

1. Data Preparation

1. Extract LAB data from TA's directory:
`% tar xzvf ~adicta/lab05d.tar.gz`
2. The extracted LAB directory (**lab05d**) contains:
 - a. **00_library/** : HSPICE Models and Cell circuits
 - b. **Demo1/** : HSPICE Reference Simulation
 - c. **Demo2/** : UltraSIM Example 2
 - d. **Demo3/** : UltraSIM Example 3
 - e. **Demo4/** : UltraSIM Example 4
 - f. **Exercise/** : PFD design

2. Run HSPICE Reference Simulation (Demo1)

1. Change to directory: **Demo1**
 2. Open and reading the demo1.sp.
`% gedit demo1.sp &`
This is a delay line circuit which has 600 inverters in the design.
 3. Execute HSPICE to perform SPICE circuit simulation of demo1.sp.
`% hspice -i demo1.sp -o demo1.lis`
 4. Open and reading the simulation log file
`% gedit demo1.lis &`
 5. 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?

 6. 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?

- Because HSPICE simulation is taken as the correct answer for BISMv3 MOS models, thus in the following UltraSIM examples, we will use the measured results from HSPICE simulation as the correct answers for comparison.
7. Open the transient simulation output (demo1.tr0) using nWave.
`% nWave &`
Select v(in) v(in_d) and v(out) to see their waveforms.

3. Run UltraSIM Demo2

1. Change to directory: **Demo2**

2. Open and reading the demo2.sp.

% gedit demo2.sp &

In this simulation, UltraSIM options is **.usim_opt sim_mode=df**. In addition, the logical probe is used to plot the v(IN) v(IN_D) v(OUT) into digital waveform.

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

% ultrasim demo2.sp

4. Open and reading the simulation log file

% gedit demo2.ulong &

5. In the simulation log file (demo2.ulong), search for “RC Reduction“, you will see the “Element Count” after “DB::RC Reduction”. How many elements are in the circuit before and after RC reduction? In addition, what are their model names?

6. In the simulation log file (demo2.ulong), search for “Device Model Statistics:”. How many NMOS use DF model? In addition, how many PMOS use DF model ?

7. In the simulation log file (demo2.ulong), search for “Partition Statistics:”, what is the partition statistics in this simulation?

8. In the simulation log file (demo2.ulong), search for “SIMULATOR OPTIONS”, UltraSim will list all simulator options for this simulation including user specified options and default options.

9. Open and reading the transient simulation measurement results file

% gedit demo2.meas0 &

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

10. 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 ?

11. Compare the UltraSIM simulation results with HSPICE simulation results

% gedit demo1.mt0 demo2.mt0 &

What is the difference between UltraSIM simulation results in DF mode and

HSPICE simulation results for tr, tf, th, tl, avg_power, and max_power ?

12. Open the transient simulation output (demo2.fsdb) using nWave.

% nWave &

Open demo2.fsdb, and select in, in_d and out in /digital to see their digital waveforms.

4. Run UltraSIM Demo3

1. Change to directory: **Demo3**

2. Open and reading the demo3_*.sp.

% gedit demo3_*.sp &

In this simulation, different netlist files are used to perform simulation for different sim_mode. The accuracy and simulation time is different for different sim_mode in UltraSIM.

3. Execute UltraSIM to perform SPICE circuit simulation of demo3_*.sp.

% ./01_run_ultrasim

The output waveforms for different simulation are demo3_01_dx.fsdb (sim_mode=dx), demo3_02_df.fsdb (sim_mode=df), demo3_03_da.fsdb (sim_mode=da), demo3_04_ms.fsdb (sim_mode=ms), demo3_05_a.fsdb (sim_mode=a), and demo3_06_s.fsdb (sim_mode=s). In addition, the measured results for different simulation are demo3_01_dx.mt0 (sim_mode=dx), demo3_02_df.mt0 (sim_mode=df), demo3_03_da.mt0 (sim_mode=da), demo3_04_ms.mt0 (sim_mode=ms), demo3_05_a.mt0 (sim_mode=a), and demo3_06_s.mt0 (sim_mode=s).

4. Open and reading the simulation log file

% gedit demo3_*.uilog &

5. In the simulation log file (demo3_*.uilog), search for “Partition Statistics:”, what is the partition statistics for different sim_mode?
-

6. In the simulation log file (demo3_*.uilog), search for “Device Model Statistics:”. What is the MOS model (DF/DA/ANALOG/SPICE) used in different sim_mode ?
-

7. Compare the UltraSIM simulation results with HSPICE simulation results

% gedit demo1.mt0 demo3_*.mt0 &

What is the difference between UltraSIM simulation results and HSPICE

simulation results for tr, tf, th, tl, avg_power, and max_power in different simulation mode?

5. Run UltraSIM Demo4

1. Change to directory: **Demo4**
2. Open and reading the demo4_*.sp.

```
% gedit demo4_*.sp &
```

In this simulation, different netlist files are used to perform simulation for different speed option. The “speed” option changes the tolerance for voltage/current change in transition analysis converge process, thus it influences the simulation accuracy and simulation time.

3. Execute UltraSIM to perform SPICE circuit simulation of demo4_*.sp.

```
% ./01_run_ultrasim
```

The output waveforms for different “speed” simulation are demo4_01_speed5.fsdb (speed=5), demo4_02_speed4.fsdb (speed=4), demo4_03_speed3.fsdb (speed=3), demo4_04_speed2.fsdb (speed=2), and demo4_05_speed1.fsdb (speed=1). In addition, the measured results for different simulation are demo4_01_speed5.mt0 (speed=5), demo4_02_speed4.mt0 (speed=4), demo4_03_speed3.mt0 (speed=3), demo4_04_speed2.mt0 (speed=2), and demo4_05_speed1.mt0 (speed=1).

4. Open and reading the simulation log file

```
% gedit demo4_*.ulog &
```

5. In the simulation log file (demo4_*.ulog), search for “Partition Statistics:”, what is the partition statistics for different speed(=1,2,3,4,5)?

6. In the simulation log file (demo4_*.ulog), search for “Device Model Statistics:”. What is the MOS model (DF/DA/ANALOG/SPICE) used in different

speed(=1,2,3,4,5)?

7. Compare the UltraSIM simulation results with HSPICE simulation results

```
% gedit demo1.mt0 demo4_*.mt0 &
```

What is the difference between UltraSIM simulation results and HSPICE simulation results for tr, tf, th, tl, avg_power, and max_power in different speed(=1,2,3,4,5)?

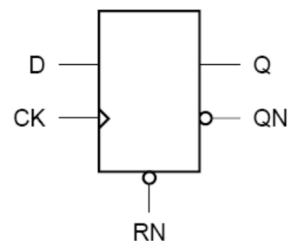
8. In this circuit, because it is a delay line with many inverters which have longer delay path than typical circuits. As a result, the speed option which determined the re-calculation in transition analysis is more important than MOS models. Thus in this circuit simulation, we can use **sim_mode=da**, and **speed=1** to get a better simulation accuracy than use **sim_mode=s**, and **speed=5**. You must learn from this lab, that the simulation mode and simulation option will have a large influence on the simulation results of the Fast-SPICE simulator. You can also try other options to get better simulation accuracy with shorter simulation time.

6. Design a Phase and Frequency Detector (PFD)

1. Change to directory: **Exercise**
2. Open demo cell circuits.

```
% gedit ultrasim_cells.sp &
```

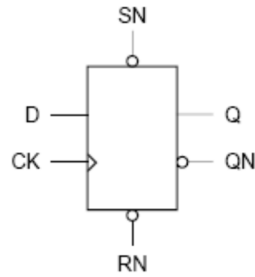
In this SPICE file, some cells are provided to help you to speed up the design process for PFD. Combinational cells are the same as in **lab05b**. The functions of the sequential cells are listed as follows:



DFFRX4:

Function Table

RN	D	CK	Q[n+1]	QN[n+1]
0	x	x	0	1
1	0		0	1
1	1		1	0
1	x		Q[n]	QN[n]



Function Table

RN	SN	D	CK	Q[n+1]	QN[n+1]
0	1	x	x	0	1
1	0	x	x	1	0
0	0	x	x	1	0
1	1	0		0	1
1	1	1		1	0
1	1	x		Q[n]	QN[n]

DFFSRX4:

3. Design a PFD with a smaller dead zone
 - a. Please use the architecture shown in **02_All-Digital_PLL.pdf, page 44** to build up your PFD. The PFD must have a dead zone smaller than 60ps in the worst case.
 - b. The rise time and fall time of input signals are 0.1ns.
 - c. **You must perform PVT (process, voltage, temperature) simulation** to verify the dead zone of your PFD, including: (SS, 1.62V, 125°C), (TT, 1.8V, 25°C), (FF, 1.98V, 0°C).
 - d. **Please use digital vector input** to simulate your PFD, and use UltraSIM with **sim_mode=ms, speed=5**, to perform the simulation of your PFD.
 - e. Please determine the max speed of input signals of your PFD.
 - f. The max power and average power of your PFD should also be reported.