

EE8074-Fall 2023

記憶體積體電路

Final Project N16 SRAM Layout:
Tutorials

Prof. E Ray Hsieh 謝易叡 副教授

Full-Custom IC Design Using Virtuoso

Professor: E Ray Hsieh

Date: 2023/12/25

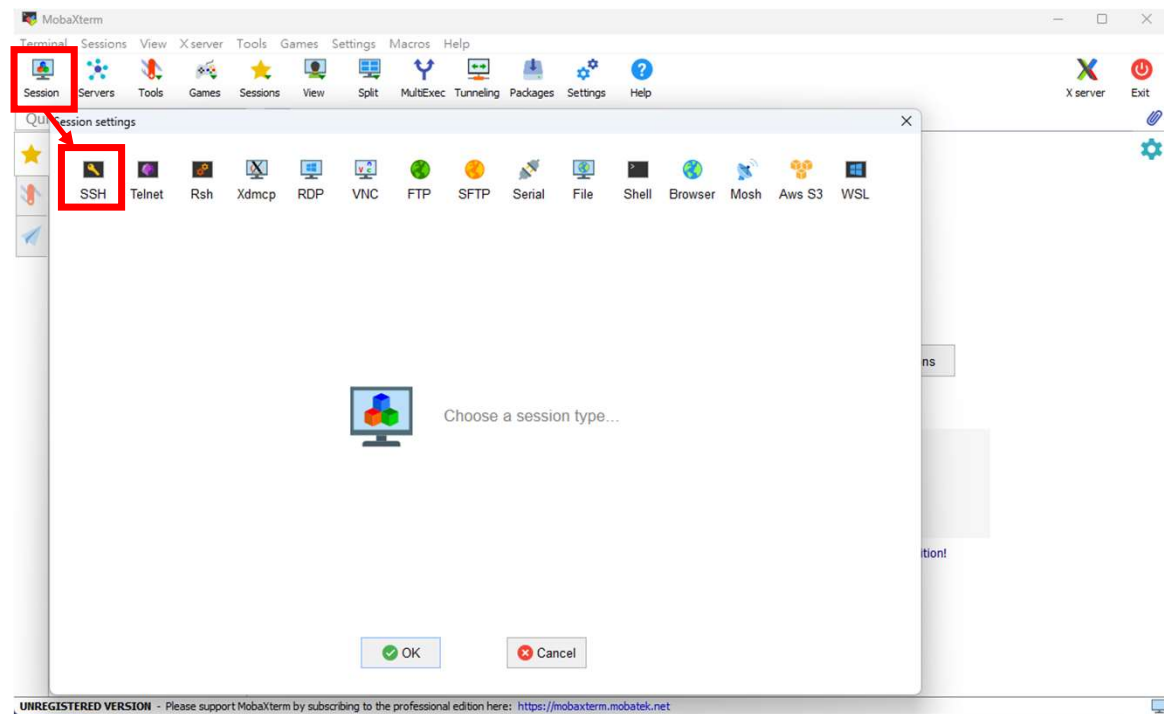
Outline

- Virtuoso Workstation Setup
- Environment Setup
- Draw Schematic using Virtuoso

Virtuoso Workstation Setup

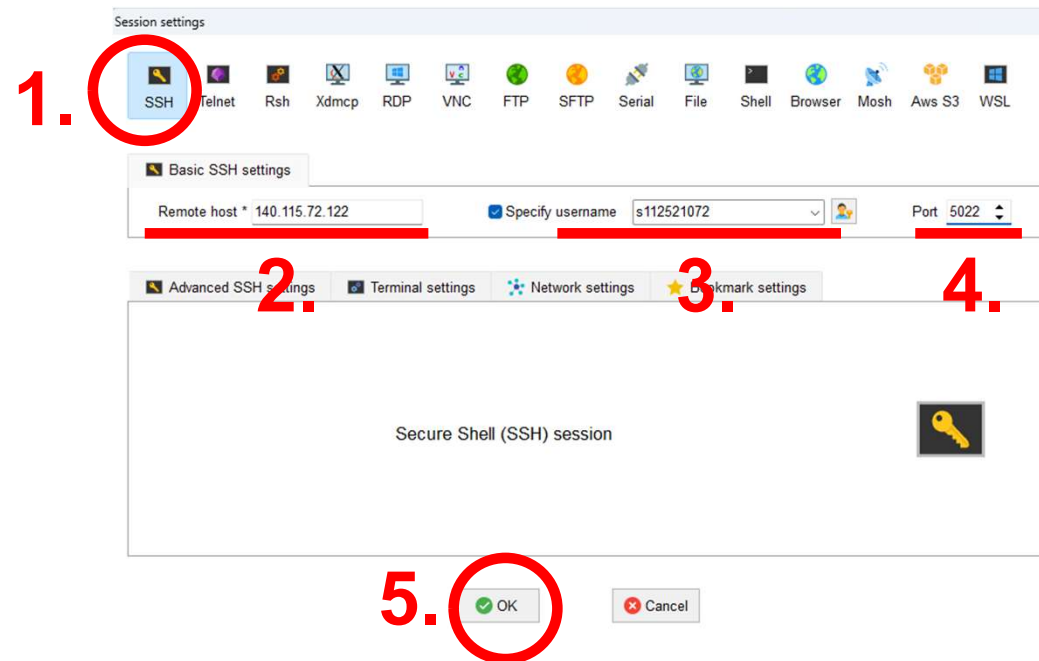
Login Server (1/5)

- Download MobaXterm 
- Choose “Session” -> “SSH”term



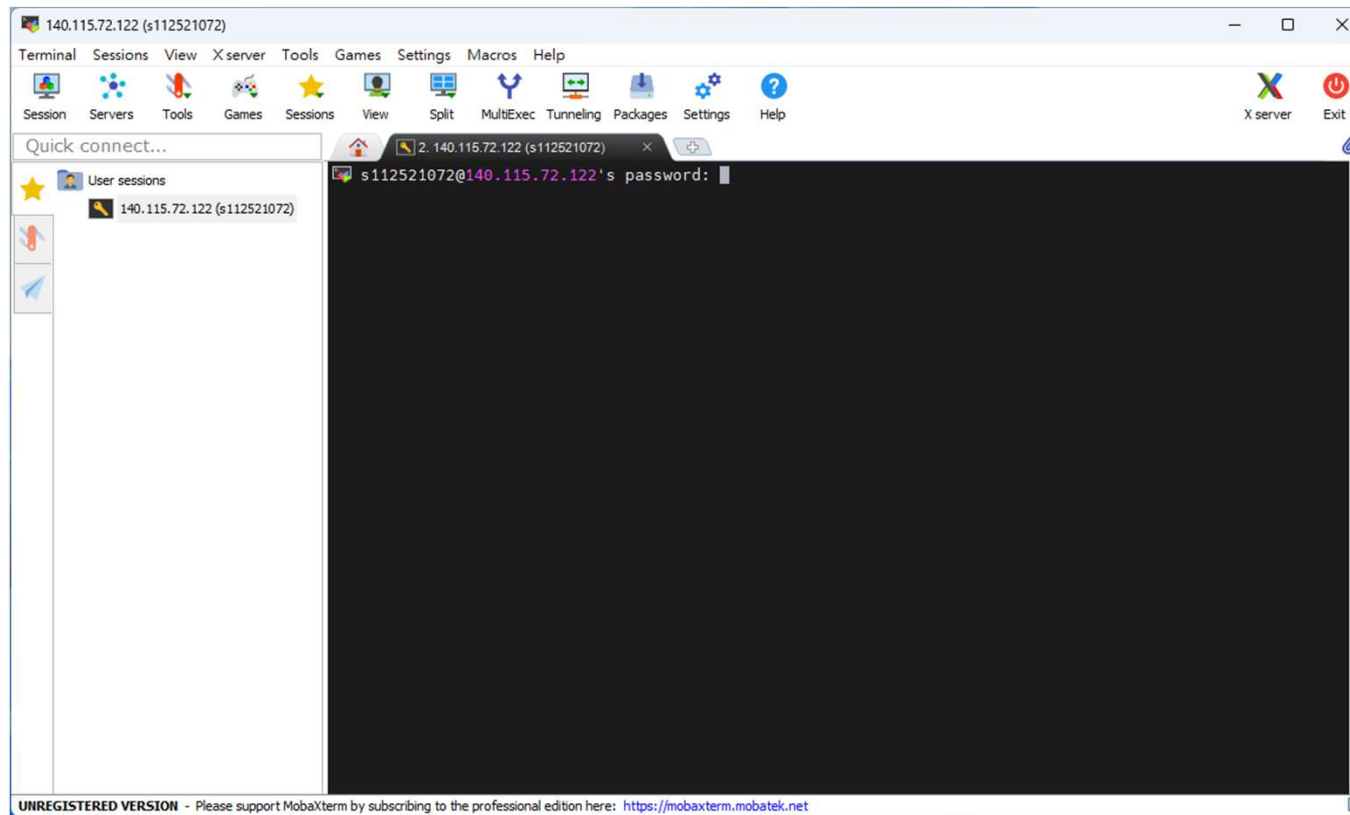
Login Server (2/5)

- Host : 140.115.72.122
- Username : s + your student ID no. (Ex: s112521072)
- Port : 5022



Login Server (3/5)

- Password(default) : ncuee



Login Server (4/5)

- Enter “passwd” for changing password in first time login.
- Or you can just use the default password, and you can press “ctrl + C” to terminate the command.

```
Last login: Fri Dec 22 03:22:26 2023 from gateway
[s112521072@centos7-rram ~]$ passwd
Changing password for user s112521072.
Changing password for s112521072.
(current) UNIX password:
New password:
BAD PASSWORD: The password is the same as the old one
New password:
Retype new password: 
```


Login Server (5/5)

- “exit” when you finishing your work

```
? MobaXterm Personal Edition v23.5 ?  
(SSH client, X server and network tools)  
  
▶ SSH session to s112521072@140.115.72.122  
  ? Direct SSH      : ✓  
  ? SSH compression : ✓  
  ? SSH-browser     : ✓  
  ? X11-forwarding  : ✓ (remote display is forwarded through SSH)  
  
▶ For more info, ctrl+click on help or visit our website.  
  
Last failed login: Fri Dec 22 05:04:25 CST 2023 from gateway on ssh:notty  
There was 1 failed login attempt since the last successful login.  
Last login: Fri Dec 22 04:55:48 2023 from gateway  
[s112521072@centos7-rram ~]$ exit  
logout  
  
Session stopped  
- Press <Return> to exit tab  
- Press R to restart session  
- Press S to save terminal output to file
```

Environment Setup

Environment

- TSMC N16 ADFP (Academic Design Foster Package)
- Cadence (including Virtuoso)
- Mentor Graphics Calibre

Start virtuoso

- At your home directory :

```
[s112521009@centos7-rram ~]$ cd sh
```

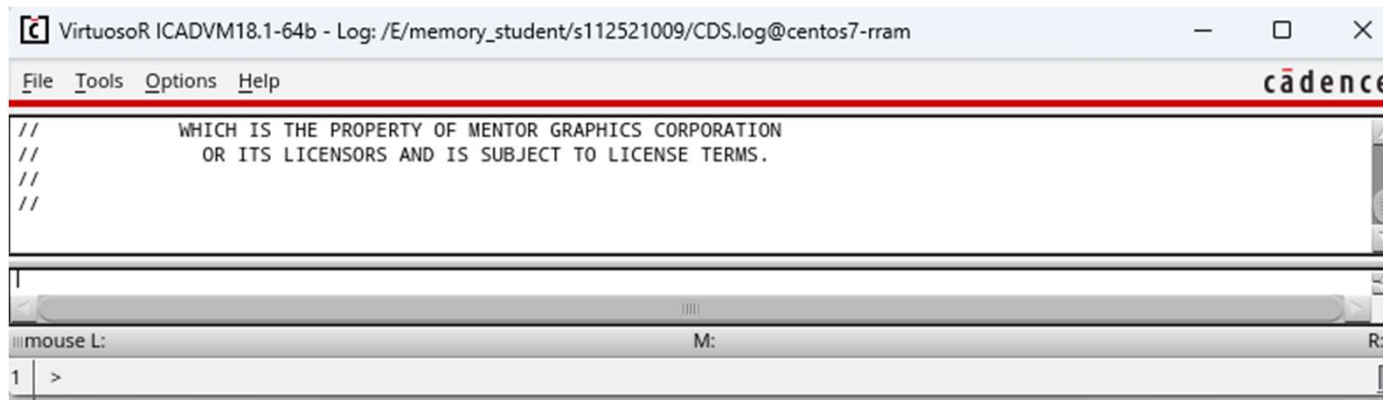
- Go into the 'sh' folder

```
[s112521009@centos7-rram sh]$ /bin/csh
```

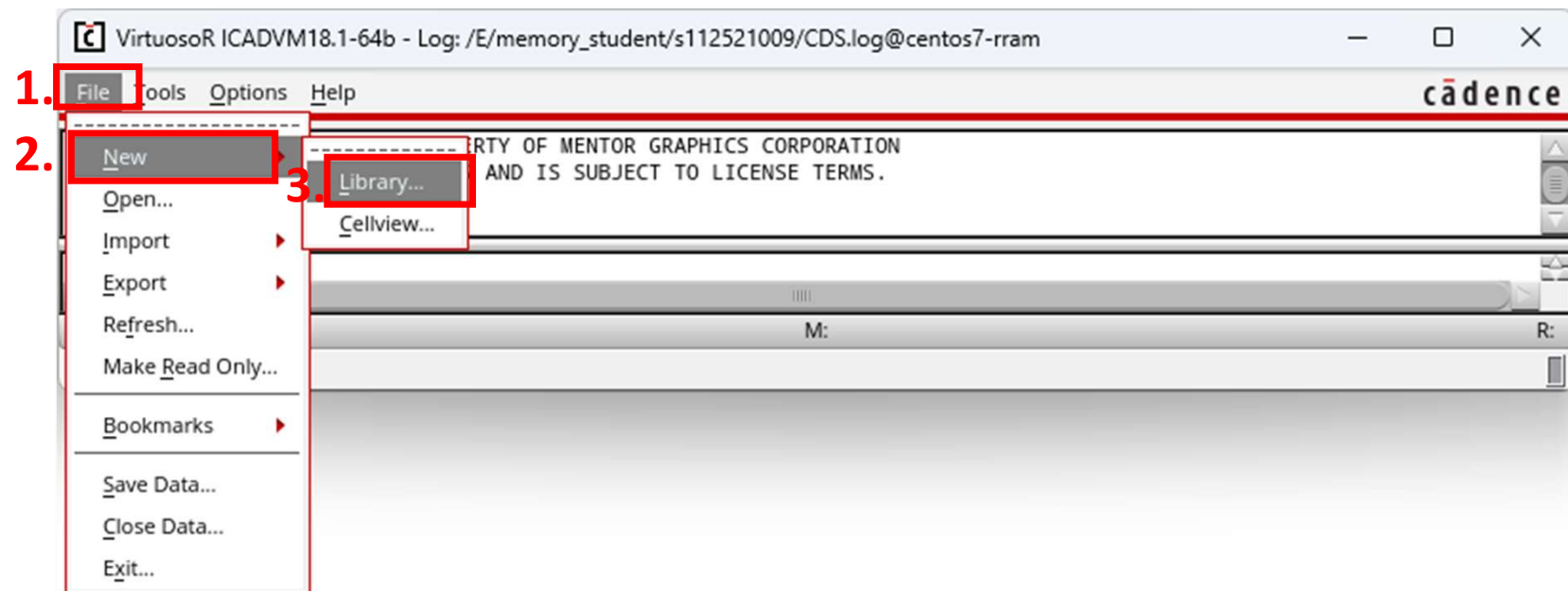
```
[s112521009@centos7-rram ~/sh]$ source memory.sh
```

```
[s112521009@centos7-rram ~/sh]$ virtuoso &
```

- ICADVM window :

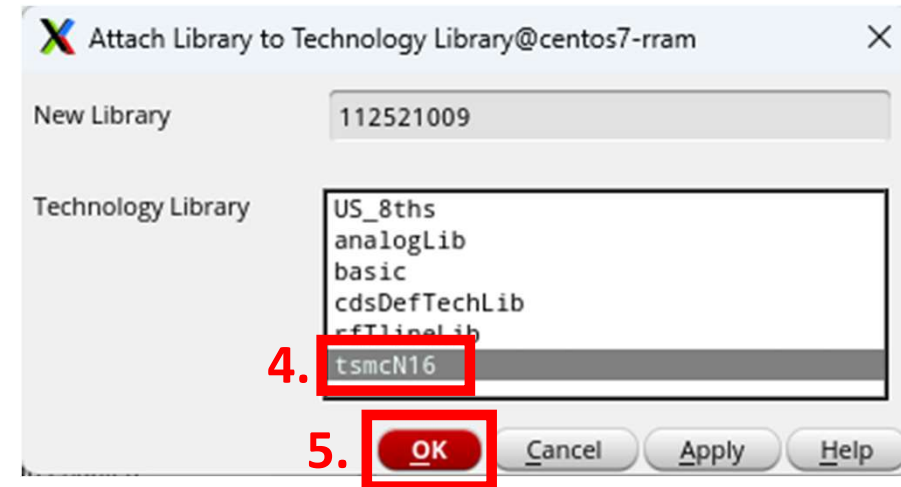
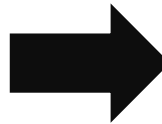
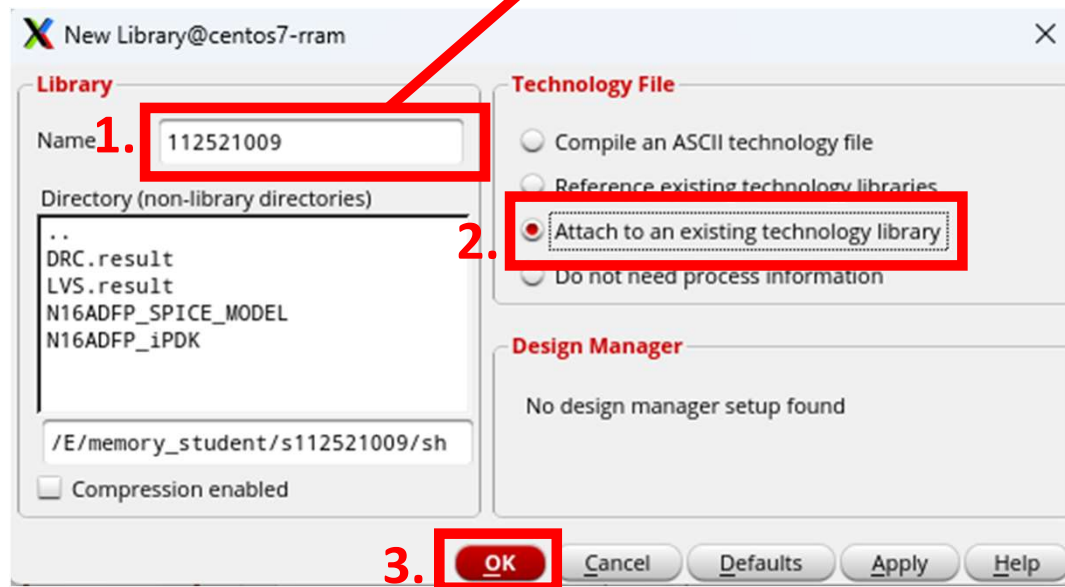


Construct New Library (1/2)



Construct New Library (2/2)

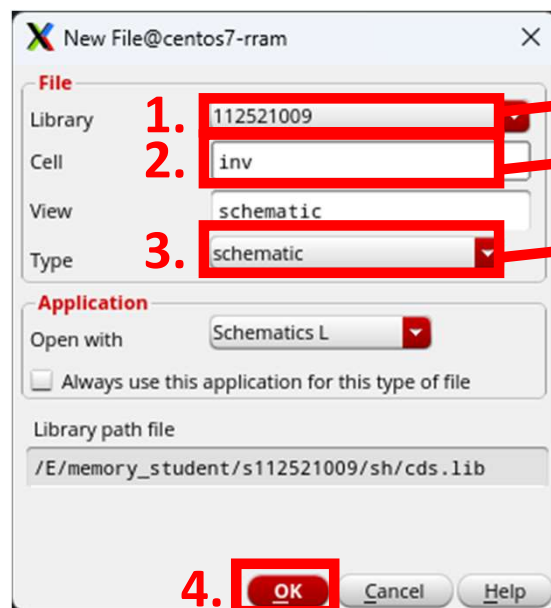
Please name it using your student ID



Draw Schematic using Virtuoso

Create Schematic View

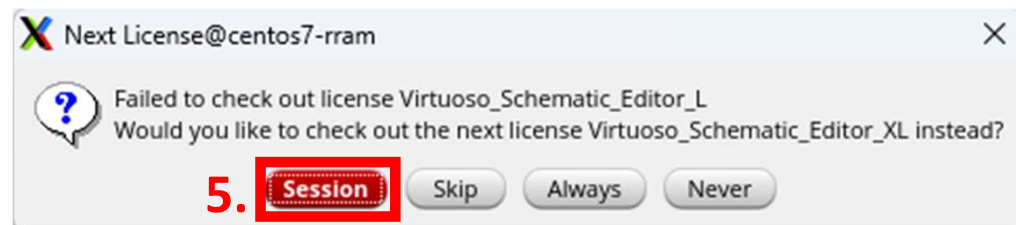
- ICADVM window : Tools -> Library Manager
- Library Manager : Click your new library -> File -> New -> Cell View...



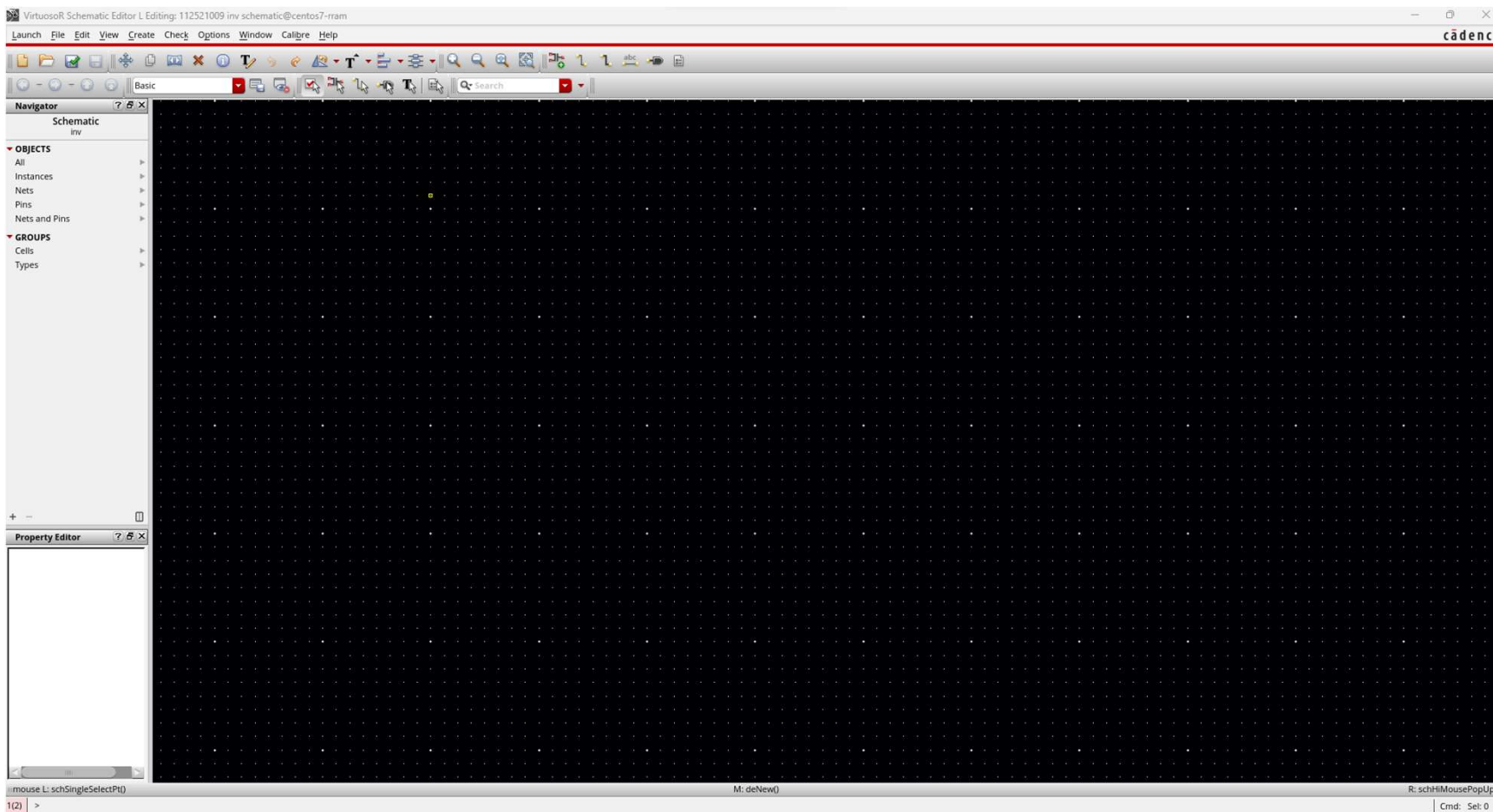
Choose your new library

Naming your new cell

Choose schematic



Cadence Virtuoso Schematic Editor

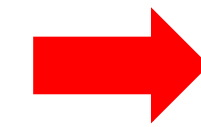
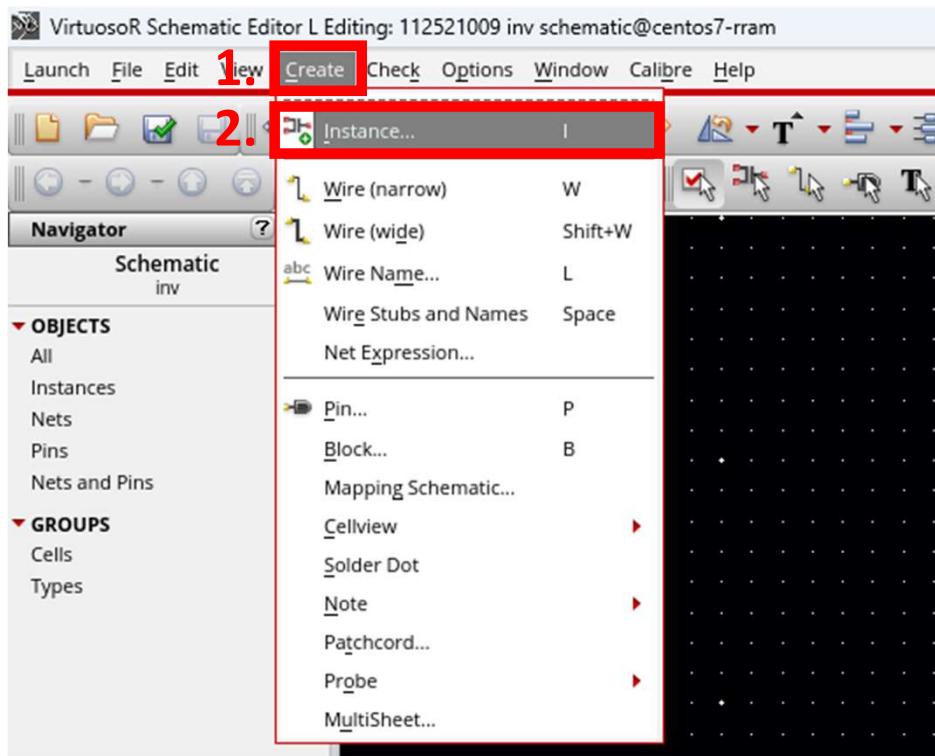


Can use
keyboard's
“↑↓←→”
to move
window

Cadence Virtuoso Schematic Editor

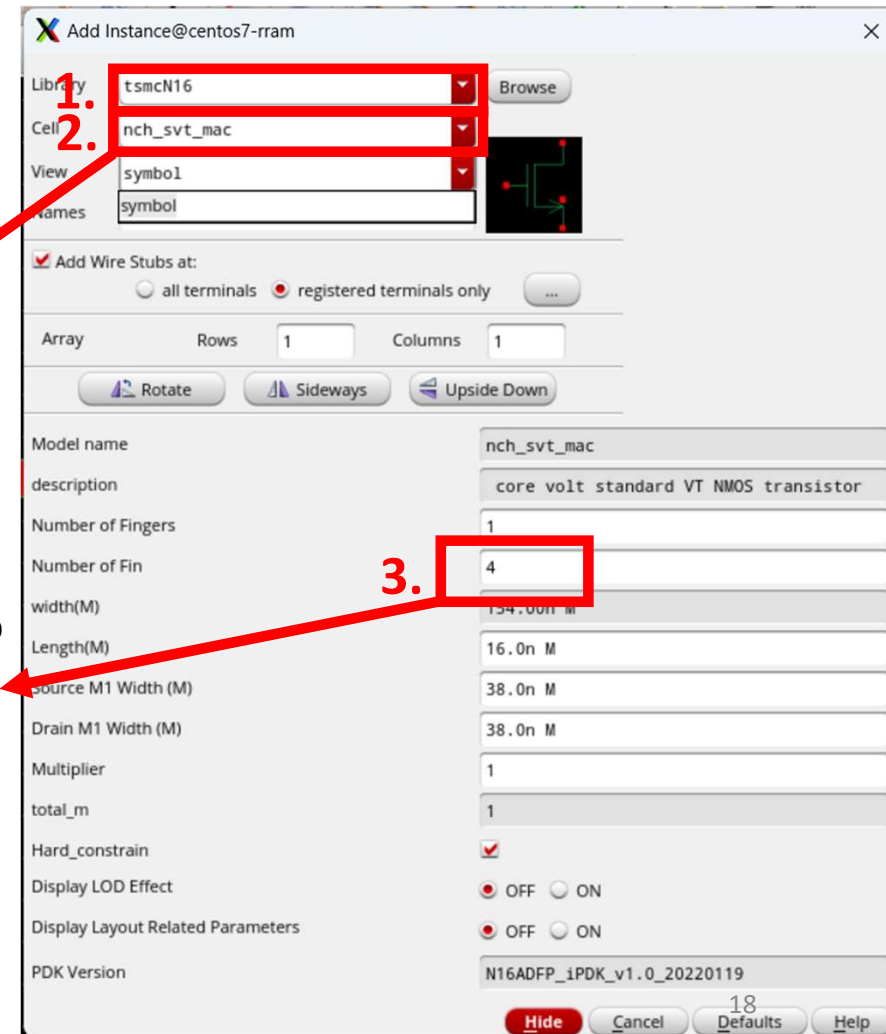
Function	Command	Function	Command
Zoom in	Ctrl + z	Properties	q
Zoom out	Shift + z	Instance	i
Zoom overall circuit	Ctrl + x	Path	p
Zoom in specific location	mouse right button	Polygon	Shift + p
Stretch	s	Label	l
Copy	c	Rectangle	r
Move	m	Ruler/Clear Ruler	k/Shift + k
Delete	del	Save	F2
Undo	u	Erase current command	esc

Another way to add instance



Choosing
nch_svt_mac
pch_svt_mac

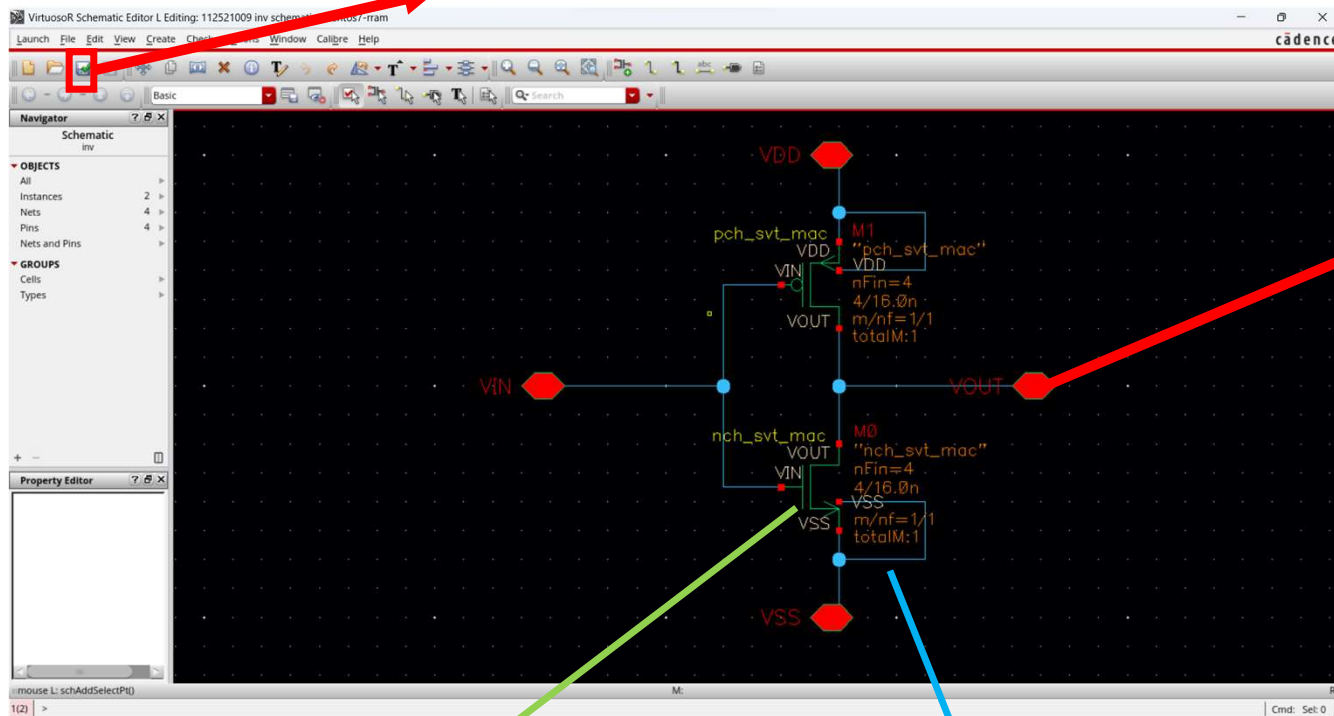
Please set the
number of fins to
be 4 or more,
and it must be
the same as the
layout quantity.



Schematic

4. After finishing the schematic, remember to click on “check and save”

3. Command “p”



The 'Create Pin@centos7-rram' dialog box is shown. It has a title bar with a close button. The main area contains several fields and checkboxes. The 'Names' field is filled with 'VSS VDD VIN VOUT'. The 'Direction' dropdown is set to 'inputOutput'. The 'Usage' dropdown is set to 'schematic'. The 'Signal Type' dropdown is set to 'signal'. There are two checkboxes: 'Expand busses' and 'Place multiple pins', both of which are unchecked. Below these is a section for 'Net Expression' with an 'Attach to pin' checkbox, which is also unchecked. At the bottom, there is a 'Rotation' section with four icons for different pin orientations. At the very bottom are four buttons: 'Hide' (red), 'Cancel', 'Defaults', and 'Help'.

1. Command “i”

2. Command “w”

Export the .sp file (1/2)

1. File

2. Export

3. CDL...

4. Library Browser...

5. inv.sp

6. Analog

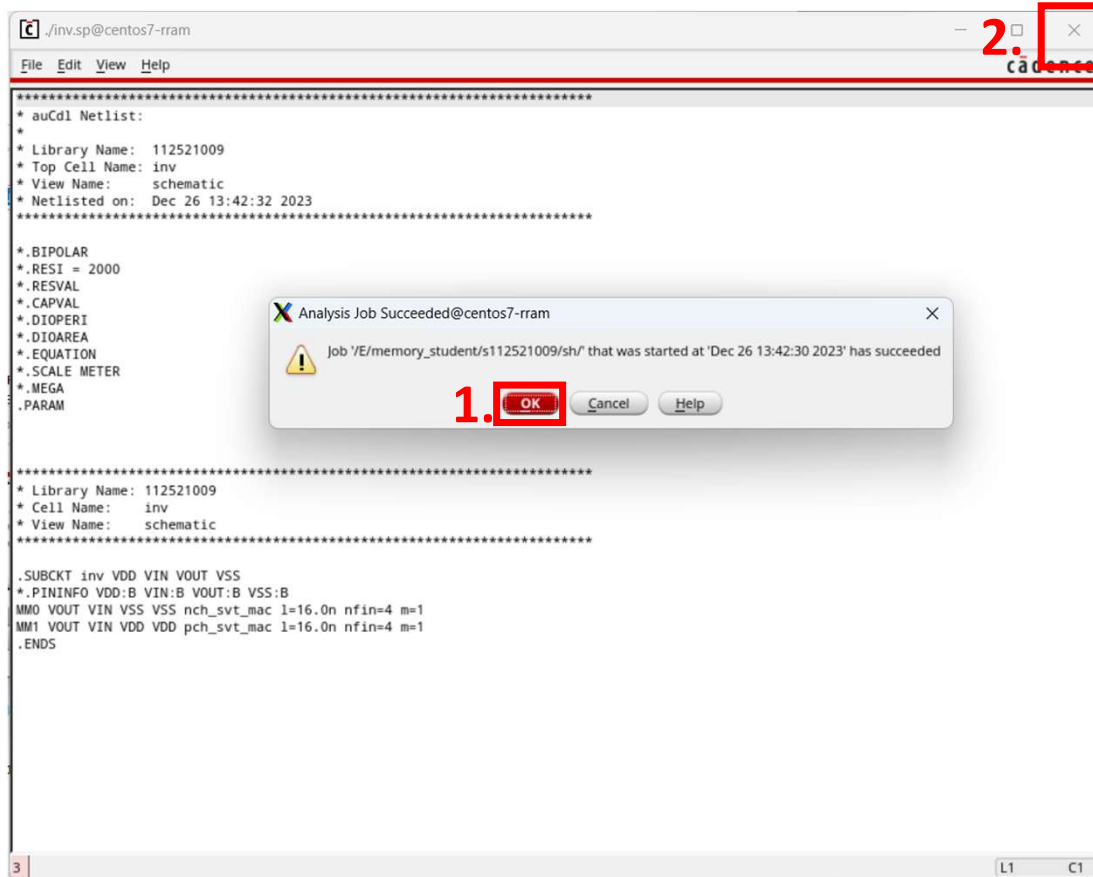
7. OK

Choosing your schematic

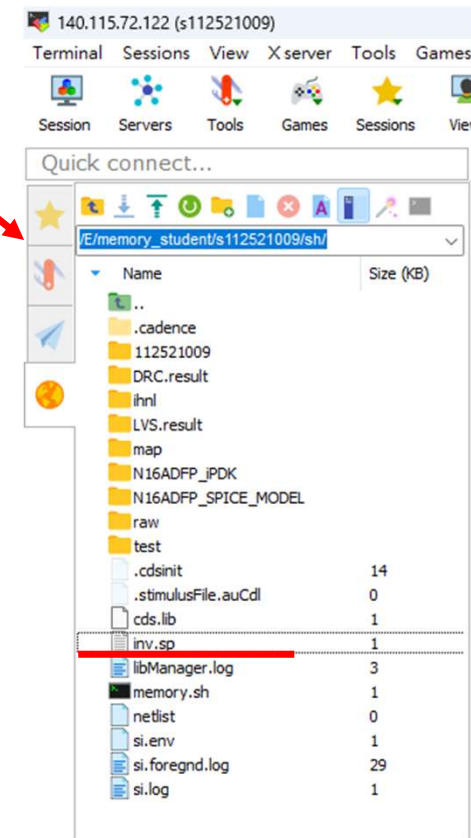
Please name it as "your_schematic_name.sp"

The image shows a sequence of steps for exporting a .sp file from Cadence Virtuoso. Step 1: Click on the 'File' menu. Step 2: Click on 'Export'. Step 3: Click on 'CDL...'. Step 4: Click on 'Library Browser...'. Step 5: Enter 'inv.sp' in the 'Output CDL Netlist File' field. Step 6: Select 'Analog' for 'Netlisting Mode'. Step 7: Click 'OK'. The 'Library Browser' dialog shows the 'Library' list with '112521009' selected, the 'Cell' list with 'inv' selected, and the 'View' list with 'schematic' selected. The 'CDL Out' dialog shows the 'Library Name' as '112521009', 'Top Cell Name' as 'inv', 'View Name' as 'schematic', 'Switch View List' as 'auCd1 schematic', and 'Stop View List' as 'auCd1'. The 'Output' section shows 'Output CDL Netlist File' as 'inv.sp' and 'Run Directory' as '.'. The 'Netlisting Mode' is set to 'Analog'.

Export the .sp file (2/2)



The .sp file will appear in the “sh” folder under your own student ID.



Edit the .sp file

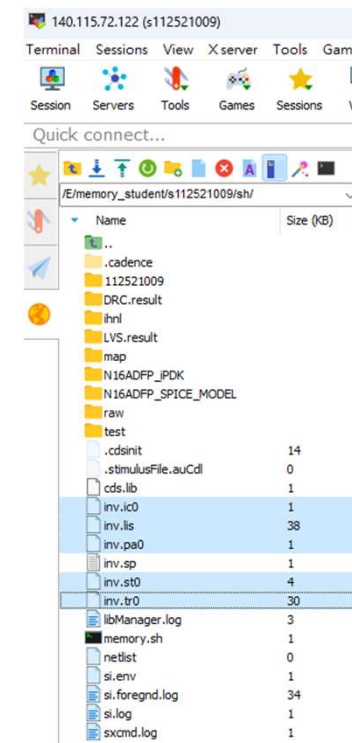
- Go into the 'sh' folder

```
[s112521009@centos7-rram ~/sh]$ hspice -i inv.sp -o inv.lis
```

```
1 *****
2 * auCdl Netlist:
3 *
4 * Library Name: 112521009
5 * Top Cell Name: inv
6 * View Name: schematic
7 * Netlisted on: Dec 26 13:31:31 2023
8 *****
9 .TEMP 25
10 .OPTION POST=2
11 .LIB '/E/memory_student/s112521009/sh/N16ADFP_SPICE_MODEL/n16adfp_spice_model_v1d0_usage.l' TTMacro_NOS_MOSCAP
12 *****
13 *.INCLUDE /E/cad_install/sh/N16ADFP_iPDK/tsmcN16/./Calibre/lvs/source.added
14 *.BIPOLAR
15 *.RESI = 2000
16 *.RESVAL
17 *.CAPVAL
18 *.DIOPERI
19 *.DIOAREA
20 *.EQUATION
21 *.SCALE METER
22 *.MEGA
23 .PARAM
24
25
26
27 *****
28 * Library Name: 112521009
29 * Cell Name: inv
30 * View Name: schematic
31 *****
32
33 .SUBCKT inv VDD VIN VOUT VSS
34 *.PININFO VDD:B VIN:B VOUT:B VSS:B
35 MM0 VOUT VIN VSS VSS nch_svt_mac l=16.0n nfin=4 m=1
36 MM1 VOUT VIN VDD VDD pch_svt_mac l=16.0n nfin=4 m=1
37 .ENDS
38
39 X1 VDD VIN VOUT VSS inv
40 VDD VDD 0 DC 1.8V
41 VIN VIN 0 PULSE(0 1.8 0.1n 0.1n 0.1n 0.5u 1u)
42 VSS VSS 0 DC 0V
43 .tran 0.1n 5u
44 .end
```

Add these three lines above INCLUDE, remember to change the LIB path to your student ID, and comment out the INCLUDE line.

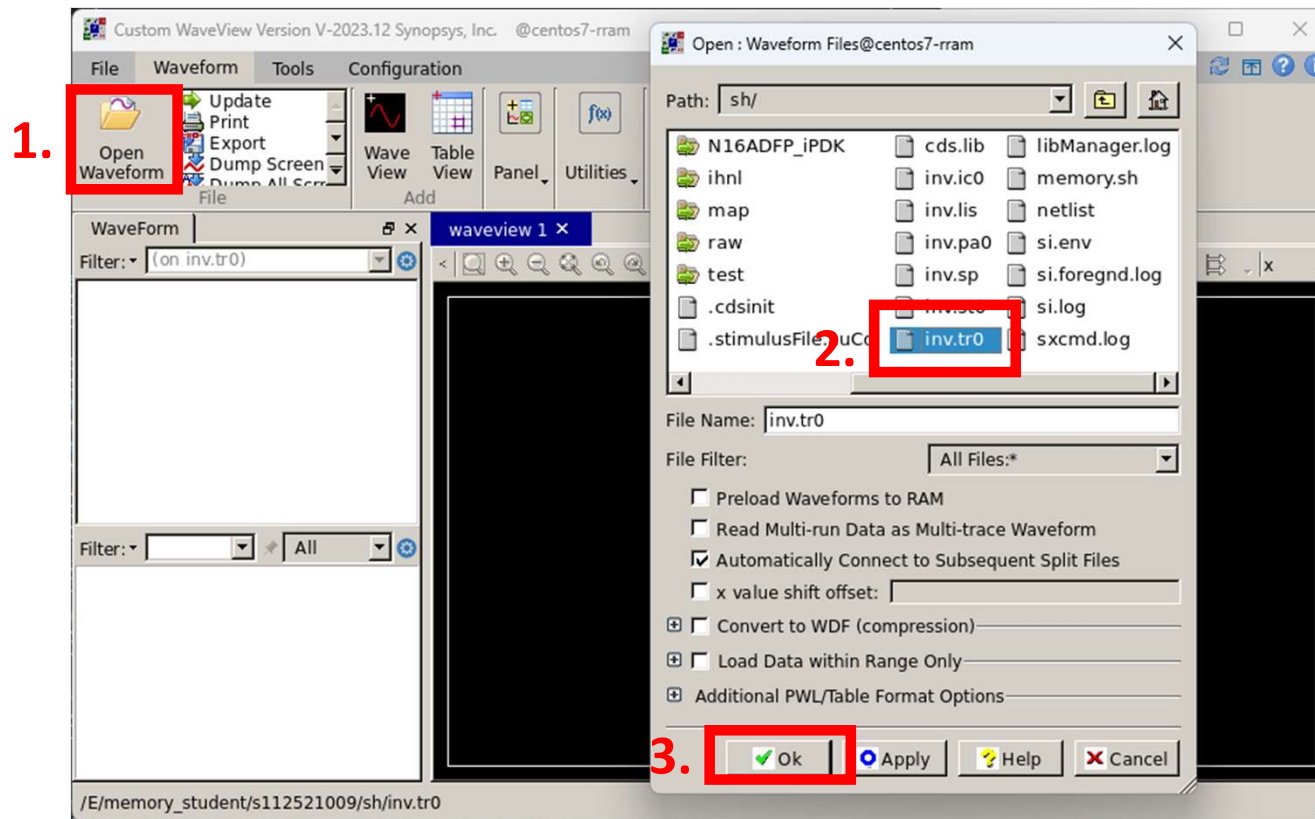
Make sure the order and names here match those in SUBCKT.



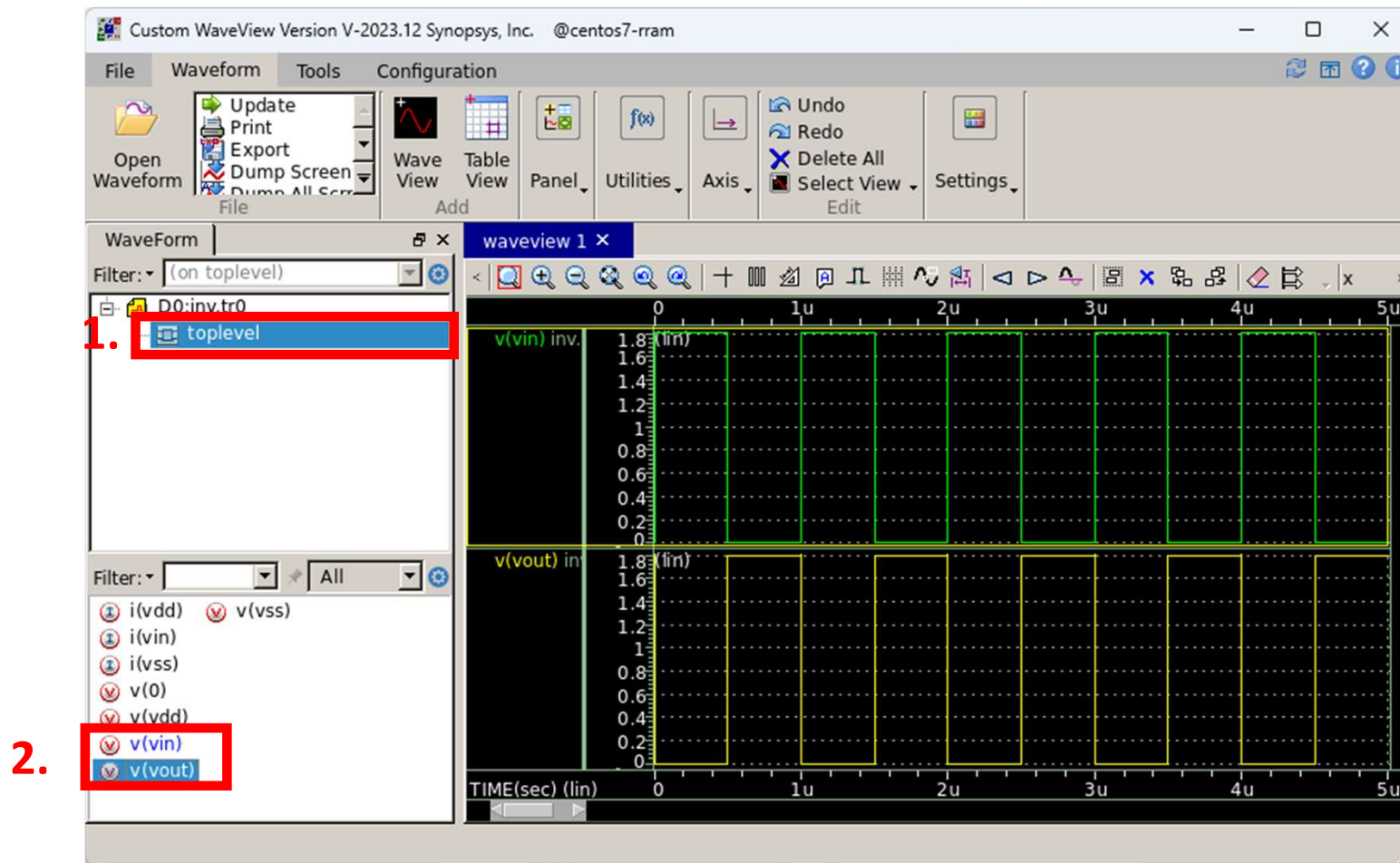
Output waveform (1/2)

- Go into the 'sh' folder

```
[s112521009@centos7-rram ~/sh]$ wv &
```



Output waveform (2/2)



Presim completed

THANK YOU FOR YOUR
ATTENTION!