Sun Nov 5 21:41:38 2023
3 Plotname: DC Transfer Characteristic
4 Flags: real
5 Abscissa: 1.000000000000000e+00 5.000000000000000e+00 lin
6 No. Variables: 4
7 No. Points: 3
8 Command: QSPICE64, Build Nov 3 2023 09:29:29
9 .param temp=27
10 Variables:
11   0 V1 voltage
12   1 V(a) voltage
13   2 I(V1) current
14   3 P(V1) power
15 Values:
16 0 1.000000000000000e+00
17 1 1.000000000000000e+00
18 1 0.000000000000000e+00
19 1 0.000000000000000e+00
20 1 3.000000000000000e+00
21 1 3.000000000000000e+00
22 1 0.000000000000000e+00
23 1 0.000000000000000e+00
```

Binary vs Ascii

Newline </n>

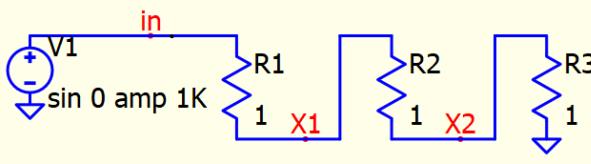
00000130	76	6f	6c	74	61	67	65	0a	09	32	09	49	28	56	31	29	voltage..2.I(V1)
00000140	09	63	75	72	72	65	6e	74	0a	09	33	09	50	28	56	31	.current..3.P(V1)
00000150	29	09	70	6f	77	65	72	0a	42	69	6e	61	72	79	3a	0a ).power.Binary:.	
00000160	00	00	00	00	00	00	f0	3f	00	00	00	00	00	00	f0	3f	.....ð?.....ð?
00000170	00	Binary Format : Float 64	00	00	00	00	00	00	00	00	00	00	00	00	00	00	.....
00000180	00	00	00	00	00	00	00	40	00	00	00	00	00	08	40	.....@.....@	
00000190	00	00	00	00	00	00	00	00	00	00	00	00	00	00	00	00	.....
000001a0	00	00	00	00	00	00	14	40	00	00	00	00	00	14	40	.....@.....@	
000001b0	00	00	00	00	00	00	00	00	00	00	00	00	00	00	00	00	.....

# QPOST.exe : Post Processor to execute .meas and .four

- Syntax for Qpost.exe

- Qpost <netlistname> -o <consoleoutput>

- This will write .meas and .four results into a file for the console output
  - This result is equivalent in Qspice Post Process Output Window after Simulation is run



.tran 10m

.step param amp 1 2 0.1      .param amp=1

.meas @amp param amp ←

If works on .out

alone, recommend

to add a .meas for

.step param

path C:\Program Files\QSPICE\

set Qname=resistor\_network

set Qpath=C:\QspiceKSKelvin\01 User Guide and  
Script\03 Qspice Reference Guide\Batch  
Command\Qpost

cd %Qpath%

QUX -Netlist "%Qname%.qsch"

QSPICE64 -binary "%Qname%.cir"

QPOST "%Qname%.cir" -o "%Qname%.out"

Line	Content
1	.meas @amp param amp:
2	0 1
3	1 1.1
4	2 1.2
5	3 1.3
6	4 1.4
7	5 1.5
8	6 1.6
9	7 1.7
10	8 1.8
11	9 1.9
12	10 2
13	.meas vrms rms v(in):
14	0 0.707107
15	1 0.777817
16	2 0.848528
17	3 0.919239
18	4 0.989949
19	5 1.06066
20	6 1.13137
21	7 1.20208
22	8 1.27279
23	9 1.34335

# QPOST.exe : Post Processor to execute .meas and .four

---

- Syntax for Qpost.exe
  - **Qpost <netlistname> -r <.qraw file> -o <consoleoutput>**
    - This is to load .meas/.four from a netlist file (\*.cir)
    - e.g. QPOST "%Qname%.cir" -r "%Qname%.qraw" -o "%Qname%.out"
  - **Qpost <meas script> -r <.qraw file> -o <consoleoutput>**
    - This is to load .meas/.four from a separated text file contains these directives
    - e.g. QPOST "meas script.txt" -r "%Qname%.qraw" -o "%Qname%.out"

# Example : Save data file (.qraw) into different directory

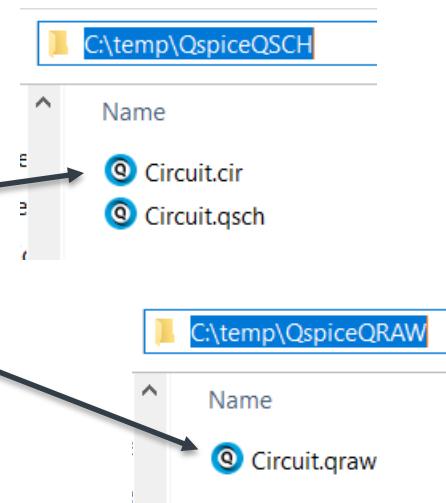
- Save data file (.qraw) into different directory

- This example assume

- Qspice installed in C:\Program Files\QSPICE\
    - Schematic file in C:\temp\QspiceQSCH\
    - Circuit schematic named Circuit.qsch
    - You want the data file to be stored in C:\temp\QspiceQRAW\

- Batch Command in CMD

- path C:\Program Files\QSPICE\
    - cd C:\temp\QspiceQSCH\
    - QUX -Netlist "Circuit.qsch"
    - QSPICE64 -binary "Circuit.cir" -r "C:\temp\QspiceQRAW\Circuit.qraw"
    - QUX "C:\temp\QspiceQRAW\Circuit.qraw"



Curly Braces {}

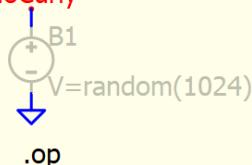
# Curly Braces { }

## Qspice : Curly Braces with Random.qsch

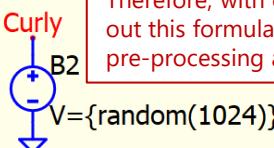
- Curly Braces { }
  - In Pspice and Ltspice, the curly braces always meant evaluate the contents before proceeding to the simulation
  - But Qspice tries to figure out what can be solved before the simulation by itself and remove the necessary of using curly braces { }
  - Therefore, Qspice can call to use a parameter or a formula without curly braces { }
  - But user can still observe effect of curly braces in Qspice, an example is B-source with random(x) [with argument] function in B-source

Curly Braces : Evaluate the contents before proceeding to the simulation

NoCurly



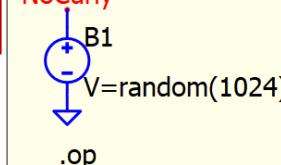
Curly



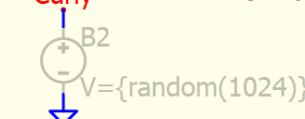
Pre-proceeded random() must be no input argument (i.e. expected 0 argument)  
Therefore, with curly braces, simulator figured out this formula is not expected to be solved in pre-processing and return a warning to user

Curly Braces : Evaluate the contents before proceeding to the simulation

NoCurly



Curly



Example of Correct usage of random with argument  
No warning anymore!

Output Window

Warning: Expected 0 argument(s) to random(), but it was called with 1 arguments.  
C:\KSkelvinQspice\01 User Guide and Script\03 Command Reference Guide\Curly Brac

Output Window

C:\KSkelvinQspice\01 User Guide and Script\03 Command Reference Guide\Curly Brac

\*\* Qspice determines it is not a pre-process function and continues to run after warning is returned

**Timestep too small**

# Timestep too small (.tran) – Guideline for .option

- Timestep too small (.tran) – Guideline for .option
  - **.option fastmath=0** : Disable fastmath (Qspice64.exe) and use Qspice80.exe. This will allow for 80-bit math calculations, which are more precise but may result in slightly slower simulations [Only for Qspice]
  - **.option trtol=7 method=gear** : If the model is a PSpice model, you may want to consider this. PSpice uses a different trtol and integration method. Be cautious as gear integration adds dampers to the circuit, suppressing oscillations but also introducing more errors. PSpice uses gear integration to prevent trapezoidal oscillations from trapezoidal integration
  - **.option gshunt=<value>** or **.option cshunt=<value>**, These options add conductance or capacitance from every node to the ground. They may help the simulation to converge by providing a ground path for high-frequency oscillations or numerical noise. Start by setting the value to 1e-12 or 1e-11
  - **.option maxstep=<value>** : Force a maximum timestep! Check your switching frequency or highest operating frequency and set a reasonable maxstep
  - **.option ITL4=<value>** : This option increases the number of transient iteration limits. Consider setting it to 100 or 1000 to see if there is any improvement by allowing more iterations
  - If a timestep too small occurs right after bias point analysis in .tran, it may be related to the initial voltage/current condition, and you may consider trying one of the following
    - Add UIC in .tran to skip the bias point analysis (.op)
    - Change the DC supply source to a PWL source, which ramps the voltage from 0 to the desired setpoint (e.g., change DC 10 to PWL 0 0 1n 10)
  - Investigate the 3rd party sub-circuit model, as it may consist of devices or equations that can cause the timestep to be too small. However, this troubleshooting normally requires a lot of experience
  - Some people may change **vntol**, **abstol**, **reltol**, but user needs deeper knowledge if they change these parameters in .option and I normally not to do that

# SPICE methods

---

- SPICE methods (All you need to know)
  - $[G][V]=[I]$ 
    - Node count is more important than component count
    - Avoid circuit elements represented as Thevenin or trans-Thevenin equivalents
  - Newton-Raphson Iteration
    - All I-V curves must be continuous in value and slope
    - Don't make anything more non-linear than necessary
    - Every nonlinear element should be bypassed with some capacitance
- \*\* SPICE-based and PWL-based
  - There are primarily two different circuit simulation concepts
    - SPICE-based (SPICE engine) : Qspice, PSpice, LTspice, TINA, NgSPICE etc...
    - PWL-based (Piecewise Linear simulation engine) : SIMPLIS, PSIM, NL5 etc...
  - SPICE-based models device equations. To simulate a near-ideal device, it generally requires more attention and a higher level of understanding of how to set up the simulation as it was not designed to handle discontinuity in I-V curves
  - PWL simulation engines model devices with piecewise linear functions and focus on resolving ideal switching circuit elements. They are designed to handle steep I-V curves

# Prevent discontinuities in the I-V curve

Qspice : SmoothIV-Diode-Epsilon.qsch | SmoothIV-SW-VH.qsch

- Remove discontinuities in the I-V curve
  - For behavioral diode model, consider the following
    - Ratio between Ron and Roff not too extreme
    - Add an instance parameter epsilon to smooth the transition
  - For switch, consider the following
    - Use -ve Vh for smoothly transition switch
    - Add ETA if further smoothen is required

