Qspice - General Reference Guide by KSKelvin

KSKelvin Kelvin Leung

Created on 8-4-2023
Last update on 1-21-2025

QSPICE

- QSPICE
  - Arthor : Mike Engelhardt
  - Download : https://www.qorvo.com/design-hub/design-tools/interactive/qspice
- Topic Included in this guideline
  - Shortcut Key
  - Hierarchical and Sub-circuit
  - Schematic Viewer
  - Waveform Viewer
  - Symbol Viewer
  - Simulation Technique

Part 1
Shortcut Key

Schematic Editor Keyboard Shortcuts

HELP > Schematic Capture > Schematic Editor > Keyboard Shortcuts

Schematic Editor Keyboard Shortcuts

Key	Command
␣ spacebar	Zoom to fit
B[1]	Behavioral source
C[1]	Capacitor
D[1]	Diode
E[1]	E-source
F	F-source
G[2]	Ground, G-source
H	H-source
I	Current Source
J[1]	JFET
L[1]	Inductor
M[1]	MOSFET
N	Place a net label
P	Toggle suspension of cross probing
Q[1]	Bipolar Transistor
R[1]	Resistor
S[1]	Voltage Controlled Switch
T[3]	Place Text
V[1]	Voltage Source
W	Start a wire
Y	Piezoelectric Crystal
Z[1]	MESFET

Ctrl+A	Draw an arc(graphical annotation)
Ctrl+B	Draw a box(graphical annotation)
Ctrl+C	Copy selected object(s) to clipboard
Ctrl+F	Find
Ctrl+G	Toggle display of grid dots
Ctrl+J	Stuff with jumper
Ctrl+L	Draw a line(graphical annotation)
Ctrl+M	Mirror selected object(s)
Ctrl+P	Print the schematic
Ctrl+R	Rotate selected object(s)
Alt+Ctrl+R	Rotate in 45° increments
Ctrl+V	Paste; whether CAD objects, bitmap, or text
Shift+Ctrl+V	Paste; whether CAD objects, bitmap, or text without incrementing net names
Ctrl+X	Cut
Ctrl+Y	Redo
Ctrl+Z	Undo
Ctrl+3	Draw a triangle(graphical annotation)
;	Toggle a text graphic's comment (or component's stuff) status.
F2	Toggle visibility of the Symbol and IP Browser pane.
F3	Toggle visibility of the Symbol Properties pane.
F4	Toggle visibility of the output console.
F5	Run the simulation.

1] Repeated depressions of the key cycles through different versions of the symbol.

2] Repeated depressions of 'G' cycles through different versions of the ground symbol and then G-source symbols.

3] The period key, '.', is accepted as a synonym for 'T'.

Symbol Editor and Waveform Viewer Keyboard Shortcuts

HELP > Waveform Viewer > Keyboard Shortcuts / HELP > Schematic Capture > Symbol Editor > Keyboard Shortcuts

Waveform Viewer Keyboard Shortcuts

Key	Command
Delete	Delete attached cursor if pointing to a readout or delete selected plot labels
F	Zoom to fit(all panes)
F4	Toggle visiblity of the console display
F5	Rerun the simulation
␣	Reload Plot configuration file
←	Move attached cursor left
→	Move attached cursor right
↑	Move attached cursor to next step } for .step
↓	Move attached cursor to previous step } for .step
Ctrl-A	Add a trace
Ctrl-C	Copy
Ctrl-D	Delete a plotting pane
Ctrl-F	Find
Ctrl-G	Turn Grid On/Off
Ctrl-V	Paste
Ctrl-P	Print
Ctrl-W	Add a plotting pane
Ctrl-X	Cut
Ctrl-Y	Redo
Ctrl-Z	Undo

Symbol Editor Keyboard Shortcuts

Key	Command
Ctrl-A	Draw an arc defined by three points
Shift-Ctrl-A	Draw an arc defined by four points
Ctrl-B	Draw a box(or a box for an image)
Ctrl-C	Copy selection(s) to clipboard
E	Draw an Ellipse
Ctrl-F	Find
Ctrl-L	Draw a line
Ctrl-M	Mirror selected objects
P	Place a pin
Ctrl-R	Rotate selected objects
T	Place a text attribute
Ctrl-V	Paste
Ctrl-X	Cut
Ctrl-Y	Redo
Ctrl-Z	Undo
Ctrl-3	Draw a triangle
F3	Toggle visibility of the Properties pane.

** Hold down ALT key (after simulation is ran and with an active waveform viewer)
Click on a node for voltage or device for current – Can prevent accidentally move wire or device
Click on a node and drag to another node – Can measure differentiate voltage, i.e. V(N001,N002)

** Hold down SHIFT key to probe current – invert the sign of a current quantity

Waveform Viewer Functions and Keywords (.func , .meas)

HELP > Waveform Viewer > Waveform Expressions

The following functions, constants, and keywords are recognized in expressions of waveform data.

Waveform Viewer Functions and Keywords

Syntax	Description
ABS(x)	Absolute value of x
ACOS(x)	Inverse cosine of x
ACOSH(x)	Inverse hyperbolic cosine of x
ARCCOS(x)	Inverse cosine of x
ARCCOSH(x)	Inverse hyperbolic cosine of x
ARCSIN(x)	Inverse sine of x
ARCSINH(x)	Inverse hyperbolic sine of x
ARCTAN(x)	Inverse tangent of x
ARCTANH(x)	Inverse hyperbolic tangent of x
ASIN(x)	Inverse sine of x
ASINH(x)	Inverse hyperbolic sine of x
ATAN(x)	Inverse tangent of x
ATAN2(x,y)[1]	Four quadrant inverse tangent of x
ATANH(x)	Inverse hyperbolic tangent of x
BUF(x)	x > .5 ? 1 : 0
CBRT(x)	Cube root of x
CEIL(x)	x rounded up to nearest integer
COS(x)	Cosine of x
COSH(x)	Hyperbolic cosine of x
COT(x)	Cotangent of x
D(x)	Derivative of x
DD(x)	Second derivative of x
D²(x)	Second derivative of x
E	2.7182818284590452354
ERF(x)	Error function of x
ERFC(x)	Complementary error function of x
EXP(x)	e raised to the x power
EXP10(x)	10 raised to the x power
FABS(x)	Absolute value of x
FLOOR(x)	x rounded down to nearest integer
FREQ	Frequency
FREQUENCY	Frequency
GAMMA(x)	Gamma function of x
HYPOT(x,y)	√(x²+y²)
IF(x,y,z)	(x > .5) ? y : z
ILOGB(x)	Unbiased exponent of x
IM(x)	Imaginary part of x
IMAG(x)	Imaginary part of x
INT(x)	x rounded to nearest integer
INV(x)	x > .5 ? 0 : 1
INVSQRT(x)	1÷√x
ISNAN(x)	One if x is not a number, otherwise zero
J	√-1
J0(x)	Zero order Bessel function of the first kind at x
J1(x)	First order Bessel function of the first kind at x
JN(x,n)	Nth order Bessel function of the first kind at x
K	1.380649e-23 J/°K
LGAMMA(x)	Log-gamma function of x
LIMIT(x,y,z)	Mutually intermediate value of x,y, and z
LN(x)	Natural logarithm of x
LOG(x)	Natural logarithm of x
LOG10(x)	Logarithm of x in base 10
LOG1P(x)	Natural logarithm of (x + 1)
LOG2(x)	Logarithm of x in base 2
LOGB(x)	LOG2(ABS(x))
MAG(x)	Absolute value of x
MAX(x,y)	Maximum of x and y
MAXMAG(x,y)	x or y with maximum magnitude
NAN	A value guaranteed to be not a number
MIN(x,y)	Minimum of x and y
MINMAG(x,y)	x or y with minimum magnitude
PH(x)	Phase of x
PHASE(x)	Phase of x
PI	3.14159265358979323846
POW(x,y)	x raised to the y power
POWN(x,y)	x raised to the nearest integer value of y
PWR(x,y)	Absolute value of x raised to the y power
PWRS(x,y)	x >= 0 ? x^y : -x^y
Q	1.602176487e-19 Coulomb
RE(x)	Real part of x
REAL(x)	Real part of x
RINT(x)	x rounded to the nearest integer
ROUND(x)	x rounded to the nearest integer
SGN(x)	Sign of x
SIGN(x)	Sign of x
SIN(x)	Sine of x
SINH(x)	Hyperbolic sine of x
SQRT(x)	Square root of x
TABLE(x,x1,y1,...)	Interpolate the table given as x1,y1, x2,y2,... at point x
TAN(x)	Tangent of x
TANH(x)	Hyperbolic tangent of x
TAUGRP(x)	Group delay of x
TBL(x,x1,y1,...)	Interpolate the table given as x1,y1, x2,y2,... at point x
TEMP	Circuit temperature
TG(x)	Group delay of x
TIME	Time
TRUNC(x)	Integer part of s
URAMP(x)	x > 0 ? x : 0
USTEP(x)	x > 0 ? 1 : 0
Y0(x)	Zero order Bessel function of the second kind at x
Y1(x)	First order Bessel function of the second kind at x
YN(x)	Nth order Bessel function of the second kind at x

[1] For complex data, the syntax is ATAN2(z). The meaning is ATAN2(IMAG(z),REAL(z)).

simulation variable (reserve word) : FREQ, FREQUENCY, TEMP, TIME

important constant (reserve word) : E, J, K, PI, Q

Function and Operators for Behavioral V and I Sources

HELP > Simulator > Device Reference > B. Behavioral Sources

Functions

Name	Description
abs(x)	Absolute value of x
acos(x)	arc cosine of x
arccos(x)	Synonym for acos()
acosh(x)	arc hyperbolic cosine of x
asin(x)	arc sine of x
arcsin(x)	Synonym for asin()
asinh(x)	Arc hyperbolic sine
atan(x)	Arc tangent of x
arctan(x)	Synonym for atan()
atan2(y,x)	Four quadrant arc tangent of y/x
atanh(x)	Arc hyperbolic tangent
buf(x)	1 if x > .5, else 0
ceil(x)	Integer equal or greater than x
cos(x)	Cosine of x
cosh(x)	Hyperbolic cosine of x
ddt(x)	Time derivitive x
delay(x,y)	x delayed by y
delay(x,y,z)[1]	x delayed by y, but store no more than z history
dlim(x,y,z)	x bounded by y which it asymptotically starts to approach at y+z as a first inverse order Laurent series
exp(x)	e to the x
floor(x)	Integer equal to or less than x
hypot(x,y)	sqrt(x^2 + y^2)
idt(x,y,z)	Time integral of x with initial condtion of y reset when z > .5
if(x,y,z)	If x > .5, then y else z
int(x)	Convert x to integer
inv(x)	0. if x > .5, else 1.
limit(x,y,z)	Intermediate value of x, y, and z
ln(x)	Natural logarithm of x
log(x)	Alternate syntax for ln()
log10(x)	Base 10 logarithm
max(x,y)	The greater of x or y
min(x,y)	The smaller of x or y
pow(x,y)	x^y
pwr(x,y)	abs(x)^y
pwrs(x,y)	sgn(x)*abs(x)^y
random(x)	Random number from 0. to 1. depending on the integer value of x. Interpolation between random numbers is linear for non-integer x.
sin(x)	Sine of x
sinh(x)	Hyperbolic sine of x
sqrt(x)	Square root of x
table (x,a,b,c,d,...)	Interpolate x from the look-up table given as a set of pairs of constant values.
tan(x)	Tangent of x.
tanh(x)	Hyperbolic tangent of x
ulim(x,y,z)	x bounded by y which it asymptotically starts to approach at y-z as a first inverse order Laurent series

Operators grouped in reverse order of precedence of evaluation

Operand	Description
&	Boolean AND
|	Boolean OR
>	True if expression on the left is greater than the expression on the right.
<	True if expression on the left is less than the expression on the right.
>=	True if expression on the left is greater than or equal the expression on the right.
<=	True if expression on the left is less than or equal the expression on the right.
+	Addition
-	Subtraction
*	Multiplication
/	Division
**	** / ^ Raise left hand side to power of right hand side. Same as '^'.
!	Boolean not the following expression.

Available Function in B source not listed

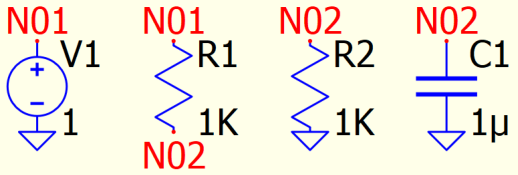
- Trunc(x) ; floor(x) ; int(x) : rounded down integer
- Rint(x) ; round(x) : rounded to nearest integer
- Ceil(x) : rounded up integer
- Ustep(x) : x > 0 ? 1 : 0
- Uramp(x) : x > 0 ? x : 0

Ctrl-Shift-V : Paste Netlist as Graphical Symbols on Schematic
Ctrl-Shift-V : Paste Without Increment Node Names

Netlist (Ctrl-C to Copy)

```
* C:\QspiceKSKelvin\01 U
V1 N01 0 1
R1 N01 N02 1K
R2 N02 0 1K
C1 N02 0 1µ
.end
```

Paste with Ctrl-Shift-V

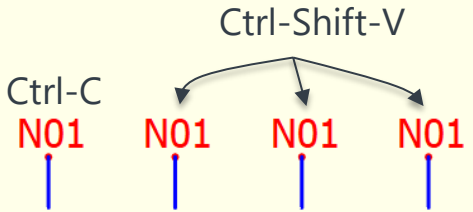


Revision History
01/13/2024 You can now paste fragments of a netlist copied to the clipboard as graphical symbols on a schematic by typing shift-control-V.

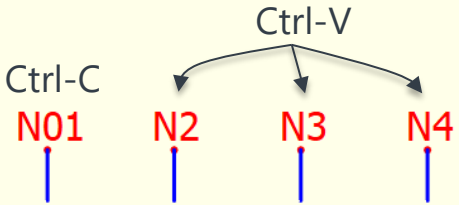
Paste with Ctrl-V

```
V1 N01 0 1
R1 N01 N02 1K
R2 N02 0 1K
C1 N02 0 1µ
```

- Increment node name
  - Shift+Ctrl+V pastes without increment node names.



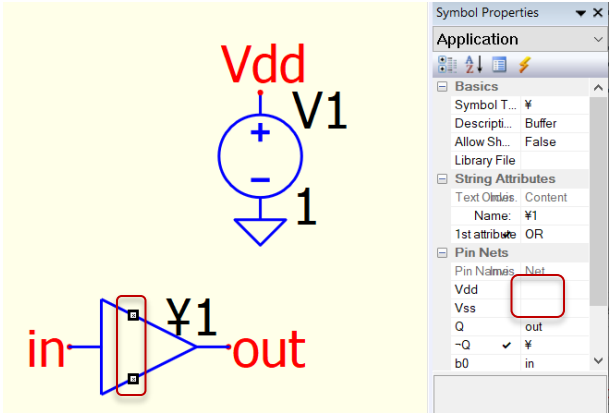
Without Increment with Ctrl-Shift-V

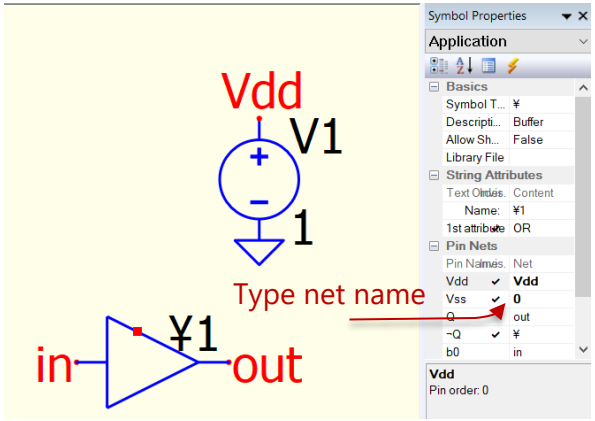


Increment with Ctrl-V

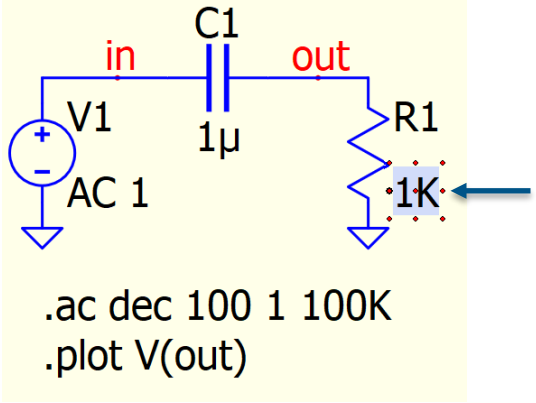
Hide Symbol Pins
ALT-Mousewheel – Delimited numbers

- Hide Symbol Pins
  - If you directly type a net name into the Net field, the corresponding pin in the symbol will be set to invisible and connected to that net name
  - This allows users to simplify the outlook of the schematic





- ALT-Mousewheel
  - Delimited numbers in schematic or netlist
  - Highlight number by selecting its (double click in schematic), key pressing ALT and rotate mousewheel

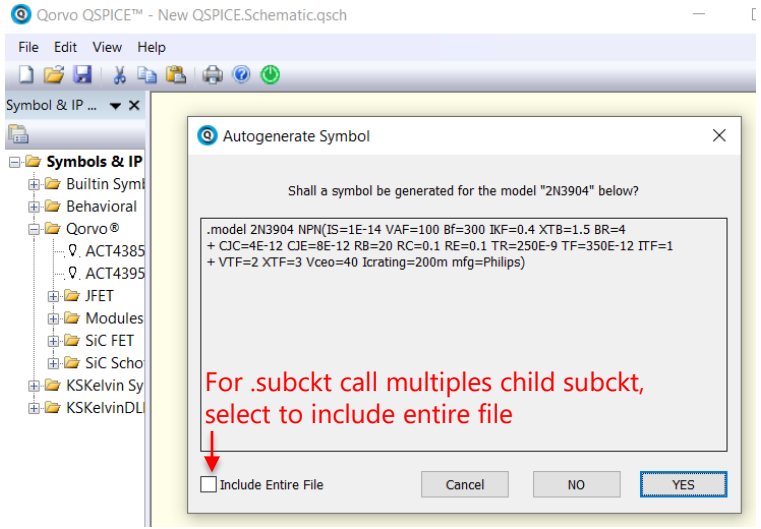


Quick Way to Create a Symbol (example from .model) Symbol (.qsym) to share for other projects

Step 1 : Assume you have a .model statement, which can copy and paste into Qspice and create a symbol in schematic, for example
.model 2N3904 NPN(IS=1E-14 VAF=100 Bf=300 IKF=0.4 XTB=1.5 BR=4 CJC=4E-12 CJE=8E-12 RB=20 RC=0.1 RE=0.1 TR=250E-9 TF=350E-12 ITF=1 VTF=2 XTF=3 Vceo=40 Icrating=200m mfg=Philips)

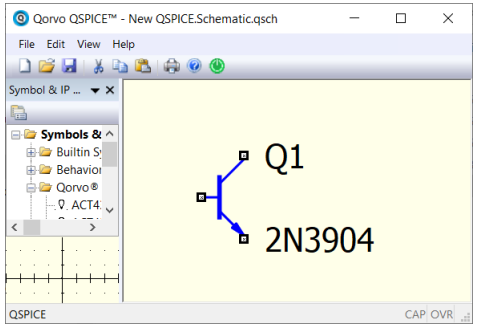
Step 2

- Copy entire .model text statement (or .subckt)
- In **Schematic Window**
  Ctrl-V to paste, Select **Yes** in **Autogenerate Symbol**



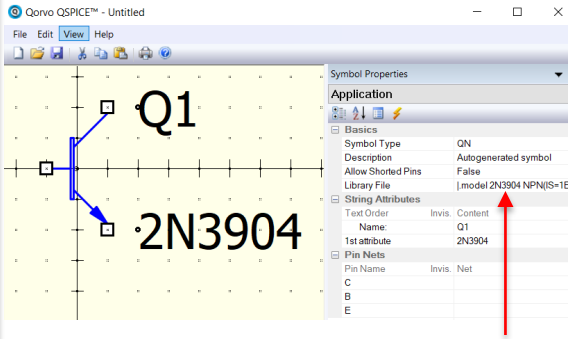
Step 3

- In **Schematic Window**, a symbol is created
- Select this symbol, **Copy-C** to copy
- In menu, **File** -> **New Symbol**



Step 4

- In **Symbol window**, **Ctrl-V** to paste
- In menu, **File** > **Save**
  Save into a .qsym (symbol file)

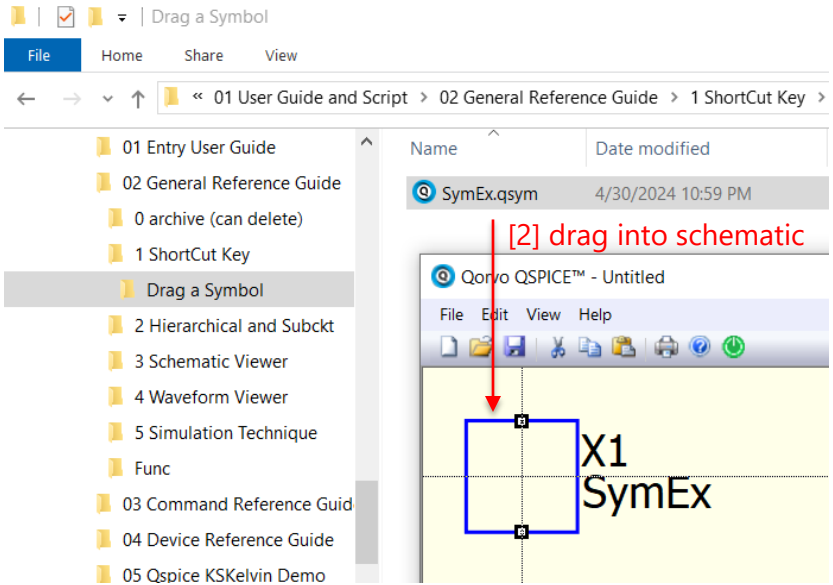


With this method, Qspice can embed .model or .subckt into symbol file, therefore, only share symbol file (.qsym) have all necessary detail to run simulation

Drag a Symbol into Schematic (Two Methods)

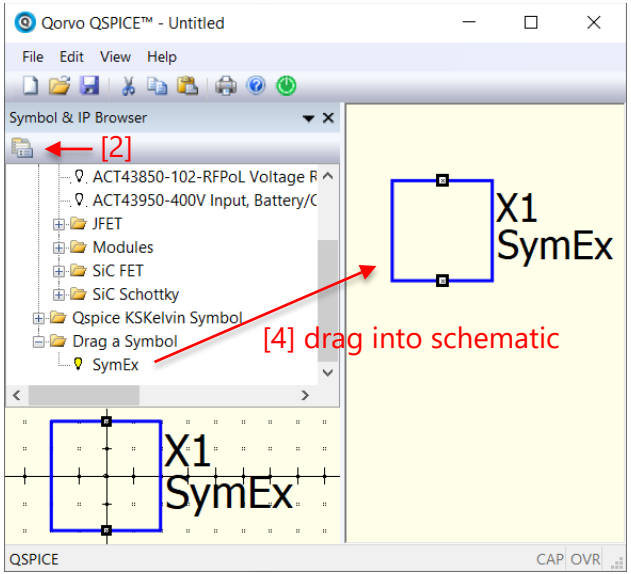
Method 1 : Drag from File Explorer

[1] Open the symbol folder using **File Explorer**
[2] **Drag** the symbol files (.qsym) into the schematic



Method 2 : Drag from Symbol & IP Browser

[1] Go to **View** > **Symbol & IP Browser** [or use the shortcut **F2**].
[2] Select the folder containing the symbol files (.qsym).
[3] If a folder with .qsym files is selected, a new tree path will be displayed.
[4] **Drag** the symbol from the **Symbol & IP Browser** into the schematic.

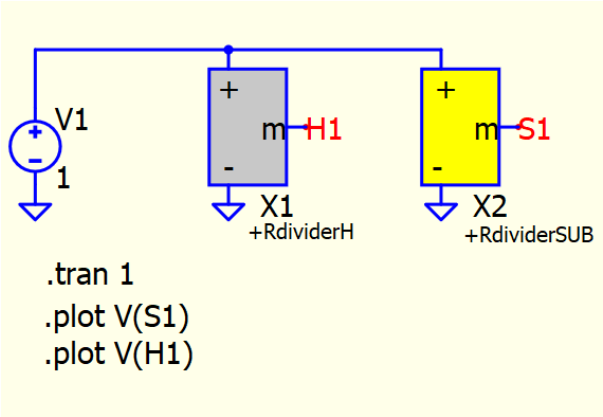


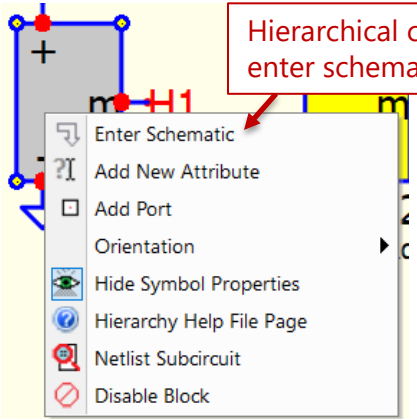
Part 2
Hierarchical and Sub-circuit

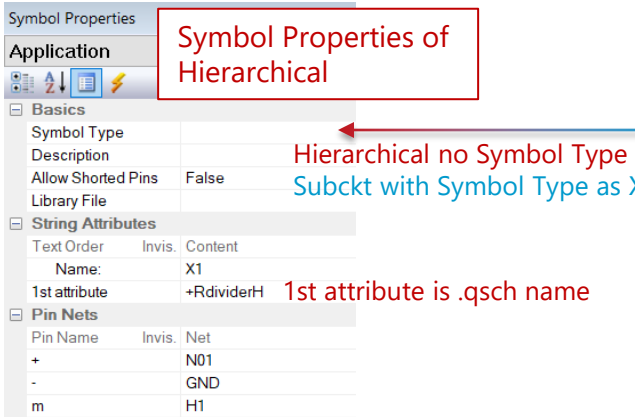
Hierarchical and Sub-circuit : Comparison

Qspice : parent - hierarchical and subckt.qsch | +RdividerH.qsch

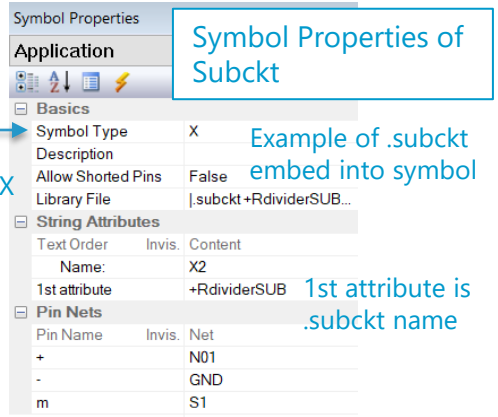
- Hierarchical and Sub-circuit
  - They are similar and both support by .qsym symbol, but two different concepts
  - Hierarchical
    - Call a child schematic (.qsch) for simulation
    - Waveform viewer can probe simulation result in child schematic directly
  - Sub-circuit (.subckt)
    - Call a sub-circuit (.subckt netlist) for simulation
    - Waveform viewer cannot probe simulation result directly
      - Results are stored, just offer to add trace indirectly
    - In Qspice, .subckt netlist can embed into a symbol in the field of library file (i.e. no need to share library file)







Hierarchical no Symbol Type
Subckt with Symbol Type as X



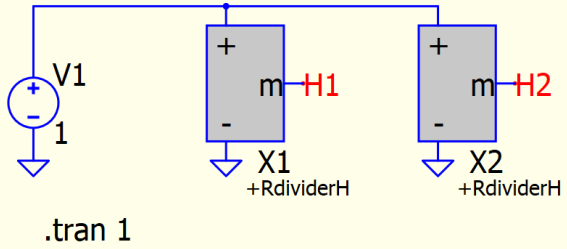
Example of .subckt embed into symbol

Hierarchical and Sub-circuit : Comparison

Qspice : parent - hierarchical and subckt (dual hierarchical/subckt).qsch

- Hierarchical and Sub-circuit
  - In netlist, both Hierarchical and Sub-circuit call .subckt syntax
  - Hierarchical
    - Child schematic is a .subckt in Parent netlist
    - Symbol calls this child schematic name
  - Sub-circuit (.subckt)
    - Each symbol calls an individual .subckt by naming its by adding prefix as Xnnn•<subckt name>

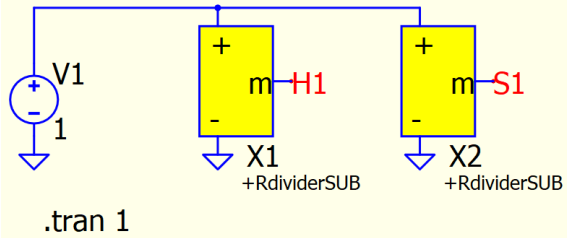
Hierarchical Block




```
* C:\QspiceKSKelvin\01 User Guide and Script\02
V1 N01 0 1
X1 N01 0 H1 +RdividerH
X2 N01 0 H2 +RdividerH

.subckt +RdividerH + - m
R1 + m 1K
R2 m - 1K
.ends +RdividerH

.tran 1
.end
```

.subckt for X1 and X2 <child schematic name>

Sub-Circuit (.subckt)




```
* C:\QspiceKSKelvin\01 User Guide and Script\02
V1 N01 0 1
.subckt X1•+RdividerSUB + - m
R1 + m 1K
R2 m - 1K
.ends +Rdivider
X1 N01 0 H1 X1•+RdividerSUB
.subckt X2•+RdividerSUB + - m
R1 + m 1K
R2 m - 1K
.ends +Rdivider
X2 N01 0 S1 X2•+RdividerSUB
.tran 1
.end
```

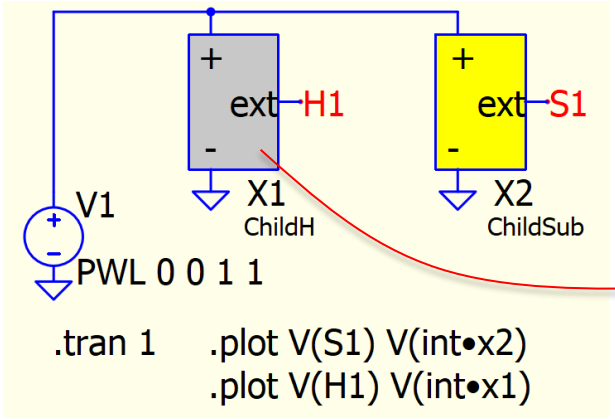
.subckt for X1 X1•<subckt name>

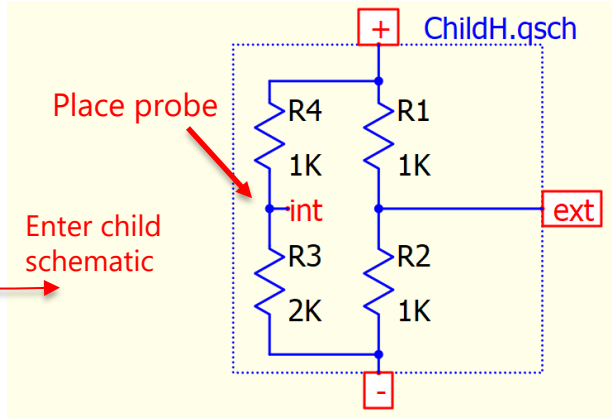
.subckt for X2 X2•<subckt name>

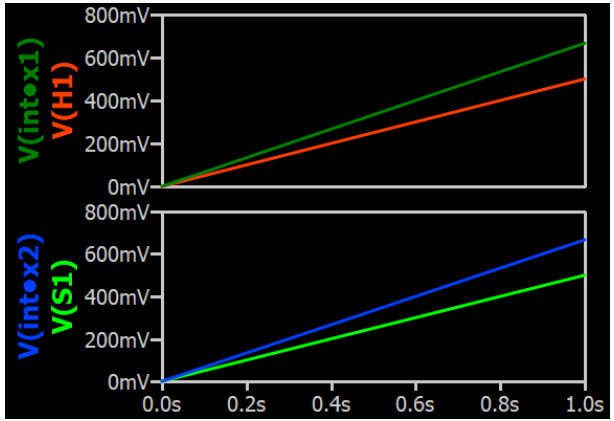
Hierarchical and Sub-circuit : Probe in Waveform Viewer

Qspice : Probe Hierarchical and Subckt.qsch | ChildH.qsch

- Probe Hierarchical and Sub-circuit
  - Expression to probe for hierarchical and subckt symbol are identical
    - V : <netname>•<Xn>
    - I : <devicename>•<Xn>
  - Hierarchical allow user to enter child schematic and add probe
  - Sub-circuit is in netlist and can only be probe by
    - .plot directive
    - In waveform viewer, Add Plot and enable "Asterisk Matches Hierarchy Separator"





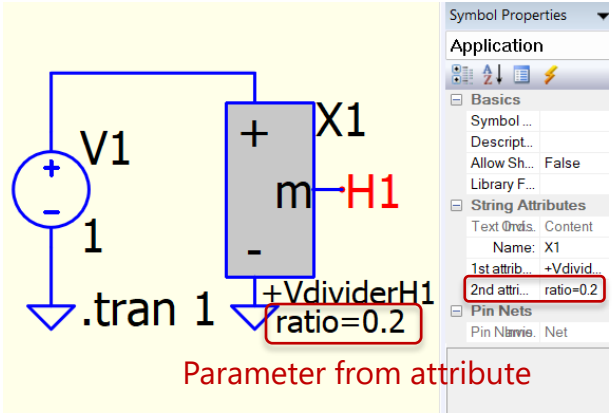


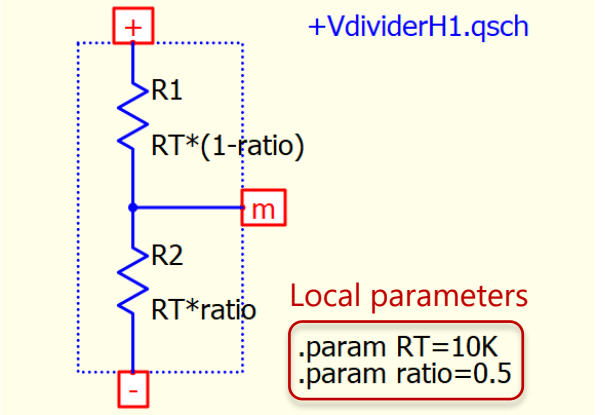
In Waveform Viewer
Right Click > Add Plot >
Select "Asterisk Matches Hierarchy Separator

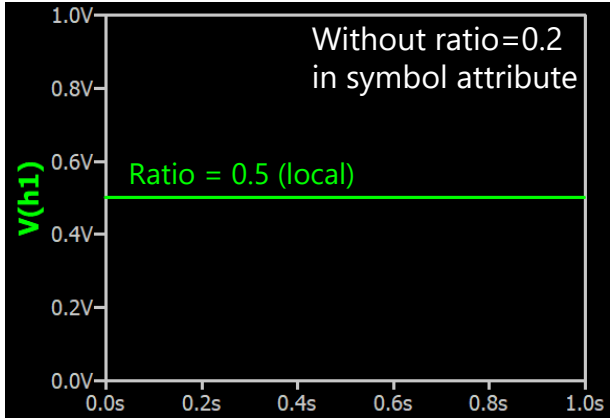
Hierarchical and Sub-circuit : Passing Parameters

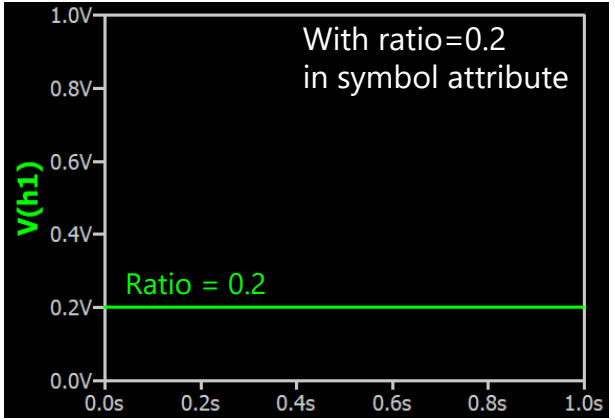
Qspice : parent-PassParamHierarchical.qsch | +VdividerH1.qsch

- Passing Parameters
  - Hierarchical and subcircuits work in the same way
  - By default, .subckt or child schematics load their local parameters from the .param (not a must to have local parameters)
  - In the parent schematic, add a new attribute to the symbol can pass parameters; this will override local parameters within the .subckt or child schematic





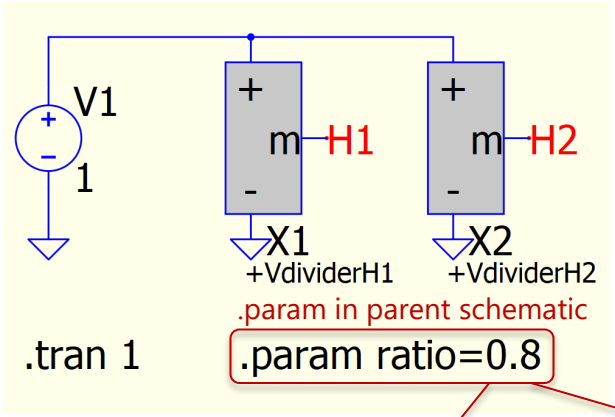


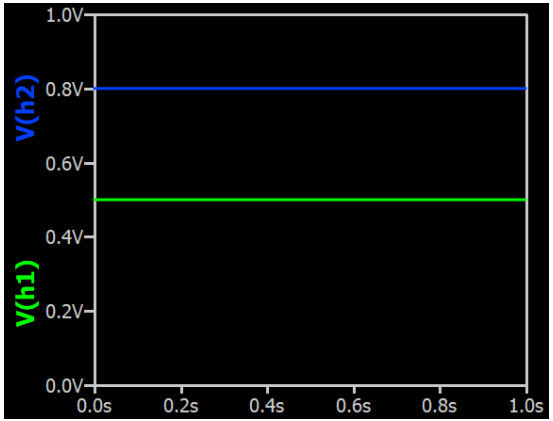


Parameter Passing with Global .param from parent

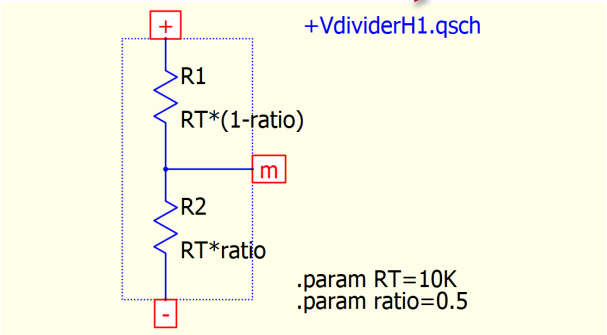
Qspice : parent-PassGlobalParam.qsch | +VdividerH1.qsch | +VdividerH2.qsch

- .param from Parent
  - Global .param passing from parent into a child schematic depends whether this child schematic has the parameter defined
  - If no such parameter is defined in child schematic, global .param override
  - If parameter is defined in child, global .param is ignored. Only string attribute in symbol has ability to override child schematic defined parameter

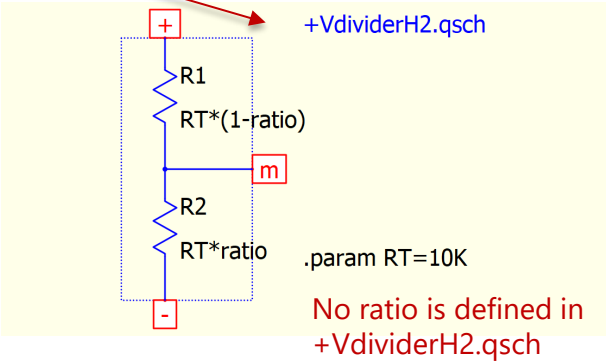




No effect



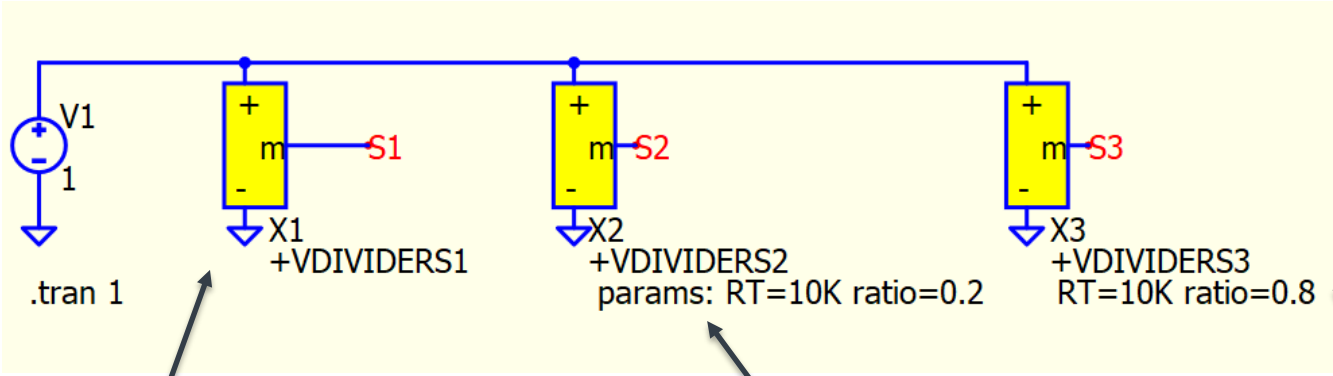
In effect



Three Way to Define Default Parameters in .subckt

Qspice : parent-PassParamSubckt.qsch | +VdividerS.txt

- Three Way to Define Default Parameters in .subckt




```
.subckt +VdividerS1 + - m
R1 + m RT*(1-ratio)
R2 m - RT*ratio
.param RT=10K
.param ratio=0.5
.ends +VdividerS1
```

```
.subckt +VdividerS2 + - m params: RT=10K ratio=0.5
R1 + m RT*(1-ratio)
R2 m - RT*ratio
.ends +VdividerS2
```

```
.subckt +VdividerS3 + - m RT=10K ratio=0.5
R1 + m RT*(1-ratio)
R2 m - RT*ratio
.ends +VdividerS3
```

For this version, if removes .param lines, param can be added in symbol in string attribute
Right Click on symbol > Add New Attribute

Three Type of Sub-Circuit (.subckt) Symbol (.qsym)

Qspice : parent - 3 type subckt symbol.qsch

Embedded SUBCKT
[Easy to share, just one .qsym]

Symbol Properties

Application

Basics		
Symbol Type	X	
Description		
Allow Shorted Pins	False	
Library File	|.subckt +RdividerH1 + - m\nR...	
String Attributes		
Text Order	Invis.	Content
Name:		X1
1st attribute		+RDIVIDERH1

.subckt is in library file properties

Link to Library
[Recommend for complex .subckt]

Symbol Properties

Application

Basics		
Symbol Type	X	
Description		
Allow Shorted Pins	False	
Library File	subckt_library.txt	
String Attributes		
Text Order	Invis.	Content
Name:		X2
1st attribute		+RDIVIDERH2

Library is in library file properties

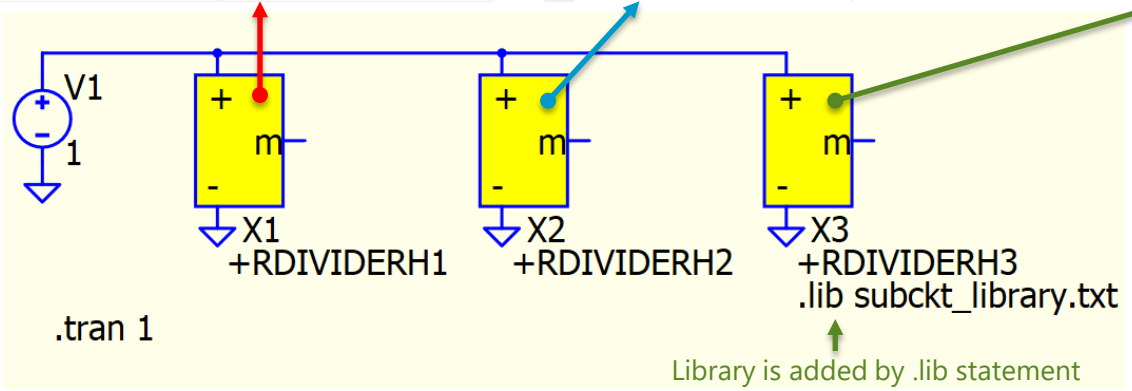
No Embed or Link
[For universal symbol]

Symbol Properties

Application

Basics		
Symbol Type	X	
Description		
Allow Shorted Pins	False	
Library File		
String Attributes		
Text Order	Invis.	Content
Name:		X3
1st attribute		+RDIVIDERH3

Nothing is in library file properties



subckt_library.txt

```
1  .subckt +RdividerH2 + - m
2  R1 + m 1K
3  R2 m - 2K
4  .ends +RdividerH2
5
6  .subckt +RdividerH3 + - m
7  R1 + m 1K
8  R2 m - 3K
9  .ends +RdividerH3
```

Hierarchical and Sub-circuit Sub-Topics

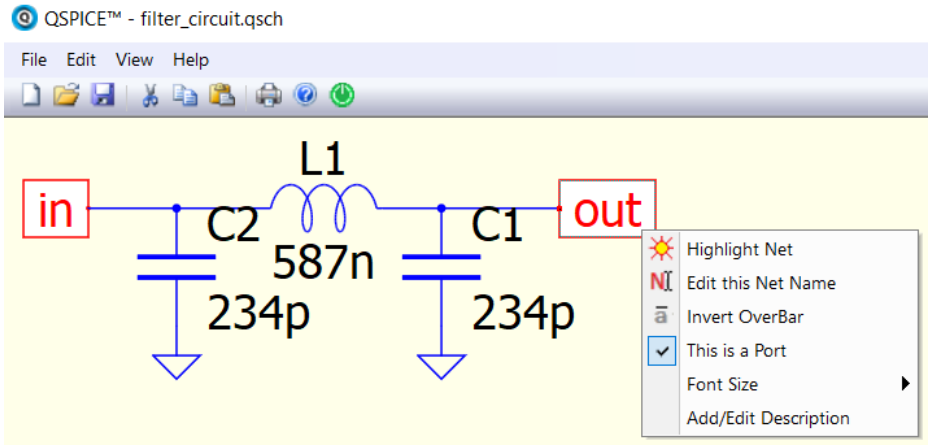
- Part 2A : Hierarchical Block
  - Create hierarchical block from child to parent or parent to child schematic
  - Create symbol for hierarchical block
  - Get .subckt from hierarchical block to convert into an embedded subckt symbol
- Part 2B : Symbol for Subckt [Embedded Subckt]
- Part 2C : Symbol for Subckt [Link to Library]
- Part 2D : Convert MOSFET M to subckt Symbol
  - Demonstrate how to convert a MOSFET M symbol into subckt to save effort in creating a MOSFET symbol for .subckt MOSFET model from 3rd party vendor
- Part 2E : Bus and Hierarchical Block

Part 2A
Hierarchical Block

#1 – Hierarchical Block : From Child to Parent

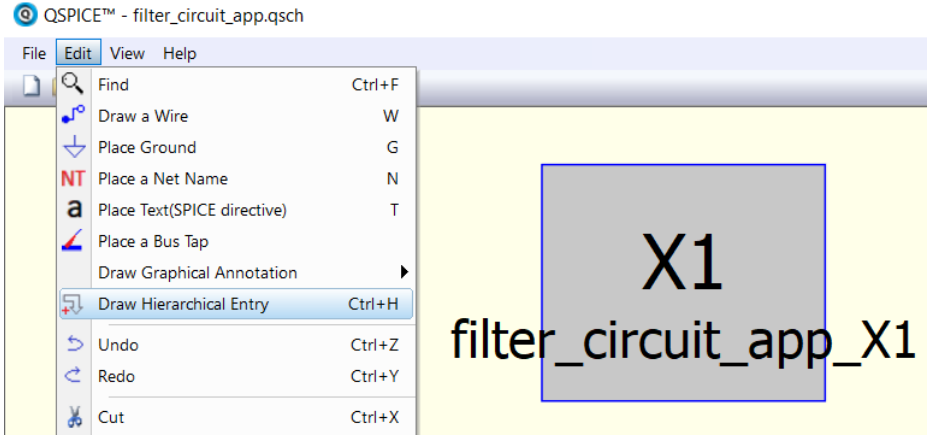
Qspice : filter_circuit.qsch | filter_circuit_app.qsch

[1] Create a child schematic (.qsch) with circuit and net label
[2] Right click on net label and select “This is a Port”



Method #1
[3] Create a new schematic which will call to use hierarchical
[4] Edit → Draw Hierarchical Entry

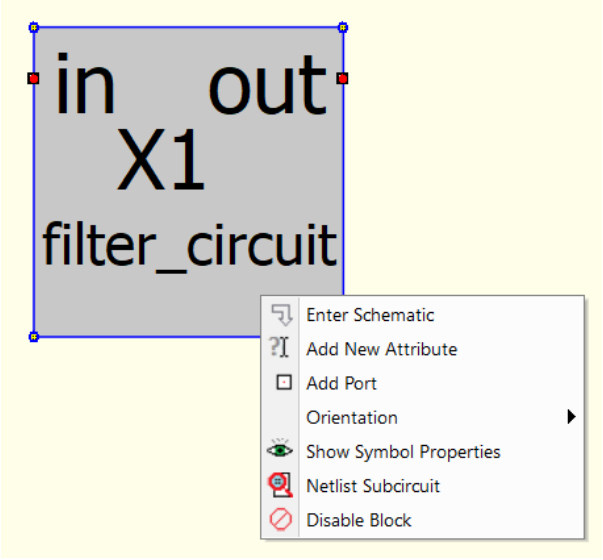
Method #2
[3] In child schematic, Right click > Open Parent Schematic
This will automatically create a parent schematic contains hierarchical symbol, with all Port automatically created



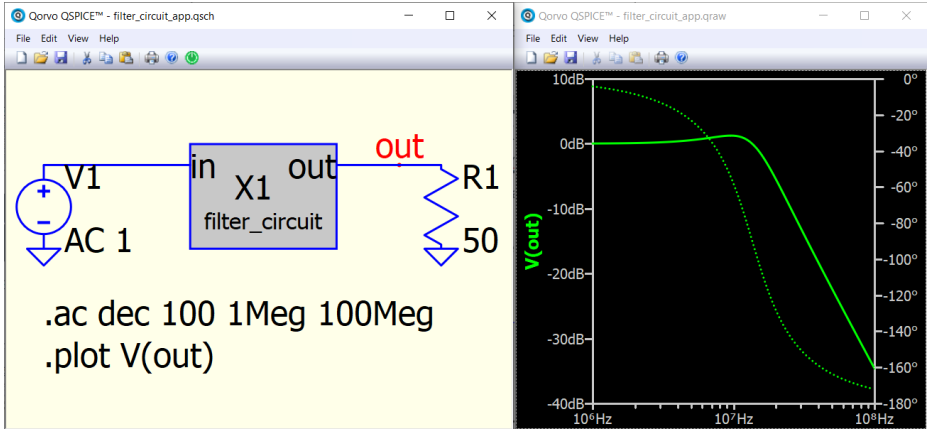
#1 – Hierarchical Block : From Child to Parent

Qspice : filter_circuit.qsch | filter_circuit_app.qsch

[5] Change component text (1st attribute) to match child schematic name
[6] Right click hierarchy component and “Add Port”
[7] Name ports as port name defined in child schematic
[8] Right click hierarchy component and “Enter Schematic” should open child schematic

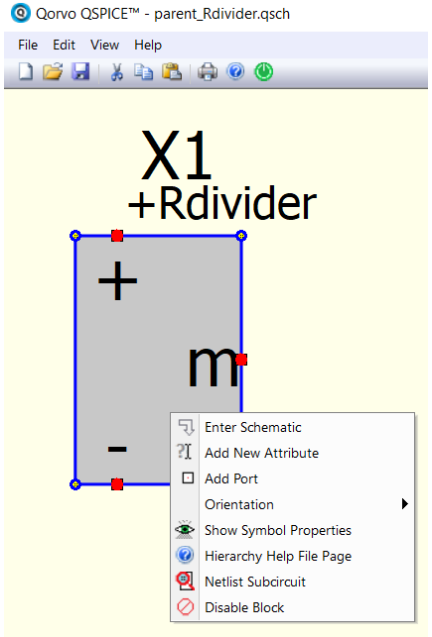


A completed example

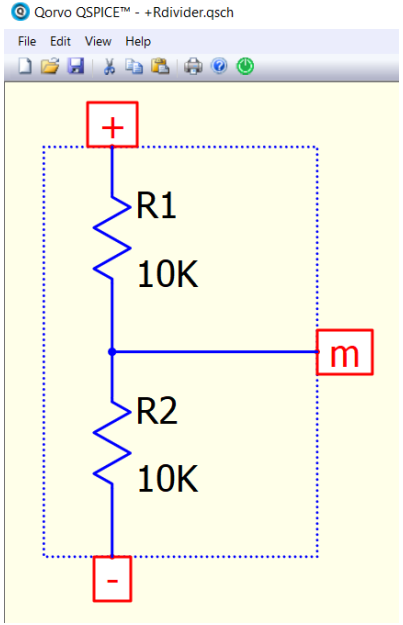


#2 – Hierarchical Block : From Parent to Child

[1] Right click > Draw Hierarchical Entry
[2] Rename component text (1st attribute) to child schematic name
** Child schematic will be created later
[3] Right click within Hierarchical Block > Add Port
[4] Right click > Enter Schematic, it will create a child .qsch



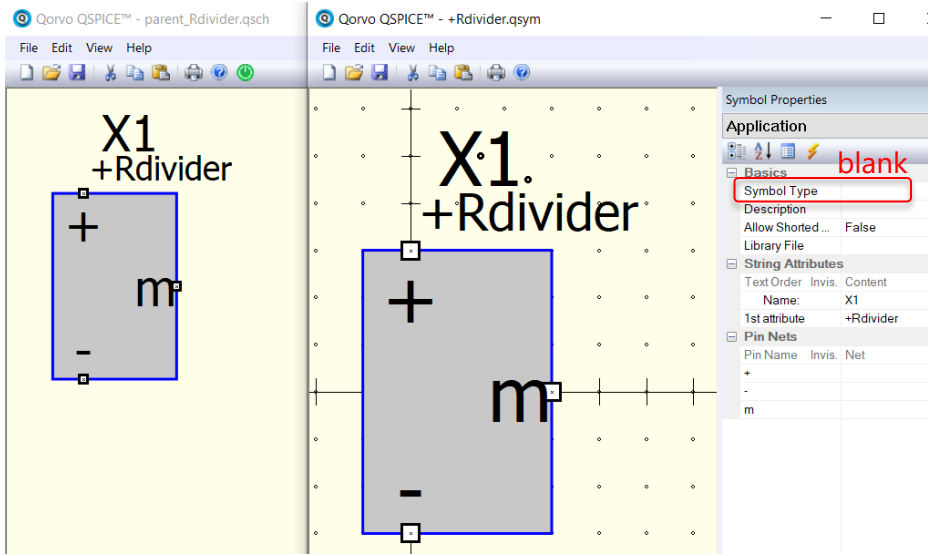
[5] Create circuit in child schematic



#3 – Hierarchical Block : Create Symbol (.qsym) for Hierarchical

Qspice : parent_Rdivider.qsch | +Rdivider.qsym

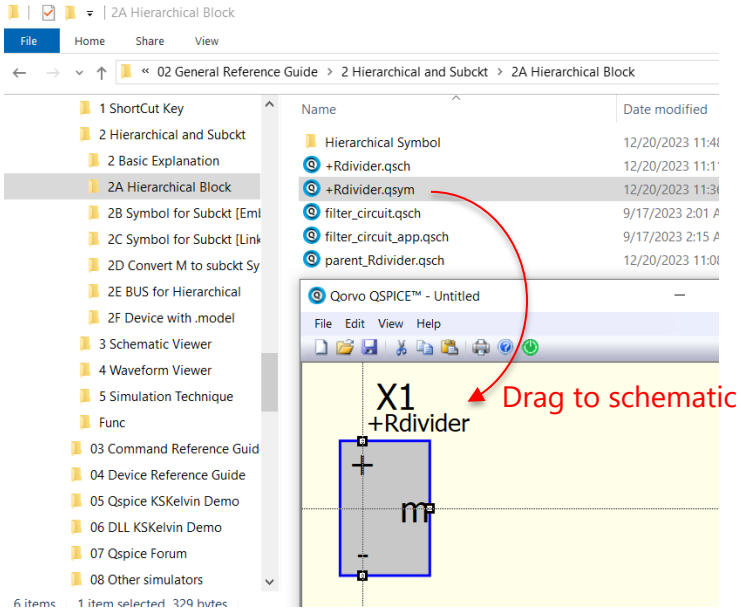
[1] In a parent schematic which contains a hierarchical block
[2] Hold Shift, draw a selection box to select Hierarchical
[3] Press Ctrl-C to copy
[4] File > New > New Symbol
[5] In New Symbol window, Press Ctrl-V to paste
[6] A symbol for a hierarchical block has been created, and now you have the option to edit this symbol. However, please keep in mind that the "Symbol Type" field must be left blank for a hierarchical symbol



[7] Save this symbol (.qsym) in the same directory as the hierarchical schematic (.qsch)

How to use .qsym with .qsch

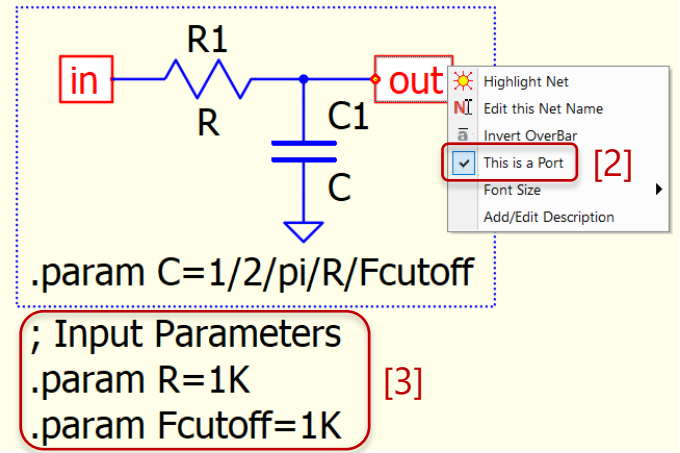
[8] To place this symbol in the schematic window, drag it from the File Explorer and drop it into the schematic window



#4 – Create Hierarchical Block from Hierarchical Schematic

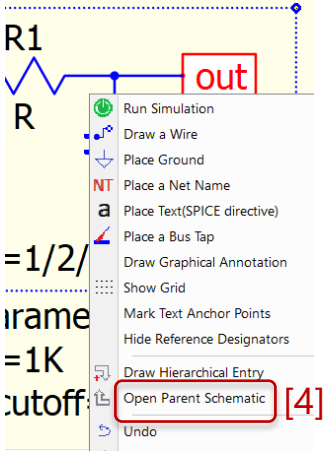
Qspice : \2A Hierarchical Block\Hierarchical Symbol\

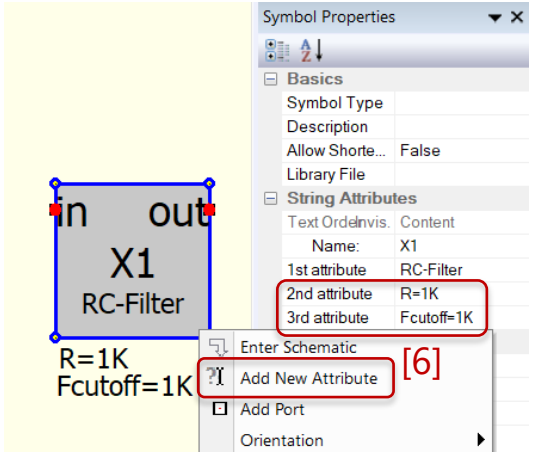
[1] **Draw a hierarchical schematic (child)** to illustrate a hierarchical circuit. This example is an RC filter
[2] **Right-click** on the pin label to set input/output pins label **as a port** for hierarchical block (only Port will appear in hierarchical block)
[3] For input parameters, you can leave them untouched or comment on them. If comment, the hierarchical block must have these parameters in its attributes. If left untouched, they act as default values if a parameter is not assigned in the hierarchical block attributes



[4] **Right click** > **Open Parent Schematic**
[5] A hierarchical block is created in its parent schematic. For the hierarchical block, its Symbol Type is blank, its Name is Xn, and the first attribute is the child schematic name (without .qsch)
[6] **Right-click** on the hierarchical block > **Add New Attribute**
Input parameters are added as attributes in the hierarchical block

Now, a hierarchical block in a parent schematic is created

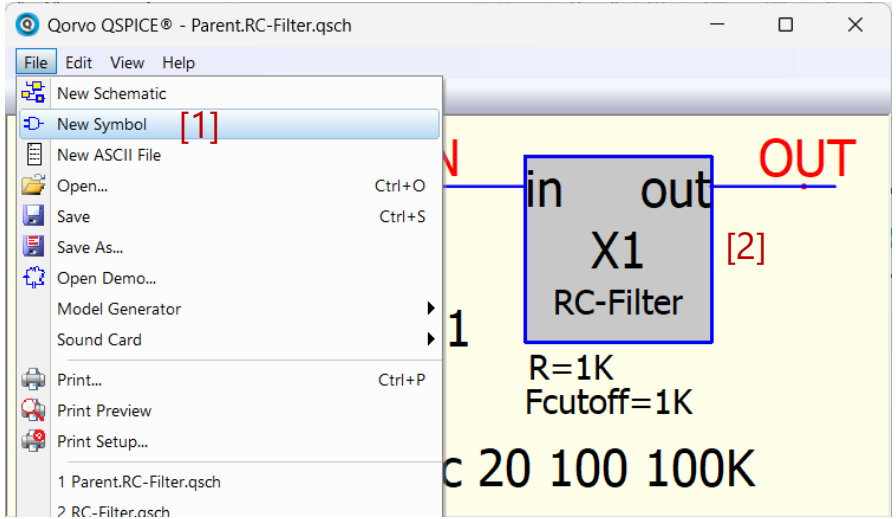


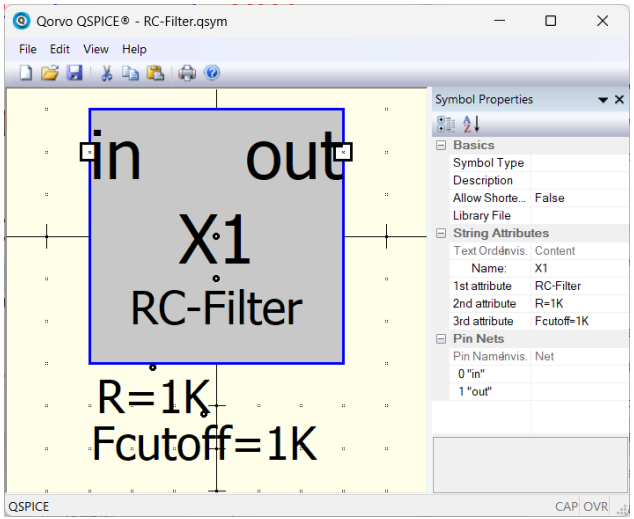


#4 – Create Hierarchical Symbol (.qsym) from Hierarchical Block

Qspice : \2A Hierarchical Block\Hierarchical Symbol\

Prerequisite : A Hierarchical Block is created in a Parent Schematic (refer to previous slide)
[1] **File > New Symbol** to open a new symbol window in a Parent schematic with a Hierarchical block
[2] **Hover the mouse over hierarchical block** in parent schematic, then **Ctrl-C** to copy it
[3] Navigate to **Symbol Window** and press **Ctrl-V** to paste it
[4] In the **Symbol Window, File > Save** to save this symbol as .qsym file
[5] Now, you can reuse this symbol in other projects. For a hierarchical symbol (Symbol Type = blank), it does not contain a subcircuit netlist and is linked to a schematic named as its [1st attribute].qsch. Therefore, it MUST be shared with this child schematic

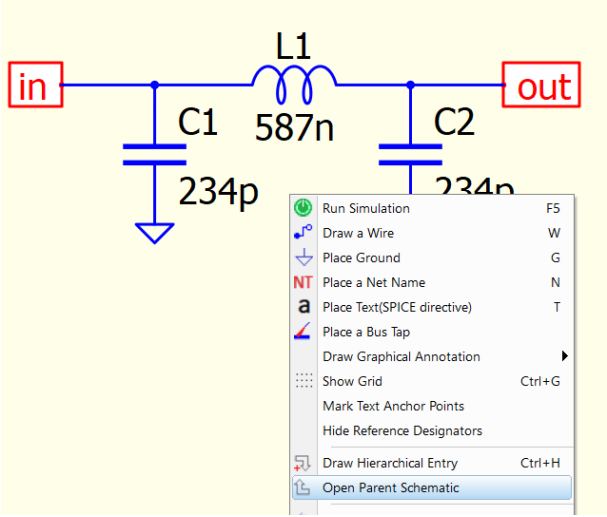




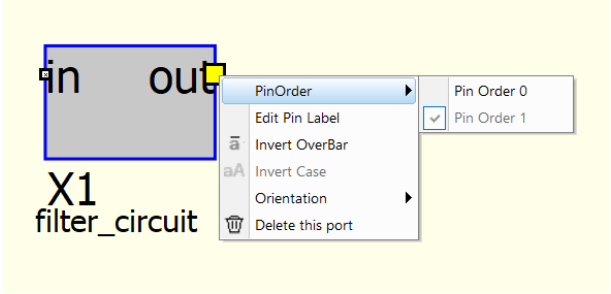
#5 – Create Subckt Symbol (.qsym) from Hierarchical Circuit (.qsch)

[Step 1 to 4]

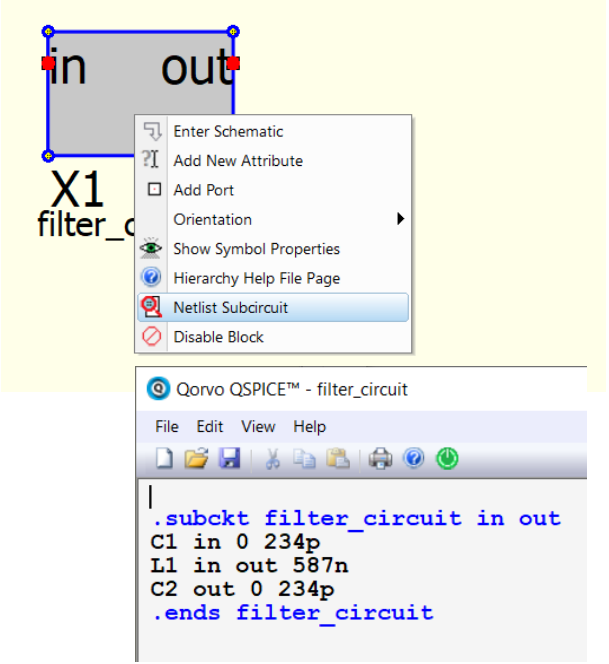
[1] In hierarchical schematic
Right Click > Open Parent Schematic
It will generate a .qsch with prefix Parent.



[2] In Parent Schematic, verify/change Pin Order if a particular pin name order you need for .subckt (not necessary step)



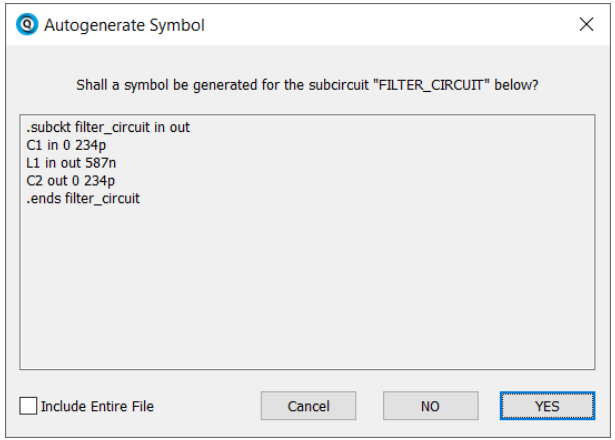
[3] **Right Click** on Hierarchical Symbol, select **Netlist Subcircuit**
[4] select all and copy .subckt netlist with Ctrl-C



#5 – Create Subckt Symbol (.qsym) from Hierarchical Circuit (.qsch)

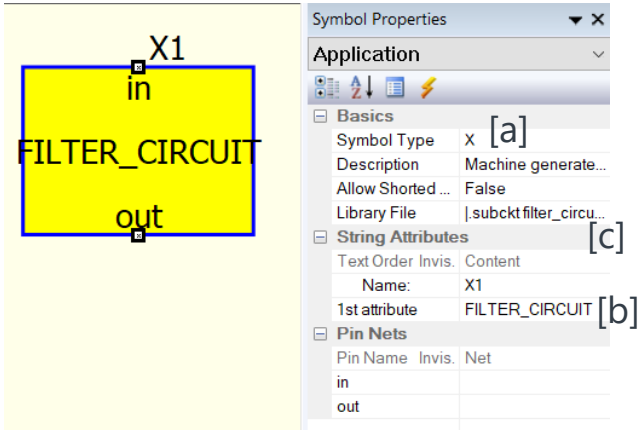
[Step 5 to 6]

[5] **Ctrl-V** to paste the text into a schematic (or a blank symbol) to invoke the Autogenerate Symbol routine



[6] A subckt symbol is generated with embedded .subckt which contain the circuitry that you can use in any schematic in any directory

[7] You can copy this symbol with **Ctrl-C**, go to **File > New Symbol**, paste it into the Symbol Window with **Ctrl-V**, and save the symbol to be used in other projects



About Subcircuit Symbol

Similar to a hierarchical symbol, users can add input parameters through the New Attribute

[a] A subcircuit symbol with Symbol Type: X
[b] Its 1st attribute is the name of the .subckt
[c] Using this method, the subcircuit netlist is embedded into the Library File in this symbol, which comprises a one-line subcircuit netlist

For subcircuit symbol, it does not require linking with a child schematic. This symbol contains both graphical and simulation netlists

Part 2B
Symbol for Subckt
[Embedded Subckt]

Symbol for Subckt – Overview

- Symbol for Subckt
  - This section explains how to create a symbol from a subcircuit (.subckt) netlist library file
  - In general, manufacturer provides their device model with .subckt netlist in a .lib or .txt file in SPICE syntax
    - Device modeled by subcircuit because no single native SPICE device can describe the circuit model
  - To use this subcircuit in schematic, user needs to create a symbol
    - With .subckt netlist embedded [describe in Part 2B]
    - With subcircuit library linked [describe in Part 2C]

- Library with Subcircuit
  - Subcircuit (.subckt) is a SPICE directive as
    - .subckt <name> <node1> <node2> …
      … <SPICE circuit netlist> …
      .ends <name>
  - The symbol will be created with <name> matched to .subckt and <node> in same order
  - Subcircuit can call another subcircuit. *If to use Autogenerate symbol feature in Qspice, **parent subcircuit** MUST in front of its **child subcircuits** in netlist file*

irf530.lib

```
1  *Feb 16, 2010
2  *Doc. ID: 90181, Rev. A
3  *File Name: part irf530_PS.txt and part irf530_PS.spi
4  *This document is intended as a SPICE modeling guideline
5  *constitute a commercial product data sheet.  Designers s
6  *appropriate data sheet of the same number for guaranteed
7  *limits.
8  .SUBCKT irf530 1 2 3
9  ****************************************
10 *      Model Generated by MODPEX       *
11 *Copyright(c) Symmetry Design Systems*
12 *         All Rights Reserved          *
13 *    UNPUBLISHED LICENSED SOFTWARE     *
14 *   Contains Proprietary Information *
15 *      Which is The Property of       *
16 *     SYMMETRY OR ITS LICENSORS       *
17 *Commercial Use or Resale Restricted *
18 *   by Symmetry License Agreement     *
19 ****************************************
20 * Model generated on
21 * Model format: SPICE
22 * Symmetry POWER MOS
23 * External Node Desig
24 * Node 1 -> Drain
25 * Node 2 -> Gate
26 * Node 3 -> Source
27 M1 9 7 8 8 MM L=100u W=100u
28 * Default values used in MM:
29 * The voltage-dependent capacitances are
30 * not included. Other default values are:
31 *   RS=0 RD=0 LD=0 CBD=0 CBS=0 CGBO=0
32 .MODEL MM NMOS LEVEL=1 IS=1e-32
33 +VTO=3.87932 LAMBDA=0.00393789 KP=7.05019
34 +CGSO=6.11314e-06 CGDO=1e-11
35 RS 8 3 0.073836
36 D1 3 1 MD
37 .MODEL MD D IS=9.70956e-10 RS=0.0137423 N=1.31938 BV=300
38 +IBV=0.00025 EG=1 XTI=4 TT=1e-07
39 +CJO=1.03141e-09 VJ=1.46661 M=0.501224 FC=0.5
40 RDS 3 1 4e+06
```

Define .subckt

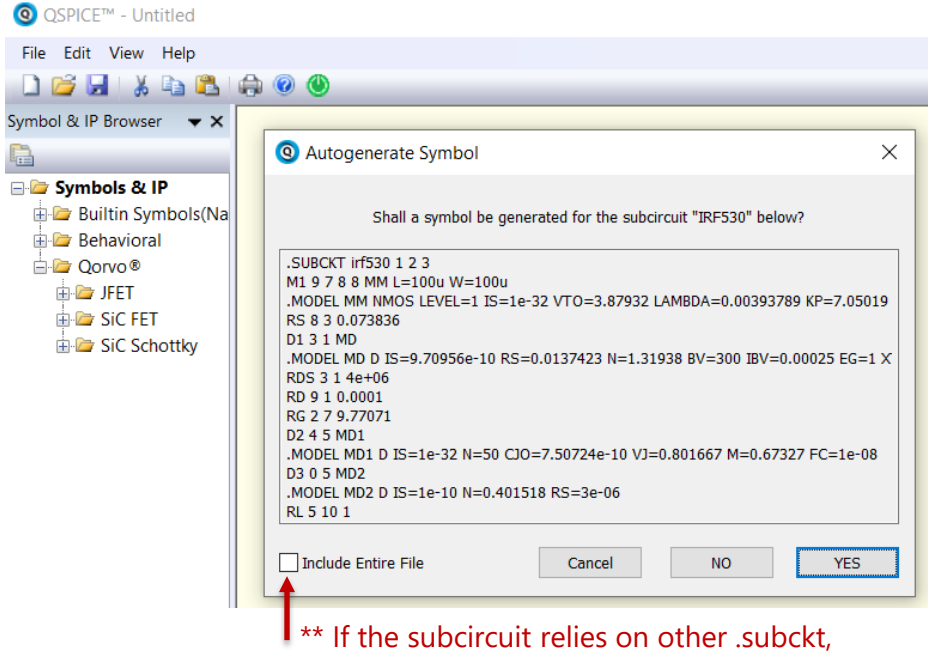
May have comment info to explain node and its physical meaning

Circuit netlist

#1 – Symbol for Subckt [Embedded Subckt] – From Subcircuit .subckt

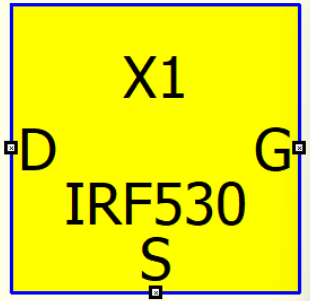
Procedure to Create embedded SUBCKT symbol from a subcircuit (.subckt) netlist

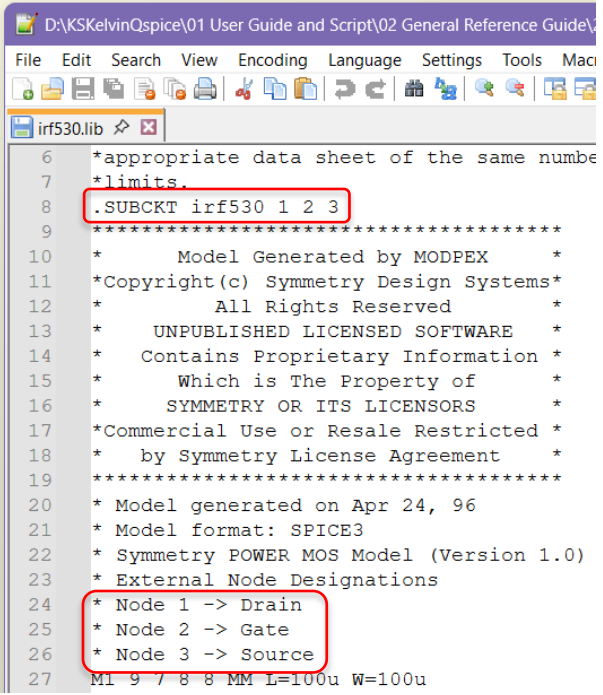
[1] Assume user has a subcircuit .subckt netlist (.txt / .lib etc)
[2] Use a text editor to open library file, copy text of subcircuit
[3] In **Schematic Window**, paste the text (**Ctrl-V**) and **Autogenerate Symbol** window will appear, **Yes** to create



** If the subcircuit relies on other .subckt, click on this block to include entire file

[4] (*Optional Step*) Rename pins by referring to descriptions in model file

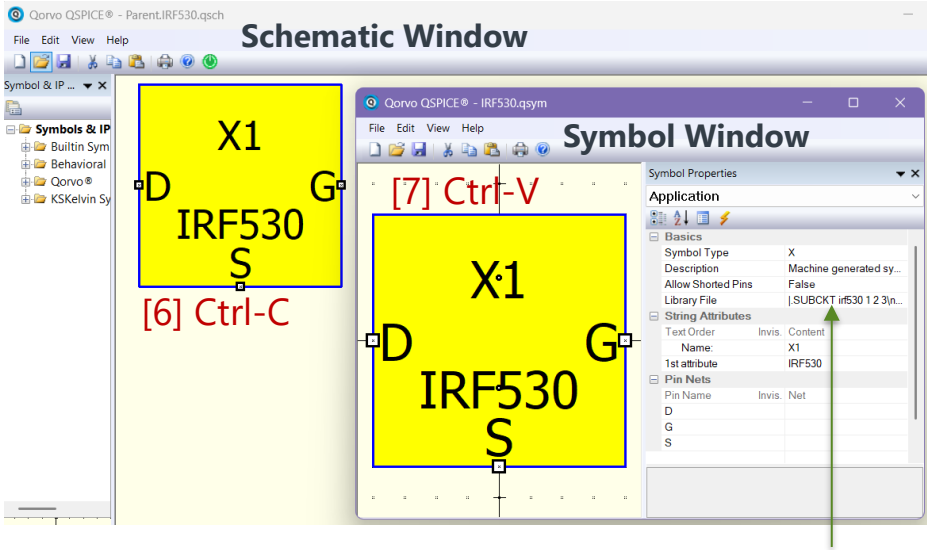




#1 – Symbol for Subckt [Embedded Subckt] – From Subcircuit .subckt

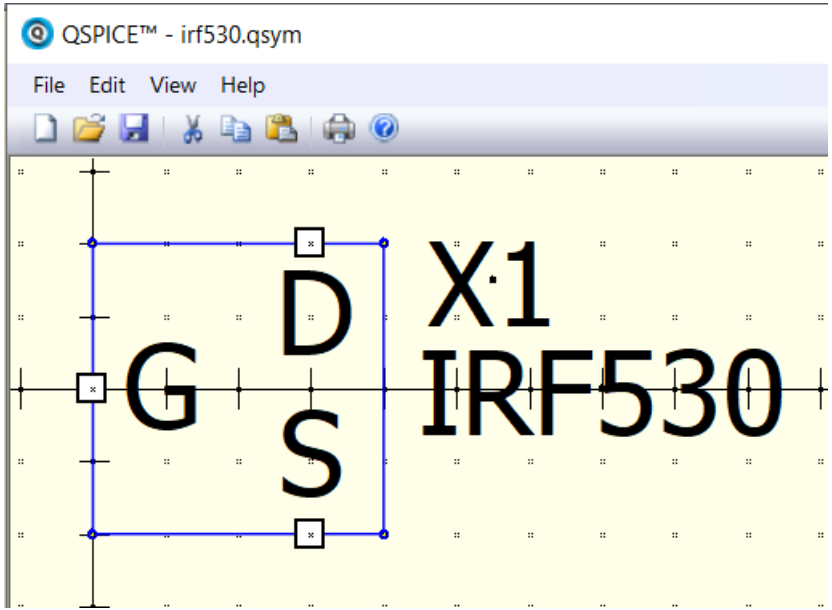
Procedure to Create embedded SUBCKT symbol from a subcircuit (.subckt) netlist

[5] **File** > **New Symbol** to open a **Symbol Window**
[6] In **Schematic Window**, **Ctrl-C** to copy device X1
[7] Goto **Symbol Window**, **Ctrl-V** to paste device



** .subckt netlist is embedded into symbol in the field of "Library File"

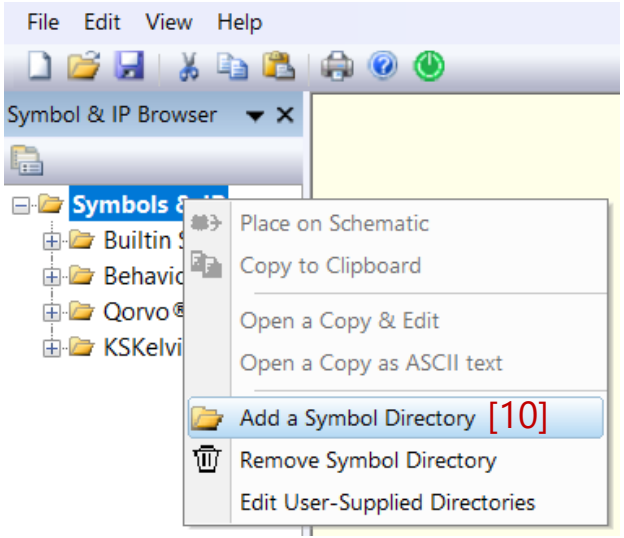
[8] Now rearrange pins location and re-draw the symbol
** But don't change pin order!
[9] Now, save as a Qspice embedded symbol (file extension as .qsym)

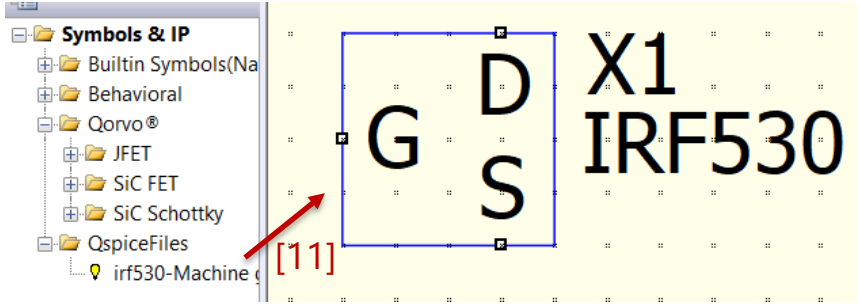


#1 – Symbol for Subckt [Embedded Subckt] – From Subcircuit .subckt

Procedure to Create embedded SUBCKT symbol from a subcircuit (.subckt) netlist

[10] In **Schematic Window**, open **Symbol & IP Browser** (shortcut F2)
Right Click "Symbols & IP" > "Add a symbol directory"
Add directory that contains your symbol (.qsym) files
[11] Now, you can drag custom symbol to schematic from this browser
(Alternative method : You can also directly drag a .qsym symbol file from **File Explorer** to schematic)
[12] ** netlist library (.lib/.txt) is no longer required as .subckt is embedded into symbol



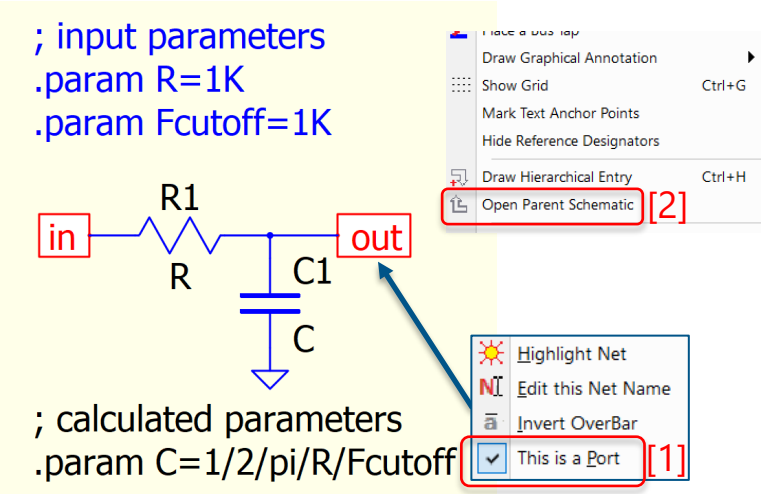


#2 – Symbol for Subckt [Embedded Subckt] – From Hierarchical Block

Procedure to Create embedded SUBCKT symbol from Hierarchy

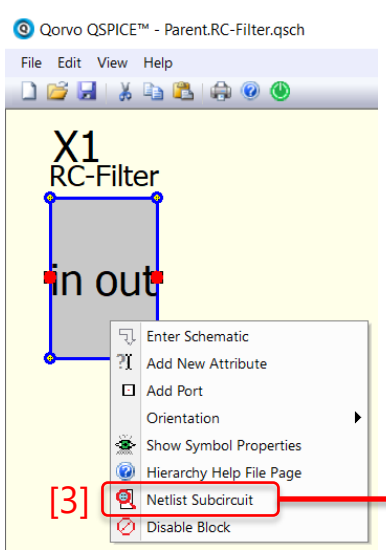
[1] Draw a circuit schematic
In this example
- Two ports : in and out
  (for subckt I/O port, right click net name, select **This is a Port**)
- Two input params : R and Fcutoff
  (comment input parameter, we will add it from symbol later)
- One calculated parameter : C

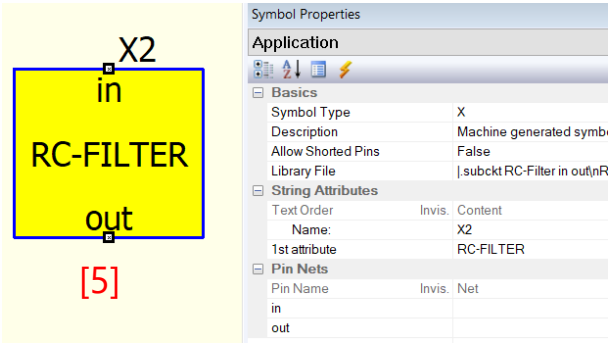
[2] Right click on schematic > **Open Parent Schematic**
- A hierarchical block with a new schematic is generated



[3] **Right click** hierarchical block > **Netlist Subcircuit**
[4] In .suckt netlist, **Ctrl-A** to select all and **Ctrl-C** to copy
[5] Return to **Schematic Viewer**, **Ctrl-V** to paste netlist and auto generate a symbol
- this is a symbol with embedded .subckt

[6] Click on this symbol, **Ctrl-C** to copy it
[7] File > New > New Symbol , **Ctrl-V** to paste symbol in **Symbol Viewer**



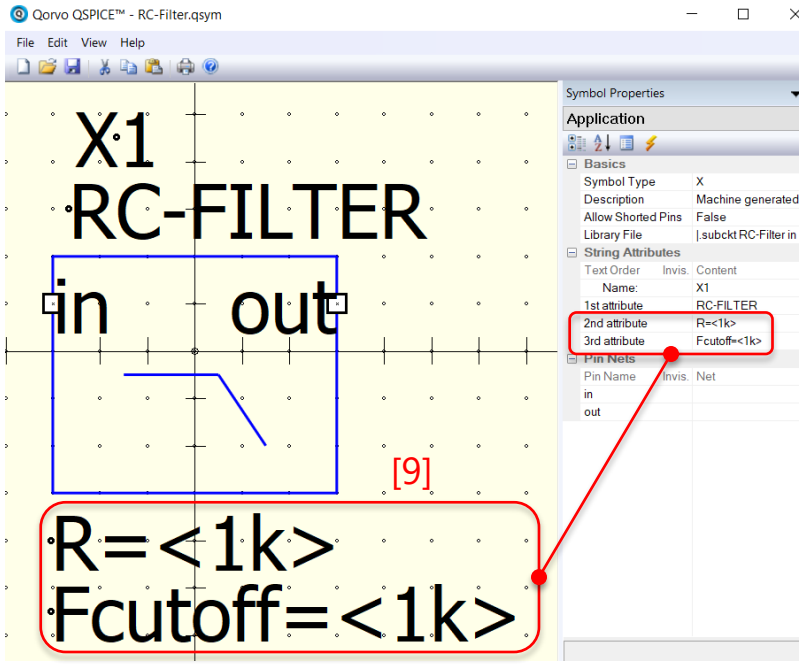



```
.subckt RC-Filter in out
R1 in out R
C1 out 0 C
; calculated parameters
.param C=1/2/pi/R/Fcutoff
.ends RC-Filter
```
[3]

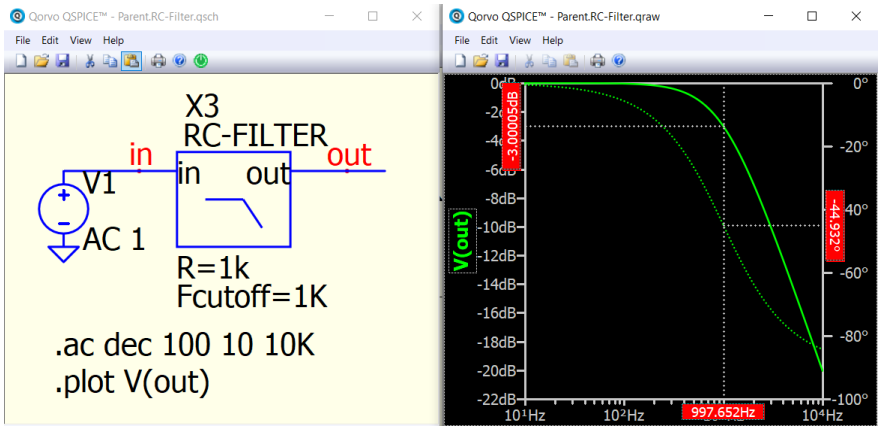
#2 – Symbol for Subckt [Embedded Subckt] – From Hierarchical Block

Procedure to Create embedded SUBCKT symbol from Hierarchy

[8] In **Symbol Viewer**, re-arrange symbol layout as you want
[9] To add input parameter R and Fcutoff, type **T** to enter new attribute. In this example, two attributes are added, and with arrow brackets <>. This syntax allows only to change value when this symbol is called



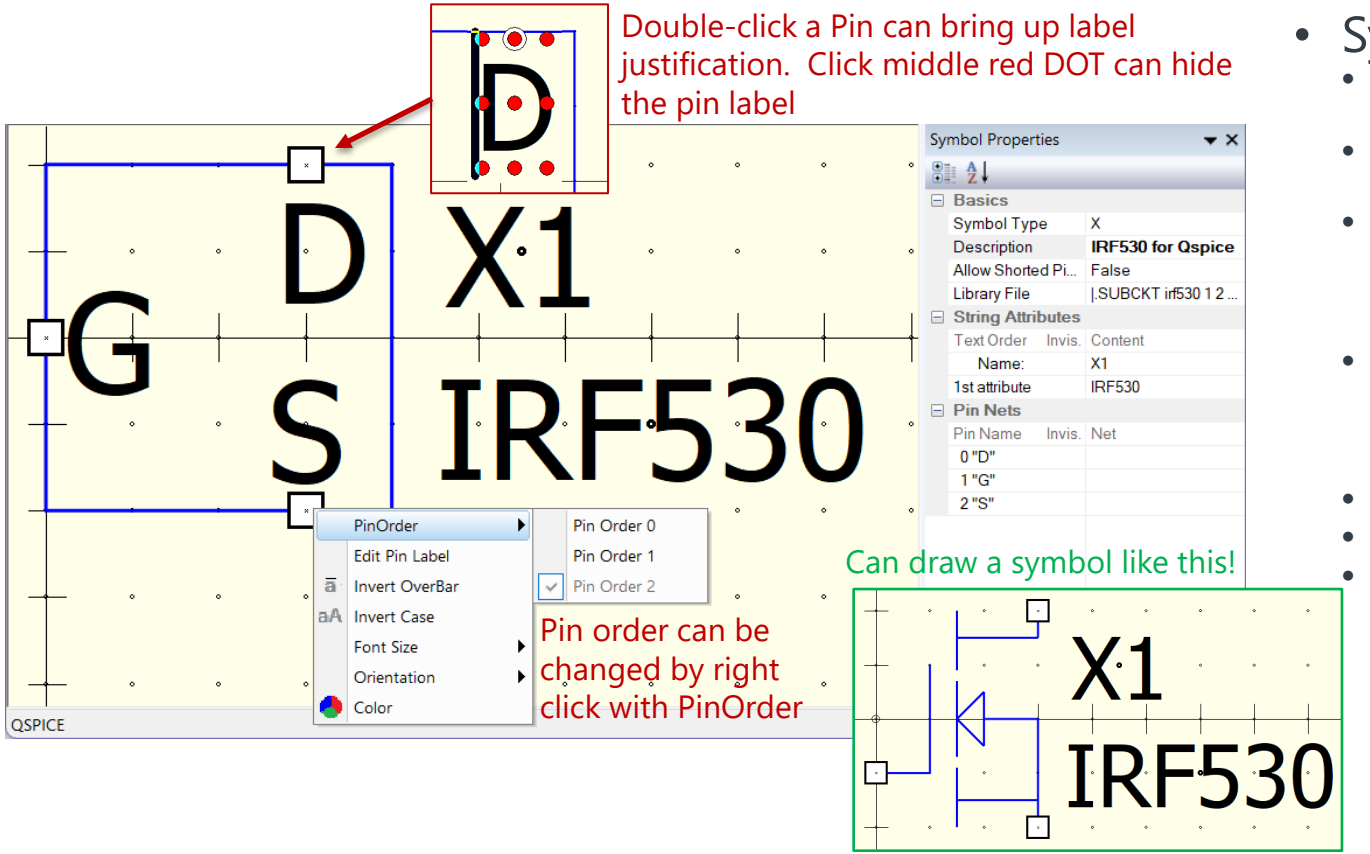
[10] Now, you can use this symbol for simulation



Technique if hierarchy circuit needs update

- It is not required to re-draw the symbol
- When hierarchy is updated, re-preform step [1] to [5]
- Right the click new auto generated symbol to show symbol properties
- Click on .subkct in library file, Ctrl-A to select all, Ctrl-C to copy it
- Open symbol .qsym, replace Library File with this new embedded .subckt line

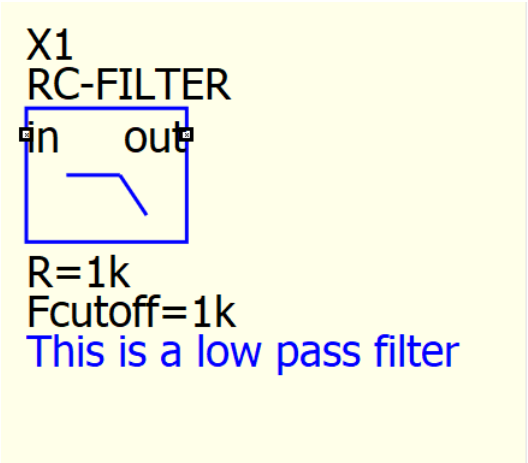
Detail about Embedded Subckt Symbol (.qsym)



- Symbol Properties
  - Symbol Type
    - X-device is Subcircuit
  - Description
    - Text to describe the device
  - Allow Shorted Pins
    - If wire connects when it is drawn directly through two pins
  - Library File
    - Embedded .subckt, embedded .model or put library file name
  - Name : Xn
  - 1st attribute : <name>
  - Pin Nets
    - Pin with order from 0,1,2,…
    - Correlation between symbol and .subckt netlist is based on pin ORDER but not its name!
    - You can give whatever name (label) you want to the pin

Detail about Embedded Subckt Symbol (.qsym)

Schematic Viewer



Symbol Properties

Basics		
Symbol Type		X
Description		RC-Filer
Allow Shorted Pins		False
Library File		|.subckt RC-Filter in out\n...
String Attributes		
Text Order	Invis.	Content
Name:		X1
1st attribute		RC-FILTER
R		1k
Fcutoff		1k
4th attribute		This is a low pass filter
Pin Nets		
Pin Name	Invis.	Net
0 "in"		
1 "out"		

View → Netlist

```
.subckt X1•RC-Filter in out
R1 in out R
C1 out 0 C
.param C=1/2/pi/R/Fcutoff
.ends RC-Filter
X1 ¥0 ¥1 X1•RC-FILTER R=1k Fcutoff=1k
.end
```

Subckt device name
<Name> • <1st attribute>

Input parameters
<attribute>

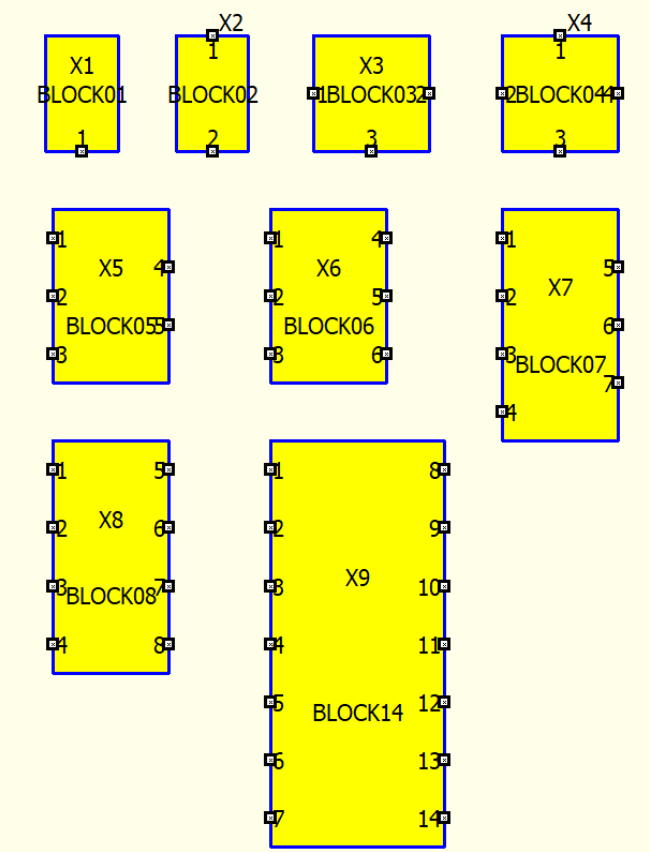
4th attribute is comment, which only appears in viewer but not netlist

In Library File : embedded .subckt (a single line format)

|.subckt RC-Filter in out\nR1 in out R\nC1 out 0 C\n.param C=1/2/pi/R/Fcutoff\n.ends RC-Filter

Autogenerate Symbol Pin Assignment

Qspice : Autogenerate Symbol Pin Assignment.qsch

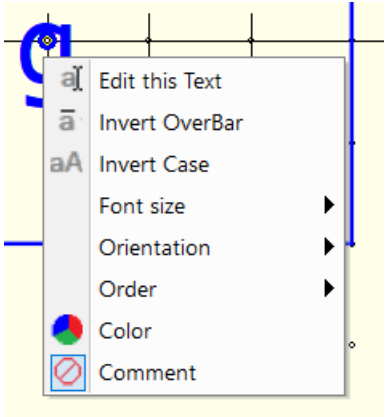


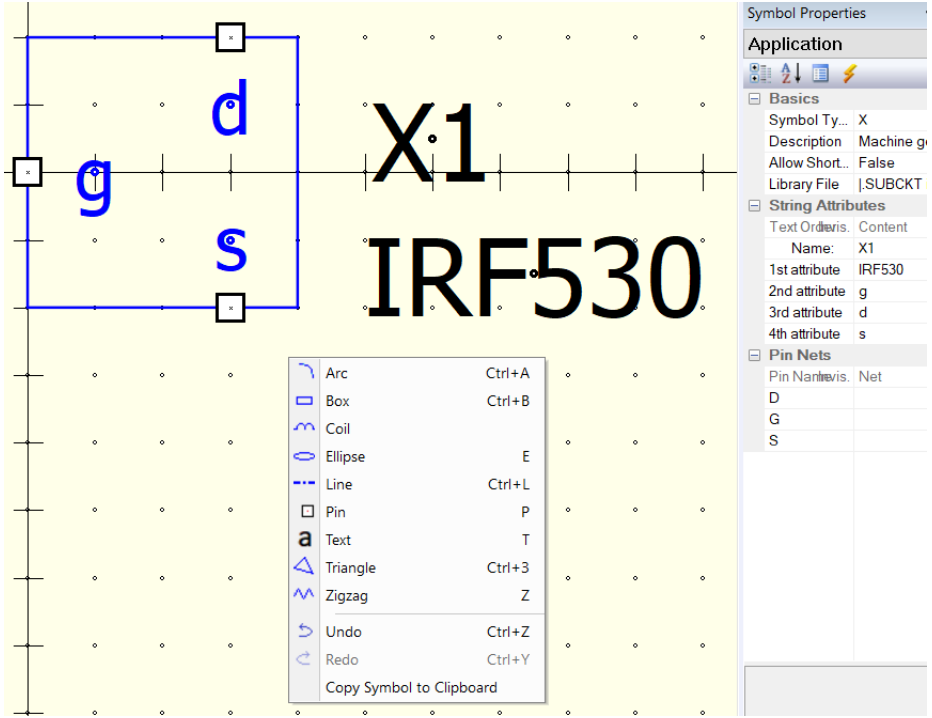

```
.subckt Block01 1
.ends Block01

.subckt Block02 1 2
.ends Block02

.subckt Block03 1 2 3
.ends Block03

.subckt Block04 1 2 3 4
.ends Block04

.subckt Block05 1 2 3 4 5
.ends Block05

.subckt Block06 1 2 3 4 5 6
.ends Block06

.subckt Block07 1 2 3 4 5 6 7
.ends Block07

.subckt Block08 1 2 3 4 5 6 7 8
.ends Block08

.subckt Block14 1 2 3 4 5 6 7 8 9 10 11 12 13 14
.ends Block14
```

Symbol for Subckt [Embedded Subckt] : Label with Text

Qspice : irf530 with text.qsym

- Text can be used in label
  - For example, instead of changing net name, you can
    - Right click > Text
    - Right click on text > Select “Comment”
      - Text not comment will become valid item in netlist
      - Can change font size and color
    - Be careful 1st attribute is device name (e.g. IRF530 in example), and doesn’t comment it

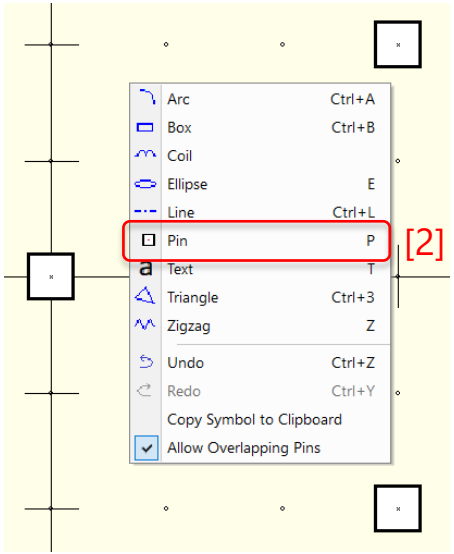


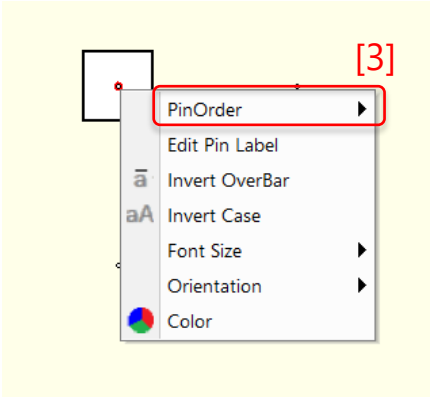


Part 2C
Symbol for Subckt
[Link to Library]

Symbol for Subckt [Create Symbol and Link to Library]

Example to create subckt symbol for irf530
[1] **File** > **New** > **New Symbol**
[2] **Right Click** > **Pin** (to add 3 pins with order D, G, S)
[3] Right Click at center of Pin to review PinOrder and PinLabel



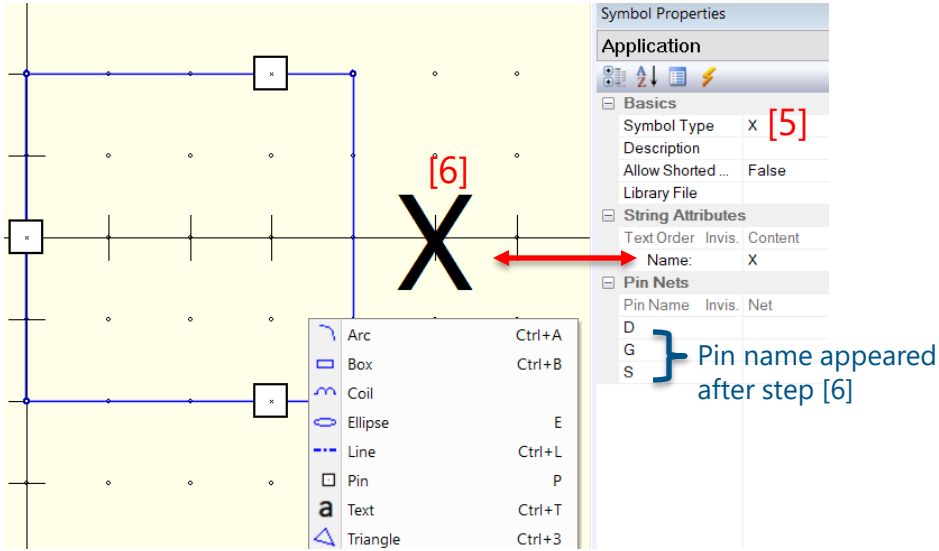


irf530.lib

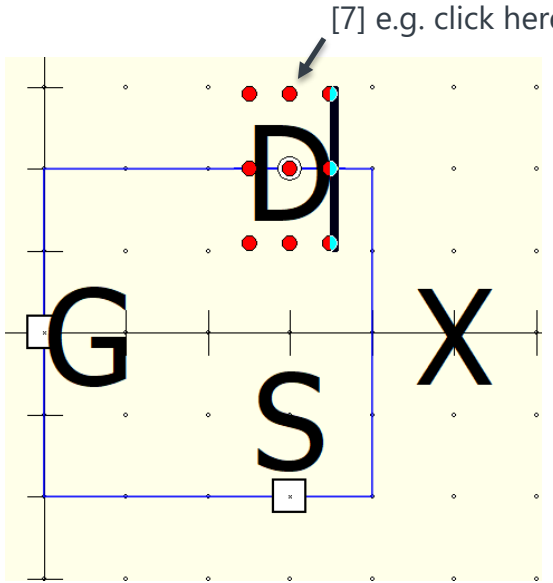
```
1  *Feb 16, 2010
2  *Doc. ID: 90181, Rev. A
3  *File Name: part irf530_PS.txt and part irf
4  *This document is intended as a SPICE model
5  *constitute a commercial product data sheet
6  *appropriate data sheet of the same number
7  *limits.
8  .SUBCKT irf530 1 2 3
9  ************************************
10 *      Model Generated by MODPEX     *
11 *Copyright(c) Symmetry Design Systems*
12 *         All Rights Reserved        *
13 *    UNPUBLISHED LICENSED SOFTWARE   *
14 *   Contains Proprietary Information *
15 *      Which is The Property of      *
16 *     SYMMETRY OR ITS LICENSORS      *
17 *Commercial Use or Resale Restricted *
18 *   by Symmetry License Agreement    *
19 ************************************
20 * Model generated on Apr 24, 96
21 * Model format: SPICE3
22 * Symmetry POWER MOS Model (Version 1.0)
23 * External Node Designations
24 * Node 1 -> Drain
25 * Node 2 -> Gate
26 * Node 3 -> Source
27 M1 9 7 8 8 MM L=100u W=100u
28 * Default values used in MM:
```

Symbol for Subckt [Create Symbol and Link to Library]

[4] Draw a box for outline
[5] In **Symbol Properties**, **Symbol Type**, select **X**
[6] In Symbol Window, **Right Click** > **Text** > Type **X**

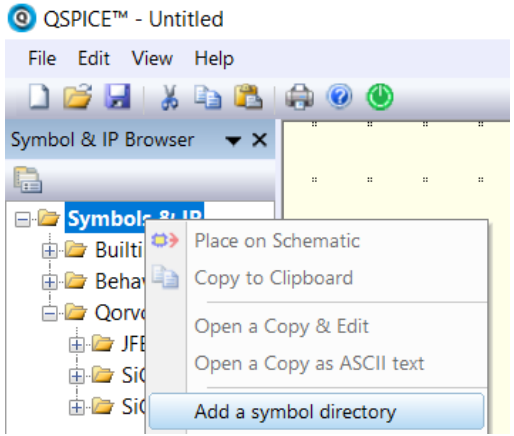


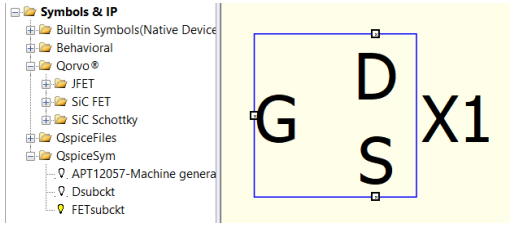
[7] To justify Pin label, double click center of Pin
[8] Click red dot other than its centered justification
[centered justification hide the pin label]
[9] Save symbol file as .qsym



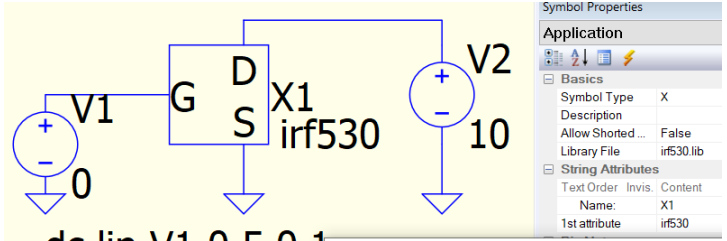
Symbol for Subckt [Create Symbol and Link to Library]

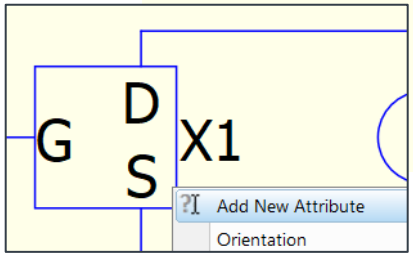
[10] In Schematic, Symbol & IP Browser, Right Click to **Add a symbol directory**
[11] Drag created component to schematic

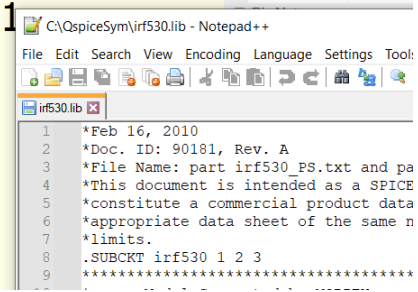




[12] **Right Click** on symbol, **Add New Attribute** as irf530 (1st attribute)
[13] In Symbol Properties, add irf530.lib to the field of "Library File" OR, add ".lib irf530.lib" into schematic for library link
**** library file is required to be put in schematic directory or symbol directory added in Symbols & IP browser**



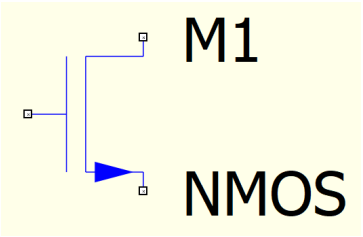




Part 2D
Convert MOSFET M to subckt Symbol

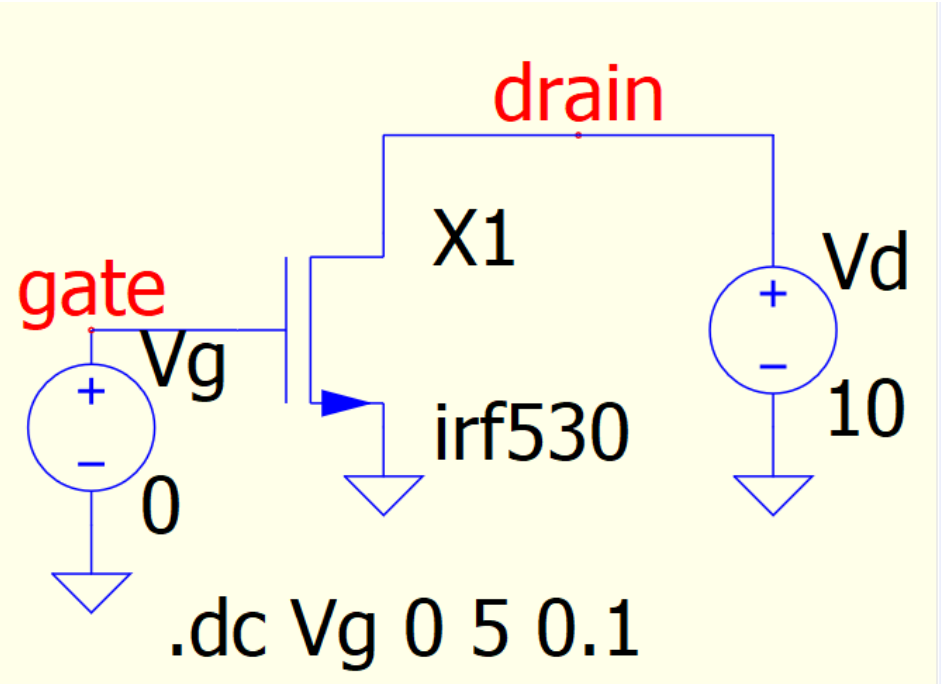
Convert MOSFET M to subckt Symbol

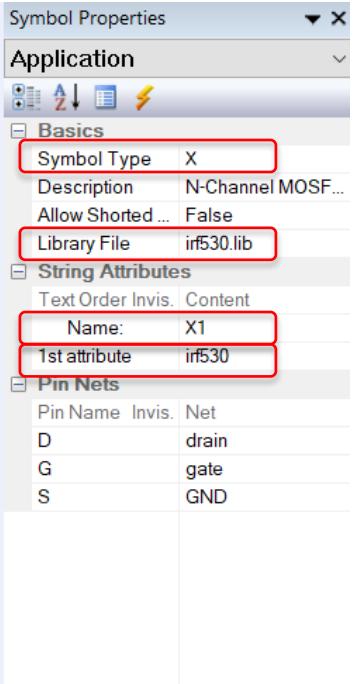
Qspice : Call Lib from M.qsch

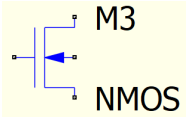


It is possible to convert MOSFET symbol M into a sub-circuit symbol by

1. Change Symbol Type from MN to X
2. Library File as subckt library file
3. 1st attribute as subckt name
4. [Optional] Change Symbol Name to X?



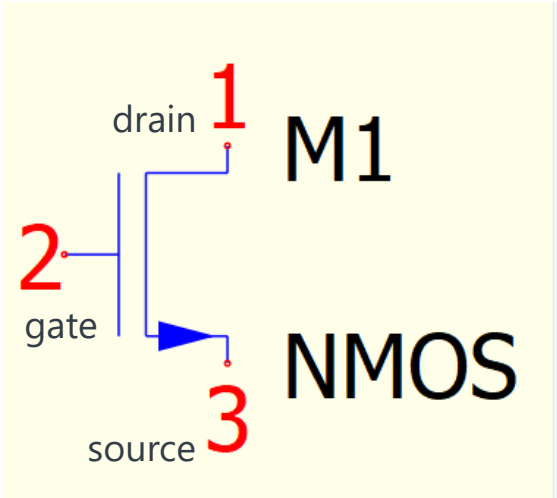




** this alternative symbol is 4 pins (+ base), which cannot support 3 pin subckt

Pin Order in Symbol MN and MP

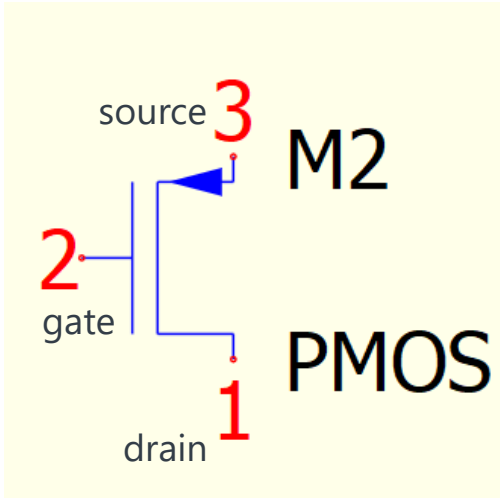
Pin Order for Symbol MN (NMOS)



Application

Basics	
Symbol Type	MN
Description	N-Channel MOSF...
Allow Shorted ...	False
Library File	NMOS.txt
String Attributes	
Text Order Invis.	Content
Name:	M1
1st attribute	NMOS
Pin Nets	
Pin Name Invis.	Net
D	1
G	2
S	3

Pin Order for Symbol MP (PMOS)



Application

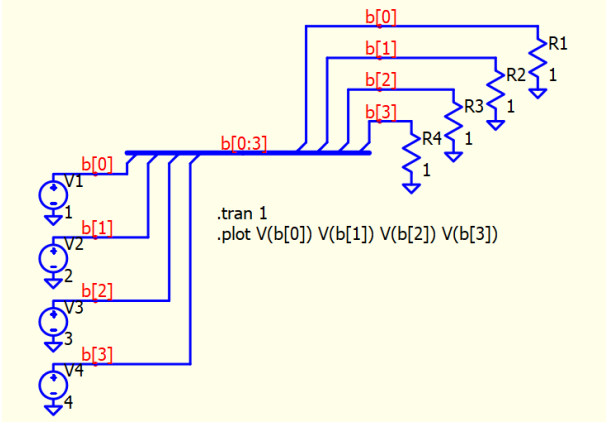
Basics	
Symbol Type	MP
Description	P-Channel MOSF...
Allow Shorted ...	False
Library File	PMOS.txt
String Attributes	
Text Order Invis.	Content
Name:	M2
1st attribute	PMOS
Pin Nets	
Pin Name Invis.	Net
D	1
G	2
S	3

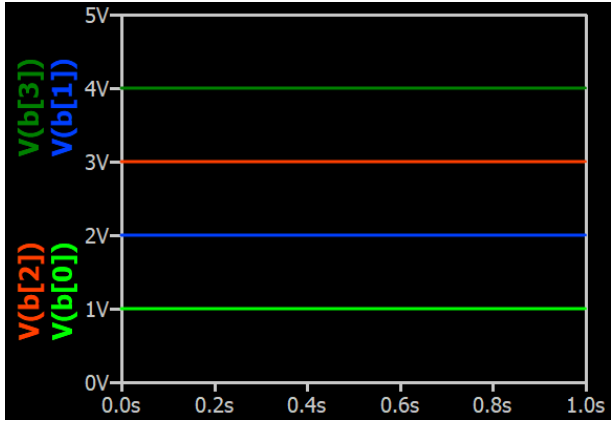
Part 2E
Bus and Hierarchical Block

Bus and Hierarchical Block

Qspice : Bus Basic.qsch

- Bus and Hierarchical Block
  - With Bus, data is defined as Data[n:m]
  - In Qspice, this net name format create a series of net names from Data[n] to Data[m]
    - If n<m, net name are Data[n], Data[n+1], Data[n+2], … , Data[m]
    - If n>m, net name are Data[n], Data[n-1], Data[n-2], … , Data[m]
  - For hierarchical block, subckt bus net names are assigned according to index sequence
  - To use data bus, it is recommending bus, hierarchical block and subckt with same data bus index, which can prevent unexpected behavior in net assignment

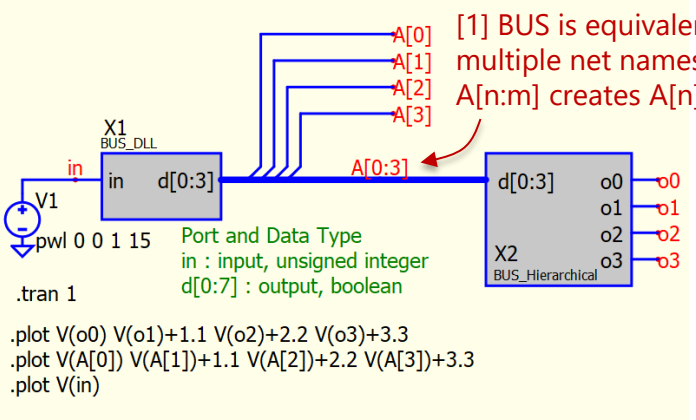




- Example
  - This example demonstrates how the bus can be set up. However, if the bus line is deleted in this example, the circuit still functions, as connections can be made with the same netname
  - The real usage of a bus is for hierarchical blocks.

Bus and Hierarchical Block

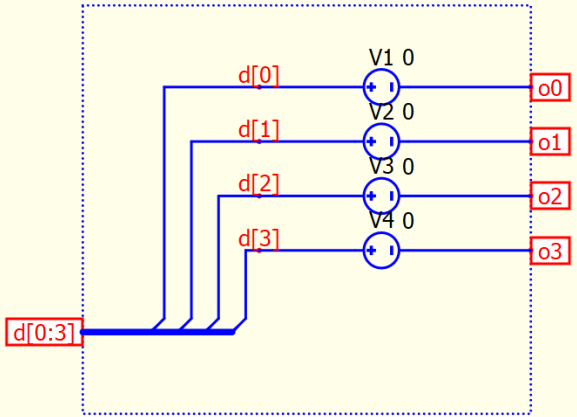
Qspice : Parent-BUS.qsch / BUS_Hierarchical.qsch / bus_dll.cpp

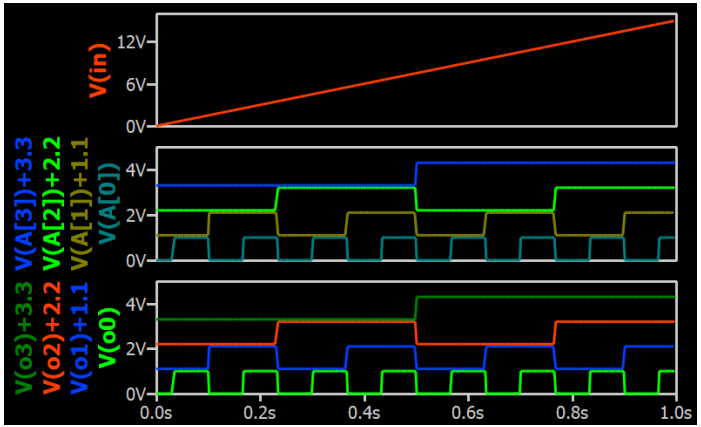


[1] BUS is equivalent to create multiple net names with []
A[n:m] creates A[n], A[n+1], ... A[m]

```
* C:\QspiceKSKelvin\01 User Guide and Script\01 Qspice Reference (
Ø†X1 «in´ui» «A[0]´b A[1]´b A[2]´b A[3]´b» «»  BUS_DLL
V1 in 0 pwl 0 0 1 15
X2 A[0] A[1] A[2] A[3] o0 o1 o2 o3 BUS_Hierarchical

.subckt BUS_Hierarchical d[0] d[1] d[2] d[3] o0 o1 o2 o3
V1 d[0] o0 0
V2 d[1] o1 0
V3 d[2] o2 0
V4 d[3] o3 0
.ends BUS_Hierarchical

.tran 1
.plot V(o0) V(o1)+1.1 V(o2)+2.2 V(o3)+3.3
.plot V(A[0]) V(A[1])+1.1 V(A[2])+2.2 V(A[3])+3.3
.plot V(in)
.end
```

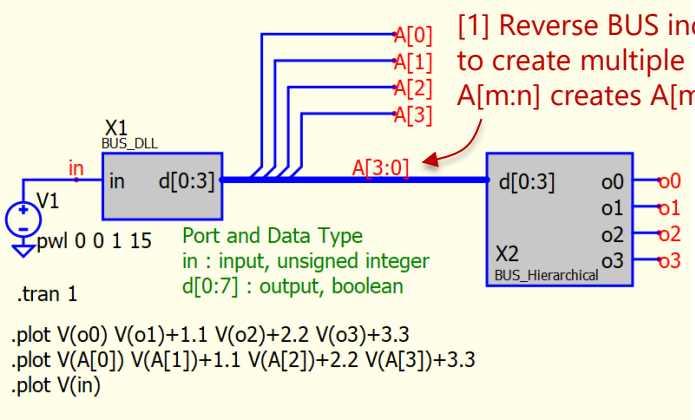
[2] Subckt X1, hierarchical X2 are all feed in same name and order





Bus and Hierarchical Block : Change BUS name order

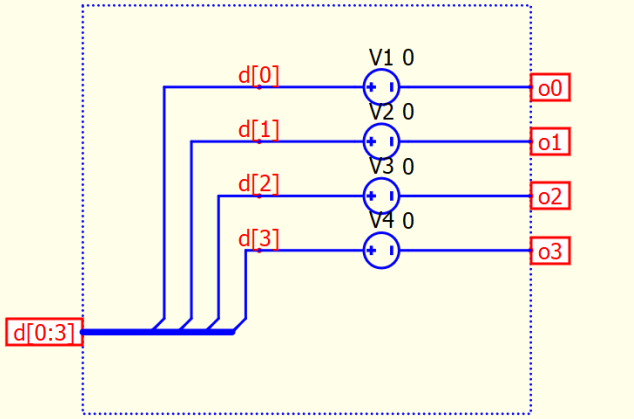
Qspice : Parent-BUS.qsch / BUS_Hierarchical.qsch / bus_dll.cpp

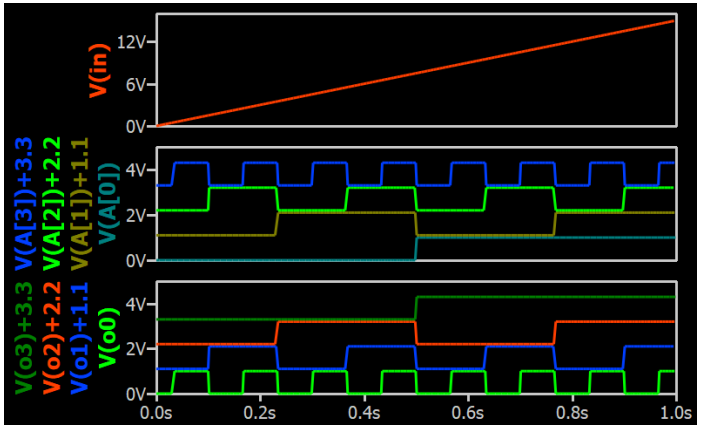


[1] Reverse BUS index is equivalent to create multiple net names with [] A[m:n] creates A[m], A[m-1], ... A[n]

```
* C:\QspiceKSKelvin\01 User Guide and Script\01 Qspice Reference (
ØtX1 «in´ui» «A[3]´b A[2]´b A[1]´b A[0]´b» «» BUS_DLL
V1 in 0 pwl 0 0 1 15
X2 A[3] A[2] A[1] A[0] o0 o1 o2 o3 BUS_Hierarchical

.subckt BUS_Hierarchical d[0] d[1] d[2] d[3] o0 o1 o2 o3
V1 d[0] o0 0
V2 d[1] o1 0
V3 d[2] o2 0
V4 d[3] o3 0
.ends BUS_Hierarchical

.tran 1
.plot V(o0) V(o1)+1.1 V(o2)+2.2 V(o3)+3.3
.plot V(A[0]) V(A[1])+1.1 V(A[2])+2.2 V(A[3])+3.3
.plot V(in)
.end
```

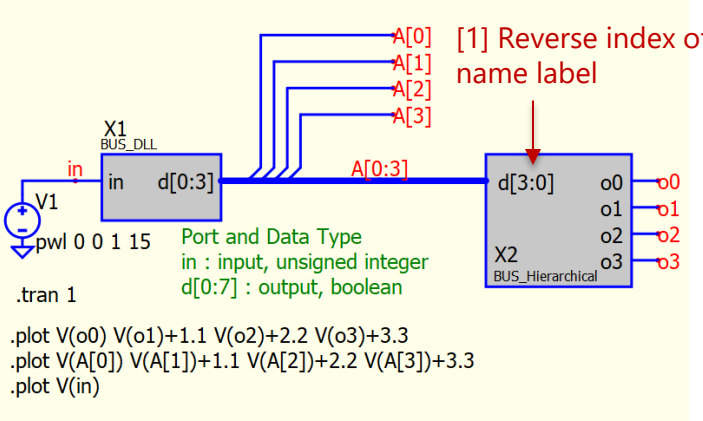
[2] Subckt X1-d[0] is connected to A[3]
[3] This order is feed into hierarchical block X2, e.g. A[3] is feed into X2-d[0]





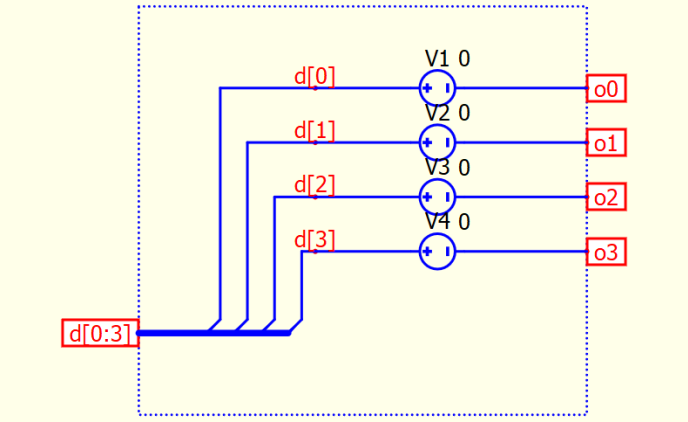
Bus and Hierarchical Block : Change Hierarchical net label order

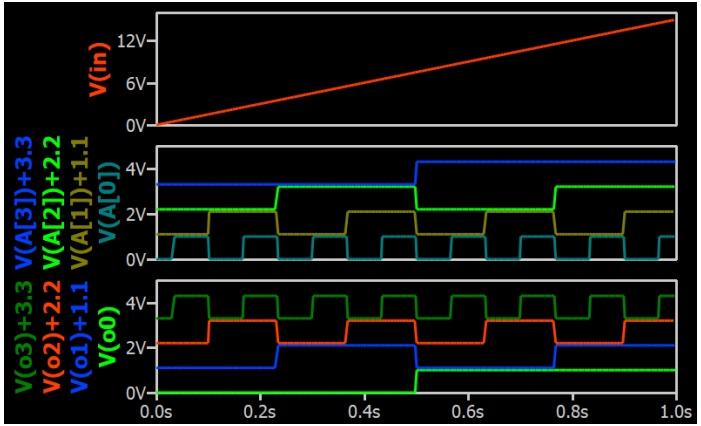
Qspice : Parent-BUS.qsch / BUS_Hierarchical.qsch / bus_dll.cpp




```
* C:\QspiceKSKelvin\01 User Guide and Script\01 Qspice Reference (
Ø†X1 «in´ui» «A[0]´b A[1]´b A[2]´b A[3]´b» «»  BUS_DLL
V1 in 0 pwl 0 0 1 15
X2 A[0] A[1] A[2] A[3] o0 o1 o2 o3 BUS_Hierarchical

.subckt BUS_Hierarchical d[3] d[2] d[1] d[0] o0 o1 o2 o3
V1 d[0] o0 0
V2 d[1] o1 0
V3 d[2] o2 0
V4 d[3] o3 0
.ends BUS_Hierarchical

.tran 1
.plot V(o0) V(o1)+1.1 V(o2)+2.2 V(o3)+3.3
.plot V(A[0]) V(A[1])+1.1 V(A[2])+2.2 V(A[3])+3.3
.plot V(in)
.end
```

[2] Hierarchical block X2 name is reversed, but Hierarchical / Subckt net assignment is based on order, therefore, A[0] is feed to hierarachical subckt d[3] in this case





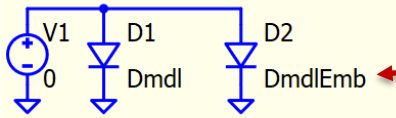
Part 2F
Device with .model

Device with .model

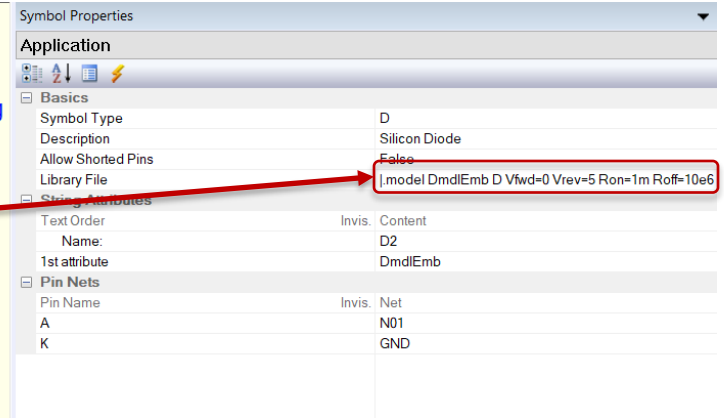
Qspice : Diode - .model and embedded .model.qsch

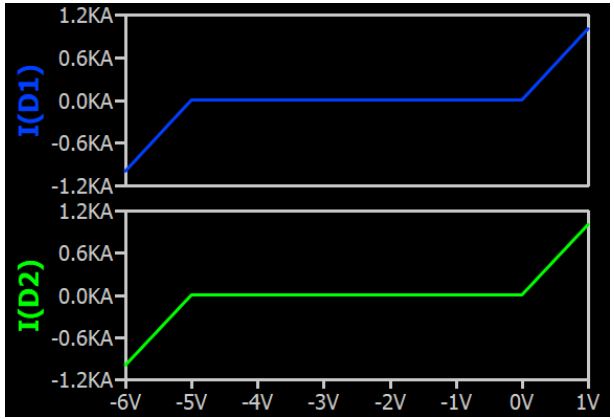
- Device with .model
  - .model can be added with a text in schematic
  - For embedded .model, the symbol character **"|"** need to be added at beginning of .model in Symbol Properties > Library File
    - Without symbol "|", .lib will be used in netlist to look for a library file name in this field
  - Alternative method
    - Copy .model statement, paste into schematic with autogenerate symbol method

Diode with Embedded .model
In Symbol Properties
Library File : Add .model with | at beginning



.model Dmdl D Vfwd=0 Vrev=5 Ron=1m Roff=10e6
.dc V1 -6 1 0.01
.plot I(D2)
.plot I(D1)



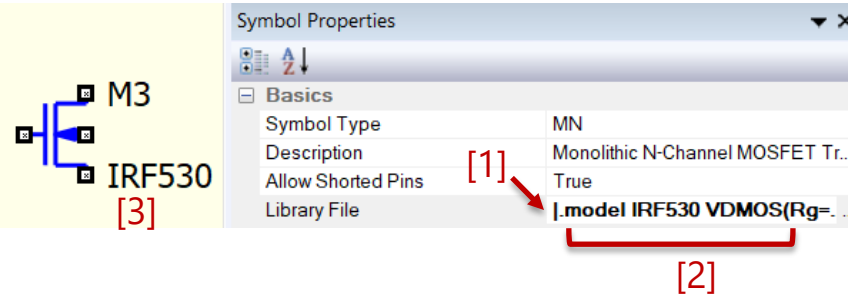


Netlist

```
V1 N01 0 0
.model D2•DmdlEmb D Vfwd=0 Vrev=5 Ron=1m Roff=10e6
D2 N01 0 D2•DmdlEmb
D1 N01 0 Dmdl
.dc V1 -6 1 0.01
.plot I(D2)
.plot I(D1)
.model Dmdl D Vfwd=0 Vrev=5 Ron=1m Roff=10e6
.lib C:\PROGRA~1\QSPICE\Diode.txt
.end
```

Import .model workflow

- Import .model workflow
  - Assume user copy a .model statement with Ctrl-C from library, for example
    - .model IRF530 VDMOS(Rg=3 Vto=4 Rd=50m Rs=12m Rb=60m Kp=5 lambda=.01 Cgdmax=1n Cgdmin=.26n Cgs=.2n Cjo=.4n Is=52p mfg=International_Rectifier Vds=100 Ron=160m Qg=26n)
  - Method #1 : Auto-generate symbol
    - In the schematic: Ctrl+V to paste text > auto-generate symbol > OK to paste an autogenerated symbol
  - Method #2 : Place a symbol and modify its [library file] field
    - In the schematic: Type M to cycle through MOS symbols, select a symbol with 3 or 4 terminals
    - Right-click on the symbol > Show Symbol Properties
    - [1] Go to "Library File" in Symbol Properties, type "|"
    - [2] After the "|" character, use Ctrl+V to paste text into "Library File" in Symbol Properties
    - [3] Change the name from NMOS to the model name, for example, in this case, IRF530



Part 2G
Special Topics about Creating Symbol

Special Topic #1 : Symbol from Multiple Hierarchical Layers

Qspice : \2G Special Topics about Creating Symbol\01 Multiple Hierarchical Layers\

Autogenerate symbol from multiple hierarchical layers

[1] X1 in parent schematic calls multiple layers of hierarchical

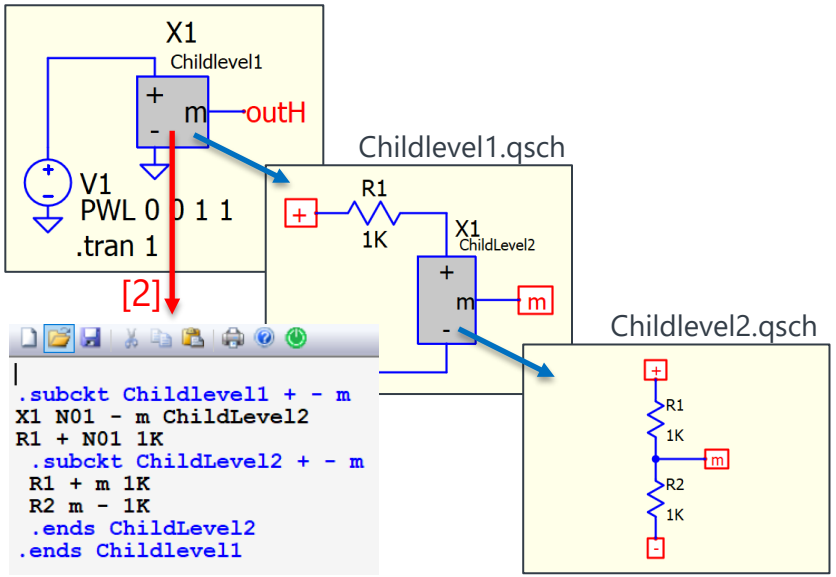
[2] In parent schematic

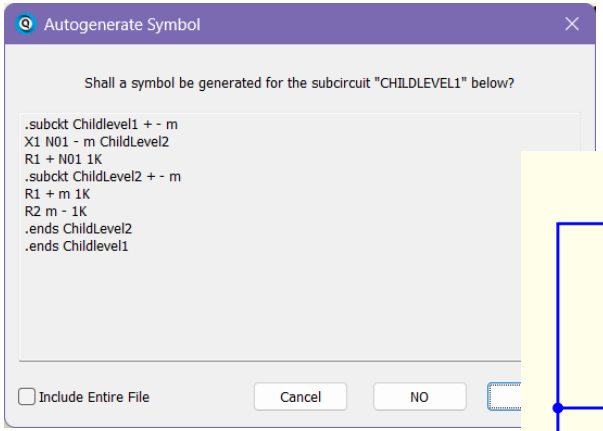
- Right Click X1 > **Netlist Subcircuit**
- **Ctrl+A** to select all text in netlist subcircuit, **Ctrl-C** to copy

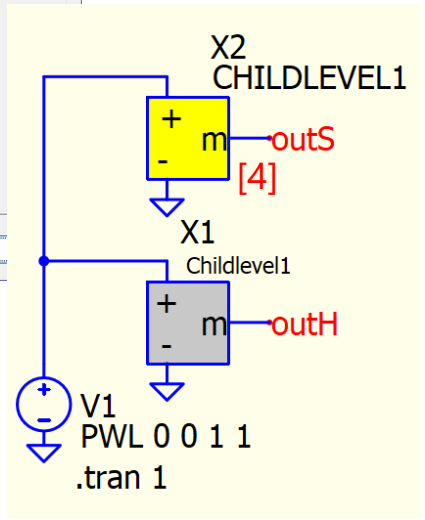
[3] In parent schematic, **Ctrl-V** to paste for autogenerate symbol

[4] An autogenerate symbol is created (subcircuits is embedded into this circuit, goto **Symbol Properties** > **Library File** to review it

Parent.TopLevelSchematic.qsch







Special Topic #2 : Symbol from Hierarchical and DLL

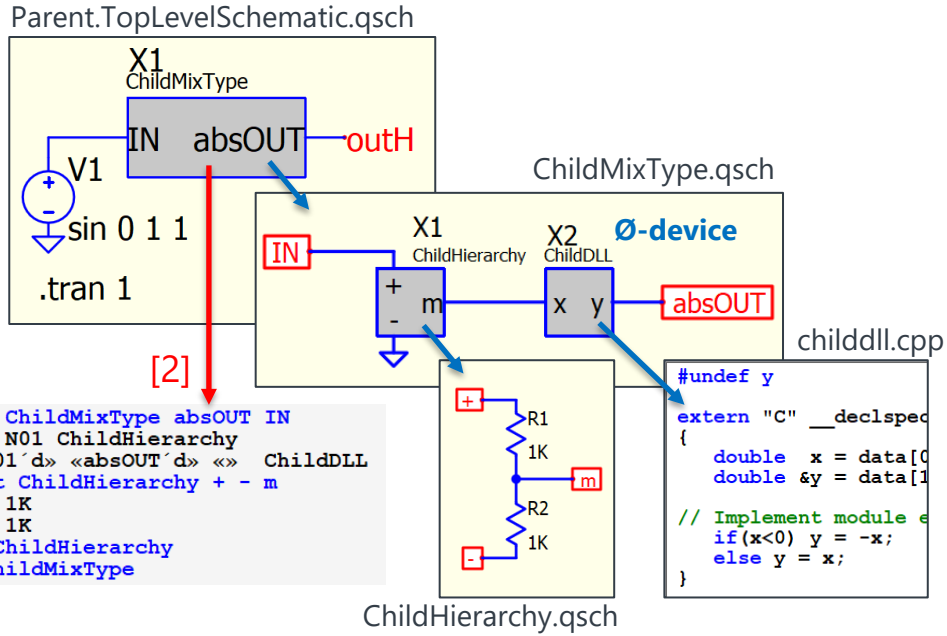
Qspice : \2G Special Topics about Creating Symbol\02 Hierarchical and DLL\

Autogenerate symbol from mixed type block

[1] X1 in parent schematic calls mixed type (hierarchical, Ø-device)

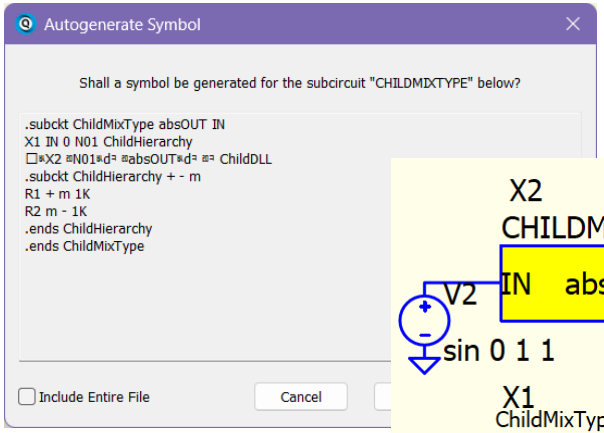
[2] In parent schematic

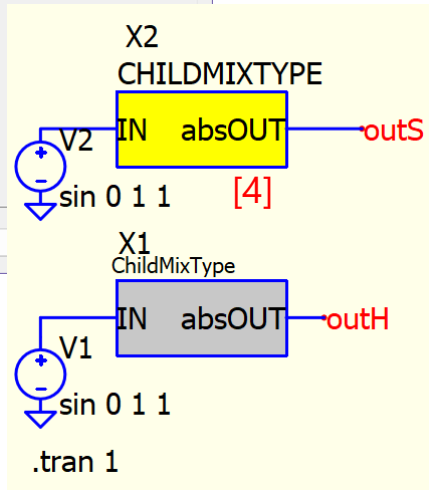
- Right Click X1 > **Netlist Subcircuit**
- **Ctrl+A** to select all text in netlist subcircuit, **Ctrl-C** to copy



[3] In parent schematic, **Ctrl-V** to paste for autogenerate symbol

[4] An autogenerate symbol is created (subcircuits is embedded into this circuit, goto **Symbol Properties** > **Library File** to review it

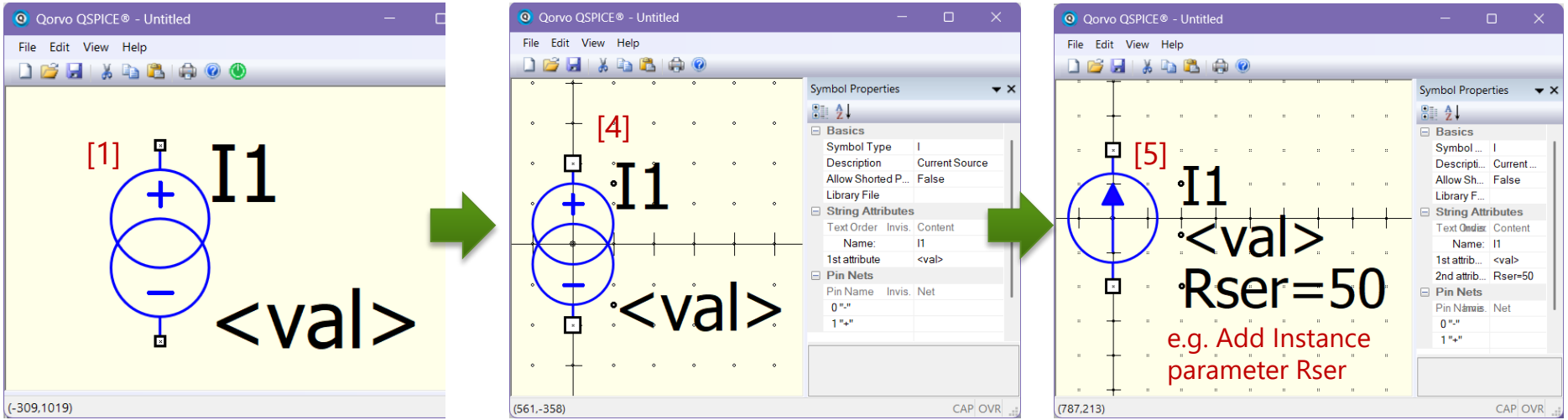




Special Topic #3 : Create an alternative native device symbol

Create an alternative Qspice native device symbol

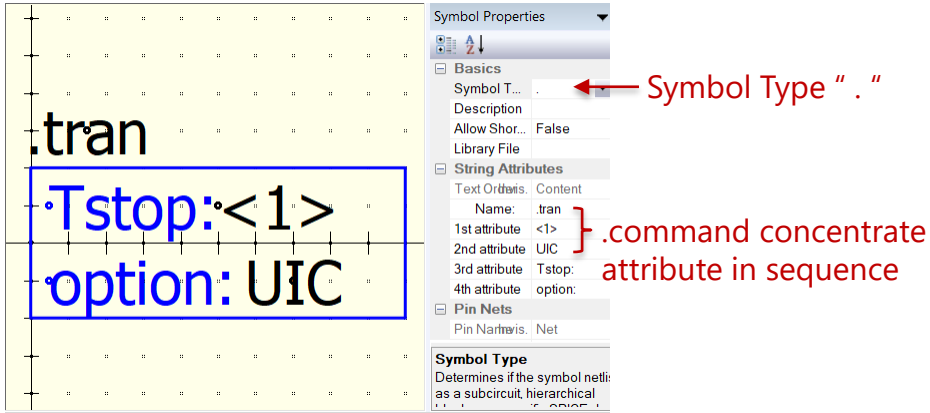
[1] In **Schematic Window**, place a Qspice native device
[2] Copy this device with **Ctrl-C**
[3] **File** > **New Symbol**
[4] In **Symbol Window**, **Ctrl-V** to paste the native device
 - In Symbol Properties, you can verify symbol type, pin nets etc…
[5] Now, redraw symbol (don't mess up pins). You can use text (**T**) to add new attribute for instance parameters
[6] **Save** this symbol in your symbol folder

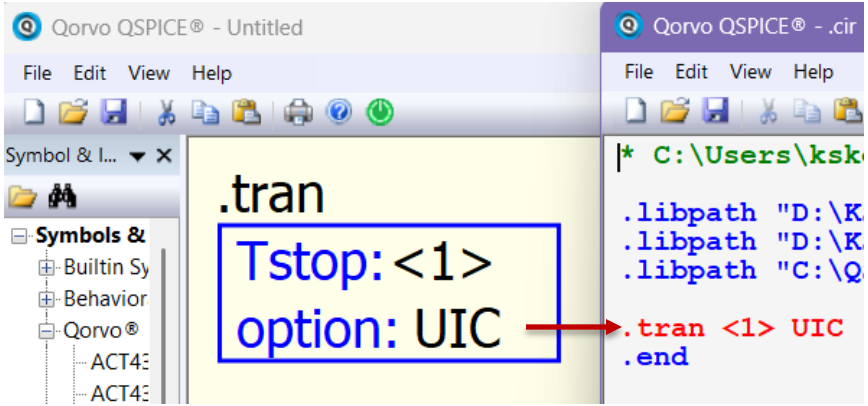


Special Topic #4 : Symbol Type "." (.command)

Qspice : \2G Special Topics about Creating Symbol\04 Symbol Type .command\

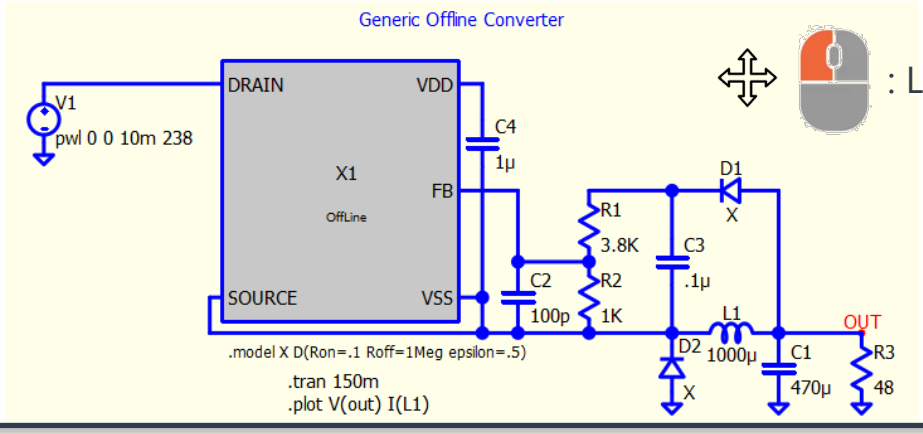
- Symbol Type "." (.command)
  - This symbol type enables the inclusion of active text attributes to generate a one-line netlist
  - No pins are required for this symbol type
  - Command syntax can be constructed using this symbol structure
  - ** Symbol Type "." doesn't exist in pull-down menu, need manually type "." in Symbol Type





Part 3
Schematic Viewer

Move, Zoom-In and Out Schematic, Copy/Cut/Paste and Selection

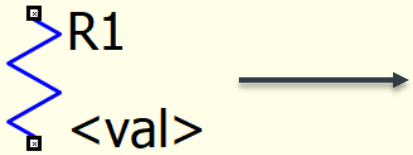


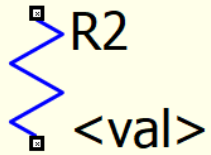
: Left Click (hold) background to move around

Space : Zoom to Fit

: Zoom In

: Zoom Out



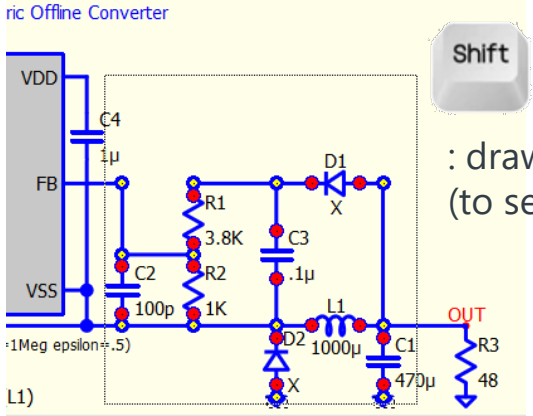


Mouse hover over device

Ctrl + C : copy

Ctrl + X : cut

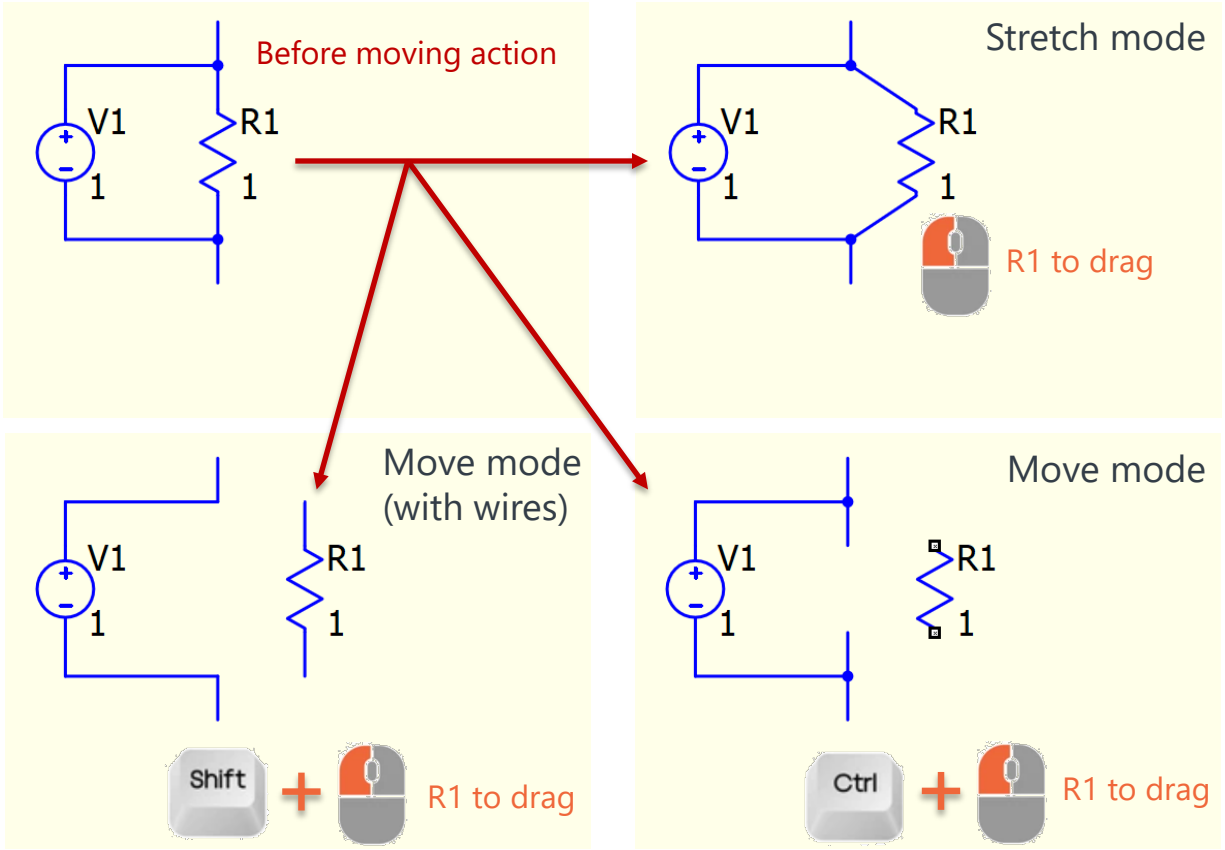
Ctrl + V : paste



Shift + : draw selection box (to select multiple elements)

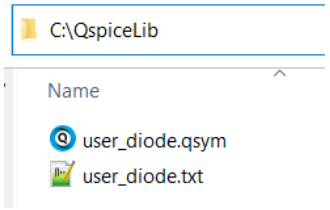
Drag a Component and handling multiple components

- Drag a Component
  - Combine with Shift or Ctrl Key, drag a single component can have different behavior
- Multiple components
  - Hold Shift Key and Left click can draw a selection box to select multiple components
  - Drag to move, or use Ctrl-X (cut) and Ctrl-V (paste) shortcut to separate it from circuit

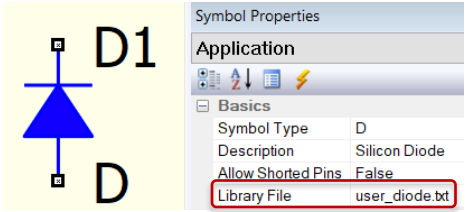


Eliminate the use of absolute library path in user-symbols

1. Assume user has a user-supplied symbols and library files in C:\QspiceLib

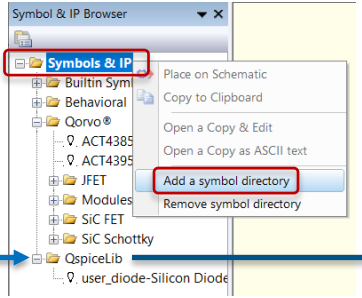


[1] Symbol and Library in same directory

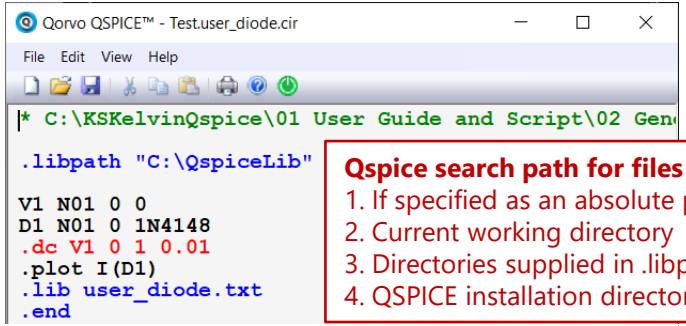


[2] Library File in user symbol is library name (Not absolute path)

2. In Qspice : View > Symbol & IP Browser
   - Right click "Symbols & IP", Add a symbol directory
   - Add C:\QspiceLib

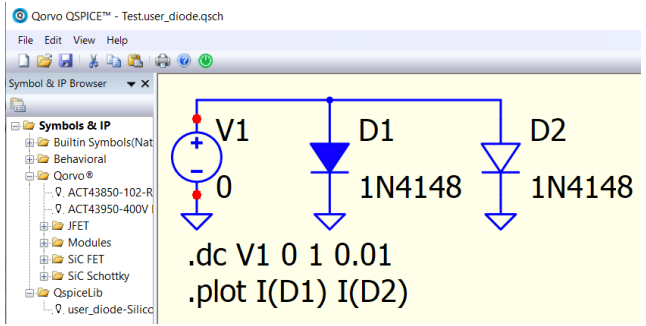


When add a symbol directory, it will add following in netlist **.libpath "C:\QspiceLib"**

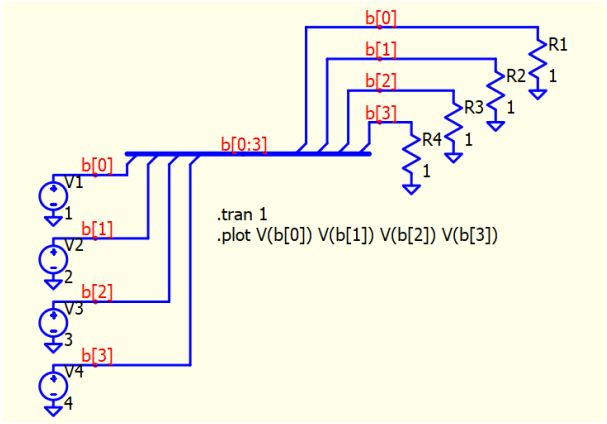


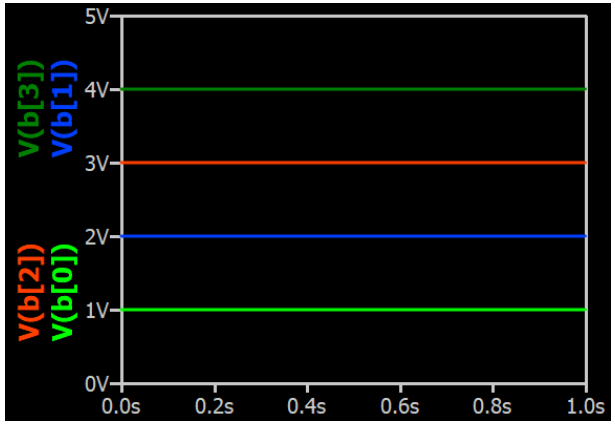
Qspice search path for files
1. If specified as an absolute path, only that is used.
2. Current working directory
3. Directories supplied in .libpath command(s)
4. QSPICE installation directory

3. Draw a schematic in other folder
   - Now you can use user-symbol or Qspice symbol but change library file from diode.txt to user_diode.txt



- Bus


b[0]
b[1]
b[2]
b[3]
R1
R2
R3
R4
1
b[0:3]
b[0]
V1
1
b[1]
V2
2
b[2]
V3
3
b[3]
V4
4
.tran 1
.plot V(b[0]) V(b[1]) V(b[2]) V(b[3])

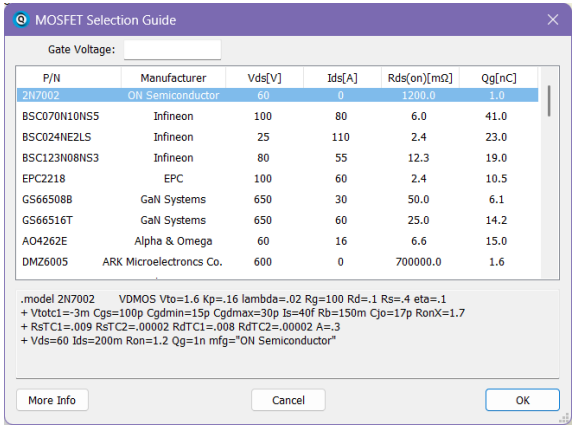

5V
4V
3V
2V
1V
0V
V(b[3])
V(b[1])
V(b[2])
V(b[0])
0.0s
0.2s
0.4s
0.6s
0.8s
1.0s

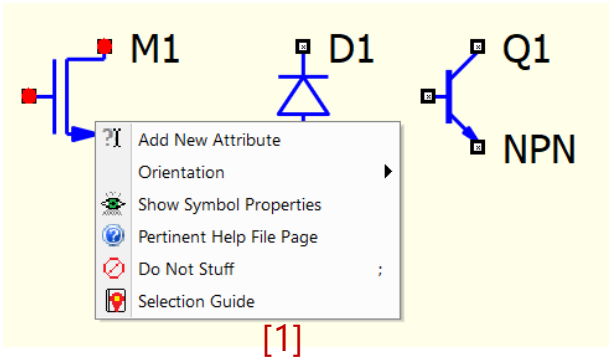
Part 3 – Appendix A
Schematic Viewer

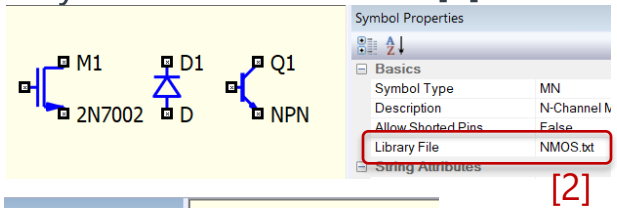
Selection Guide

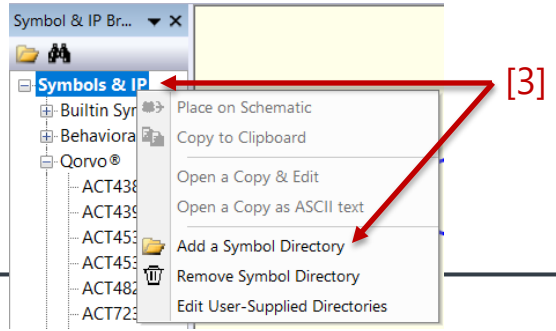
Selection Guide for Qspice Native Devices

- Selection Guide
  - Selection Guide allows user to select a device model by right click a device with a selection guide window [1] for libraries in Qspice library search path.
  - Selection Guide window only available for 4 type of native devices MOSFET (M), Transistor (Q), JFET (J), Diode (D)
  - Without a library file name specified in Library File property, selection guide will not be available, as models listed in selection guide are from this specified library file [2]
    - For selection guide to work, this must be *library file name only*, and *no directory path included*
  - This library file MUST be in Qspice library search path : Installation Directory (C:\Program Files\QSPICE) or its **directory path** added into Symbol Directory in Symbols & IP Broswer [3]



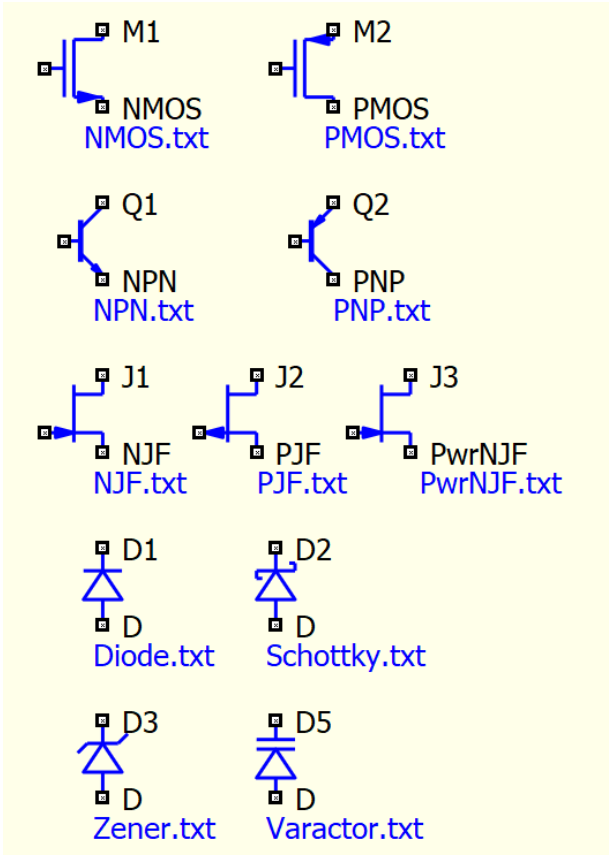


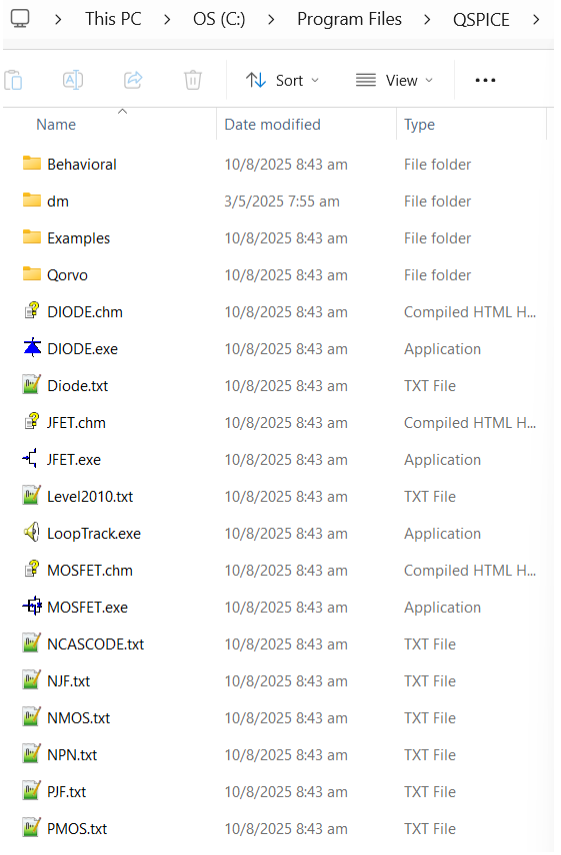




Selection Guide for Qspice Native Devices (Default Library)

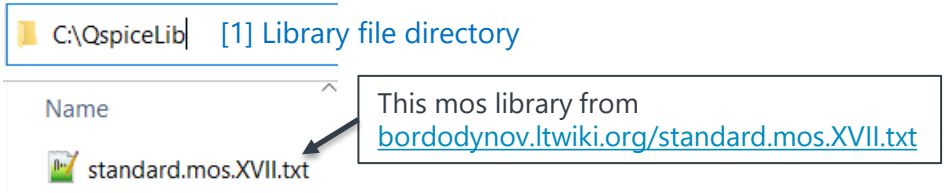
- Default Library
  - Qspice default libraries are located in its installation directory (C:\Program Files\QSPICE)
  - Native devices reference with their library file names in their library file properties
  - User can directly modify these library files to expand the selection guide list, but beware Qspice default library will be deleted during update or reinstalling Qspice
  - User can copy library files into installation directory. Update or reinstalling Qspice will NOT delete these files
  - User can also put file in other directory and add search path from Symbol & IP





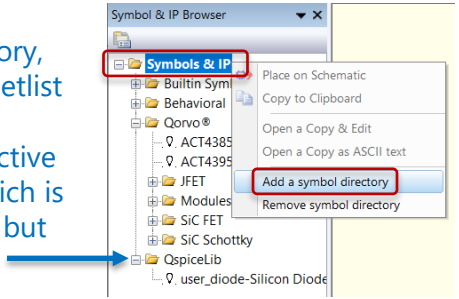
#1: Selection Guide for native symbols with user-supplied library files

1. Assume user has a user-supplied library file in C:\QspiceLib



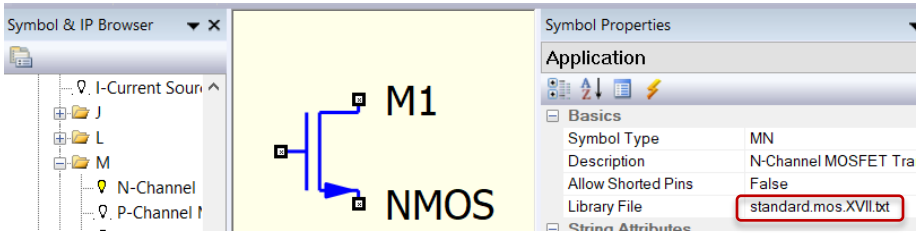
2. In Qspice : View > Symbol & IP Browser
   - Right click “Symbols & IP”, Add a symbol directory
   - Add C:\QspiceLib

When add a symbol directory, it represents following in netlist
.libpath "C:\QspiceLib"
[in View > Netlist, this directive is automatically added, which is not visible from schematic but only netlist]



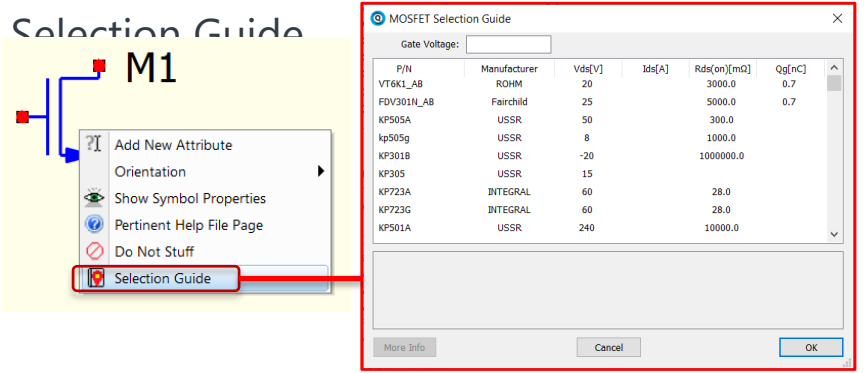
** If a path without .qsym, it will not show in Symbols & IP browser. But the path can still be added. A successfully added path can be found by **Right Click > Edit User-Supplied Directories**

3. Place Qspice symbol to schematic
   - Place Qspice symbol, replace Library File name with user-supplied library name (must be **filename only, no directory path!**)



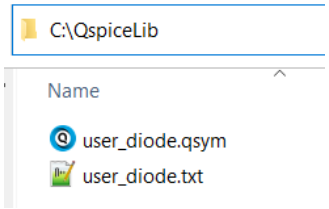
In this example, replace NMOS.txt to standard.mos.XVII.txt

4. Selection Guide

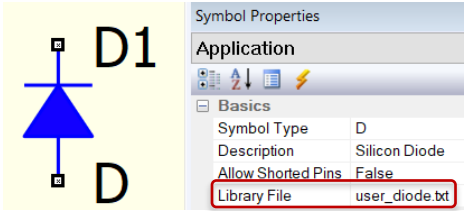


#2: Selection Guide for user-supplied symbols and library files

1. Assume user has a user-supplied symbol and library files in C:\QspiceLib

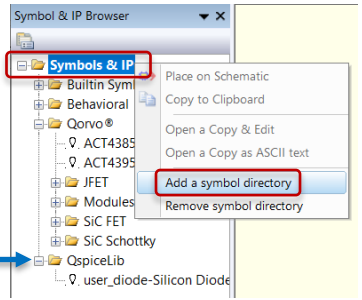


[1] Symbol and Library in same directory



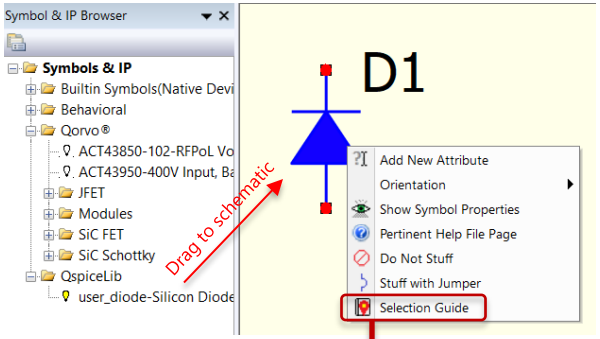
[2] Library File in user symbol is library name (e.g. user_diode.txt)

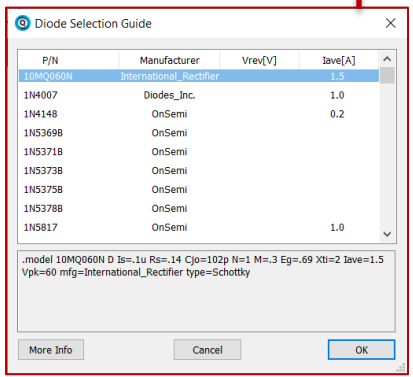
2. In Qspice : View > Symbol & IP Browser
   - Right click “Symbols & IP”, Add a symbol directory
   - Add C:\QspiceLib



When add a symbol directory, it will add following in netlist
.libpath "C:\QspiceLib"

3. Drag user symbol to schematic
   - Right click on symbol, “Selection Guide” is available

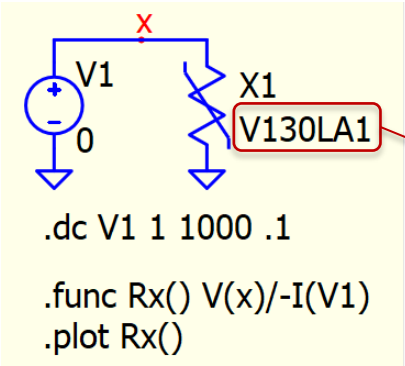


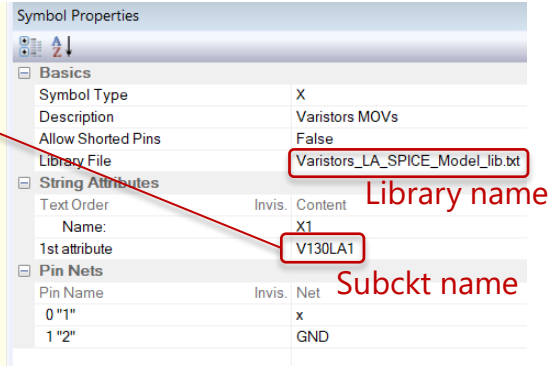


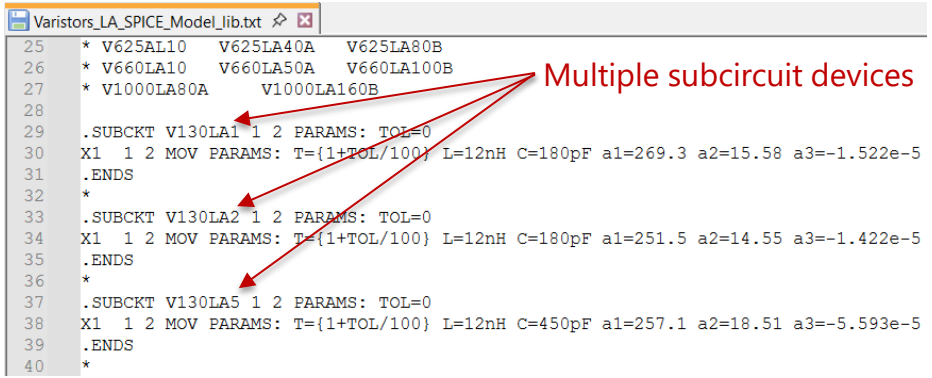
User-supplied symbols still have to be M, Q, J, or D. However, it allows for a pre-written library file to be assigned to these symbols, eliminating the need for the user to rename the library file from the native device each time.

Subcircuit Picker for libraries in library search path

- Selection Guide to pick from subcircuits (.subckt)
  - Selection Guide supports subcircuit libraries for X-devices
  - It is common for a subcircuit library (.lib or .txt) to contain multiple subcircuit devices, all of which have the same pin net numbers and sequence. Users only need to change the 1st attribute of the symbol to select a device
  - Instead of opening the library to navigate a .subckt model name and typing its into the 1st attribute, the Selection Guide in QSpice can act as a subcircuit picker if the library is in the library search path

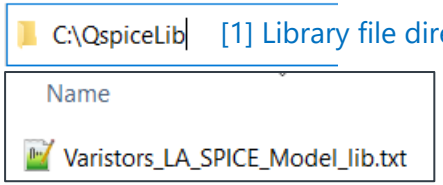






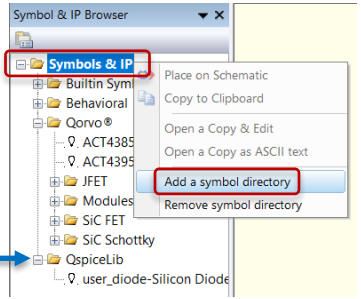
Subcircuit Picker for libraries in library search path

1. Assume user has a subckt library file in C:\QspiceLib



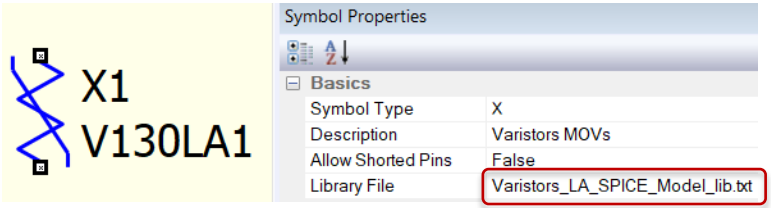
2. In Qspice : View > Symbol & IP Browser
   - Right click “Symbols & IP”, Add a symbol directory
   - Add C:\QspiceLib

When add a symbol directory, it represents following in netlist **.libpath "C:\QspiceLib"** [in View > Netlist, this directive is automatically added, which is not visible from schematic but only netlist]

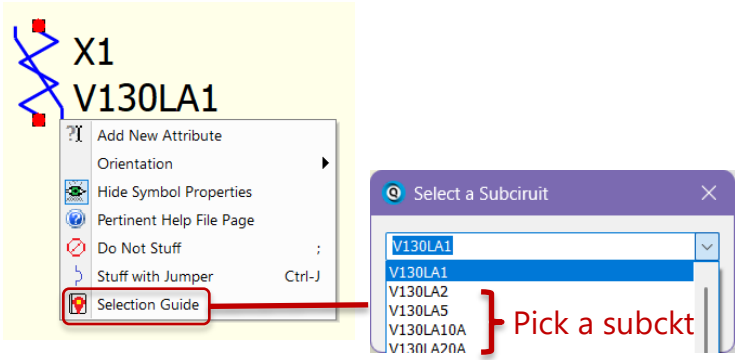


** If a path without .qsym, it will not show in Symbols & IP browser. But the path can still be added. A successfully added path can be found by **Right Click** > **Edit User-Supplied Directories**

3. Place subckt symbol to schematic
   - Place suckt symbol with Library File name included in its Symbol Properties (must be **filename only, no directory path!**)



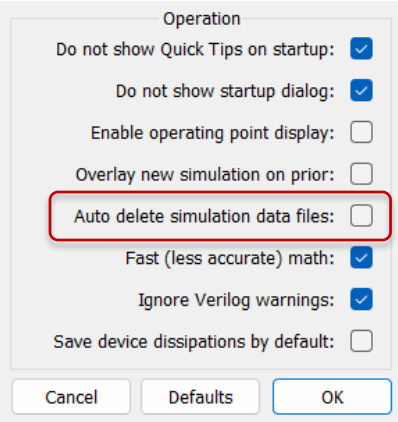
4. Selection Guide

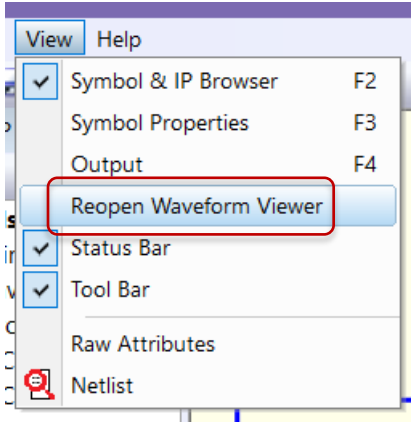


Part 4
Waveform Viewer

Waveform Viewer Workflow from GUI

- Waveform Viewer Workflow from GUI
  - When schematic (.qsch) is run, QUX.exe will convert it into a netlist (.cir) and pass to Qspice64/Qspice80.exe for simulation. Output results is stored to waveform data file (.qraw)
  - QUX.exe executes a waveform viewer and load this .qraw file to plot waveform data
  - .qraw remains in current directory until rerun or auto delete when schematic is closed (this option can be disable in Edit > Preference > Auto delete simulation data files
  - If waveform viewer is closed but schematic remain open, .qraw will still exist. Waveform viewer can be reopened by View > Reopen Waveform Viewer





Waveform Viewer Plot Config File (*.pfg) and .plot directive

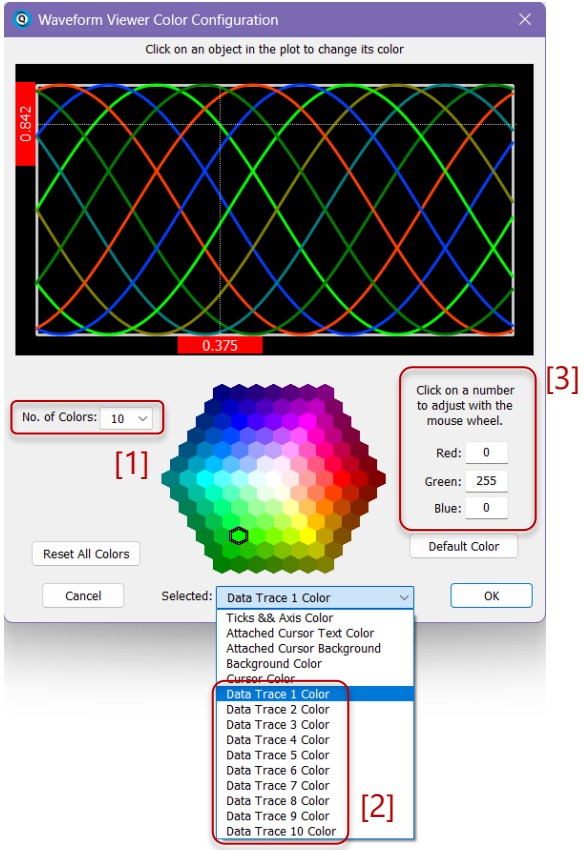
- Waveform Viewer Config File (*.pfg) and .plot
  - In waveform viewer, plot config can be saved with File > Save Config : [qschname].pfg
    - This config file save windows, traces and axis setting
    - Press spacebar in waveform viewer can re-load config file [qschname].pfg
  - Two unique feature [qschname].pfg can provide but not support by .plot
    - Pre-define x-axis Quantity
    - Pre-define x and y-axis range

Waveform Viewer	Plot Config File [1] [qschname].pfg	.plot command in schematic	Outcome
Closed before Simulation	No	No	A blank waveform viewer
	No	Yes	Plot according to .plot command
	Yes	[ignore]	Plot according to [qschname].pfg config
Opened before Simulation	[ignore]	[ignore]	Keep windows and traces setting from last plot, reset x and y-axis
[1] Save plot config in Waveform Viewer : File > Save Config			
** In Waveform Viewer, press Spacebar to reload [qschname].pfg plot config file			
** to use .plot, delete [qschname].pfg and close waveform viewer before run of simulation			

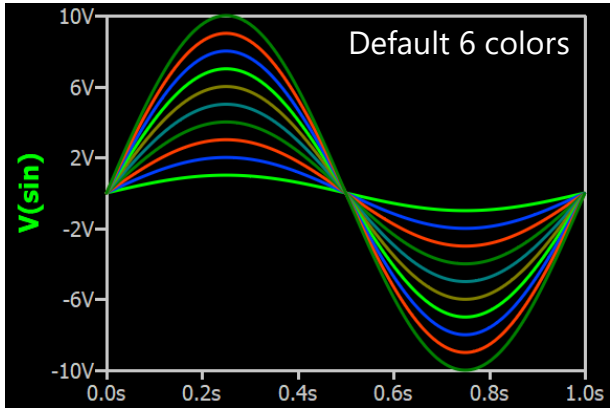
Waveform Viewer – Colors for Trace

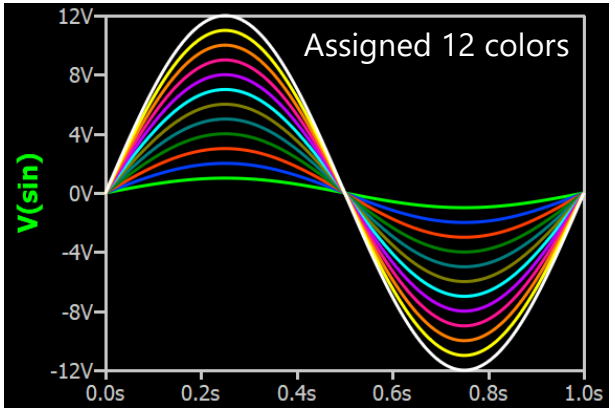
Qspice : \color curve\color curve.qsch

- Colors of Trace
  - In default, no. of colors is 6 and trace color repeat after that
- Change Trace colors
  - Maximum of 24 colors can be assigned to data traces
    - Users have to assign colors themselves starting from trace 7 onwards
  - In waveform viewer
    - Edit > Color Preference
    - Select No. of Colors [1]
    - In Selected, select "Data Trace n Color" [2] and manually assign a color code [3]



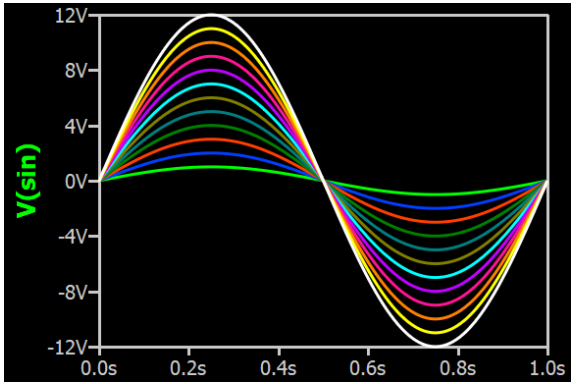
[3]





Waveform Viewer – Colors for Trace (Color RGB) Dark theme

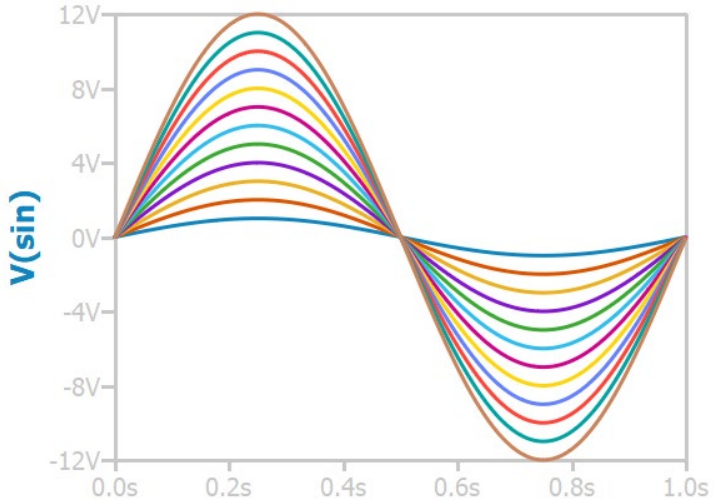
- Dark background color traces example
  - Here is the list of default data trace 1 to 6 (HEX code is for Registry modification)
    - Pure Green (1) : RGB(0,255,0) or 0x00FF00
    - Bright Blue (2) : RGB(0,63,255) or 0xFF3F00
    - Vivid Orange-Red (3) : RGB(255,63,0) or 0x003FFF
    - Green (4) : RGB(0,128,0) or 0x008000
    - Teal (5) : RGB(0,127,128) or 0x807F00
    - Olive Gold (6) : RGB(128,127,0) or 0x007F80
  - Here is the list of colors demonstrated in the previous slide for data trace 7 to 12
    - Neon Cyan (7) : RGB(0,255,255) or 0xFFFF00
    - Electric Purple (8) : RGB(191,0,255) or 0xFF00BF
    - Bright Pink (9) : RGB(255,20,147) or 0x9314FF
    - Vivid Orange (10) : RGB(255,128,0) or 0x0080FF
    - Glowing Yellow (11) : RGB(255,255,0) or 0x00FFFF
    - White (12) : RGB(255,255,255) or 0xFFFFFF
    - Electric Blue : RGB(0,150,255) or 0xFF9600
    - Blood Red : RGB(230,0,50) or 0x3200E6



Waveform Viewer – Colors for Trace (Color RGB) Light theme

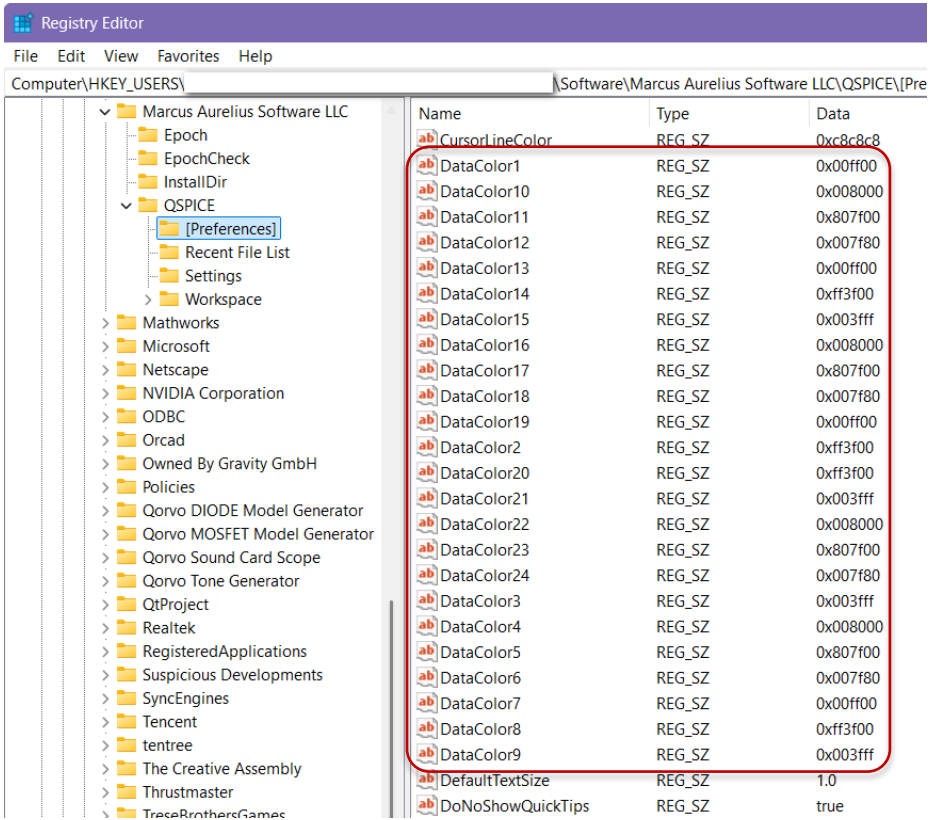
- Light background color traces
  - Reference to Matlab color palettes with orderedcolors(“gem12”)

- (1) : RGB(17,133,190) or 0xbe8511
- (2) : RGB(221,84,0) or 0x0054dd
- (3) : RGB(237,177,32) or 0x20b1ed
- (4) : RGB(133,22,209) or 0xd11685
- (5) : RGB(59,170,50) or 0x32aa3b
- (6) : RGB(47,190,239) or 0xefbe2f
- (7) : RGB(209,4,139) or 0x8b04d1
- (8) : RGB(255,214,10) or 0x0ad6ff
- (9) : RGB(101,130,253) or 0xfd8265
- (10) : RGB(255,69,58) or 0x3a45ff
- (11) : RGB(0,163,163) or 0xa3a300
- (12) : RGB(203,132,92) or 0x5c84cb



Waveform Viewer – Colors for Trace (in Registry)

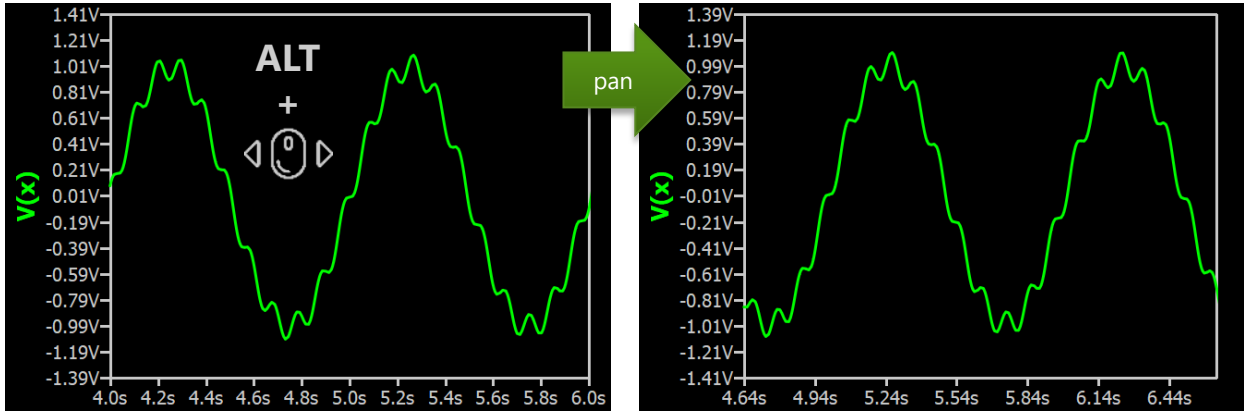
- Colors for Trace in Registry
  - Waveform Viewer are stored in regedit
  - Procedure to find regedit location
    - Run Registry Editor
    - Select HKEY_USERS
    - Edit > Find (Ctrl-F)
      - Find what: qspice
      - Look at: Keys only
    - Registry location:
      - Computer\HKEY_USERS\%%%%%\SOFTWARE\Marcus Aurelius Software LLC\QSPICE
    - Colors of Trace : DataColor1 to DataColor24
  - Default Color Code (0xBBGGRR)
    - DataColor1 : 0x00FF00 (B=0, G=255, R=0)
    - DataColor2 : 0xFF3F00 (B=255, G=63, R=0)
    - DataColor3 : 0x003FFF (B=0, G=63, R=255)
    - DataColor4 : 0x008000 (B=0, G=128, R=0)
    - DataColor5 : 0x807F00 (B=128, G=127, R=0)
    - DataColor6 : 0x007F80 (B=0, G=127, R=128)



Waveform Viewer – Panning and Zooming

Qspice : waveform - panning.qsch

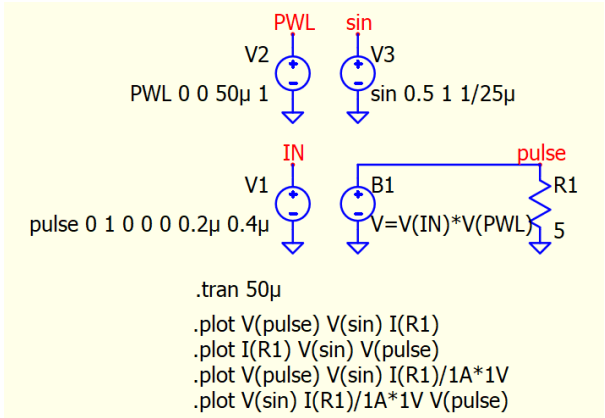
- Panning
  - In plot window, hold ALT + move cursor to pan (mutli-direction)
  - The plot must be zoom before it can pan
- Zooming
  - In plot window, hold ALT + mouse scroll



Waveform Viewer – Trace Display Sequence

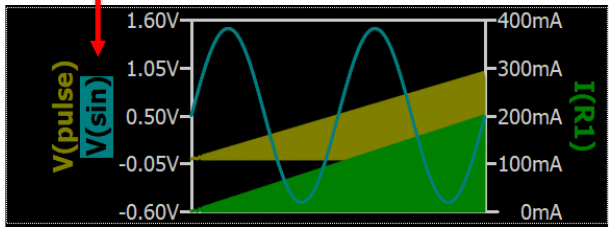
Qspice : waveform - trace overlay sequence.qsch

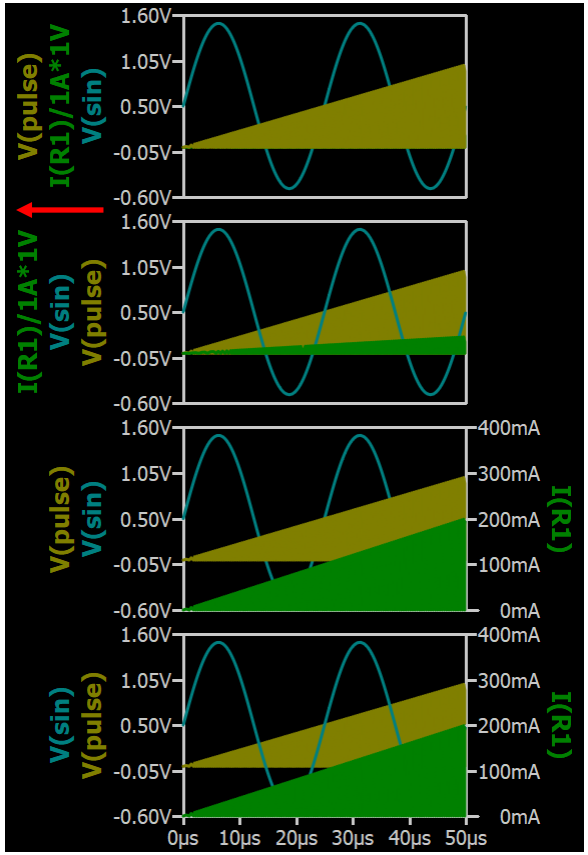
- Trace Display Sequence
  - Adding trace by cross probing
    - The first added trace is at the bottom, and the last added trace is at the top
  - Adding trace with .plot
    - Expression sequence in .plot determines display sequence
    - The leftmost expression is at the bottom, and the rightmost expression is at the top (**except when traces are in different units)
  - Temporary move to top
    - Click trace label in waveform viewer can temporary move pieces to top



Bottom to Top
(except they have different units)

Click the trace to move it on top temporary

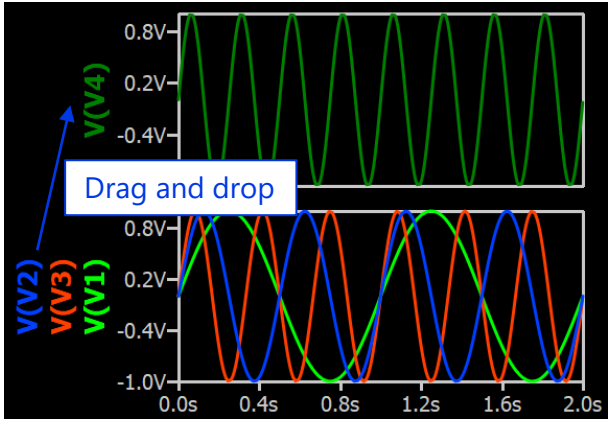


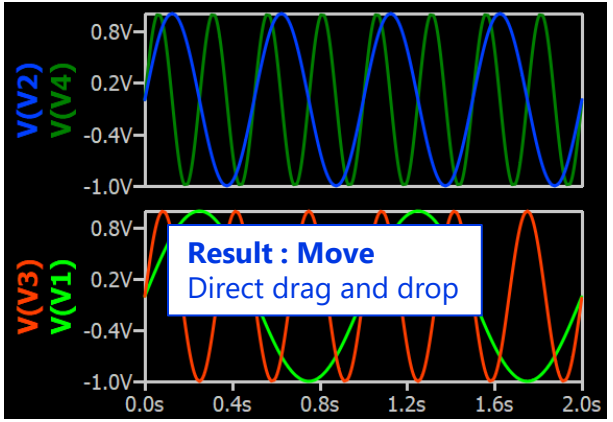


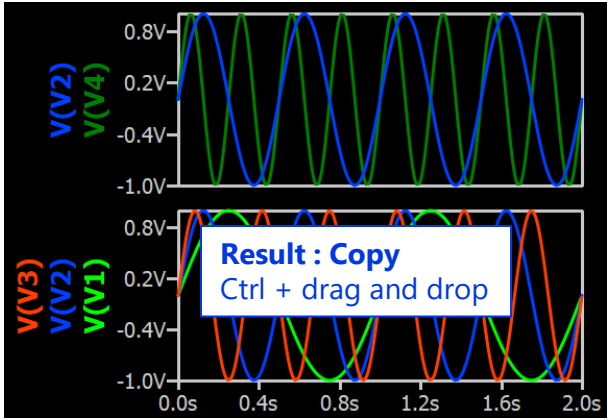
Traces manipulation

Qspice : waveform - drag traces.qsch

- Traces manipulation
  - Delete
    - Click a trace label, press delete
  - Cut and Paste
    - Click a trace label
    - Copy : Ctrl+C
    - Cut : Ctrl+X
    - Paste : Ctrl+V
  - Drag and Drop (Move)
    - Left click to drag and drop
  - Drag and Drop (Copy)
    - Hold Ctrl + Lef click to drag and drop

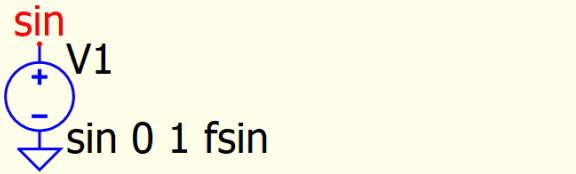






Data Export in Waveform Viewer – with @ in expression for .step

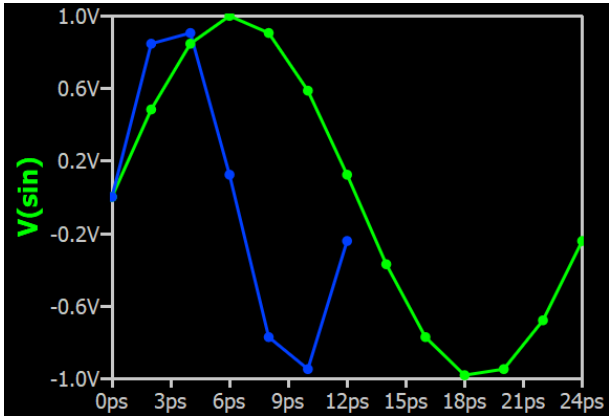
Qspice : waveform - with @ for step.qsch




```
.step param fsin list 40G 80G
.tran 1/fsin
.plot V(sin)
```

- Data Export
- Setup Data Export
  - File > Export Data
  - Number Points : All
  - Expression(s) : V(sin),FSIN

- Data Export with @
- Setup Data Export
  - File > Export Data
  - Number Points : All
  - Expression(s) : V(sin)@1,V(sin)@2



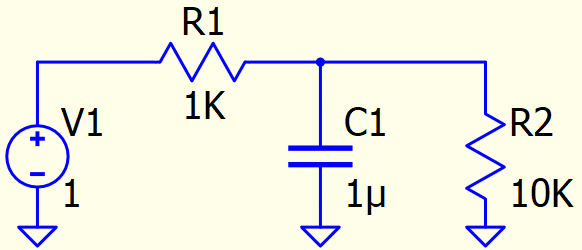

```
Time,V(sin),FSIN
0,0,40000000000
2.001953125e-12,0.482183772079123,40000000000
4.00390624999999e-12,0.844853565249706,40000000000
6.00585937500001e-12,0.998118112900149,40000000000
8.00781250000003e-12,0.903989293123441,40000000000
1.0009765625e-11,0.58579785745643,40000000000
1.20117187500001e-11,0.122410675199201,40000000000
1.40136718750001e-11,-0.371317193951856,40000000000
1.6015625e-11,-0.773010453362737,40000000000
1.80175781249999e-11,-0.983105487431211,40000000000
2.00195312499998e-11,-0.949528180593055,40000000000
2.20214843749997e-11,-0.680600997795516,40000000000
2.40234374999995e-11,-0.242980179903377,40000000000
0,0,80000000000
2.001953125e-12,0.844853565249706,80000000000
4.00390625000001e-12,0.903989293123441,80000000000
6.00585937500003e-12,0.122410675199201,80000000000
8.0078125e-12,-0.773010453362736,80000000000
1.00097656249999e-11,-0.949528180593055,80000000000
1.20117187499998e-11,-0.242980179903377,80000000000
```

```
Time,V(sin)@1,V(sin)@2
0,0,0
2.001953125e-12,0.482183772079123,0.844853565249706
4.00390624999999e-12,0.844853565249706,0.903989293123441
6.00585937500001e-12,0.998118112900149,0.122410675199201
8.00781250000003e-12,0.903989293123441,-0.773010453362739
1.0009765625e-11,0.58579785745643,-0.949528180593
1.20117187500001e-11,0.122410675199201,-0.596254180248215
1.40136718750001e-11,-0.371317193951856,-0.596254180248215
1.6015625e-11,-0.773010453362737,-0.596254180248215
1.80175781249999e-11,-0.983105487431211,-0.596254180248215
2.00195312499998e-11,-0.949528180593055,-0.596254180248215
2.20214843749997e-11,-0.680600997795516,-0.596254180248215
2.40234374999995e-11,-0.242980179903377,-0.596254180248215
0,0,0
2.001953125e-12,0.482183772079122,0.844853565249706
4.00390625000001e-12,0.844853565249707,0.903989293123441
6.00585937500003e-12,0.998118112900148,0.122410675199201
8.0078125e-12,0.903989293123442,-0.773010453362736
1.00097656249999e-11,0.585797857456455,-0.949528180593055
1.20117187499998e-11,0.122410675199269,-0.242980179903377
```

Snapshot Data Method – Export Data with Single Number Points

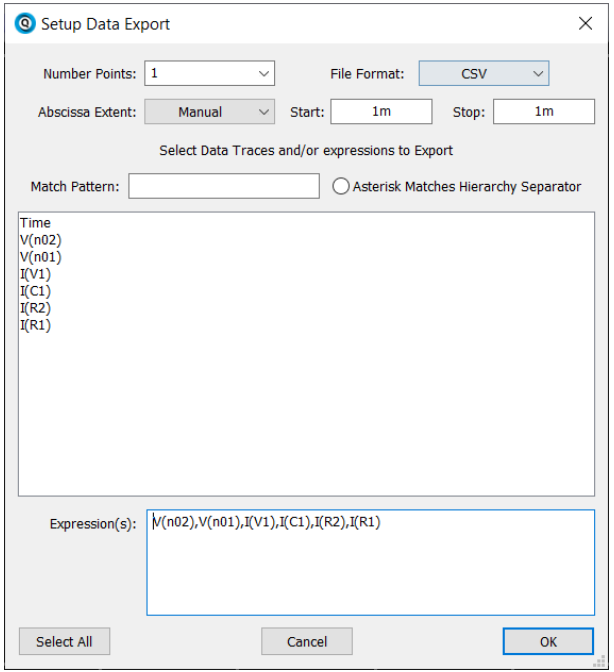
Qspice : waveform - time snapshot.qsch

- Snapshot Data
  - To create a snapshot dataset (e.g. all calculated results at particular time)
  - This example demonstrate a snapshot data in csv format with export data method
  - Idea is to force number points in data export to 1
    - Output two row but both are identical if start and stop are same
    - If start and stop are not same, output two row with time=start and time=stop



In waveform viewer

1. File > Export Data
2. Change Number Points to 1
3. File Format : CSV
4. Abscissa Extent : 1, Start = Stop
5. Select All



Result in exported csv

Time	V(n02)	V(n01)	I(V1)	I(C1)	I(R2)	I(R1)
0.001	1	0.606398	-0.00039	0.000333	6.06E-05	0.000394
0.001	1	0.606398	-0.00039	0.000333	6.06E-05	0.000394

Data export in .ac analysis – Complex format

Qspice : waveform - complex format (.ac).qsch

- Data export in .ac analysis
  - Data export in waveform viewer from .ac analysis is complex format
    - In .qraw data file, Flags : complex
  - Complex format data in ascii (.csv) is as R,X
    - where R is real and X is imaginary
  - If assume data is complex voltage $V_{complex} = V_r + jV_x$
    - $|V_{complex}| = \sqrt{V_r^2 + V_x^2} = \text{abs}(V_{complex})$
    - Magnitude In dB : $V_{complex,dB} = 20\log_{10}|V_{complex}| = 20 * \text{log10}\left(\text{abs}(V_{complex})\right)$
    - $\angle V_{complex} = \tan^{-1}\frac{V_x}{V_r} = \text{atan2d}\left(\text{imag}(V_{complex}), \text{real}(V_{complex})\right) = \text{angle}(V_{complex}) * 180/\text{pi}$

Editor - C:\KSKelvinQspice\01 User Guide and Script\02

waveform_complex_example.m

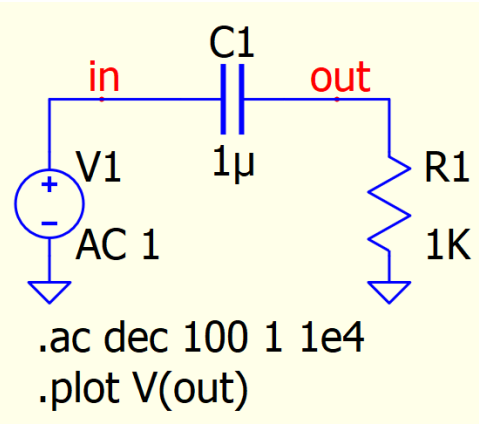
```
1  vout = 0.28304+1i*0.45048
2  vout_db = 20*log10(abs(vout))
3  vout_ph = angle(vout)*180/pi
```

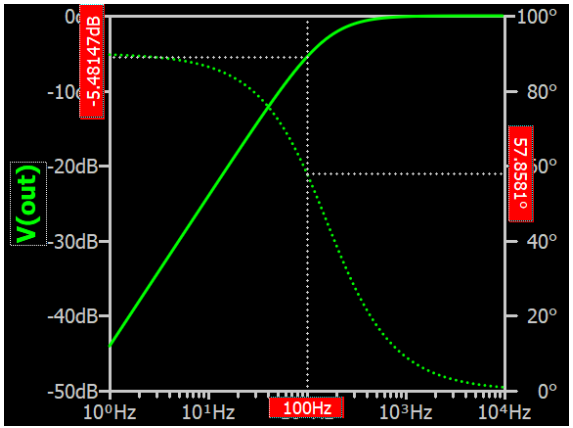
Command Window

```
>> waveform_complex_example

vout =

   0.2830 + 0.4505i


vout_db =

   -5.4815

vout_ph =

   57.8585
```

dB @ 100Hz

Phase @ 100Hz





waveform - complex format (.ac).csv

```
1  Frequency,V(out)
2  1   3.94768591214272e-05,0.00628293726675837
3  1.02329299228075    4.13372692659287e-05,0.00642927371443294
4  :
5  :
6  97.7237220955805    0.273792005193308,0.445903513201606
7  99.9999999999995    0.2830431996751,0.450477243368387
8  102.329299228075    0.292481088087185,0.45490207869224
9  104.712854805089    0.302101072770688,0.459168830171961
```

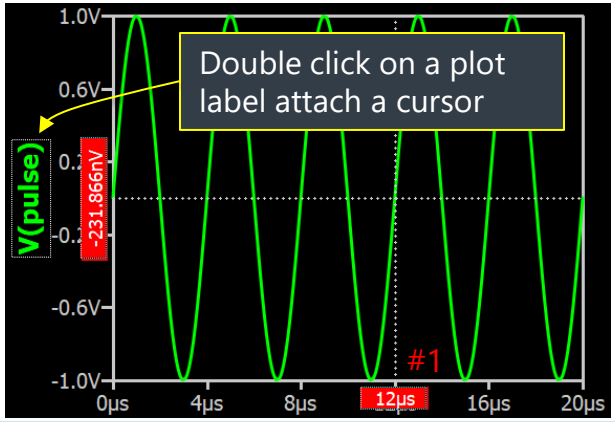
@100Hz, V(out) = 0.28304 + j*0.45048

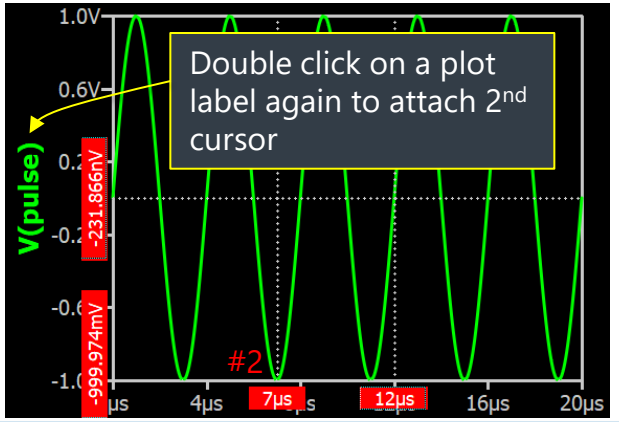
Cursors – Attach a cursor and Cursor Delta

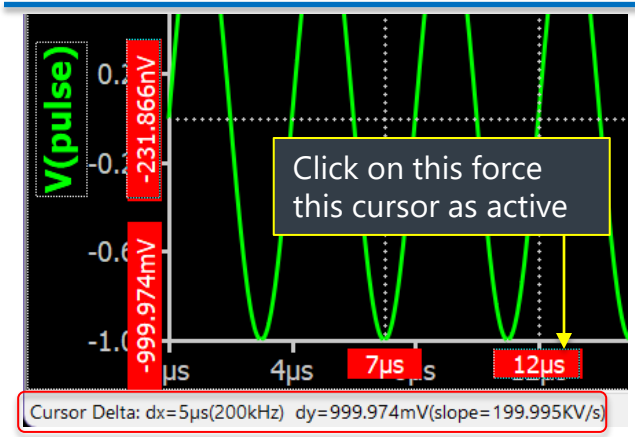
Qspice : waveform - cursors (.tran).qsch

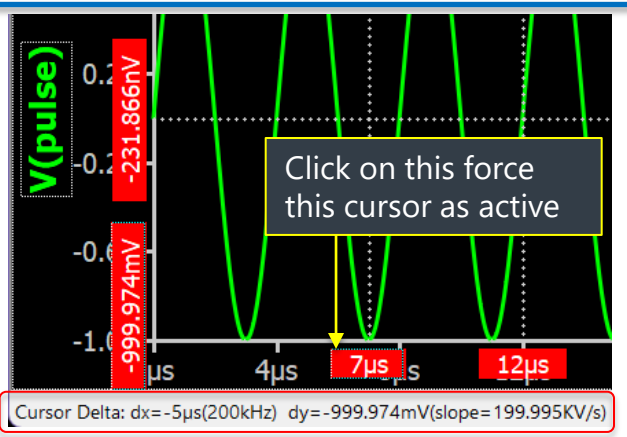
- Attaching a cursor
  - To attach a cursor, double-click on a plot label
  - A maximum of 2 cursors can be attached

- Cursor Delta
  - When 2 cursors are attached, move the mouse to a blank area, and the status bar will display the cursor delta
  - Cursor delta represents the difference in x and y coordinates between the active and inactive cursor
    - The active cursor is the last cursor that was clicked





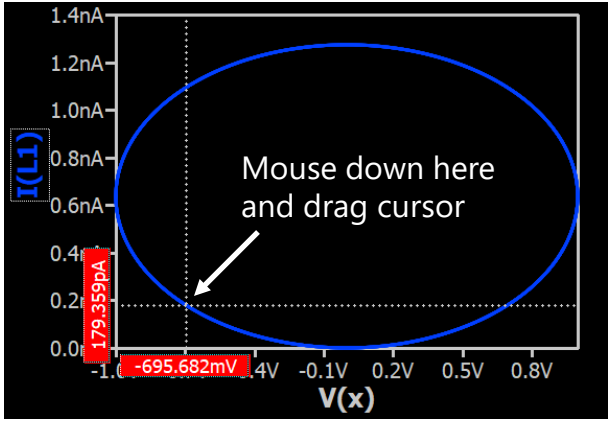




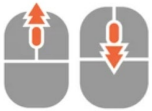
Cursors – Parametric Plot and Cursor movement with mouse wheel

Qspice : waveform – cursors in parametric plot

- Cursor in parametric
  - For parametric plot, the cursor drag operation is different
  - Mouse down where the cursor meets the data and drag along the data



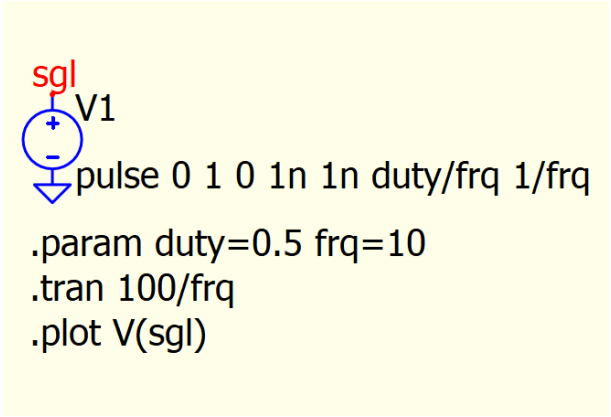
- Cursor movement
  - Mouse wheel moves attached cursor from datapoint to datapoint
  - Hold SHIFT / CTRL for faster movement

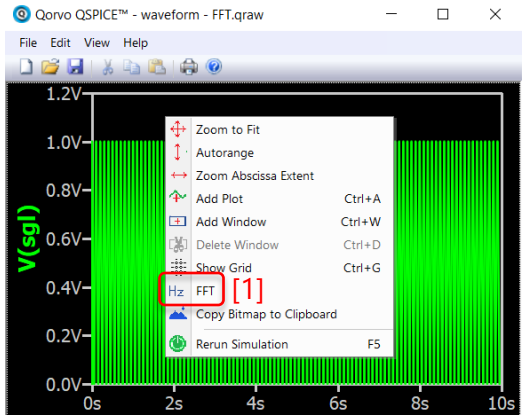


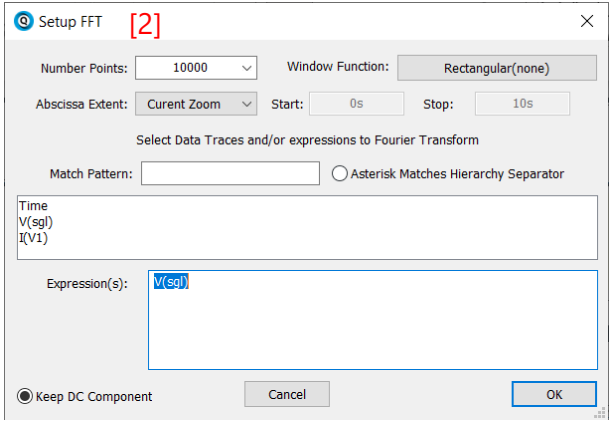
Waveform Viewer – FFT (Fast Fourier Transform)

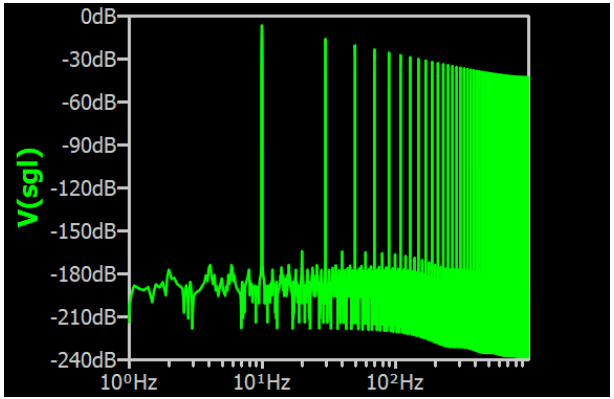
Qspice : waveform - FFT.qsch

- FFT
  - Fast Fourier Transform (FFT) is a right-click option in waveform viewer
  - A Setup FFT window will pop up to select which expression for FFT, number of points (affect post-processing time), window function etc…









Rescale to 1Hz-1000Hz

Waveform Viewer – Run a .meas script

Qspice : waveform - meas script.qsch | waveform - meas script.txt

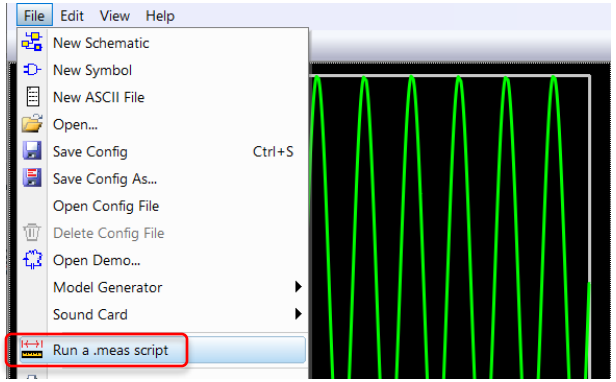
- Run a .meas script
  - File > Run a .meas script
    - load a .meas script from a text file to post-processing (QPOST.exe) the waveform
  - Preparation of .meas script
    - First line needs to be comment (add a * at beginning) as this line will be ignored

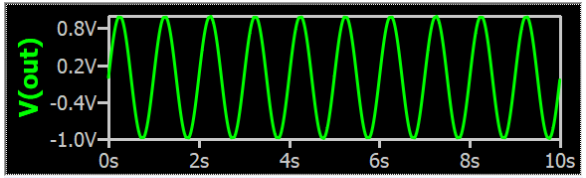
.meas script in text file

** first line needs to be comment

waveform - 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)
```





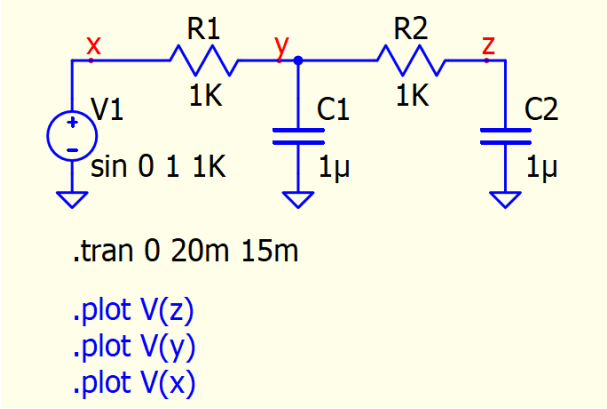
Measurement Window

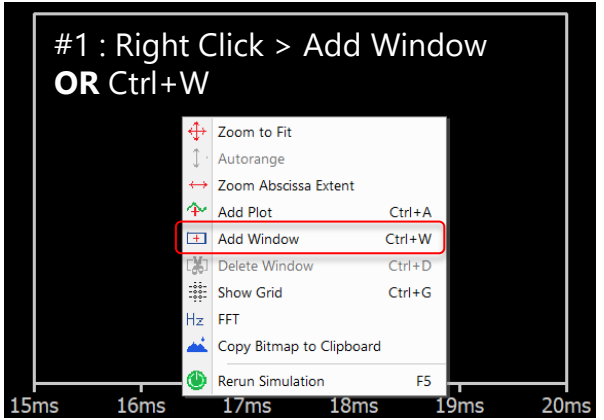
```
.meas vavg avg v(out):
     8.52067e-09
.meas vrms rms v(out):
      0.707106
.meas vmax max v(out):
      0.999994 (at Time=2.24944)
.meas vmin min v(out):
     -0.999994 (at Time=4.74944)
.meas vpp pp v(out):
      1.99999
```

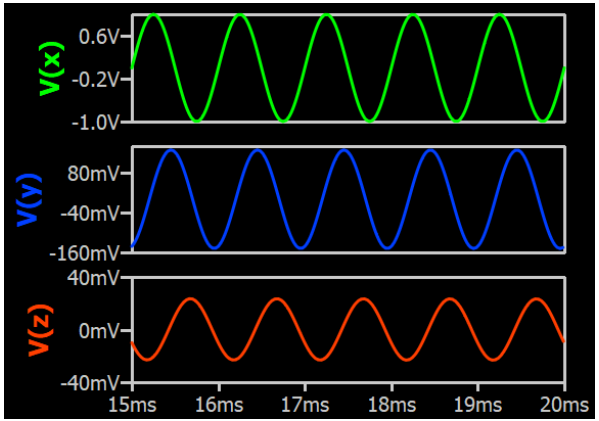
Waveform Viewer – Add Multiple Windows

Qspice : waveform - multiple windows.qsch

- Add Window
  - To plot multiple windows in a single waveform viewer, follow these steps in the Waveform Viewer
    - [1] Right Click > Add Window (or Ctrl+W)
    - [2] Click on the corresponding window and probe or add a plot
  - This can be automatically done with multiple .plot commands



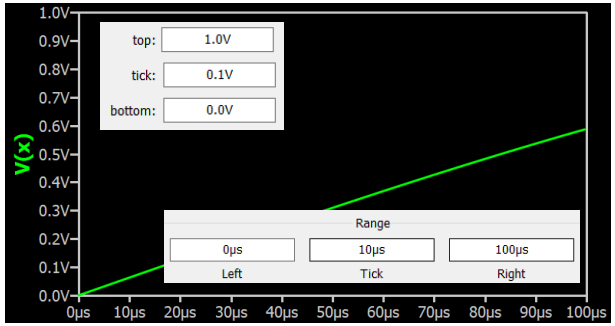


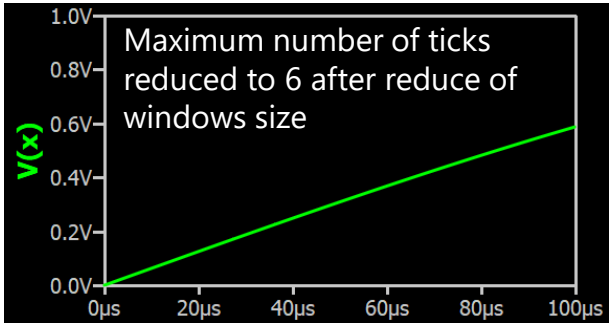


#2 : Click on corresponding windows and probe or add plot

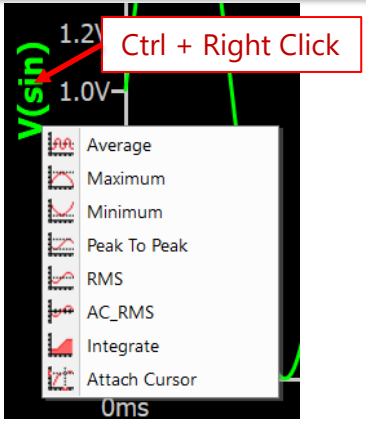
Waveform Viewer – Minimum Tick and Waveform Measure

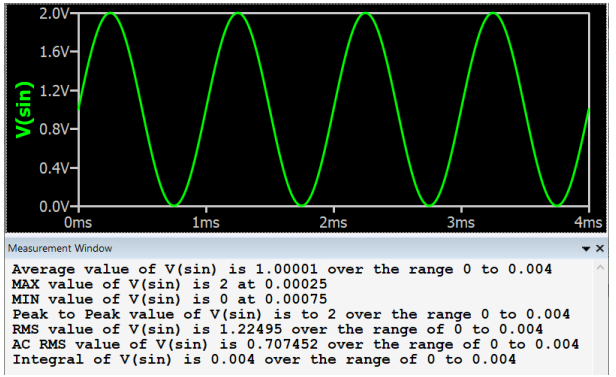
- Minimum Tick
  - Maximum number of ticks in x- and y-axis are 11
  - Depends on windows size, maximum number of ticks can reduce to 6
  - Therefore, minimum allowable tick is $\frac{\text{Right}-\text{Left}}{10}$ or $\frac{\text{top}-\text{bottom}}{10}$





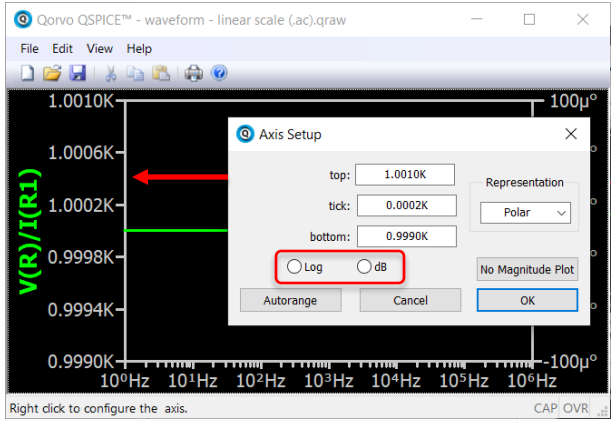
- Waveform Measure
  - Ctrl + Right Click on plot label
  - 7-types of measurement
    - Average
    - Maximum
    - Minimum
    - Peak-to-Peak
    - RMS
    - AC_RMS : remove DC
    - Integrate



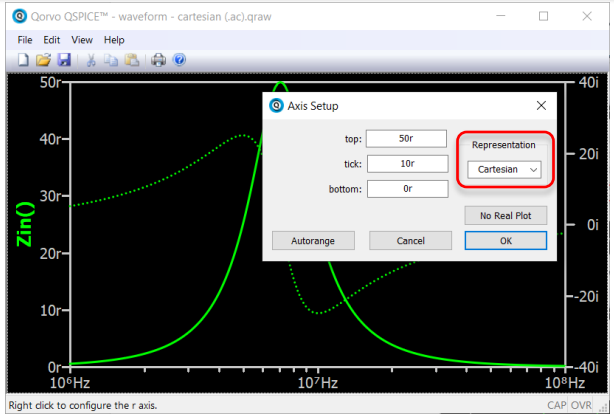


Waveform Viewer – Linear Scale in .AC, Polar vs Cartesian

- Linear Scale in .ac
  - Right Click on Y-axis
  - In Axis Setup, de-select both Log and dB
  - In Polar representation, left y-axis is magnitude which represent value is always absolute (no negative value)



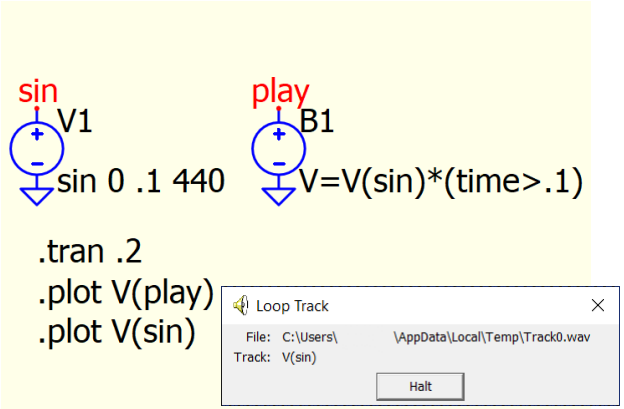
- Polar vs Cartesian
  - Waveform viewer default representation of .ac is Polar representation (mag,phase)
  - Right Click on Y-axis to change to Cartesian representation (real,imag)
  - Polar is normally for bode plot and cartesian is for impedance plot

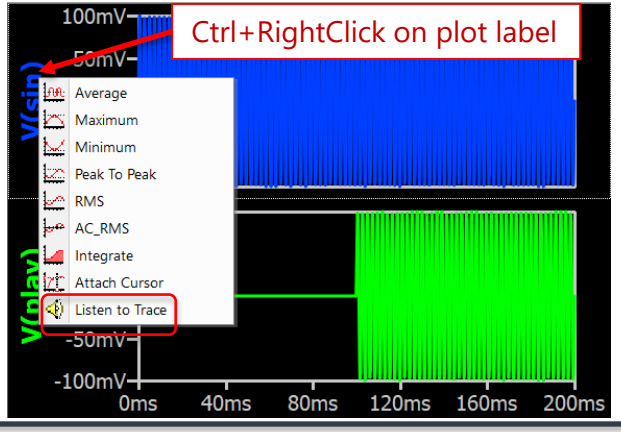


Waveform Viewer – Listen to Trace | Phase Unraveling (Unwrapping)

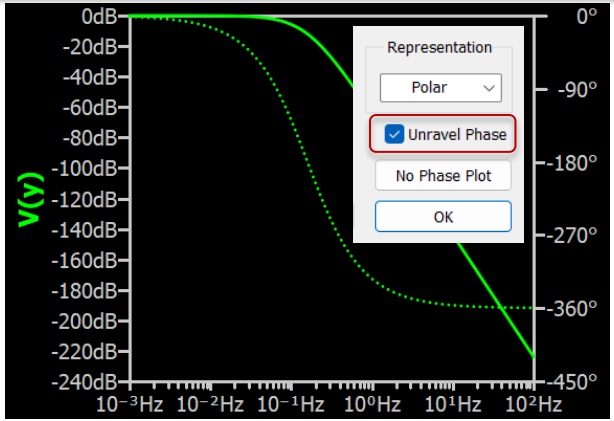
Qspice : waveform - Listen to Trace.qsch | waveform - unraveling.qsch

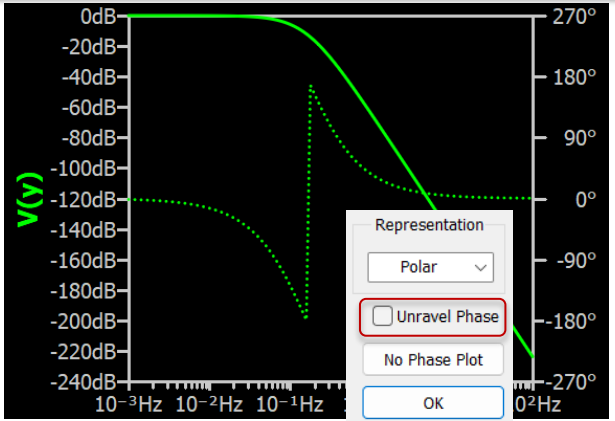
- Listen to Trace
  - Send a time domain data trace to the sound card
    - This option is only available for data >= 1ms
  - In waveform window, **Ctrl** + **Right Click** on the plot label, then select **Listen to Trace**
  - The playback keeps looping





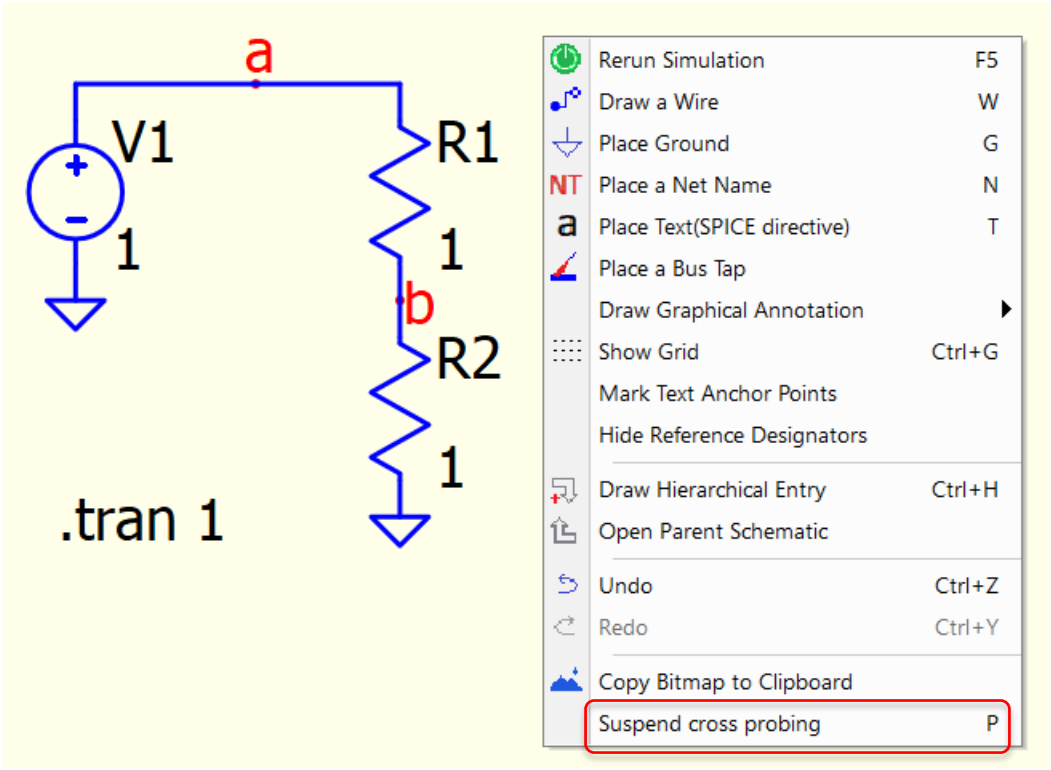
- Phase unraveling
  - Also called Phase wrapping (±180°) and unwrapping
  - Only for complex data plot (e.g. .ac analysis), right click phase axis, enable or disable unravel phase





Waveform Viewer – Suspend Cross Probing (Shortcut : P)

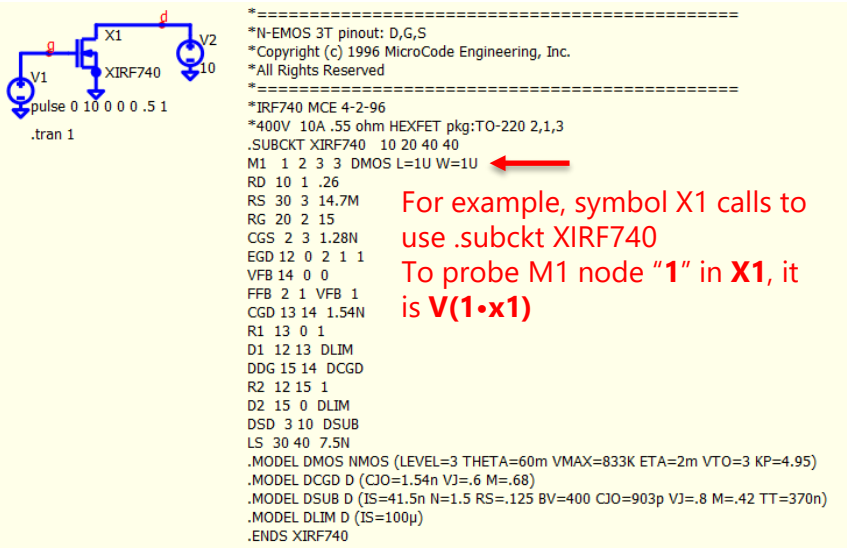
- Suspend cross probing
  - After a simulation run, a new trace can be inserted into the waveform viewer by clicking on a node or device, but this action may unintentionally disrupt the waveform configuration
  - To edit a schematic without altering the waveform setup, right-click in the schematic window and select "Suspend Cross-Probing"
    - This option will only appear in an active waveform window that has cross-probing to the schematic after a simulation run

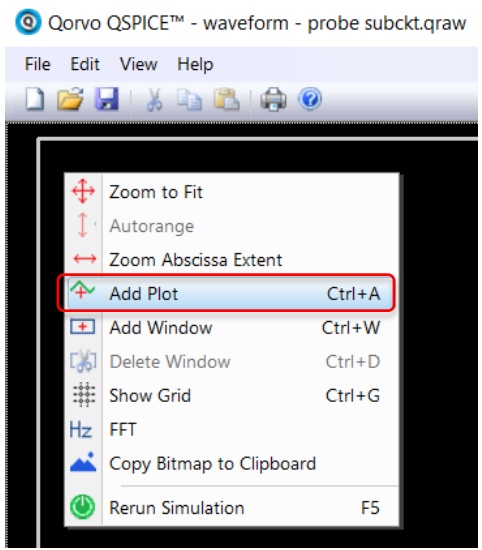


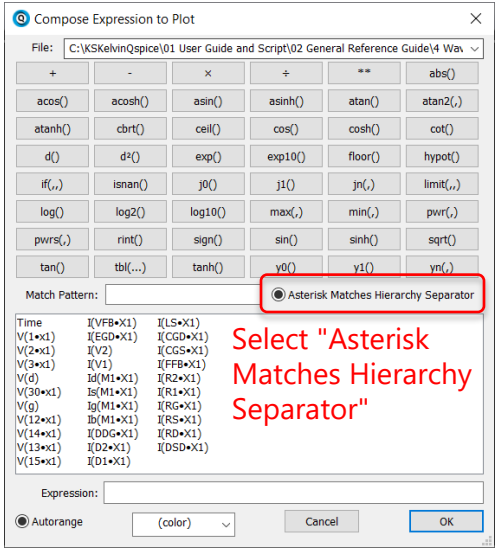
Probe .subckt or hierarchy voltage and current

Qspice : waveform - probe subckt.qsch

- Probe .subckt or hierarchy voltage and current
  - In default, Qspice only returns parent schematic voltage and current expression
  - However, if you open a .qraw data file, you can observe that the voltage and current expressions for .subckt or hierarchical components are stored, such as in the format V(1•x1)
  - In the waveform viewer, **right-click > add plot > "Asterisk Matches Hierarchy Separator"** to enable probing into subckt or hierarchical components



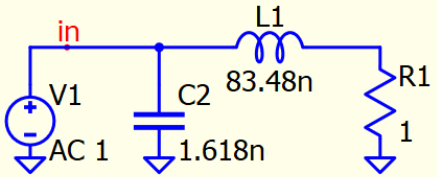




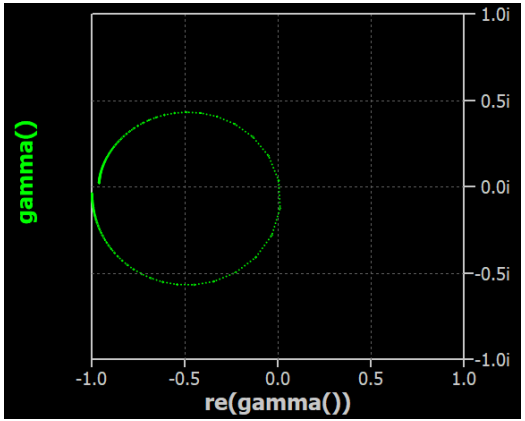
Waveform Viewer – Plot Reflection Coefficient (Γ)

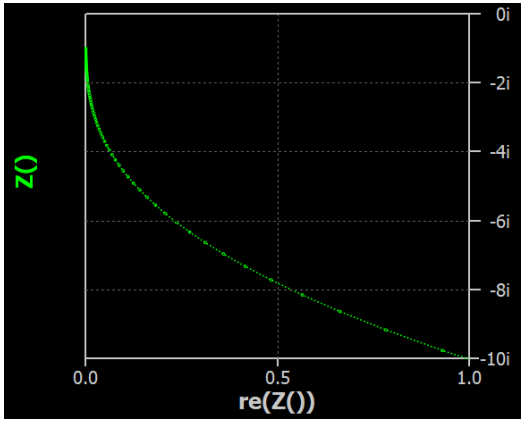
Qspice : waveform - cartesian (Gamma).qsch

- Impedance Trajectory
  - .ac analysis can be used to calculate impedance Z or reflection coefficient $\Gamma = \frac{Z-Z_o}{Z+Zo}$
  - Trajectory can be plot with Cartesian plot by setting x-axis to plot real part and y-axis to plot imaginary part
    - Right-axis : Cartesian
    - Left-axis : No Real Plot
    - X-axis : Quantity changed to Re(<var>)




```
.ac dec 100 1Meg 100Meg

; Calculate Reflection Coefficient
.func Z() V(in)/-I(V1)
.func gamma() (Z()-50)/(Z()+50)
.plot gamma()
```



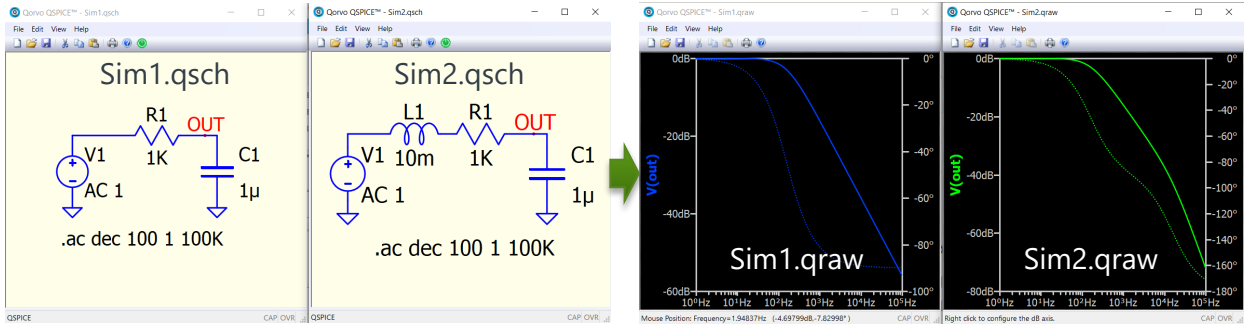


Waveform Viewer – Overlay (method 1)

Qspice : (Folder : overlay) Sim1.qsch | Sim2.qsch

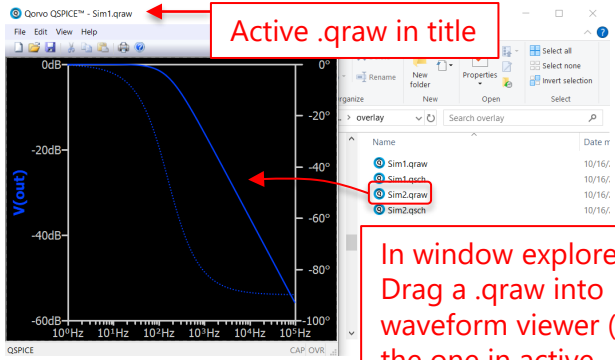
- Overlay – method 1
  - In waveform viewer, multiple .qraw data files can be drag into waveform viewer for overlay comparison
  - In an overlay condition
    - Ctrl-L : toggle plotting overlay data, it can toggle .qraw data which drag into active waveform viewer
    - Ctrl-O : manage overlay datasets

Step 1 : Run simulation (e.g. two different .qsch) to generate multiple .qraw data files

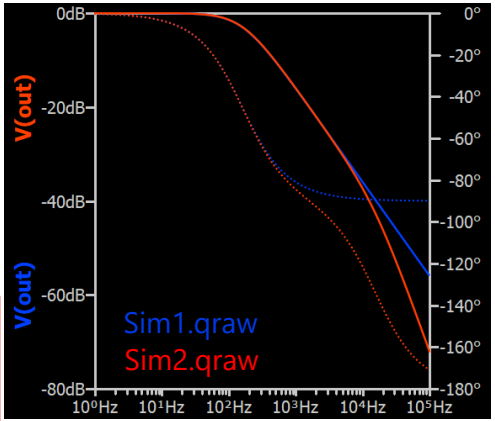


Step 2

- Keep one active waveform viewer
- Drag other .qraw into this waveform viewer



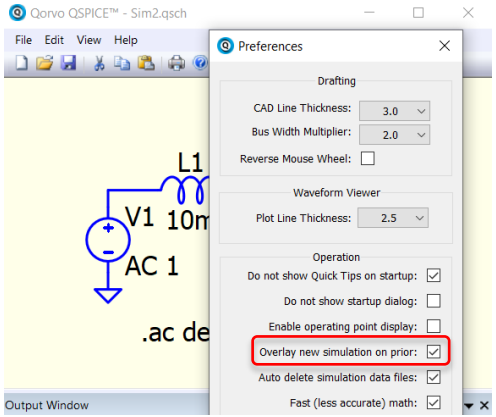
In window explorer Drag a .qraw into waveform viewer (not the one in active waveform viewer)

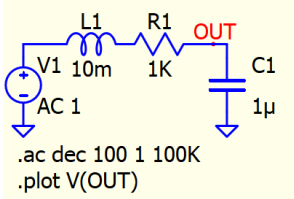


Waveform Viewer – Overlay (method 2)

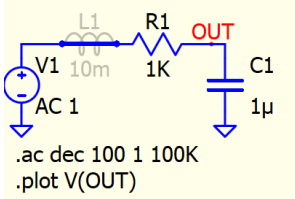
Qspice : (Folder : overlay) Sim2.qsch

- Overlay – method 2
  - In Preferences, enable “Overlay new simulation on prior”
  - Now, each time you run simulation, it will generate a new .qraw with [fname].n.qraw, where n = 0,1,2,3,…



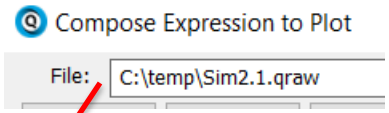


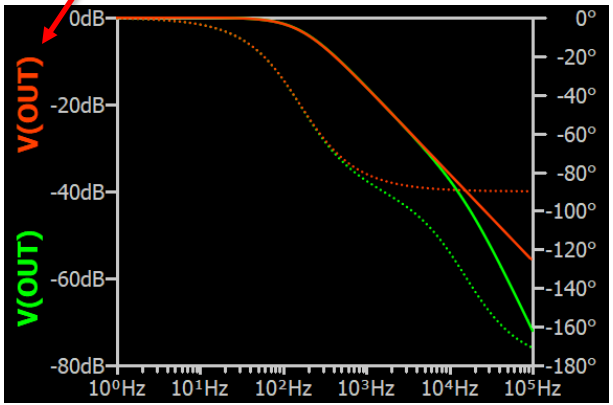
Run simulation to generate xxx.0.qraw



Change something, Run simulation to generate xxx.1.qraw

** Right click can view its .qraw name

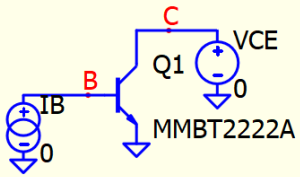




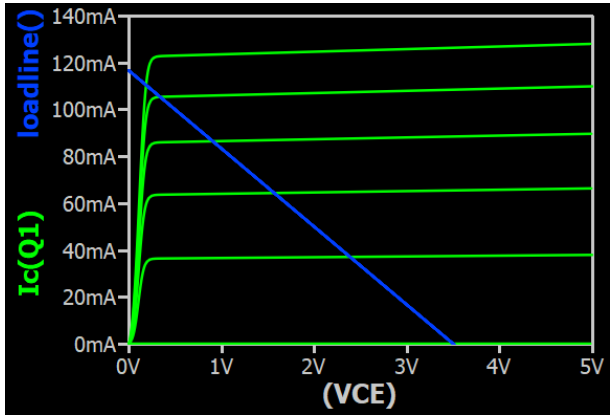
Waveform Viewer – Draw a Line

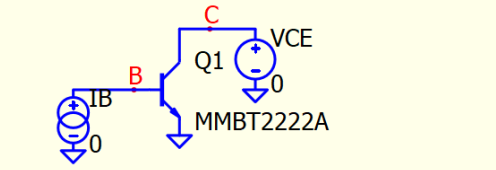
Qspice : waveform - draw loadline.qsch | waveform - draw line with table.qsch

- Draw a Line
  - .func can be used to plot a line in the waveform viewer where this line follows a mathematical formula or table relationship

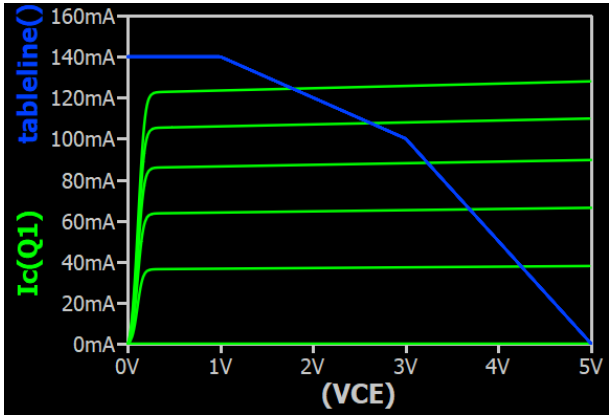



```
.dc  VCE 0 5 0.01 IB 0 1m 1m/5
.param VCC=3.5
.param RL=30
.func loadline() (-1/RL*V(C)+VCC/RL)/1V*1A
.plot Ic(Q1) loadline()
```





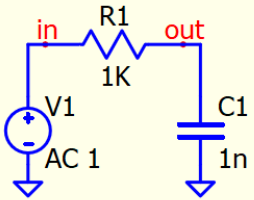

```
.dc  VCE 0 5 0.01 IB 0 1m 1m/5
.plot Ic(Q1) tableline()
```



Waveform Viewer – Draw a Line

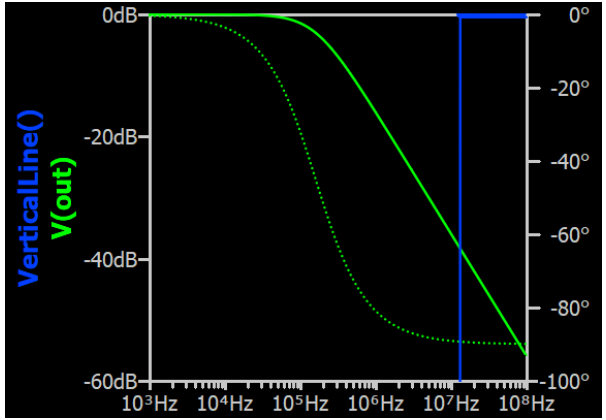
Qspice : waveform - draw vertical line.qsch

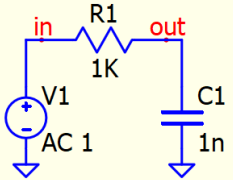
- Draw a Line
  - Example to draw vertical line with if(x,y,z) function



This formula only for a vertical line in Mag plot

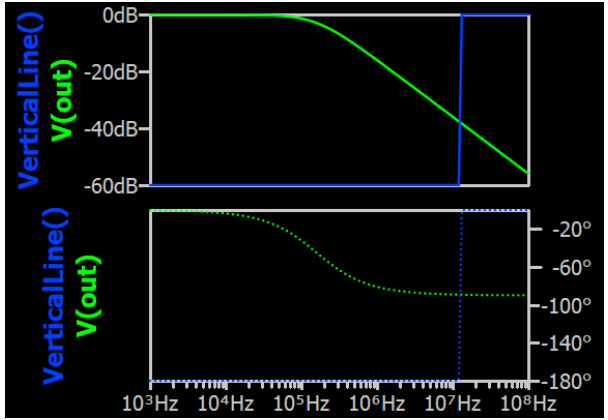
```
.ac dec 100 1K 100Meg
.plot V(out) VerticalLine()
.func VerticalLine() if(frequency>13Meg,1,NaN)
```





This formula can have a vertical line in Mag and Phase plot

```
.ac dec 100 1K 100Meg
.plot V(out) VerticalLine()
.func VerticalLine() if(frequency>13Meg,1,NaN)
.func VerticalLine() if(frequency>13Meg,1,-0.001j)
```



Part 4 – Appendix
Waveform Viewer Functions

Waveform Viewer Functions and Keywords

The following functions, constants, and keywords are recognized in expressions of waveform data.

Waveform Viewer Functions and Keywords

Syntax	Description
ABS(x)	Absolute value of x
ACOS(x)	Inverse cosine of x
ACOSH(x)	Inverse hyperbolic cosine of x
ARCCOS(x)	Inverse cosine of x
ARCCOSH(x)	Inverse hyperbolic cosine of x
ARCSIN(x)	Inverse sine of x
ARCSINH(x)	Inverse hyperbolic sine of x
ARCTAN(x)	Inverse tangent of x
ARCTANH(x)	Inverse hyperbolic tangent of x
ASIN(x)	Inverse sine of x
ASINH(x)	Inverse hyperbolic sine of x
ATAN(x)	Inverse tangent of x
ATAN2(x,y)[1]	Four quadrant inverse tangent of x
ATANH(x)	Inverse hyperbolic tangent of x
BUF(x)	x > .5 ? 1 : 0
CBRT(x)	Cube root of x
CEIL(x)	x rounded up to nearest integer
COS(x)	Cosine of x
COSH(x)	Hyperbolic cosine of x
COT(x)	Cotangent of x
D(x)	Derivative of x
DD(x)	Second derivative of x
D²(x)	Second derivative of x
E	2.7182818284590452354
ERF(x)	Error function of x
ERFC(x)	Complementary error function of x
EXP(x)	e raised to the x power
EXP10(x)	10 raised to the x power
FABS(x)	Absolute value of x
FREQ	Frequency
FREQUENCY	Frequency
GAMMA(x)	Gamma function of x
HYPOT(x,y)	$\sqrt{(x^2+y^2)}$
IF(x,y,z)	(x > .5) ? y : z
ILOGB(x)	Unbiased exponent of x
IM(x)	Imaginary part of x
IMAG(x)	Imaginary part of x
INT(x)	x rounded to nearest integer closest to zero
INV(x)	x > .5 ? 0 : 1
INVSQRT(x)	$1 \div \sqrt{x}$
ISNAN(x)	One if x is not a number, otherwise zero
J	$\sqrt{-1}$
J0(x)	Zero order Bessel function of the first kind at x
J1(x)	First order Bessel function of the first kind at x
JN(x,n)	N^{th} order Bessel function of the first kind at x
K	1.380649e-23 J/°K
LGAMMA(x)	Log-gamma function of x
LIMIT(x,y,z)	Mutually intermediate value of x,y, and z
LN(x)	Natural logarithm of x
LOG(x)	Natural logarithm of x
LOG10(x)	Logarithm of x in base 10
LOG1P(x)	Natural logarithm of (x + 1)
LOG2(x)	Logarithm of x in base 2
LOGB(x)	LOG2(ABS(x))
MAG(x)	Absolute value of x
MAX(x,y)	Maximum of x and y
MAXMAG(x,y)	x or y with maximum magnitude
NAN	A value guaranteed to be not a number
MIN(x,y)	Minimum of x and y
MINMAG(x,y)	x or y with minimum magnitude
PH(x)	Phase of x
PHASE(x)	Phase of x
PI	3.14159265358979323846
POW(x,y)	x raised to the y power
POWN(x,y)	x raised to the nearest integer value of y
PWR(x,y)	Absolute value of x raised to the y power
PWRS(x,y)	$x >= 0 ? x^y : -x^y$
Q	1.602176487e-19 Coulomb
RE(x)	Real part of x
REAL(x)	Real part of x
RINT(x)	x rounded to the nearest integer
ROUND(x)	x rounded to the nearest integer
SGN(x)	Sign of x
SIGN(x)	Sign of x
SIN(x)	Sine of x
SINH(x)	Hyperbolic sine of x
SQRT(x)	Square root of x
TABLE(x,x1,y1,...)	Interpolate the table given as x1,y1, x2,y2,... at point x
TAN(x)	Tangent of x
TANH(x)	Hyperbolic tangent of x
TAUGRP(x)	Group delay of x
TBL(x,x1,y1,...)	Interpolate the table given as x1,y1, x2,y2,... at point x
TEMP	Circuit temperature
TG(x)	Group delay of x
TIME	Time
TRUNC(x)	Integer part of x
URAMP(x)	x > 0 ? x : 0
USTEP(x)	x > 0 ? 1 : 0
Y0(x)	Zero order Bessel function of the second kind at x
Y1(x)	First order Bessel function of the second kind at x
YN(x)	N^{th} order Bessel function of the second kind at x

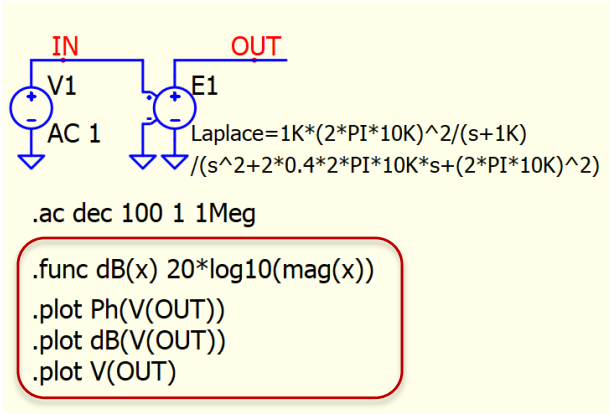
[1] For complex data, the syntax is ATAN2(z). The meaning is ATAN2(IMAG(z),REAL(z)).

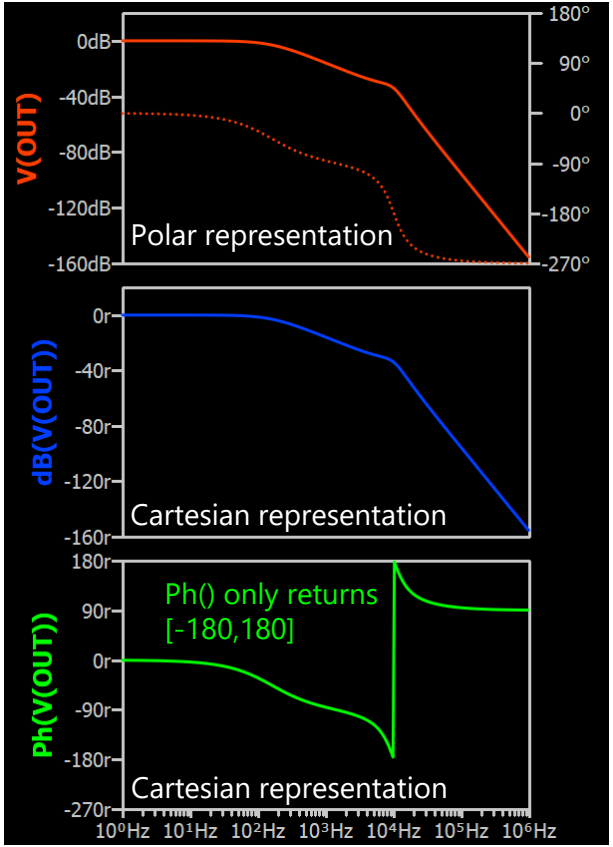
Waveform Viewer Functions : dB() and Phase() | Ph()

Qspice : waveform func - dB and Ph (.ac).qsch

- dB in .ac
  - dB() function only available in .meas but not in waveform viewer function
  - dB() in waveform viewer requires user-defined function with $\mathrm{dB} = 20\log_{10}|v|$

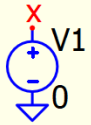
- Phase in .ac
  - Phase() or Ph() function returns angle between -180 to +180 degree
    - In waveform viewer polar representation, phase is unwrap
    - This is important in using .meas as search over this range is unavailable



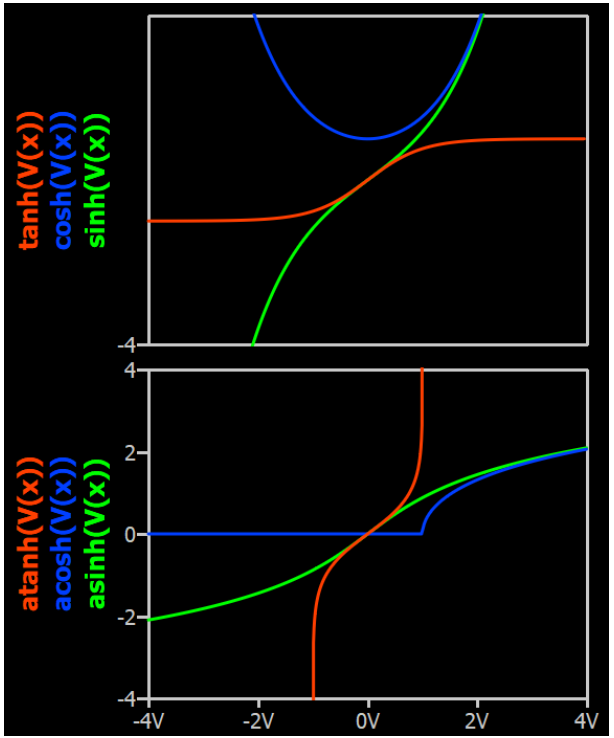


Waveform Viewer Functions : Hyperbolic Sin, Cos and Tan

- Hyperbolic Sin Cos Tan
  - $\sinh x = \frac{e^x - e^{-x}}{2}$
  - $\cosh x = \frac{e^x + e^{-x}}{2}$
  - $\tanh x = \frac{\sinh x}{\cosh x} = \frac{e^x - e^{-x}}{e^x + e^{-x}}$
- Inverse functions
  - $\sinh^{-1} x = \ln\left(x + \sqrt{x^2 + 1}\right)$
  - $\cosh^{-1} x = \ln\left(x + \sqrt{x^2 - 1}\right)$
  - $\tanh^{-1} x = \frac{1}{2} \ln \frac{1+x}{1-x}$



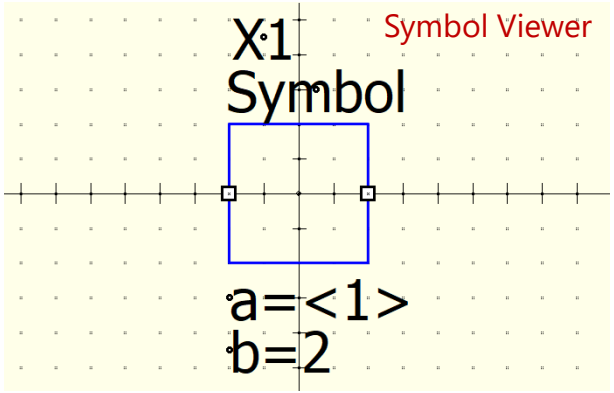

```
.dc V1 -4 4 0.01
.plot asinh(V(x)) acosh(V(x)) atanh(V(x))
.plot sinh(V(x)) cosh(V(x)) tanh(V(x))
```

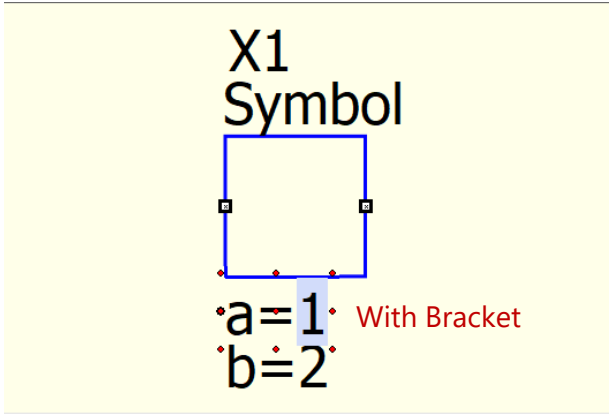


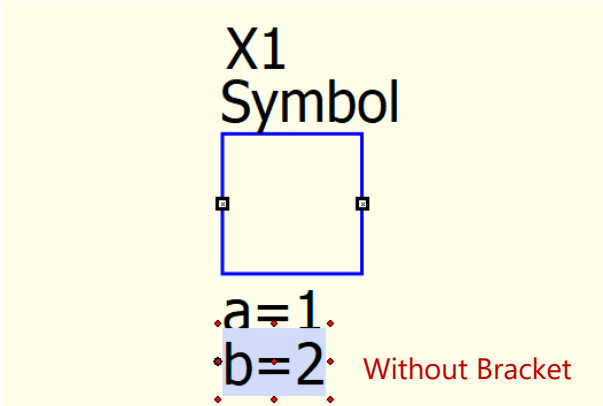
Part 5
Symbol Viewer

Attribute with Angle Brackets < >

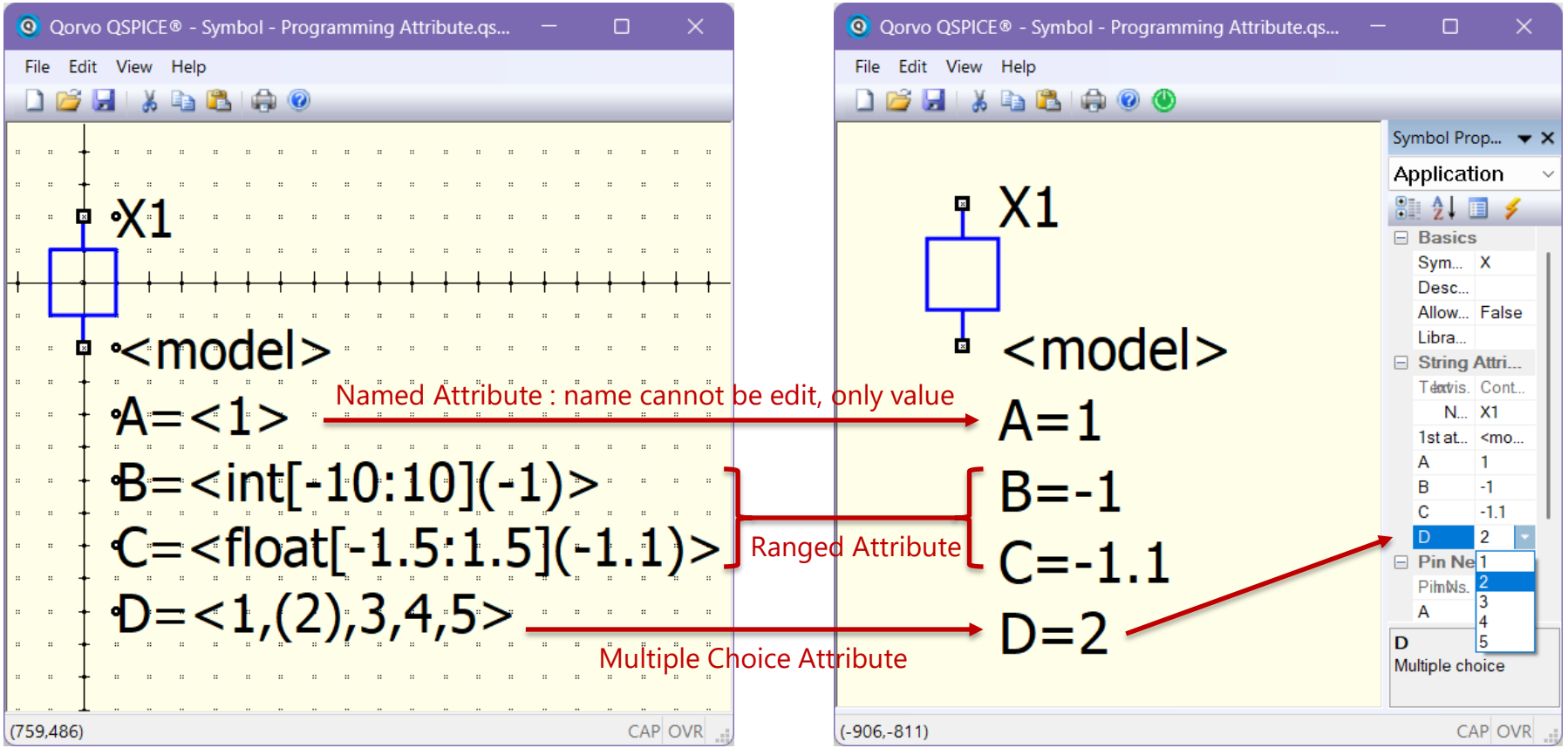
- Attribute with brackets
  - The 1st attribute of a symbol is its model name, and starting from the 2nd attribute are the instance parameters
  - When angle brackets (< >) are used for instance parameters, they can restrict the user-editable section of the text attribute
    - Each attribute only allows for one bracket; any text after the bracket is ignored





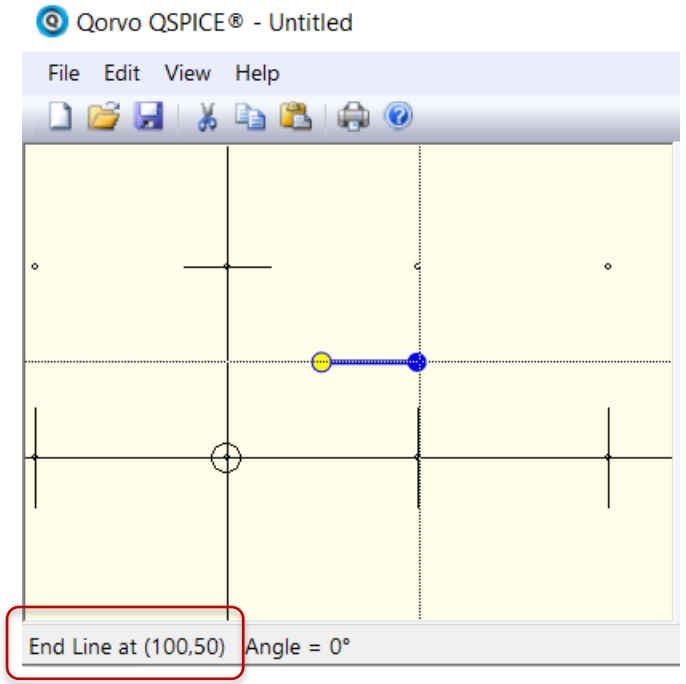


Programming Attributes



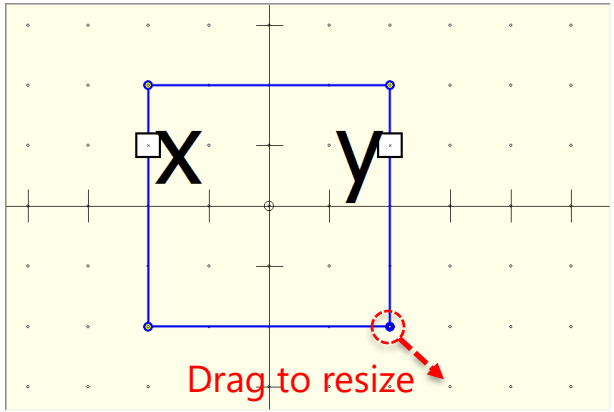
Placement Resolution in Symbol Window

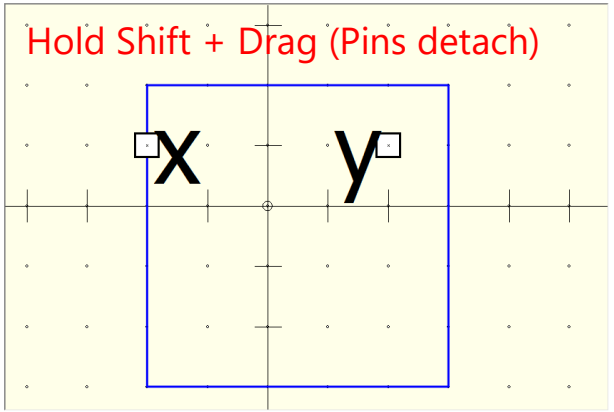
- Placement Resolution
  - In the symbol window, coordinates are displayed in status bar
  - To place an arc, box, line etc., the minimum resolution can be changed by holding the **Ctrl**, **Alt** or **Shift** key as follows
    - No holding key : step by 50
    - Holding one key : step by 10
    - Holding any two keys : step by 5
    - Holding all three keys : step by 1
  - However, Pin always stick to the major grid, which is set at a step of 100

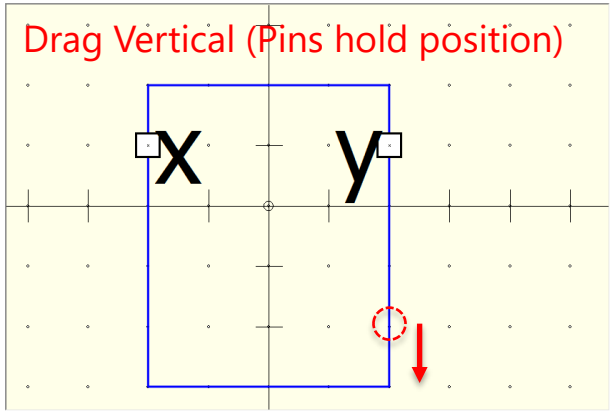


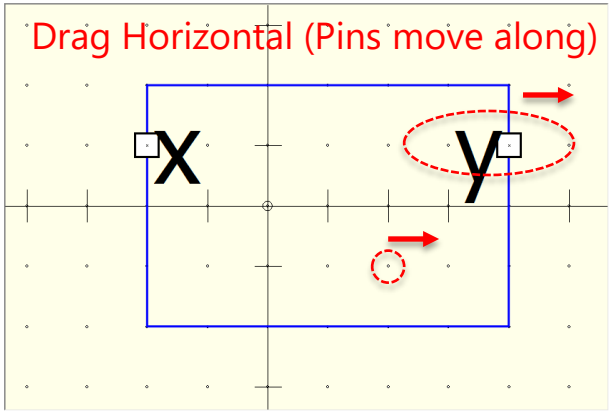
Resize Rectangle symbol with Pins on the edge

- Resize rectangle with pins on the edge
  - Pins move along during horizontal resizing of the rectangle symbol
  - Pins hold position during vertical resizing of the rectangle symbol
  - Holding down the Shift key disables this feature, allowing the pins and symbol to detach
  - This apply to both schematic and symbol window



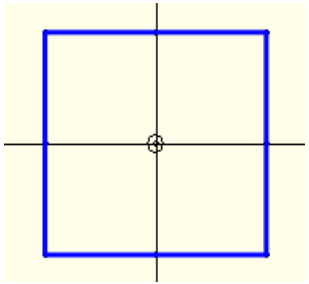






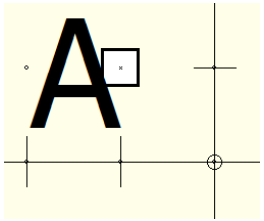
Symbol File (.qsym) Syntax (Edit with Text Editor)

- .qsym (BOX)
  - «rect (-100,100) (100,-100) 0 0 0 0x1000000 0x1000000 -1 0 -1»
  - «rect ([top left]) ([bottom right]) 0 [line width] [line type] [line color] [fill color] -1 [hierarchal block] -1»
- Line width : 0 is default
- Line type : 0 is solid line, 1 to 4 are various dotted line
- Line color
  - 0x1000000 is default color
  - 0xBBGGRR is RGB color where RR, GG and BB are HEX from 00 to FF
- Fill color : 0x1000000 is Not filled
- Hierarchal block : 1 is hierarchal block

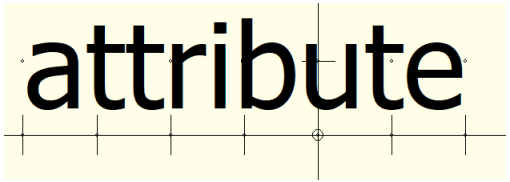


Symbol File (.qsym) Syntax (Edit with Text Editor)

- .qsym (Pin)
  - «pin (-100,100) (0,0) 1 0 0 0x0 -1 "A"»
  - «pin ([center]) ([pin label position]) [font size] [justification+orientation] 0 [color] -1 "[pin label]"»
  - Font size : 1 is default
  - Justification+orientation
    - In normal orientation : 15 is center (or 0), 7 is right, 11 is left, 14 is top, 13 is bottom
  - Color : 0x0 is default color, 0xBBGGRR is RGB color
  - Pin label : ¬ character is invert overbar



- .qsym (Pin) for Ø(.DLL)
  - «pin (-100,100) (0,0) 1 0 0 0x0 -1 "A"»
  - «pin (-100,100) (0,0) 1 0 [Port Type +Data Type] 0x0 -1 "A"»



- .qsym (Attribute | Text)
  - «text (-100,100) 1 15 0 0x0 -1 -1 "attribute"»
  - «text ([center]) [font size] [justification+orientation] 0 [color] -1 -1 "[text]"»

Symbol File (.qsym) Syntax (Edit with Text Editor)

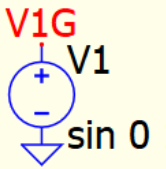
- [Port Type + Data Type] in .qsym (Pin) for Ø(.DLL)
  - Only accept Decimal value
  - In HEX value, it can observe that bit 0-3 defines port type (Input,Output,Dll's GND) and bit 4-7 defines data type

Port Type	Data Type	DEC	HEX
Input		1	00000001
	Boolean	**17**	**00000011**
	Char	33	00000021
	Unsigned Char	49	00000031
	Short	65	00000041
	Unsigned Short	81	00000051
	Integer	**97**	**00000061**
	Unsigned Integer	113	00000071
	Short Float32	129	00000081
	Float	**145**	**00000091**
	Integer64	161	000000A1
	Unsigned Integer 64	177	000000B1
	Bit Vector[0:3]	196801	000300C1
Output		2	00000002
	Boolean	**18**	**00000012**
	Char	34	00000022
	Unsigned Char	50	00000032
	Short	66	00000042
	Unsigned Short	82	00000052
	Integer	**98**	**00000062**
	Unsigned Integer	114	00000072
	Short Float32	130	00000082
	Float	**146**	**00000092**
	Integer64	162	000000A2
	Unsigned Integer 64	178	000000B2
	Bit Vector[0:3]	196802	000300C2
DLL's GND		3	00000003

Part 6
Simulation Technique

Max Time Step in .tran (and .bode) : Two methods

Qspice : MaxTimeStep.qsch



sin 0 1 1G .plot V(V1G)

Method 1 : Traditional Berkeley Syntax

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

.tran 0 10n 0 10p

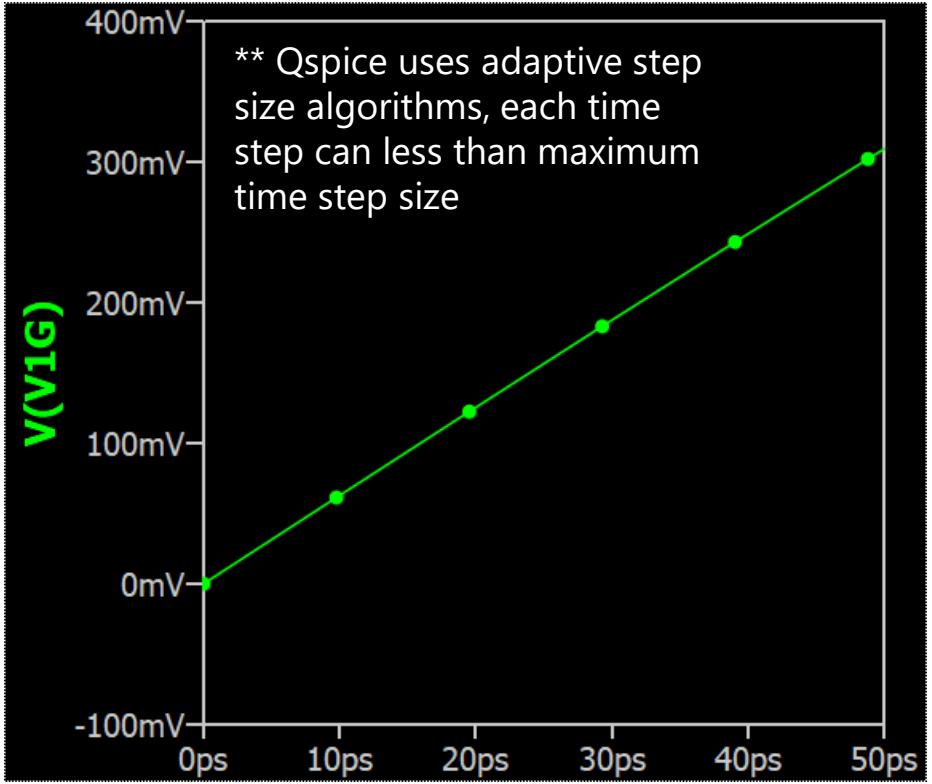
↑ Max Time Step

Method 2 : MAXSTEP in Simulator Option

.tran 10n

.options MAXSTEP=10p

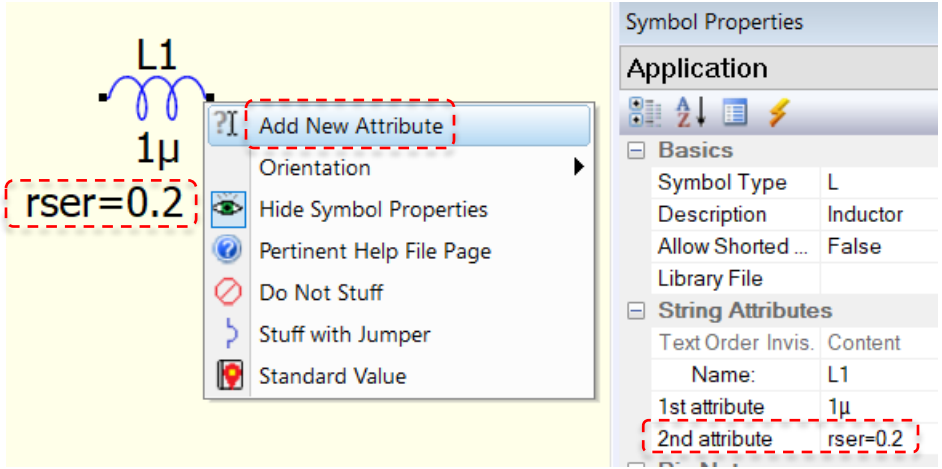
↑ Max Time Step
(this method apply to .tran and .bode)



Add Attribute [e.g. add instance parameters]

1. Right Click on Component
2. Select “Add New Attribute”
3. Type parameter name and value [refer to help for full list of instance parameters]

This is an example to assign 0.2 ohms series resistance to inductor L1



Inductor Instance Parameters

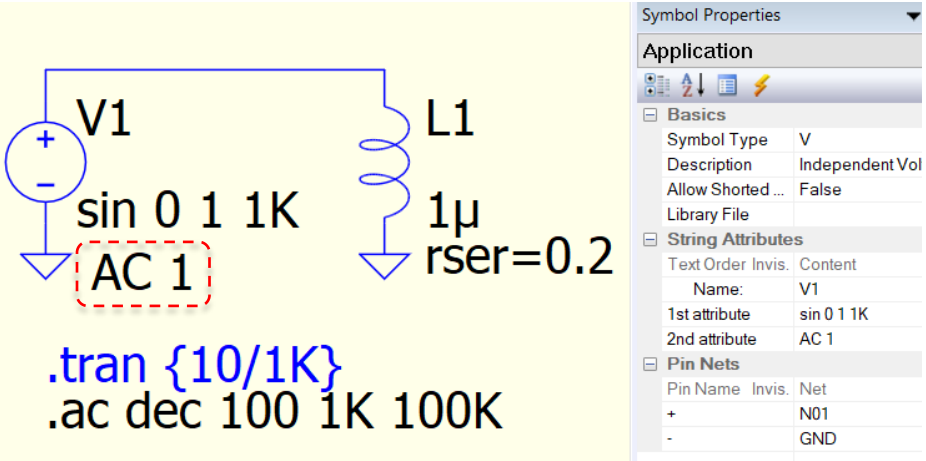
Name	Description	Units	Default
AG	Wire or stripline is made of gold		
AL	Wire or stripline is made of aluminum		see below
AU	Wire or stripline is made of silver		
BEND	Fractional inductance correction for wire bend or proximity effects		1.
CPAR	Parallel capacitance	F	0.
CU	Wire or stripline is made of copper		see below
DIAMETER	Diameter of wire or air coil	m	
FREQUENCY	Frequency at Q. Also used to compute Rser due to skin effect		
HEIGHT	Height of PCB stripline above ground plane	m	
IC	Initial current if uic is specified on .tran statement	A	none
INDUCTANCE	Inductance of inductor	H	0.0
ISAT	Current causing inductance to drop to SATFRAC×INDUCTANCE	A	Infinite
LENGTH	Length of wire, stripline, or air coil	m	
LSAT	Inductance asymptotically approached in saturation	H	10% of INDUCTANCE
M	Number of parallel inductors		1.0
NI	Wire is made of nickel		
Q	Quality factor at FREQUENCY		
RPAR	Equivalent parallel resistance	Ω	Infinite
RSER	Equivalent series resistance	Ω	0.0
SATFRAC	Fractional drop in inductance at ISAT		0.7
THICK	Thickness of stripline on top of a PCB	m	0.0
TURNS	Number of turns of an air coil		
VERBOSE	Print wire L, Rser, Rpar results on the console		(not set)
WIDTH	Width of stripline on top of a PCB	m	

AC and DC Attribute in Source

Qspice : AC with Transient Source.qsch ; AC with Bias.qsch

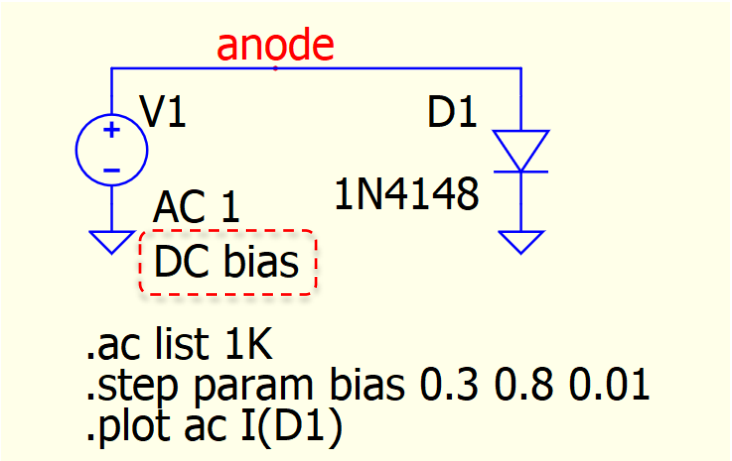
Technique to perform AC analysis with a transient source

1. Right Click on Voltage/Current source
2. Select “Add New Attribute”
3. Type “AC 1” to define a 1V source for AC sweep
4. Add a .ac analysis statement, and comment transient analysis



Technique to perform AC analysis with DC in source

1. Right Click on Voltage/Current Sourec, Add New Attribute
2. To add DC source, type “DC …”
   - If without DC, simulator may not interpret the DC voltage during simulation. Best practice is to add DC



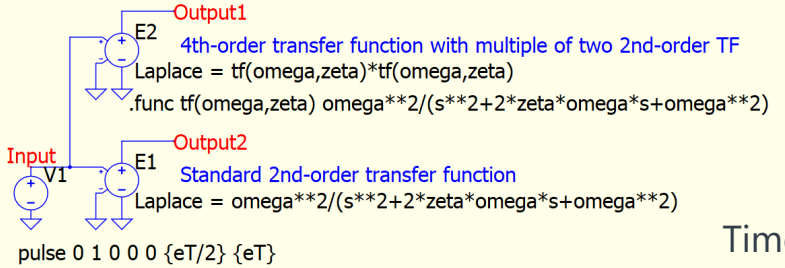
Laplace Time and Frequency Domain Simulation

Qspice : Laplace Simulation - Fdomain.qsch ; Laplace Simulation - Tdomain.qsch

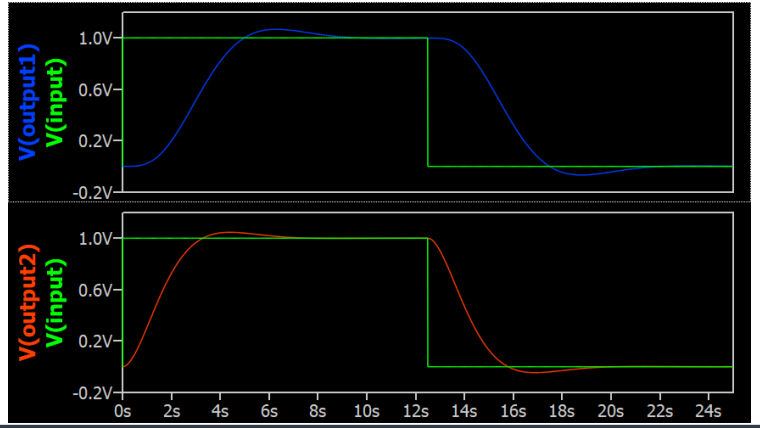
2nd-order system step response
transfer function : omega^2/(s^2+2*zeta*omega+omega^2)

.param zeta = 0.7
.param omega = 1

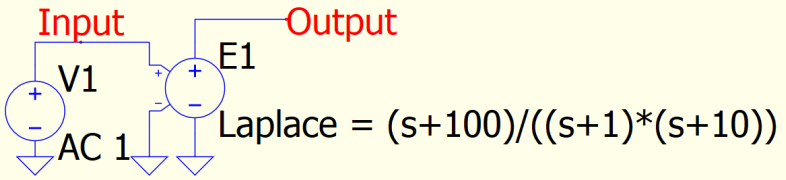
.tran {eT}
.param eT=25



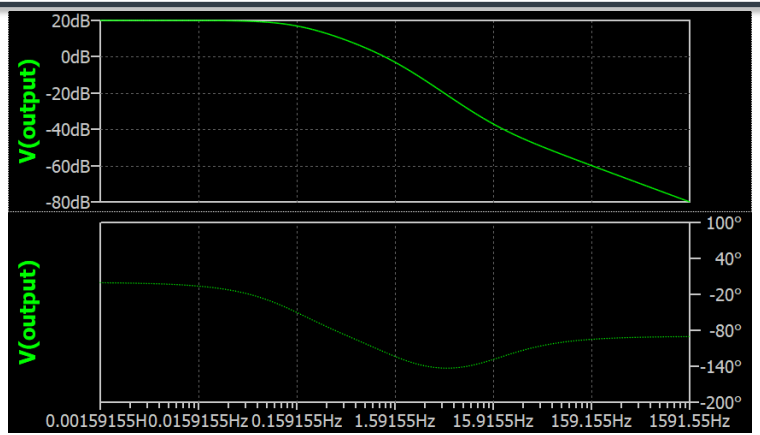
Time Domain



Frequency Domain



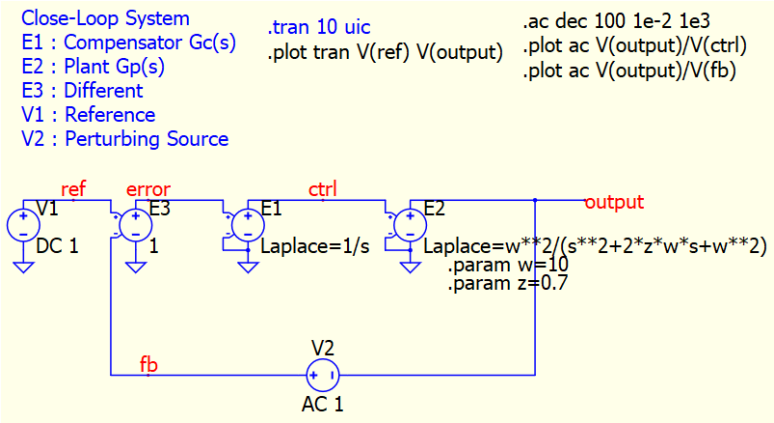
.ac dec 100 {1e-2/2/pi} {1e4/2/pi}

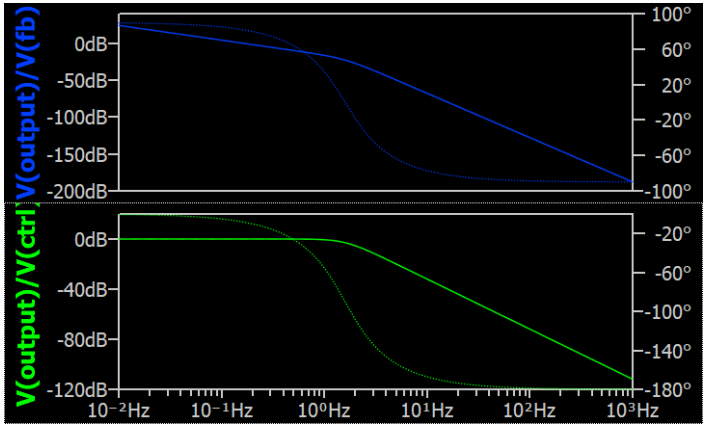


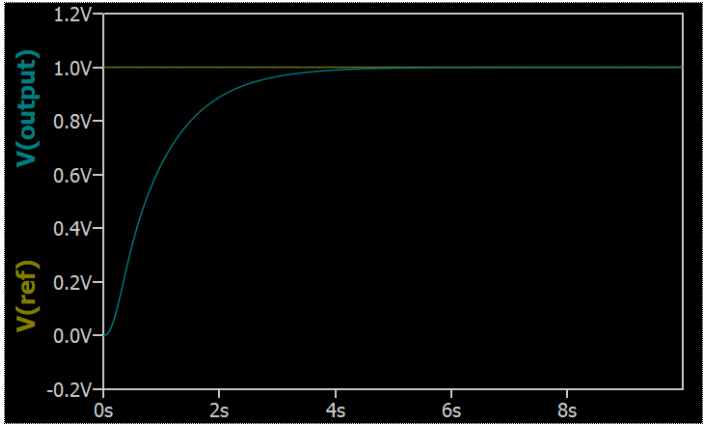
Laplace Time and Frequency Domain Simulation

Qspice : Laplace Close Loop.qsch

- Close Loop System Time and Bode
  - A technique to get Gp(s) and GH(s) is to add a perturbing source between output and feedback and perform ac analysis
  - In this example, Laplace function can collect in series for both .tran and .ac directive

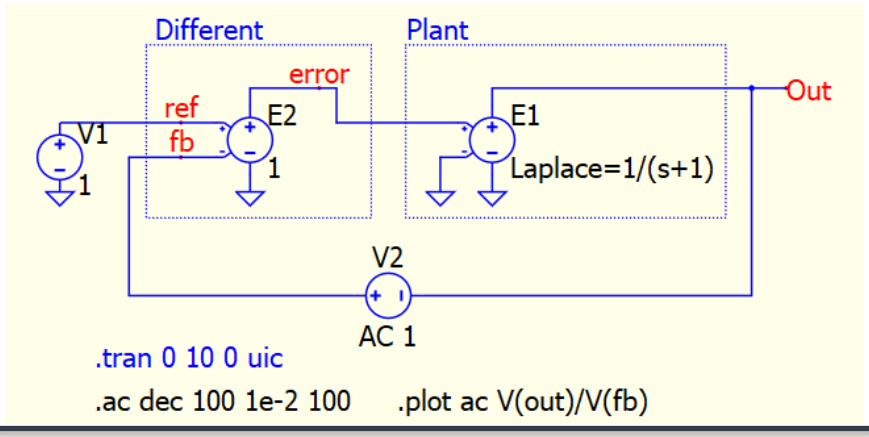


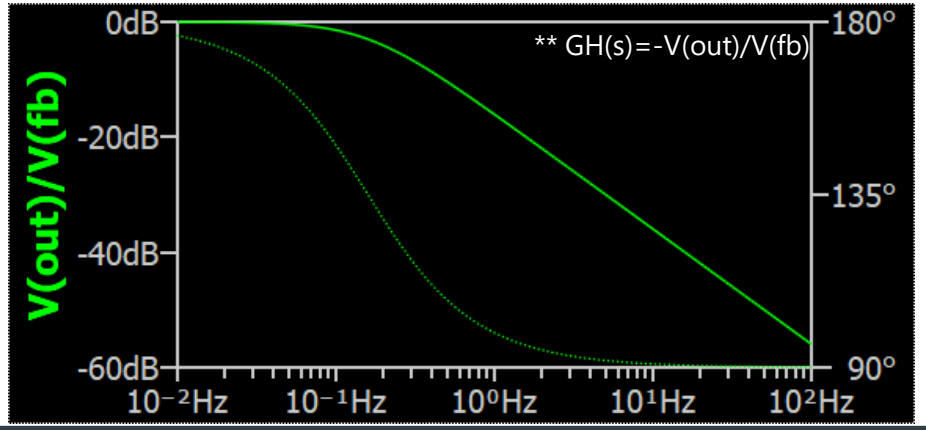


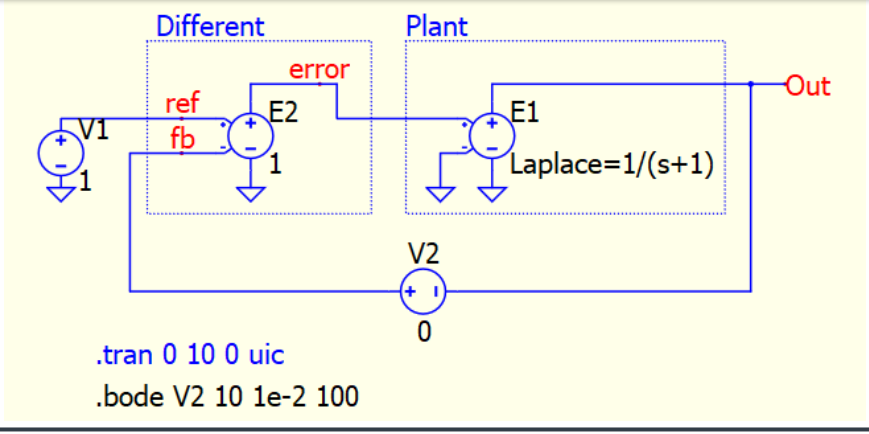


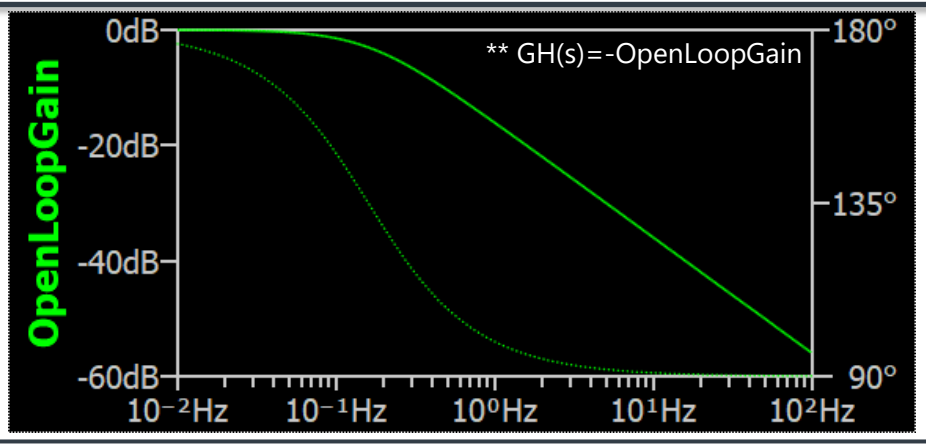
AC (.ac) and Frequency Response Analysis (.bode)

Qspice : ACmethod.qsch ; BODEmethod.qsch





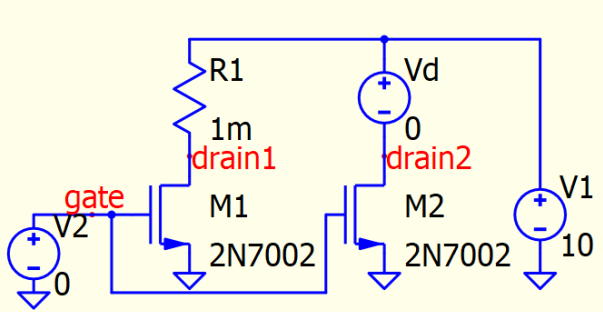


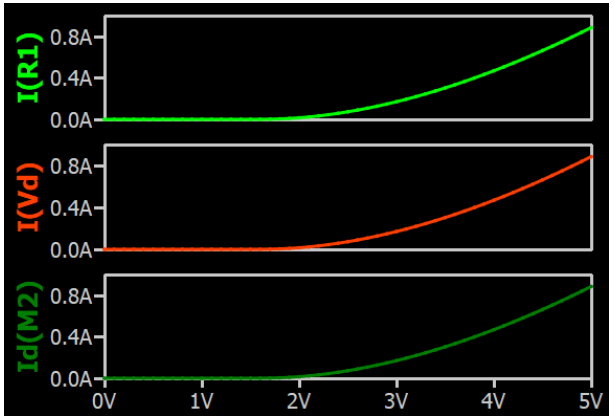


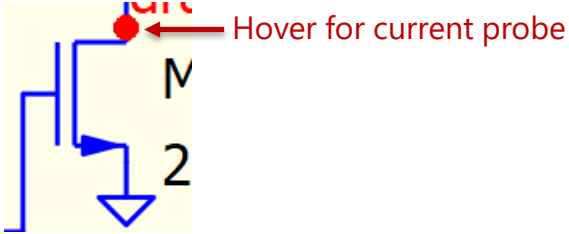
Technique to Probe NMOS Drain Current / General Current Probe

Qspice : Current Probe Method.qsch

- Current probe method
  - Method #1 : Resistor
    - Add a series resistor (small value) and probe R current
  - Method #2 : 0V Voltage
    - Add 0V voltage source and probe current of this voltage source.
    - +ve represent current flow from + to – direction within symbol (i.e. +ve current represent current flow downward in this example)
    - Hold SHIFT key in Qspice to probe voltage source current can give default +ve current, otherwise it will assign –ve to current
  - Method #3 : Probing
    - Probe current by hover point at device terminal
    - Not support sub-circuit yet
    - OR Ctrl-A (Add Plot) in waveform viewer and select Id(Mnnn)



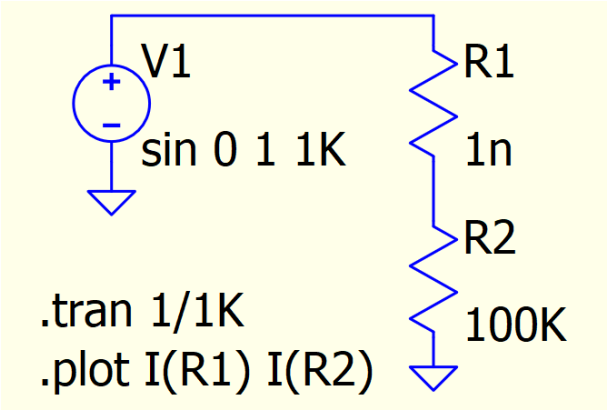


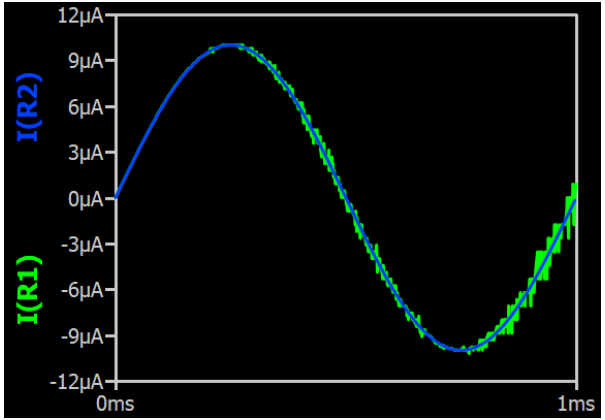


Technique to Probe NMOS Drain Current / General Current Probe

Qspice : Current Probe with Resistor - Limitation

- Limitation in R current
  - If sensing resistor with extremely small value compares to its measurement object, current reading from this resistor will be incorrect
  - Use 0V voltage source approach as replacement





- Explanation – by frank.widmann in Qspice forum
  - https://forum.qorvo.com/t/persistent-bug-warning-singular-matrix-check-node-b/17636/21

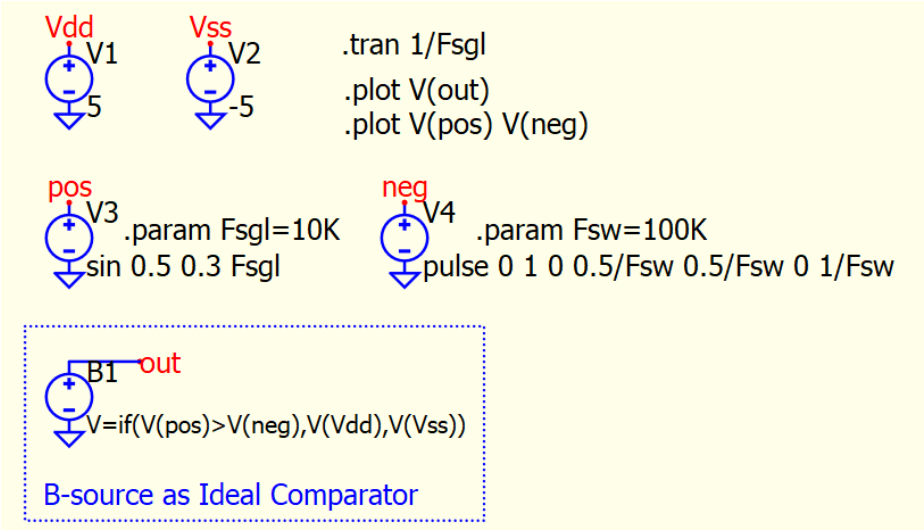
frank.wiedmann 30m

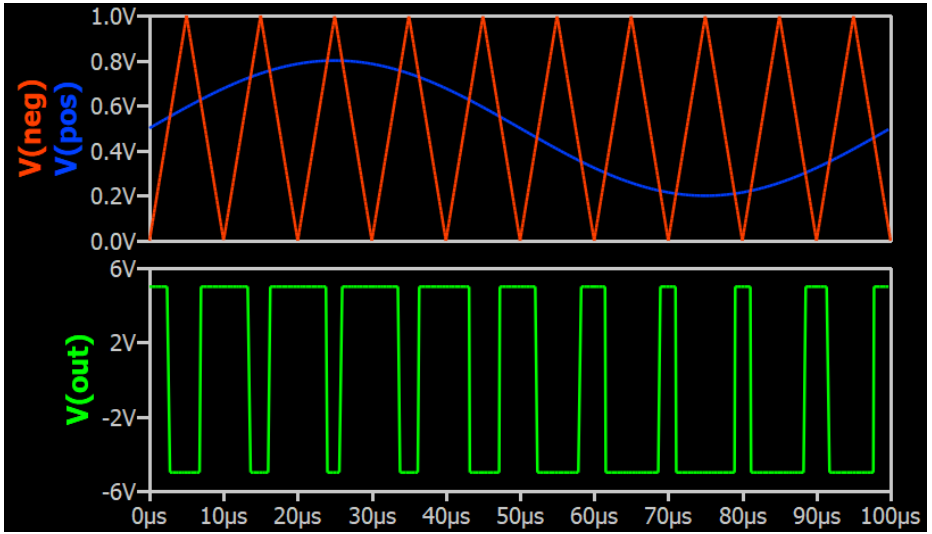
Here is the explanation for the behavior observed by @KSKelvin : SPICE uses Modified nodal analysis - Wikipedia which directly calculates the node voltages and the currents through voltage sources. The current through the small resistor, on the other hand, is calculated indirectly by dividing the voltage difference between its terminals by its resistance. With a resistance of 1u, the tolerances of the voltages are almost as large as the voltage difference, causing the observed imprecise results.

B-Source as Comparator

Qspice : B-Source as Comparator.qsch

- Concept of Ideal Comparator with Behavioral Voltage Source
  - Formula of B-source is : if(V(pos)>V(neg),V(Vdd),V(Vss))
  - Practical comparator output normally is open-drain configuration, this is just for simulation purpose

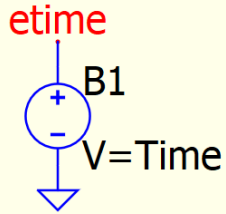




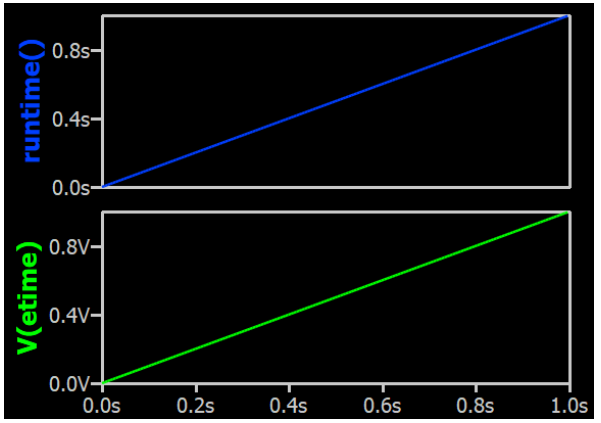
Time in .tran and Logic Diagram in Waveform Viewer with .plot

Qspice : Time in .tran.qsch ; Logic Signal Plot.qsch

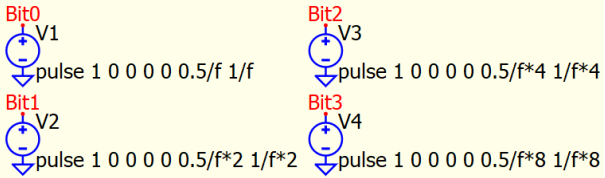
- Time in .tran
  - In .tran, simulation time is stored as a parameter named **Time**
  - Therefore, use a B-source can convert Time into a voltage
  - Time can also be used in function



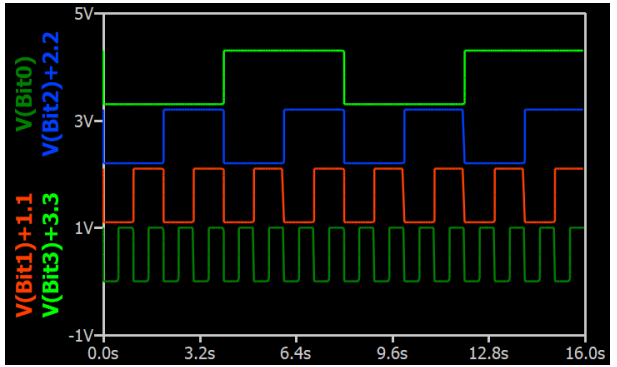

```
.tran 1    .func runtime() Time
.plot V(etime)
.plot runtime()
```



- Logic Diagram
  - A simple idea to plot logic signal into logic diagram format
  - Idea is to add an offset for each logic in .plot



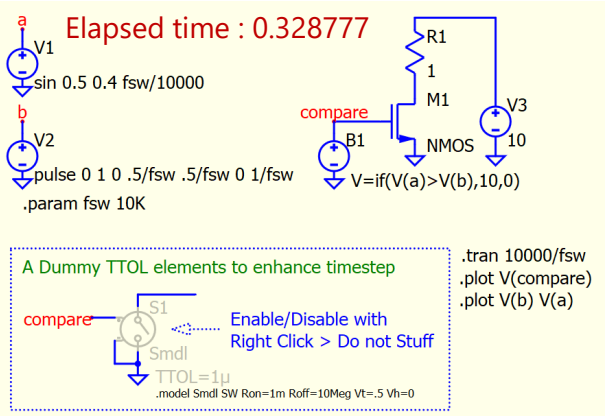

```
.param f=1
.tran 1/f*16
.plot V(Bit3)+3.3 V(Bit2)+2.2 V(Bit1)+1.1 V(Bit0)
```

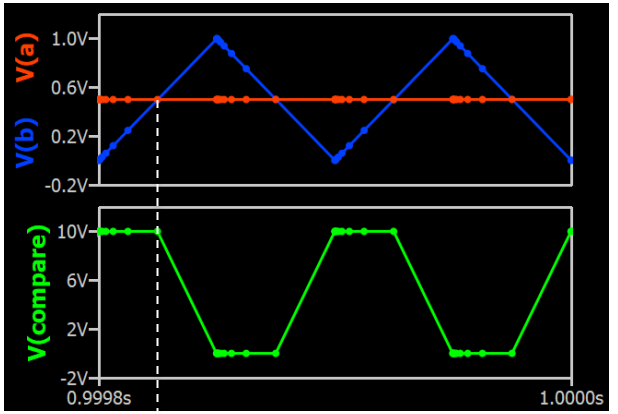


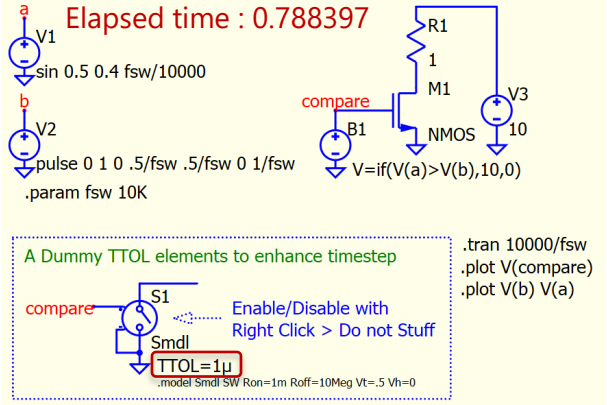
Dummy TTOL device to help in adaptive timestep

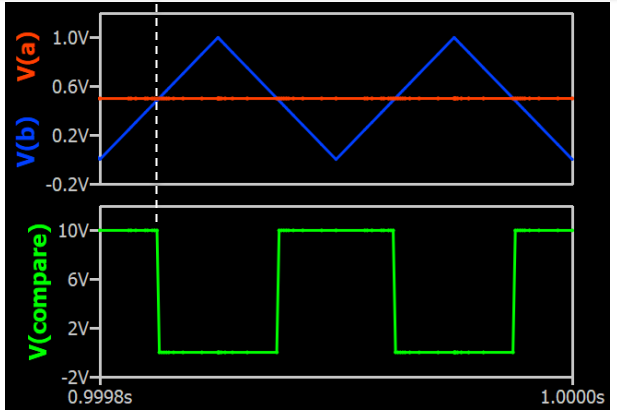
Qspice : TTOL - Dummy TTOL element - Enhance Timestep.qsch

- Dummy TTOL device
  - Qspice uses adaptive timestep
  - If a circuit uses a B-source, if(x,y,z) as a comparator, without TTOL device, its simulation timestep can far from compare instance and output looks weird
    - Example on Top Row
    - Precise time instance at compare action, but as no extra timestep after compare action, output looks like ramping as next timestep is far away (interpolation)
  - To resolve this without using MAXSTEP to limit timestep, a dummy TTOL device can be used (e.g. Switch), with TTOL instance parameters included
    - Example on Bottom Row
    - Extra time steps are added after V(compare) flip the switch, with additional time steps, output looks reasonable
    - Smaller TTOL value can yield a better results but with longer elapsed time





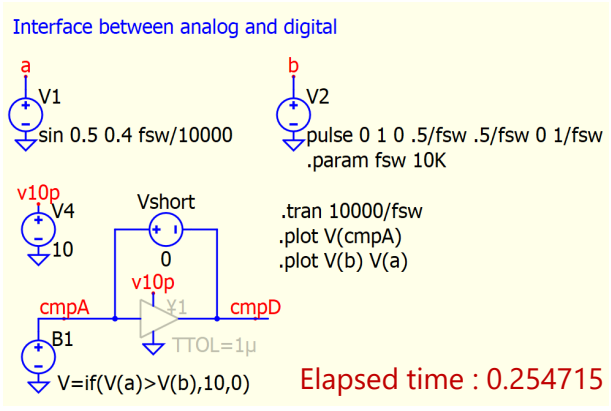


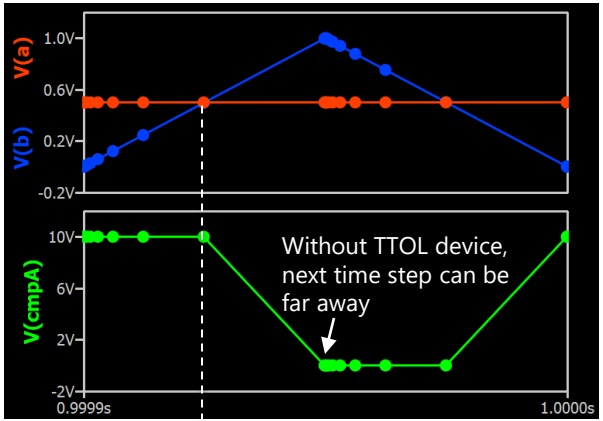


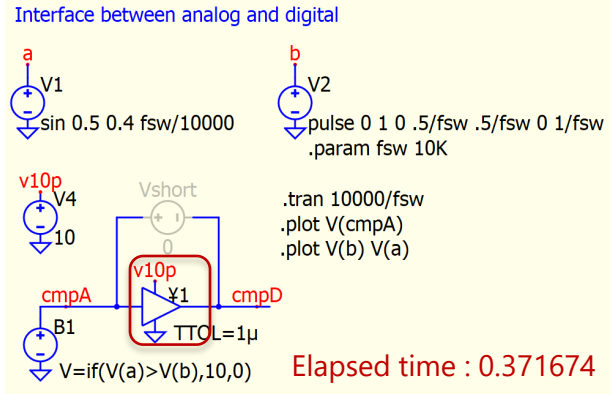
TTOL device to help in adaptive timestep (e.g. function IF)

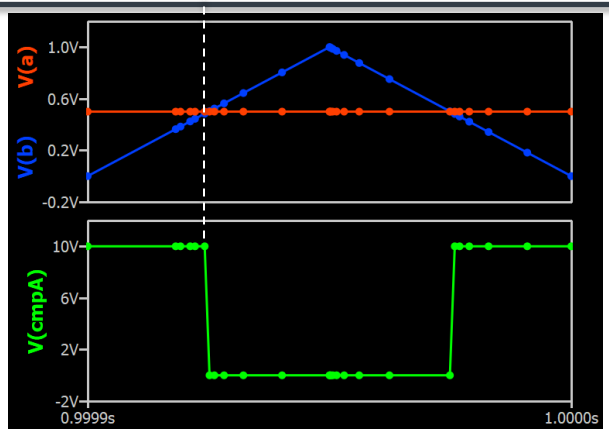
Qspice : TTOL - TTOL device to Interface Analog and Digital.qsch

- TTOL device interface
  - Qspice uses adaptive timestep
  - If a circuit uses a B-source, if(x,y,z) as a comparator, without TTOL device, its simulation timestep can far from compare instance and output looks weird
    - Example in Top Figure
    - Precise time instance at compare action, but as no extra timestep at compare action, output looks like trapezoidal as next timestep is far away
  - To resolve this without using MAXSTEP to limit timestep, a TTOL device can be used (e.g. buffer, with default TTOL=1u)
    - Example in Bottom Figure
    - Extra time steps are added after V(cmpA) flip the buffer, with additional time steps, output looks square waveform
    - Smaller TTOL value can yield a better results but with longer elapsed time





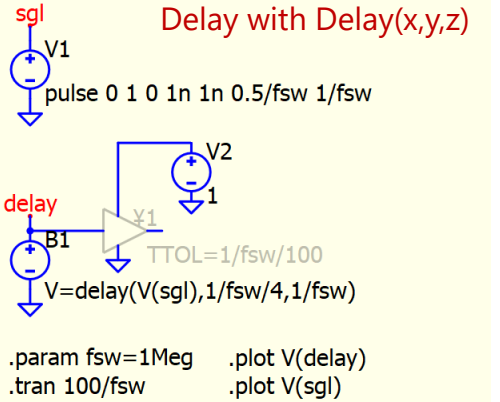


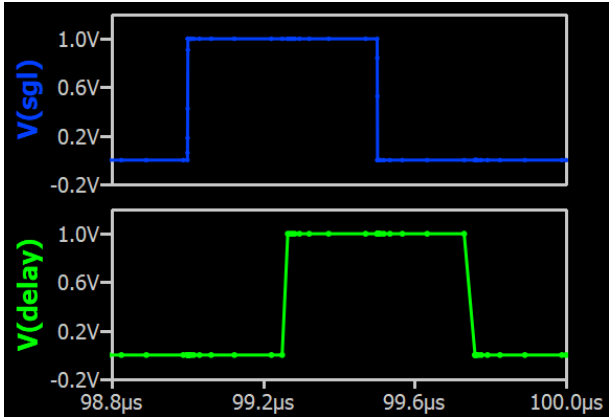


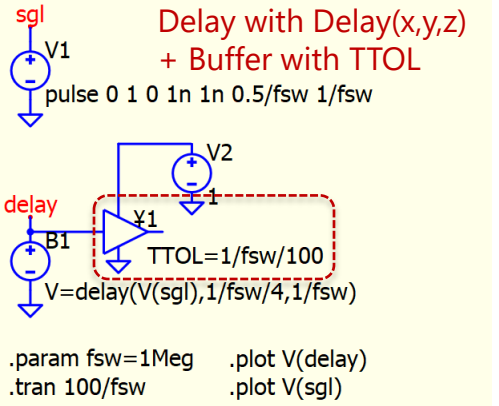
TTOL device to help in adaptive timestep (e.g. function DELAY)

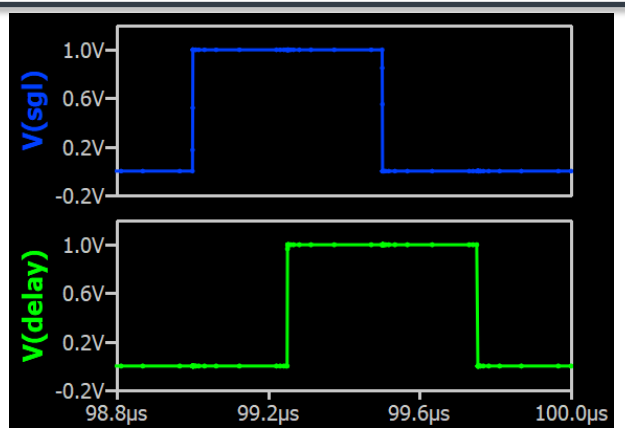
Qspice : TTOL - TTOL for Pulse Delay.qsch

- TTOL device interface
  - Pulse source has instance parameter TimeCtrl to determine timestep at each breakpoint (default is TimeCtrl=Limits), therefore, extra timestep at its rising/falling edge
    - Example in Top Figure
    - But the edge of delayed signal from behavioral source has no information of extra timestep is required, therefore, delayed signal looks like trapezoid
  - A buffer with TTOL is used to improve sharpness of pulse edge
    - Example in Bottom Figure
    - Buffer is triggered when its input cross REF voltage, with TTOL instance parameter, extra timestep is added at such moment





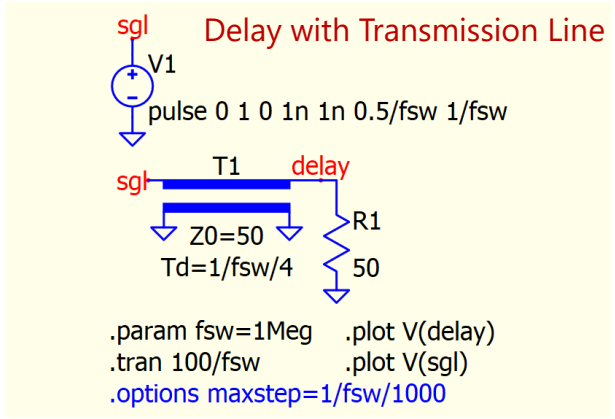


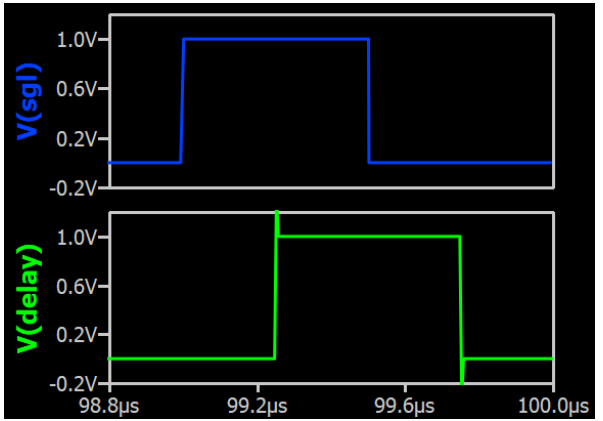


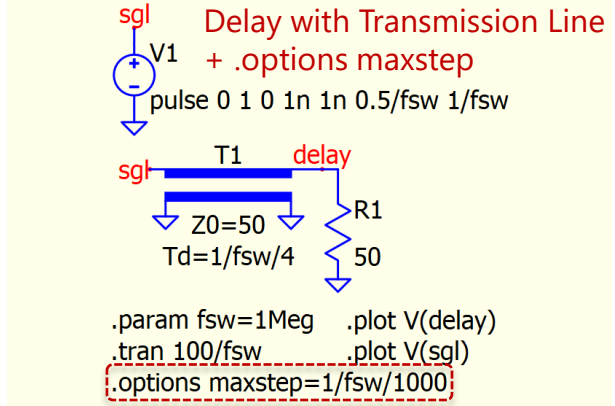
Delay with Transmission Line (alternative way for delay function)

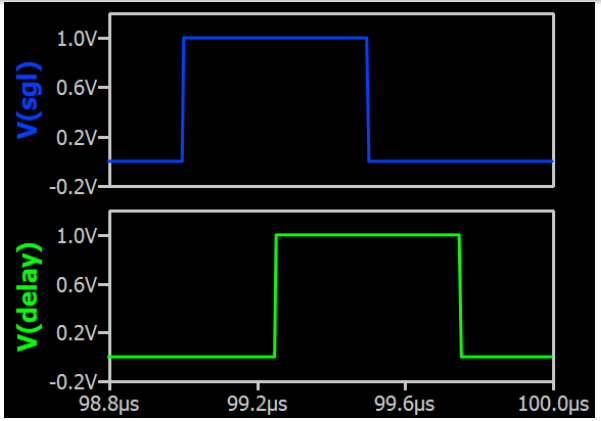
Qspice : Transmission Line for Pulse Delay.qsch

- Delay with Transmission Line
  - Beside of delay(x,y,z) function in behavioral source, delay can be generated with transmission line terminate with Zo
  - However, this approach may generate overshoot/undershoot if maximum time step is not defined, this crux is related to Qspice design as a trade between simulation time and accuracy





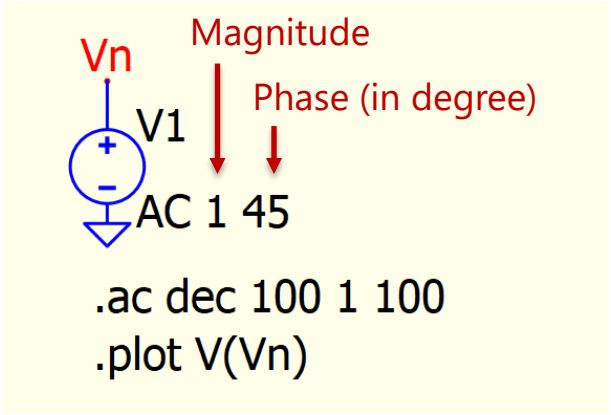


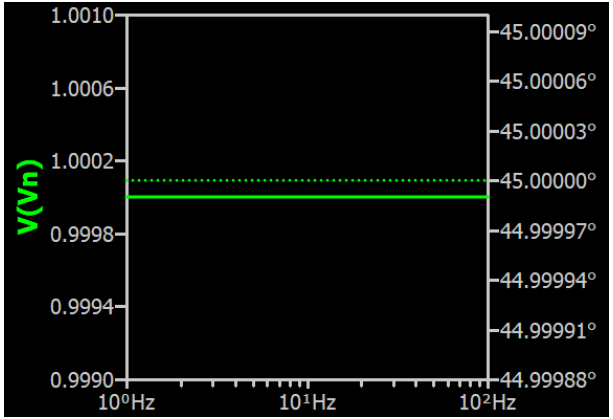


Phasor simulation technique in .ac

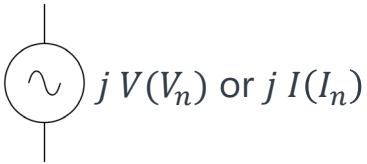
Qspice : Phasor - Source.qsch | Phasor - Dependent Source with j.qsch

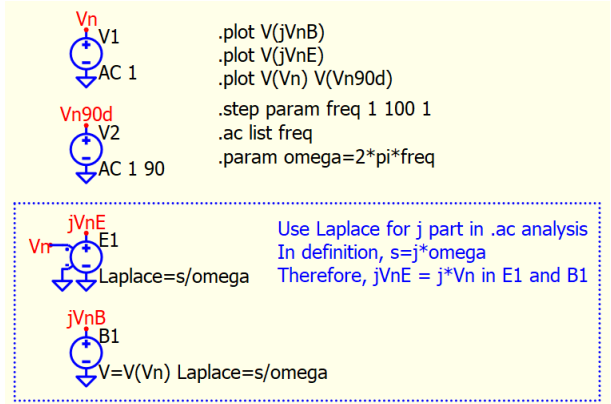
- Active Source
  - Voltage / Current Source
  - With formula $A\angle\theta$

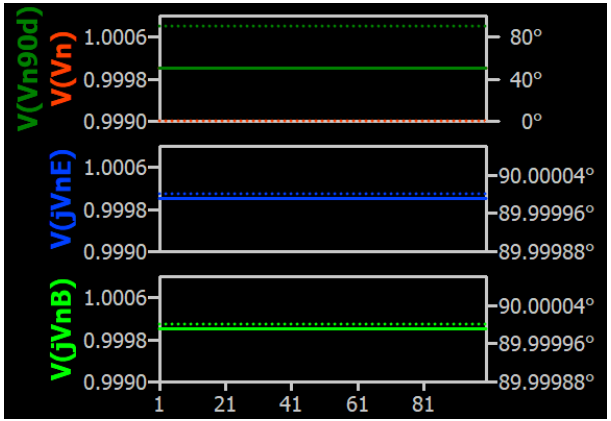




- Dependent Source
  - A dependent source with j to an active source



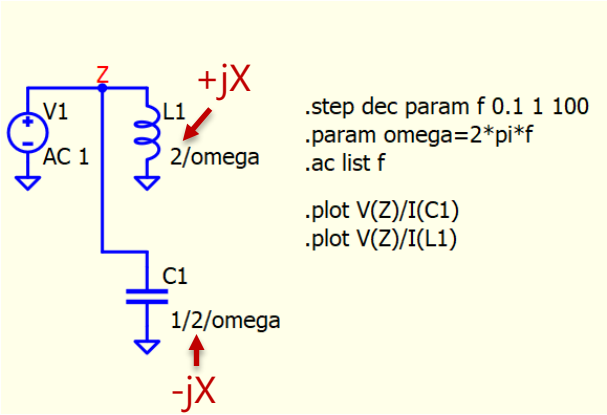


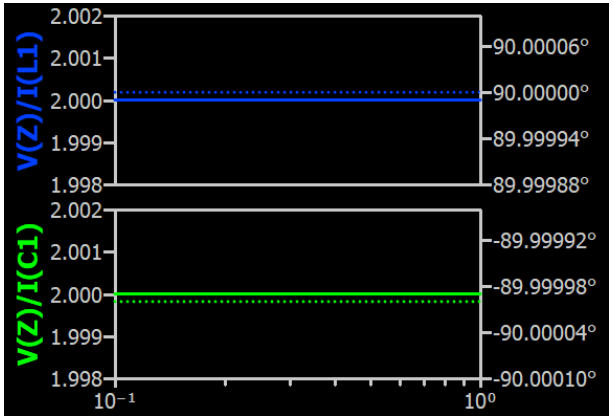


Phasor simulation technique in .ac

Qspice : Phasor - jX.qsch

- +jX or –jX
  - jX and –jX can be simulate with normalized inductance and capacitance
  - $jX_L = j\omega L \rightarrow L = \frac{X_L}{\omega}$
  - $-jX_C = -j\frac{1}{\omega C} \rightarrow C = \frac{1}{\omega X_C}$

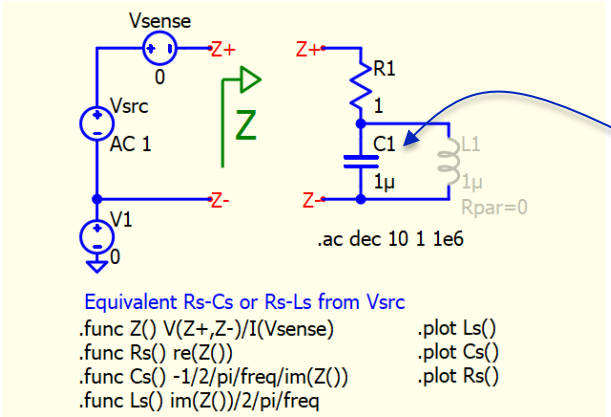


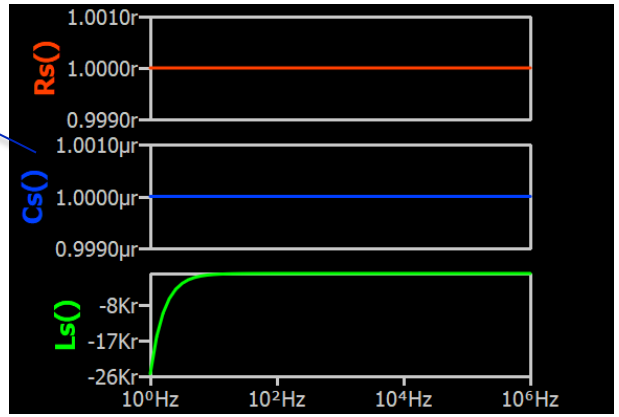


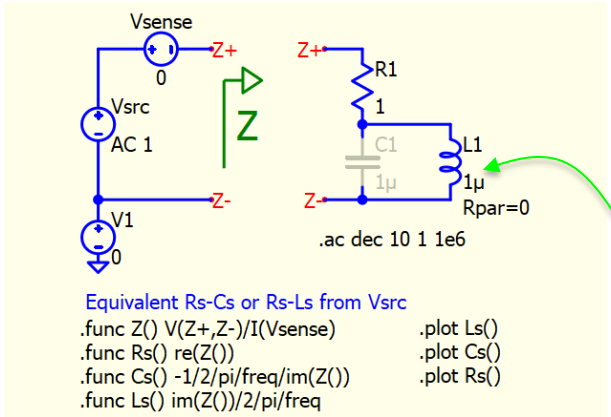
General form of impedance calculation in .ac (series model)

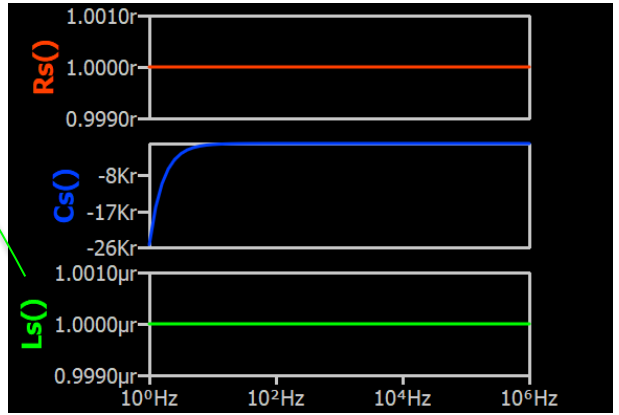
Qspice : General Form Rs-Cs Or Rs-Ls in .ac.qsch

- Impedance in .ac
  - Impedance
    - $Z = \frac{V}{I}$
  - Rs-Cs model
    - $R_s = \mathrm{re}(Z)$
    - $C_s = -\frac{1}{2\pi f \times \mathrm{im}(Z)}$
  - Rs-Ls model
    - $L_s = \frac{im(Z)}{2\pi f}$





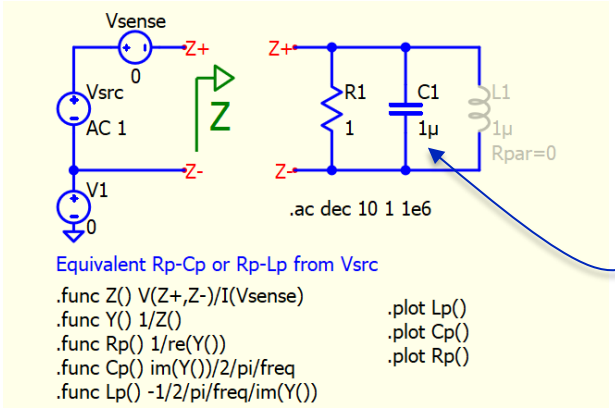


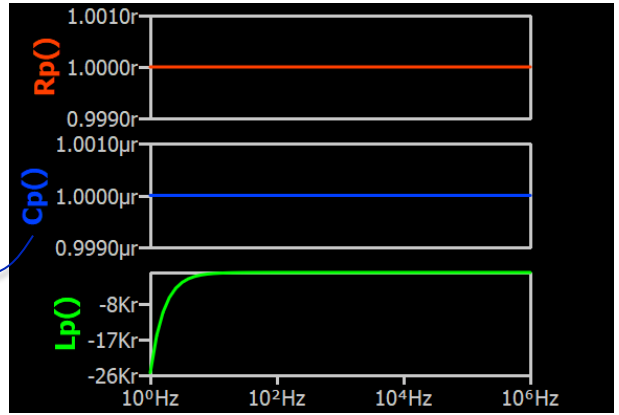


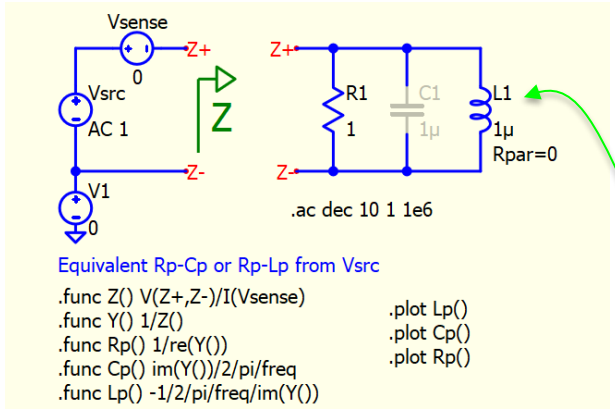
General form of admittance calculation in .ac (parallel model)

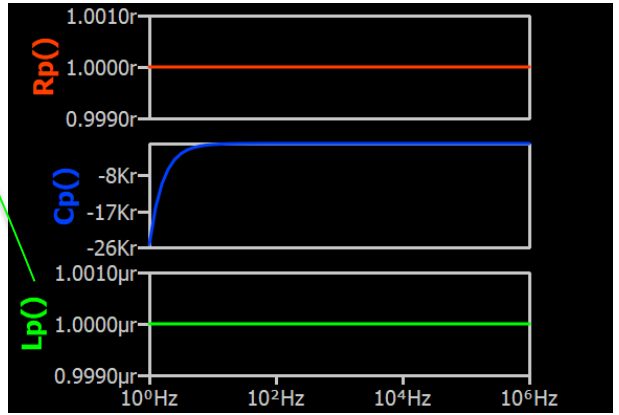
Qspice : General Form Rp-Cp Or Rp-Lp in .ac.qsch

- Admittance in .ac
  - Admittance
    - $Z = \frac{V}{I}$
    - $Y = \frac{1}{Z}$
  - Rp-Cp model
    - $R_p = \frac{1}{\mathrm{re}(Y)}$
    - $C_p = \frac{im(Y)}{2\pi f}$
  - Rp-Lp model
    - $L_p = -\frac{1}{2\pi f \times \mathrm{im}(Y)}$









General form of impedance / admittance calculation in .ac

- Qspice function formula : 2 elements series model

```
.func Z() V(Z+,Z-)/I(Vsense)
.func Rs() re(Z())
.func Cs() -1/2/pi/freq/im(Z())
.func Ls() im(Z())/2/pi/freq
```

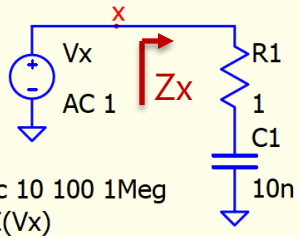
- Qspice function formula : 2 elements parallel model

```
.func Z() V(Z+,Z-)/I(Vsense)
.func Y() 1/Z()
.func Rp() 1/re(Y())
.func Cp() im(Y())/2/pi/freq
.func Lp() -1/2/pi/freq/im(Y())
```

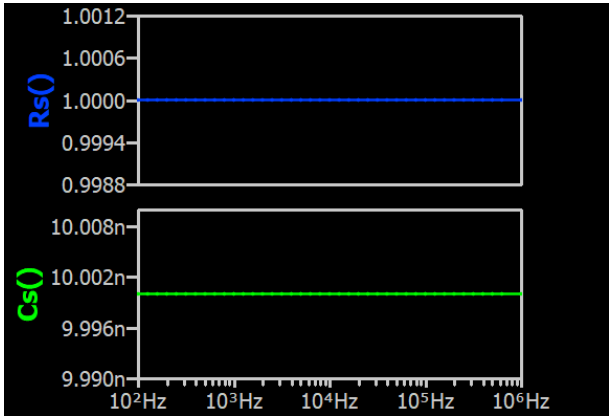
Impedance calculation in .ac with frequency or DC sweep

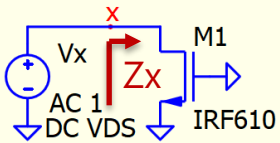
Qspice : Rs-Cs in .ac (vs freq).qsch | Rs-Cs in .ac (vs dc).qsch

- Calculate Rs-Cs in .ac
  - Impedance of Rs-Cs model
    - $Z = \frac{V}{I} = R_s + \frac{1}{j2\pi f C_s}$
    - $R_s = \mathrm{re}\left(\frac{V}{I}\right)$
    - $C_s = -\frac{1}{2\pi f \times \mathrm{im}\left(\frac{V}{I}\right)}$
  - Example #1
    - In .ac analysis, FREQ is a reserved variable. Utilize it to assist in calculating Cs
  - Example #2
    - To sweep DC voltage, set .ac to a single frequency measurement (.ac list 1Meg) and employ .step to sweep the DC voltage

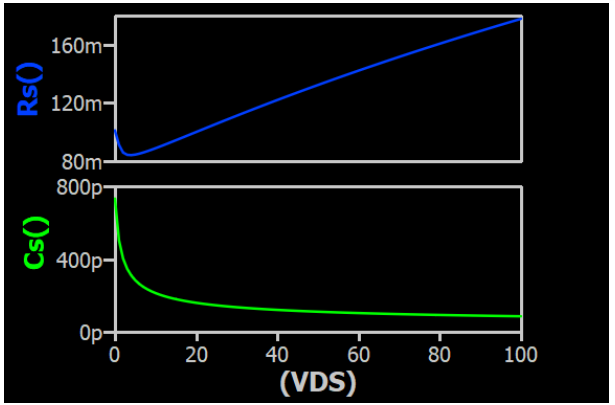



```
.ac dec 10 100 1Meg
;.plot I(Vx)

.func Rs() re(V(x)/-I(Vx))
.func Cs() -1/2/pi/freq/im(V(x)/-I(Vx))
.plot Cs()
.plot Rs()
```





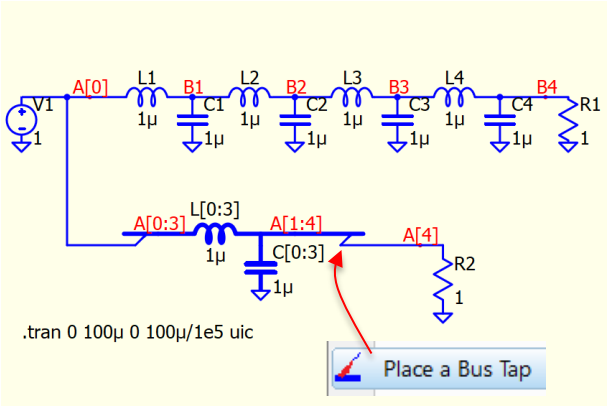

```
.step param VDS 0 100 1
.param frq=1Meg
.ac list frq
;.plot I(Vx)

.func Rs() re(V(x)/-I(Vx))
.func Cs() -1/2/pi/frq/im(V(x)/-I(Vx))
.plot Cs()
.plot Rs()
```

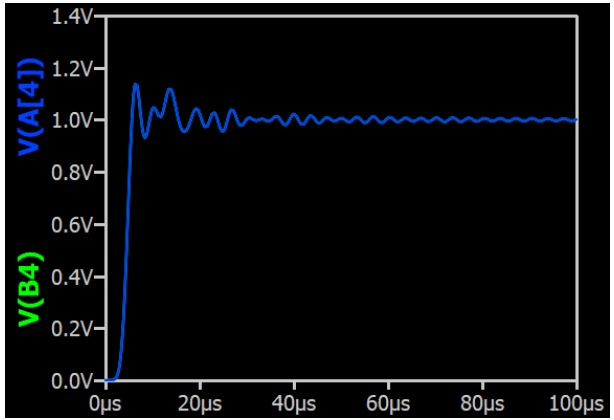


Bussing of Connections and Array of Components

Qspice : Array - LC example.qsch

- Bus and Array
  - Arrays can be defined with square brackets [n:m], which can be used for bus connections or component definitions
  - This example demonstrates using the array method to replicate a lumped LC filter network
  - To tap a single node from a bus, right-click and place a bus tap on the bus wire
  - It is recommended for the user to verify the array structure from the netlist



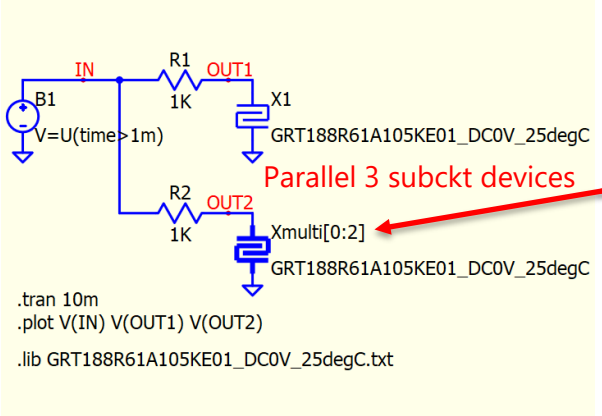


Netlist

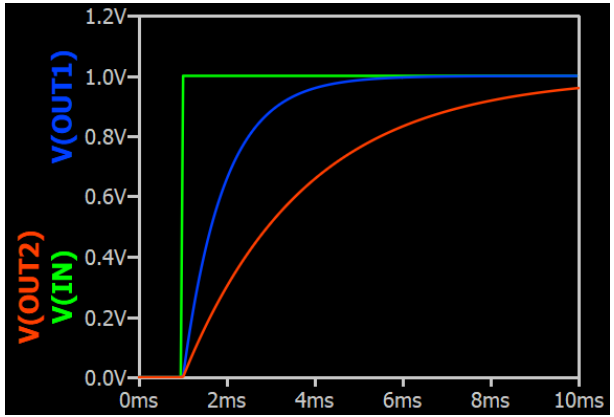
```
L1 A[0] B1 1µ
C1 B1 0 1µ
L2 B1 B2 1µ
C2 B2 0 1µ
L3 B2 B3 1µ
C3 B3 0 1µ
L4 B3 B4 1µ
C4 B4 0 1µ
R1 B4 0 1
L[0] A[0] A[1] 1µ
L[1] A[1] A[2] 1µ
L[2] A[2] A[3] 1µ
L[3] A[3] A[4] 1µ
C[0] A[1] 0 1µ
C[1] A[2] 0 1µ
C[2] A[3] 0 1µ
C[3] A[4] 0 1µ
R2 A[4] 0 1
V1 A[0] 0 1
.tran 0 100µ 0 100µ/1e5 uic
.plot V(B4) V(A[4])
.end
```

Bussing of Connections and Array of Components

- Array
  - An array of components can be used to simplify paralleling .subckt X-device




```
R1 IN OUT1 1K
B1 IN 0 V=U(time>1m)
R2 IN OUT2 1K
Xmulti[0] OUT2 0 GRT188R61A105KE01_DC0V_25degC
Xmulti[1] OUT2 0 GRT188R61A105KE01_DC0V_25degC
Xmulti[2] OUT2 0 GRT188R61A105KE01_DC0V_25degC
X1 OUT1 0 GRT188R61A105KE01_DC0V_25degC
.lib GRT188R61A105KE01_DC0V_25degC.txt
.tran 10m
.plot V(IN) V(OUT1) V(OUT2)
.end
```



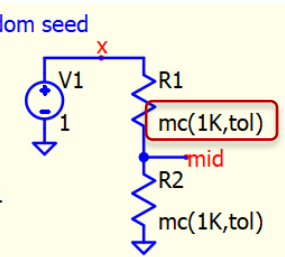
Part 6
Simulation Technique (Monte Carlo)

Monte Carlo – Basic

Qspice : \6 Simulation Technique\Monte Carlo\Monte-Carlo (.op with .meas).qsch

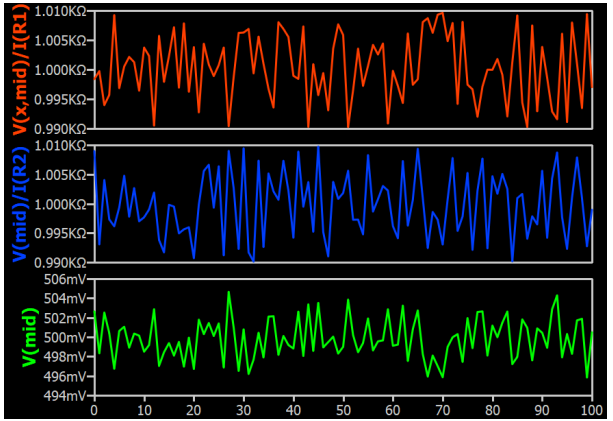
- Monte Carlo
  - Monte Carlo Analysis uses random sampling and statistical analysis to predict the outcomes of complex systems
  - MC(x,y) is Monte Carlo function
    - Output a random number between $[x(1-y), x(1+y)]$
    - mc(x,y) is equivalent to x*(1+y*(random()*2-1))

```
; use system clock as random seed
.option SEEDCLOCK

.param tol=1/100


.op
.step param LOOP 0 100 1
.plot V(mid)
.plot V(mid)/I(R2)
.plot V(x,mid)/I(R1)


.meas mid_max max V(mid)
.meas mid_min min V(mid)
.meas mid_worst param (mid_max - mid_min)
```



Output Window

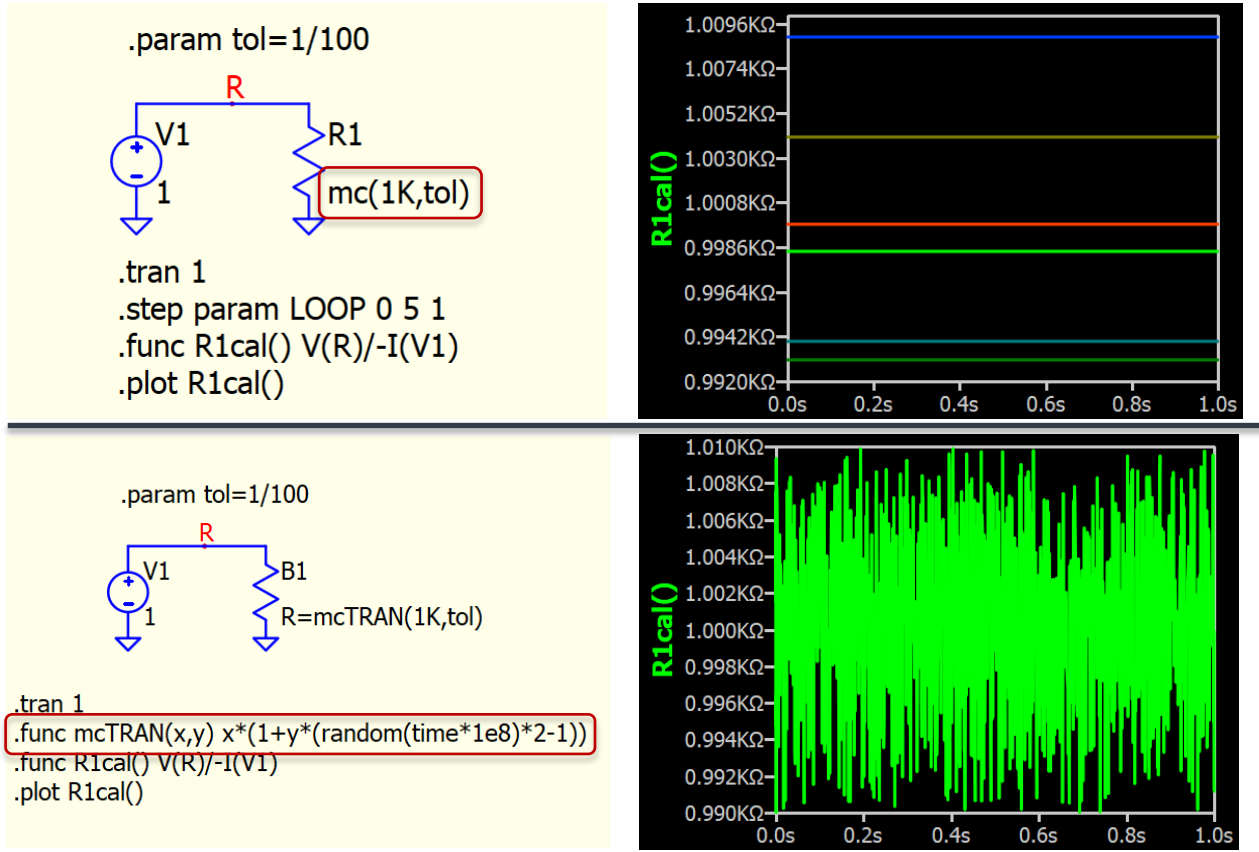
```
.meas mid_max max v(mid):
       0.504645 (at LOOP=27)
.meas mid_min min v(mid):
       0.495837 (at LOOP=99)
.meas mid_worst param (mid_max - mid_min):
     0.00880854
```



Monte Carlo – Preprocessor function

Qspice : \6 Simulation Technique\Monte Carlo\Monte-Carlo (.tran).qsch

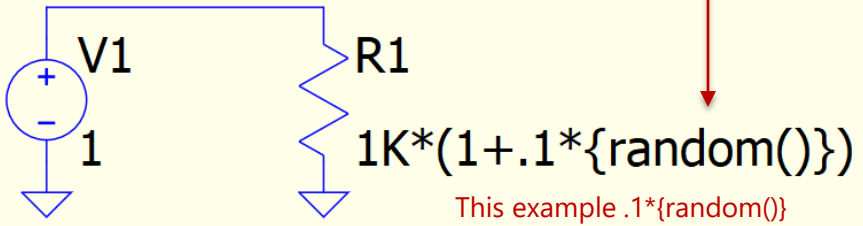
- Preprocessor function
  - MC(x,y) is a preprocessor function which is understood during the preprocessor
  - For example, in .tran, R1 is assigned a value before each step with mc() function. Therefore, behavorial resistor cannot support mc()
  - A device value changes over time requires user to write a random(seed) function where seed should change over time, where $(2\ \mathrm{random} - 1)\ y$ can generate a value between $[-y, +y]$



Monte Carlo

Qspice : Monte Carlo.qsch

Random number from 0 to 1 depending on the seed

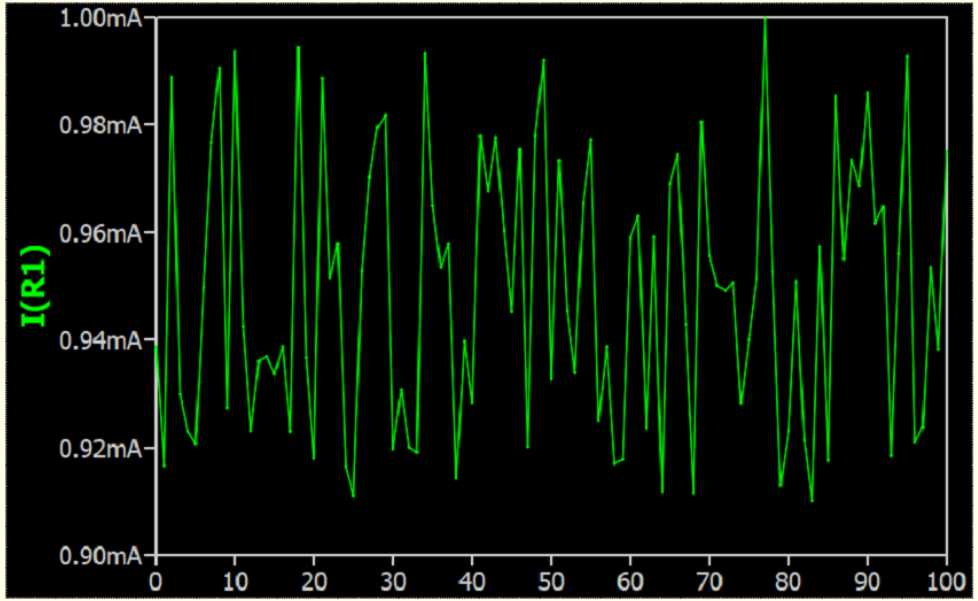


This example .1*{random()}
equivalent +0% to +10% change

```
.op
.step param dummy 0 100 1
.plot I(R1)

.options seedclock
.options seed=5
```

This enable random seed to be generated

This assign manual seed



Engelhardt

9-4-2023

OK, I just implemented

.options seedclock

It convolutes a 10MHz system clock with the simulation process ID to generate a physically random integer to seed the Mersenne Twister.

Flat(x) and MC(x,y) functions equivalent to Ltspice

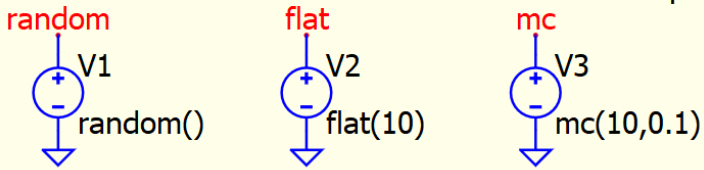
Qspice : Flat and MC Function.qsch

- Uniform random distribution
  - LTspice offers flat(x) and mc(x,y) functions, but not in Qspice (last check 10-3-2023)

- Function for flat(x) and mc(x,y)
  - .func flat(x) x*((random()*2)-1) ← Generate random [-x, x]
  - .func mc(x,y) x*(1+y*(random()*2-1)) ← Generate random [x*(1-y), x*(1+y)]

```
.step param x 1 200 1    Dummy For Loop
.op
```

```
.plot V(mc)
.plot V(flat)
.plot V(random)
```



flat(x) : Random number between -x and x with uniform distribution

```
.func flat(x) x*((random()*2)-1)
```

mc(x,y) : A random number between
x*(1+y) and x*(1-y) with uniform distribution

```
.func mc(x,y) x*(1+y*(random()*2-1))
```