



臺灣大學

# IC Design HW2 Tutorial

Ti-Yu Chen

Advisor: Tzi-Dar Chiueh

Oct. 06, 2022



臺灣大學

# Outline

- Connect to workstations
- Flow chart
- Transistor-level Simulation: HSPICE
- Debug tool: nWave



臺灣大學

# Connect to Workstations

MobaXterm



臺灣大學

# Preparatory Works

- Download MobaXterm

- <http://mobaxterm.mobatek.net/download-home-edition.html>
- MobaXterm Home Edition v20.4 Portable edition

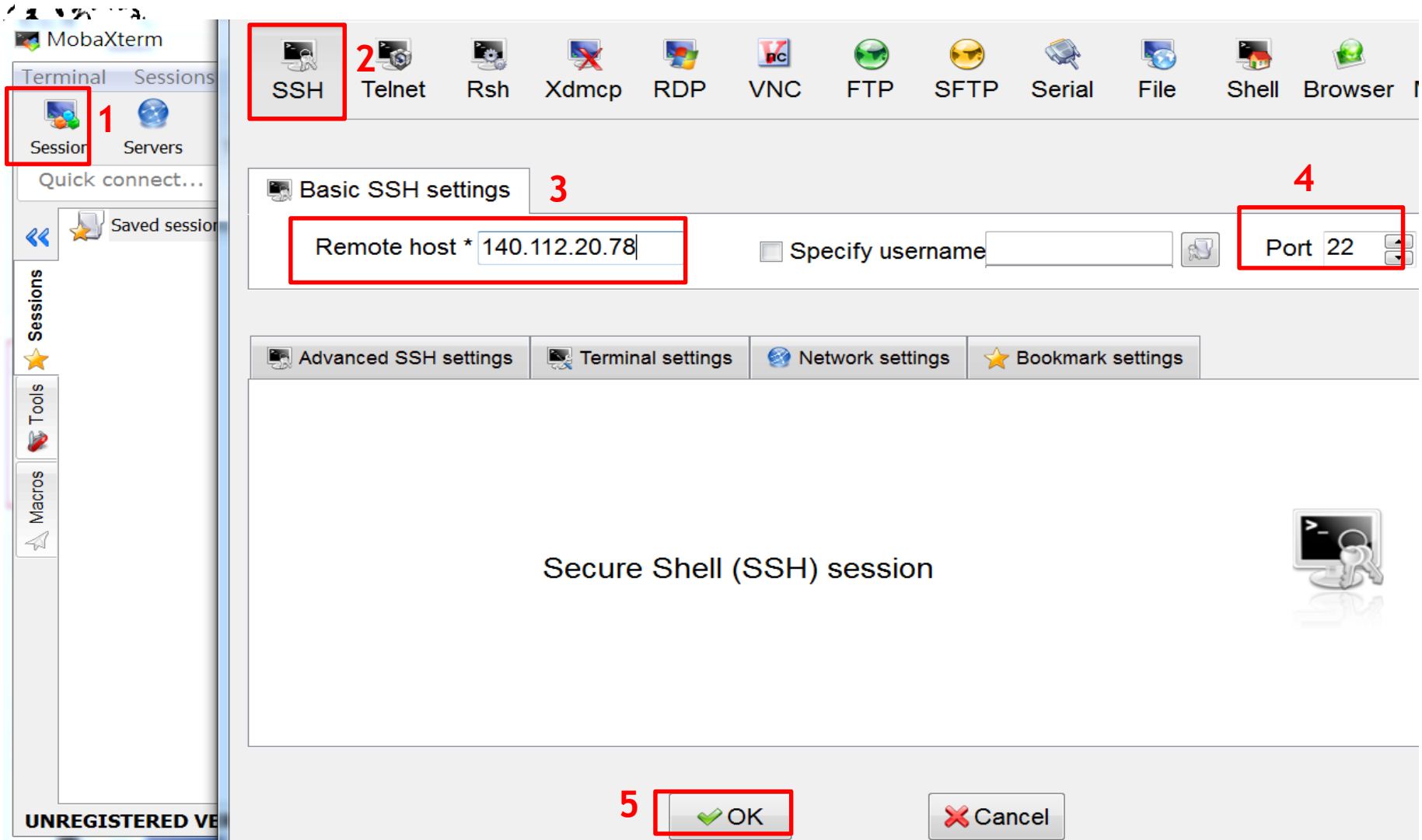
- Choose an Workstation IP

- <http://cad.ee.ntu.edu.tw/wordpress/?p=33>
- Use 140.112.20.72/74/83/84
- You can try it and find appropriate IP



# Connect Workstation

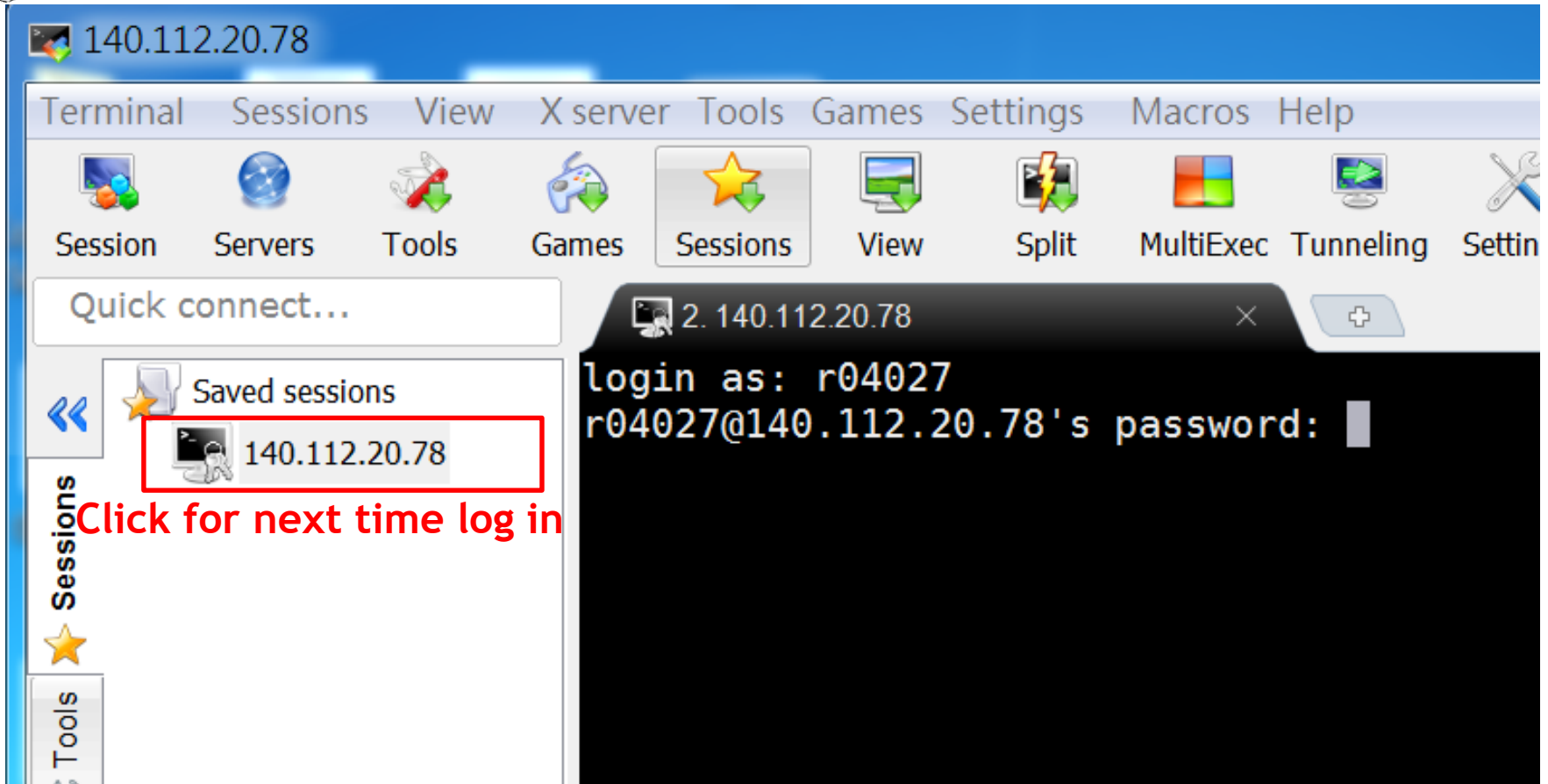
- MobaXterm → Session → SSH → Remote host: IP Port: 22





臺灣大學

# Log-In (1/2)





臺灣大學

# Log-In (2/2)

- User Name

- Ex: For NTUEE(901) and GIEE(943) student, remove 901/943

r04943027 -> r04027

b01901123 -> b01123

- Ex: For other NTU student, remove number right after first English character

b96502040 -> b6502040

- Password

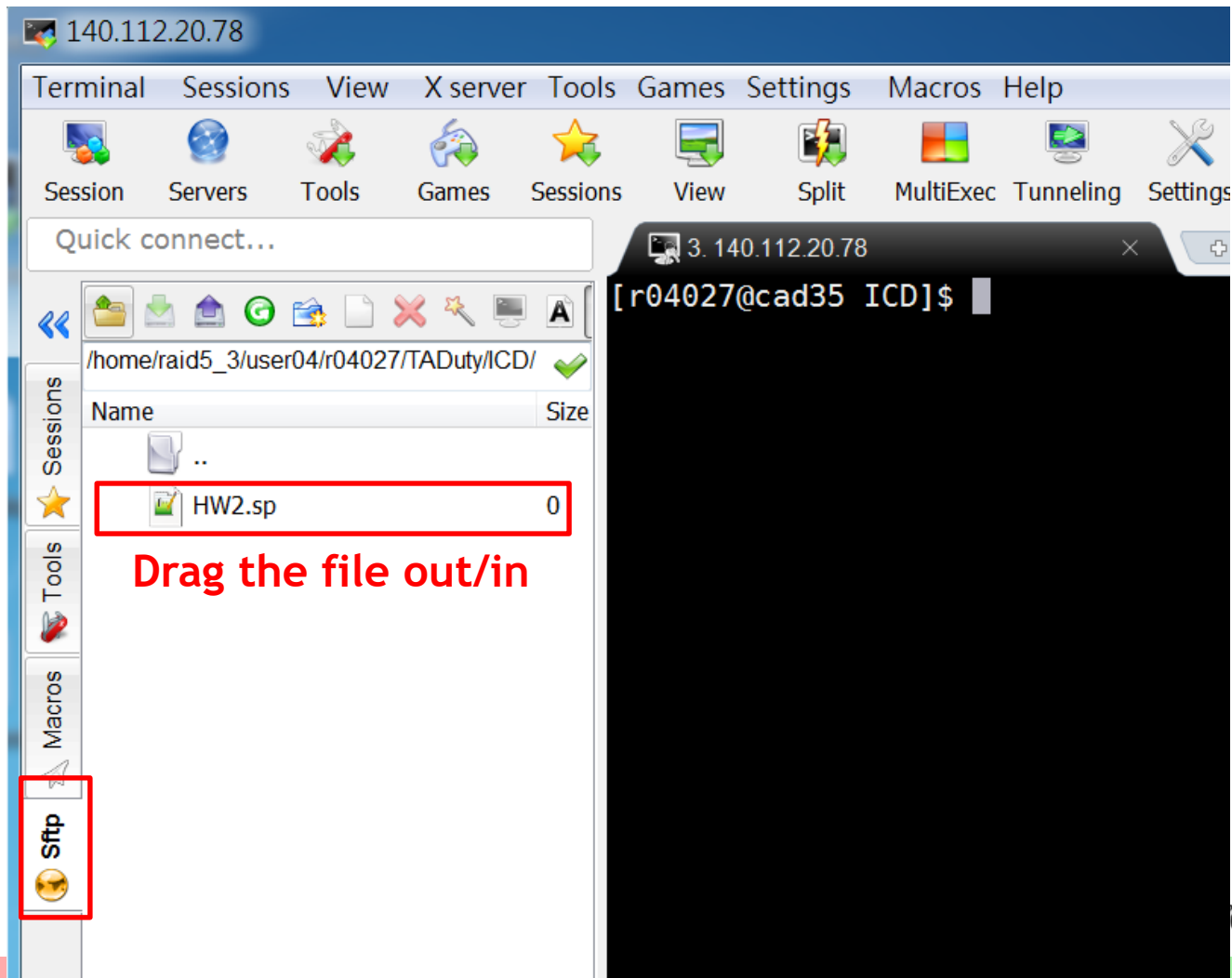
- What you fill in application form



臺灣大學

# Down/Upload File From Workstation

- Just drag the file in/out left hand side file list







臺灣大學

# Simple Linux command

- Source the setting file
  - source
- Document management
  - cd [directory name/..] : Move to other directory
  - cd .. : Move to upper directory
  - ls [-a/-l] : List all files in current directory
  - mkdir [directory name] : Create new directory
  - cp [source] [destination] : Copy file
  - rm [file/directory] : Remove file
  - mv [source] [destination] : Rename/Move file
- Text Editor
  - vim
- More in “鳥哥的Linux 私房菜” <http://linux.vbird.org/>



臺灣大學

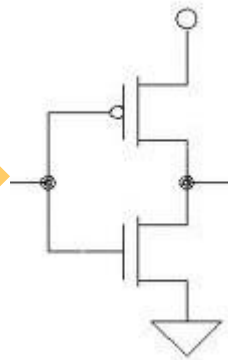
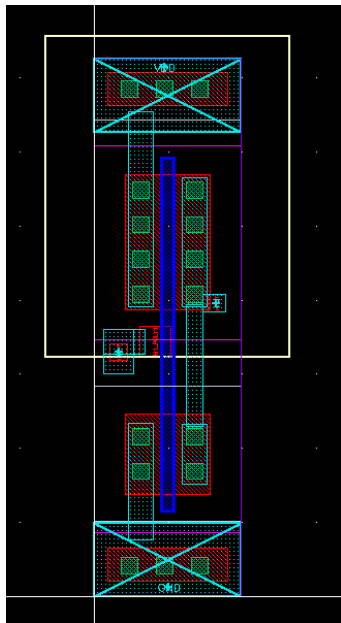
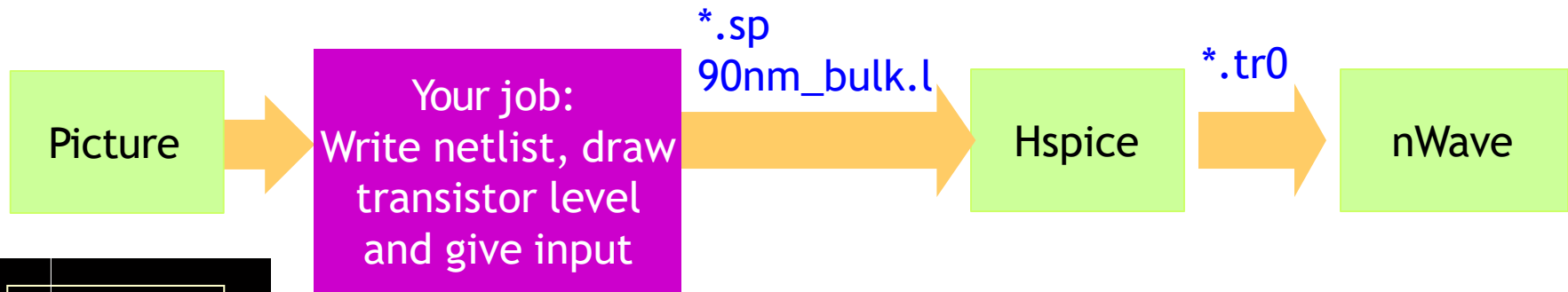
# Source the needed files

- Need to do it every time when you log in
- Cadence tool
  - `source /usr/cadence/cshrc`
- HSPICE
  - `source /usr/cad/synopsys/CIC/hspice.cshrc`
- nWave
  - `source /usr/spring_soft/CIC/verdi.cshrc`

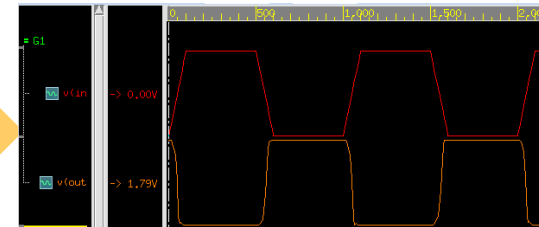


臺灣大學

# Flow Chart



```
.SUBCKT Inv DVDD GND In Out
MM1 Out In GND GND NMOS l=0.1u w=0.25u m=1
MM2 Out In DVDD DVDD PMOS l=0.1u w=0.5u m=1
.ENDS
```

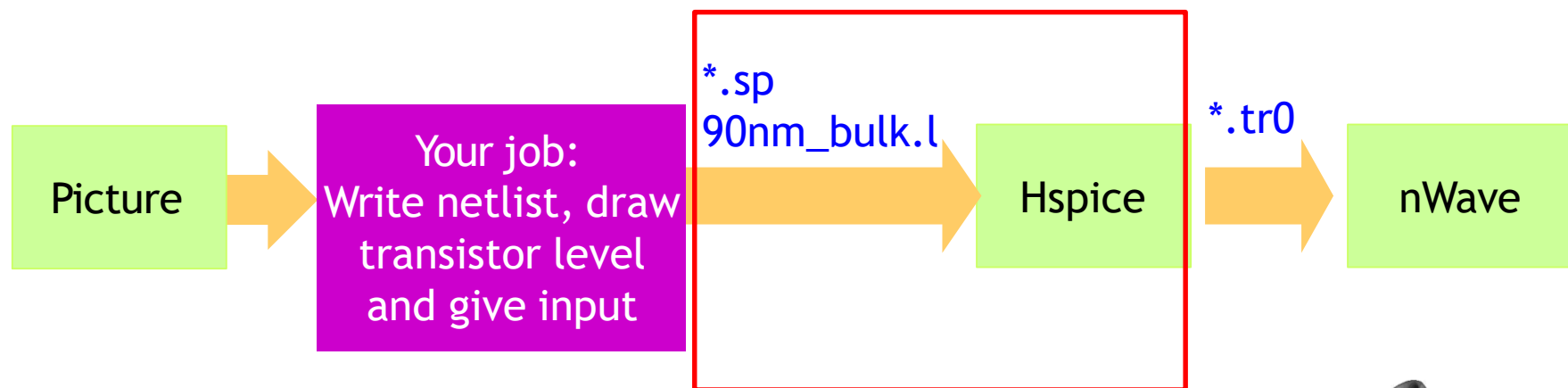




臺灣大學

# HSPICE

## Transistor-level Simulation





# Hspice Syntax(1/7)

- Create an Hspice file “\*.sp”
- Edit with text editors such as WordPad or Notepad++
- First line must be a comment line or be left blank.
- Comment start with \*
- Remember to .inc '90nm\_bulk.l'
- Case **insensitive**
- “0” means ground
- Transistor name must start with “M”
- Sub-circuit name must start with “x”



臺灣大學

# Hspice Syntax (2/7)

- Hspice codes compose of three parts:

(1)include lib file (2)define sub-circuit (3)input signal

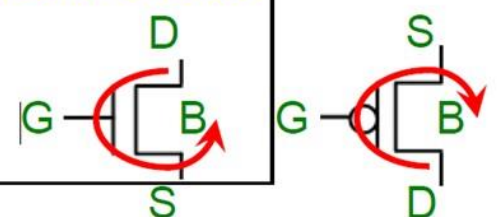
```
*****
.inc '90nm_bulk.l'
.SUBCKT Inv DVDD GND In Out
*.PININFO DVDD:I GND:I In:I Out:O
MM1 Out In GND GND NOMS l=0.1u w=0.25u m=1
MM0 Out In DVDD DVDD PMOS l=0.1u w=0.5u m=1
.ENDS D G S B Type L W
*****

Vdd DVDD 0 1.8
Vss GND 0 0

Vin In 0 pulse(0 1.8 0 100n 100n 0.4u 1u)
x1 DVDD GND In Out Inv
.tran 10n 1.1u
.op
.option post
.end
```

Annotations:

- (1)source library (points to `.inc '90nm_bulk.l'`)
- (2)transistors (points to the MOSFET model definitions)
- (3)Include a sub-circuit (points to `x1`)
- (3)Input a square wave (points to the `pulse` function)
- (3)scan transient (points to `.tran`)





# Hspice Syntax (3/7)

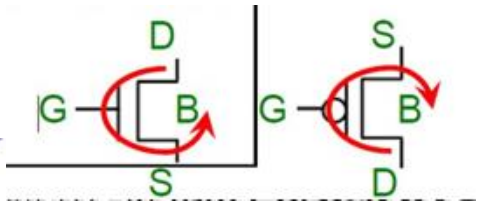
臺灣大學

- Define sub-circuit (Transistor name must start with “M”)
- Start with .SUBCKT, end with .ENDS

Sub-circuit name    Port name(Input first, Output last)

```
.SUBCKT  Inv  DVDD  GND  In  Out
*.PININFO DVDD:I  GND:I  In:I  Out:O
MM1 Out  In  GND  GND  NOMS  l=0.1u  w=0.25u  m=1
MM0 Out  In  DVDD DVDD PMOS  l=0.1u  w=0.5u  m=1
.ENDS  D  G  S  B  Type  L  W
```

Transistor name



- Set VDD GND (DVDD and GND are nodes' name)

Node name    ground

Vdd	DVDD	0	1.8
Vss	GND	0	0

Voltage

- Call sub-circuit (Sub-circuit name must start with “x”)

sub-circuit port(Same order as defined)

```
x1  DVDD  GND  In  Out  Inv
```

sub-circuit name                      sub-circuit type



# Hspice Syntax(4/7)

臺灣大學

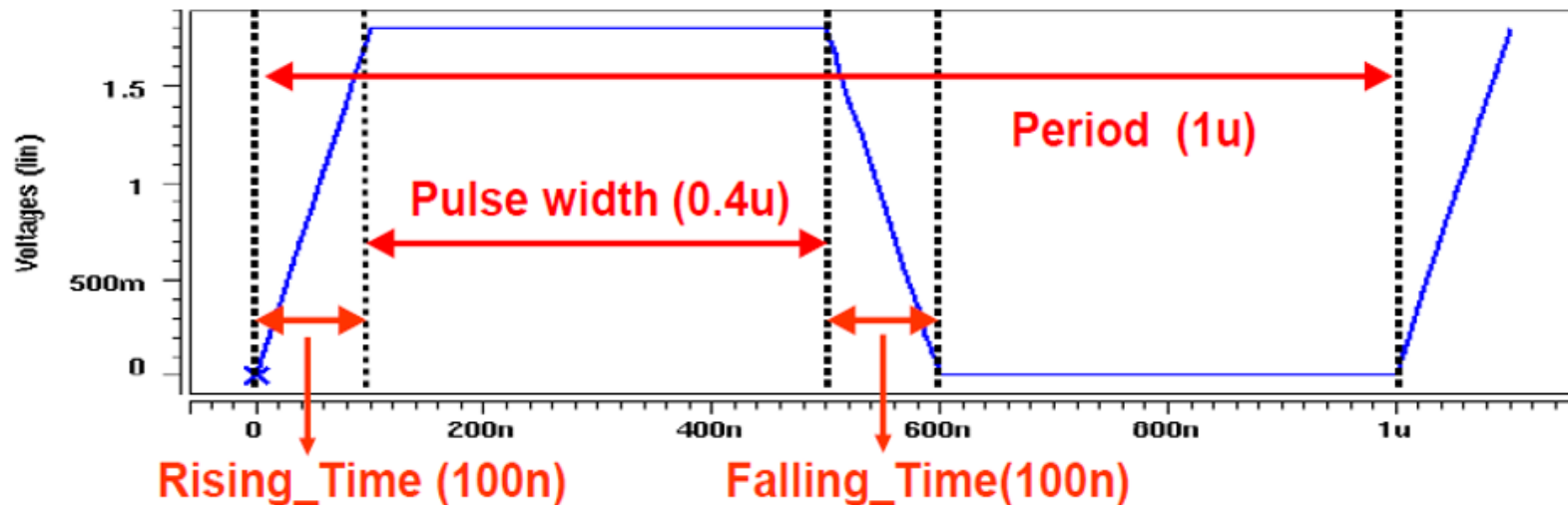
- Model input AC signal: pulse

```
Vin In 0 pulse(0 1.8 0 100n 100n 0.4u 1u)
```

voltage\_name node1 node2 pulse( GND VDD delay\_time  
rising\_time falling\_time pulse\_width period)

Example : Vin In 0 pulse(0 1.8 0 100n 100n 0.4u 1u)

Node name







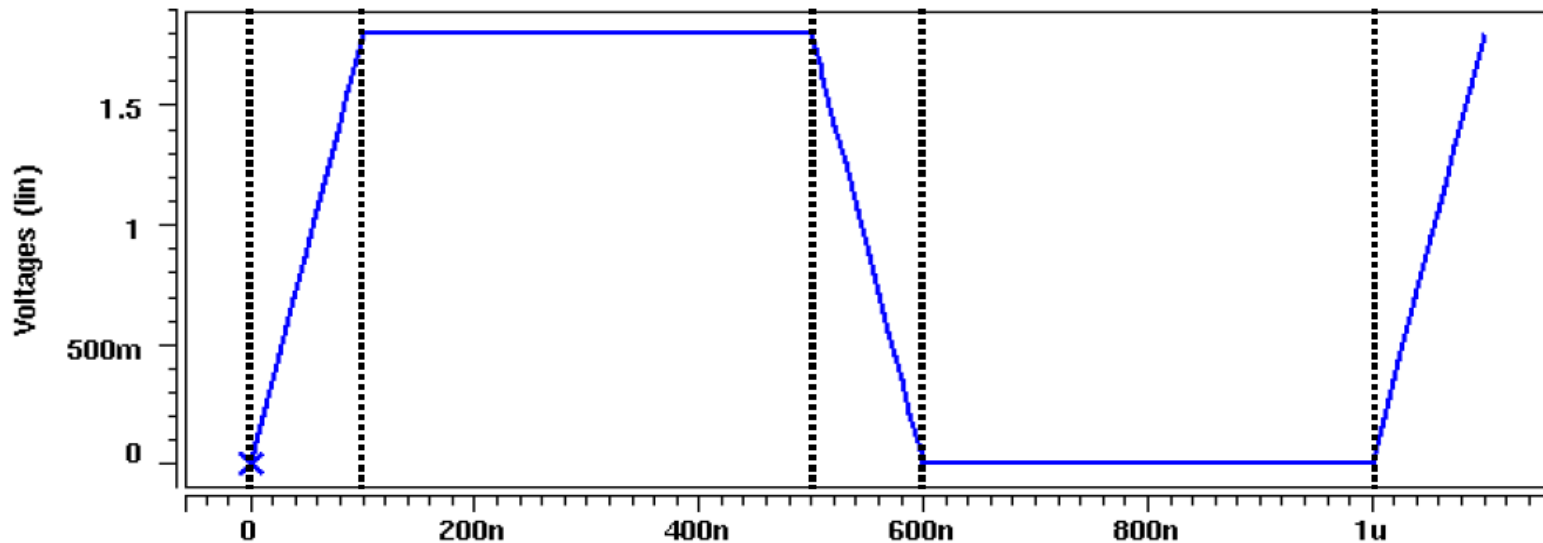
臺灣大學

# Hspice Syntax (5/7)

- Another signal waveform: pwl (recommended)

PWL( {time1} {v1} {time2} {v2} ... {time3} {v3} )

Example : Vin In 0 pwl(0n 0v 100n 1.8v 500n 1.8v 600n 0v ...)





臺灣大學

# Hspice Syntax(6/7)

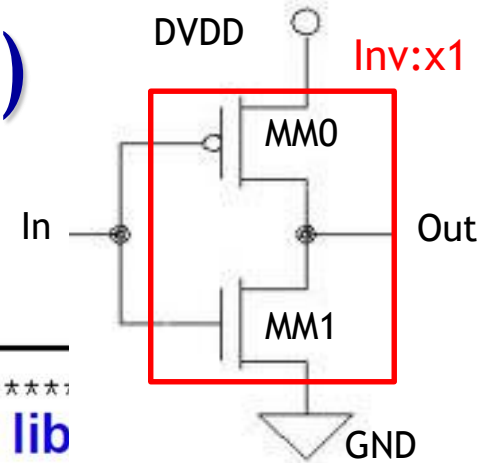
- Define DC voltage source: Vvdd(name) vdd(vddport) gnd 1.8V
- .tran 多久取樣一次 總共模擬多久，可改動，Ex .tran 10n 1u
- .op (計算操作點電壓 operation point，基本上都要加)
- .option post (轉出檔案(ex: .tr0 file...)給scope用，基本上都要加)
- .end



臺灣大學

# Hspice Syntax(7/7)

- It an inverter circuit



\*\*\*\*\*;

.inc '90nm\_bulk.l'

← (1)source lib

.SUBCKT Inv DVDD GND In Out

\*.PININFO DVDD:I GND:I In:I Out:O

MM1 Out In GND GND NOMS |l=0.1u w=0.25u m=1

MM0 Out In DVDD DVDD PMOS |l=0.1u w=0.5u m=1

← (2)transist

.ENDS **D G S B Type L W**

\*\*\*\*\*

Vdd DVDD 0 1.8

Vss GND 0 0

Vin In 0 pulse(0 1.8 0 100n 100n 0.4u 1u)

← (3)Input a square wave

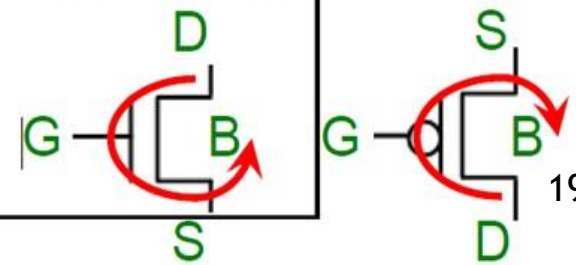
x1 DVDD GND In Out Inv ← (2)Include a sub-circuit

.tran 10n 1.1u ← (3)scan transient

.op

.option post

.end





臺灣大學

# Hspice Simulation (1/2)

- SPICE is generally a circuit analysis tool for simulation of electrical circuits in steady-state, transient, and frequency domains
- Source your environment setting file
  - `source /usr/cad/synopsys/CIC/hspice.cshrc`
- Upload .sp file and 90nm\_bulk.l to workstation by dragging into MobaXterm



臺灣大學

# Hspice Simulation (2/2)

- Save Hspice file and run simulation
  - hspice [hspice file] , Ex. hspice hw2.sp

- Error

```
>info: ***** job aborted
***** hspice job aborted
lic: Release hspice token(s)
```

- Successful

```
>info: ***** hspice job concluded
lic: Release hspice token(s)
real 0.56
```

- Output wave file(.tr0)

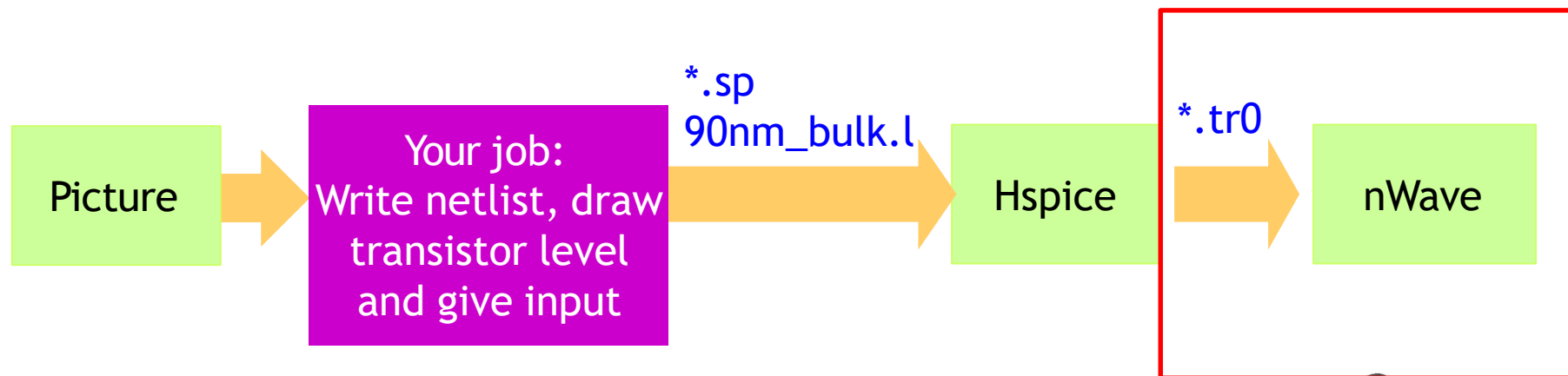
inv_hspice.ic0	1
inv_hspice.pa0	0
inv_hspice.sp	0
inv_hspice.st0	2
inv_hspice.tr0	40



臺灣大學

# Debug tool

## nWave

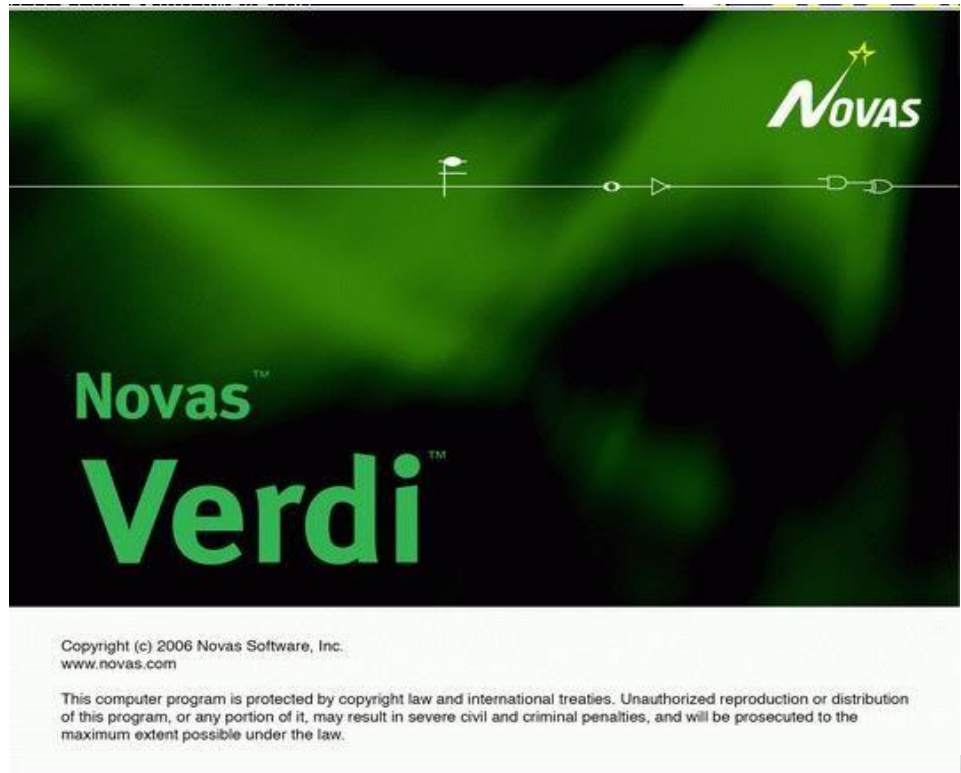




臺灣大學

# nWave: Source File and Execute

- Source
  - source /usr/spring\_soft/CIC/verdi.cshrc
- Execute nWave
  - nWave &

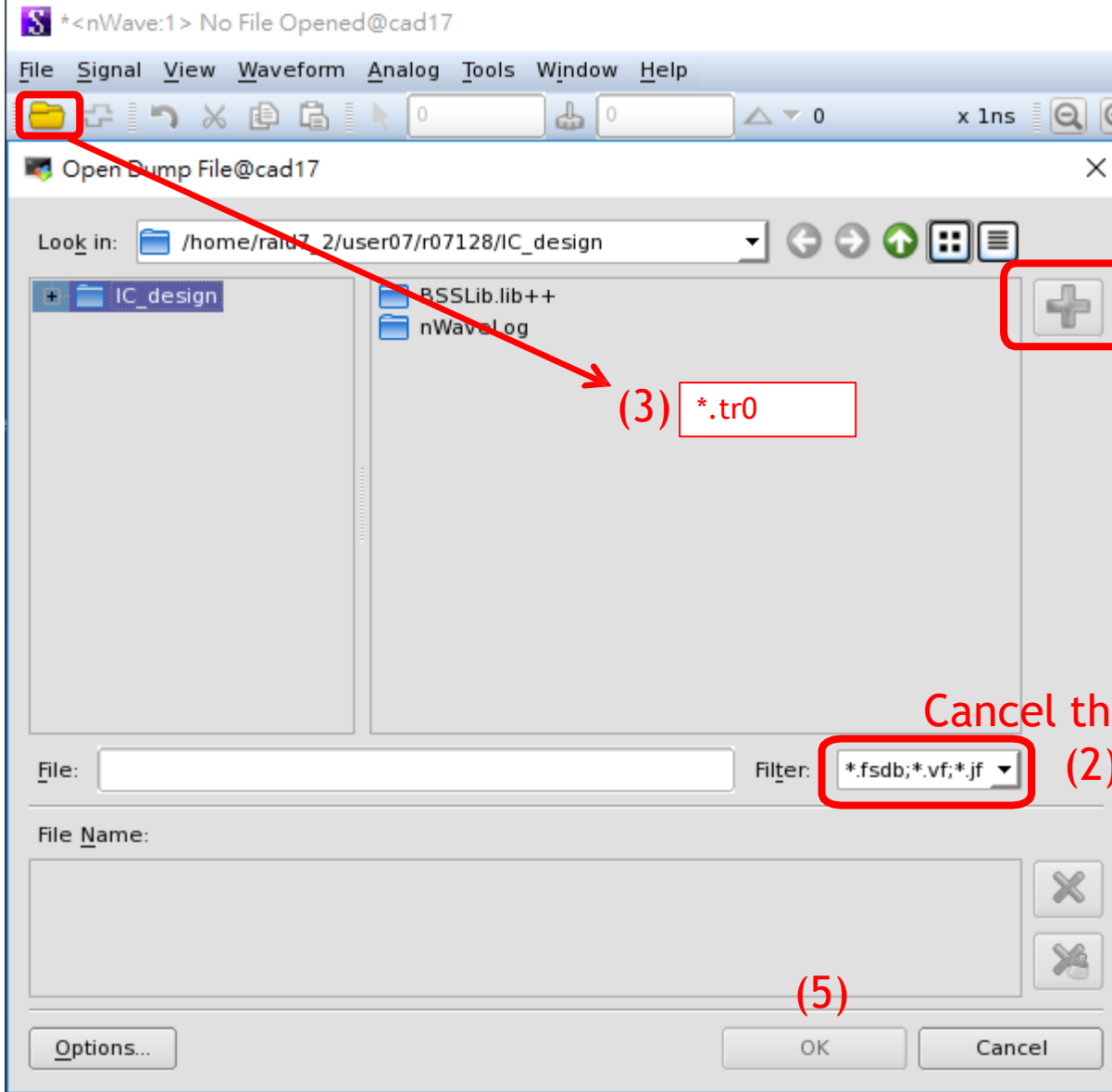




臺灣大學

# Select Output File Generated by Hspice

(1)





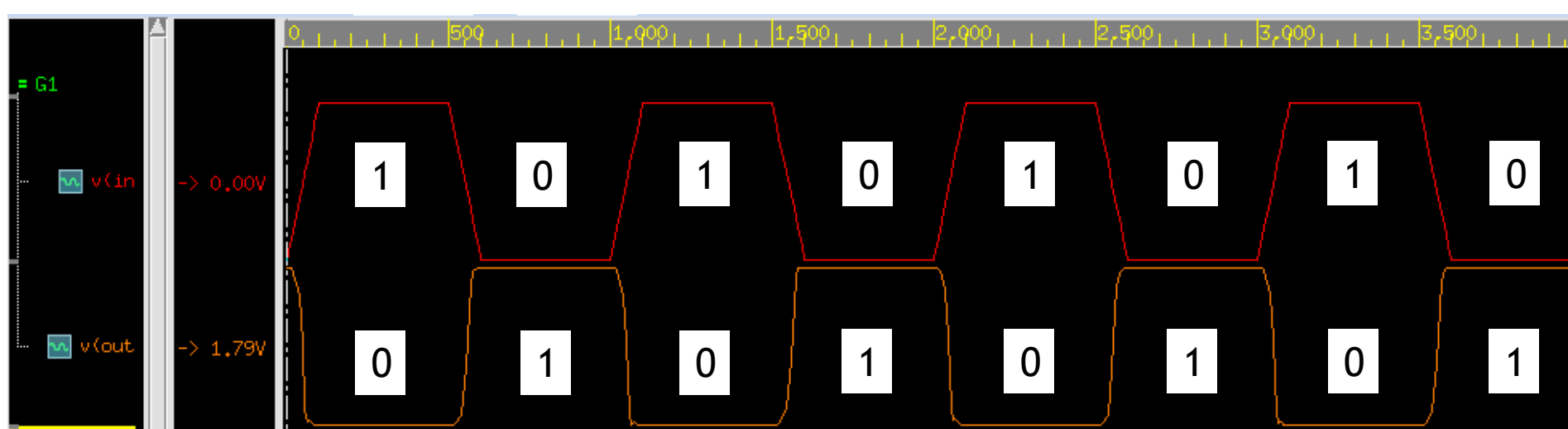




臺灣大學

# Check Waveform

- Remember to check all possible input
- Record the waveform into your report
- Ex: Inverter





臺灣大學

# Reminder

- Be patient and careful about each step!
- References
  - [1] “SPICE,” CIC handout, 2001
  - [2] “鳥哥的Linux 私房菜” <http://linux.vbird.org/>
- If there’s any workstation account/password problem, please directly contact workstation administrator
  - 邱茂菱，d01943010@ntu.edu.tw
- If you have any questions, please contact TA
  - 陳帝宇，EE2-329，tp62u4m3@gmail.com



臺灣大學

Thanks for your attention!

Q & A