Laboratory for Reliable Computing

Signal Sensing and Application Laboratory

HSPICE and Waveform

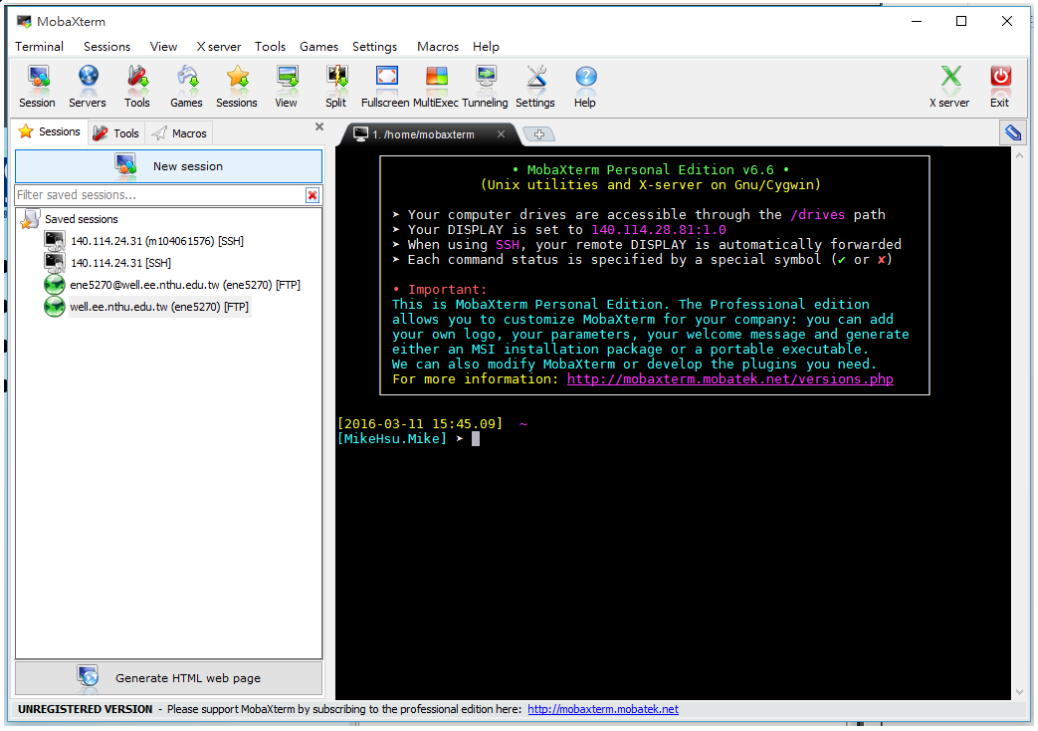
2016.03.17

Outline

- Login method
  - MobaXterm
- Elements and Device Models
- Input Sources
- Analysis Types
- Simulation Step and Graphic Tools
  - Run HSPICE & Waveform explorer

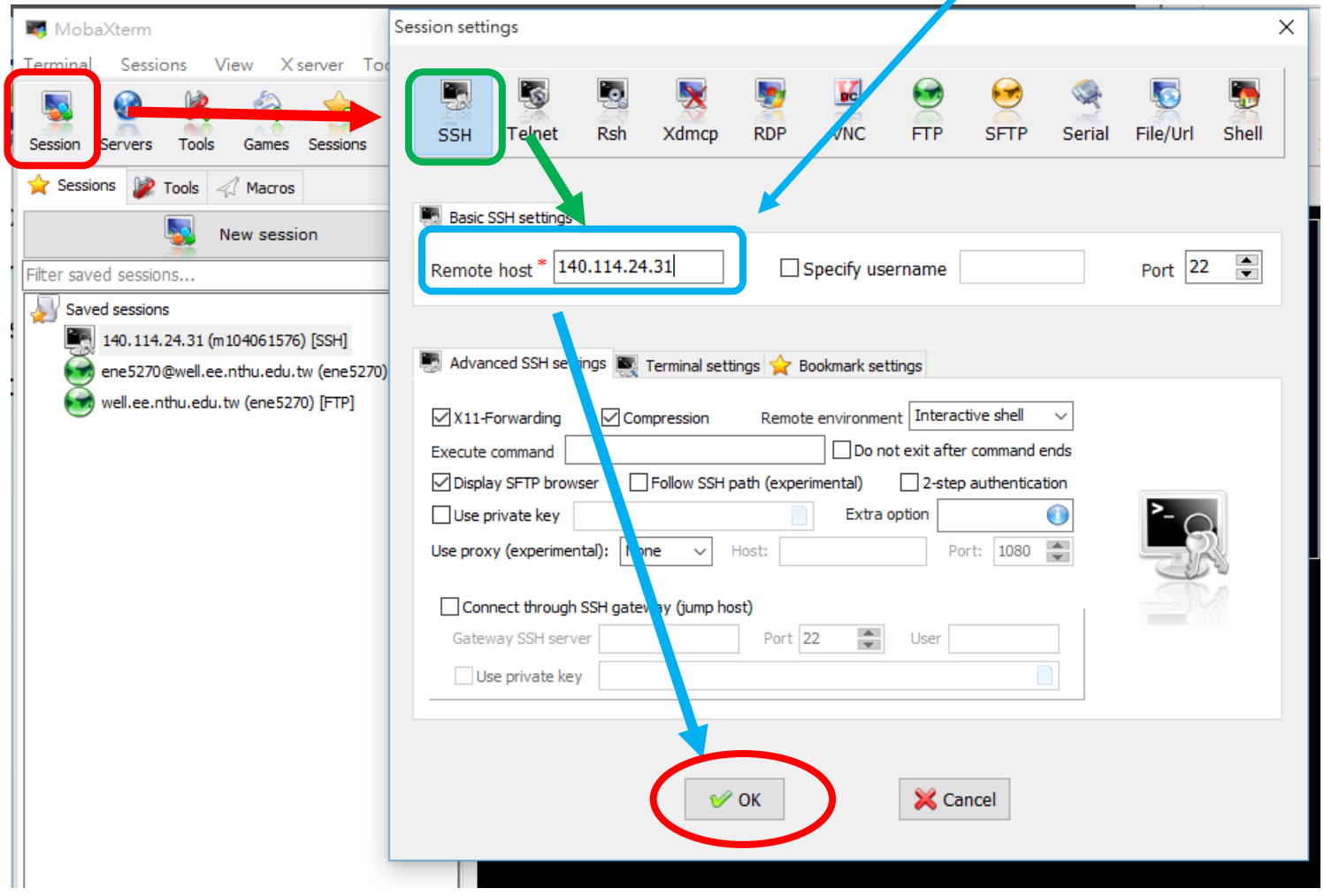
Login Method

- Use MobaXterm for example
- Download website：
  - http://mobaxterm.mobatek.net/download-home-edition.html
  - Download the free version
  - Execute the .exe file



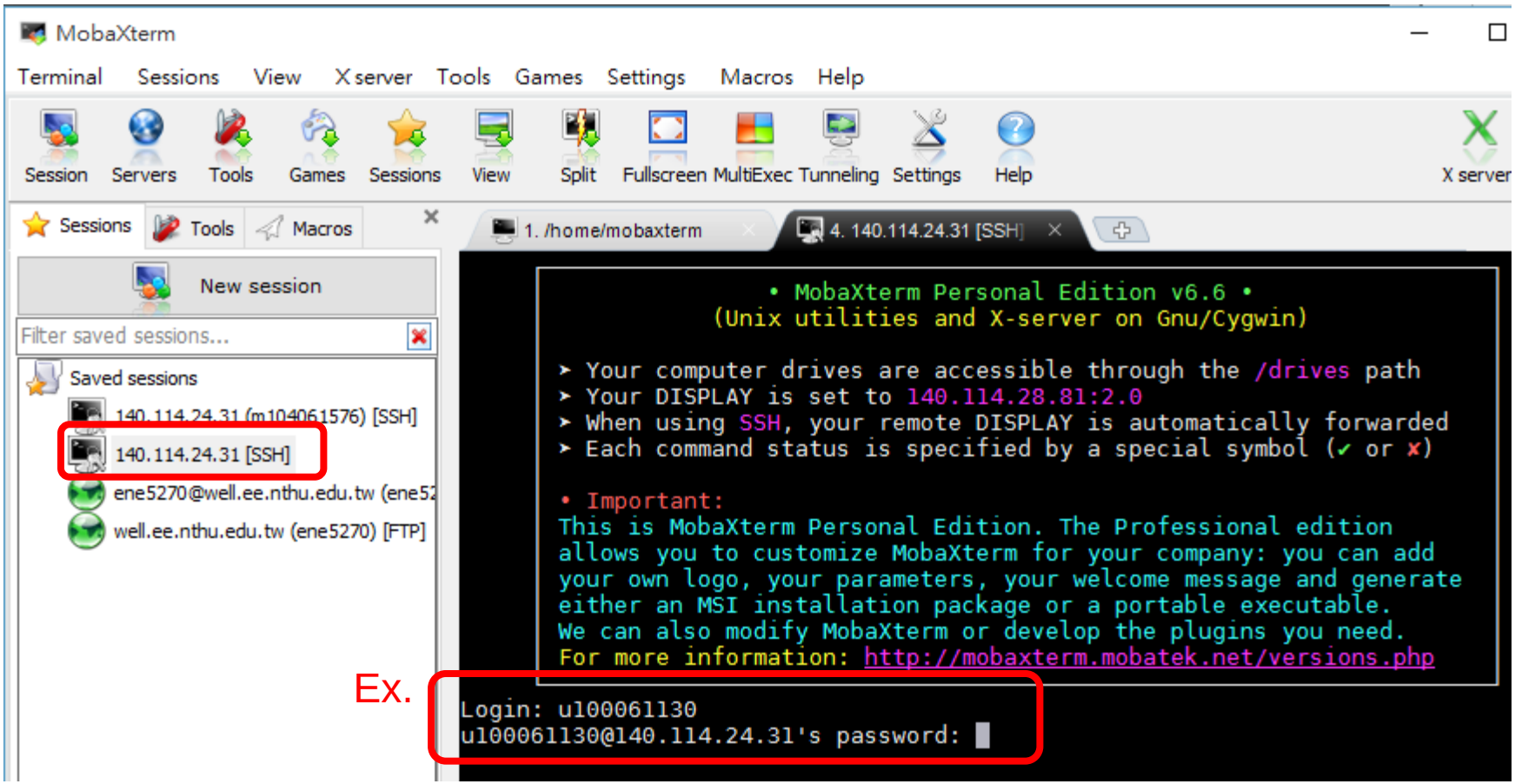
Login Method

- Session → SSH → Remote host 140.114.24.31



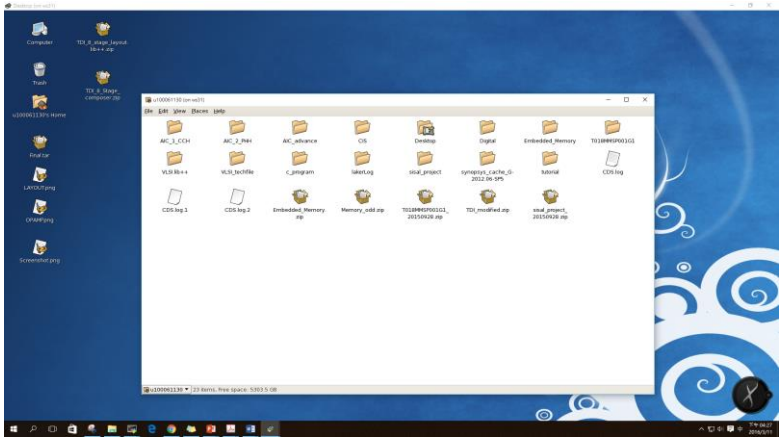
Login Method

- Login：<your account>
- Password：<your password>



Login Method

- Type ssh -X ws31
  - (Note：X need to be capital)
  - ws：choose the workstation from the list
- Type your password
- Enter the host ws31….
- When using your PC…
  You can use remote desktop by typing：nautilus &

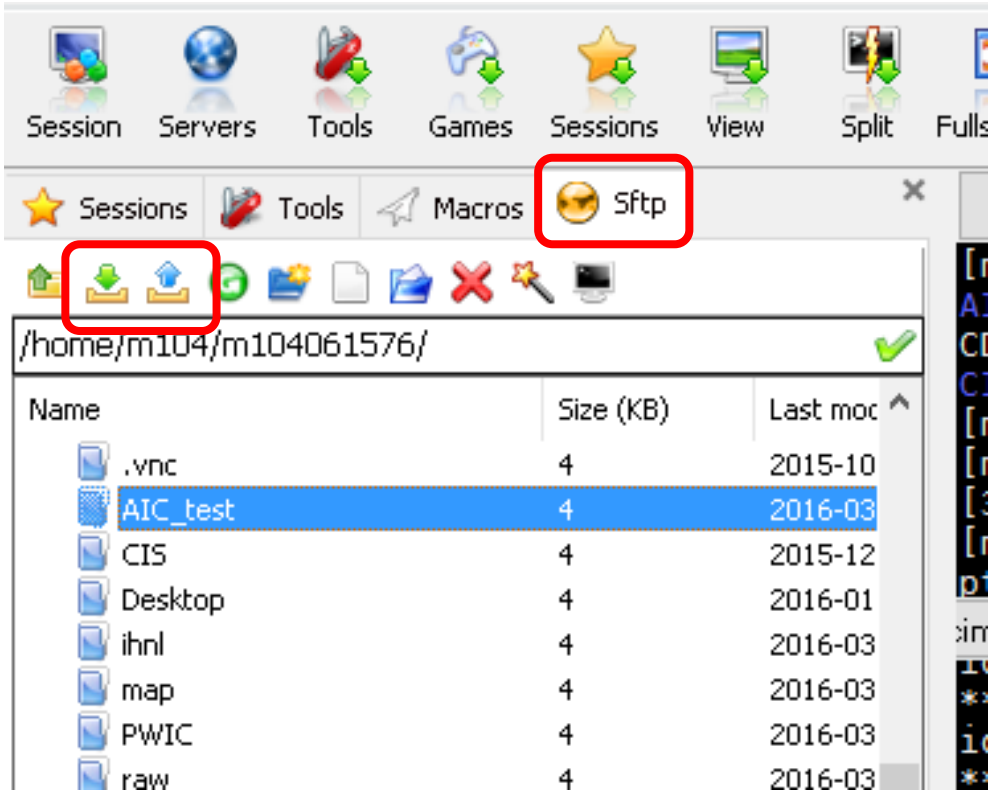


```
--------users---load average-----+----------users---load average---
ws26     0   0.00, 0.00, 0.00    |  ws38        2   0.11, 0.03, 0.01
ws27 is down                     |  ws39        2   0.03, 0.01, 0.00
ws28 is down                     |  ws40 is down
ws29 is down                     |  ws41        0   0.00, 0.00, 0.00
ws30 is down                     |  ws42        1   6.46, 6.14, 6.03
ws31     1   0.47, 0.11, 0.03    |  ws43        0   15.28, 14.99, 14.93
ws32     2   1.05, 1.05, 1.01    |  ws44        2   0.00, 0.00, 0.00
ws33 is down                     |  ws45        3   0.00, 0.00, 0.00
ws34     0   0.00, 0.00, 0.00    |  ws46        2   2.22, 2.07, 2.02
ws35     2   0.88, 0.18, 0.06    |  ws47        4   0.36, 0.24, 0.20
ws36     3   0.75, 0.17, 0.05    |  ws48        4   0.00, 0.00, 0.00
ws37     3   3.06, 3.03, 3.00    |

/usr/cadtool/cad/synopsys/SAED32_EDK/saed_mc/saed_mc.csh: No such file o
ory.
[u100061130@daisy ~]$ ssh -X ws31
u100061130@ws31's password:
```

Ex.

```
---------------Resource Monitoring Bulletin Board---------------
The following users please go check and kill your dead
process as soon as possible. The administrator will clear the
processes/jobs running exceeding 4 days. If you must run for
a long period, please notify the administrator for keeping your
process alive thru email (opr@ee.nthu.edu.tw or ylchen@ee.nthu.edu.tw).
Thanks.

Note:
1. Commands 'ps -aux' or 'top' could help look up the process ID (PID)
   of a running process.
2. Command 'kill -9 PID' can kill the process with PID.
3. Command 'grep' helps extract wanted information.
   For example, 'ps aux | grep u1234567' extracts all process of the user
   u1234567
----------------------------------------------------------------
/usr/cadtool/cad/synopsys/SAED32_EDK/saed_mc/saed_mc.csh: Permission denied.
[u100061130@ws31 ~]$
[u100061130@ws31 ~]$
[u100061130@ws31 ~]$
[u100061130@ws31 ~]$
[u100061130@ws31 ~]$ nautilus &
```

- Download & Upload files



Create a New Directory

- mkdir ##### (新增"#####"資料夾)
- ls (查看當前目錄下之檔案)
- cd ##### (進入"#####"資料夾)

```
[u100061130@ws31 ~]$ mkdir AIC_Example
[u100061130@ws31 ~]$ ls
AIC_1_CCH/     Desktop/                    VLSI.lib++/
AIC_2_PHH/     Digital/                    VLSI_techfile/
AIC_Example/   Embedded_Memory/            c_program/
AIC_advance/   Embedded_Memory.zip         lakerLog/
CDS.log        Memory_odd.zip              sisal_project/
CDS.log.1      T018MMSP001G1/              sisal_project_20150928.zip
CDS.log.2      T018MMSP001G1_20150928.zip  synopsys_cache_G-2012.06-SP5/
CIS/           TDI_modified.zip            tutorial/
[u100061130@ws31 ~]$ cd A
AIC_1_CCH/   AIC_2_PHH/   AIC_Example/ AIC_advance/
[u100061130@ws31 ~]$ cd AIC_Example/
[u100061130@ws31 ~/AIC_Example]$
```

Prepare for simulation in Synopsys HSPICE

- Library file：cic018.l
  - SPICE model
  - Syntax: .lib “cic018.l” TT/SS/FF/SF/FS

- Simulation file：XXX.sp (a netlist from schematic tool, ex: Composer)
  - Setup
  - Main circuit
    - Can include another netlist by using .inc ‘XXX.spi’.
  - Analysis

SPICE Netlist Example

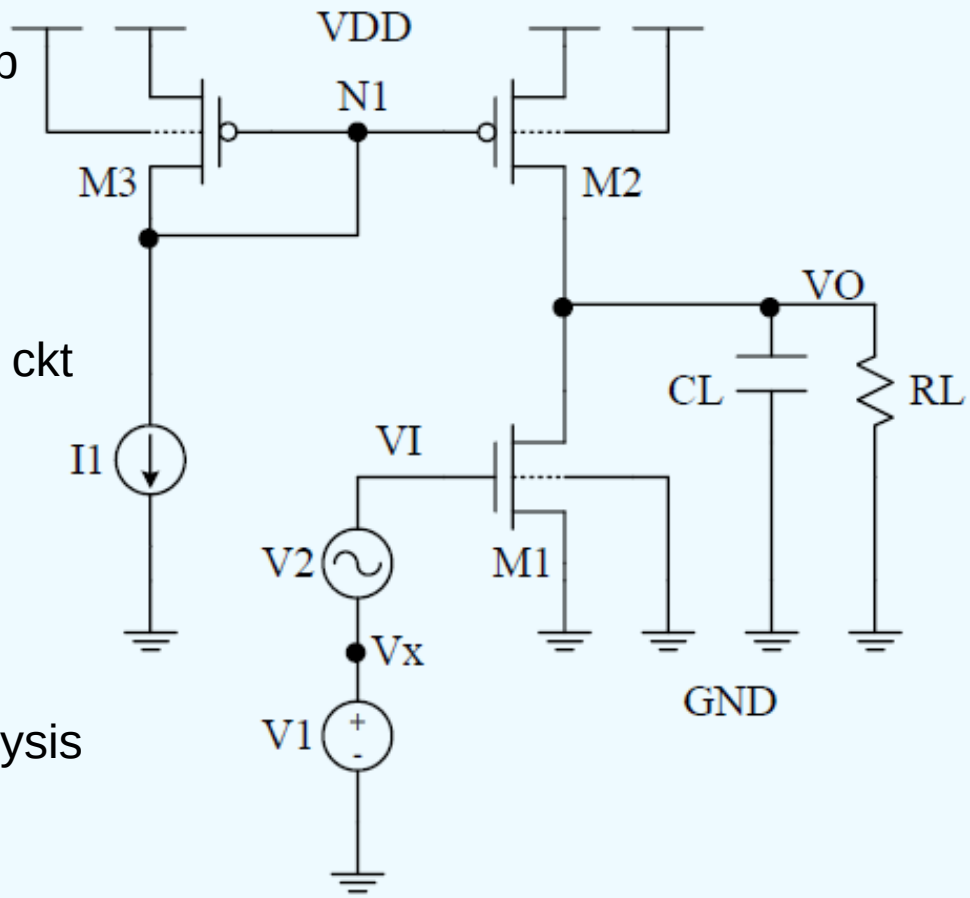
```
*a common source amplifier with active load

.prot
.lib "cic018.l" TT
.unprot
.option post=1 ACCT CAPTAB

**** Netlist ****

M1 VO VI GND GND N_18 W=4.2u L=1u M=1
M2 VO N1 VDD VDD P_18 W=5u   L=1u M=2
M3 N1 N1 VDD VDD P_18 W=5u   L=1u M=1

RL VO GND 10MEG
CL VO GND 0.1p

**** Sourcec ****

Vsup VDD GND DC=1.8
V1   Vx  GND DC=1
V2   VI  Vx  AC=1
I1   N1  GND DC=100u

**** Analysis ****

.OP
.DC V1 0 1.8 0.01
.AC DEC 100 1K 1G

.PRINT DC V(VO)
.PLOT  DC V(VO)
.PROBE AC VDB(VO)

.END
```

setup

Main ckt

Analysis

Different lib. has different MOS model name.



```
*a common source amplifier with active load

.prot
.lib "cic018.l" TT
.unprot
.option post=1 ACCT CAPTAB

**** Netlist ****

M1 VO VI GND GND N_18 W=4.2u L=1u M=1
M2 VO N1 VDD VDD P_18 W=5u   L=1u M=2
M3 N1 N1 VDD VDD P_18 W=5u   L=1u M=1

RL VO GND 10MEG
CL VO GND 0.1p

**** Sourcec ****

Vsup VDD GND DC=1.8
V1   Vx  GND DC=1
V2   VI  Vx  AC=1
I1   N1  GND DC=100u

**** Analysis ****

.OP
.DC V1 0 1.8 0.01
.AC DEC 100 1K 1G

.PRINT DC V(VO)
.PLOT  DC V(VO)
.PROBE AC VDB(VO)

.END
```

The first line is always a comment.

library

.prot/.unprot
Things btw will not appears in results file

Simulation options

註解方式
可用"******"(限單行開頭)
或"$"(行中)

不分大小寫

(0, GND, GND!)Always refer to global ground

.op 分析節點偏壓

Always ".end"

Scale Factor

Prefix	Scale Factor	Multiplying Factor
Tera	T	1e+12
Giga	G	1e+9
Mega	MEG or X	1e+6
Kilo	K	1e+3
Milli	M	1e-3
Mikro	u	1e-6
Nano	n	1e-9
Pico	p	1e-12
Femto	f	1e-15
Atto	a	1e-18

Device

Passive Devices
- Resistor – R
- Capacitor – C
- Inductor – L

Rx node1 node2 value

Cx node1 node2 value

Lx node1 node2 value

Active Devices
- Diode – D
- BJT – Q
- MOSFET – M

Qx C B E m_name

*m_name: model name
P_18/N_18 in cic018.l

Mx D G S B m_name W=value L=value m=value

Other Devices
- Subcircuit – X
- Source – V,I
- Behavoral – E, G, H, F, B
- Transmission Lines – T, U, O

Xx node1 node2 … nodeN name

Subcircuits

- Use hierarchical structure to simplify complex connection
- Definition with .subckt and .ends
- Use X<subckt_nanme> to call subcircuit.

Example:

subckt_name

```
.subckt CSAmp VI VO NI VDD GND
M1 VO VI GND GND N_18 W=4.2u L=1u M=1
M2 VO NI VDD VDD P_18 W=5u L=1u M=2
M3 NI NI VDD VDD P_18W=5u l=1u M=1
.ends
```

Call subckt `X1 VI VO NI VDD GND CSAmp`

注意subckt_name位置

注意! subckt若沒接出VDD GND 在setup時須加上.global VDD GND

- Access nodes of subcircuits by "(.)" extension
- Ex：.print V(X1.node)

Input Source

- Independent source elements --- DC/AC
  - Syntax：

```
Vxx n1 n2 <DC=dcval> <AC=acval>, <ac.phase>
Ixx n1 n2 <DC=dcval> <AC=acval>, <ac.phase>
```

  - Ex：

```
V1 net1 net2 DC=1.8v
V2 net3 net4 3.3
I3  net5 net6 1uA
```

```
Vinp Vinp 0 DC common AC 0.5 0
Vinn Vinn 0 DC common AC 0.5 180
```

*而外定義 common mode
.param common=0.9

  - DC sweep range is specified in .DC analysis statement.
  - AC frequency sweep range is specified in .AC analysis statement.

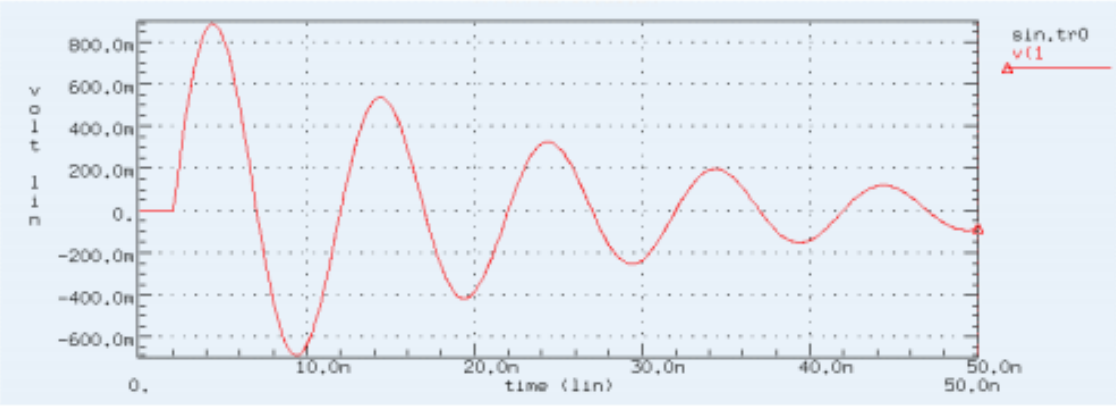
Independent source elements

- Independent source elements --- Transient
  - SIN：

```
SIN ( Voffset  Vacmag < Freq Tdelay Dfactor > )
```

  - Example：

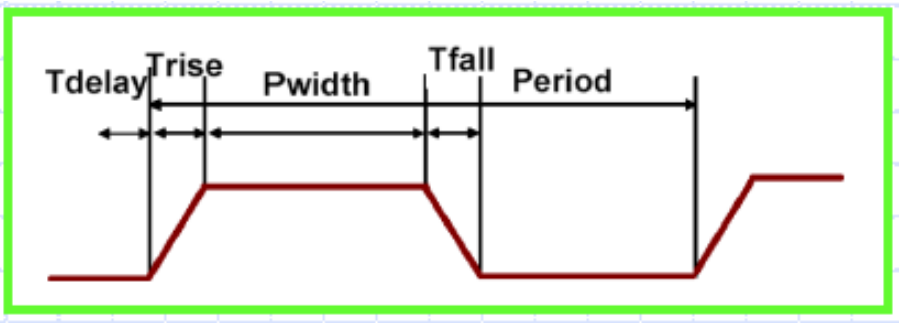
```
Vin   3   0   SIN   (  0V   1V   100Meg   2ns   5e7 )
```



Independent source elements

– PULSE (pulse waveform)：

PULSE (V1 V2 < Tdelay Trise Tfall Pwidth Period >)



Independent source elements

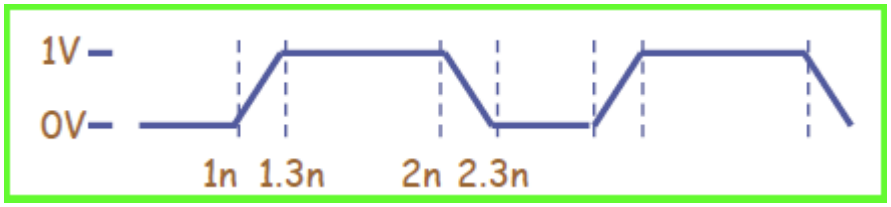
– PWL (piece-wise linear waveform)：

PWL (<t1 v1 t2 v2 .....> <R<=repeat>> <Tdelay=delay>)

$ R=repeat_from_what_time TD=time_delay_before_PWL_start

– Example：

V3 10 5 PWL 0n 0V, 1n 0V, 1.3n 1V, 2n 1V, 2.3n 0V, R 0



Analysis Types

- .OP：Operating point analysis
  - In .tran simulation, resulting DC operating point is initial estimate)
  - syntax： .OP
  - Example：.OP
- .DC：sweep parameter, source and temperature values
  - syntax： .DC <var1> <start> <stop> <step>
  - Example： .DC Vin 0 1.8 0.1
- .AC：sweep frequency
  - syntax： .AC <DEC/LIN> <np> <start> <stop>
  - Example： .AC DEC 10 1kHz 10MHz
- .Tran：sweep time
  - syntax： .TRAN <step> <stop>
  - Example： .TRAN 1ns 10us

Analysis Types

- .Probe：probe the observation will not show in the result file but can be seen at waveform
  - syntax： .probe V(net) I(device)
  - Example：.probe V(Vout) or .probe I(MCS)
- .Print：Print the observation in result file
  - syntax： .print V(net) I(device)
- .Plot：plot the observation in the result file
  - syntax： .plot V(net) I(device)

Simulation Step and Graphic Tools

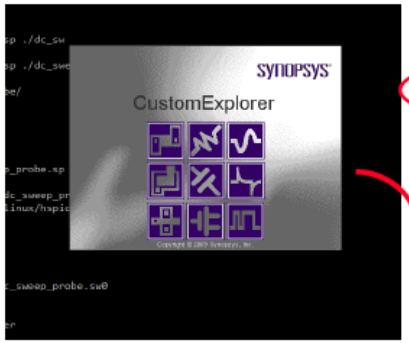
- Running HSPICE：
  - .l file (library)
  - .sp file
  - Make sure all files are in the same folder.
- 指令：
  - Type hspice XXX.sp >! XXX.lis at terminal

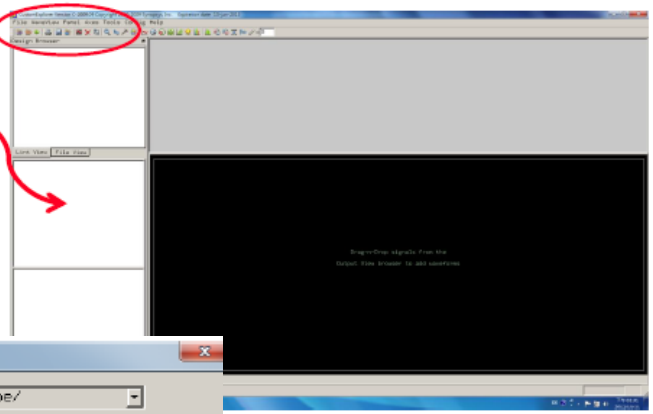
```
[m9761564@ws23 ~/microelectronics]$ hspice hw3.sp >! hw3.lis
>info:          ***** hspice job concluded
real 0.42
user 0.06
sys 0.00
[m9761564@ws23 ~/microelectronics]$ ls
cic018.l  hw3.ac0  hw3.ic0  hw3.lis  hw3.sp  hw3.sp~  hw3.st0  hw3.sw0
```

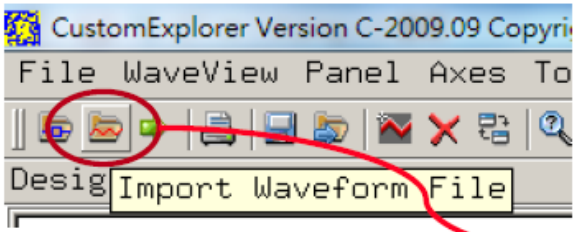
  - Hspice job concluded → circuit run correctly
  - Hspice job aborted → circuit has error (error will show in .lis file)

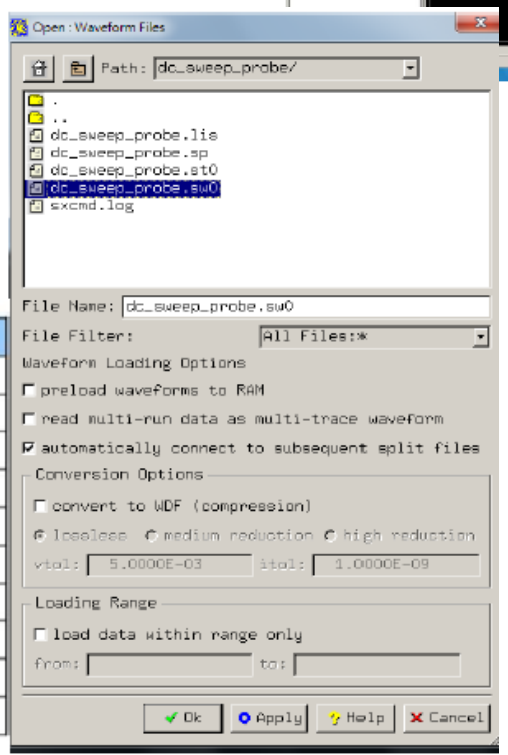
Simulation Step and Graphic Tools

- SPICE explorer
  - Type wv & at terminal
  - Or sx &



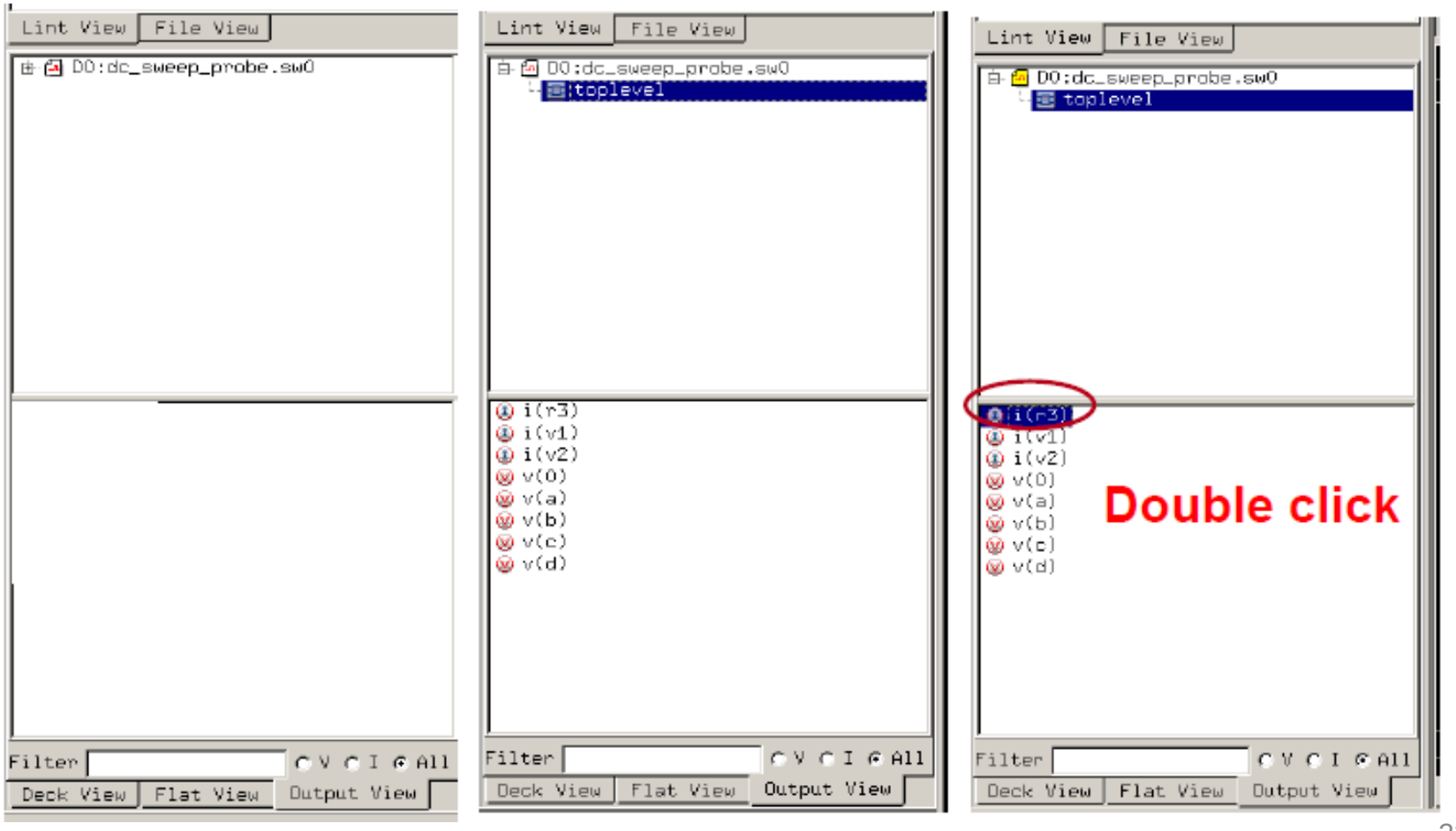




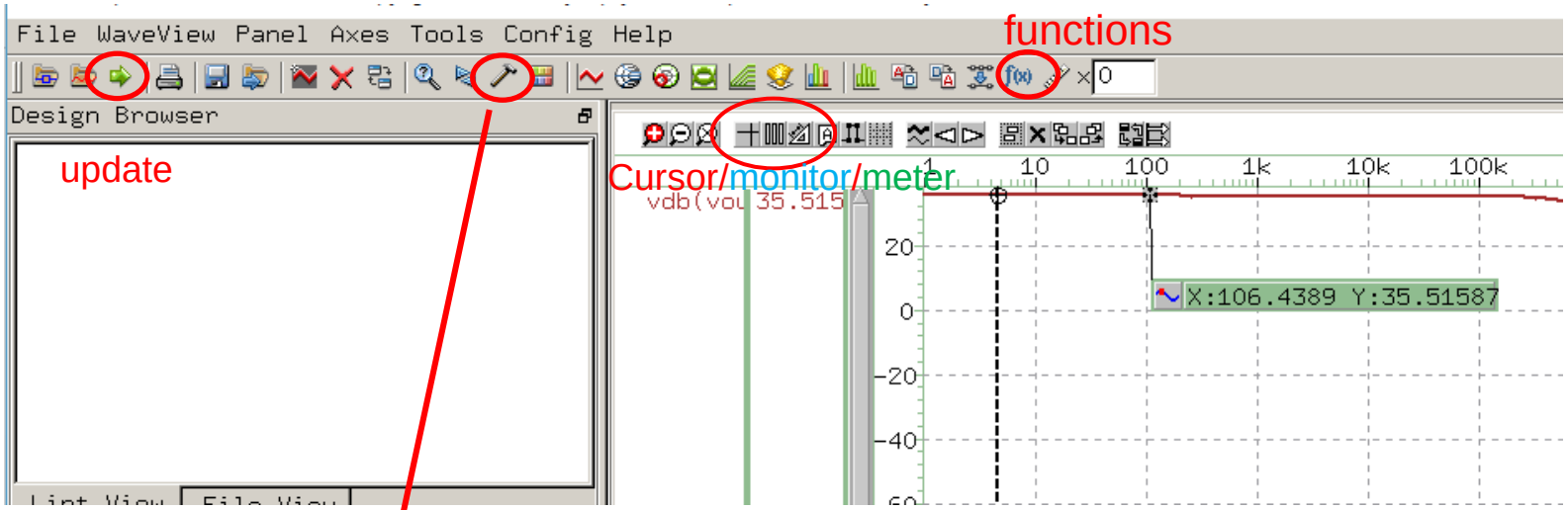


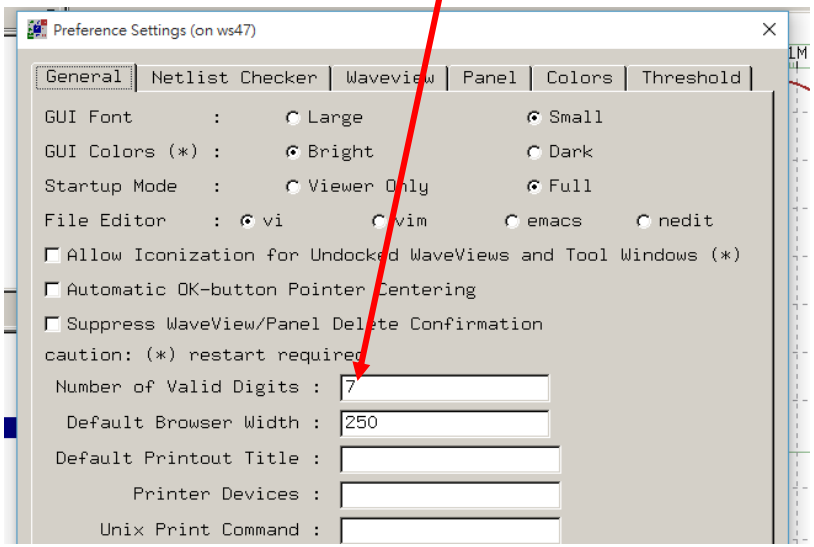
Ouput File Type	Extension
Output listing	.lis
Transient analysis results	.tr#
DC analysis results	.sw#
AC analysis results	.ac#
Transient analysis measurement results	.mt#
DC analysis measurement results	.ms#
AC analysis measurement results	.ma#
FFT analysis graph data files	.ft#
Output status files	.st#
Nets operation voltages	.ic#

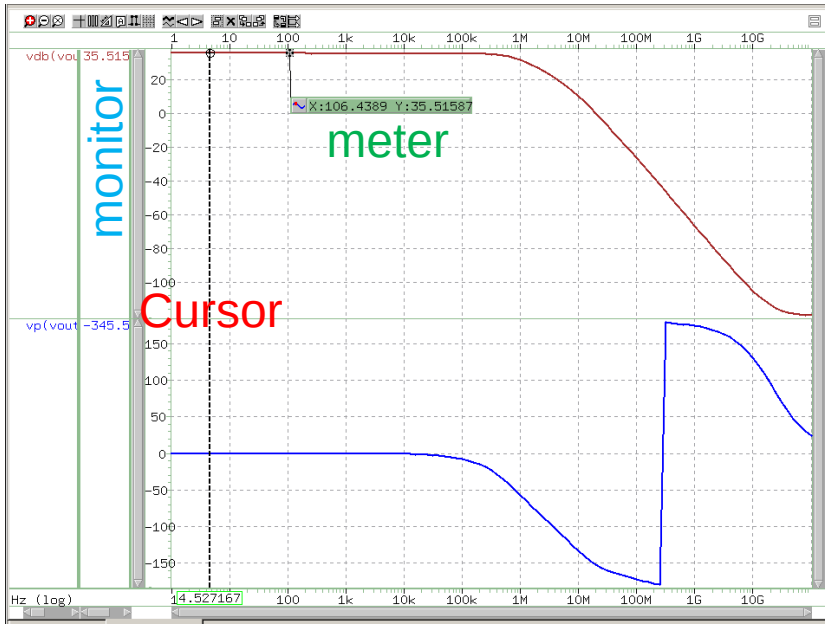
- Choose the node



Tools


File WaveView Panel Axes Tools Config Help
functions
Design Browser
update
Cursor/monitor/meter
X:106.4389 Y:35.51587


Preference Settings (on ws47)
General
Netlist Checker
Waveview
Panel
Colors
Threshold
GUI Font : Large Small
GUI Colors (*) : Bright Dark
Startup Mode : Viewer Only Full
File Editor : vi vim emacs nedit
Allow Iconization for Undocked WaveViews and Tool Windows (*)
Automatic OK-button Pointer Centering
Suppress WaveView/Panel Delete Confirmation
caution: (*) restart required
Number of Valid Digits : 7
Default Browser Width : 250
Default Printout Title :
Printer Devices :
Unix Print Command :


monitor
meter
Cursor
X:106.4389 Y:35.51587

Analysis Type - OP

- .OP statement prints out the following items in xxx.lis file.
  - Node voltage
  - Source current
  - Power dissipation
  - Device information
  - …..

- Specify time at which operating point is to be calculated
  - .OP at 3us (show the operating point at 3us in transient simulation)

Analysis Type - DC

- Sweep for param/temp/supply voltage…
- Syntax：

```
.DC var1 start1 stop1 incr1 < var2 start2 stop2 incr2 >
.DC var1 start1 stop1 incr1 < SWEEP var2 DEC/OCT/LIN np start2 stop2 >
```

- Example：

```
.DC  VIN  0.25  5.0  0.25
.DC  VDS  0  10  0.5  VGS  0  5  1
.DC  xval  1k  10k  0.5k  SWEEP  TEMP  LIN   5  25  125
```

- Syntax：

```
.PZ  V(OUT)  VIN      **output-variable, input-source
**NOTE: Compute information about pole/zero analysis
```

- Results：

```
******   pole/zero analysis  tnom=  25.000 temp=  25.000
....
     poles (rad/sec)             poles ( hertz)
**************************************************************************
   real          imag           real          imag
  -1.0393x       0.           -165.4096k       0.
 -12.7888g       0.             -2.0354g       0.
.....
     zeros (rad/sec)             zeros ( hertz)
**************************************************************************
   real          imag           real          imag
 -12.7834g       0.             -2.0345g       0.
 -45.4697g    20.0122g          -7.2367g       3.1850g
```

Analysis Type – AC & Transient

- Syntax：

```
.AC  DEC/OCT/LIN  np  fstart  fstop
.AC DEC/OCT/LIN  np  fstart  fstop < SWEEP var start stop incr >
```

- Example：

```
.AC  DEC  10  1K  100MEG
```

Frequency sweep 10 point per decade from 1kHz to 100MHz

- Syntax：

```
.TRAN  tincr1  tstop1 < tincr2  tstop2 ..... > < START=val>
.TRAN  tincr1  tstop1 < tincr2  tstop2 ..... > < START=val> UIC <SWEEP..>
```

- Example：

```
.TRAN   1NS  100NS
.TRAN  10NS  1US  UIC
.TRAN  10NS  1US  UIC  SWEEP  TEMP  -55  75  10  $ step=10
```

Simulation output and controls

- Output commands
  - .PRINT – print numeric analysis results in .lis file
  - .PLOT
  - .PROBE – Allows save output variables only into graphic data files
  - .MEAS – Print numeric results of measured specifications

- Output file type

Output File Type	Extensi
Output Lis	.lis
DC Analysis Results	.sw#
DC Analysis Measurement Results	.ms#
AC Analysis Results	.ac#
AC Analysis Measurement Results	.ma#
Transient Analysis Results	.tr#
Transient Analysis Measurement Results	.mt#
Subcircuit Cross-Listing	.pa#
Operating Point Node Voltages (Initial Condition)	.ic

Output variable Example

- DC and Transient analysis：
  - Nodal Voltage Output : V(1), V(3,4), V(X3.5)
  - Current Output (Voltage Source) : I(VIN), I(X1.VSRC)
  - Current Output (Element Branches) : I2(R1), I1(M1), I4(X1.M3)
- AC analysis：
  - AC : V(2), VI(3), VM(5,7), VDB(OUT), IP(9)

R : Real
I : Imaginary
M : Magnitude
P : Phase
DB : Decibels

- Element templates：(see HSPICE simulation and Analysis user guide)
  - mn1[vth] → LV9(mn1)
  - mn1[gds] → LX8(mn1)
  - mn1[gm] → LX7(mn1)

CGGBO	LX18	CGGBO = ∂Qg/∂Vgb=CGS + CGD + CGB
CGDBO	LX19	CGDBO = ∂Qg/∂Vdb, (for Meyer CGD=-CGDBO)
CGSBO	LX20	CGSBO = ∂Qg/∂Vsb, (for Meyer CGS=-CGSBO)

*Hint for HW1

Table 38 MOSFET

Name	Alias	Description
L	LV1	Channel length (L).
W	LV2	Channel width (W).
AD	LV3	Area of the drain diode (AD).
AS	LV4	Area of the source diode (AS).

*Need another .option DCCAP

Thank you !!