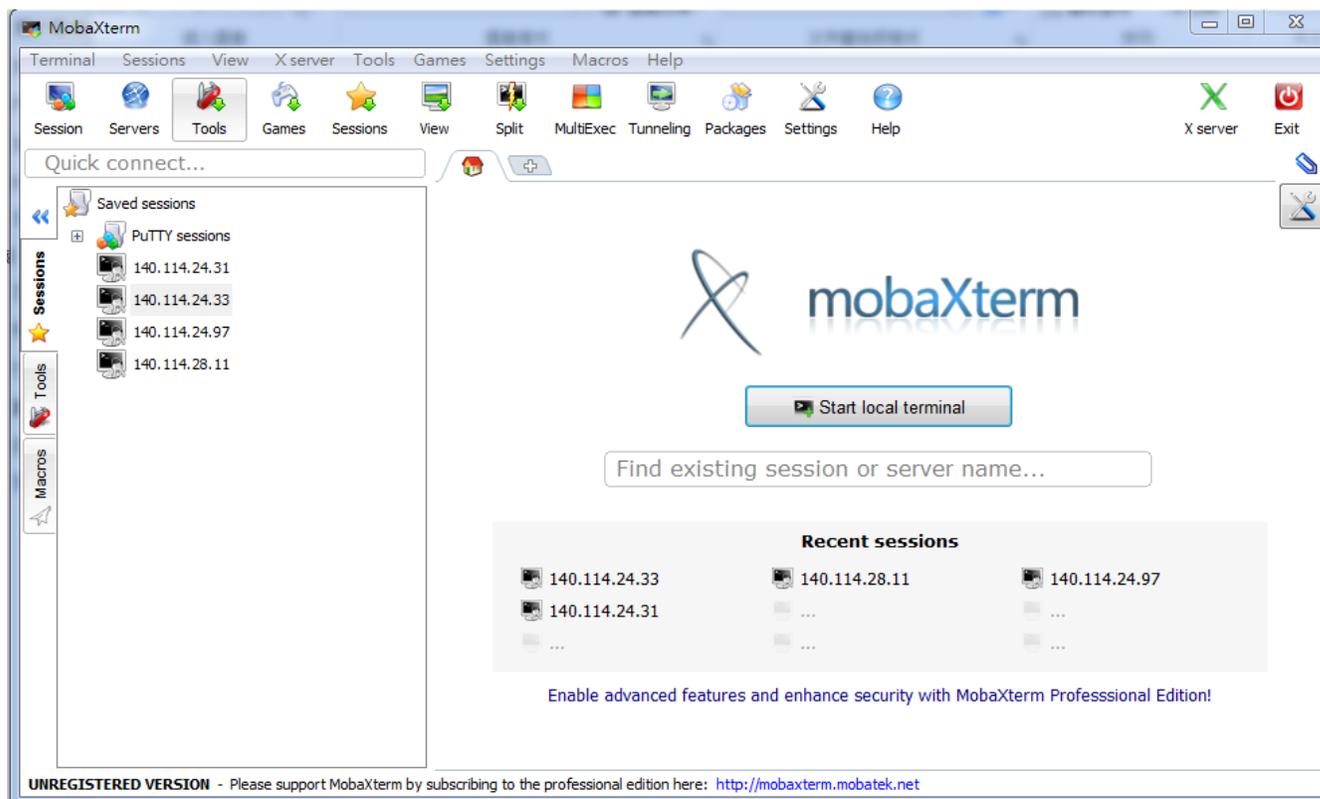


# Workstation Tutorial

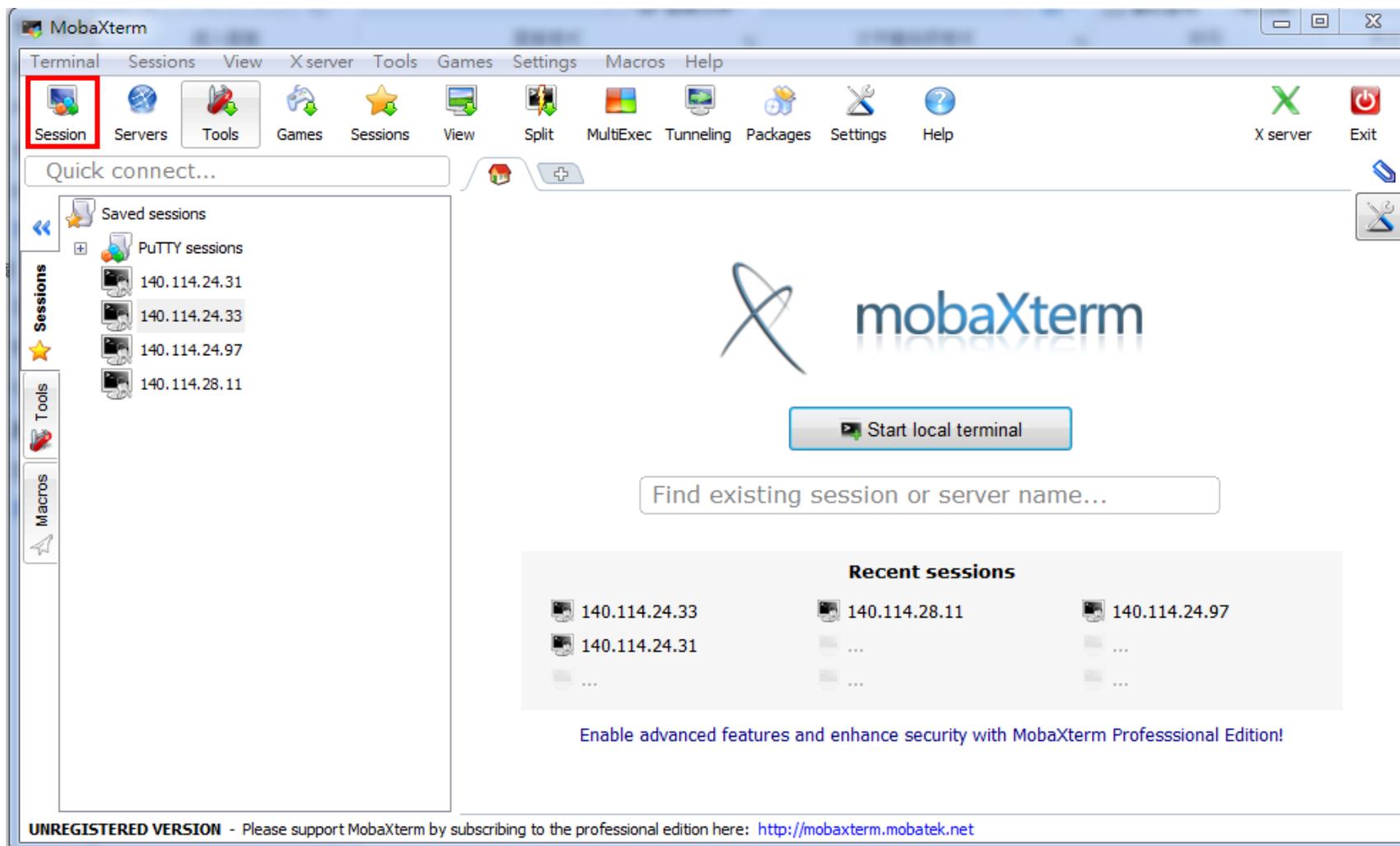
# 遠端連線程式

- 先執行遠端連線程式(putty, MobaXterm,... 這裡用MobaXterm為範例)

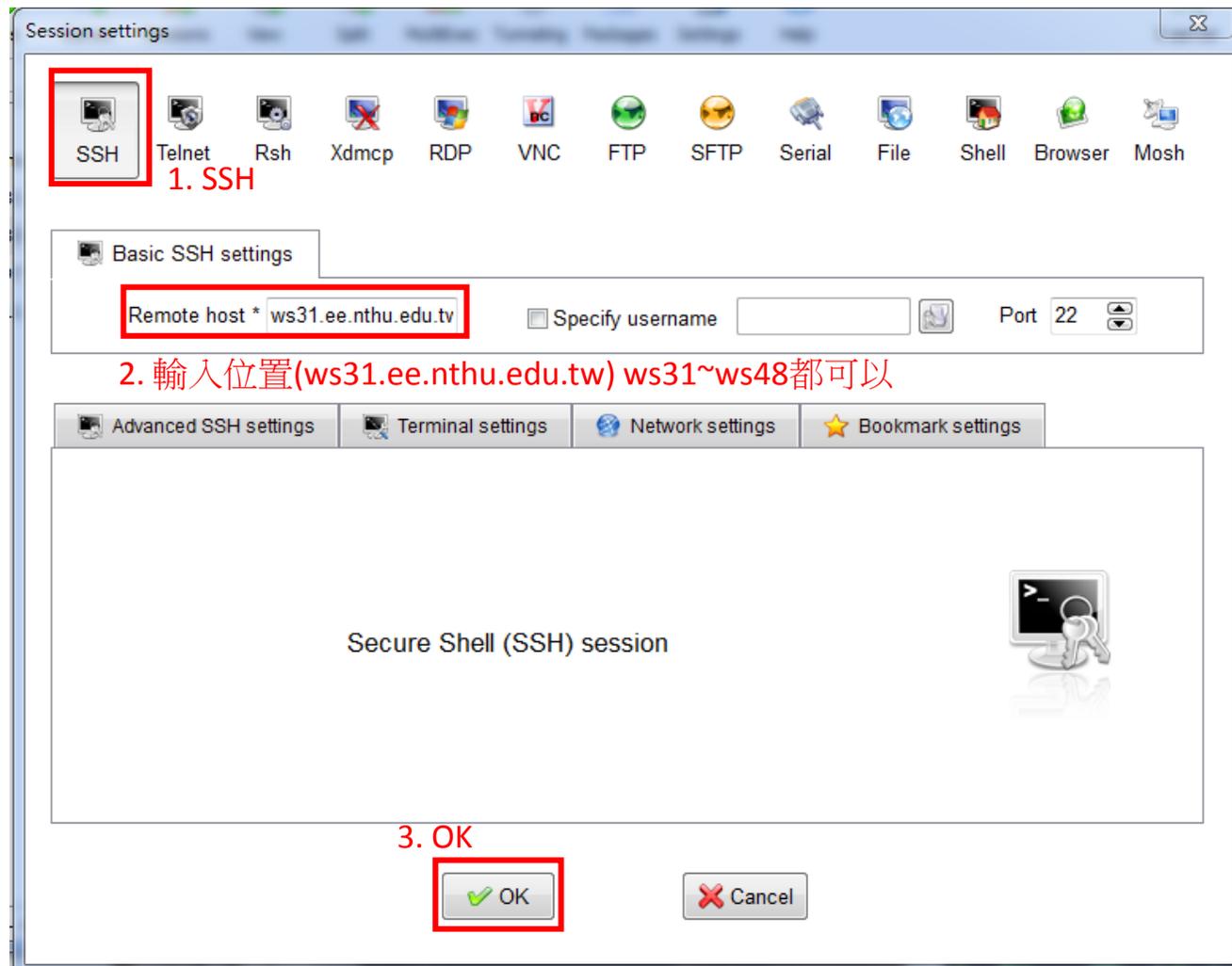


# 遠端連線程式

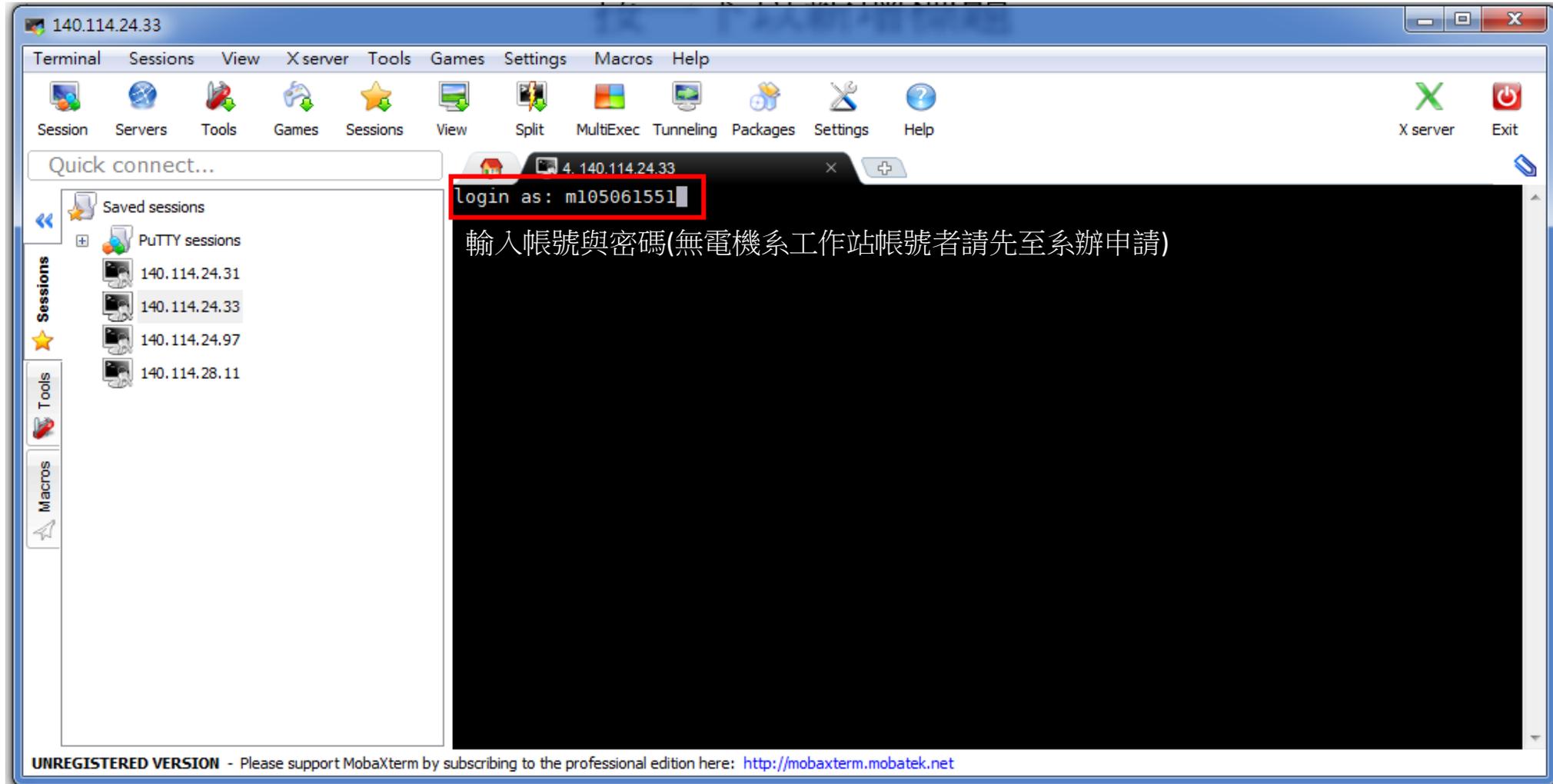
Session



# 遠端連線程式



# Login Workstation



# Simulation Environment(Directory)

1. mkdir vlsi(創造資料夾)

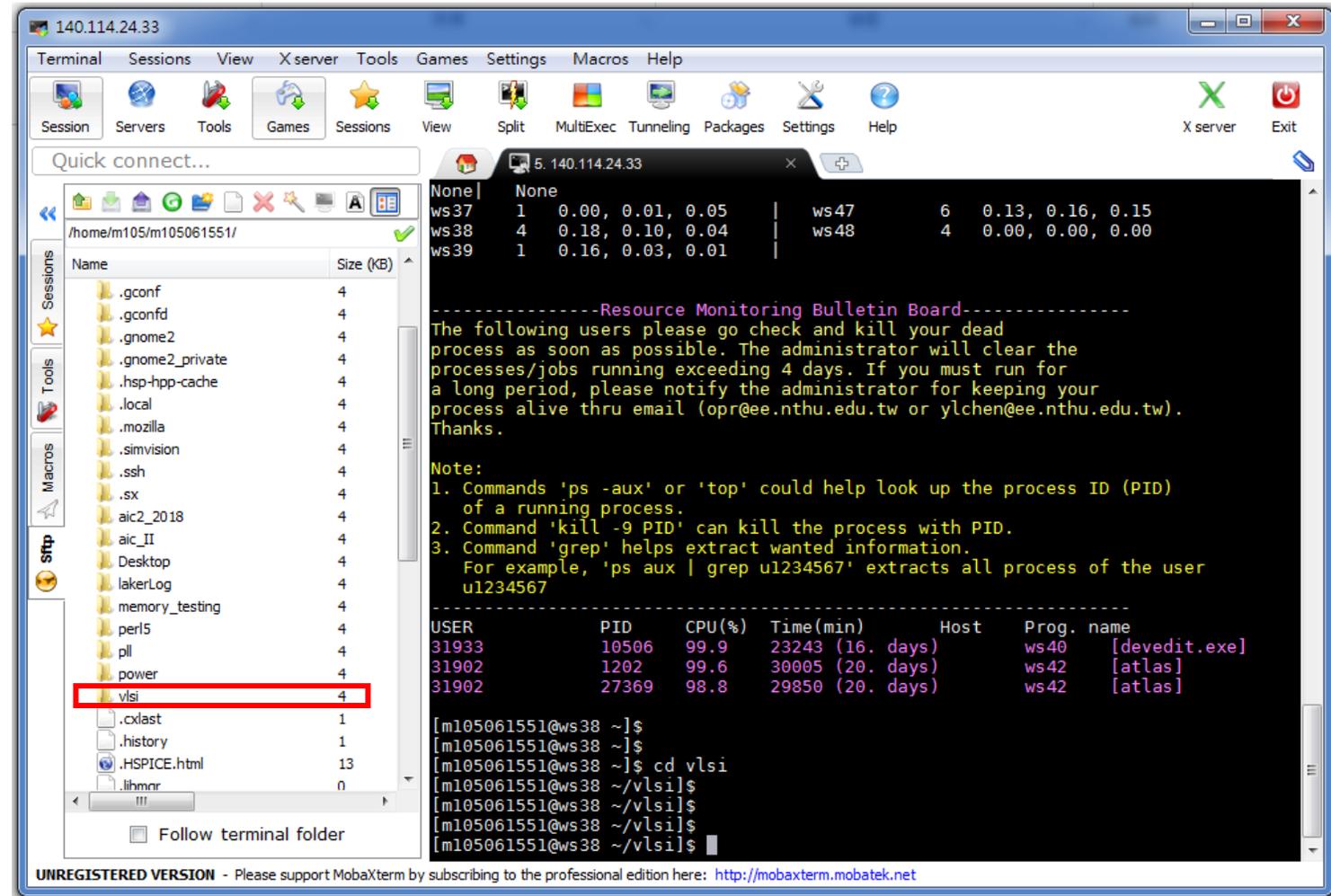
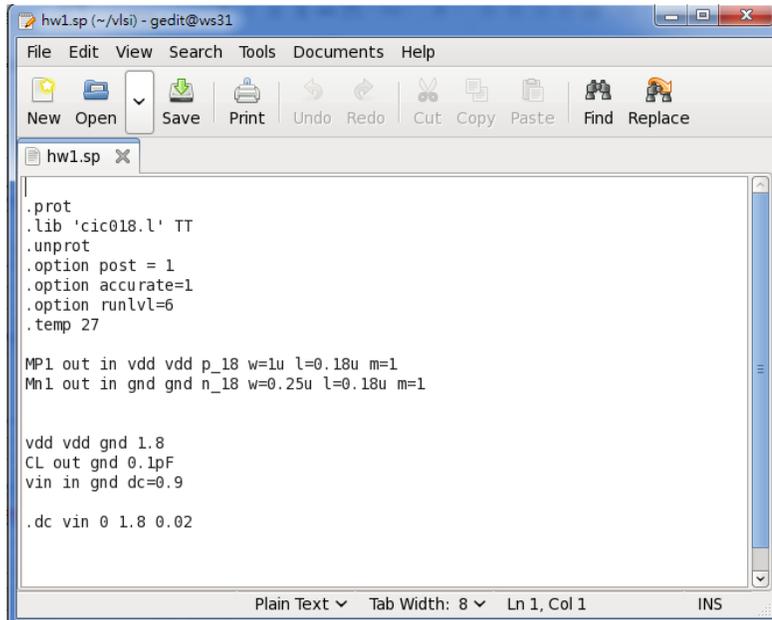
```
[m105061551@ws31 ~]$ mkdir vlsi
```

2. cd xxx(進入xxx資料夾) ※cd .. 為回上一層

```
[m105061551@ws31 ~]$ cd vlsi  
[m105061551@ws31 ~/vlsi]$
```

3. gedit hw1.sp & (可編輯名為xxx.sp的檔案)

```
[m105061551@ws31 ~/vlsi]$ gedit hw1.sp &  
[1] 6566
```



從這邊也可以看到你的資料夾

# Prepare for Simulation

※記得將製程檔案上傳到同一個資料夾，才能跑模擬

The screenshot shows the MobaXterm interface with a terminal window and a file manager pane. The file manager pane shows a directory listing for `/home/m105/m105061551/vlsi/`. A file named `circ018.l` with a size of 90 KB is highlighted with a red box. A red label `上傳檔案` (Upload File) is positioned above the file manager pane. The terminal window shows the following commands and output:

```
USER          PID    CPU(%)  Time(min)   Host    Prog. name
31933         10506   99.9    23303 (16. days)  ws40    [devedit.
exe]
31902         1202    99.6    30065 (20. days)  ws42    [atlas]
31902         27369   98.8    29910 (20. days)  ws42    [atlas]

[m105061551@ws31 ~]$
[m105061551@ws31 ~]$
[m105061551@ws31 ~]$ cd vlsi
[m105061551@ws31 ~/vlsi]$
[m105061551@ws31 ~/vlsi]$
[m105061551@ws31 ~/vlsi]$ gedit hw1.sp &
[1] 5767
[m105061551@ws31 ~/vlsi]$
[m105061551@ws31 ~/vlsi]$ gedit test.sp &
[2] 5786
[1] Done
    gedit hw1.sp
[m105061551@ws31 ~/vlsi]$
[m105061551@ws31 ~/vlsi]$ cd ..
[2] Done
    gedit test.sp
[m105061551@ws31 ~]$ cd vlsi
[m105061551@ws31 ~/vlsi]$ gedit hw1.sp &
[1] 6566
[m105061551@ws31 ~/vlsi]$
```

At the bottom of the terminal window, there is a notice: `UNREGISTERED VERSION - Please support MobaXterm by subscribing to the professional edition here: http://mobaxterm.mobatek.net`

# Example for .sp file

```
#####Simulation Setup#####  
.prot  
.lib 'cic018.l' TT  
.unprot  
.option post = 1  
.option accurate=1  
.option runlvl=6  
.temp 27  
  
#####Circuit or Sub-circuit Description#####  
M1 out in vdd vdd p_18 w=1u l=0.18u m=1  
Mn1 out in gnd gnd n_18 w=0.25u l=0.18u m=1  
  
#####Voltage or Current Source Description#####  
vdd vdd gnd 1.8  
CL out gnd 0.1pF  
vin in gnd dc=0.9  
  
#####Simulation to Run#####  
.op  
.dc vin 0 1.8 0.02  
  
.end
```

描述製程檔案和模擬設定

電路(子電路)描述，以一個inverter為例

電壓/電流源描述

模擬類型和設定

Plain Text Tab Width: 8 Ln 24, Col 19 INS

# Run Hspice

1. 輯完.sp檔的內容後，可以執行模擬
2. Hspice -i hw1.sp -o hw1.lis (-i為input file, -o為simulation info output file)

```
[m105061551@ws31 ~/vlsi]$ hspice -i hw1.sp -o hw1.lis
```

3. 若結果顯示hspice job aborted, 則模擬未完成，請查看hw1.lis檔案的內容，裡面會顯示error information，請將錯誤更正

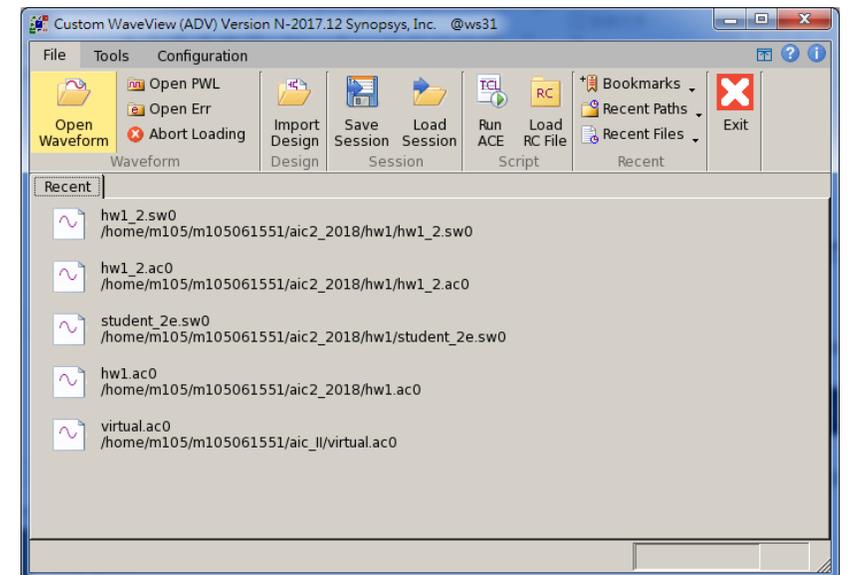
```
>info:          ***** hspice job aborted  
[m105061551@ws31 ~/vlsi]$
```

4. 若顯示hspice job concluded, 表示模擬正確的執行完成，可以看hw1.lis來查看部分結果

```
>info:          ***** hspice job concluded
```

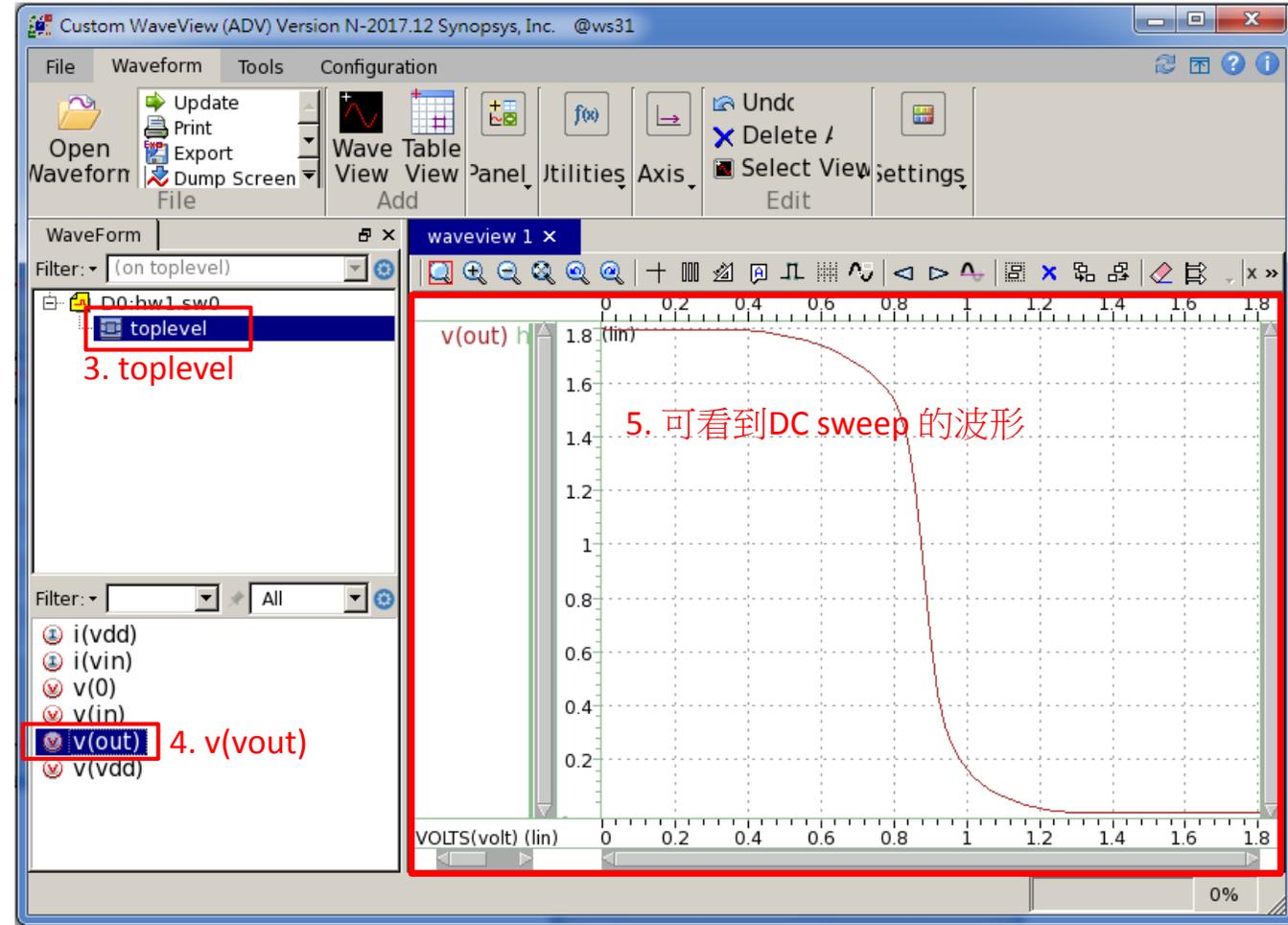
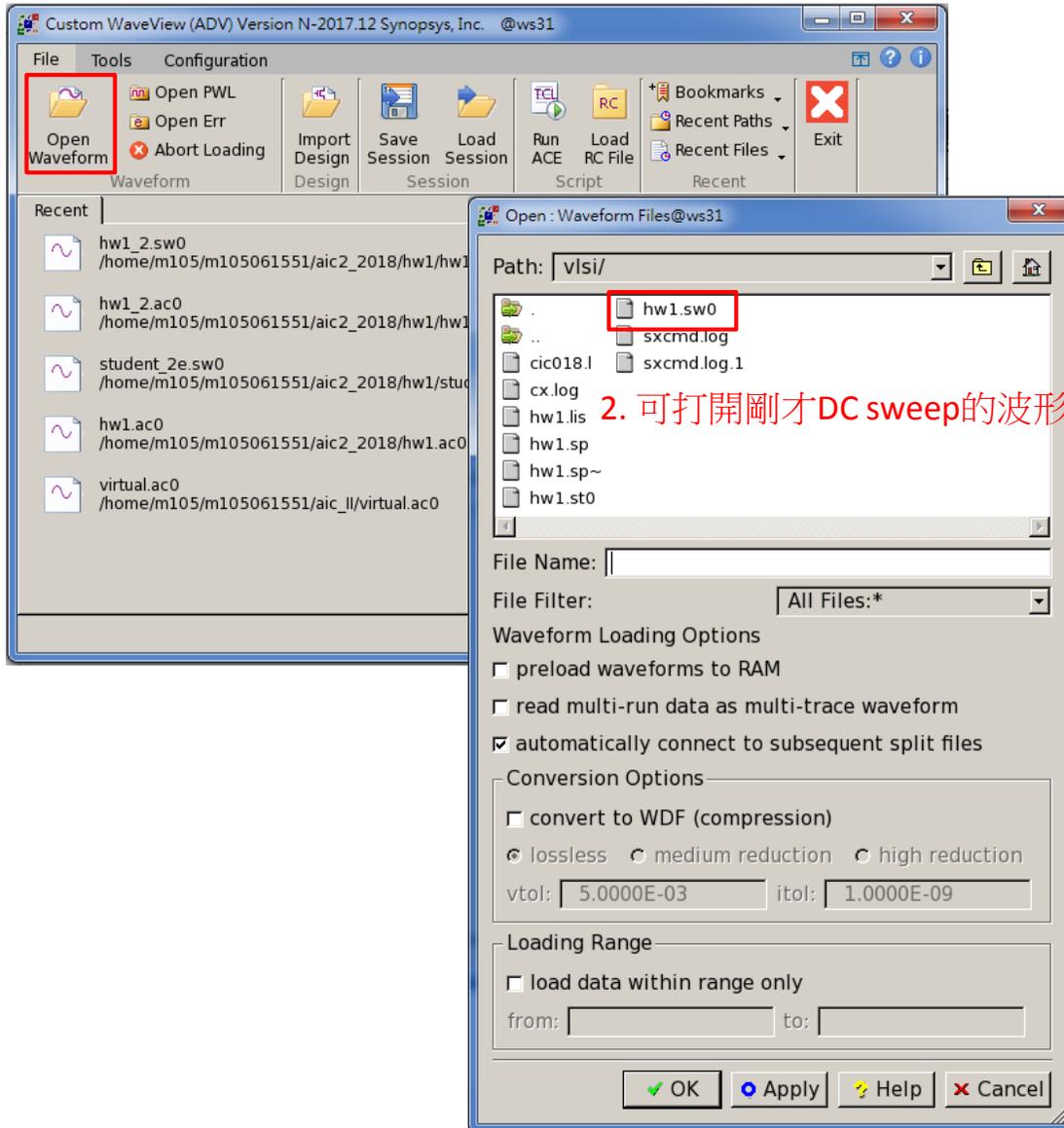
5. 或用cx & 打開waveview來看波形

```
[m105061551@ws31 ~/vlsi]$ cx &  
[4] 8260  
[m105061551@ws31 ~/vlsi]$  
[m105061551@ws31 ~/vlsi]$ 'spice_explorer' license is checked out for Custom WaveView ADV feature.
```



# Waveform Check

## 1. Open Waveform



Thank you