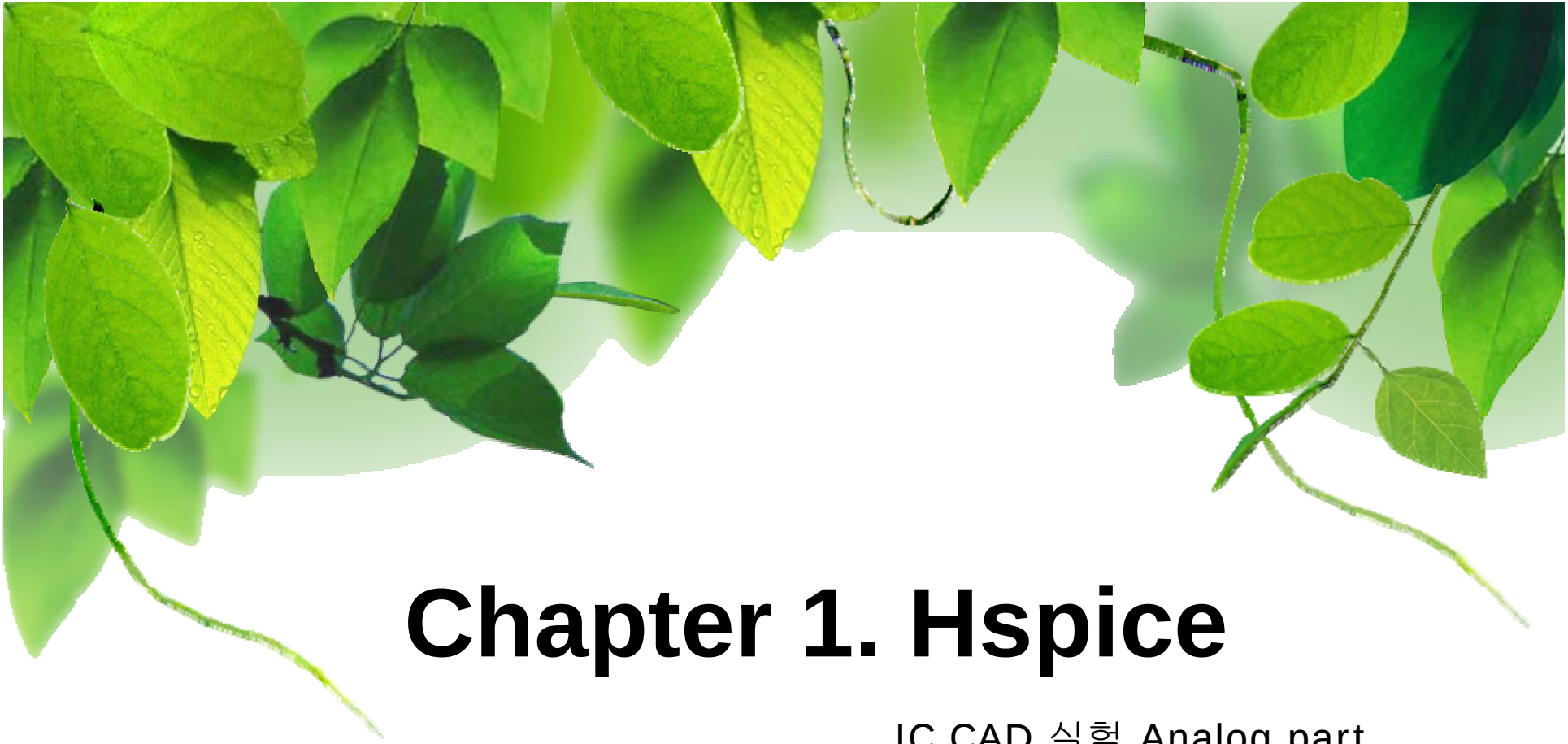


Chapter 1. Hspice

IC CAD 실험 Analog part

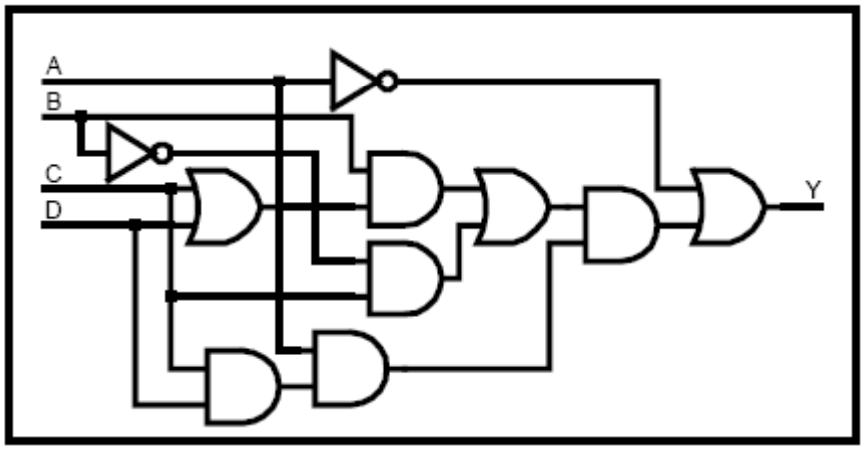
YONSEI UNIVERSITY

Digital circuit design

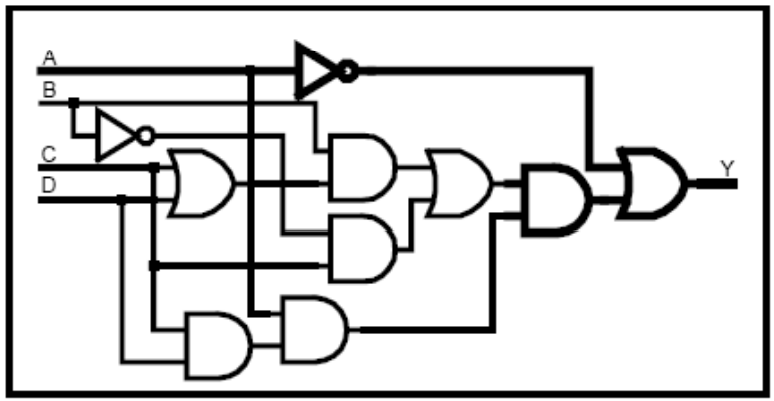
```
if(A == 1)
  Y = C & D;
else if(B==1)
  Y = C | D;
else
  Y = C;
```

RTL description

Synthesis

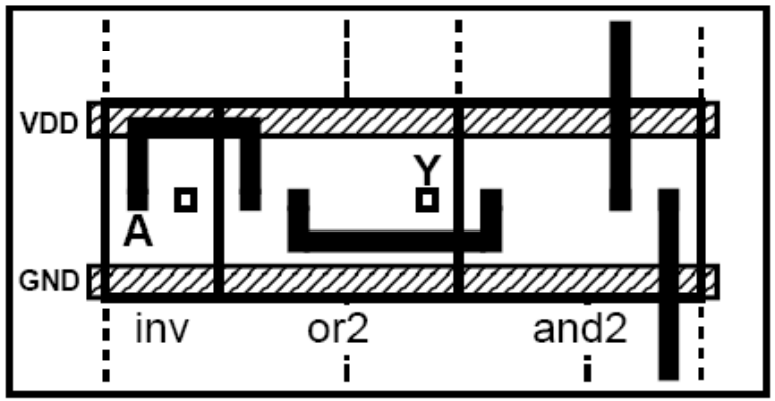


Gate-level netlist



Gate-level netlist

P&R



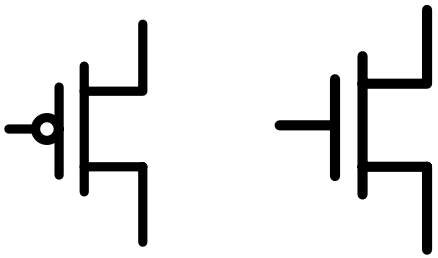
Layout

YONSEI UNIVERSITY

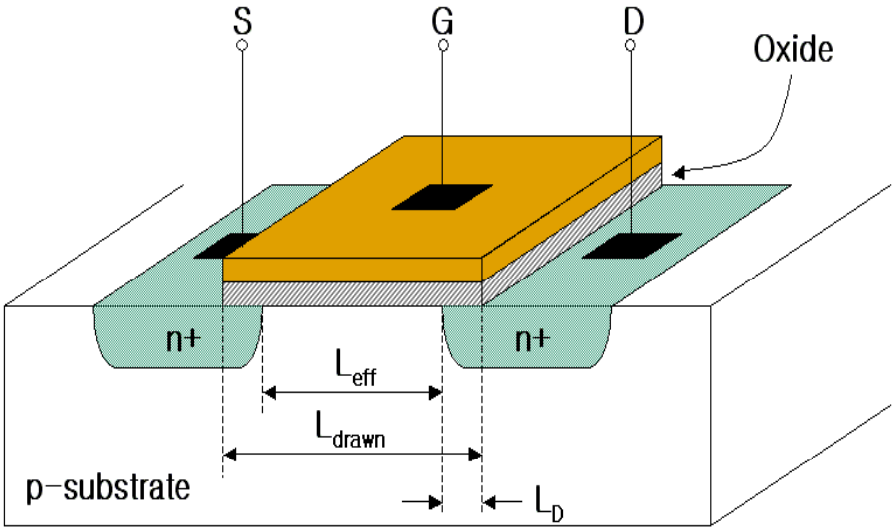
Layout?

MOSFET!

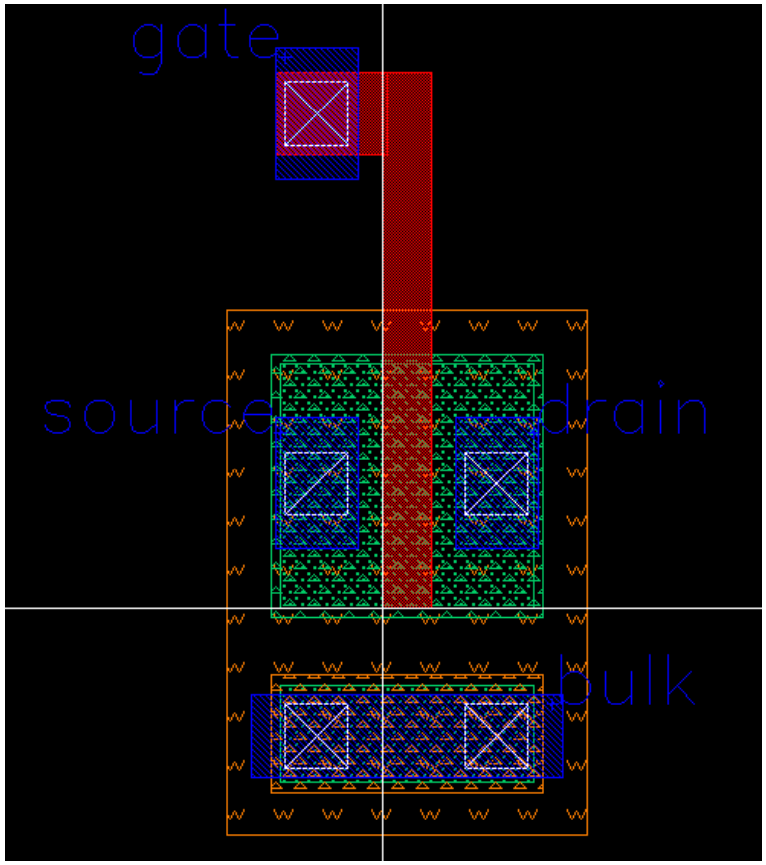
Symbol



Physical structure

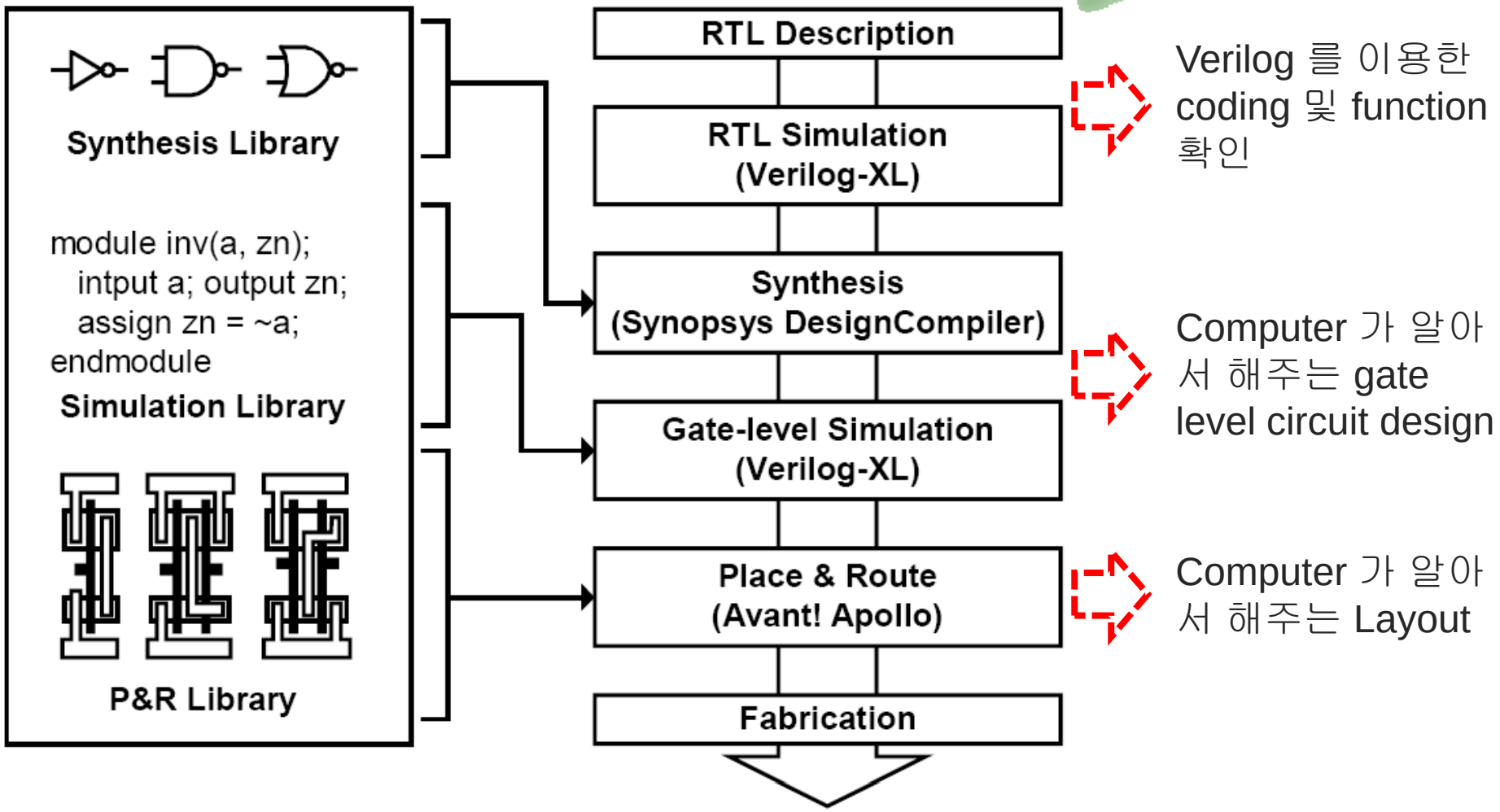


Layout



YONSEI UNIVERSITY

Digital circuit design



Analog circuit design

TR level circuit design

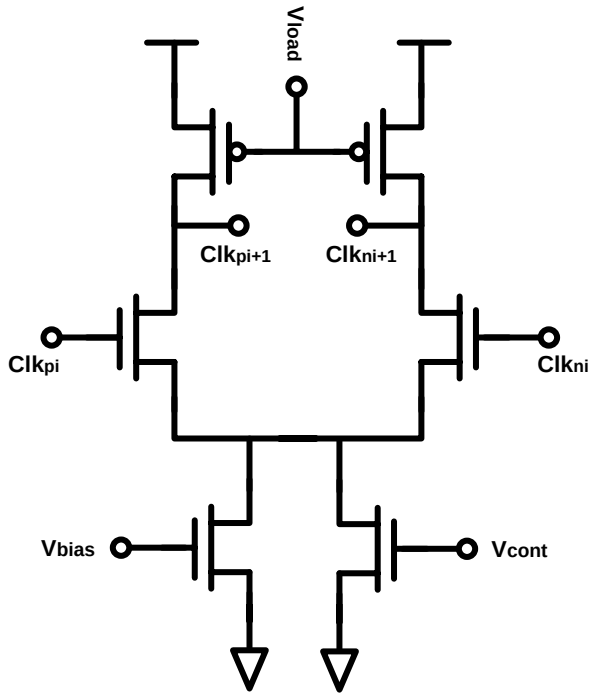
TR level simulation

Layout

Post layout simulation

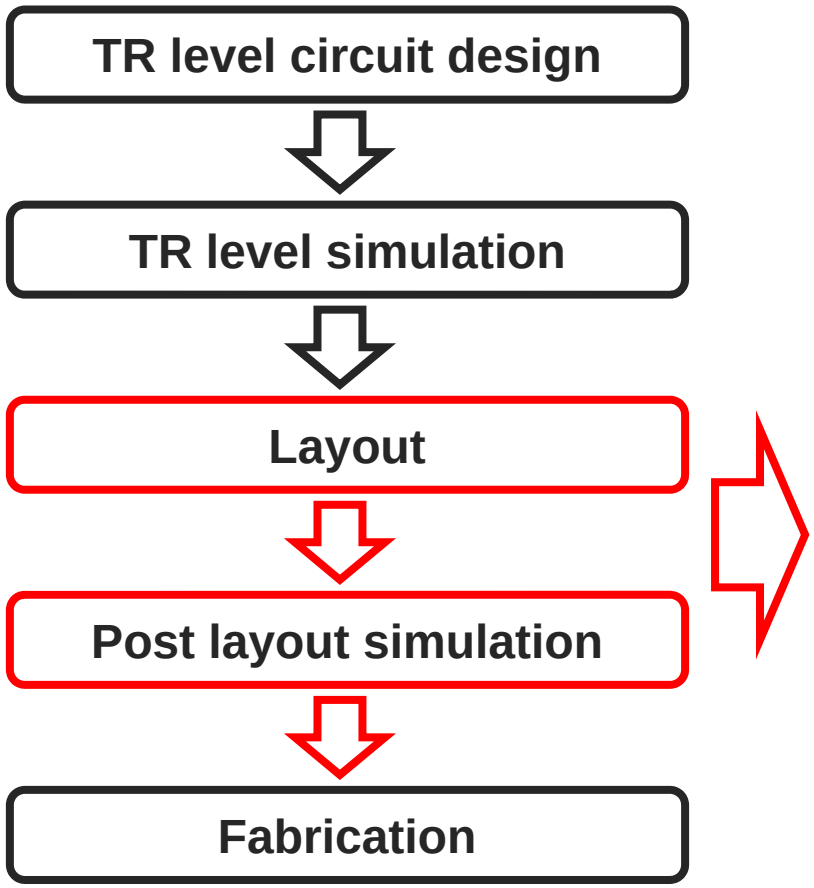
Fabrication

Hspice, cadence 를 이용한 TR level circuit design & simulation

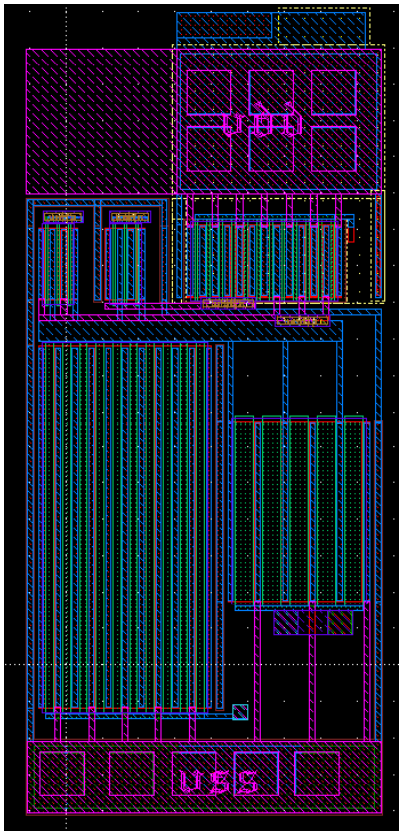


YONSEI UNIVERSITY

Analog circuit design

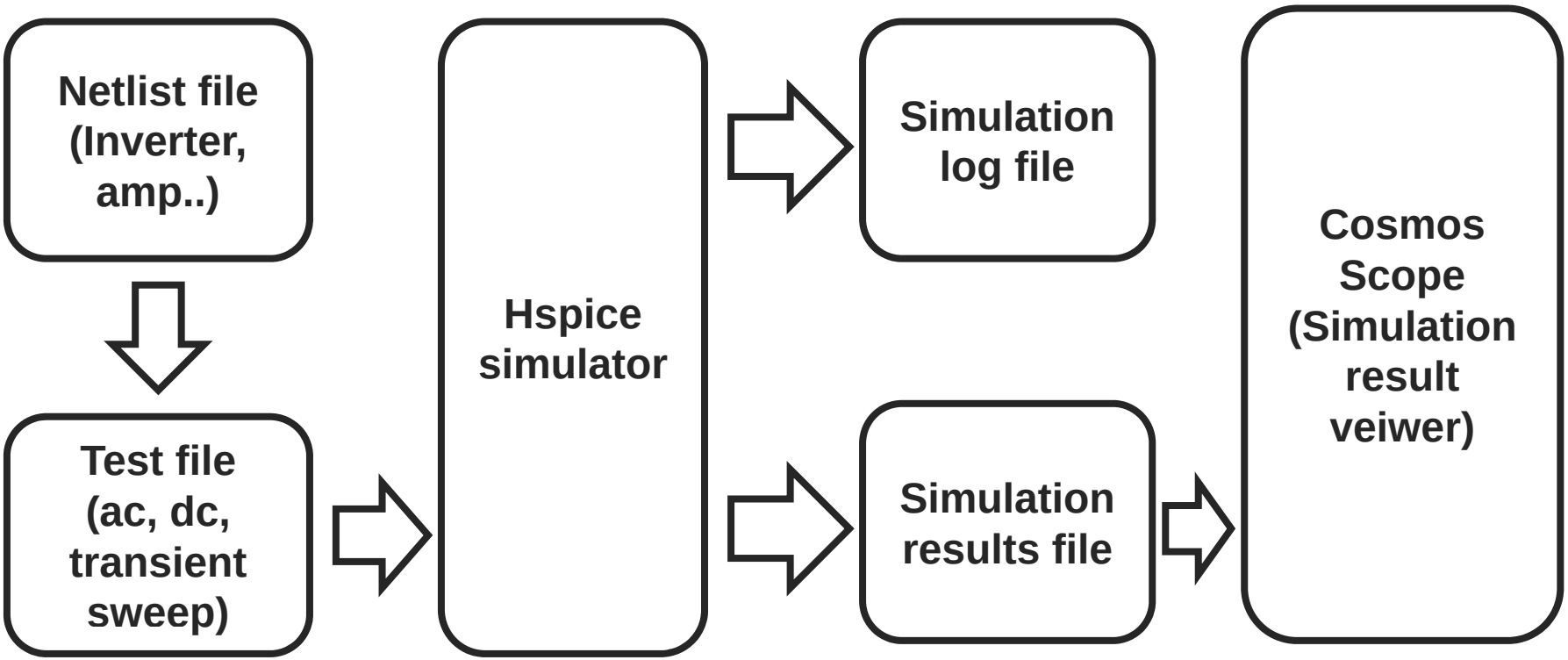


Cadence layout editor 를 이용한 손으로 하는~layout, Hspice, cadence 를 이용한 post layout simulation



Hspice simulation

TR level circuit simulator! – text 기반



Hspice simulation

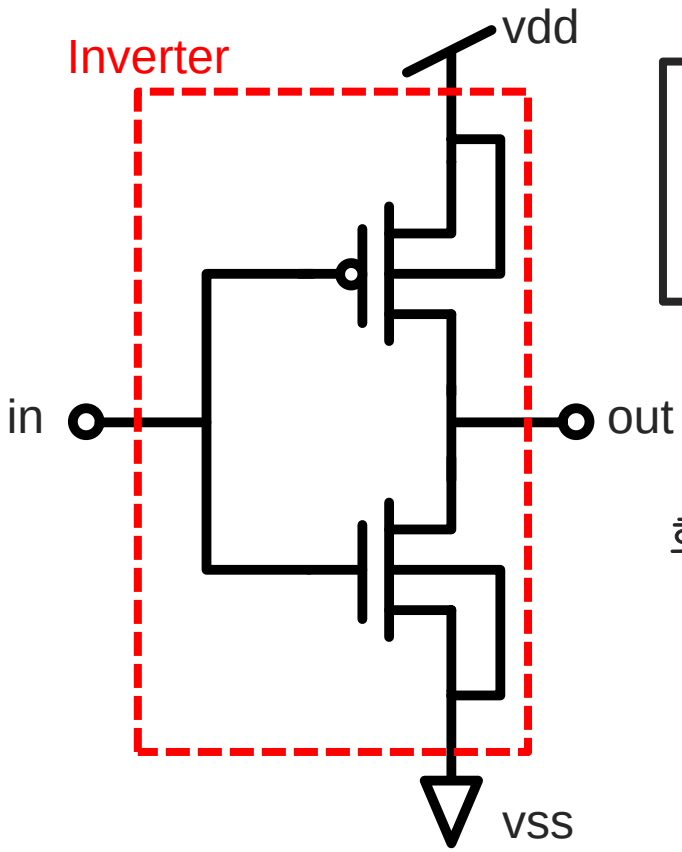
Example> Inverter 설계 및 simulation

[ICCAD@train##]/user1/train##/ > mkdir hspice
[ICCAD@train##]/user1/train##/ > cd hspice
[ICCAD@train##]/user1/train##/ hspice>
cp /user1/master/hspice/netlist.sp .
cp /user1/master/hspice/PMOS_VTL.inc .
cp /user1/master/hspice/NMOS_VTL.inc .
ls

[ICCAD@train##]/user1/train##/ hspice> vi netlist.sp

Hspice simulation

Example> Inverter 설계 및 simulation



Netlist

```
.subckt Inverter out in vdd vss
M_p out in vdd vdd PMOS_VTL w=10u l=0.05u
M_n out in vss vss NMOS_VTL w=5u l=0.05u
.ends
```

.subckt **Inverter** **out** **in** **vdd** **vss**

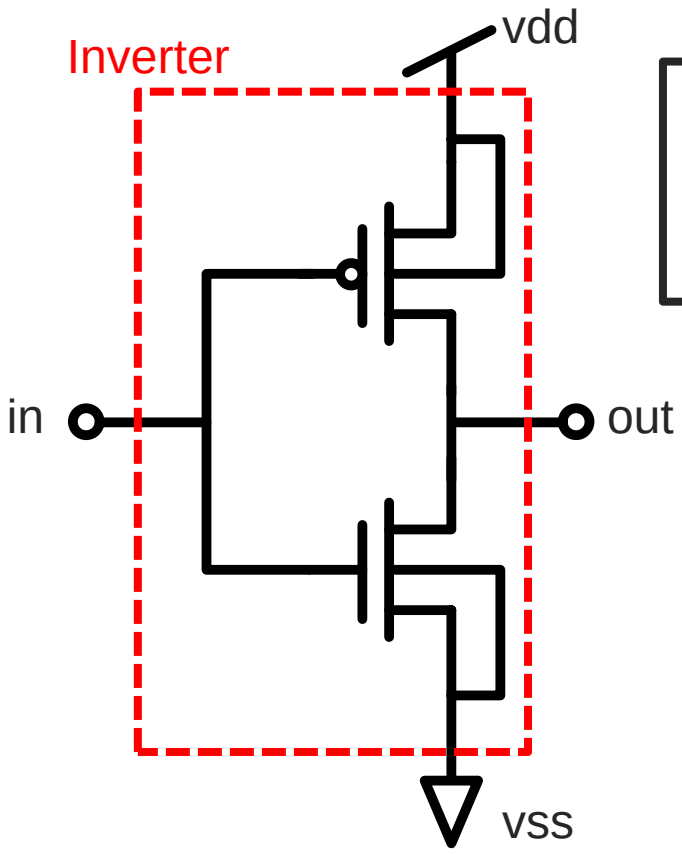
회로 만들기

회로 이름

회로 바깥에서 보이는 **port** 들

Hspice simulation

Example> Inverter 설계 및 simulation



Netlist

```
.subckt Inverter out in vdd vss
M_p out in vdd vdd PMOS_VTL w=10u l=0.05u
M_n out in vss vss NMOS_VTL w=5u l=0.05u
.ends
```

Mosfet 의 이름 | **Drain 의 node 명** | **Gate 의 node 명** | **Source 의 node 명** | **Bulk 의 node 명**

M_p | **out** | **in** | **vdd** | **vdd**

PMOS_VTL | **w=10u** | **l=0.05u**

Mosfet 의 type

Mosfet 의 width 와 length

Inverter 설계 끝!

Hspice simulation

Example> Inverter 설계 및 simulation

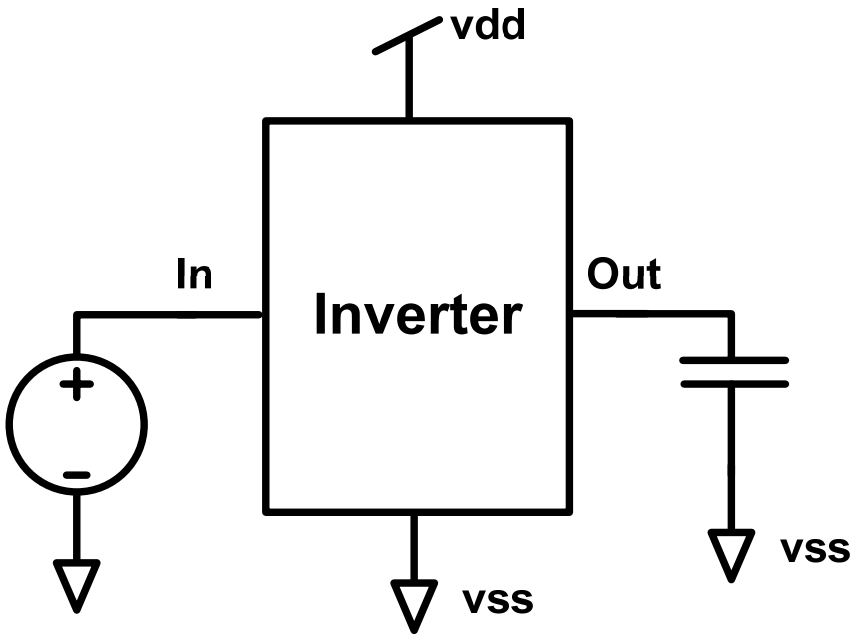
[ICCAD@train##]/user1/train##/ hspice>
cp /user1/master/hspice/inv_sim.sp .

[ICCAD@train##]/user1/train##/ hspice> vi inv_sim.sp

Hspice simulation

Example> Inverter 설계 및 simulation

DC simulation




```
.include 'PMOS_VTL.inc'
.include 'NMOS_VTL.inc'
.include 'netlist.sp'
```

Model 및 만들어 놓은 회로 불러오기

```
.OPTIONS POST NODE LIST

V1 vdd 0 1
V2 vss 0 0
Vin    in     vss    0.5
```

Voltage 정의해 주기

Inverter 라는 회로 불러오기

```
x_inv   out    in     vdd    vss    Inverter
c_out   out    vss    1p
```

Capcitor 정의

```
.dc    Vin    0      1      0.001
.END
```

Vin 이라는 voltage source 를 0.001 단위로 0에서 1까지 변화시키면서 각각의 node 들의 값이 어떻게 되는지 simulation 해라.

Hspice simulation

Example> Inverter 설계 및 simulation

[ICCAD@train##]/user1/train##/ hspice>
hspice inv_sim.sp > a.lis &

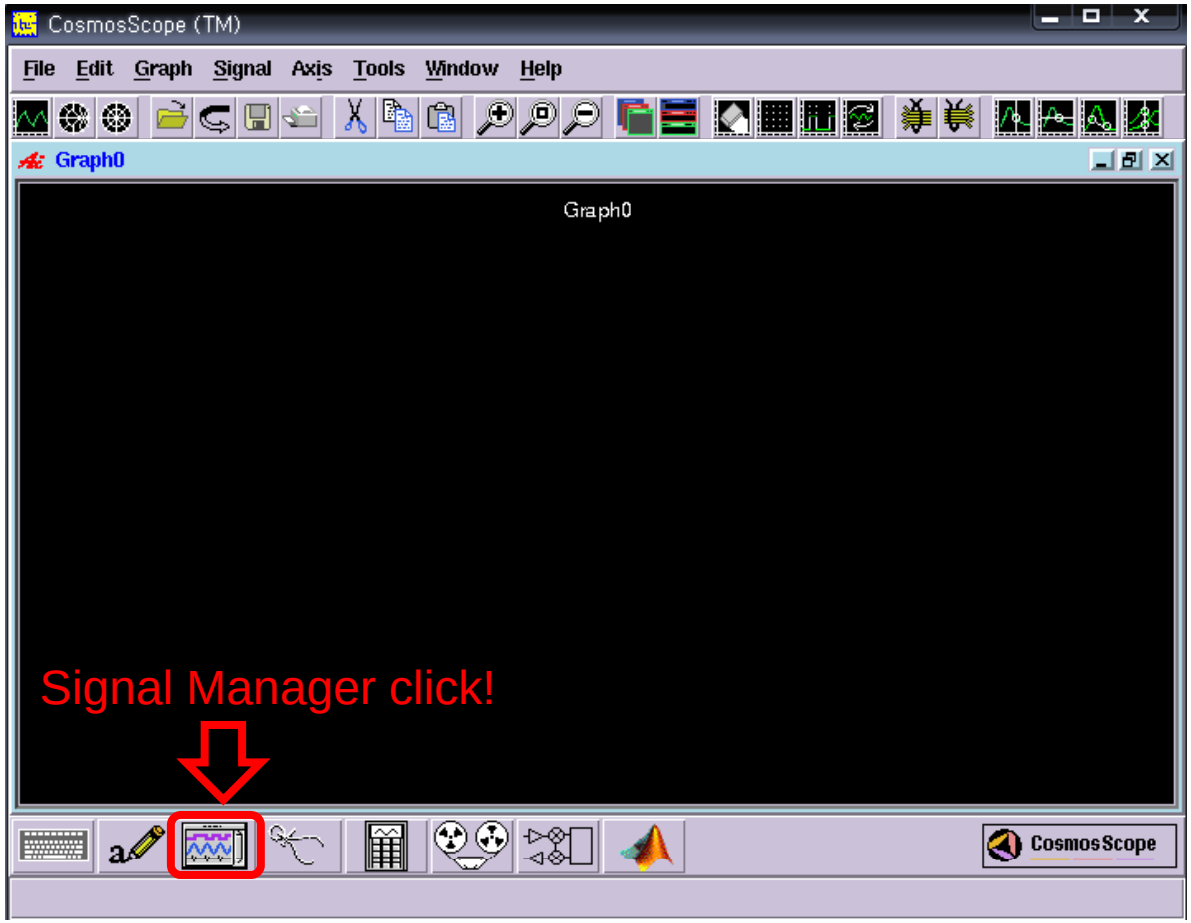
[ICCAD@train##]/user1/train##/ hspice> >info :
**** hspice jab concluded

[ICCAD@train##]/user1/train##/ hspice> cscope &

Hspice simulation

Example> Inverter 설계 및 simulation

파형 확인하기



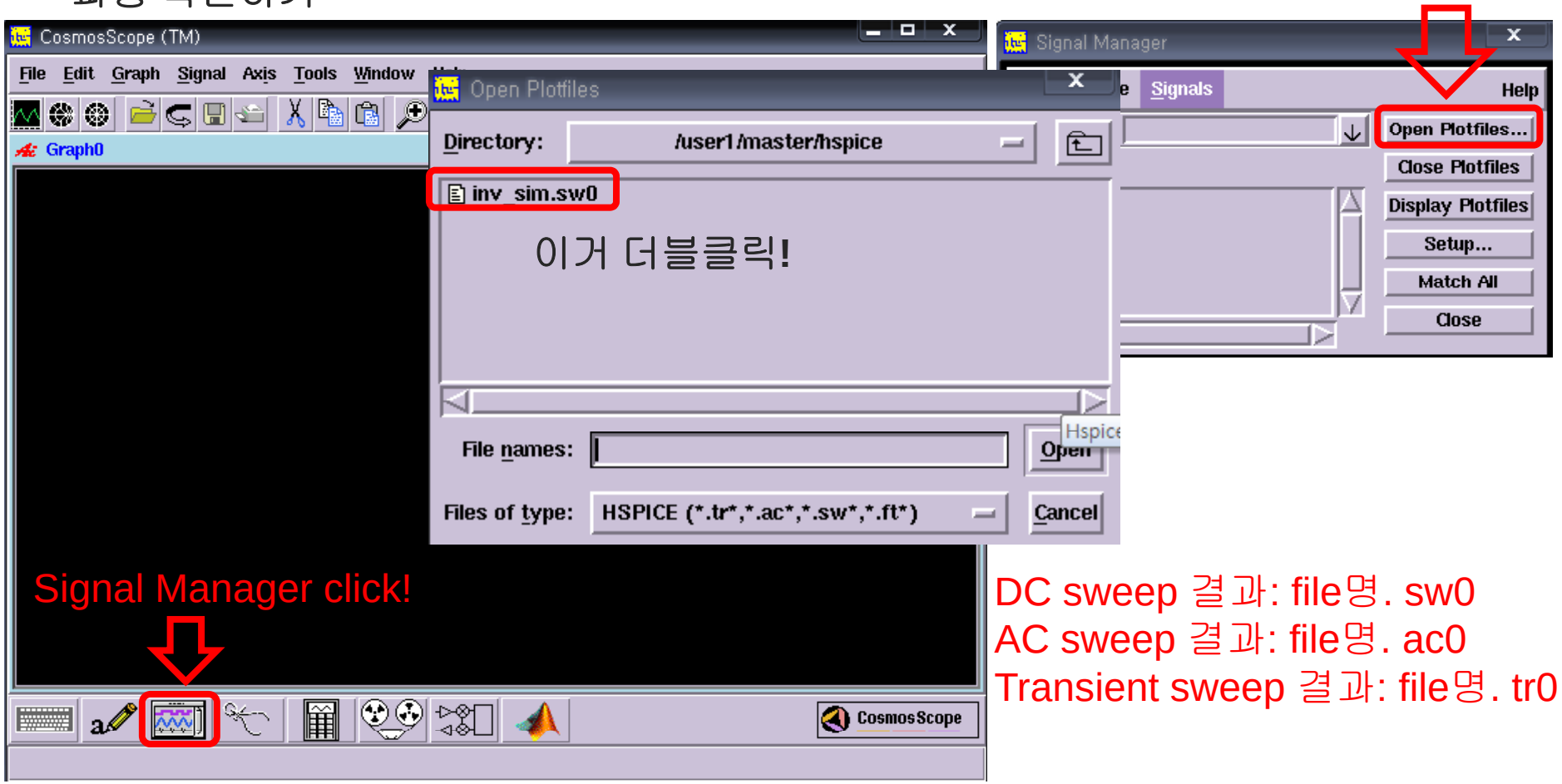
파형 열어보자!

Hspice simulation

Example> Inverter 설계 및 simulation

파형 확인하기

파형 열어보자!

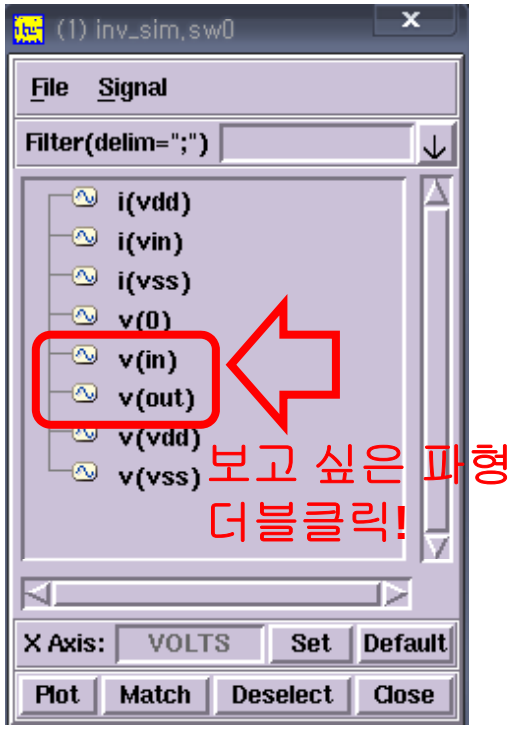


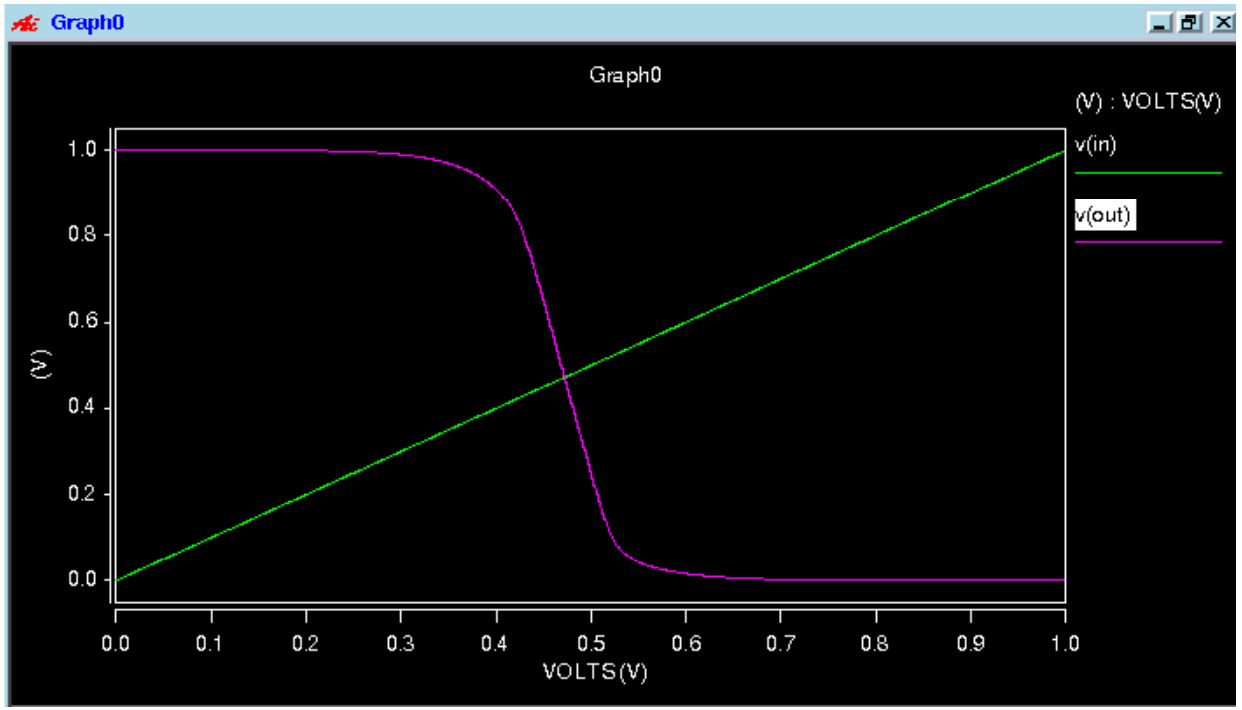
DC sweep 결과: file명. sw0
AC sweep 결과: file명. ac0
Transient sweep 결과: file명. tr0

YONSEI UNIVERSITY

Hspice simulation

Example> Inverter 설계 및 simulation

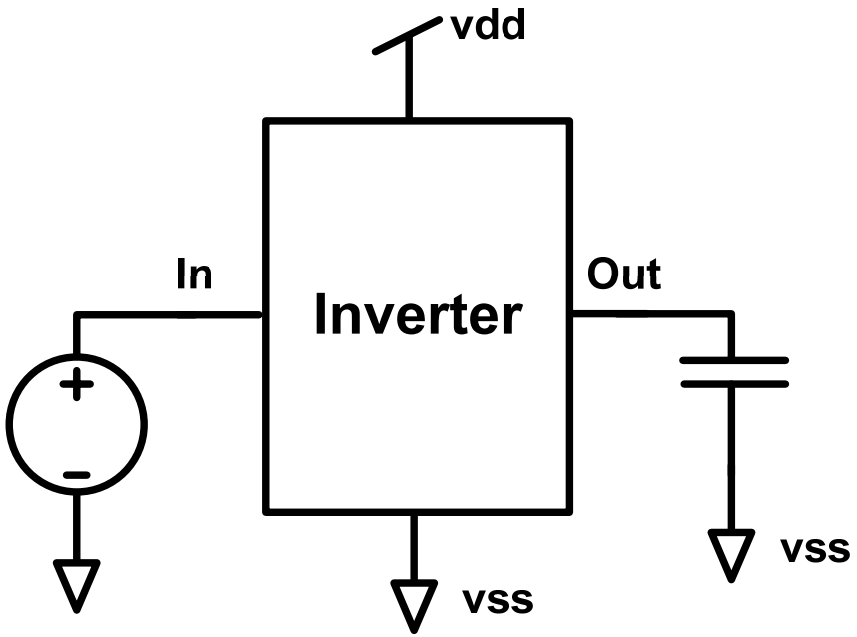




Hspice simulation

Example> Inverter 설계 및 simulation

AC simulation



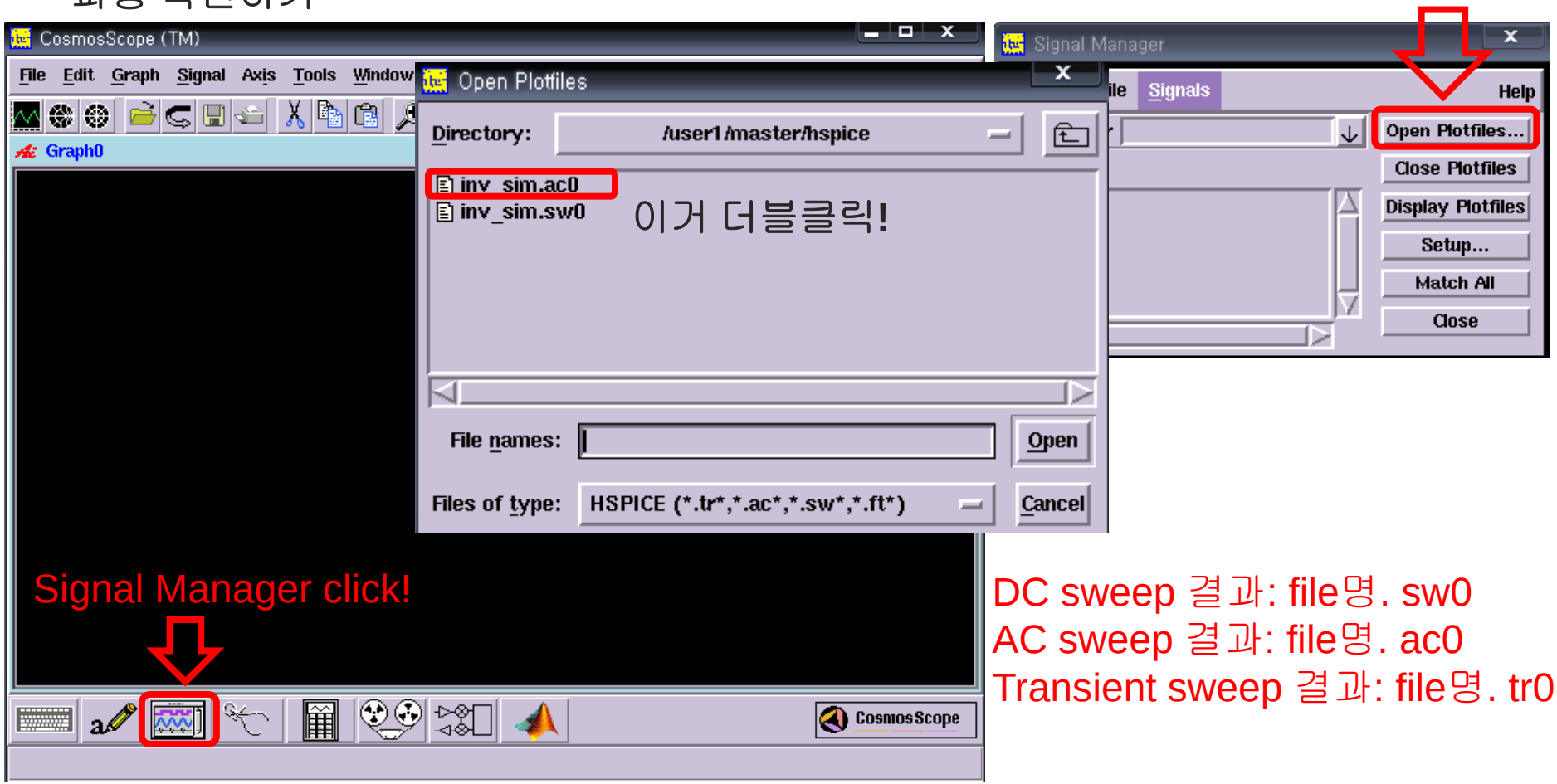

```
.include 'PMOS_VTL.inc'
.include 'NMOS_VTL.inc'
.include 'netlist.sp'

.OPTIONS POST NODE LIST

V1 vdd 0 1
V2 vss 0 0
Vin    in     vss    0.5      ac=1


x_inv  out    in     vdd    vss    Inverter
c_out  out    vss    1p

.ac    dec    1000     100k          10g
.END
```

Ac sweep 을 frequency 축을 decade 단위로 하여, 각 decade 마다 1000개의 data 를 찍고, 100KHZ 에서 부터 10GHz 까지의 주파수 응답을 simulation 해라

Hspice simulation

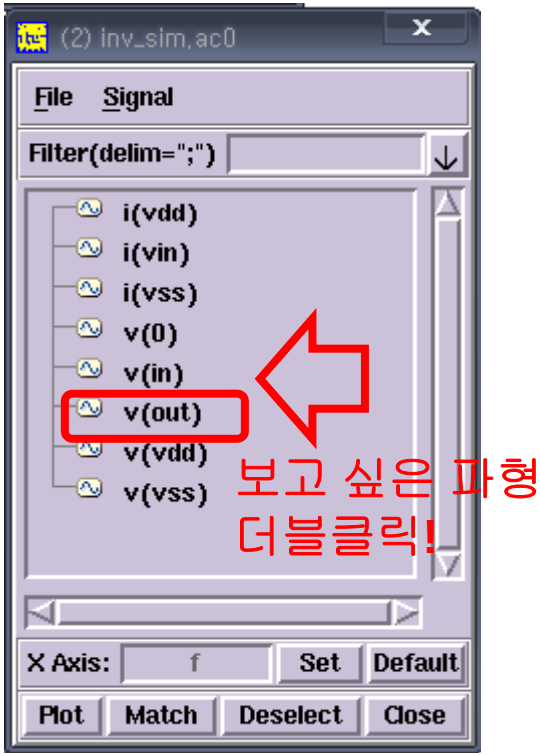
Example> Inverter 설계 및 simulation

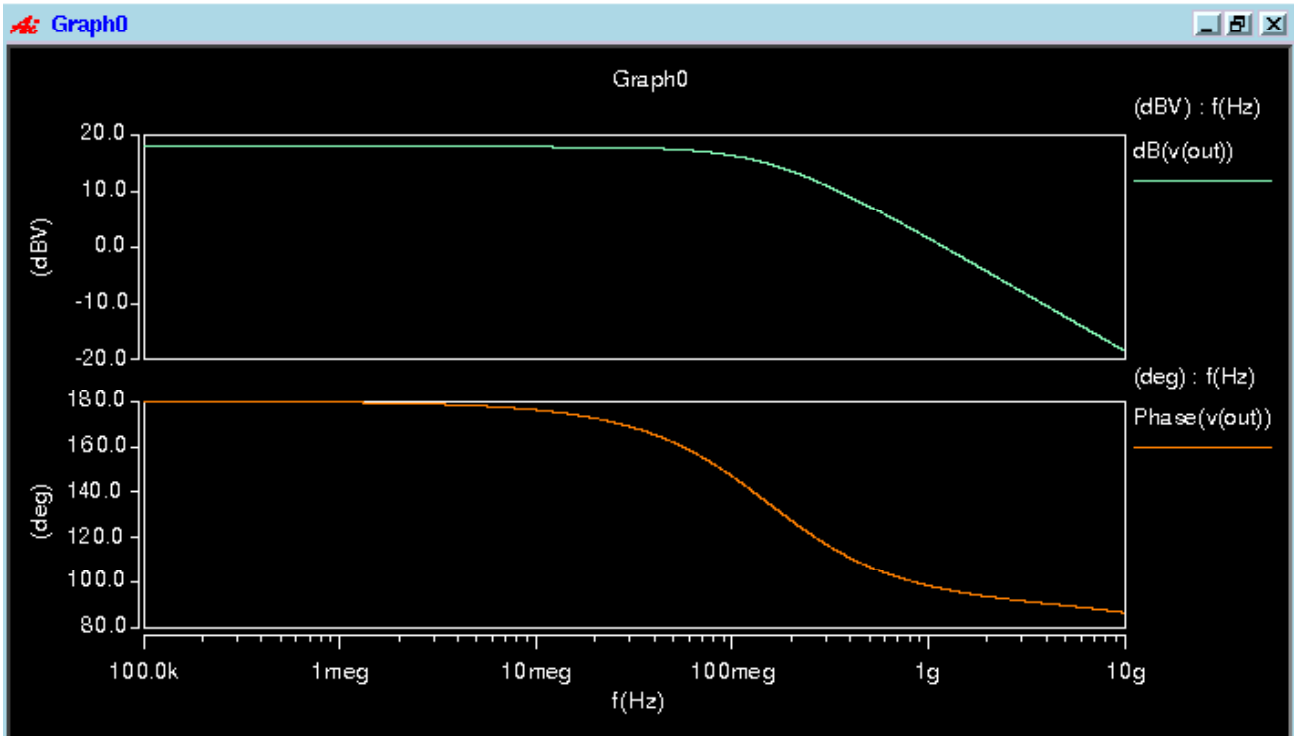
파형 확인하기



Hspice simulation

Example> Inverter 설계 및 simulation

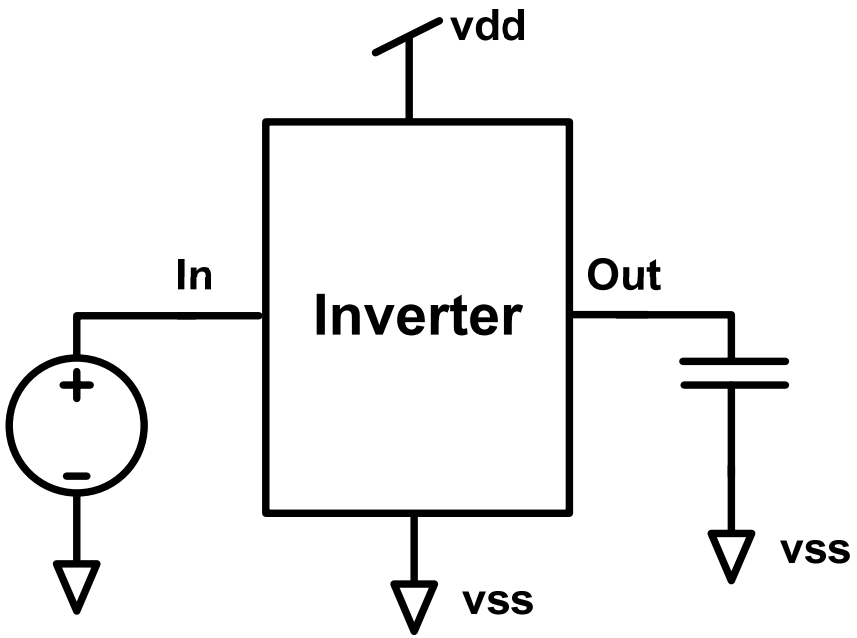




Hspice simulation

Example> Inverter 설계 및 simulation

transient simulation




```
.include ‘PMOS_VTL.inc'
.include 'NMOS_VTL.inc'
.include 'netlist.sp'

.OPTIONS POST NODE LIST

V1 vdd 0 1
V2 vss 0 0
Vin     in      vss     sin(0.5 0.5 100x)
0.5 V ± 0.5V 의 swing 을 가지는
100MHz 의 sin 파 신호 생성
x_inv   out     in      vdd     vss     Inverter
c_out   out     vss     1p

.tran       1p          100n
.END
```

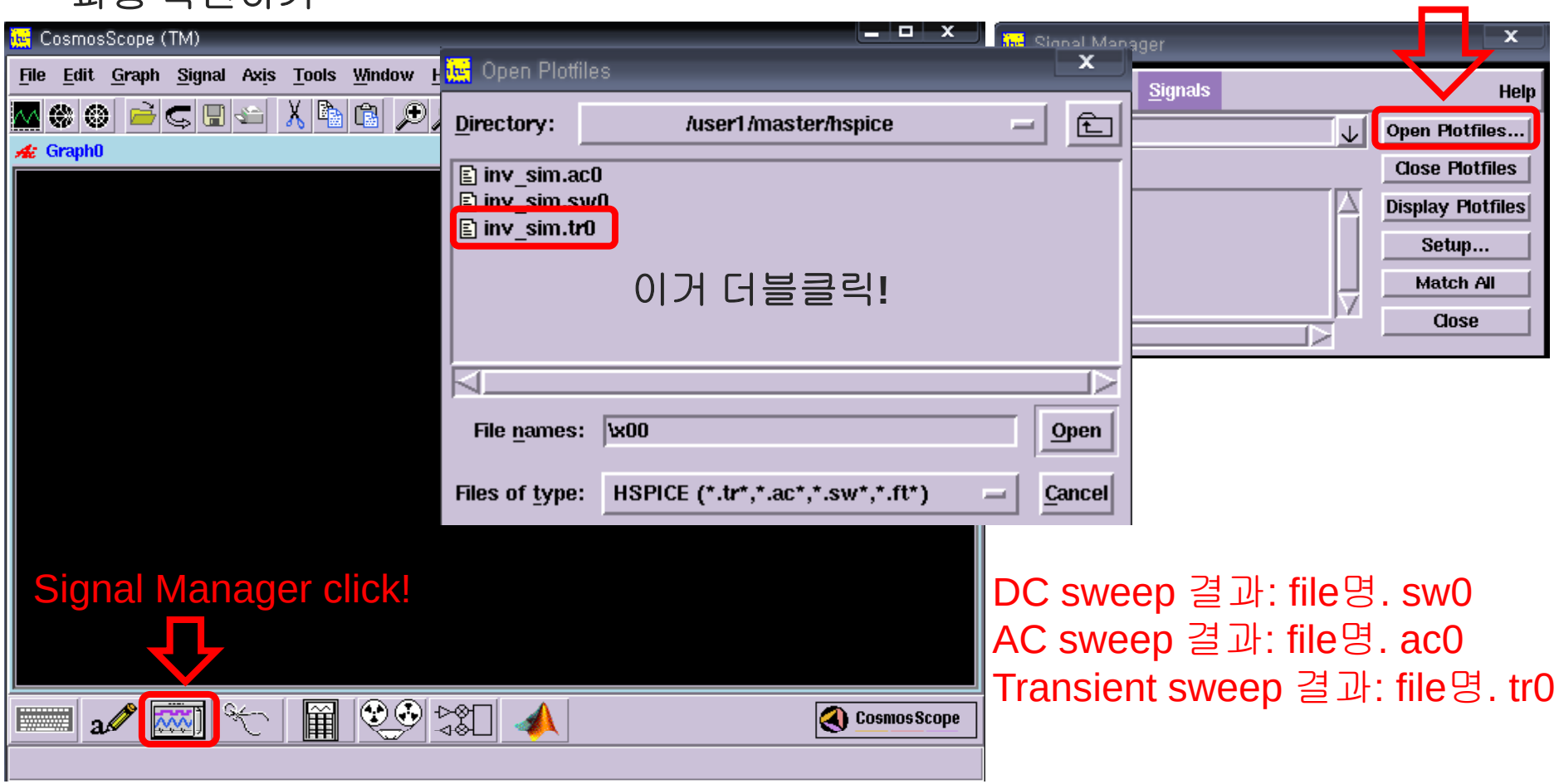
Transient sweep 을 1ps 마다 data 를 찍어서 100ns 까지 simulation 해라

Hspice simulation

Example> Inverter 설계 및 simulation

파형 확인하기

파형 열어보자!

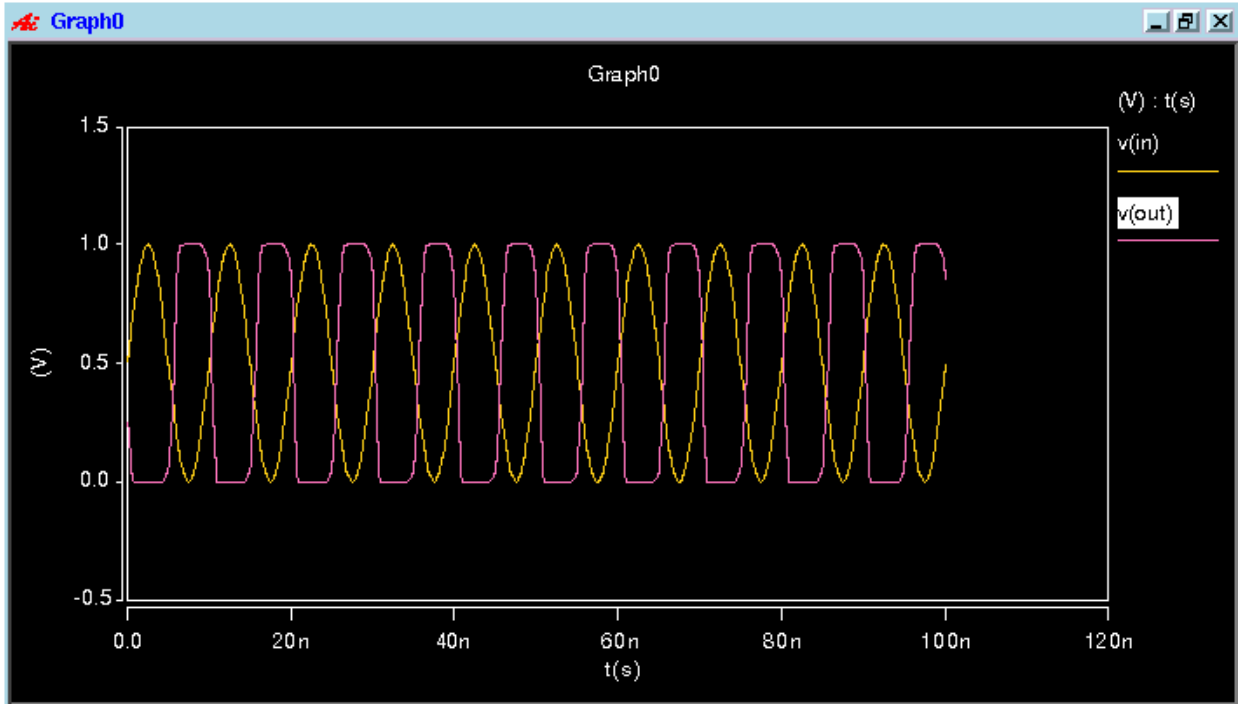


DC sweep 결과: file명. sw0
AC sweep 결과: file명. ac0
Transient sweep 결과: file명. tr0

YONSEI UNIVERSITY

Hspice simulation

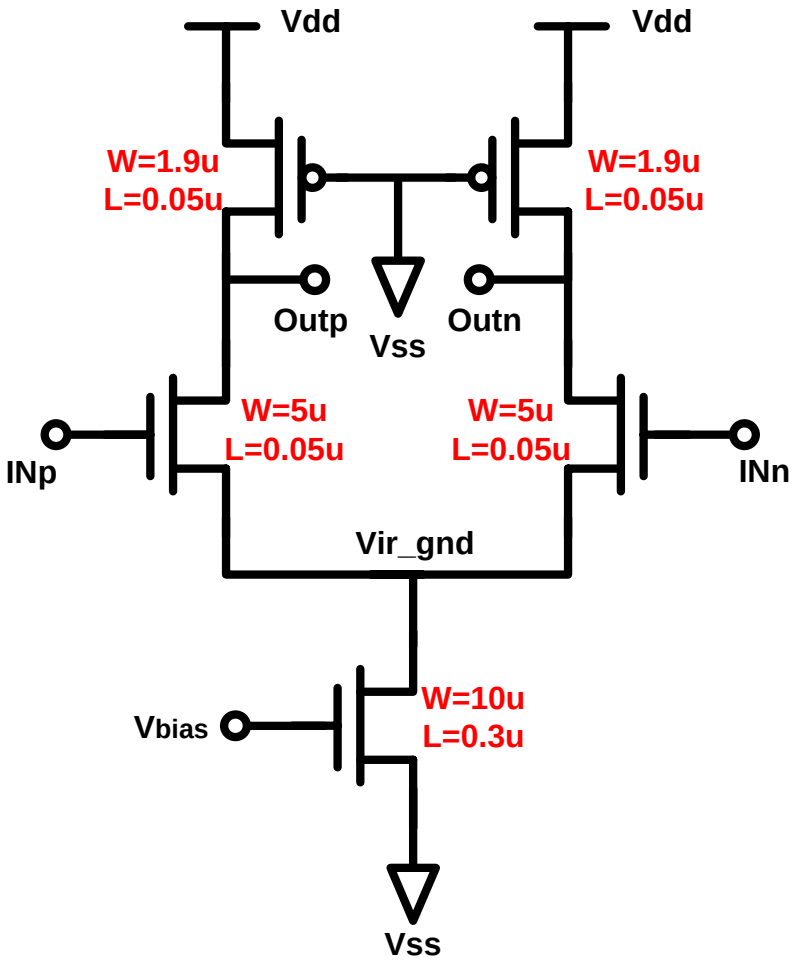
Example> Inverter 설계 및 simulation



YONSEI UNIVERSITY

실습

차동 증폭기 (differential amplifier) 의 설계 및 시뮬레이션



DC, AC, Transient sweep 후 파형 확인

Differential input signal generator

```
vin in vss ac=1
Voff off vss 0.75
Einp inp off in vss 0.5
Einn inn off in vss -0.5

Vvbias   vbias    vss      0.7

.dc vin -0.5 0.5 0.001
.ac dec 1000 100k 10g
.tran 1p 100n
```

모든 NMOS 의 bulk 는 vss 로
모든 PMOS 의 bulk 는 vdd 로 연결!

Appendix

• **Parameter 설정 및 parameter sweep**

Hspice netlist 에서 parameter 설정은 다음과 같이 한다. 예를 들어 어떤 dc voltage 값을 cont 라는 parameter (변수) 값으로 지정하고 싶다면,

```
Vcont    cont    vss    'cont'
```

와 같이 ' ' 안에 변수 값을 써 주면 된다. 그리고

```
.param   cont=0.6
```

과 같이 선언해주면 cont 에는 0.6 이란 값이 들어가게 된다.

Cont 값을 바꿔가며 parameteric sweep 을 하려면,

```
.dc      vx      0      1      0.001    sweep   cont    0       1       0.1
```

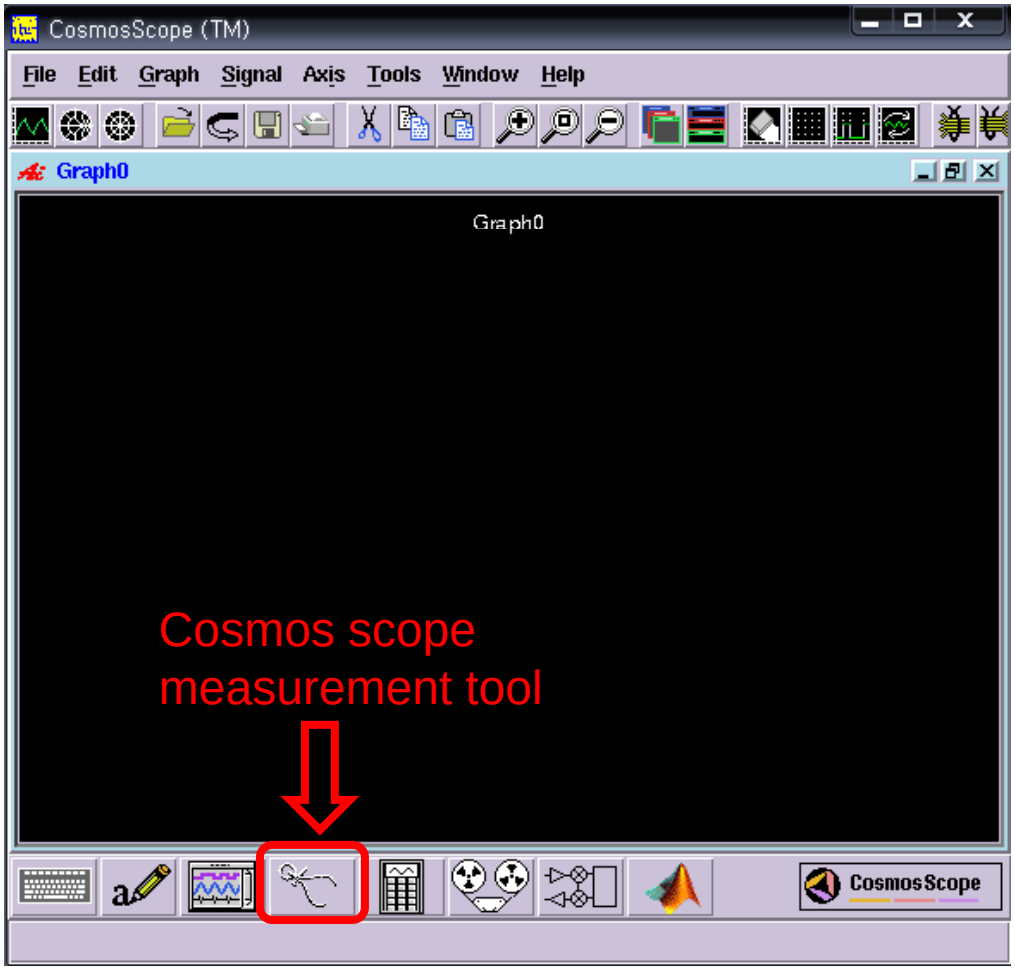
과 같이 설정해준다. 이렇게 해 주면, cont 값을 0 에서 1 까지 0.1 씩 변화시키며 10번 반복하여 dc sweep 을 하게 된다.

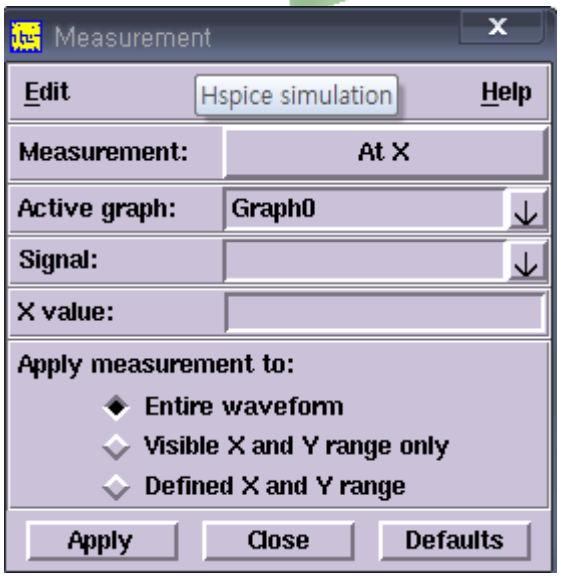
이는 ac sweep, transient sweep 에서도 마찬가지로 적용할 수 있다.

```
.ac      dec     1000   100k    10g      sweep   cont    0       1       0.1
.tran    1p      100n   sweep   cont     0       1       0.1
```

Appendix

- **Cosmos scope measurement tool**





Measurement tool 을 잘 사용하면 파형에 대한 거의 모든 정보를 얻을 수 있다. 예를 들어 파형의 주파수 혹은 swing 폭, rising time, falling time 등을 알고 싶을 땐, 직접 재려고 하지말고 measurement tool 을 이용하라