

1. Data Preparation

1. Extract LAB data from TA's directory:
`% tar xzvf ~adicta/lab05b.tar.gz`
2. The extracted LAB directory (**lab05b**) contains:
 - a. **00_library/** : HSPICE models and Cell circuits
 - b. **Demo1/** : HSPICE example 1
 - c. **Demo2/** : HSPICE example 2
 - d. **Demo3/** : HSPICE example 3
 - e. **Demo4/** : HSPICE example 4
 - f. **Exercise/** : DCO design

2. Run HSPICE Demo1

1. Change to directory: **Demo1**
2. Open and reading the HSPICE MOS models file.
`% gedit hspice_model.122 &`
How many **LIBRARY NAME** for MOSFET devices are provided in this model files? In addition, what is the **MODEL NAME** for MOSFET devices?

3. Open and reading the demo1.sp.
`% gedit demo1.sp &`
Please explain why in the measurement statement for max power, the “min” function is used rather than the “max” function? _____
4. Execute HSPICE to perform SPICE circuit simulation of demo1.sp.
`% hspice -i demo1.sp -o demo1.lis`
5. Open and reading the simulation log file
`% gedit demo1.lis &`
Search for “nodal capacitance table”, in the log file, please find out the maximum nodal capacitance is in which node? In addition, what is its nodal capacitance value? _____
6. In the simulation log file (demo1.lis), search for “transient analysis”, what is the measured value for tr, tf, th, tl, avg_power, and max_power?

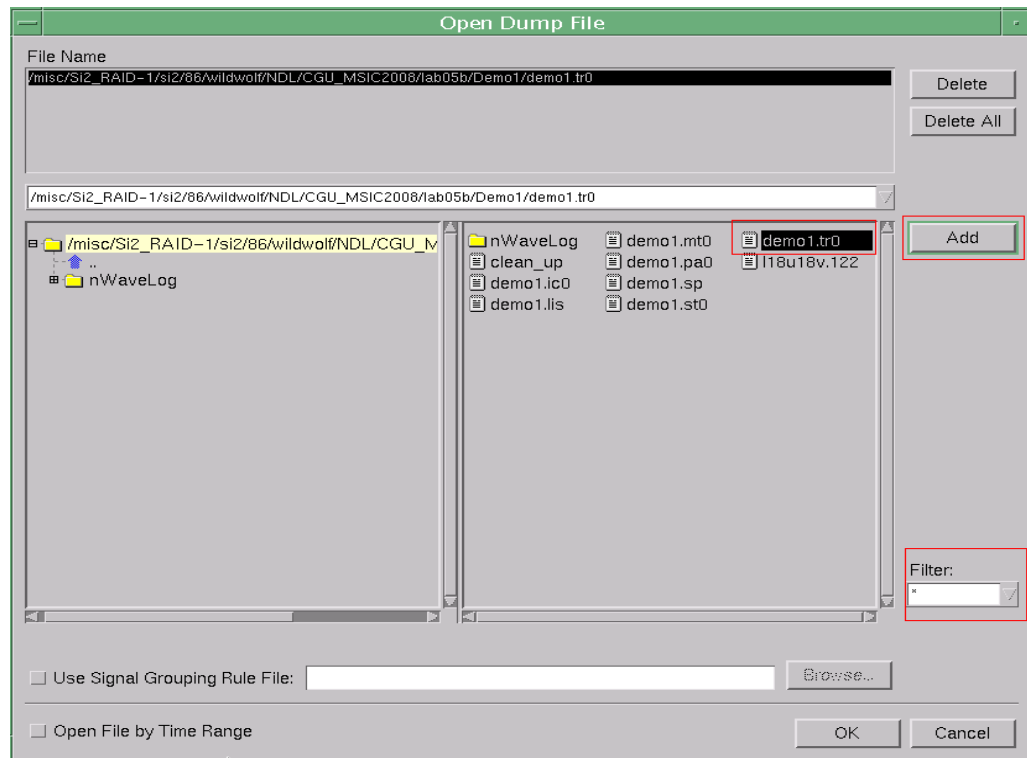
7. Open and reading the transient simulation measurement results file
`% gedit demo1.mt0 &`

What is the measured value for tr, tf, th, tl, avg_power, and max_power?

8. Open the transient simulation output (demo1.tr0) using nWave.

% nWave &

Select **File -> Open**, then change the “Filter” from “*.fsdb” to “*”. Then select demo1.tr0 and click “Add” button, and then click “OK” to complete this form.



Select **Signal -> Get Signals**. Then select v(in) v(in_d) and v(out) and press “OK” to see their waveforms.

3. Run HSPICE Demo2

1. Change to directory: **Demo2**
2. Open and reading the demo2.sp.

% gedit demo2.sp &

In this simulation, **SWEEP** is used to perform simulation for different loading parameters. Please write down the loading parameters in this simulation? _____

3. Execute HSPICE to perform SPICE circuit simulation of demo2.sp.

% hspice -i demo2.sp -o demo2.lis

4. Open and reading the simulation log file

% gedit demo2.lis &

In the end of simulation log file, you will find the measured results for

different loading, and you will see the statistical information summarized these simulations. What is the max/min values for tr, tf, th, tl, avg_power and max_power ? _____

5. Open and reading the transient simulation measurement results file

```
% gedit demo2.mt0 &
```

What is the measured value for tr, tf, th, tl, avg_power, and max_power with 0.03p loading? _____

4. Run HSPICE Demo3

1. Change to directory: **Demo3**

2. Open and reading the demo3.sp.

```
% gedit demo3.sp &
```

In this simulation, **.ALTER** is used to perform simulation for different loading parameters. Please write down the loading parameters in this simulation? _____

3. Execute HSPICE to perform SPICE circuit simulation of demo3.sp.

```
% hspice -i demo3.sp -o demo3.lis
```

4. Open and reading the simulation log file

```
% gedit demo3.lis &
```

In the simulation log file, you can see five simulations are performed, and the measured results for different loading are listed in the simulation log file.

5. Open and reading the transient simulation measurement results file

```
% gedit demo3.mt* &
```

What is the measured value for tr, tf, th, tl, avg_power, and max_power in #4 alter? In addition, what is the loading parameter in #4 alter? _____

5. Run HSPICE Demo4

1. Change to directory: **Demo4**

2. Open and reading the demo4.sp.

```
% gedit demo4.sp &
```

In this simulation, **.VEC** is used to generate digital vector input. What is the rise/fall time of vinput and what is the period of vinput? (See **demo4.vec**) _____

3. Execute HSPICE to perform SPICE circuit simulation of demo4.sp.

```
% hspice -i demo4.sp -o demo4.lis
```

4. Open and reading the simulation log file

% gedit demo4.lis &

In the end of simulation log file, you will find the measured results for different loading, and you will see the statistical information summarized these simulations. What is the max/min values for tr, tf, th, tl, avg_power and max_power ? _____

5. Open and reading the transient simulation measurement results file

% gedit demo4.mt0 &

6. What is the measured value for tr, tf, th, tl, avg_power, and max_power with 0.03p loading? _____

6. Design a Digital-Controlled Oscillator (DCO)

1. Change to directory: **Exercise**

2. Open demo cell circuits.

% gedit hspice_cells.sp &

In this SPICE file, some cells are provided to help you to speed up the design process for DCO. The functions of these cells are listed as follows:

Function Table

A	B	Y
0	x	0
x	0	0
1	1	1

AND2X1:

Function Table

A	Y
0	0
1	1

BUFX1, BUFX2, BUFX3, BUFX4:

Function Table

A	Y
0	0
1	1

CLKBUFX1, CLKBUFX2, CLKBUFX3, CLKBUFX4:

Function Table

A	Y
0	1
1	0

CLKINVX1, CLKINVX2, CLKINVX3, CLKINVX4:

Function Table

A	Y
0	1
1	0

INVX1, INVX2, INVX3, INVX4:

Function Table

S0	A	B	Y
0	0	x	0
0	1	x	1
1	x	0	0
1	x	1	1

MX2X1:

Function Table

A	B	Y
0	x	1
x	0	1
1	1	0

NAND2X1:

Function Table

OE	A	Y
0	x	Z
1	0	1
1	1	0

TBUFIX1, TBUFIX2, TBUFIX3, TBUFIX4:

Function Table

OE	A	Y
0	x	Z
1	0	0
1	1	1

TBUF1, TBUF2, TBUF3, TBUF4:

3. Design a DCO with at least 128 different frequencies output
 - a. Please use the architecture shown in **02_All-Digital_PLL.pdf, page 31** to build up your DCO. This DCO must provide at least 128 different frequencies output.
 - b. **You must perform PVT (process, voltage, temperature) simulation** to verify the operating range of your DCO, including: (SS, 1.62V, 125°C), (TT, 1.8V, 25°C), (FF, 1.98V, 0°C). Note: Operating range means the overlapped region in PVT simulation.
 - c. You must perform PVT simulation to verify the resolution of your DCO.
 - d. **Please use digital vector input** to control your DCO, the input to DCO can be a thermometer code control or any other control mechanism which make you become more easily to design your DCO.

- e. Use EXCEL or MATLAB to collect your HSPICE simulation results, and **plot the figure for DCO code vs. DCO period in PVT.**
- f. The max power and average power of your DCO should also be reported.

