

11
YEARS
OF UNIVERSITY
RECOGNITION
21 YEARS OF
ACADEMIC
EXCELLENCE



REVA
UNIVERSITY
Bengaluru, India



Semester - 6
Course Code - B20EN0606

School of Electronics & Communications Engineering

B.Tech. in Electronics & Communication Engineering (ECE)

CMOS VLSI Circuits Lab

2021-25

**RUKMINI EDUCATIONAL
Charitable Trust**

www.reva.edu.in

Vision of the University

"REVA University aspires to become an innovative university by developing excellent human resources with leadership qualities, ethical and moral values, research culture and innovative skills through higher education of global standards"

Mission of the University

- To create excellent infrastructure facilities and state-of-the-art laboratories and incubation centres
- To provide student-centric learning environment through innovative pedagogy and education reforms
- To encourage research and entrepreneurship through collaborations and extension activities
- To promote industry-institute partnerships and share knowledge for innovation and development
- To organize society development programs for knowledge enhancement in thrust areas
- To enhance leadership qualities among the youth and enrich personality traits, promote patriotism and moral values.

Vision of the School

The School of Electronics and Communication Engineering is envisioned to be a leading centre of higher learning with academic excellence in the field of electronics and communication engineering blended by research and innovation in tune with changing technological and cultural challenges supported with leadership qualities, ethical and moral values.

Mission of the School

- Establish a unique learning environment to enable the students to face the challenges in the field of Electronics and Communication Engineering and explore multidisciplinary which serve the societal requirements.
- Create state-of-the-art laboratories, resources, and exposure to the current industrial trends to enable students to develop skills for solving complex technological problems of current times and provide a framework for promoting collaborative and multidisciplinary activities.
- Promote the establishment of Centres of Excellence in niche technology areas to nurture the spirit of innovation and creativity among faculty and students.
- Offer ethical and moral value-based education by promoting activities which inculcate the leadership qualities, patriotism and set high benchmarks to serve the society

Program Educational Objectives (PEOs)

The Program Educational Objectives of B. Tech in Electronics and Communication Engineering are as follows:

- PEO -1: To have successful professional careers in industry, government, academia, and military as innovative engineers.
- PEO -2: To successfully solve engineering problems associated with the lifecycle of Electronics and Communication Systems either leading a team or as a team member.
- PEO -3: To continue to learn and advance their careers through activities such as participation in professional organizations, attainment of professional certification for lifelong learning and seeking higher education.
- PEO -4: To be active members ready to serve the society locally and internationally and will undertake entrepreneurship for the growth of economy and to generate employment.

Program Outcomes (POs)

On successful completion of the program, the graduates of B. Tech. (Electronics and Communication Engineering) program will be able to:

- **PO-1: Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals for the solution of complex problems in Electronics and communication Engineering.
- **PO-2: Problem analysis:** Identify, formulate, research literature, and analyze engineering problems to arrive at substantiated conclusions using first principles of mathematics, natural, and engineering sciences.
- **PO-3: Design/development of solutions:** Design solutions for complex engineering problems and design system components, processes to meet the specifications with consideration for the public health and safety, and the cultural, societal, and environmental considerations.
- **PO-4: Conduct investigations of complex problems:** Use research-based knowledge including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
- **PO-5: Modern tool usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
- **PO-6: The engineer and society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal, and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
- **PO-7: Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
- **PO-8: Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
- **PO-9: Individual and team work:** Function effectively as an individual, and as a member or leader in teams, and in multidisciplinary settings.
- **PO-10: Communication:** Communicate effectively with the engineering community and with society at large. Be able to comprehend and write effective reports documentation. Make effective presentations, and give and receive clear instructions.
- **PO-11: Project management and finance:** Demonstrate knowledge and understanding of engineering and management principles and apply these to one's own work, as a member and leader in a team. Manage projects in multidisciplinary environments.
- **PO-12: Life-long learning:** Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

Programme Specific Outcomes (PSOs)

On successful completion of the program, the graduates of B. Tech. (Electronics and Communication Engineering) program will be able to:

PSO-1: Isolate and solve complex problems in the domains of Electronics and Communication Engineering using latest hardware and software tools and technologies, along with analytical and managerial skills to arrive at cost effective and optimum solutions either independently or as a team.

PSO-2: Implant the capacity to apply the concepts of electronics, communications, signal processing, VLSI, embedded systems, etc. in the design, development and implementation of application oriented engineering systems.

PSO-3: Design, Model, Analyze and Build Electronics and Communication Systems to solve real life and industry problems.

CONTENTS

SI. No.	PRACTICE SESSION	Page No.
PART A:		
General procedure for Cadence Design Tool Digital design		5
1	Write Verilog Code for CMOS Inverter and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation	8
2	Write Verilog Code for CMOS Buffer and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation	9
3	Write Verilog Code for transmission gate and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation	10
4	Write Verilog Code for basic/universal gate and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation	11
5	Write Verilog Code flip flops-RS,D,JK,MS,T and their Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation	14
6	Write Verilog Code for serial and parallel adder and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation	20
7	Write Verilog Code for 4-bit counter (synchronous and asynchronous counter) and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation.	22
8	Write Verilog Code for adder circuits(full adder cascading to build 4- bit parallel adder-RCA) and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation	24
PART B:		
General Notes for Cadence Design Tool Analog Virtuoso		31
1	Design the circuit of CSA with given specifications, completing the design flow mentioned below: a. Draw the schematic and verify the following i) DC Analysis ii) AC Analysis iii) Transient Analysis b. Draw the Layout and verify the DRC, ERC c. Check for LVS	33
2	Design the circuit of CDA with given specifications, completing the design flow mentioned below: a. Draw the schematic and verify the following i) DC Analysis ii) AC Analysis iii) Transient Analysis b. Draw the Layout and verify the DRC, ERC c. Check for LVS.	57
3	Design an op-amp with given specification using given differential amplifier Common source amplifier in library and completing the design flow mentioned below: a. Draw the schematic and verify the following i) DC Analysis ii). AC Analysis iii) Transient Analysis b. Draw the Layout and verify the DRC, ERC	62
4	Design a 4 bit R-2R based DAC for the given specification and completing the design flow mentioned using given op-amp in the library. a. Draw the schematic and verify the following i) DC Analysis ii) AC Analysis iii) Transient Analysis b. Draw the Layout and verify the DRC, ERC	67

Course Title	CMOS VLSI Circuits lab				Course Type		HC	
Course Code	B20EN0606	Credits	01		Class		VII Semester	
	LTP	Credits	Contact Hours	Work Load	Total Number of Classes Per Semester		Assessment in Weightage	
	Lecture	-	-	-				
	Tutorial	-	-	-	Theory	Practical	IA	SEE
	Practical	1	2	2				
	Total	1	2	2	-	28	50%	50%

COURSE OVERVIEW:

The course introduces basic theories and techniques of digital VLSI design using CMOS and its variants. The student will understand how the digital circuits can be integrated into the semiconductor chip (ICs). The students will develop the skills required to become VLSI designers, researchers, and design tool builders. The course is conceptual, problematic and application oriented.

COURSE OBJECTIVES:

The objectives of this course are:

- Understand the design of sequential and combinational circuit design using Verilog HDL
- Illustrate the power, delay and area estimation of CMOS circuits using CADENCE tool
- Develop the CMOS Digital and Analog circuits using schematic and layout
- Study post layout RC extraction and power analysis process

COURSE OUTCOMES (COs)

On successful completion of this course; the student shall be able to:

CO#	Course Outcomes	POs	PSOs
CO1	Develop the digital circuits using Verilog HDL, perform Area, power, and delay analysis.	1,2,3,4,5	1,2
CO2	Demonstrate the physical design process of Digital Integrated circuits using ASIC Design flow	1,2,3,4,5,9,10	1,2,3
CO3	Design the analog CMOS circuits and explore, analyze the electrical characteristics using CADENCE tool	1,2,3,4,5,9,10	1,2,3
CO4	Develop schematic of various digital and analog CMOS circuits and perform various simulations using CADENCE tools	1,2,3,4,5,9,10	1,2,3
CO5	Design layout of various digital and analog CMOS circuits and perform various simulations using CADENCE tools	1,2,3,4,5,9,10	1,2,3
CO6	Illustrate the post layout RC and power estimation of CMOS circuits using CADENCE tools	1,2,3,4,5,9,10	1,2,3

BLOOM'S LEVEL OF THE COURSE OUTCOMES

CO#	Bloom's Level					
	Remember (L1)	Understand (L2)	Apply (L3)	Analyze (L4)	Evaluate (L5)	Create (L6)
CO1	✓	✓	✓	✓		

CO2	✓	✓	✓	✓			
CO3	✓	✓	✓	✓	✓		
CO4	✓	✓	✓	✓	✓		
CO5	✓	✓	✓	✓	✓		
CO6	✓	✓	✓	✓	✓		

COURSE ARTICULATION MATRIX

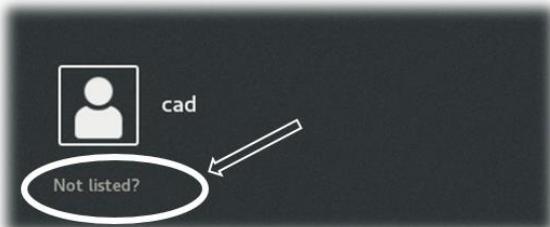
CO#/ POs	PO1	PO2	PO3	PO4	PO5	PO6	PO7	PO8	PO9	PO10	PO11	PO12	PSO1	PSO2	PSO3
CO1	1	2	3	3	3				2	2			1	2	3
CO2	1	2	3	3	3				2	2			1	2	
CO3	2	1	3	3	3				2	2			1	2	
CO4	3	2	3	3	3				2	2			1	2	3
CO5	3	2	3	3	3				2	2			1	2	
CO6	1	2	3	3	3				2	2			2	1	

Note:1-Low,2-Medium,3-High

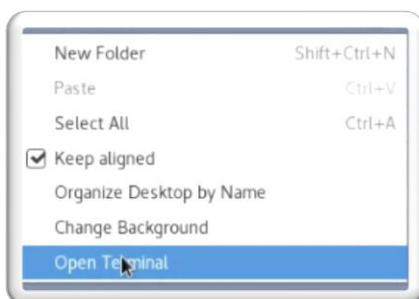
Part A: Digital Design

General procedure for Digital design:

1. Log in to workstation by clicking on **Not listed?**



2. User name: **root**
3. Password: **root123**
4. Right click on the Desktop
5. Click on Open Terminal



6. Type **csh** <press enter button>
7. Type **source ~/cshrc** <press enter button> the “**Welcome to cadence tools**” will display in the next line.
8. Type **cd labs**<press enter button> changes the directory to labs (folder).
9. Type **cd digital** <press enter button> changes the directory to digital(folder)
10. Type **cd <batch_name>** (ex: a1batch, b1batch etc..) <press enter button> changes to directory_name(folder)
11. Type **gedit <file_name.v>** <press enter button> (file name followed with the extension .v(dot v) ex: **inv.v**)
12. repeat the step 11 to create test bench gedit <file_name_test.v>)

```

root@cad24:~#
[root@cad24 ~]# csh
[root@cad24 ~]# source ~/cshrc

Welcome to Cadence Tools Suite

[root@cad24 ~]# cd labs
[root@cad24 ~/labs]# cd digital
[root@cad24 digital]# cd albatch
[root@cad24 albatch]# gedit inv.v

```

13. Type the program, click on save, go to file menu click on quit.



Procedure for simulation:**command for simulation:**

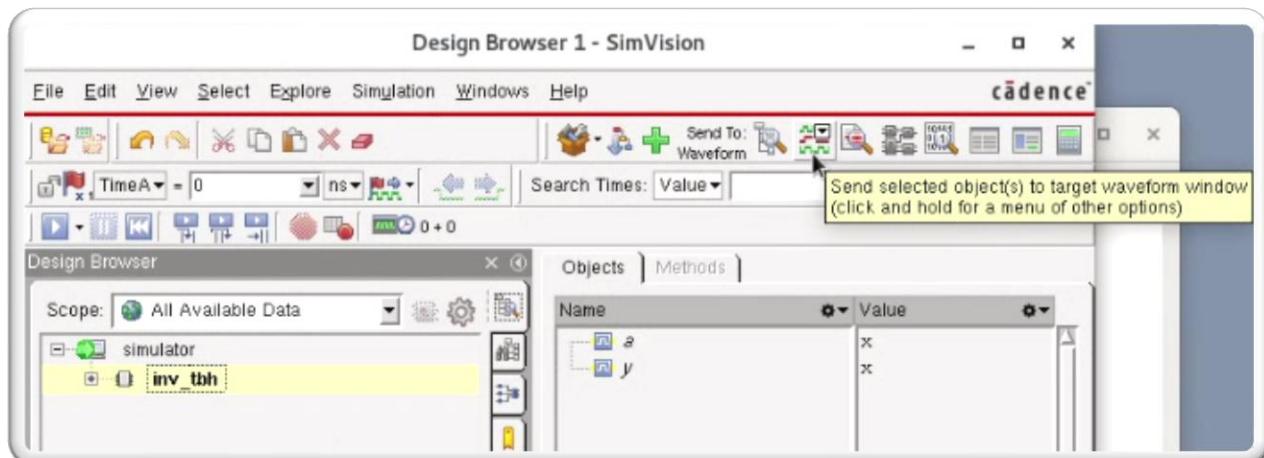
```
albatch]# irun inv.v -access +rwc -mess
```

```
irun <filename.v> -access +rwc -mess
```

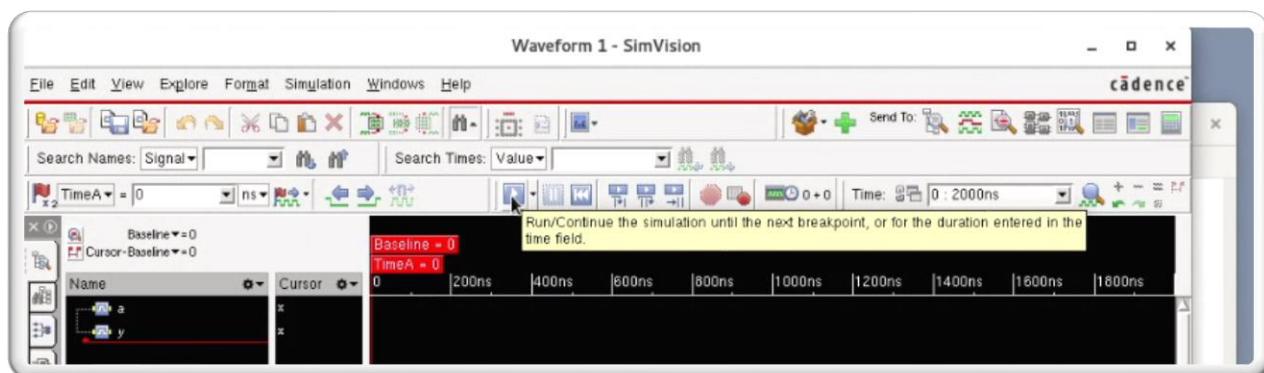
```
albatch]# irun inv_tb.v -access +rwc -mess -gui
```

```
irun <filename_test.v> -access +rwc -mess -gui
```

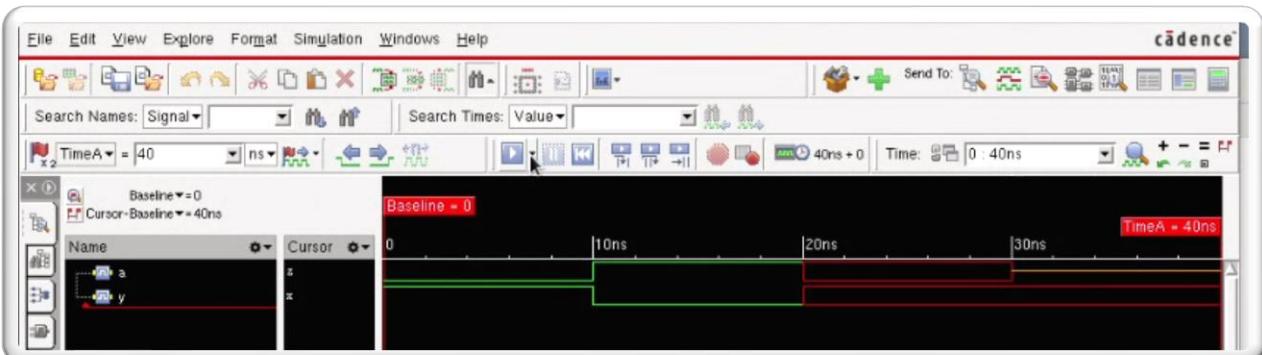
1. Click on the icon Send selected object(s) to target waveform window (click and hold for a menu of other options)



2. Click on the icon Run/Continue the simulation until the next breakpoint, or for the duration entered in the time field.



3. The o/p wave form window opens.



Procedure for synthesis (gate level only):

(top level program and test bench should be in separate file)

close all the windows except terminal window, then in terminal window type the following:

```

genus
set_db library slow_normal.lib
read_hdl <filename.v> (top level program) / for vhdl code: read_hdl -vhdl <filename.vhd>
check_design –unresolved (optional)
elaborate
read_sdc constraints.sdc (optional for combinational circuits)
synthesize -to_mapped -effort medium
report area
report timing -unconstrained (for gates and combinational circuits)
report timing (for sequential circuits)
report power
report gates
write_hdl > <filename_netlist.v>
gui_show
gui_hide
exit
#####
# slow_normal.lib file should be in the same folder where the .v files are kept.
# constraints_top.g file(which is in /rclabs/work) is to be renamed as constraints.sdc and this file should be
kept in the same folder where the .v files are kept, and then edit the i/o pins, clk etc. in this file.
#for synthesize the top module file and test bench should be in different files.
#####
the constraints file contents are:
create_clock -name clk -period 10 -waveform {0 5} [get_ports "clk"]
set_clock_transition -rise 0.1 [get_clocks "clk"]
set_clock_transition -fall 0.1 [get_clocks "clk"]
set_clock_uncertainty 1.0 [get_ports "clk"]
set_input_delay -max 1.0 [get_ports "M"] -clock [get_clocks "clk"]
set_input_delay -max 1.0 [get_ports "rst"] -clock [get_clocks "clk"]
set_input_delay -max 1.0 [get_ports "rw"] -clock [get_clocks "clk"]
set_input_delay -max 1.0 [get_ports "data_in"] -clock [get_clocks "clk"]
set_input_delay -max 1.0 [get_ports "addr_load"] -clock [get_clocks "clk"]
set_input_delay -max 1.0 [get_ports "load"] -clock [get_clocks "clk"]
set_output_delay -max 1.0 [get_ports "data_out"] -clock [get_clocks "clk"]

```

in place of "M" "rst" "rw" "data_in" "addr_load" "load" are the editable names as for ex: dff:- M-clk, rst-n_rst, data_in-din, data_out-q like this we have to comment the line if the name is not there.

1. Write Verilog Code for CMOS Inverter and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation

CMOS Inverter:

```
module inv(a,y);
  input a;
  output y;

  assign y = ~a;

endmodule
```

Test bench:

```
module invtb;
  reg a;
  wire y;

  inv i1(a,y);

  initial
    begin
      a = 0; #10;
      a = 1; #10;
      #50 $finish;
    end

  endmodule
```

2. Write Verilog Code for CMOS Buffer and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation

```
module buffer (a,y);
input a;
output y;
assign y=a;
endmodule
```

Test Bench:

```
module buffer_tb;
reg a;
wire y;

buffer i1(a,y);

initial
begin
a=0; #10;
a=1; #10;
a=0; #10;
a=1;
end
endmodule
```

3. Write Verilog Code for Transmission gate and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation

```
module tg(a,ctrl,y);
input a,ctrl;
output y;
reg y;
always @(ctrl or a)
begin if(ctrl==1)
y<=a;
else
y<=1'bz;
end
endmodule
```

Test Bench:

```
module tg_tb;
wire y;
tg tg_i1(a,ctrl,y);
initial
begin a=0; #10
a=1; #10;
a=0; #10 ;
a=1; #5;
$finish
end
initial
begin
ctrl=0; #15;
ctrl=1; #15;
ctrl=0; #25;
ctrl=1; #5;
$finish;
end
endmodule
```

4. Write Verilog Code for basic/universal gate and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation

AND gate:

```
module and_gate(a,b,y);
    input a,b;
    output y;
    assign y = a & b;
endmodule
```

Test Bench:

```
module and_gate_tb;
    reg a,b;
    wire y;
    and_gate i1(a,b,y);
    initial
    begin
        a = 0; b = 0; #10;
        b = 0; b = 1; #10;
        a = 1; b = 0; #10;
        b = 1; b = 1; #10;
        #50; $finish;
    end
endmodule
```

OR gate:

```
module or_gate_d(a,b,y);
    input a,b;
    output y;
    assign y = a | b;
endmodule
```

Test Bench:

```
module or_gate_tb;
    reg a,b;
    wire y;
    or_gate i1(a,b,y);
    initial
    begin
        a = 0; b = 0; #10;
        b = 0; b = 1; #10;
        a = 1; b = 0; #10;
        b = 1; b = 1; #10;
        #50; $finish;
    end
endmodule
```

NAND gate:

```
module nand_gate(a,b,y);
    input a,b;
    output y;
    assign y = ~(a & b);
endmodule
```

Test Bench:

```
module nand_gate_tb;
    reg a,b;
    wire y;
    nand_gate i1(a,b,y);
    initial
    begin
        a = 0; b = 0; #10;
        b = 0; b = 1; #10;
        a = 1; b = 0; #10;
        b = 1; b = 1; #10;
        #50; $finish;
    end
endmodule
```

NOR gate:

```
module nor_gate(a,b,y);
    input a,b;
    output y;
    assign y = ~(a | b);
endmodule
```

Test Bench:

```
module nor_gate_tb;
    reg a,b;
    wire y;
    nor_gate i1(a,b,y);
    initial
    begin
        a = 0; b = 0; #10;
        b = 0; b = 1; #10;
        a = 1; b = 0; #10;
        b = 1; b = 1; #10;
        #50; $finish;
    end
endmodule
```

EX-OR gate:

```
module xor_gate(a,b,y);
    input a,b;
    output y;
    assign y = a ^ b;
endmodule
```

Test Bench:

```
module xor_gate_tb;
    reg a,b;
    wire y;
    xor_gate i1(a,b,y);
    initial
    begin
        a = 0; b = 0; #10;
        b = 0; b = 1; #10;
        a = 1; b = 0; #10;
        b = 1; b = 1; #10;
        #50; $finish;
    end
endmodule
```

EX-NOR gate:

```
module xnor_gate(a,b,y);
    input a,b;
    output y;
    assign y = ~(a ^ b);
endmodule
```

Test Bench:

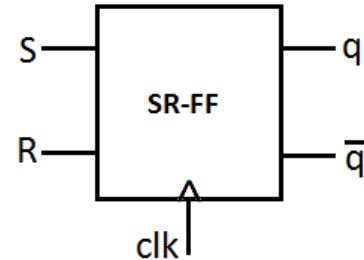
```
module xnor_gate_tb;
    reg a,b;
    wire y;
    xnor_gate i1(a,b,y);
    initial
    begin
        a = 0; b = 0; #10;
        b = 0; b = 1; #10;
        a = 1; b = 0; #10;
        b = 1; b = 1; #10;
        #50; $finish;
    end
endmodule
```

5. Write Verilog Code flip flops-RS, D, JK, MS, T and their Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation.

SR Flip-flop:

```
module SR_ff(q,qbar,s,r,clk);
output q,qbar;
input clk,s,r;
reg tq;

always @(posedge clk or tq)
begin
    if (s == 1'b0 && r == 1'b0)
        tq <= tq;
    else if (s == 1'b0 && r == 1'b1)
        tq <= 1'b0;
    else if (s == 1'b1 && r == 1'b0)
        tq <= 1'b1;
    else if (s == 1'b1 && r == 1'b1)
        tq <= 1'bx;
end
assign q = tq;
assign qbar = ~tq;
endmodule
```



Test Bench:

```
module SR_ff_test;
reg clk,s,r;
wire q,qbar;

SR_ff sr1(q,qbar,s,r,clk);

initial
clk = 1'b0;

always
#10 clk = ~clk;

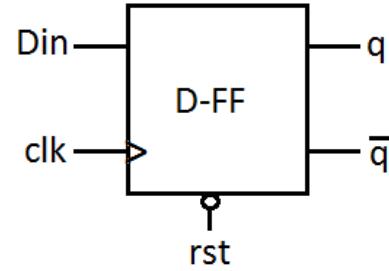
initial
begin
s = 1'b0; r = 1'b0;
#30 s = 1'b1;
#29 s = 1'b0;
#1 r = 1'b1;
#30 s = 1'b1;
#30 r = 1'b0;
#20 s = 1'b0;
#19 s = 1'b1;
```

Truth Table					
clk	S	R	q	qbar	
↑	0	0	Previous state		
↑	0	1	0	1	reset
↑	1	0	1	0	set
↑	1	1	forbidden		

```
#200 s = 1'b1; r = 1'b1;  
#50 s = 1'b0; r = 1'b0;  
#50 s = 1'b1; r = 1'b0;  
#10 ;  
end  
  
initial  
#500 $finish;  
  
endmodule
```

D Flip-Flop:

```
Module d_ff(q,clk,n_rst,din);
    output q;
    input clk,din,n_rst;
    reg q;
    always @(posedge clk or negedge n_rst)
    begin
        if(!n_rst)
            q <= 1'b0;
        else
            q <= din;
    end
endmodule
```



Truth Table				
rst	clk	Din	q	qbar
0	↓	x	0	1
1	↓	1	1	0
1	↓	0	0	1

Test Bench:

```
module d_ff_test;
    reg clk, din, n_rst;
    wire q;
    d_ff df1 (q, clk, n_rst, din);

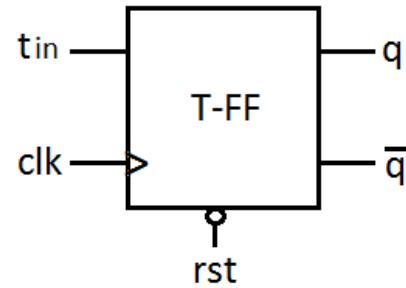
    initial
        clk = 1'b0;
    always
        #10 clk = ~clk;
    initial
        begin
            din = 1'b0;
            n_rst = 1'b1;
            #20 n_rst = 1'b0;
            #10 din = 1'b1;
            #20 n_rst = 1'b1;
            #18 din = 1'b0;
            #1 din = 1'b1;
            #20 din = 1'b0;
            #10 ;
        end
        initial
            #100 $finish;
    endmodule
```

T Flip-Flop:

```

module t_ff(q,qbar,clk,tin,rst);
    output q,qbar;
    input clk,tin,rst;
    reg tq;
    always @(posedge clk or negedge rst)
    begin
        if(!rst)
            tq <= 1'b0;
        else
            begin
                if (tin)
                    tq <= ~tq;
            end
        end
        assign q = tq;
        assign qbar = ~q;
    endmodule

```



Truth Table			
rst	clk	tin	q
0	↑	x	0
1	↑	0	q
1	↑	1	toggle

Test Bench:

```

module t_ff_test;
    reg clk,tin,rst;
    wire q,qbar;
    t_ff t1(q,qbar,clk,tin,rst);
    initial
        clk = 1'b0;
    always
        #10 clk = ~clk;
    initial
    begin
        rst = 1'b0; tin = 1'b0;
        #30 rst = 1'b1;
        #10 tin = 1'b1;
        #205 tin = 1'b0;
        #300 tin = 1'b1;
        #175 tin = 1'b0;
        #280 rst = 1'b0;
        #20 rst = 1'b1;
        #280 tin = 1'b1;
        #10 ;
    end
    initial
        #2000 $finish;
endmodule

```

JK Flip-Flop:

```

module jk_ff(q,qbar,clk,rst,j,k);
input clk,rst,j,k;
output q,qbar;
reg q,tq;
begin
    always @(posedge clk or negedge rst)
        begin
            if (!rst)
                begin
                    q <= 1'b0;
                    tq <= 1'b0;
                end
            else
                begin
                    if (j == 1'b1 && k == 1'b0)
                        q <= j;
                    else if (j == 1'b0 && k == 1'b1)
                        q <= 1'b0;
                    else if (j == 1'b1 && k == 1'b1)
                        begin
                            tq <= ~tq;
                            q <= tq;
                        end
                    end
                end
            assign qbar = ~q;
        endmodule

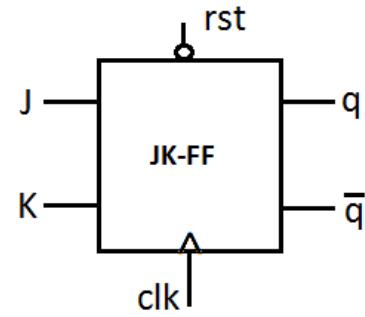
```

Test Bench:

```

module jk_ff_test;
reg clk,rst,j,k;
wire q,qbar;
jk_ff inst(q,qbar,clk,rst,j,k);
initial
clk=1'b0;
always
#10 clk=~clk;
initial
begin
j=1'b0;k=1'b0;rst=1'b0;
#10 rst=1'b1;
#10 j=1'b0; k=1'b0;
#10 j=1'b0; k=1'b1;
#10 j=1'b1; k=1'b0;
#50 j=1'b1; k=1'b1;
#10 j=1'b0; k=1'b0;
#50 j=1'b1; k=1'b1;
#10;
end
initial
#300 $finish;
endmodule

```



Truth Table						
clk	rst	J	K	q	qbar	
↓	0	x	x	0	1	
↓	1	0	0			previous state
↑	1	0	1	0	1	
↓	1	1	0	1	0	
↓	1	1	1			toggle

MS JK Flip-Flop:

```
module ms_jkff(q,q_bar,clk,j,k);
output q,q_bar;
input clk,j,k;
reg tq,q,q_bar;
always @(clk)
begin
if (!clk)
begin
if (j==1'b0 && k==1'b1)
tq <= 1'b0;
else if (j==1'b1 && k==1'b0)
tq <= 1'b1;
else if (j==1'b1 && k==1'b1)
tq <= ~tq;
end
if (clk)
begin
q <= tq;
q_bar <= ~tq;
end
end
endmodule
```

Test Bench:

```
module tb_ms_jkff;
reg clk,j,k;
wire q,q_bar;
ms_jkff inst(q,q_bar,clk,j,k);
initial
clk = 1'b0;
always #10
clk = ~clk;
initial
begin
j = 1'b0; k = 1'b0;
#60 j = 1'b0; k = 1'b1;
#40 j = 1'b1; k = 1'b0;
#20 j = 1'b1; k = 1'b1;
#40 j = 1'b1; k = 1'b0;
#5 j = 1'b0; #20 j = 1'b1;
#10;
end
initial
#250 $finish;
endmodule
```

- 6. Write Verilog Code for serial adder and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation**

SERIAL ADDER

```

module serial_adder ( A,B, reset, clock, sum);
  input [7:0] A,B;
  input reset,clock;
  output [7:0] sum;
  reg [3:0] count;
  reg s,y,Y;
  wire [7:0] qa,qb,sum;
  wire run;
  parameter G=0,H=1;

  shiftrne shift_A (A,reset,1,0,clock,qa);
  shiftrne shift_B (B,reset,1,0,clock,qb);
  shiftrne shift_sum (0,reset,run,s,clock,sum);

  //adder fsm, output and next state combinational circuitalways @(qa or qb or y)
  case (y)
    G: begin
      s = qa[0]^qb[0];
      if (qa[0] & qb[0])Y = H;
      else
        Y = G;
      end
    H: begin
      s = qa[0] ~^qb[0];
      if (~qa[0] & ~qb[0])Y =G;
      else
        Y = H;
      end
    default : Y = G;
  endcase

  //sequential block
  always @(posedge clock)if (reset)
    y <= G;
  else
    y <= Y;

  //control the shifting processalways @(posedge clock)
  if (reset) count = 8;
  else if (run) count = count - 1;
  assign run=|count;
endmodule

```

SERIAL ADDER TEST

```
module serial_adder_t ;reg [7:0] A,B;
reg reset,clock;wire [7:0] sum ;

initial
clock = 1'b0;

always
#5 clock =~clock;
serial_adder s1 (A,B,reset,clock,sum);initial
begin
reset = 1'b0;A = 8'b10101010; B = 8'b11111111;
#20 reset = 1'b1;#20 reset = 1'b0;
#150 reset = 1'b1; A = 11110000; B = 8'b11110011;
#20 reset = 1'b0;#200 $finish;
end
endmodule
```

- 7. Write Verilog Code for 4-bit counter (synchronous and asynchronous counter) and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation**

Asynchronous Counter:

```
Module rip_ctr (clock, toggle, reset, count);input clock, toggle, reset;
output [3:0] count;
reg [3:0] count;wire c0, c1, c2;
assign c0 = count[0], c1 = count[1], c2 = count[2];

always @ (posedge reset or posedge clock)
if (reset == 1'b1) count[0] <= 1'b0;
else if (toggle == 1'b1) count[0] <= ~count[0];

always @ (posedge reset or negedge c0)
if (reset == 1'b1) count[1] <= 1'b0;
else if (toggle == 1'b1) count[1] <= ~count[1];

always @ (posedge reset or negedge c1)
if (reset == 1'b1) count[2] <= 1'b0;
else if (toggle == 1'b1) count[2] <= ~count[2];

always @ (posedge reset or negedge c2)
if (reset == 1'b1) count[3] <= 1'b0;
else if (toggle == 1'b1) count[3] <= ~count[3];
endmodule
```

TEST BENCH:

```
module rip_ctr_t ;
reg clock,toggle,reset;
wire [3:0] count ;
rip_ctr a1 (clock,toggle,reset,count);
initial
clock = 1'b0;
always
#5 clock = ~clock;

Initial
begin
reset = 1'b0;
toggle = 1'b0;
#10 reset = 1'b1;
toggle = 1'b1;
#10 reset = 1'b0;
#190 reset = 1'b1;
#20 reset = 1'b0;
#100 reset = 1'b1;
#40 reset = 1'b0;
#450 $finish;
end
endmodule
```

Synchronous Counter:

```
module counter_behav ( count,reset,clk);
input wire reset, clk;
output reg [3:0] count;

always @(posedge clk)if (reset)
count = 4'b0000;else
count = count + 4'b0001;
endmodule
```

TEST BENCH:

```
module mycounter_t ;
wire [3:0] count;
reg reset,clk;

initial
clk = 1'b0;

always
#5 clk = ~clk;
counter_behav m1 ( count,reset,clk);
initial
begin
reset = 1'b0 ;

#15 reset =1'b1;
#30 reset =1'b0;

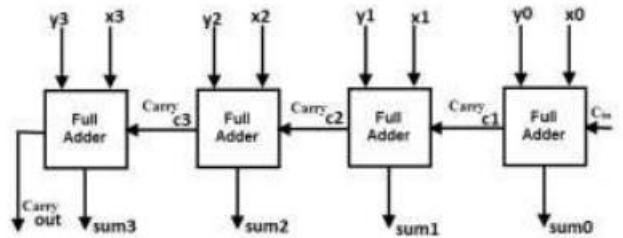
#300 $finish;

end
endmodule
```

8. Write Verilog Code for parallel adder and its Test Bench for verification, observe the waveform and synthesize the code with technological library with given constraints. Do the initial timing verification with gate level simulation

Block diagram of a 4 bit Ripple Carry Adder:

```
// FULL ADDER
module fulladd (cin, x, y, s, cout);
input cin, x, y;
output s, cout;
assign s = x ^ y ^ cin;
assign cout = (x & y) | (x & cin) | (y & cin);
endmodule
```

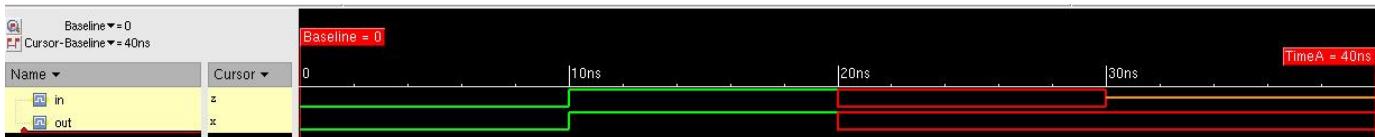
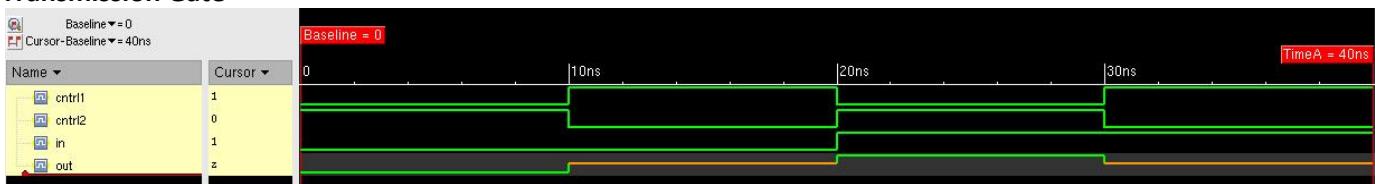
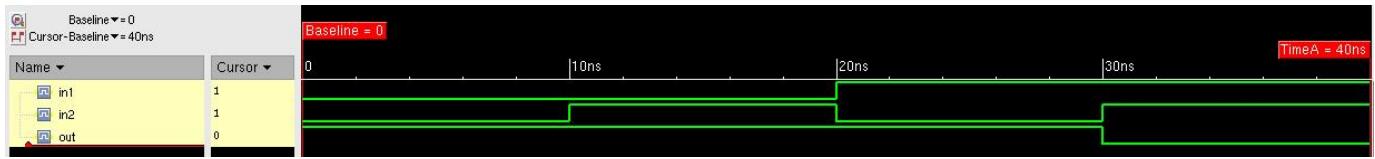
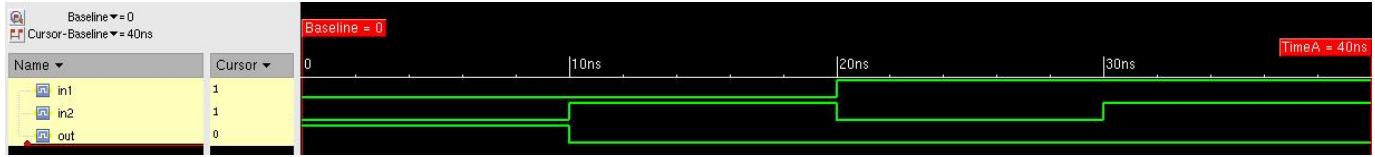
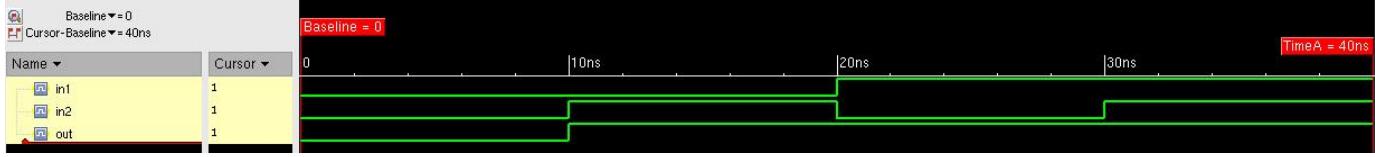
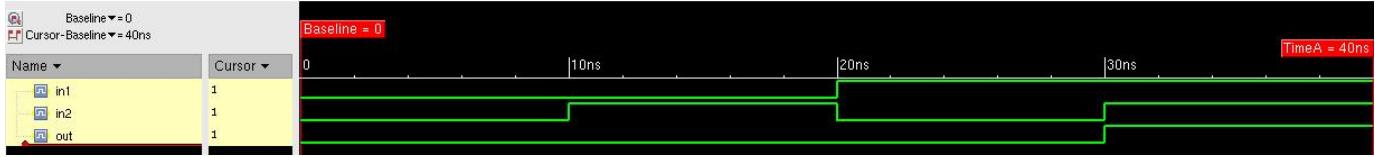
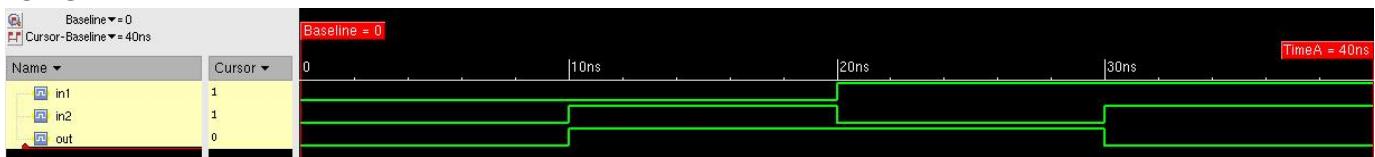


```
// PARALLEL ADDER
module adder4 (carryin, x, y, sum, carryout);
input carryin;
input [3:0] x,y;
output [3:0] sum;
output carryout;
fulladd stage0 (carryin, x[0],y[0],sum[0],c1);
fulladd stage1 (c1,x[1],y[1],sum[1],c2);
fulladd stage2 (c2,x[2],y[2],sum[2],c3);
fulladd stage3 (c3,x[3],y[3],sum[3],carryout);
endmodule
```

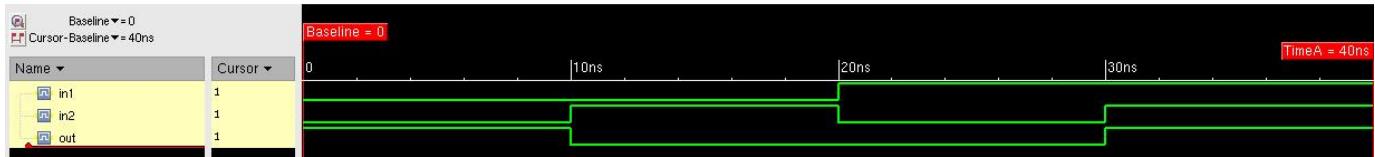
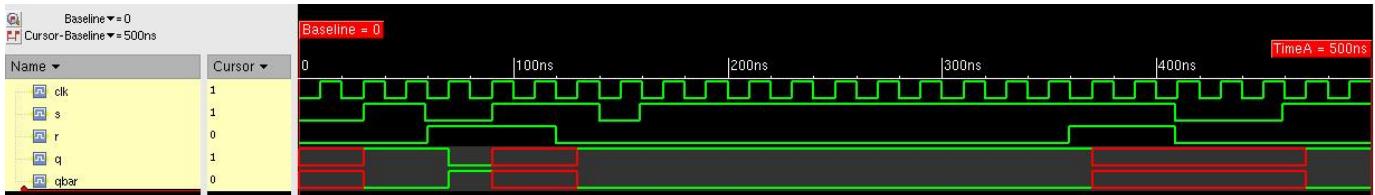
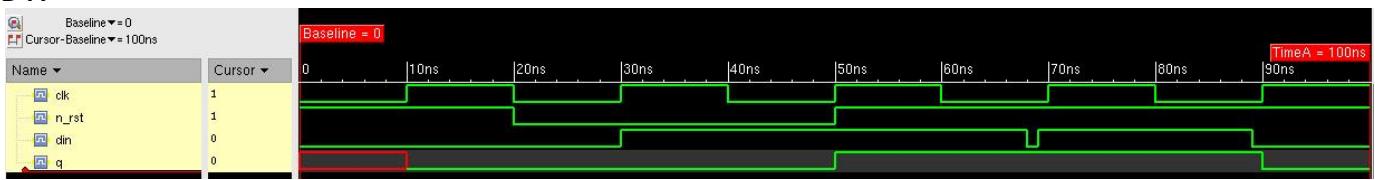
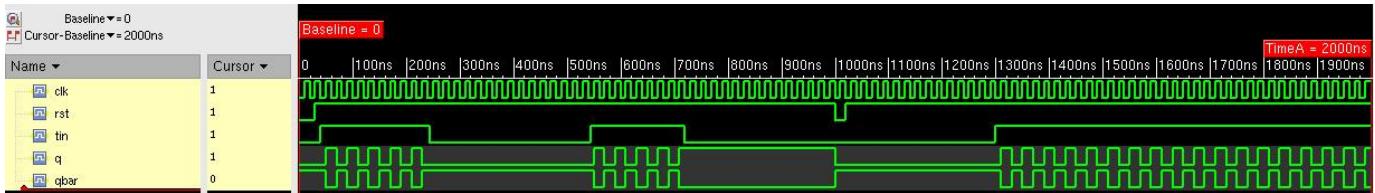
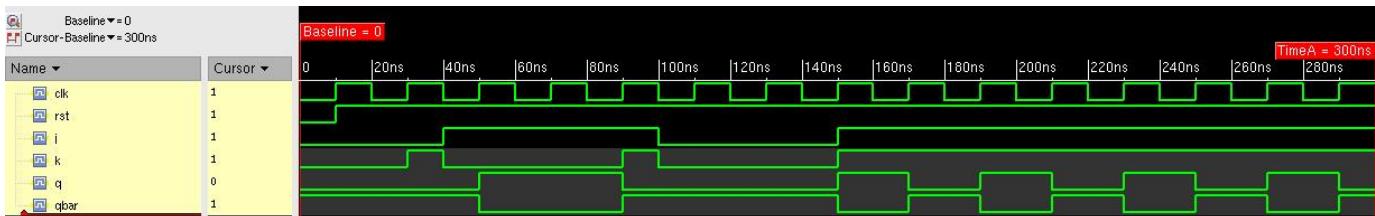
TEST BENCH :

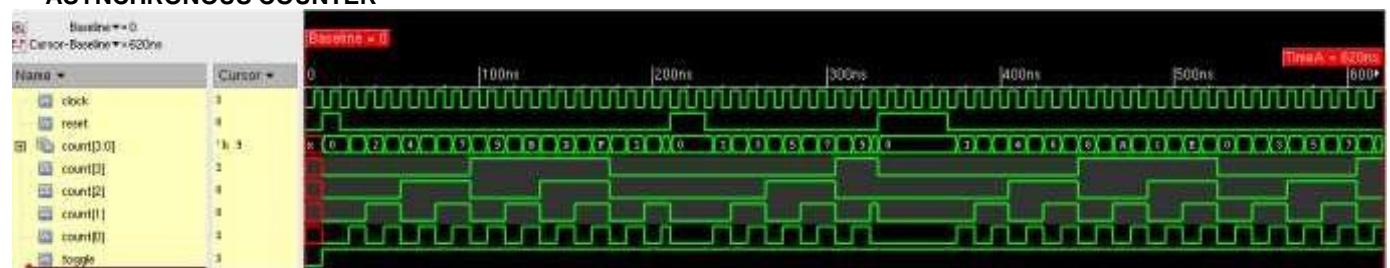
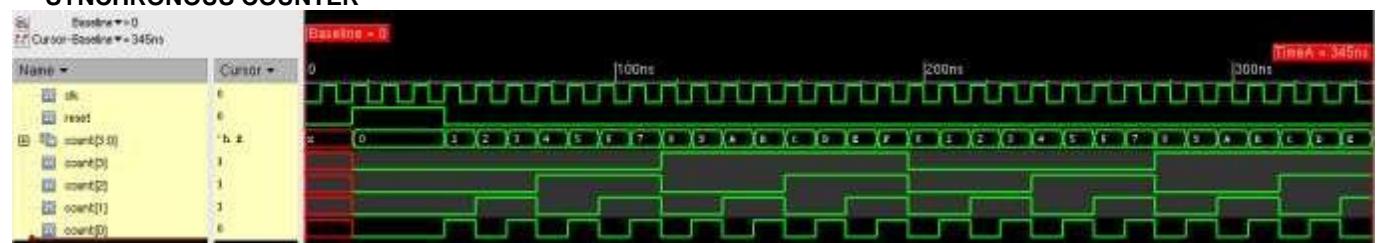
```
module adder4_t ;
reg [3:0] x, y;
reg carryin;
wire [3:0] sum;
wire carryout;
adder4 a1 (carryin, x, y, sum, carryout);
initial
begin
x = 4'b0000;
y= 4'b0000;
carryin = 1'b0;
#20 x =4'b1111; y = 4'b1010;
#40 x =4'b1011; y =4'b0110;
#40 x =4'b1111; y=4'b1111;
#150 $finish;
end
endmodule
```

This page is intentionally left blank

Inverter**Buffer****Transmission Gate****NAND Gate****NOR GATE****OR Gate****AND Gate****XOR GATE:**

This page is intentionally left blank

XNOR GATE:**SR FF****D FF****T FF****JK FF****MS JK FF**

SERIAL ADDER**PARALLEL ADDER****ASYNCHRONOUS COUNTER****SYNCHRONOUS COUNTER**

Adder circuits – full adder cascading to build 4-bit parallel adder (RCA)

Part B: Analog Virtuoso

General Notes for analog virtuoso

Lab Getting Started

1. Log in to your workstation using the username and password.
2. In a terminal window, type **csh** at the command prompt to invoke the C shell.

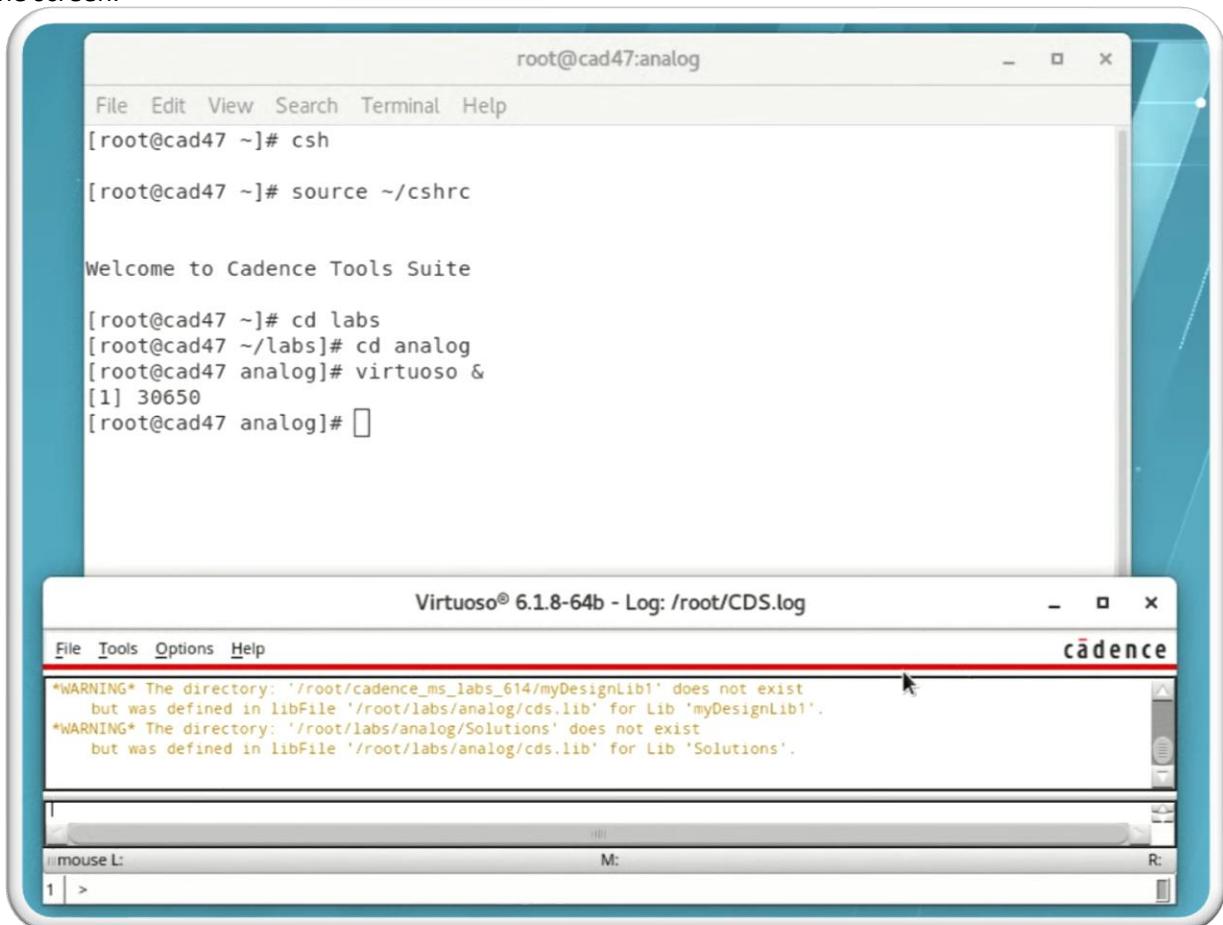
```
>csh  
>source ~/cshrc  
>cd labs  
>cd analog
```

Starting the Cadence Software

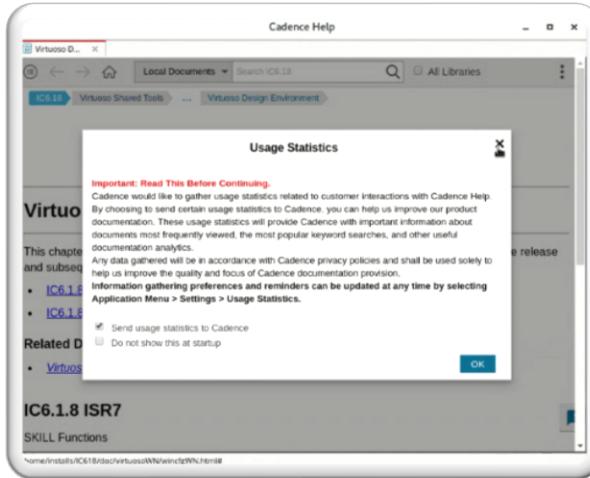
1. In the terminal window, enter:

```
>virtuoso &
```

The virtuoso or Command Interpreter Window (CIW) appears at the bottom of the screen.



2. If the “What’s New ...” if window appears, close the window.

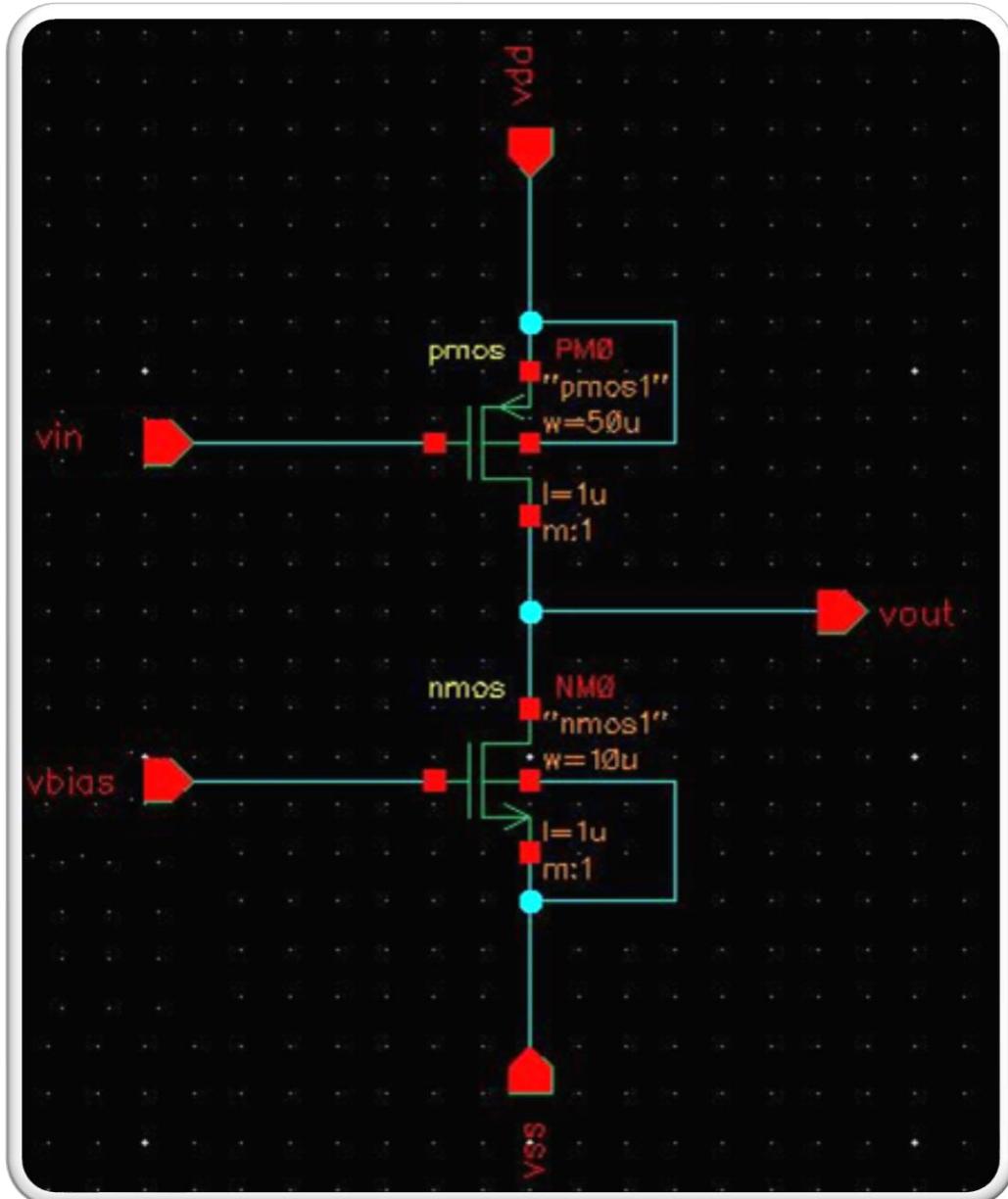


3. Keep opened CIW window for the labs.

End of General Notes

1. Design the circuit of CSA with given specifications, completing the design flow mentioned below:
 - a. Draw the schematic and verify the following.
 - i) DC Analysis ii) AC Analysis iii) Transient Analysis
 - b. Draw the Layout and verify the DRC, ERC
 - c. Check for LVS

Schematic Capture



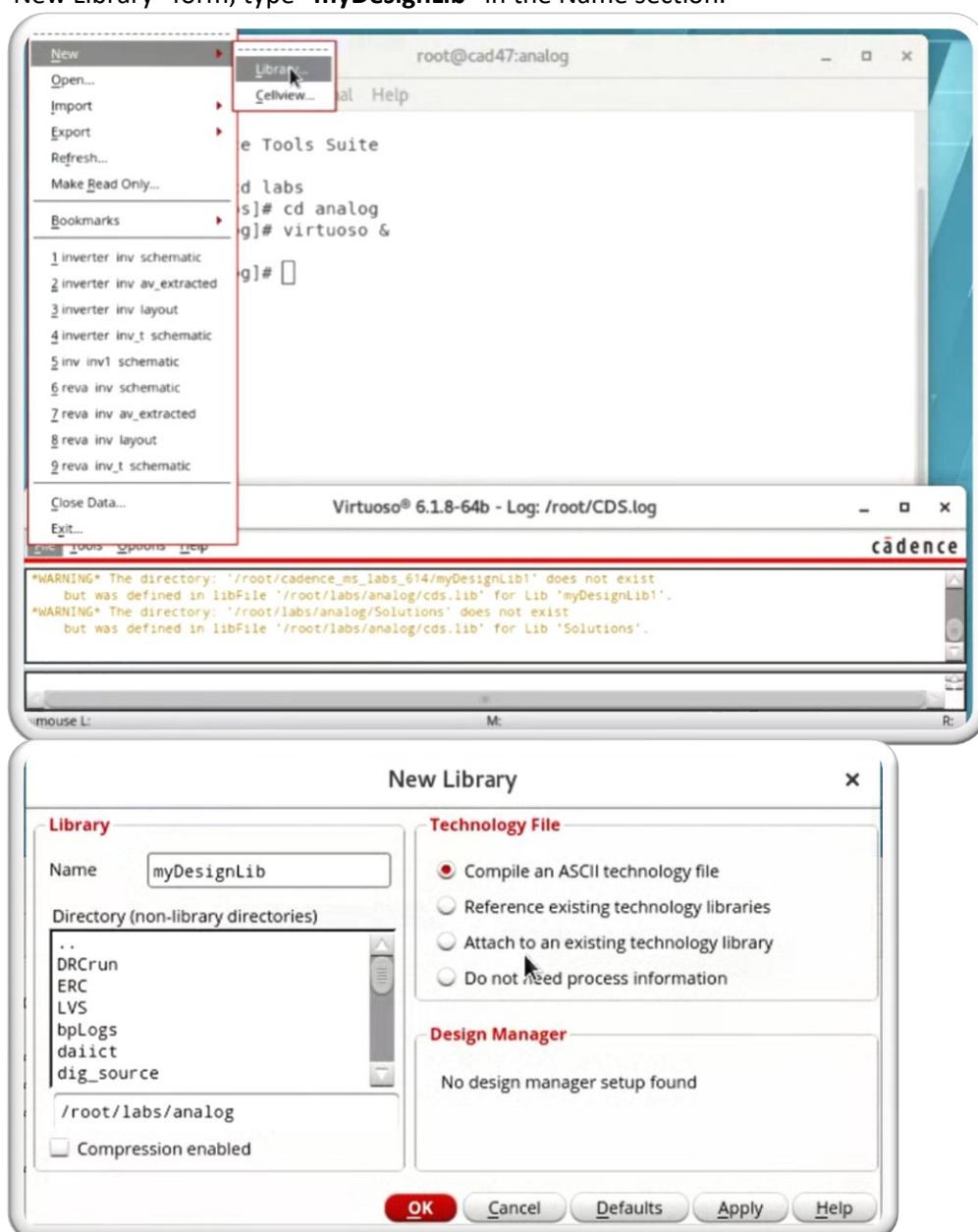
Schematic Entry

Objective: To create a library and build a schematic of an CS Amplifier

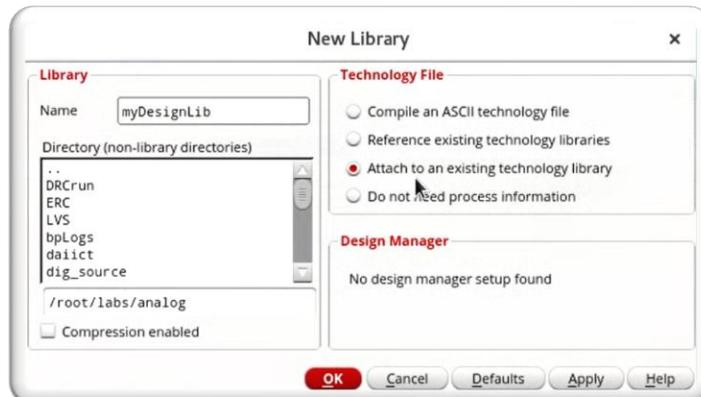
Below steps explain the creation of new library “**myDesignLib**” and we will use the same throughout this course for building various cells that we going to create in the next labs. Execute **Tools – Library Manager** in the CIW or Virtuoso window to open Library Manager.

Creating a New library

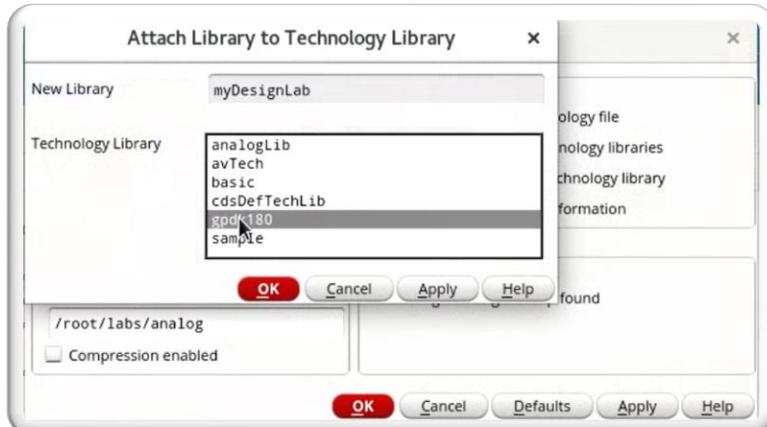
1. In the Library Manager, execute **File - New – Library**. The new library form appears.
2. In the “New Library” form, type “**myDesignLib**” in the Name section.



3. In the next “Technology File for New library” form, select option **Attach to an existing techfile** and click **OK**.

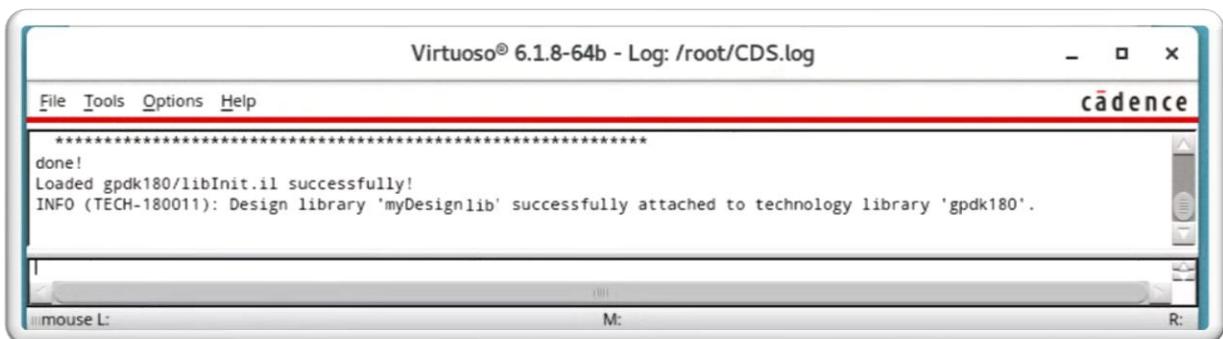


4. In the “Attach Design Library to Technology File” form, select **gpdk180** from the cyclic field and click **OK**.



5. After creating a new library you can verify it from the library manager.

6. If you right click on the “**myDesignLib**” and select properties, you will find that **gpdk180** library is attached as techlib to “**myDesignLib**”.

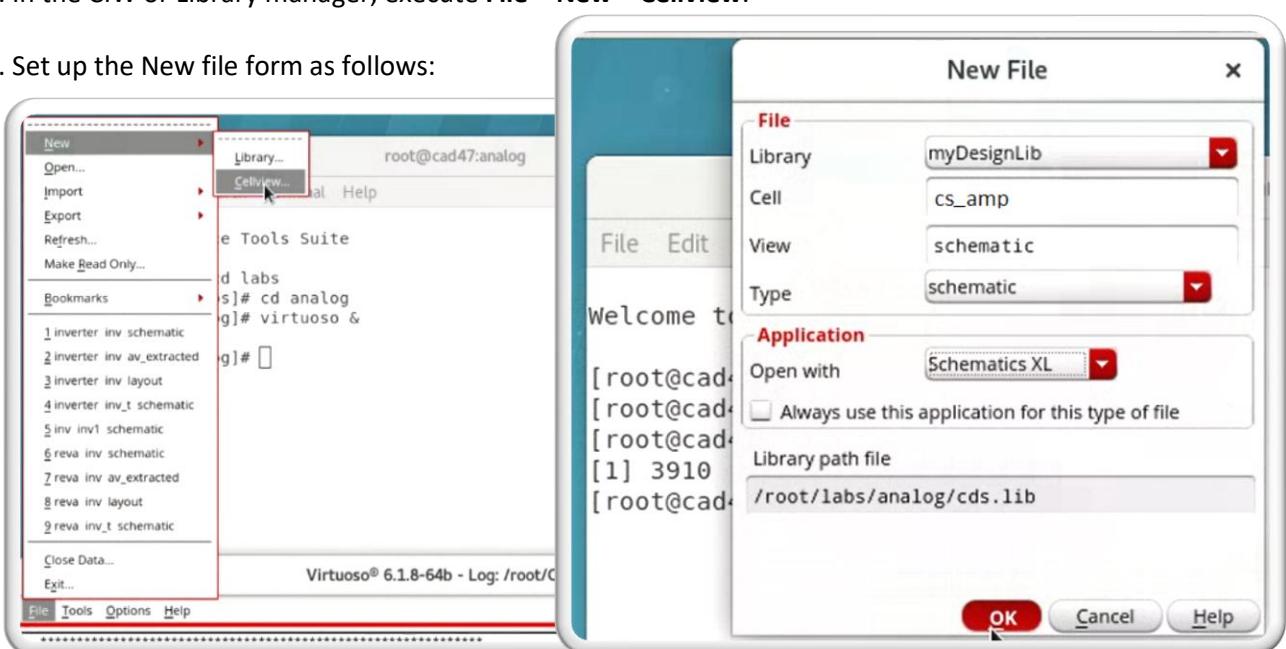


Creating a Schematic Cellview

In this section we will learn how to open new schematic window in the new “**myDesignLib**” library and build the CS Amplifier schematic as shown in the figure at the start of this lab.

1. In the CIW or Library manager, execute **File – New – Cellview**.

2. Set up the New file form as follows:



Do not edit the **Library path file** and the one above might be different from the path shown in your form.

3. Click **OK** when done the above settings. A blank schematic window for the **CS Amplifier** design appears.

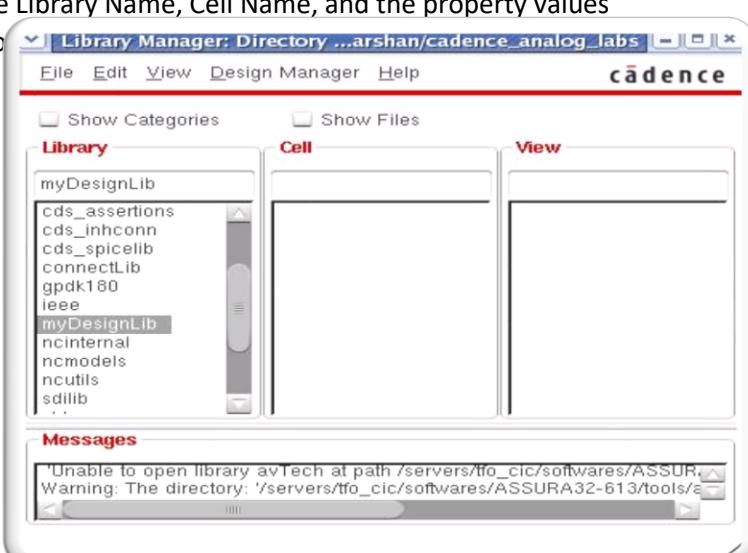
Adding Components to schematic

1. In the CS Amplifier schematic window, click the **Instance** fixed menu icon to display the Add Instance form.

Tip: You can also execute **Create — Instance** or press **i**.

2. Click on the **Browse** button. This opens up a Library browser from which you can select components and the **symbol view**.

You will update the Library Name, Cell Name, and the property values given in the table of

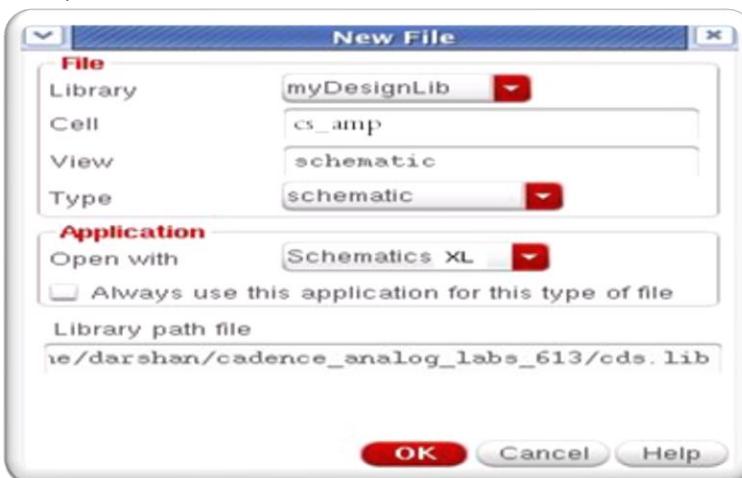


Creating a Schematic Cellview

In this section we will learn how to open new schematic window in the new “**myDesignLib**” library and build the CS Amplifier schematic as shown in the figure at the start of this lab.

1. In the CIW or Library manager, execute **File – New – Cellview**.

2. Set up the New file form as follows:



Do not edit the **Library path file** and the one above might be different from the path shown in your form.

3. Click **OK** when done the above settings. A blank schematic window for the **CS Amplifier** design appears.

Adding Components to schematic

1. In the CS Amplifier schematic window, click the **Instance** fixed menu icon to display the Add Instance form.

Tip: You can also execute **Create — Instance** or press **i**.



2. Click on the **Browse** button. This opens up a Library browser from which you can select components and the **symbol** view .

You will update the Library Name, Cell Name, and the property values given in the table on the next page as you place each component.

3. After you complete the Add Instance form, move your cursor to the schematic window and click **left** to place a component.

This is a table of components for building the CS Amplifier schematic.

Library name	Cell Name	Properties/Comments
gdk180	Pmos	Model Name = pmos1; W= 50u ; L= 1u
gdk180	Nmos	Model Name =nmos1; W= 10u ; L=1u

If you place a component with the wrong parameter values, use the **Edit— Properties— Objects** command to change the parameters. Use the **Edit— Move** command if you place components in the wrong location.



You can rotate components at the time you place them, or use the **Edit— Rotate** command after they are placed.

4. After entering components, click **Cancel** in the Add Instance form or press **Esc** with your cursor in the schematic window.

Adding pins to Schematic

1. Click the **Pin** fixed menu icon in the schematic window.

You can also execute **Create — Pin** or press **p**.

The Add pin form appears.

2. Type the following in the Add pin form in the exact order leaving space between the pin names.

Pin Names	Direction
vin vbias	Input
vout	Output
vdd vss	Input

Make sure that the direction field is set to **input/output/input Output** when placing the **input/output/inout** pins respectively and the Usage field is set to **schematic**.

3. Select **Cancel** from the Add – pin form after placing the pins.

In the schematic window, execute **Window— Fit** or press the **f** bind key.



Adding Wires to a Schematic

Add wires to connect components and pins in the design.

1. Click the **Wire (narrow)** icon in the schematic window.



You can also press the **w** key, or execute **Create — Wire (narrow)**.

2. In the schematic window, click on a pin of one of your components as the first point for your wiring. A diamond shape appears over the starting point of this wire.
3. Follow the prompts at the bottom of the design window and click **left** on the destination point for your wire. A wire is routed between the source and destination points.
4. Complete the wiring as shown in figure and when done wiring press **ESC** key in the schematic window to cancel wiring.

Saving the Design

1. Click the **Check and Save** icon in the schematic editor window.



2. Observe the CIW output area for any errors.

Symbol Creation

Objective: To create a symbol for the CS Amplifier

In this section, you will create a symbol for your CS Amplifier design so you can place it in a test circuit for simulation. A symbol view is extremely important step in the design process. The symbol view must exist for the schematic to be used in a hierarchy. In addition, the symbol has attached properties (cdsParam) that facilitate the simulation and the design of the circuit.

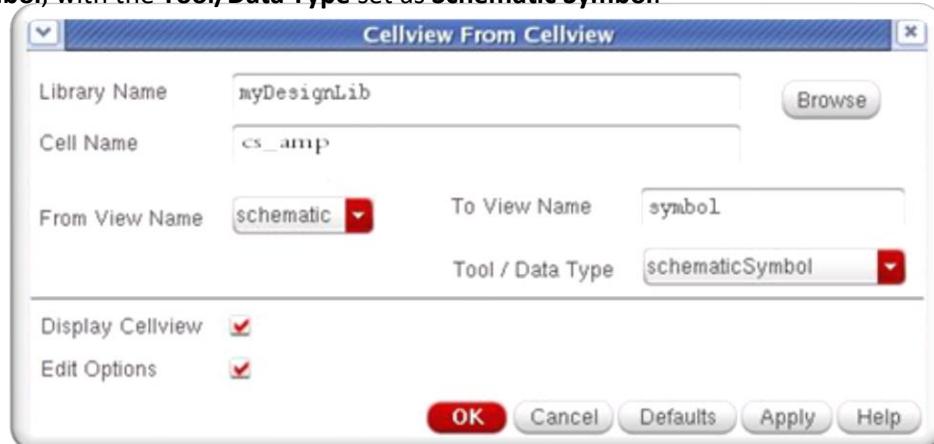
1. In the CS Amplifier schematic window, execute

Create — Cellview— From Cellview.



The **Cellview From Cellview** form appears. With the Edit Options function active, you can control the appearance of the symbol to generate.

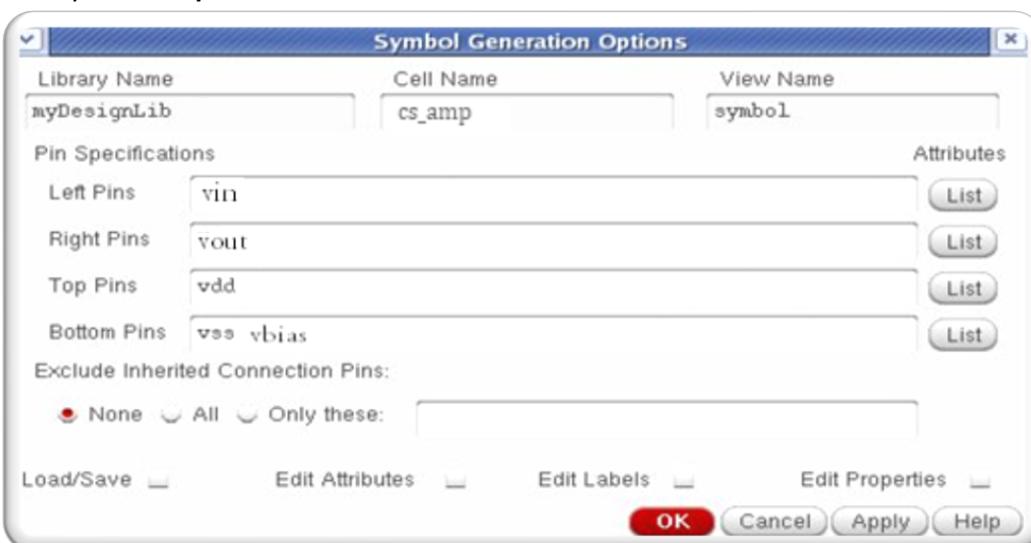
2. Verify that the **From View Name** field is set to **schematic**, and the **To View Name** field is set to **symbol**, with the **Tool/Data Type** set as **Schematic Symbol**.



3. Click **OK** in the **Cellview From Cellview** form.

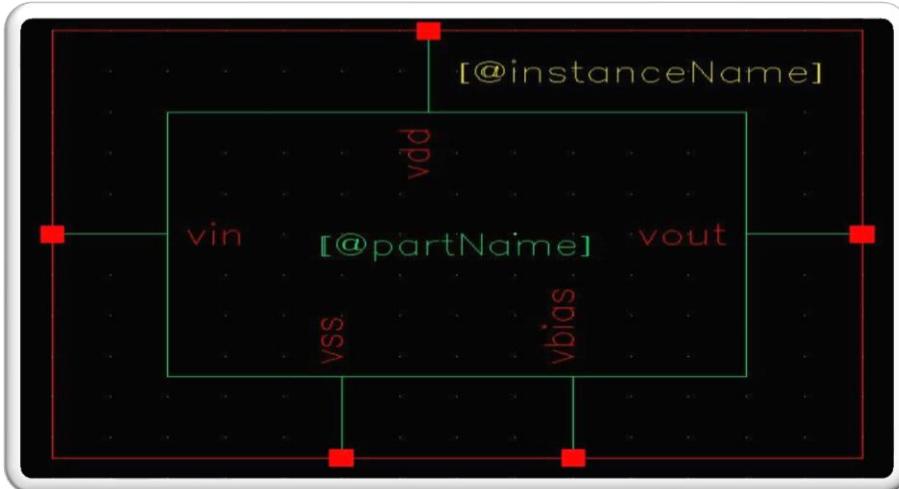
The Symbol Generation Form appears

4. Modify the **Pin Specifications** as follows:



5. Click **OK** in the **Symbol Generation Options** form.

6. A new window displays an automatically created CS Amplifier symbol as shown here.



Editing a Symbol

In this section we will modify the CS Amplifier symbol to look like a CS Amplifier gate symbol.

1. Move the cursor over the automatically generated symbol, until the green rectangle is highlighted, click **left** to select it.
2. Click **Delete** icon in the symbol window, similarly select the red rectangle and delete that.
3. Execute **Create – Shape – polygon**, and draw a shape similar to triangle.
4. After creating the triangle press **ESC** key.
5. Execute **Create – Shape – Circle** to make a circle at the end of triangle.
6. You can move the pin names according to the location.
7. Execute **Create — Selection Box**. In the Add Selection Box form, click **Automatic**. A new red selection box is automatically added.
8. After creating symbol, click on the **save** icon in the symbol editor window to save the symbol. In the symbol editor, execute **File — Close** to close the symbol view window.

Building the CS Amplifier Test Design

Objective: To build an CS Amplifier Test circuit using your CS Amplifier

Creating the CS Amplifier Test Cellview

You will create the CS Amplifier_Test cellview that will contain an instance of the CS Amplifier cellview. In the next section, you will run simulation on this design

1. In the CIW or Library Manager, execute **File— New— Cellview**.

2. Set up the New File form as follows:



3. Click **OK** when done. A blank schematic window for the **CS Amplifier_Test** design appears.

Building the CS Amplifier_Test Circuit

1. Using the component list and Properties/Comments in this table, build the **CS Amplifier_Test** schematic.

Library name	Cellview name	Properties/Comments
myDesignLib	cs_amplifier	Symbol
analogLib	vsin	Define pulse specification as AC Magnitude= 1; DC Voltage= 0; Offset Voltage= 0; Amplitude= 5m; Frequency= 1K
analogLib	vdd,vss,gnd	vdd=2.5 ; vss= -2.5 vbias=2.5

Note: Remember to set the values for **VDD** and **VSS**. Otherwise, your circuit will have no power.

2. Add the above components using **Create — Instance** or by pressing **I**.

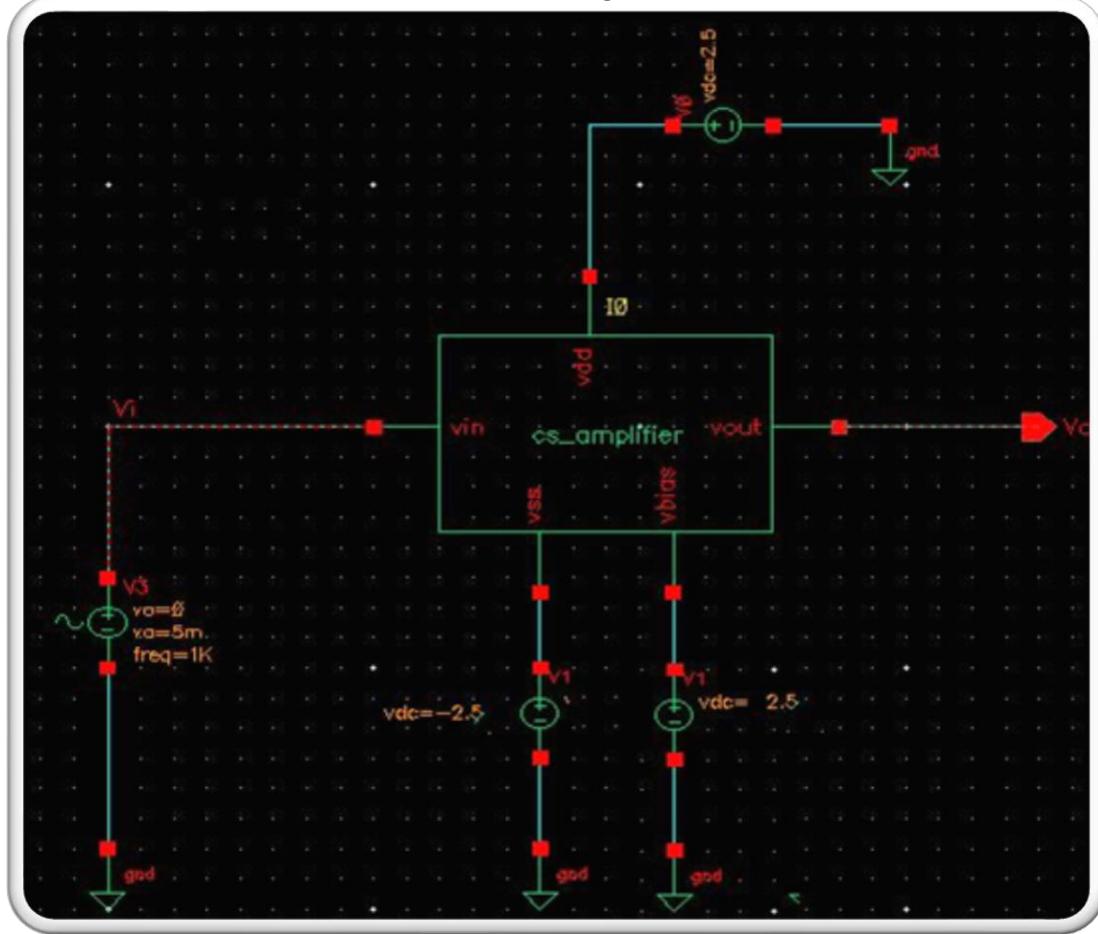
3. Click the **Wire (narrow)** icon and wire your schematic.

Tip: You can also press the **w** key, or execute **Create— Wire (narrow)**.

4. Click **Create — Wire Name** or press **L** to name the input (**Vin**) and output (**Vout**)

wires as in the below schematic.

5. Click on the ***Check and Save*** icon to save the design.



6. Leave your **CS Amplifier_Test** schematic window open for the next section.

Analog Simulation with Spectre

Objective: To set up and run simulations on the **CS Amplifier_Test** design

In this section, we will run the simulation for CS Amplifier and plot the transient, DC characteristics and we will do Parametric Analysis after the initial simulation.

Starting the Simulation Environment

Start the Simulation Environment to run a simulation.

1. In the **CS Amplifier_Test** schematic window, execute
Launch – ADE L

The **Virtuoso Analog Design Environment (ADE)** simulation window appears.

Choosing Analyses

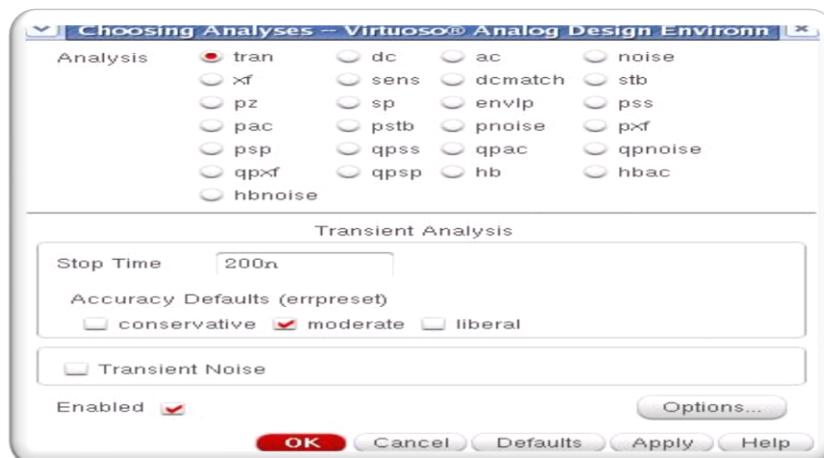
This section demonstrates how to view and select the different types of analyses to complete the circuit when running the simulation.

1. In the Simulation window (ADE), click the **Choose - Analyses** icon.
You can also execute **Analyses - Choose**.

The Choosing Analysis form appears. This is a dynamic form, the bottom of the form changes based on the selection above.

2. To setup for transient analysis

- In the Analysis section select **tran**
- Set the stop time as **5m**
- Click at the **moderate** or **Enabled** button at the bottom, and then click **Apply**.



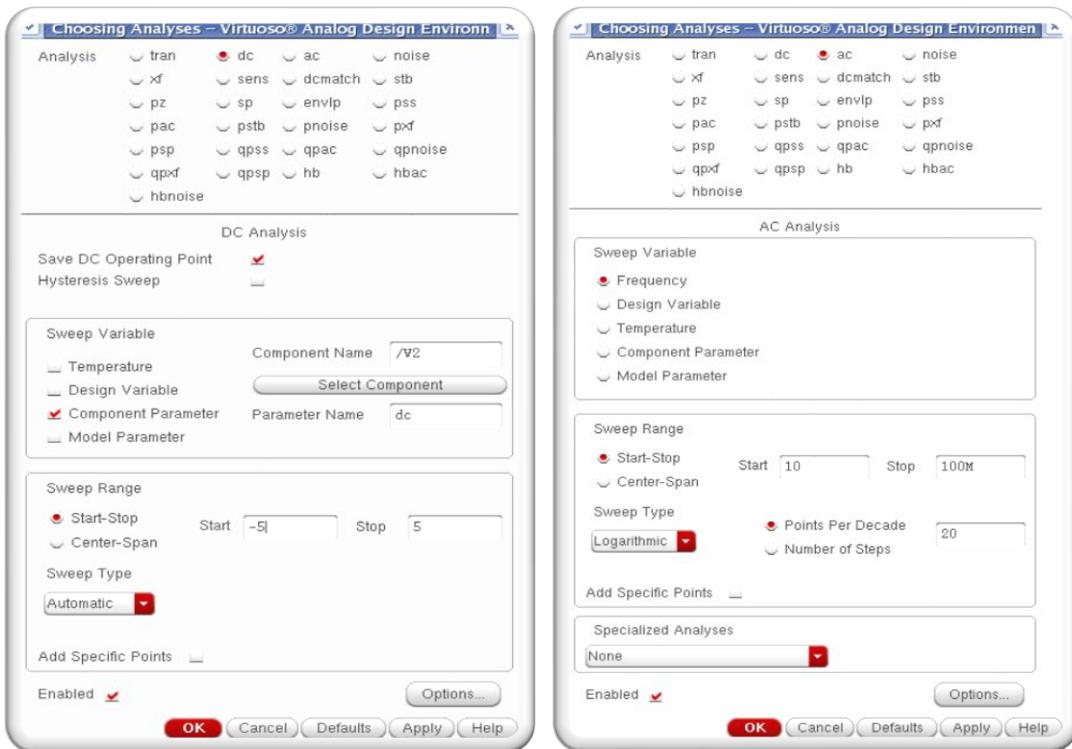
3. To set up for DC Analyses:

- In the Analyses section, select **dc**.
- In the DC Analyses section, turn **on** Save DC Operating Point.
- Turn on the **Component Parameter**
- Double click the **Select Component**, Which takes you to the schematic window.
- Select input signal **Vsin** for dc analysis.
- In the analysis form, select **start** and **stop** voltages as 0 to 2.5 or **-2.5 to 2.5** respectively.
- Check the enable button and then click **Apply**.

4. To set up for AC Analyses form is shown in the previous page.

- In the Analyses section, select **ac**.
- In the AC Analyses section, turn on **Frequency**.
- In the Sweep Range section select **start** and **stop** frequencies as **150 to 100M**
- Select Points per Decade as **20**.
- Check the enable button and then click **Apply**.

5. Click **OK** in the Choosing Analyses Form.



4. Click **OK** in the Choosing Analyses Form.

Setting Design Variables

Set the values of any design variables in the circuit before simulating. Otherwise, the simulation will not run.

1. In the Simulation window, click the **Edit Variables** icon.

The Editing Design Variables form appears.

2. Click **Copy From** at the bottom of the form.

The design is scanned and all variables found in the design are listed.

In a few moments, the **wp** variable appears in the Table of Design variables section.

3. Set the value of the **wp** variable:

With the **wp** variable highlighted in the Table of Design Variables, click on the variable name **wp** and enter the following:

Value(Expr)	2u
-------------	----

Click **Change** and notice the update in the Table of Design Variables.

3. Click **OK** or **Cancel** in the Editing Design Variables window.

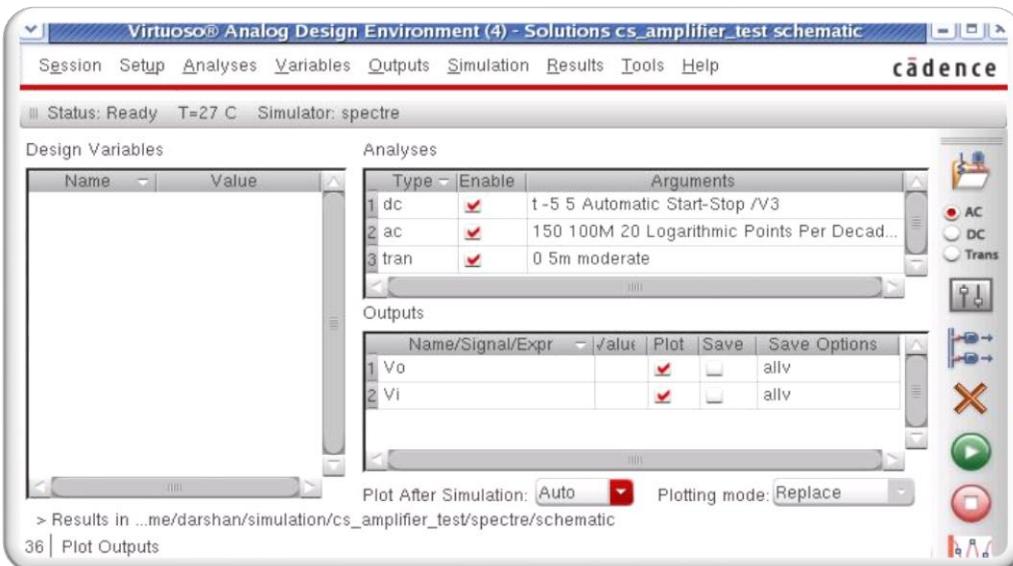
Selecting Outputs for Plotting

Select the nodes to plot when simulation is finished.

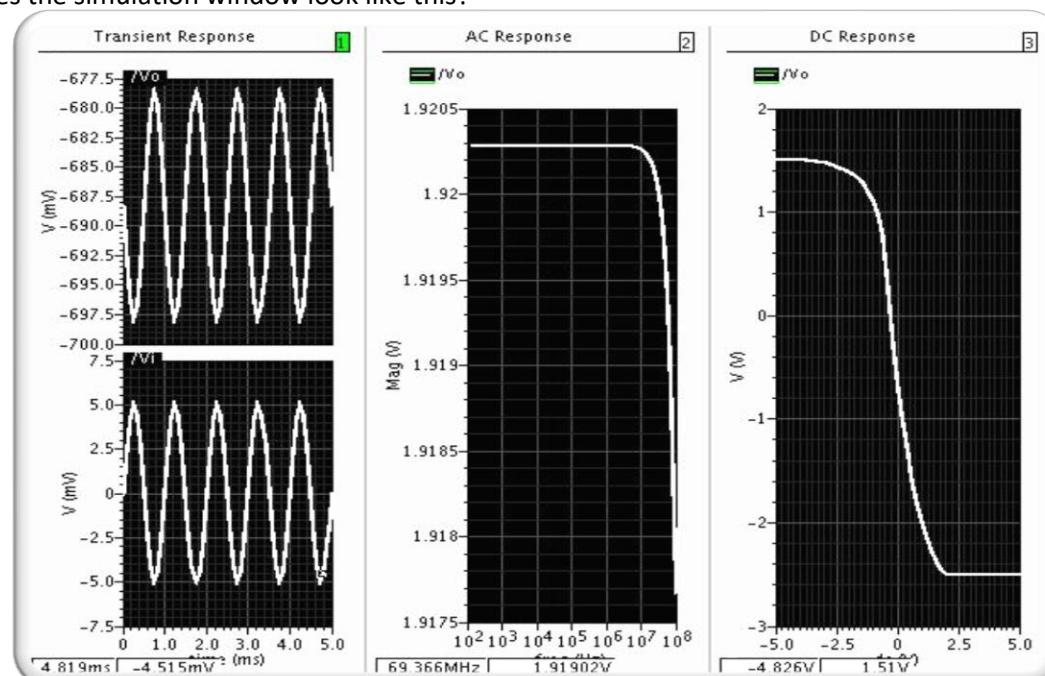
1. Execute **Outputs – To be plotted – Select on Schematic** in the simulation window.
2. Follow the prompt at the bottom of the schematic window, Click on output net **V_o**, input net **V_{in}** of the Diff_amplifier. Press **ESC** with the cursor in the schematic after selecting node.

Running the Simulation

1. Execute **Simulation – Netlist and Run** in the simulation window to start the simulation or the icon, this will create the netlist as well as run the simulation.
2. When simulation finishes, the Transient, DC plots automatically will be popped up along with log file.



Does the simulation window look like this?



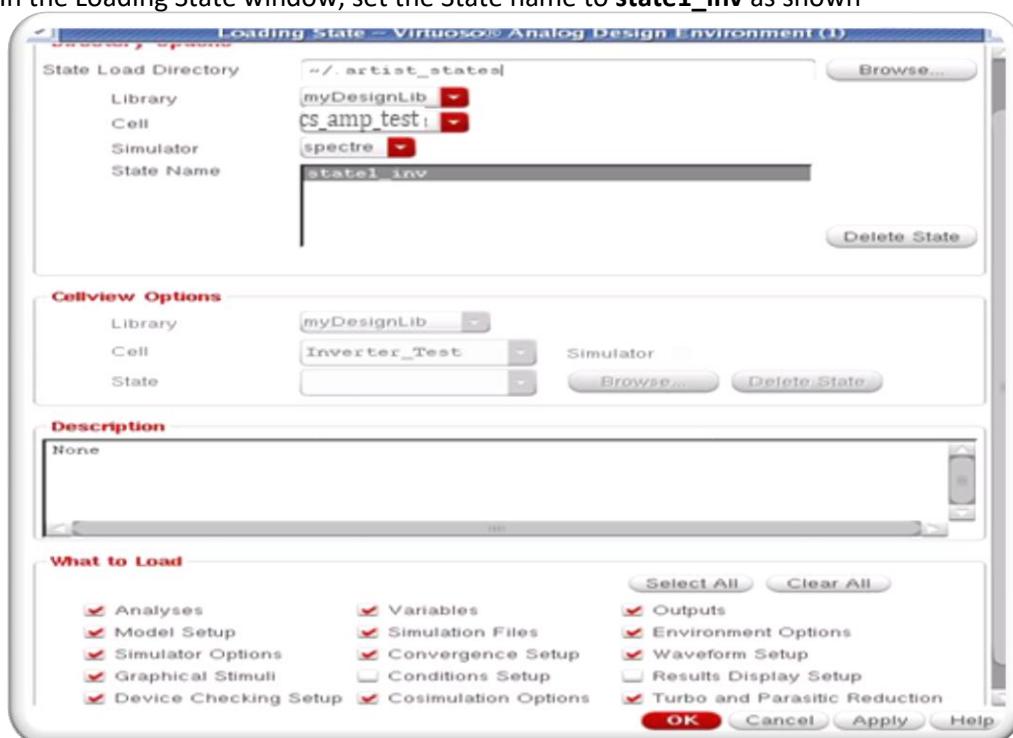
Saving the Simulator State

We can save the simulator state, which stores information such as model library file, outputs, analysis, variable etc. This information restores the simulation environment without having to type in all of setting again.

1. In the Simulation window, execute **Session – Save State**.
The Saving State form appears.
2. Set the **Save as** field to **state1_inv** and make sure all options are selected under what to save field.
3. Click **OK** in the saving state form. The Simulator state is saved.

Loading the Simulator State

1. From the ADE window execute **Session – Load State**.
2. In the Loading State window, set the State name to **state1_inv** as shown



3. Click **OK** in the Loading State window

Parametric Analysis

Parametric Analysis yields information similar to that provided by the Spectre® sweep feature, except the data is for a full range of sweeps for each parametric step. The Spectre sweep feature provides sweep data at only one specified condition.

You will run a parametric DC analysis on the **wp** variable, of the PMOS device of the CS Amplifier design by sweeping the value of **wp**.

Run a simulation before starting the parametric tool. You will start by loading the state from the previous simulation run.

Run the simulation and check for errors. When the simulation ends, a single waveform in the waveform window displays the DC Response at the **Vout** node.

Starting the Parametric Analysis Tool

1. In the Simulation window, execute **Tools—Parametric Analysis**.

The Parametric Analysis form appears.

2. In the Parametric Analysis form, execute **Setup—Pick Name For Variable—Sweep 1**.

A selection window appears with a list of all variables in the design that you can sweep. This list includes the variables that appear in the Design Variables section of the Simulation window.

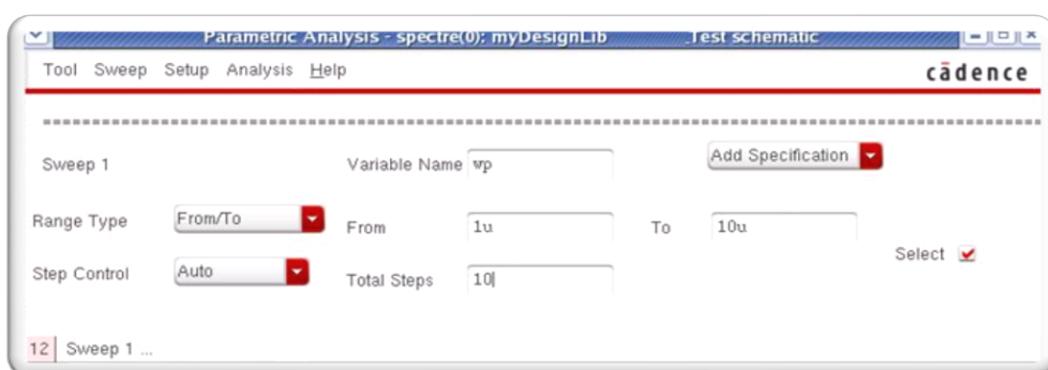
3. In the selection window, double click left on **wp**.

The Variable Name field for Sweep 1 in the Parametric Analysis form is set to **wp**.

4. Change the Range Type and Step Control fields in the Parametric Analysis form as shown below:

Range Type	From/To	From	1u	To	10u
Step Control	Auto	Total Steps	10		

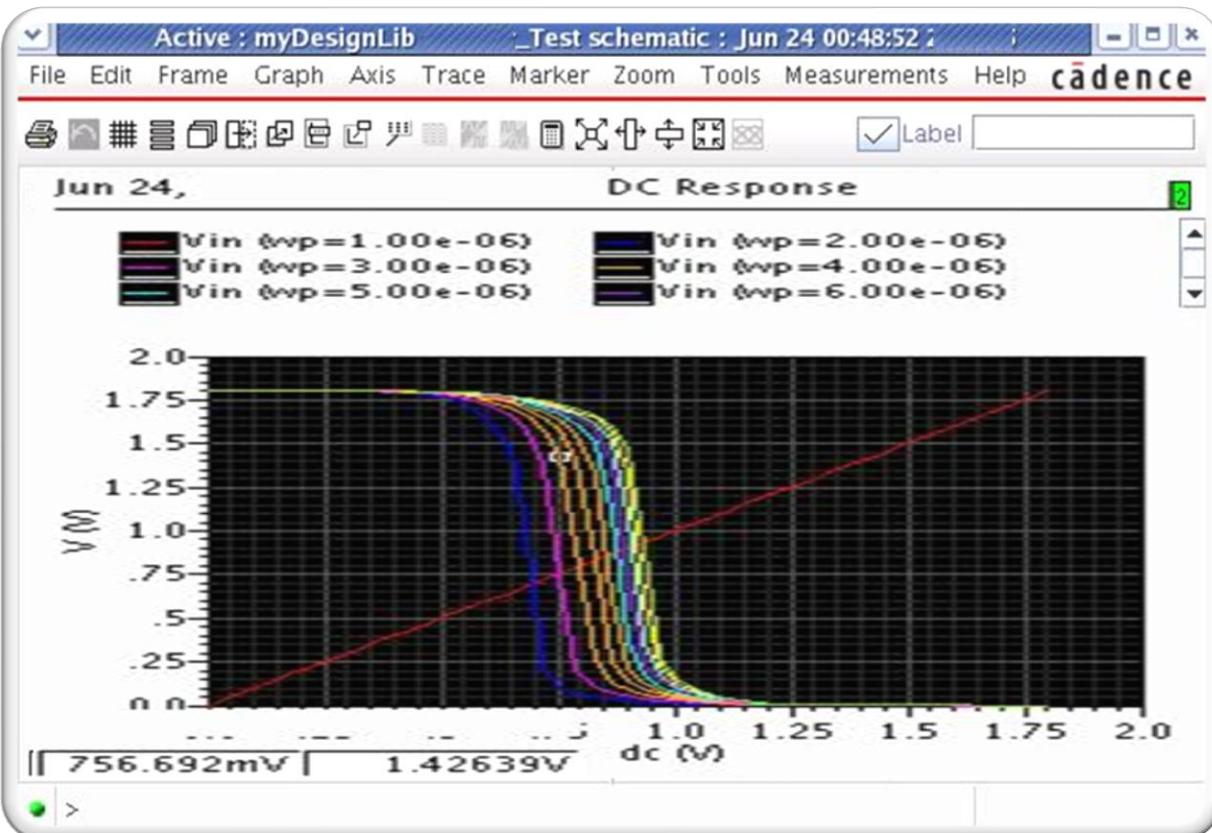
These numbers vary the value of the **wp** of the pmos between 1um and 10um at ten evenly spaced intervals.



5. Execute Analysis—Start.



The Parametric Analysis window displays the number of runs remaining in the analysis and the current value of the swept variable(s). Look in the upper right corner of the window. Once the runs are completed the wave scan window comes up with the plots for different runs.



Note: Change the wp value of pmos device back to 2u and save the schematic before proceeding to the next section of the lab. To do this use edit property option.



Creating Layout View of CS Amplifier

1. From the **CS Amplifier** schematic window menu execute **Launch – Layout XL**. A **Startup Option** form appears.
2. Select **Create New** option. This gives a **New Cell View Form**
3. Check the **Cellname (CS Amplifier)**, **Viewname (layout)**.
4. Click **OK** from the **New Cellview** form. LSW and a blank layout window appear along with schematic window.

Adding Components to Layout.

1. Execute **Connectivity – Generate – All from Source** or click the icon in the layout editor window, **Generate Layout** form appears. Click **OK** which imports the schematic components in to the Layout window automatically.



2. Re arrange the components with in PR-Boundary as shown in the next page.

Press shift+F to change the view,

Place –pin placement, click on place as in schematic,

Create H-rail-select vdd, vss click on H-rail.

Create wire **

3. To rotate a component, Select the component and execute **Edit –Properties**.

now select the degree of rotation from the property edit form.

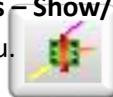


4. To Move a component, Select the component and execute **Edit -Move** command.

5. To Extend the PR boundary right click stretch & click on the line which is to be extend.

Making interconnection

1. Execute **Connectivity –Nets – Show/Hide selected Incomplete Nets** or click the icon in the Layout Menu.



2. Move the mouse pointer over the device and click **LMB** to get the connectivity information, which shows the guide lines (or flight lines) for the inter connections of the components.

3. From the layout window execute **Create – Shape – Path/ Create wire** or **Create – Shape – Rectangle** (for vdd and gnd bar) and select the appropriate Layers from the **LSW** window and Vias for making the inter connections
Shift-z, Ctrl-z –zoom.

Creating Contacts/Vias

You will use the contacts or vias to make connections between two different layers.

1. Execute **Create — Via** or select command to place different Contacts, as given in below table

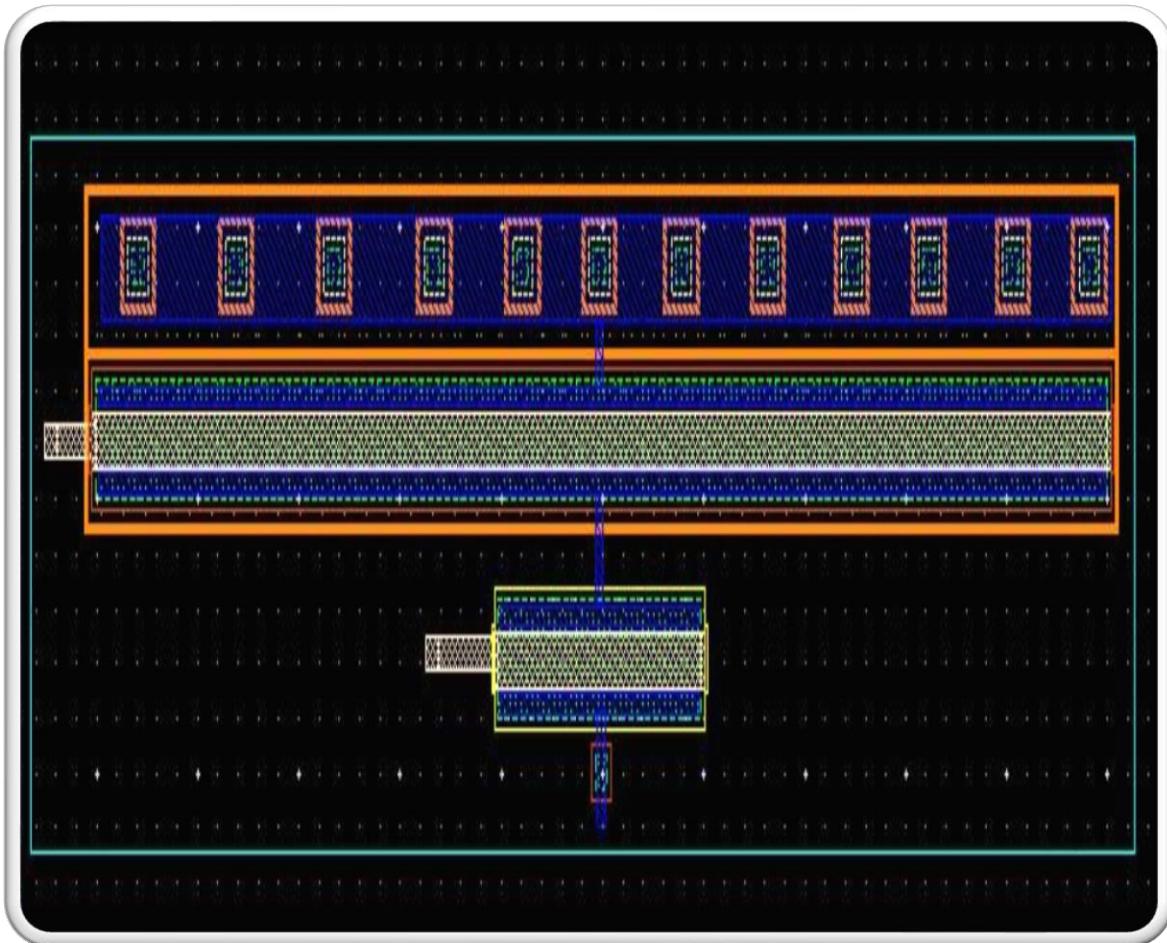
**

Connection	Contact Type
For Metal1- Poly Connection	Metal1-Poly
For Metal1- Psubstrate Connection	Metal1-Psub
For Metal1- Nwell Connection	Metal1-Nwell

(to extend the PR boundary right click stretch and click on the line which is to be Extended)

Saving the design

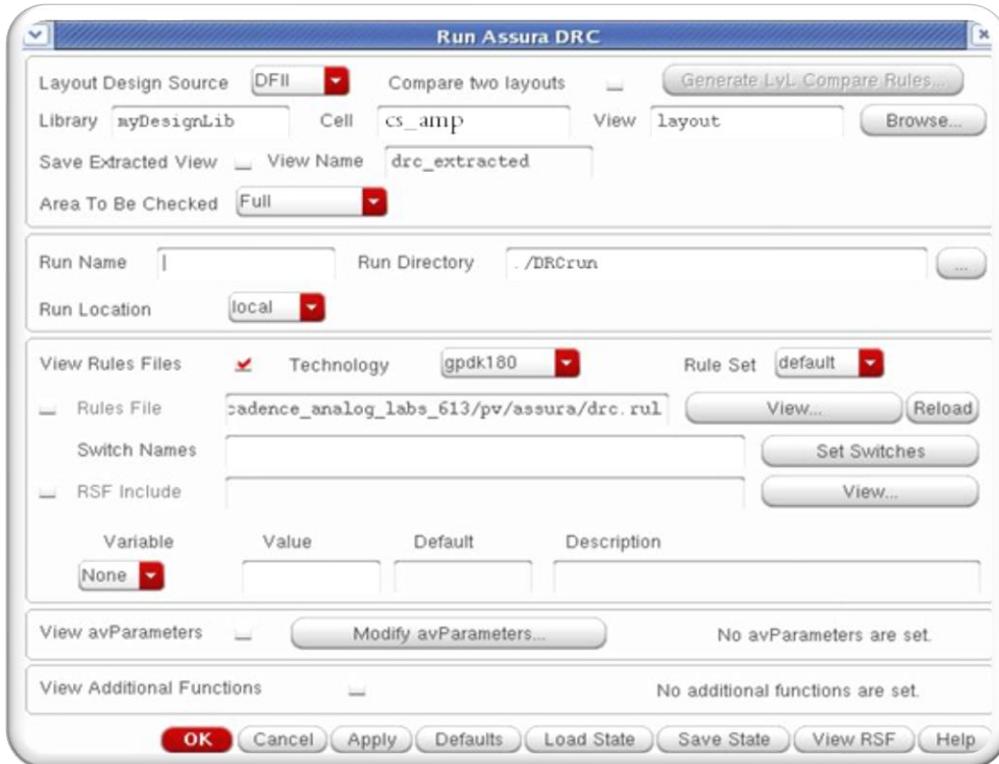
1. Save your design by selecting **File — Save** or click  to save the layout, and layout should appear as below

**Physical Verification****Assura DRC****Running a DRC**

1. Open the CS Amplifier layout form the CIW or library manger if you have closed that.
Press **shift – f** in the layout window to display all the levels.
2. Select **Assura - Run DRC** from layout window.

The DRC form appears. The Library and Cellname are taken from the current design window, but rule file may be missing. Select the Technology as **gpdk180**. This automatically loads the rule file.

Your DRC form should appear like this

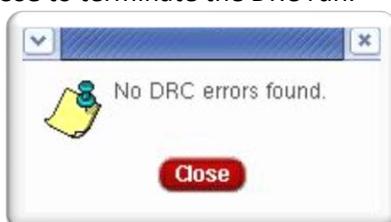


3. Click **OK** to start DRC.
 4. A Progress form will appear. You can click on the watch log file to see the log file.
 5. When DRC finishes, a dialog box appears asking you if you want to view your DRC results, and then click **Yes** to view the results of this run.
- 'cs_amplifier" has completed SUCCESSFULLY!'**

The DRC run "cs_amplifier" has completed successfully.

Do you want to view the results of this run?

Yes **No** **Help**
6. If there any DRC error exists in the design **View Layer Window** (VLW) and **Error Layer Window** (ELW) appears. Also the errors highlight in the design itself.
 7. Click **View – Summary** in the ELW to find the details of errors.
 8. You can refer to rule file also for more information, correct all the DRC errors and **Re – run** the DRC.
 9. If there are no errors in the layout then a dialog box appears with **No DRC errors found** written in it, click on **close** to terminate the DRC run.



ASSURA LVS

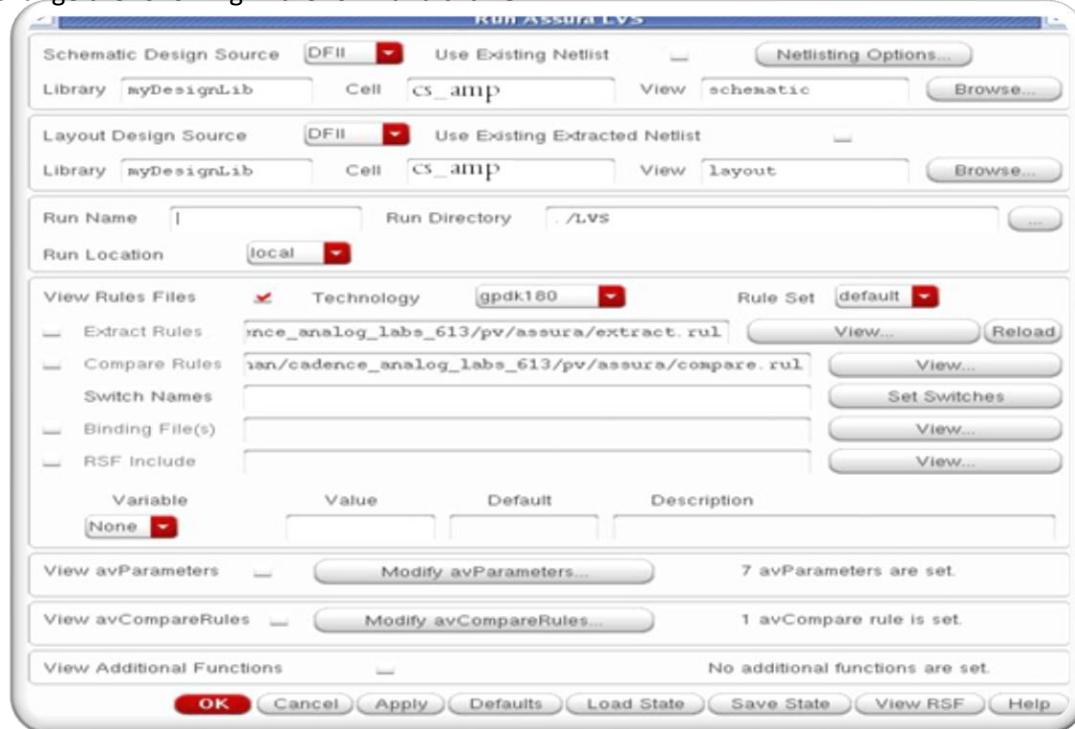
In this section we will perform the LVS check that will compare the schematic netlist and the layout netlist.

Running LVS

1. Select **Assura – Run LVS** from the layout window.

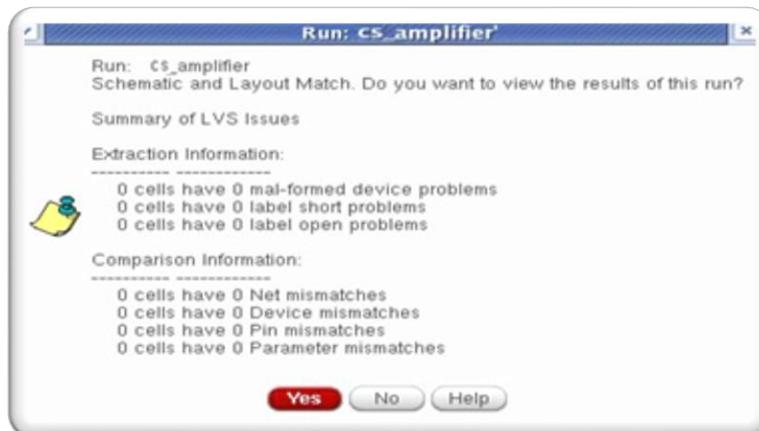
The Assura Run LVS form appears. It will automatically load both the schematic And layout view of the cell.

2. Change the following in the form and click **OK**.



3. The LVS begins and a Progress form appears.

4. If the schematic and layout matches completely, you will get the form displaying **Schematic and Layout Match**.



5. If the schematic and layout do not matches, a form informs that the LVS

Completed successfully and asks if you want to see the results of this run.

6. Click **Yes** in the form. LVS debug form appears, and you are directed into LVS debug environment.
7. In the **LVS debug form** you can find the details of mismatches and you need to correct all those mismatches and **Re – run** the LVS till you will be able to match the schematic with layout.

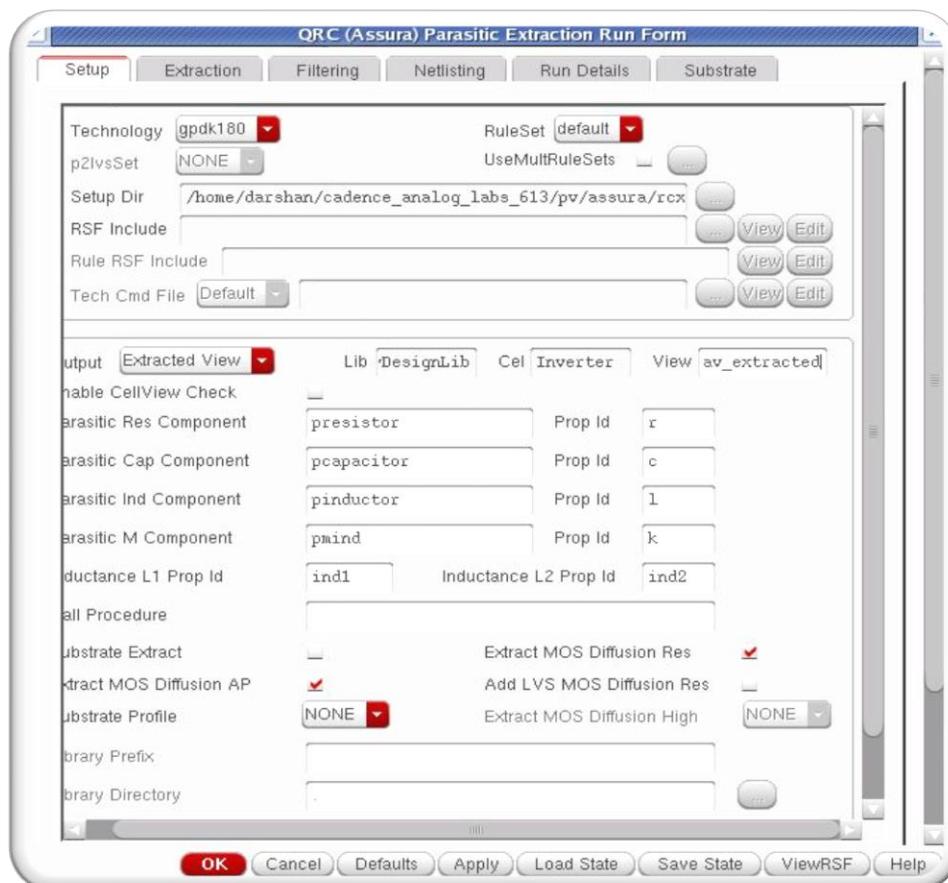
Assura RCX

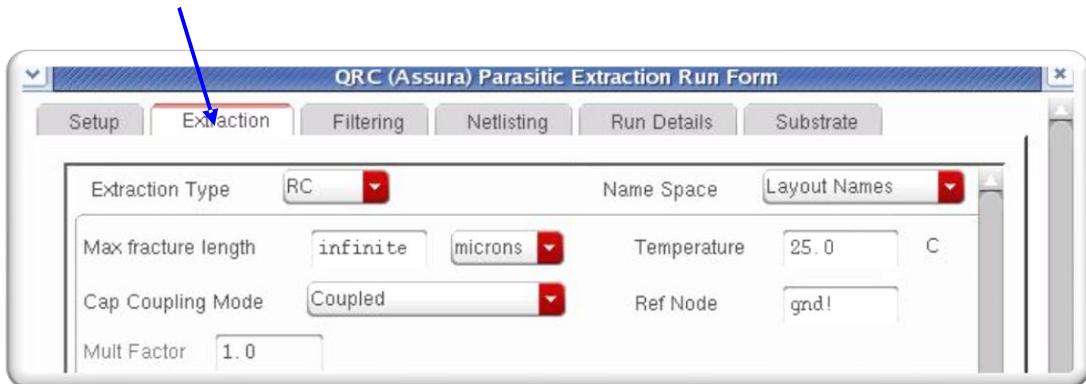
In this section we will extract the RC values from the layout and perform analog circuit simulation on the designs extracted with RCX.

Before using RCX to extract parasitic devices for simulation, the layout should match with schematic completely to ensure that all parasites will be back annotated to the correct schematic nets.

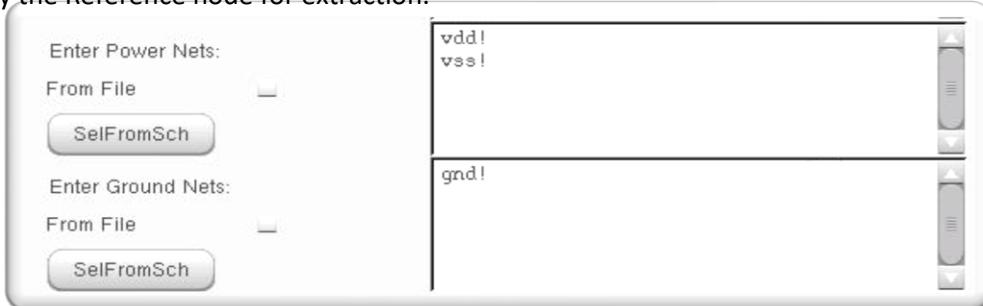
Running RCX

1. From the layout window execute **Assura – Run RCX**. or **QRC**
2. Change the following in the Assura parasitic extraction form. Select **output** type under **Setup** tab of the form.





- In the **Extraction** tab of the form, choose Extraction type, Cap Coupling Mode and specify the Reference node for extraction.



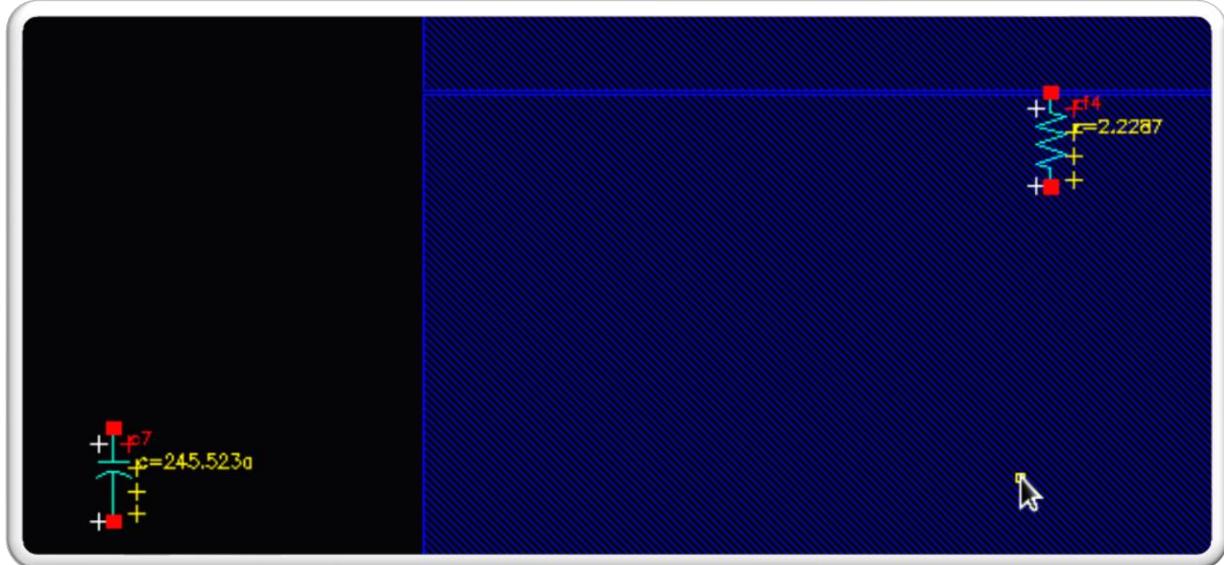
- In the **Filtering** tab of the form, **Enter Power Nets** as **vdd!, vss!** and **Enter Ground Nets** as **gnd!**

- Click **OK** in the Assura parasitic extraction form when done.

The RCX progress form appears, in the progress form click **Watch log file** to see the output log file.

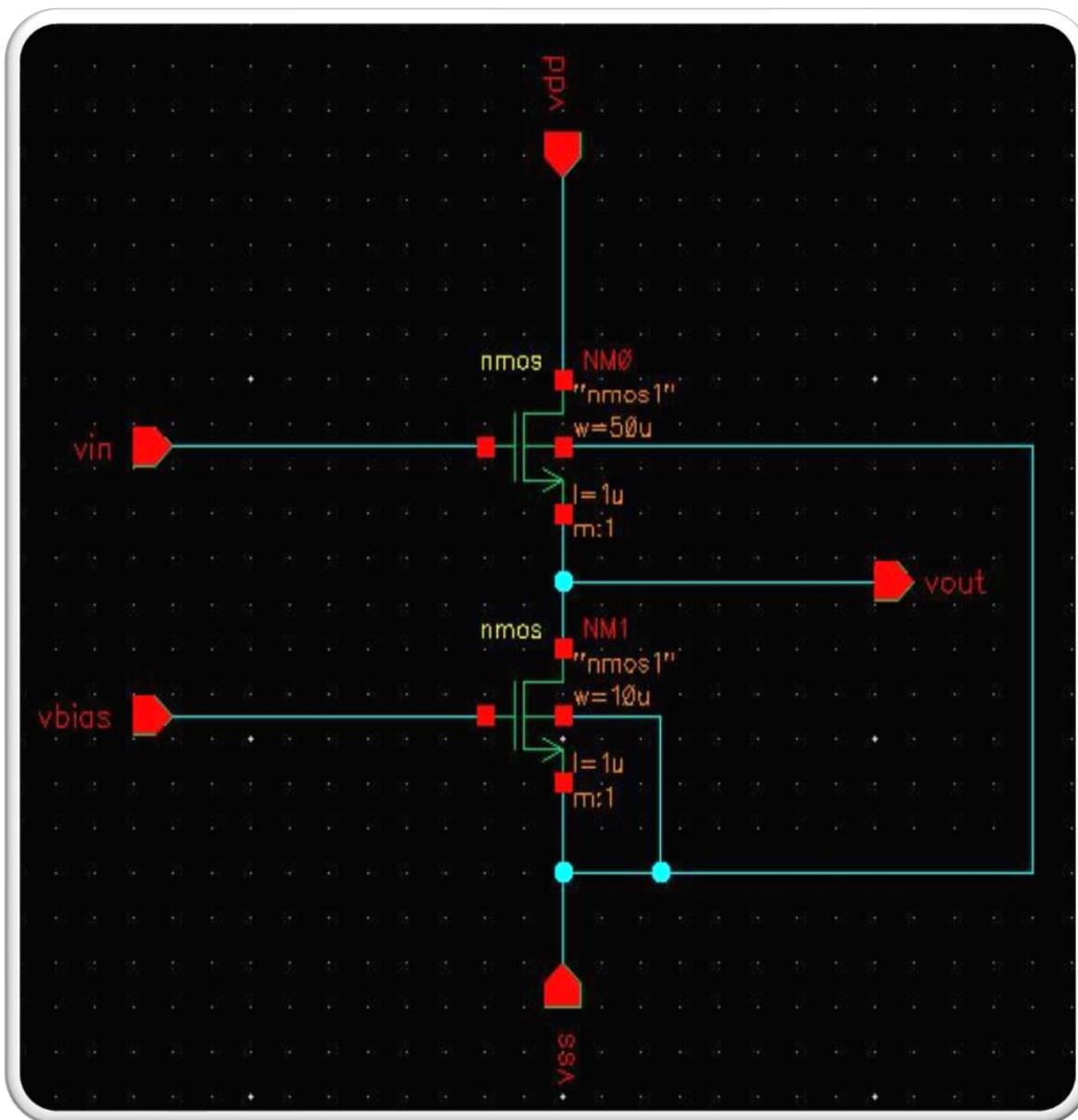
- When RCX completes, a dialog box appears, informs you that **Assura RCX run Completed successfully.**

6. You can open the **av_extracted** view from the library manager and view the parasitic. (Shift FC)



2. Design the circuit of CDA with given specifications, completing the design flow mentioned below:
- Draw the schematic and verify the following.
 - DC Analysis
 - AC Analysis
 - Transient Analysis
 - Draw the Layout and verify the DRC, ERC
 - Check for LVS

Schematic Capture



Schematic Entry

Objective: To create a new cell view and build Common Drain Amplifier

Use the techniques learned in the Lab1 and Lab2 to complete the schematic of Common Drain Amplifier.

This is a table of components for building the Common Drain Amplifier schematic.

Library name	Cell Name	Properties/Comments
gdk180	nmos	Model Name = nmos1; W= 50u ; L= 1u
gdk180	nmos	Model Name = nmos1; W= 10u ; L= 1u

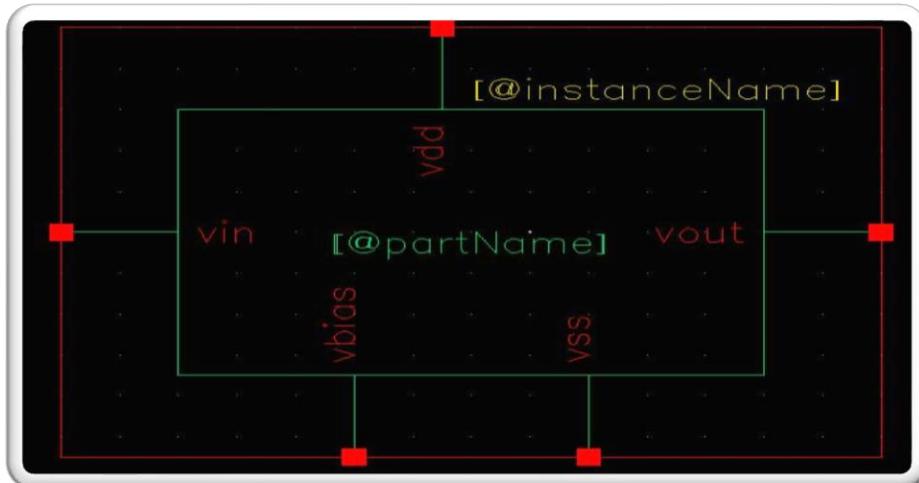
Type the following in the ADD pin form in the exact order leaving space between the pin names.

Pin Names	Direction
vin, vbias	Input
vout	Output
vdd vss	Input

Symbol Creation

Objective: To create a symbol for the Common Drain Amplifier

Use the techniques learned in the Lab1 and Lab2 to complete the symbol of cd-amplifier



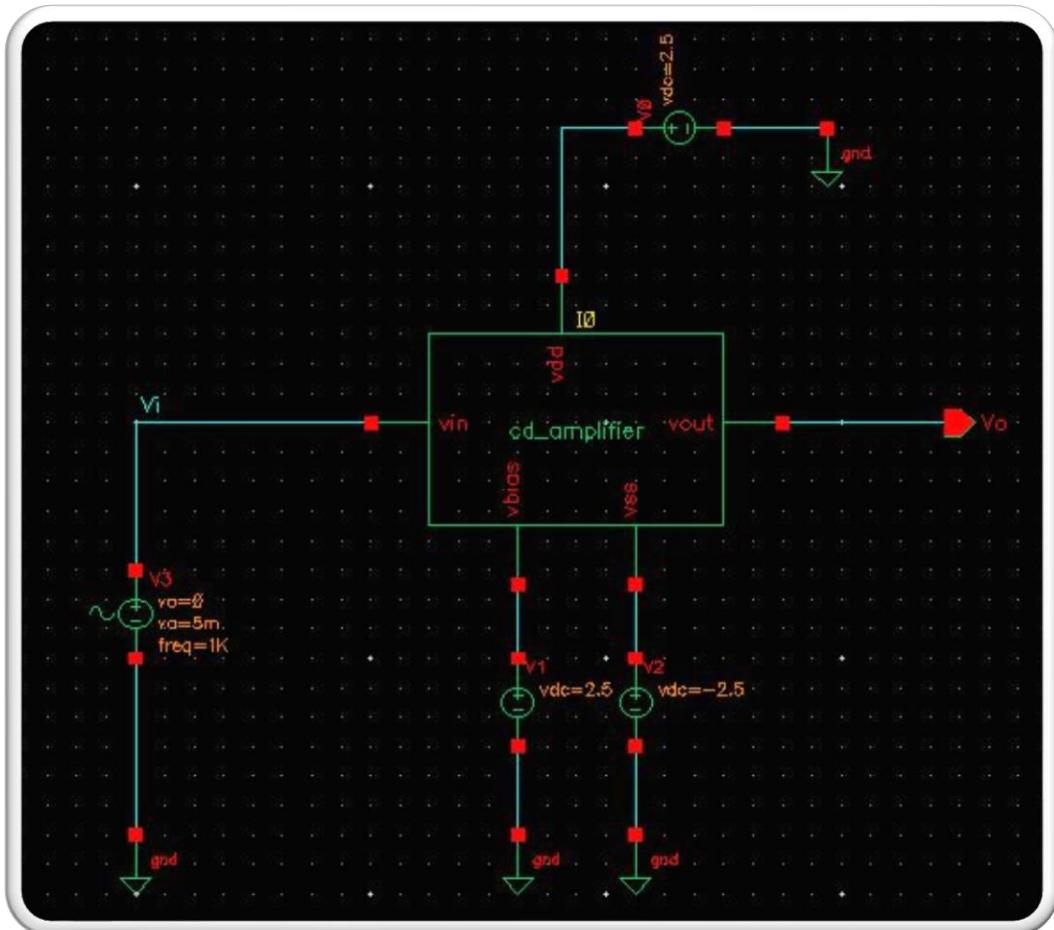
Building the Common Drain Amplifier Test Design

Objective: To build cd_amplifier_test circuit using your cd_amplifier

Using the component list and Properties/Comments in the table, build the cd-mplifier_test

schematic as shown below.

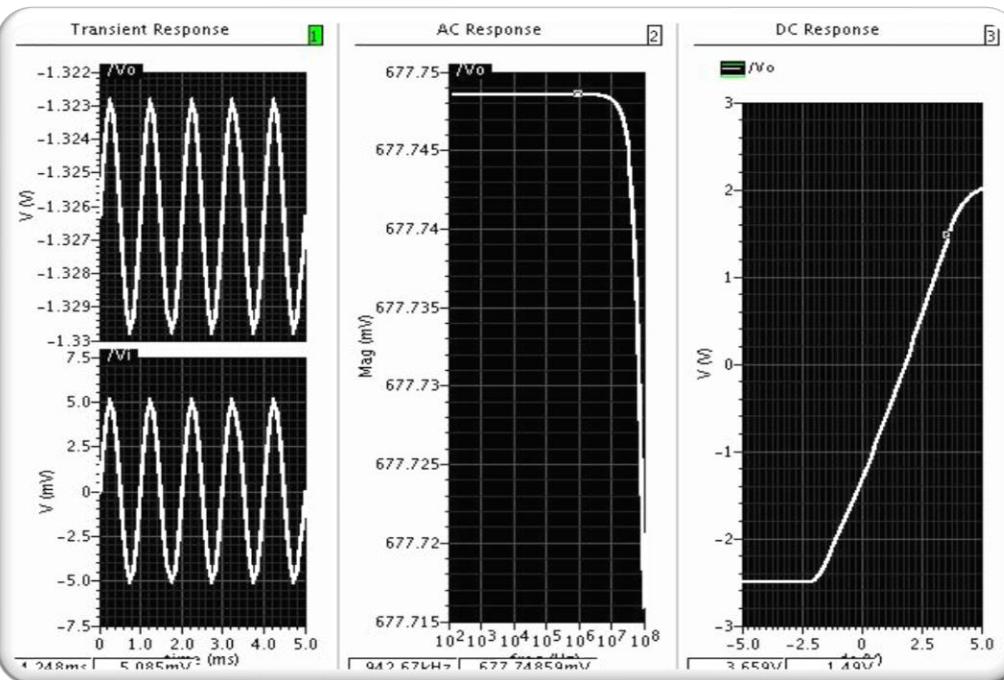
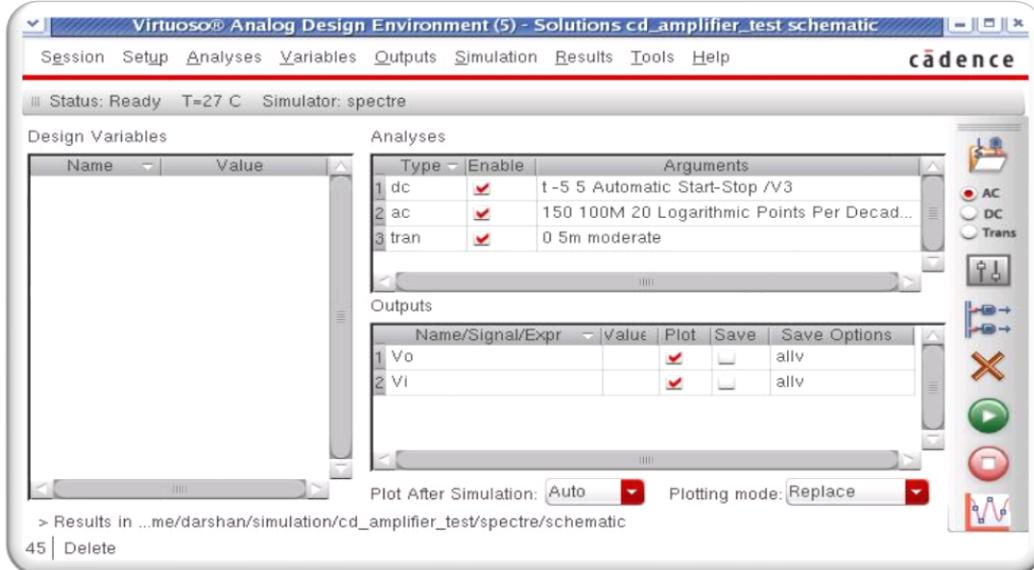
Library name	Cell Name	Properties/Comments
myDesignLib	cd_amplifier	Symbol
analogLib	vsin	Define pulse specification as AC Magnitude= 1; DC Voltage= 0; Offset Voltage= 0; Amplitude= 5m; Frequency= 1K
analogLib	vdd,vss,gnd	vdd=2.5 ; vss= -2.5



Analog Simulation with Spectre

Objective: To set up and run simulations on the cd_amplifier_test design.

Use the techniques learned in the Lab1 and Lab2 to complete the simulation of cd_amplifier, ADE window and waveform should look like below.

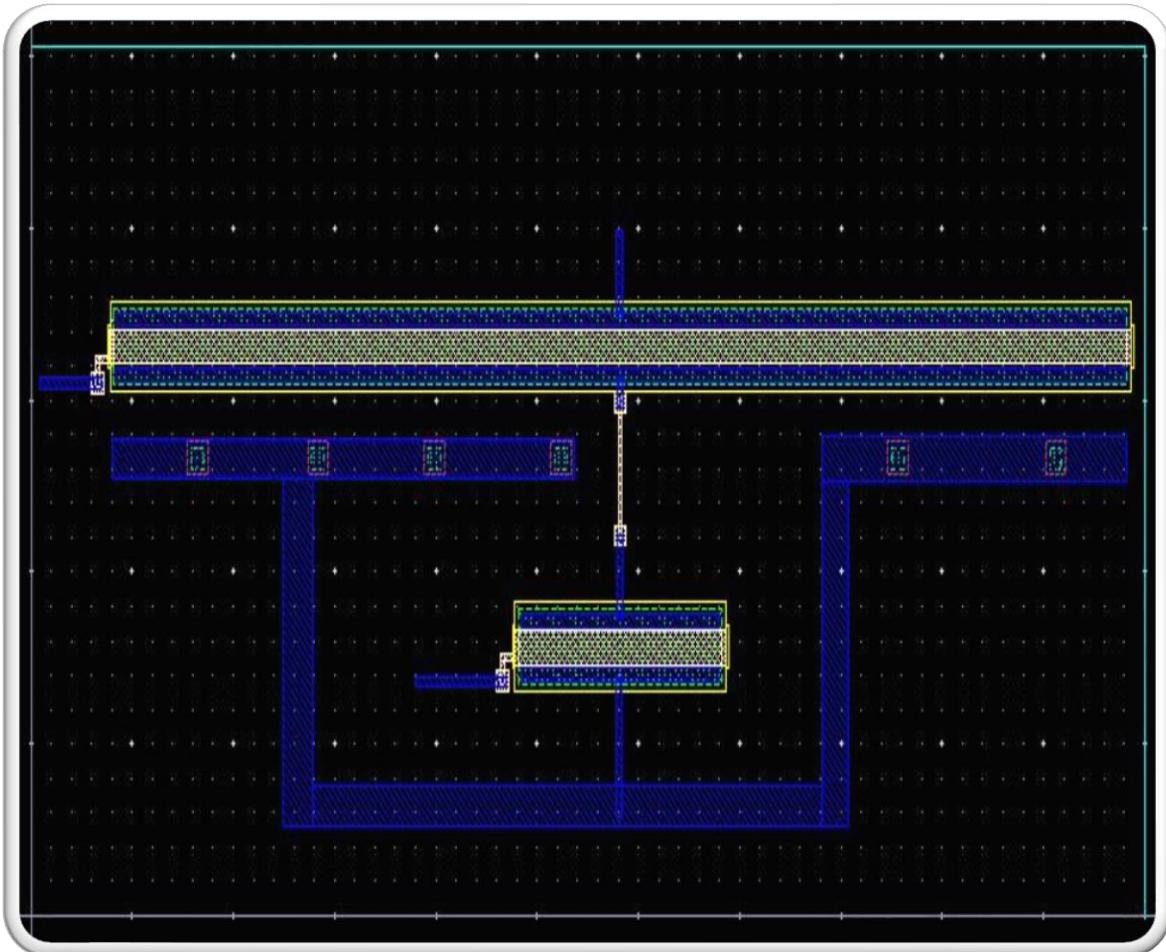


Creating a layout view of Common Drain Amplifier

Use the techniques learned in the Lab1 and Lab2 to complete the layout of cd_amplifier.

Complete the DRC, LVS check using the assura tool.

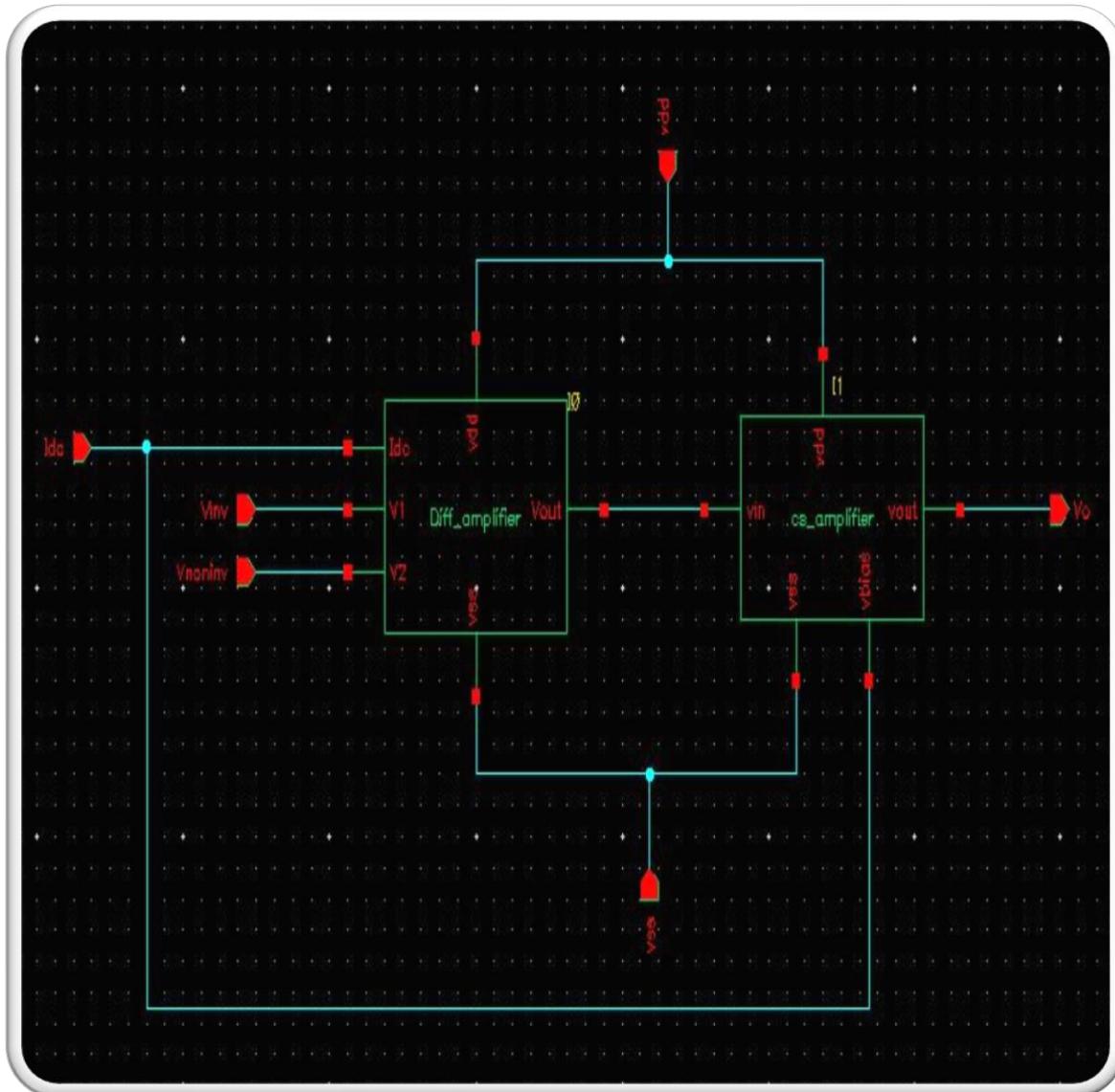
Extract RC parasites for back annotation and Re-simulation.



3. Design an op-amp with given specification using given differential amplifier

Common source amplifier in library and completing the design flow mentioned below:

- Draw the schematic and verify the following
i) DC Analysis ii). AC Analysis iii) Transient Analysis
- Draw the Layout and verify the DRC, ERC

Schematic Capture

Schematic Entry

Objective: To create a new cell view and build Operational Amplifier

Use the techniques learned in the Lab1 and Lab2 to complete the schematic of Operational Amplifier. This is a table of components for building the Operational Amplifier schematic.

Library name	Cell Name	Properties/Comments
myDesignLib	Diff_amplifier	Symbol
myDesignLib	cs_amplifier	Symbol

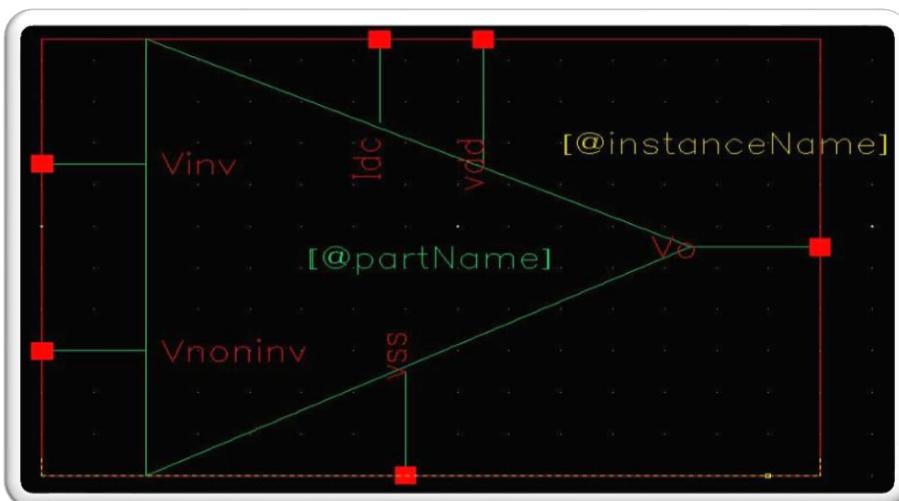
Type the following in the ADD pin form in the exact order leaving space between the pin names.

Pin Names	Direction
Idc,Vinv,Vnoninv	Input
Vo	Output
vdd, vss	Input

Symbol Creation

Objective: To create a symbol for the Operational Amplifier

Use the techniques learned in the Lab1 and Lab2 to complete the symbol of op-amp.



Building the Operational Amplifier Test

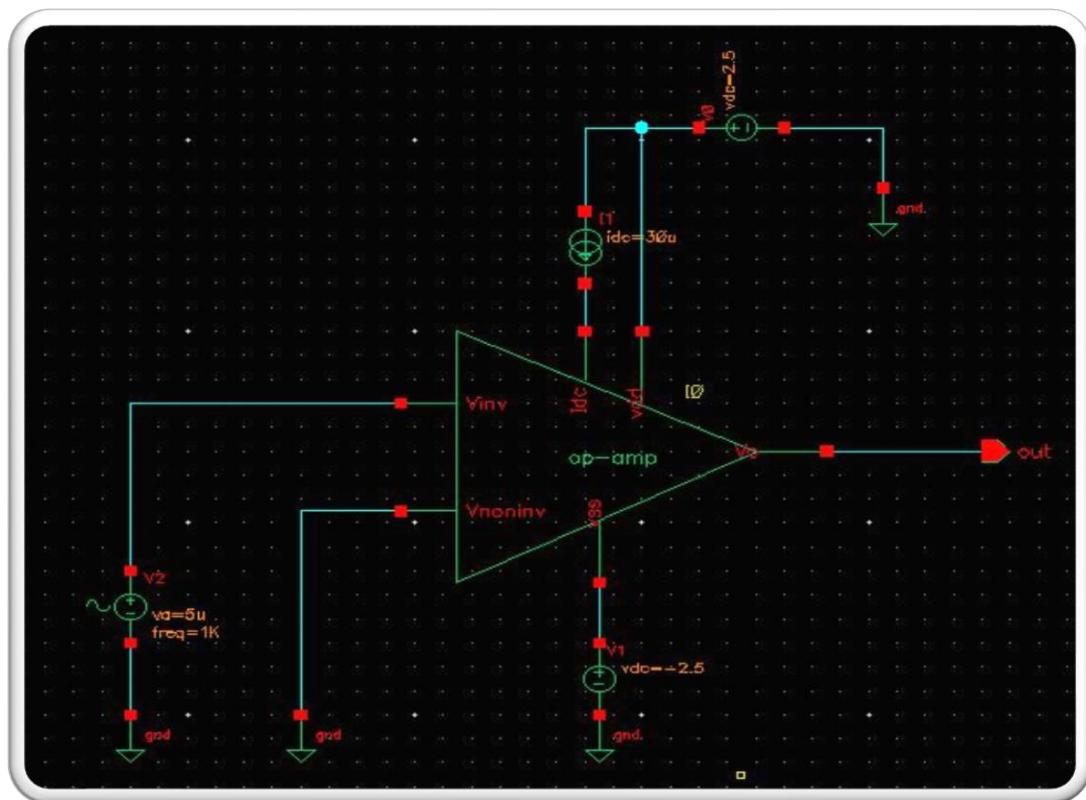
Design

Objective: To build op-amp_test circuit using your op-amp

Using the component list and Properties/Comments in the table, build the op-amp_test schematic as shown below.

Library name	Cellview name	Properties/Comments
myDesignLib	op-amp	Symbol
analogLib	vsin	Define pulse specification as AC Magnitude= 1; DC Voltage= 0; Offset Voltage= 0; Amplitude= 5m; Frequency= 1K
analogLib	vdc, gnd	vdd=2.5 ; vss= -2.5
analogLib	Idc	Dc current = 30u

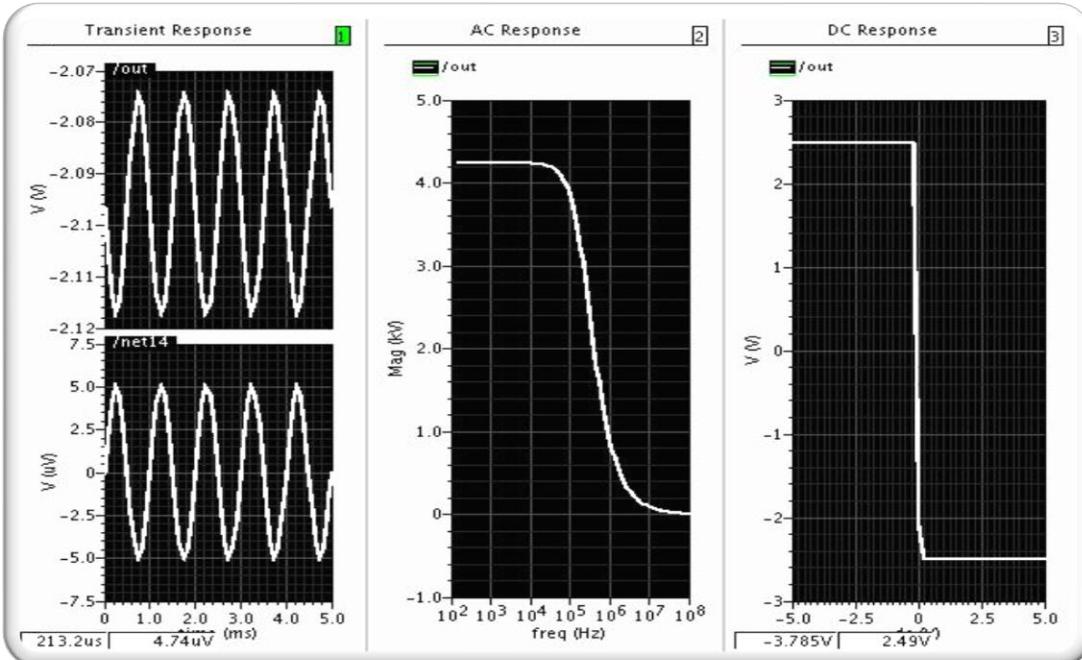
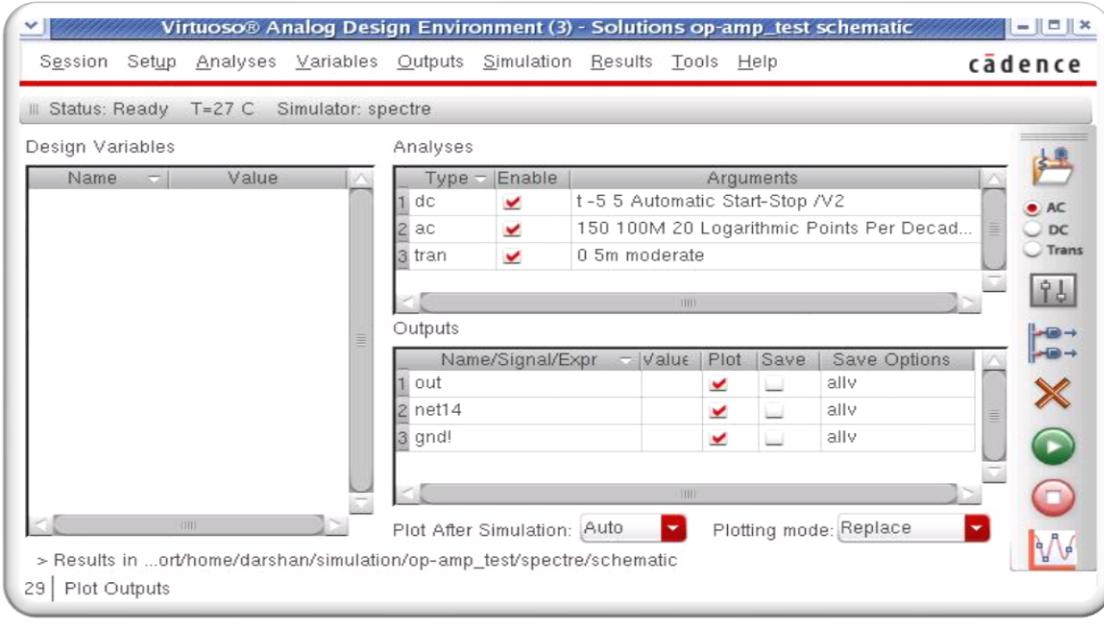
Note: Remember to set the values for **vdd** and **vss**. Otherwise your circuit will have no power.



Analog Simulation with Spectre

Objective: To set up and run simulations on the op-amp_test design.

Use the techniques learned in the Lab1 and Lab2 to complete the simulation of op-amp, ADE window and waveform should look like below.

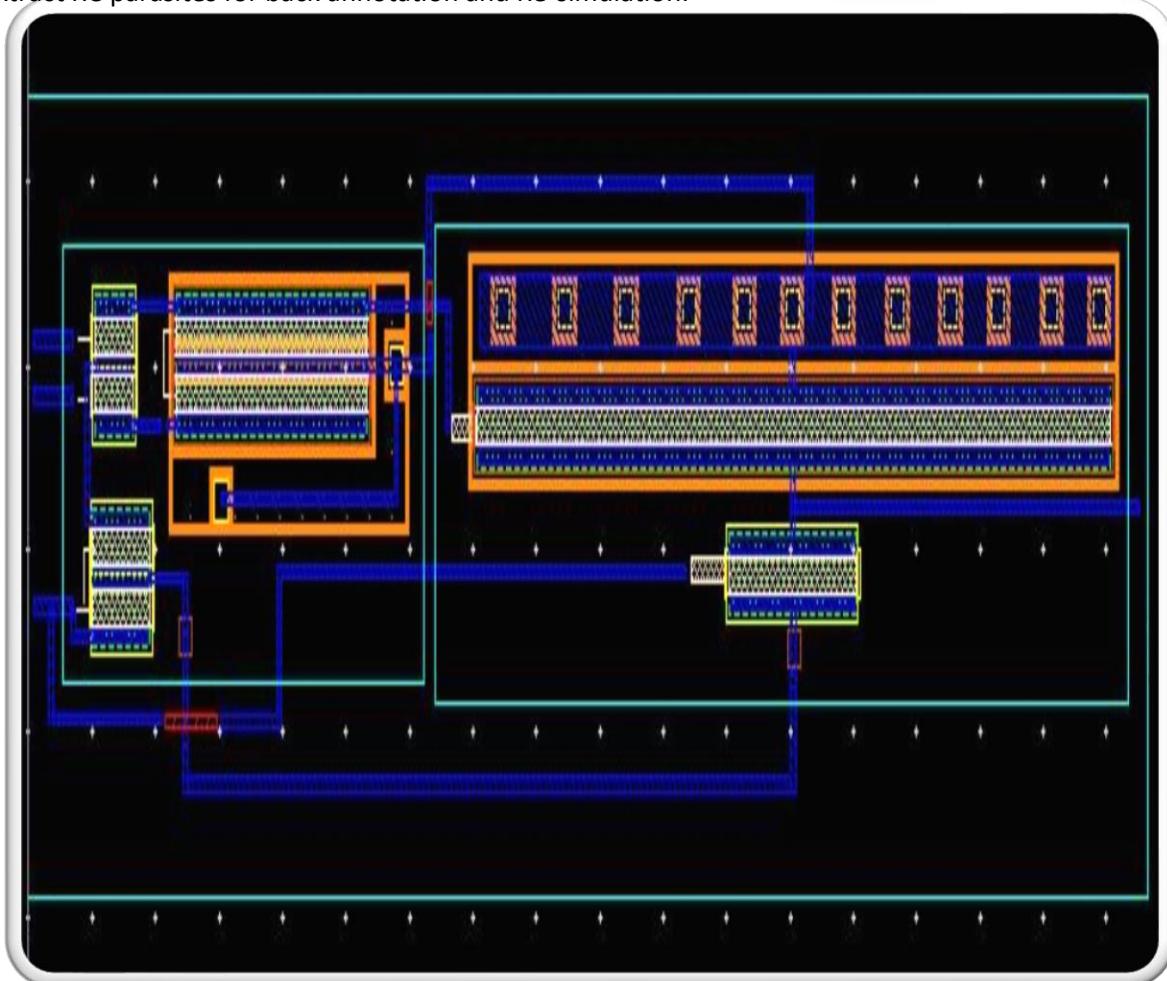


Creating a layout view of Operational**Amplifier**

Use the techniques learned in the Lab1 and Lab2 to complete the layout of op-amp.

Complete the DRC, LVS check using the assura tool.

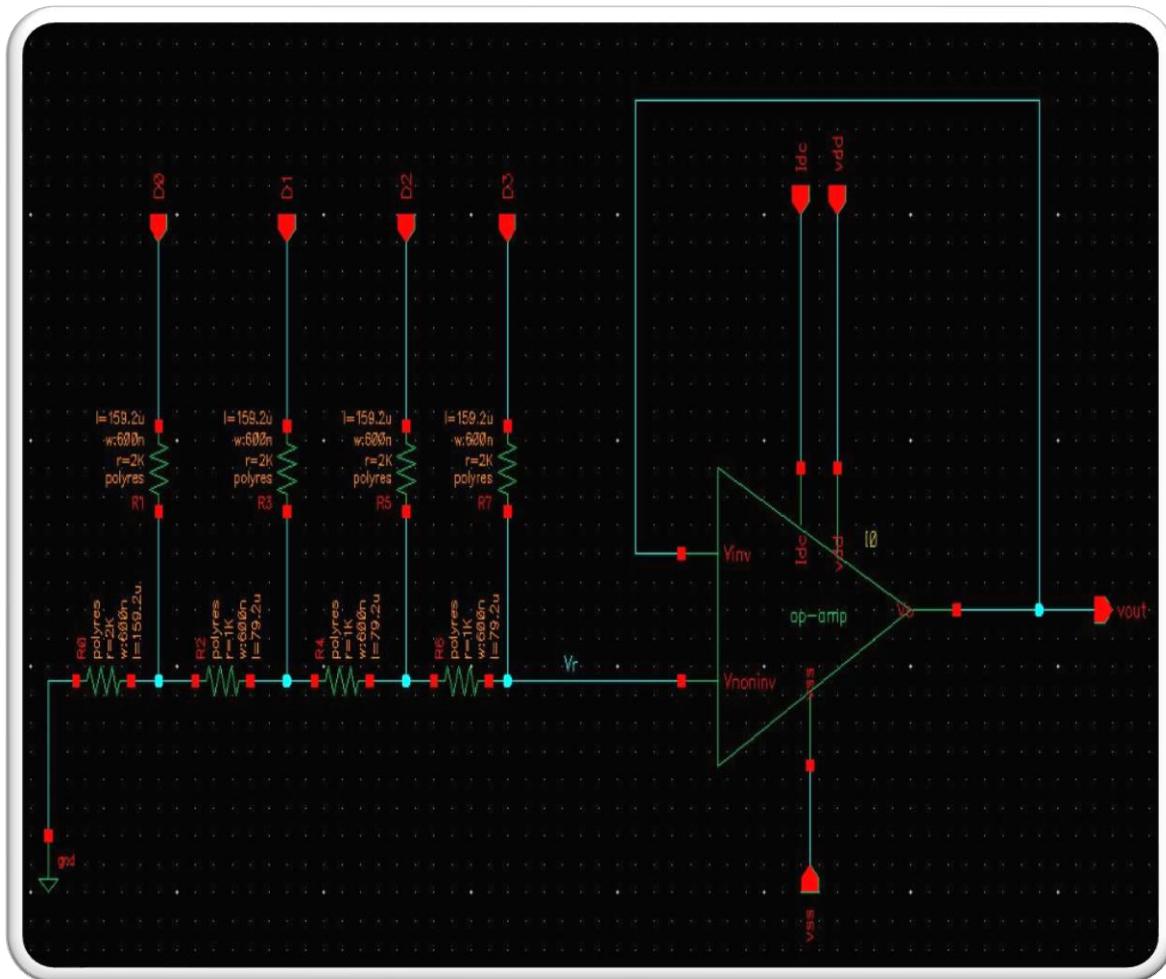
Extract RC parasites for back annotation and Re-simulation.



4. Design a 4 bit R-2R based DAC for the given specification and completing the design flow mentioned using given op-amp in the library.

- a. Draw the schematic and verify the following
 - i) DC Analysis ii) AC Analysis iii) Transient Analysis
- b. Draw the Layout and verify the DRC, ERC

Schematic Capture



Schematic Entry

Objective: To create a new cell view and build R-2R DAC

Use the techniques learned in the Lab1 and Lab2 to complete the schematic of R-2R DAC. This is a table of components for building the R-2R DAC schematic.

Library name	Cell Name	Properties/Comments
gpdk180	polyres	R = 2k
gpdk180	polyres	R = 1k
MyDesignLib	op-amp	Symbol
analogLib	ldc, gnd	idc = 30u

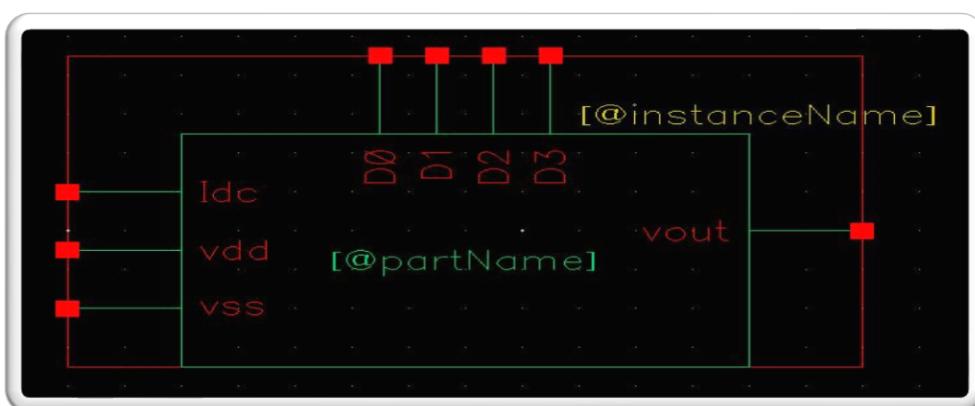
Type the following in the ADD pin form in the exact order leaving space between the pin names.

Pin Names	Direction
Do D1 D2 D3	Input
Vout	Output
vdd, vss	Input

Symbol Creation

Objective: To create a symbol for the R-2R DAC

Use the techniques learned in the Lab1 and Lab2 to complete the symbol of R-2R DAC.



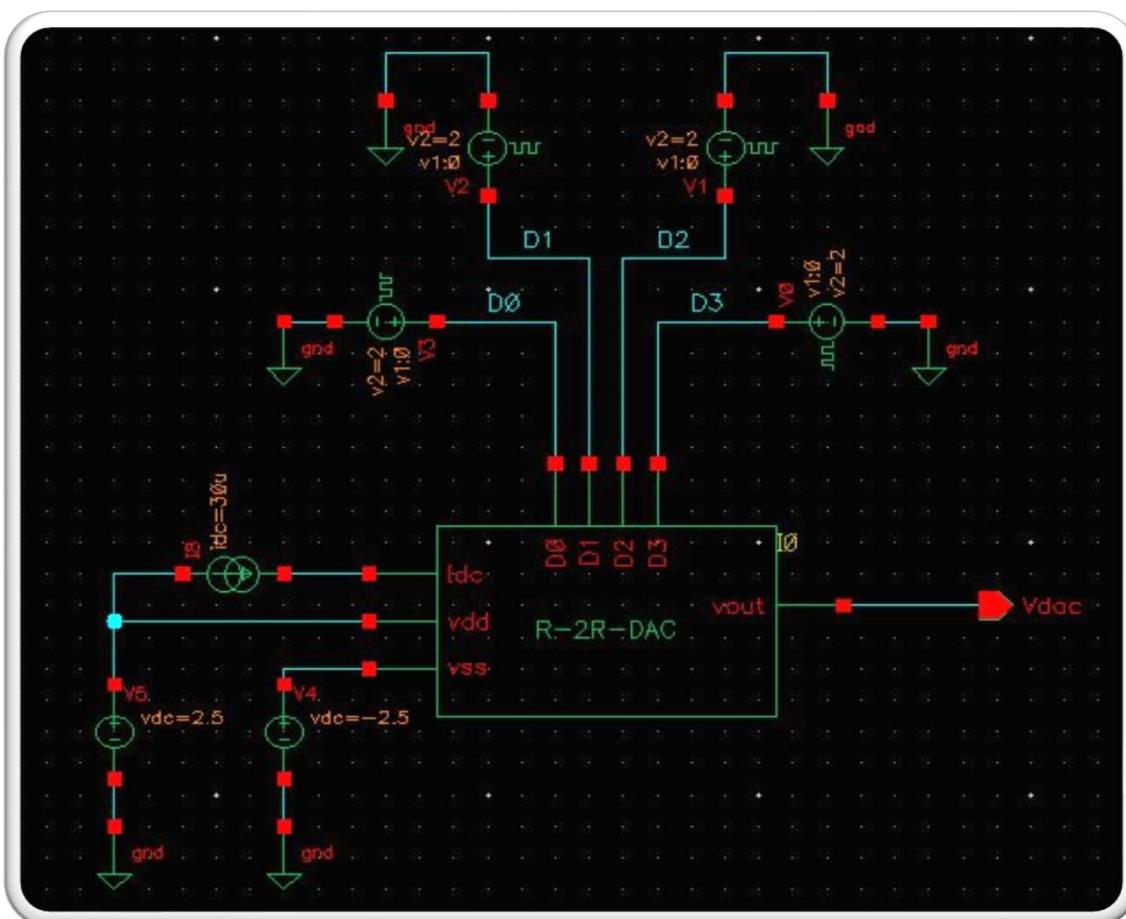
Building the R-2R DAC Test Design

Objective: To build R-2R DAC Test circuit using your R-2R DAC

Using the component list and Properties/Comments in the table, build the R-2R-AC_test schematic as shown below.

Library name	Cell Name	Properties/Comments
myDesignLib	R-2R-DAC	Symbol
analogLib	vpulse	For V0: v1= 0 v2 = 2 Ton = 5n T = 10n
		For V1: v1= 0 v2 = 2 Ton = 10n T = 20n
		For V2: v1= 0 v2 = 2 Ton = 20n T = 40n
		For V3: v1= 0 v2 = 2 Ton = 40n T = 80n
analogLib	vdc, gnd	vdd = 2 vss = -2

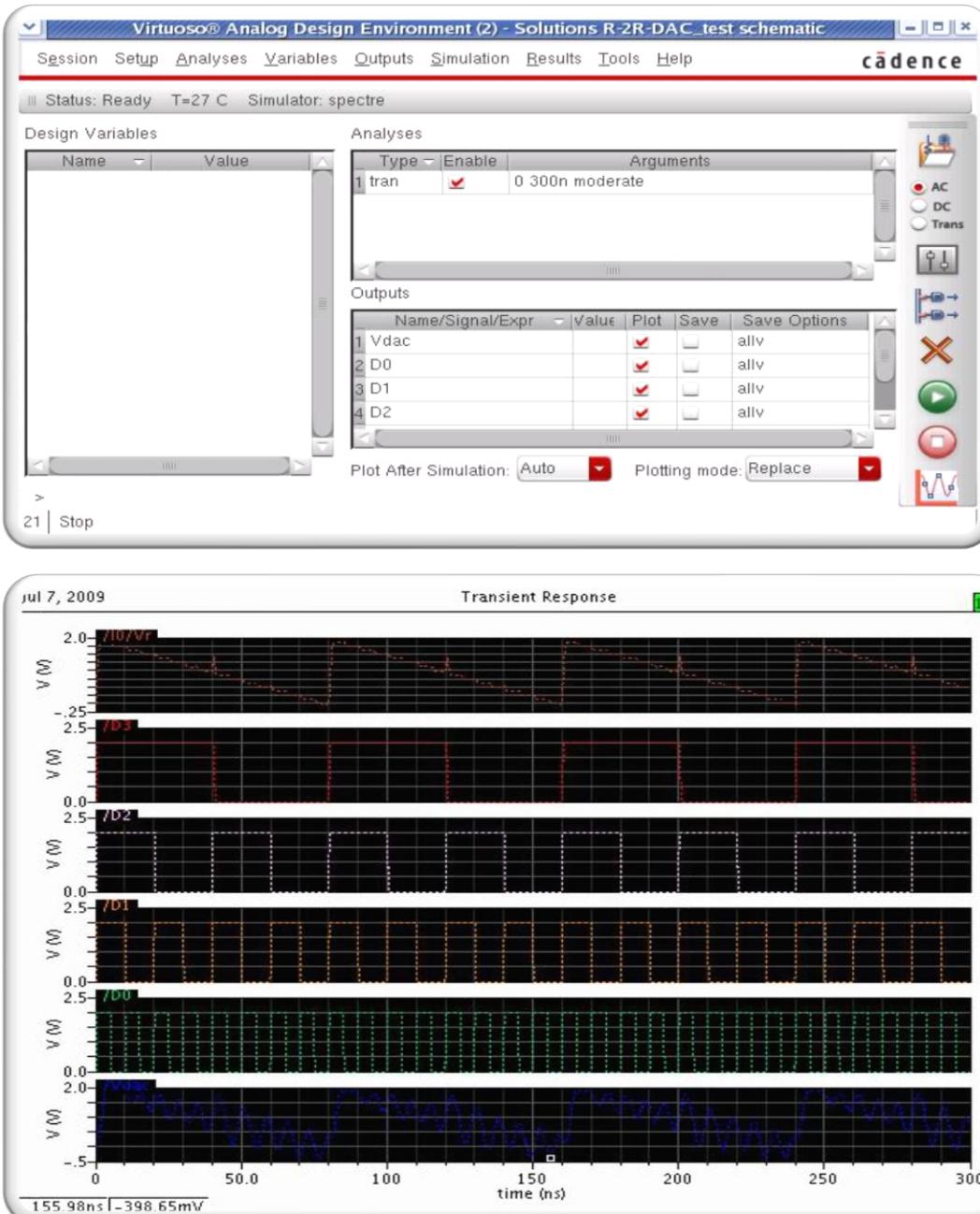
Note: Remember to set the values for **vdd** and **vss**. Otherwise your circuit will have no power.



Analog Simulation with Spectre

Objective: To set up and run simulations on the R-2R-DAC_Test design.

Use the techniques learned in the Lab1 and Lab2 to complete the simulation of R-2R- DAC and ADE window and waveform should look like below.

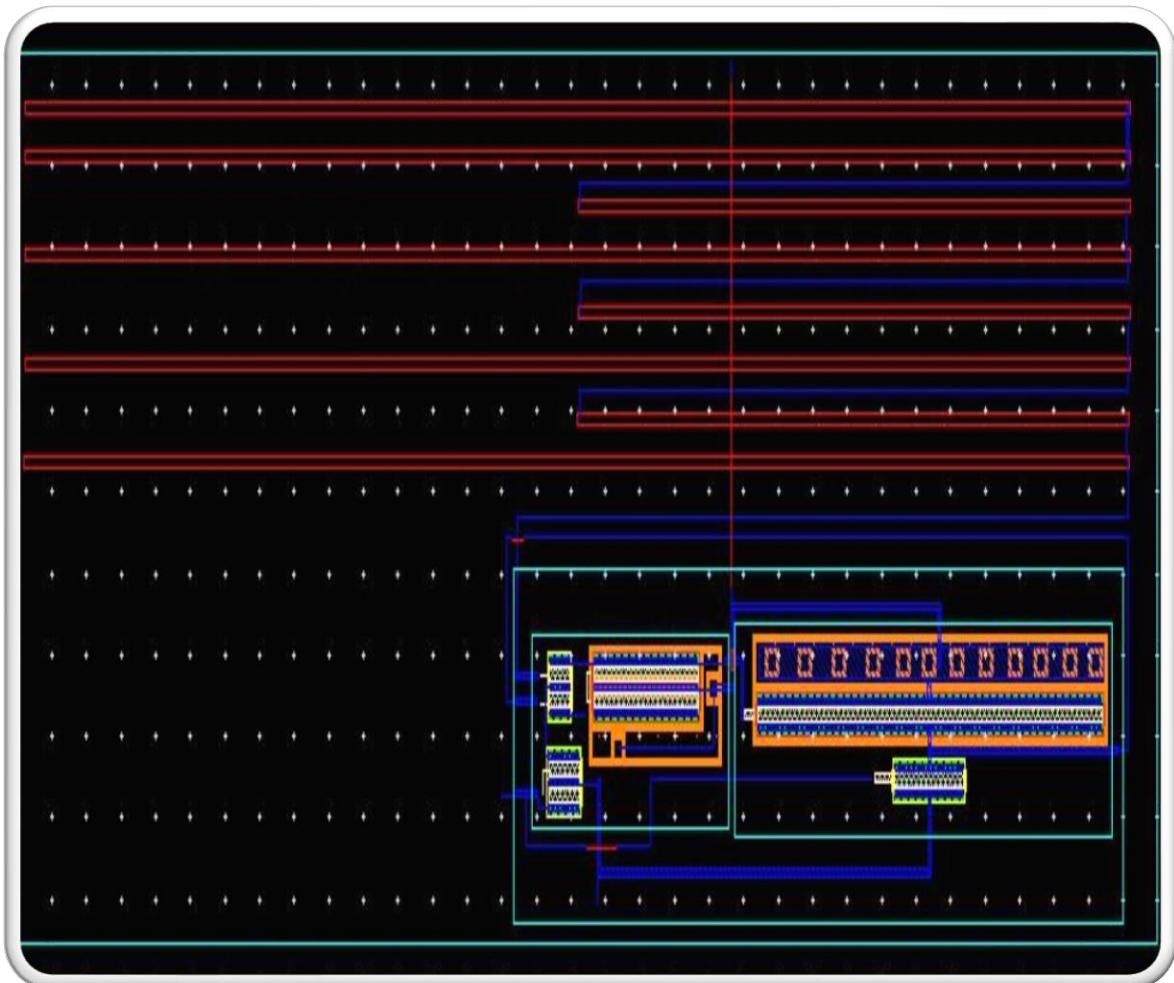


Creating a layout view of R-2R DAC

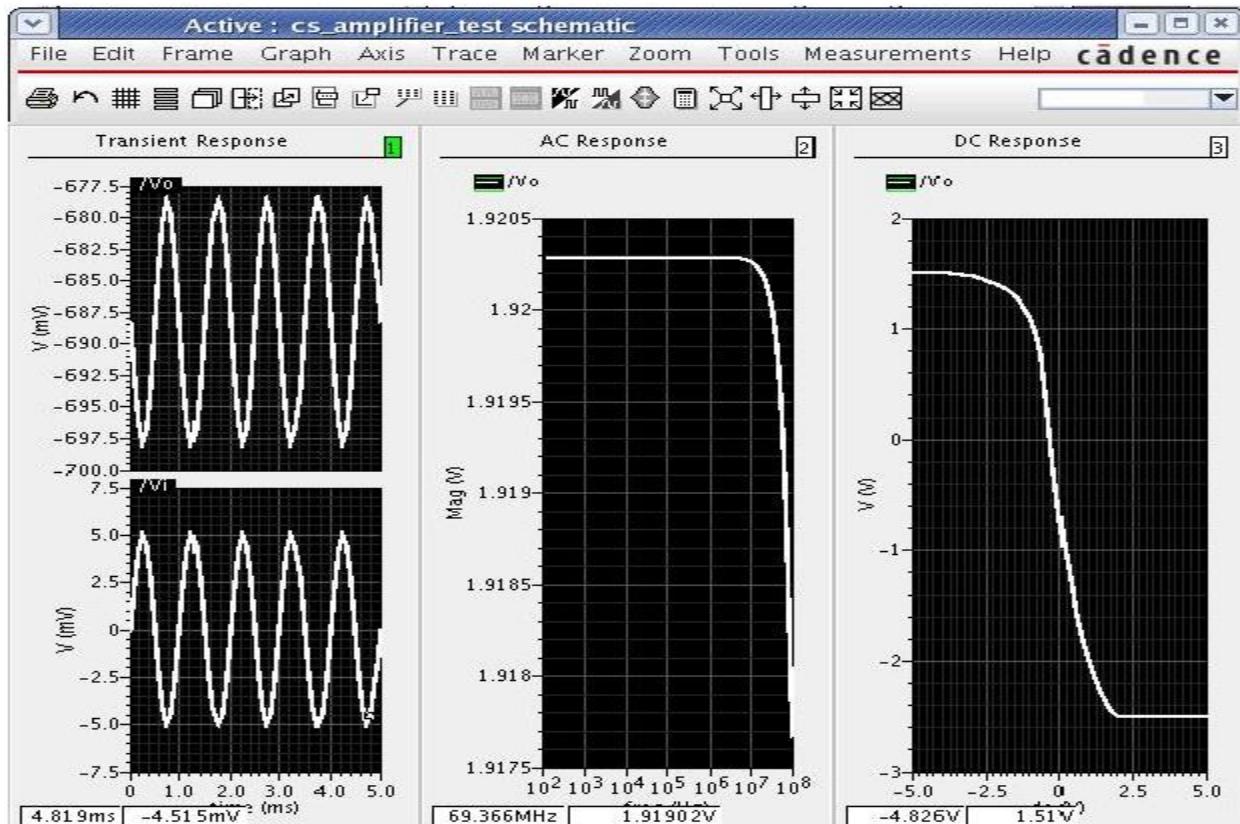
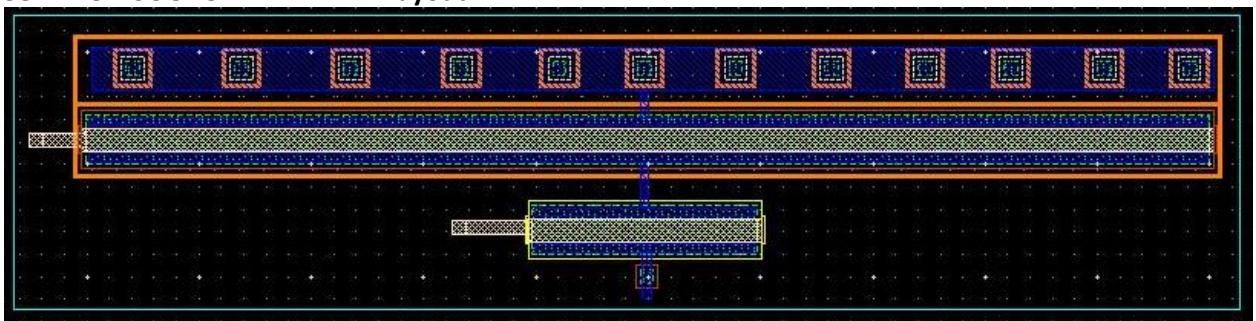
Use the techniques learned in the Lab1 and Lab2 to complete the layout of R-2R-DAC.

Complete the DRC, LVS check using the assura tool.

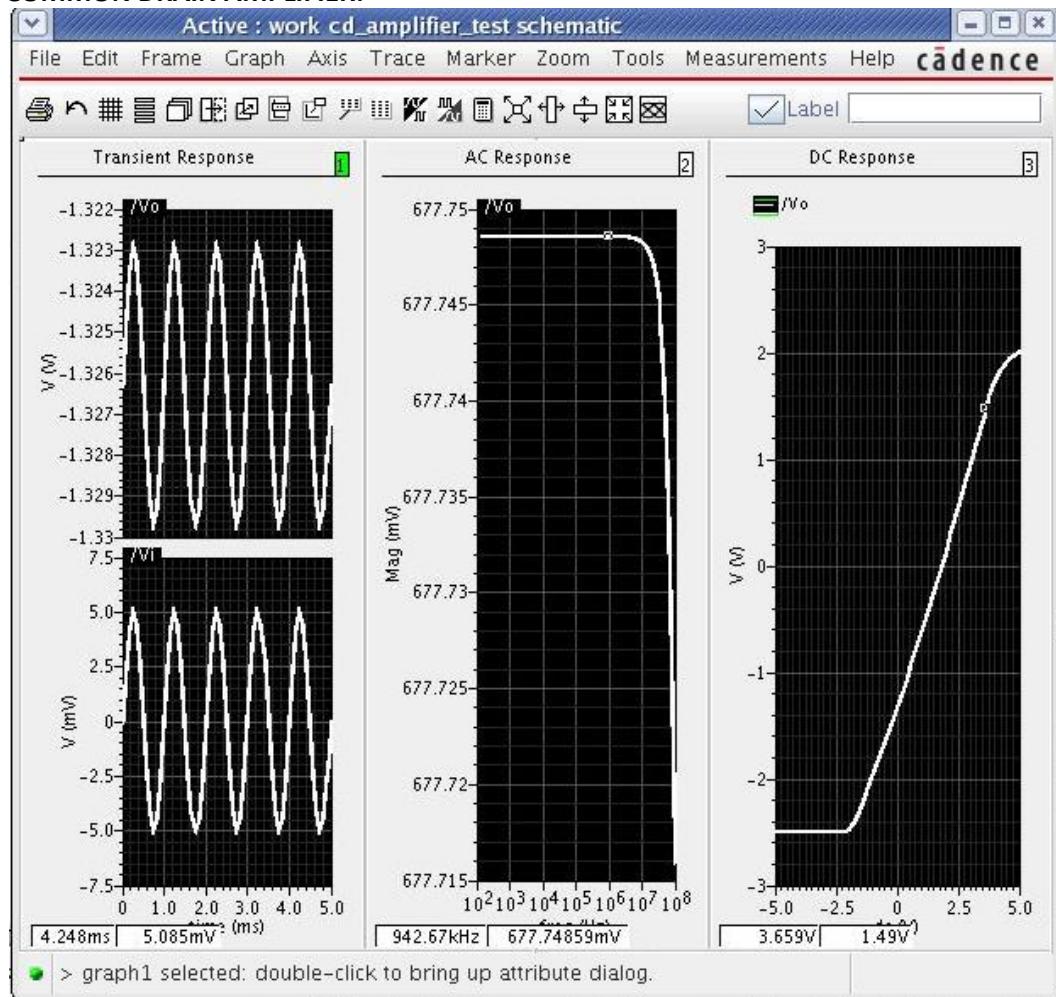
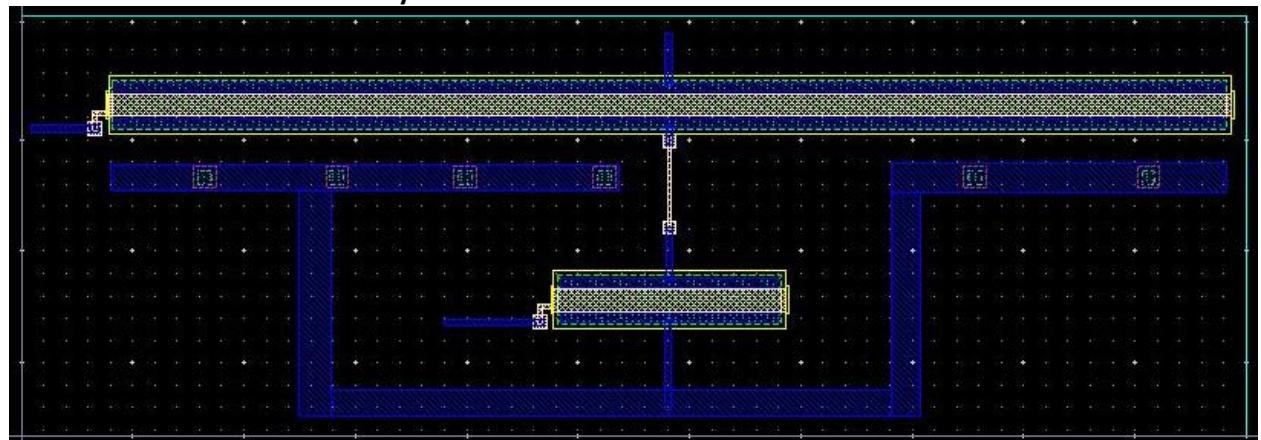
Extract RC parasites for back annotation and Re-simulation.



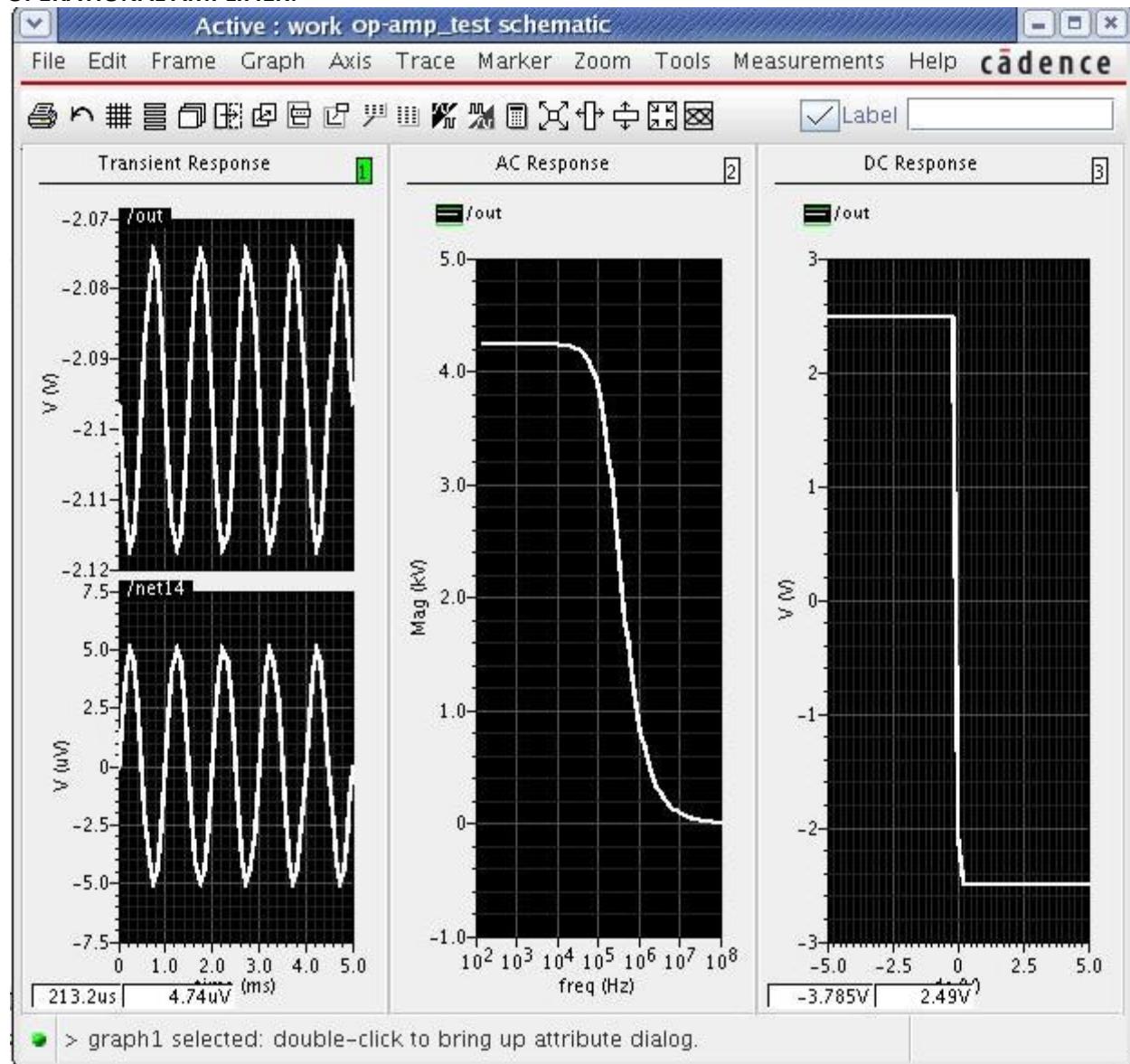
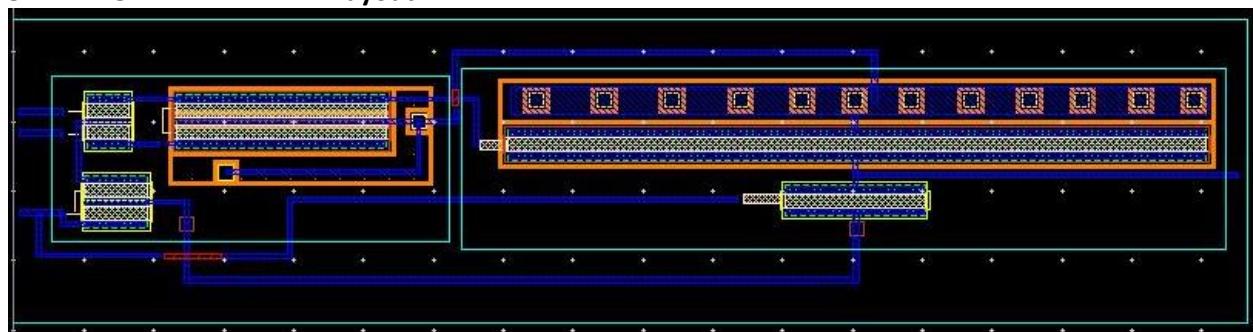
This page is intentionally left blank

COMMON SOURCE AMPLIFIER:**COMMON SOURCE AMPLIFIER: Layout**

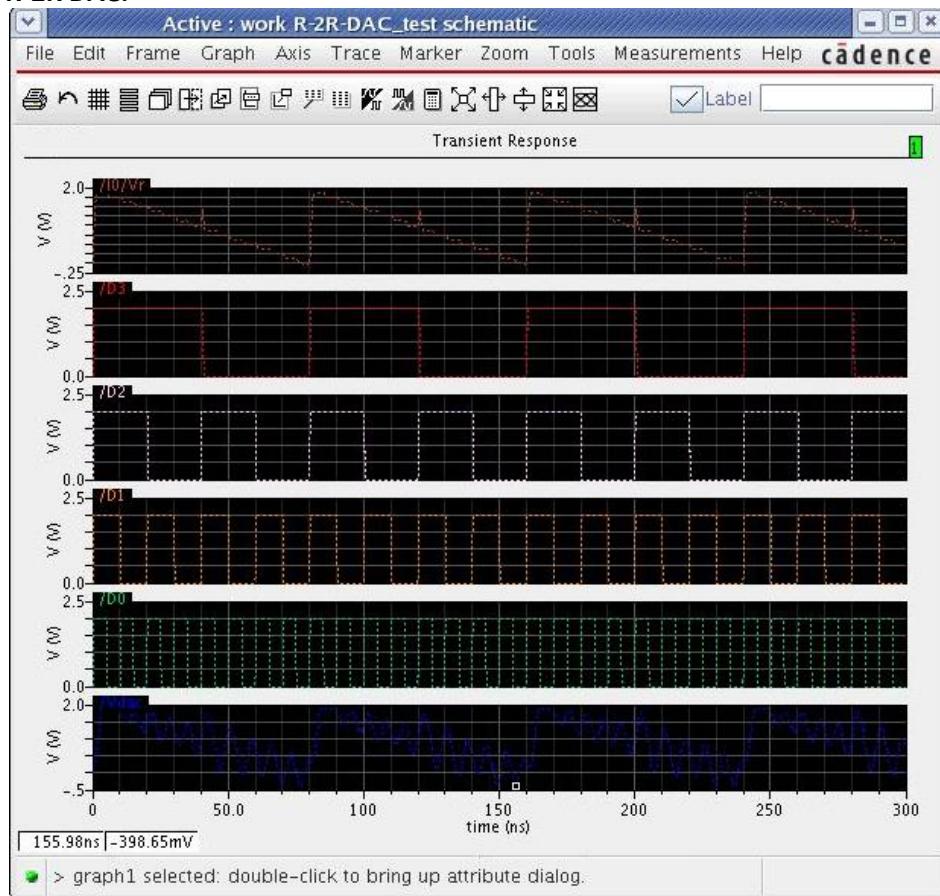
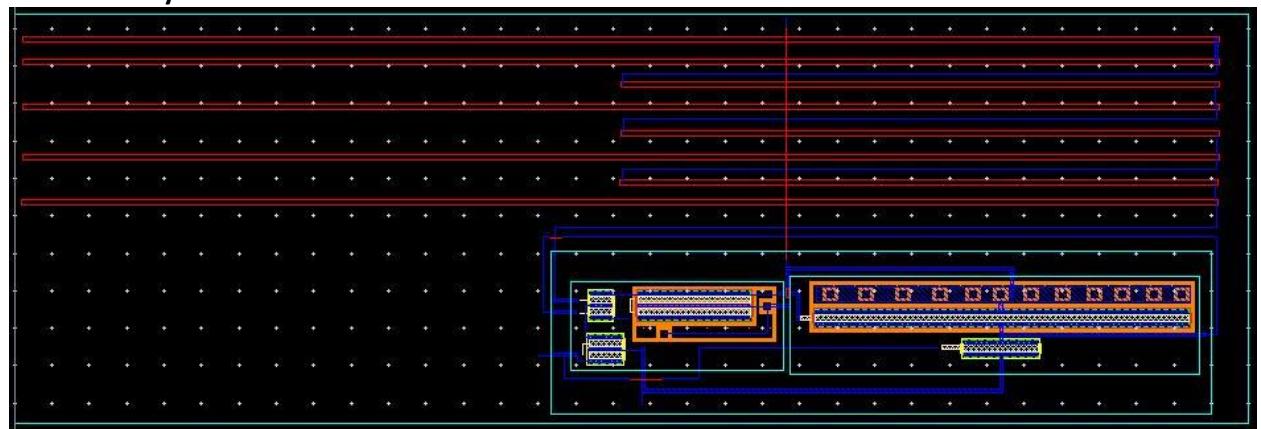
This page is intentionally left blank

COMMON DRAIN AMPLIFIER:**COMMON DRAIN AMPLIFIER: Layout**

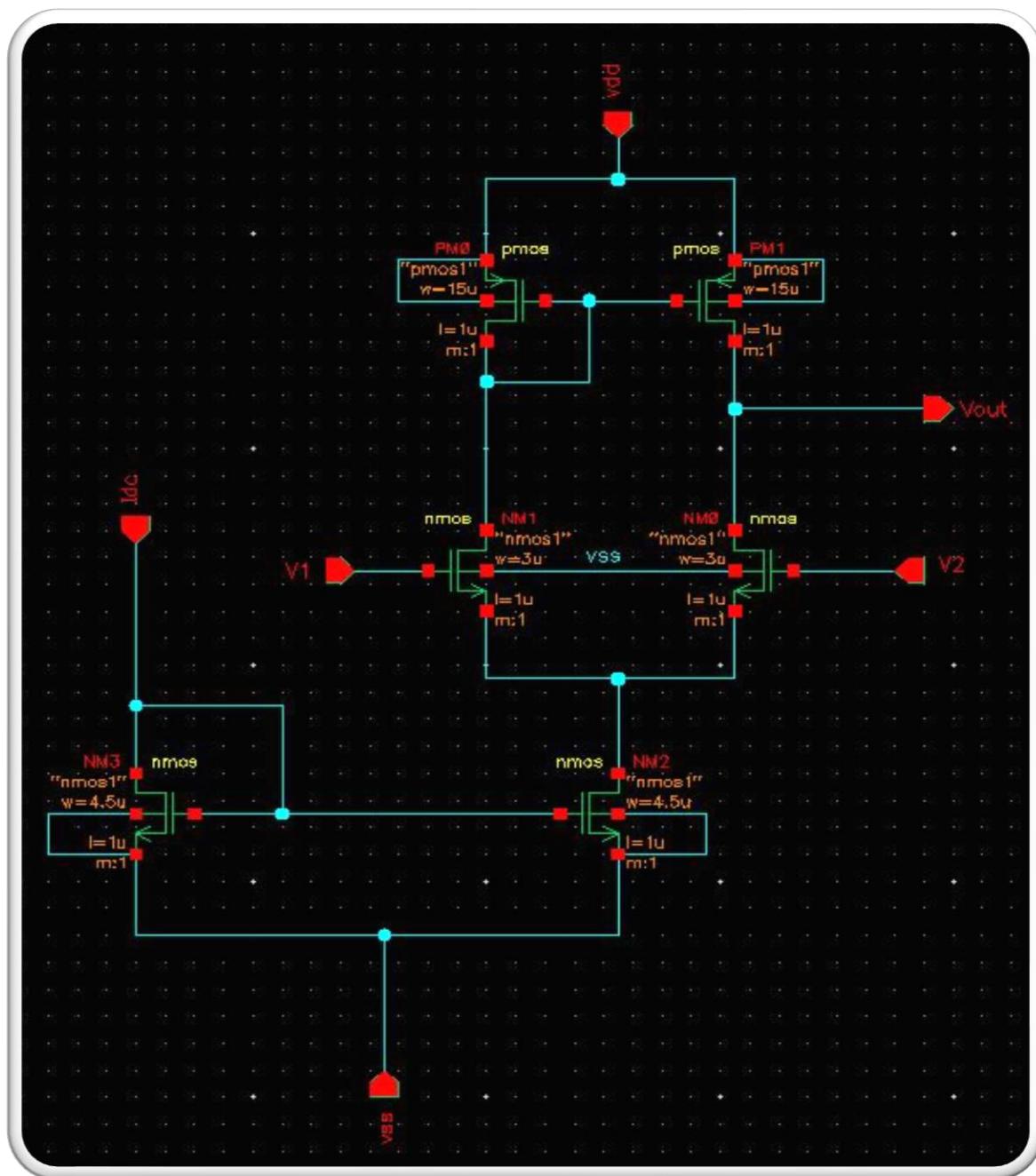
This page is intentionally left blank

OPERATIONAL AMPLIFIER:**OPERATIONAL AMPLIFIER: Layout**

This page is intentionally left blank

R-2R DAC:**R-2R DAC: Layout**

This page is intentionally left blank

DIFFERENTIAL AMPLIFIER**Schematic Capture**

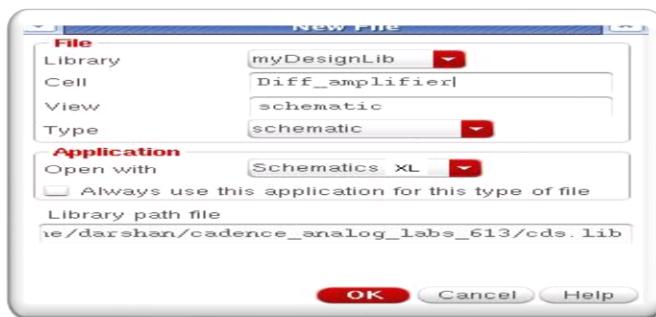
Schematic Entry

Objective: To create a new cell view and build Differential Amplifier

Creating a Schematic cellview

Open a new schematic window in the **myDesignLib** library and build the Differential_Amplifier design.

1. In the CIW or Library manager, execute **File – New – Cellview**. Set up the Create New file form as follows:



3. Click **OK** when done. A blank schematic window for the design appears.

Adding Components to schematic

1. In the Differential Amplifier schematic window, execute **Create— Instance** to display the Add Instance form. (use sideways option to rotate the transistor before placing)

2. Click on the **Browse** button. This opens up a Library browser from which you can select components and the **Symbol** view .

You will update the Library Name, Cell Name, and the property values given in the table on the next page as you place each component. (Select sideways to rotate the transistor)

3. After you complete the Add Instance form, move your cursor to the schematic window and click **left** to place a component.

This is a table of components for building the Differential Amplifier schematic.

Library name	Cell Name	Properties/Comments
gdk180	pmos	Model Name =pmos1 (PM0, PM1); W= 15u ; L= 1u
gdk180	nmos	Model Name = nmos1 (NM0, NM1) ; W= 3u ; L= 10u
gdk180	nmos	Model Name = nmos1 (NM0, NM1) ; W= 4.5u ; L= 10u

4. After entering components, click **Cancel** in the Add Instance form or press **Esc** with your cursor in the schematic window.

Adding pins to Schematic

Use **Create – Pin** or the menu icon to place the pins on the schematic window.

1. Click the **Pin** fixed menu icon in the schematic window.

You can also execute **Create – Pin** or press **p**. The Add pin form appears.

2. Type the following in the Add pin form in the exact order leaving space between the pin names.

Direction	Pin Names
Input	Idc,V1,V2
Output	Vout
Input	vdd, vss,

Make sure that the direction field is set to **input/output/inputoutput** when placing the **input/output/inout** pins respectively and the Usage field is set to **schematic**.

3. Select **Cancel** from the Add pin form after placing the pins.

In the schematic window, execute **View— Fit** or press the **f** bindkey.

Adding Wires to a Schematic

Add wires to connect components and pins in the design.

1. Click the **Wire (narrow)** icon in the schematic window.

You can also press the **w** key, or execute **Create - Wire (narrow)**.

2. Complete the wiring as shown in figure and when done wiring press **ESC** key in the schematic window to cancel wiring.

Saving the Design

1. Click the **Check and Save** icon in the schematic editor window.

2. Observe the **CIW** output area for any errors.

Symbol Creation

Objective: To create a symbol for the Differential Amplifier

1. In the Differential Amplifier schematic window, execute

Create — Cellview— From Cellview.

The **Cellview from Cellview** form appears. With the Edit Options function active, you can control the appearance of the symbol to generate.

2. Verify that the **From View Name** field is set to **schematic**, and the **To View Name** field is set to **symbol**, with the Tool/Data Type set as **SchematicSymbol**.

3. Click **OK** in the Cellview from Cellview form. The **Symbol Generation Form** appears.

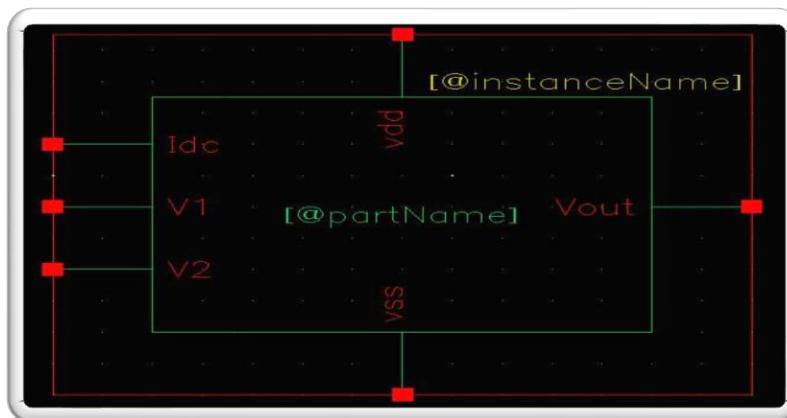
4. Modify the **Pin Specifications** as in the below symbol.

5. Click **OK** in the Symbol Generation Options form.

6. A new window displays an automatically created Differential Amplifier symbol.

7. Modifying automatically generated symbol so that it looks like below Differential Amplifier symbol.

8. Execute **Create— Selection Box**. In the Add Selection Box form, click **Automatic**. A new red selection box is automatically added.



9. After creating symbol, click on the **save** icon in the symbol editor window to save the symbol. In the symbol editor, execute **File— Close** to close the symbol view window.

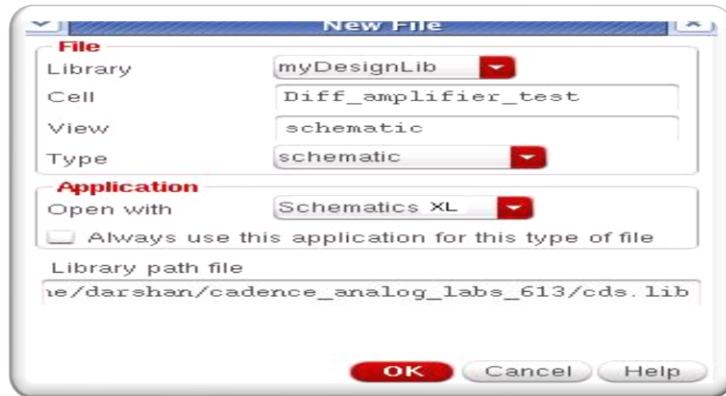
Building the Diff_amplifier_test Design

Objective: To build Differential Amplifier Test circuit using your Differential Amplifier

Creating the Differential Amplifier Test Cellview

1. In the CIW or Library Manager, execute **File— New— Cellview**.

2. Set up the Create New File form as follows:



3. Click **OK** when done. A blank schematic window for the Diff_amplifier_test design appears.

Building the Diff_amplifier_test Circuit

1. Using the component list and Properties/Comments in this table, build the Diff_amplifier_test schematic.

Library name	Cellview name	Properties/Comments
myDesignLib	Diff_amplifier	Symbol
analogLib	vsin	Define specification as AC Magnitude= 1; Amplitude= 5m; Frequency= 1K
analogLib	vdd, vss, gnd	Vdd=2.5 ; Vss= -2.5
analogLib	ldc	Dc current = 30u

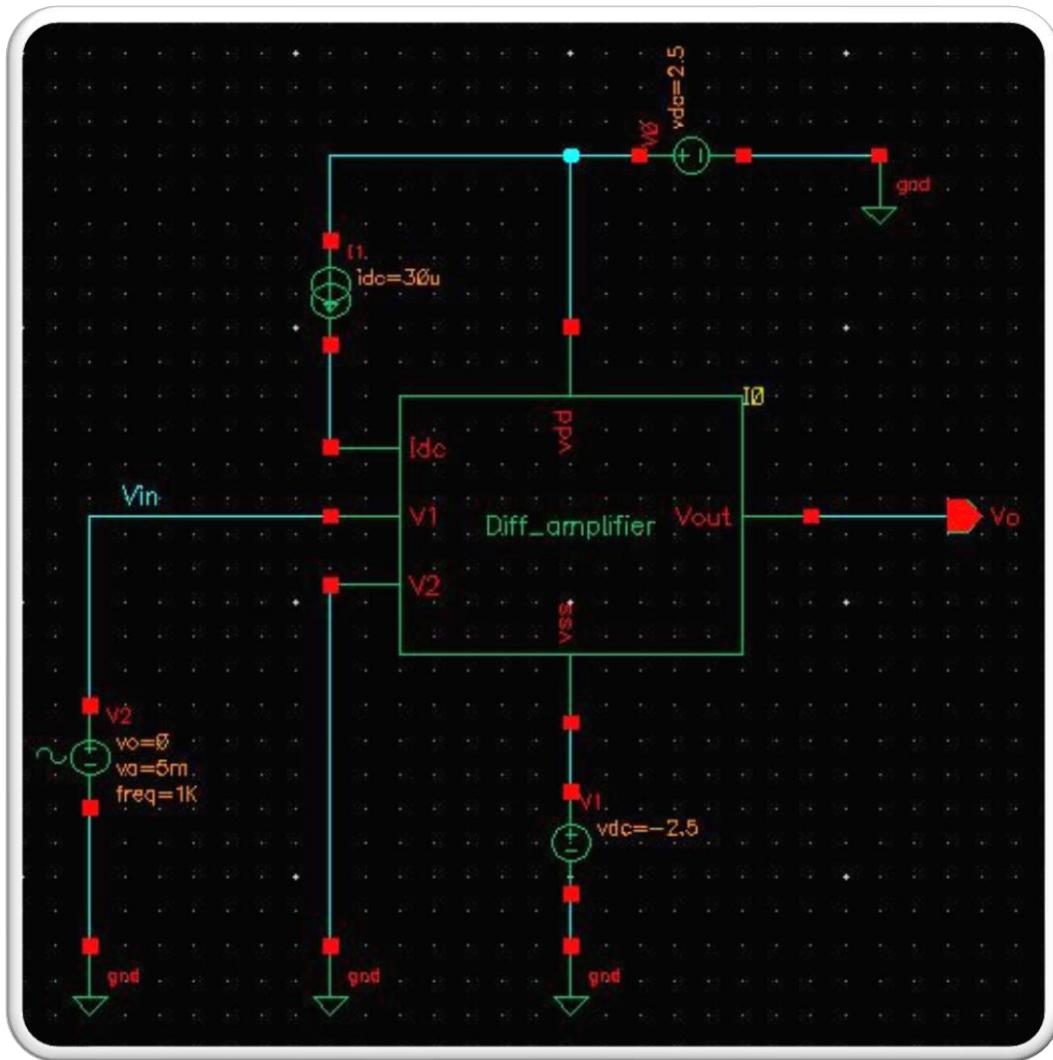
Note: Remember to set the values for **VDD** and **VSS**. Otherwise your circuit will have no power.

2. Click the **Wire (narrow)** icon and wire your schematic.

Tip: You can also press the **w** key, or execute **Create— Wire (narrow)**.

3. Click on the **Check and save** icon to save the design.

4. The schematic should look like this.



5. Leave your Diff_amplifier_test schematic window open for the next section.

Analog Simulation with Spectre

Objective: To set up and run simulations on the Differential Amplifier Test design.

In this section, we will run the simulation for Differential Amplifier and plot the transient, DC and AC characteristics.

Starting the Simulation Environment

1. In the Diff_amplifier_test schematic window, execute **Launch – ADE L**. The Analog Design Environment simulation window appears.

Choosing a Simulator

1. In the simulation window or ADE, execute

Setup— Simulator/Directory/Host.

2. In the **Choosing Simulator** form, set the Simulator field to **spectre** (Not spectreS) and click **OK**.

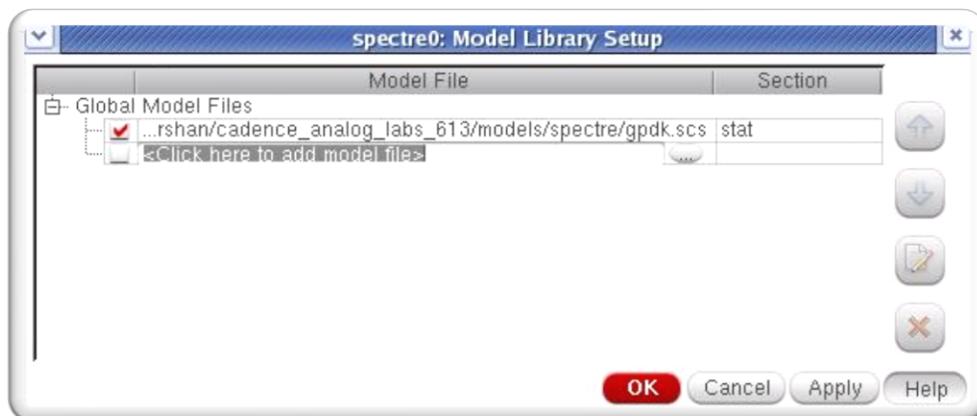
Setting the Model Libraries

1. Click **Setup - Model Libraries**.

Note: Step 2 should be executed only if the model file not loaded by default.

2. In the Model Library Setup form, click **Browse** and find the **gpdk180.scs** file in the **./models/spectre** directory.

Select **stat** in Section field, click **Add** and click **OK**.



Choosing Analyses

1. In the Simulation window, click the **Choose - Analyses** icon.

You can also execute **Analyses - Choose**.

The Choosing Analysis form appears. This is a dynamic form, the bottom of the form changes based on the selection above.

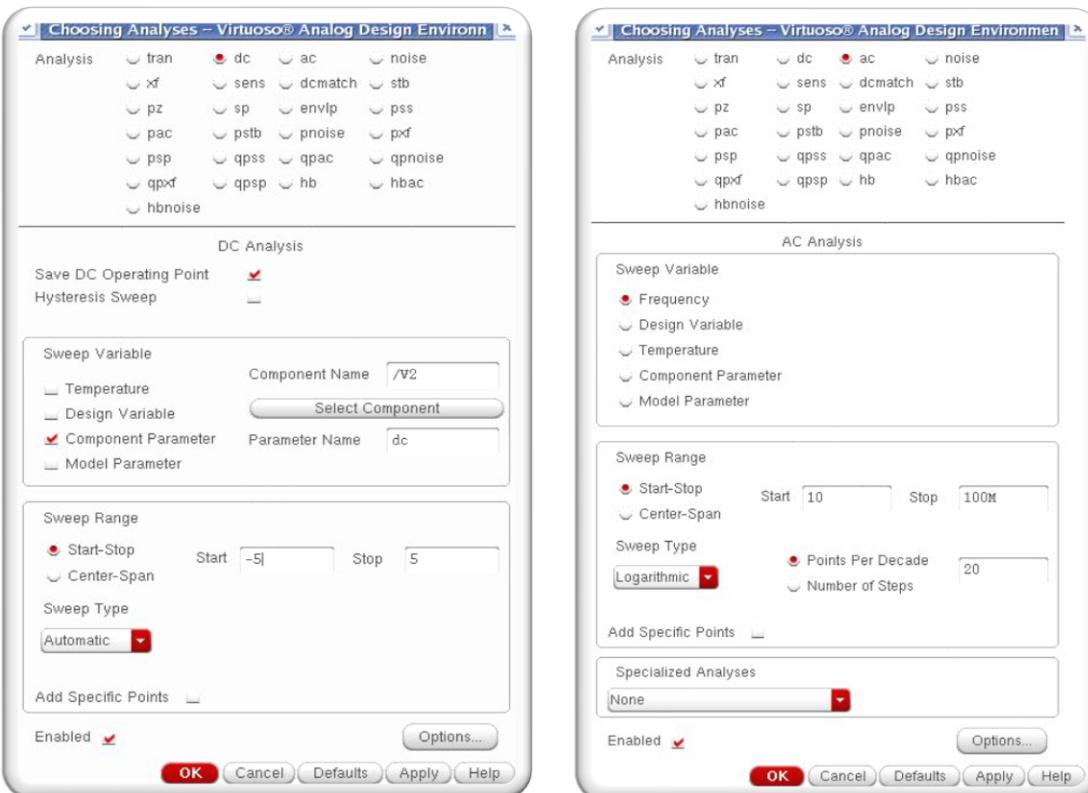
2. To setup for transient analysis

- a. In the Analysis section select **tran**
- b. Set the stop time as **5m**
- c. Click at the **moderate** or **Enabled** button at the bottom, and then click **Apply**.

3. To set up for DC Analyses:

- a. In the Analyses section, select **dc**.
- b. In the DC Analyses section, turn **on** Save DC Operating Point.
- c. Turn on the **Component Parameter**

- d. Double click the **Select Component**, Which takes you to the schematic window.
e. Select input signal **Vsin** for dc analysis.
f. In the analysis form, select **start** and **stop** voltages as **-5** to **5** respectively.
g. Check the enable button and then click **Apply**.



4. To set up for AC Analyses form is shown in the previous page.

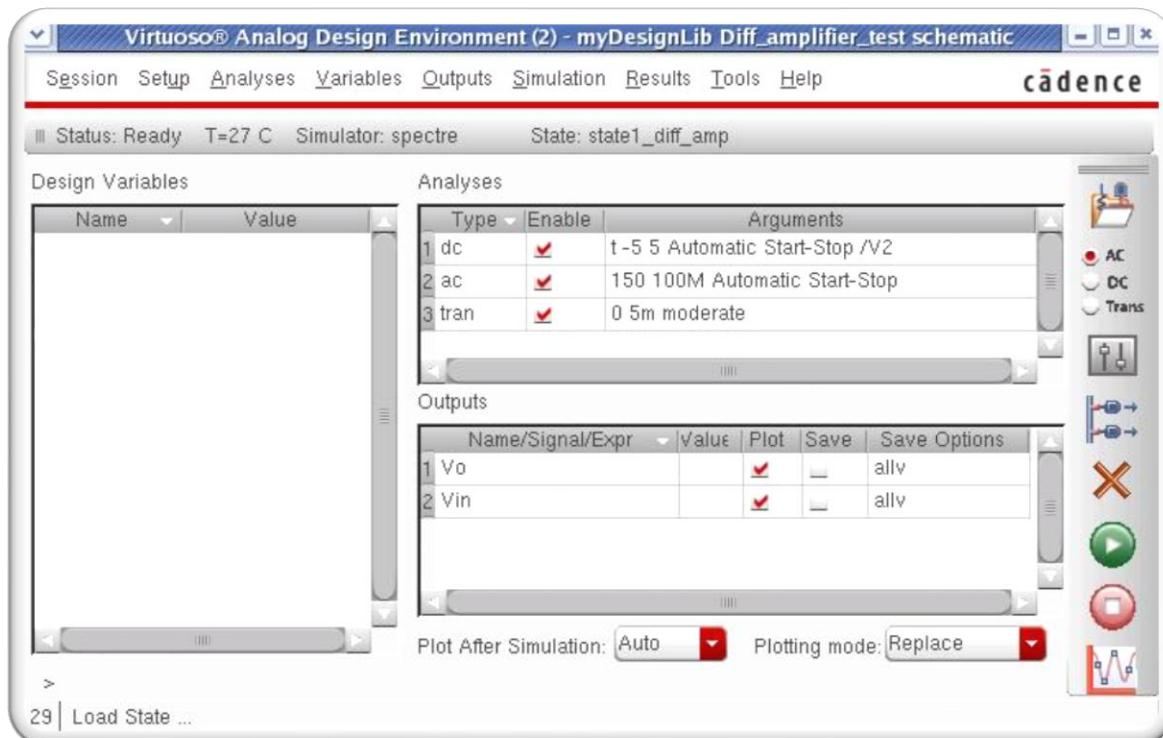
- In the Analyses section, select **ac**.
- In the AC Analyses section, turn on **Frequency**.
- In the Sweep Range section select **start** and **stop** frequencies as **150** to **100M**
- Select Points per Decade as **20**.
- Check the enable button and then click **Apply**.

5. Click **OK** in the Choosing Analyses Form.

Selecting Outputs for Plotting

Select the nodes to plot when simulation is finished.

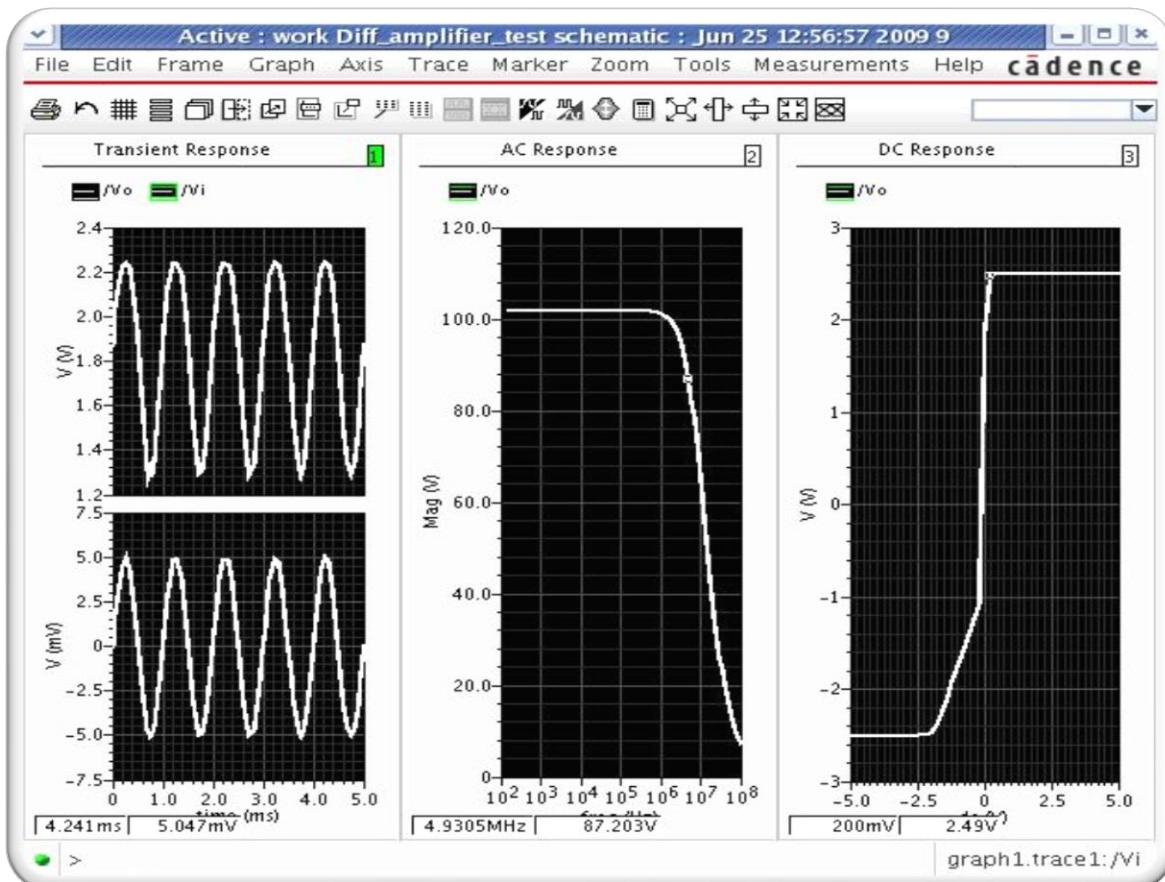
1. Execute **Outputs – To be plotted – Select on Schematic** in the simulation window.
2. Follow the prompt at the bottom of the schematic window, Click on output net **V_o**, input net **V_{in}** of the Diff_amplifier. Press **ESC** with the cursor in the schematic after selecting node.



Running the Simulation

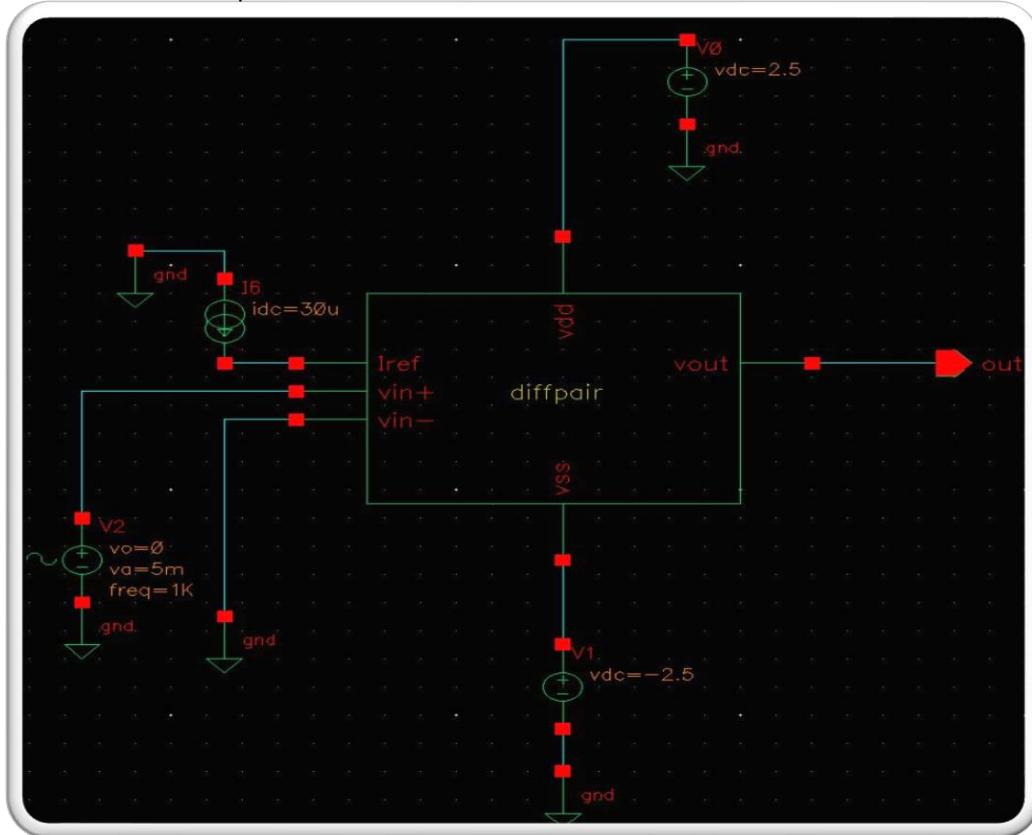
1. Execute **Simulation – Netlist and Run** in the simulation window to start the simulation, this will create the netlist as well as run the simulation.
2. When simulation finishes, the Transient, DC and AC plots automatically will be popped up along with netlist.

Does the simulation window look like this?

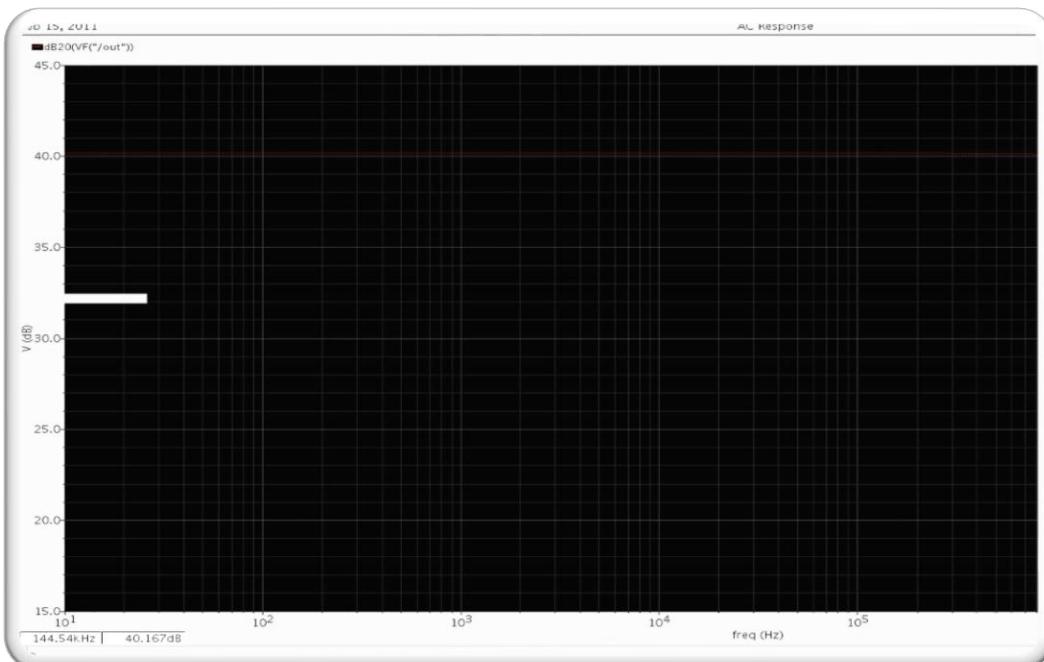


To Calculate the gain of Differential pair:

Configure the Differential pair schematic as shown below –

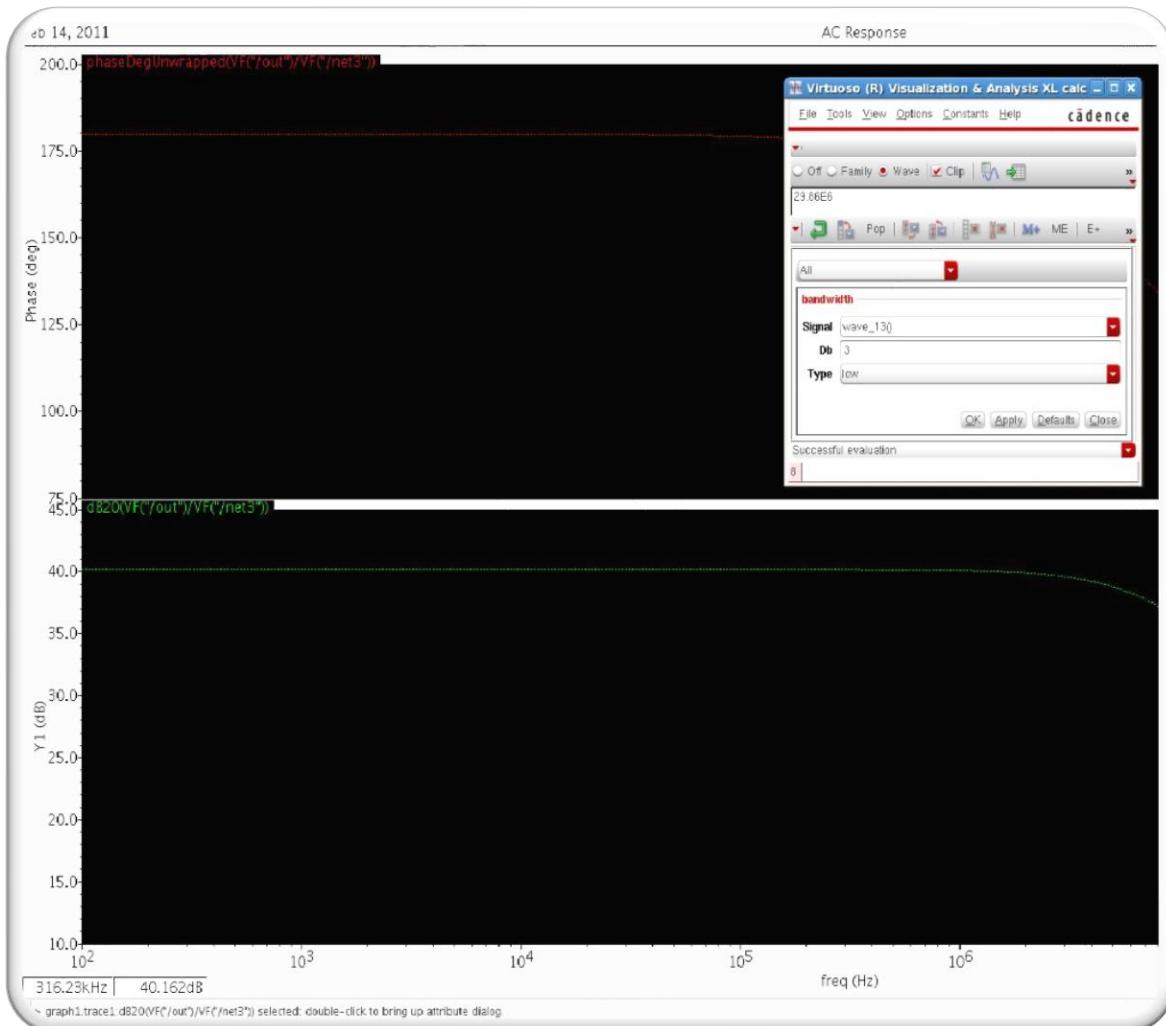


Now, open the ADE L, from **LAUNCH ADE L**, choose the analysis set the ac response and run the simulation, from **Simulation Run**. Next go to **Results Direct plot** select AC dB20 and output from the schematic and press escape. The following waveform appears as shown below –



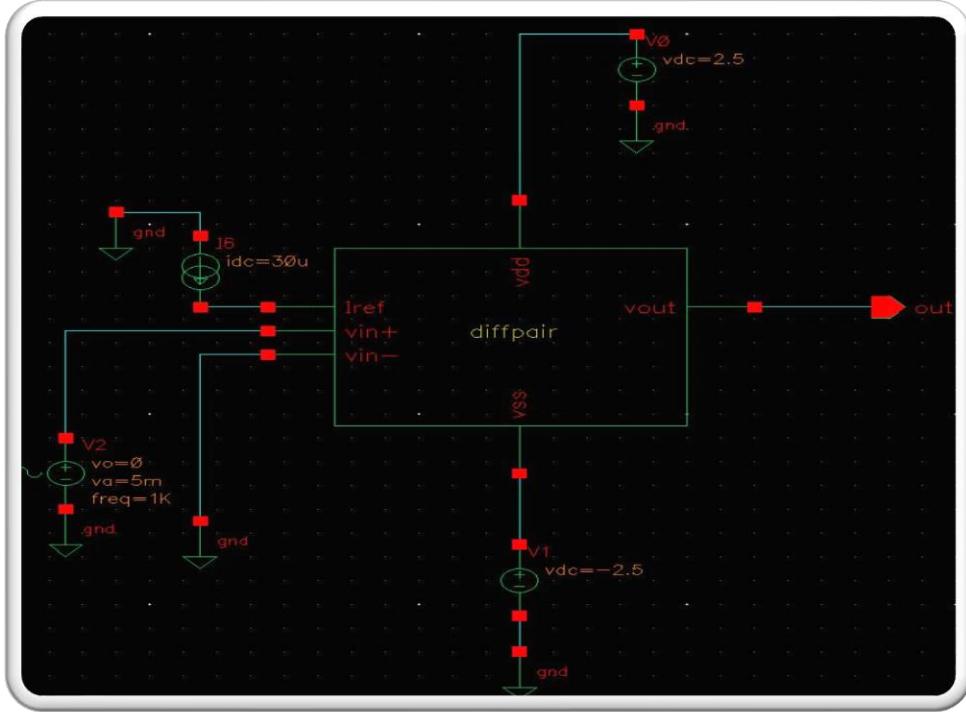
To Calculate the BW of the Differential pair:

Open the calculator and select the bandwidth option, select the waveform of the gain in dB and press Evaluate the buffer –

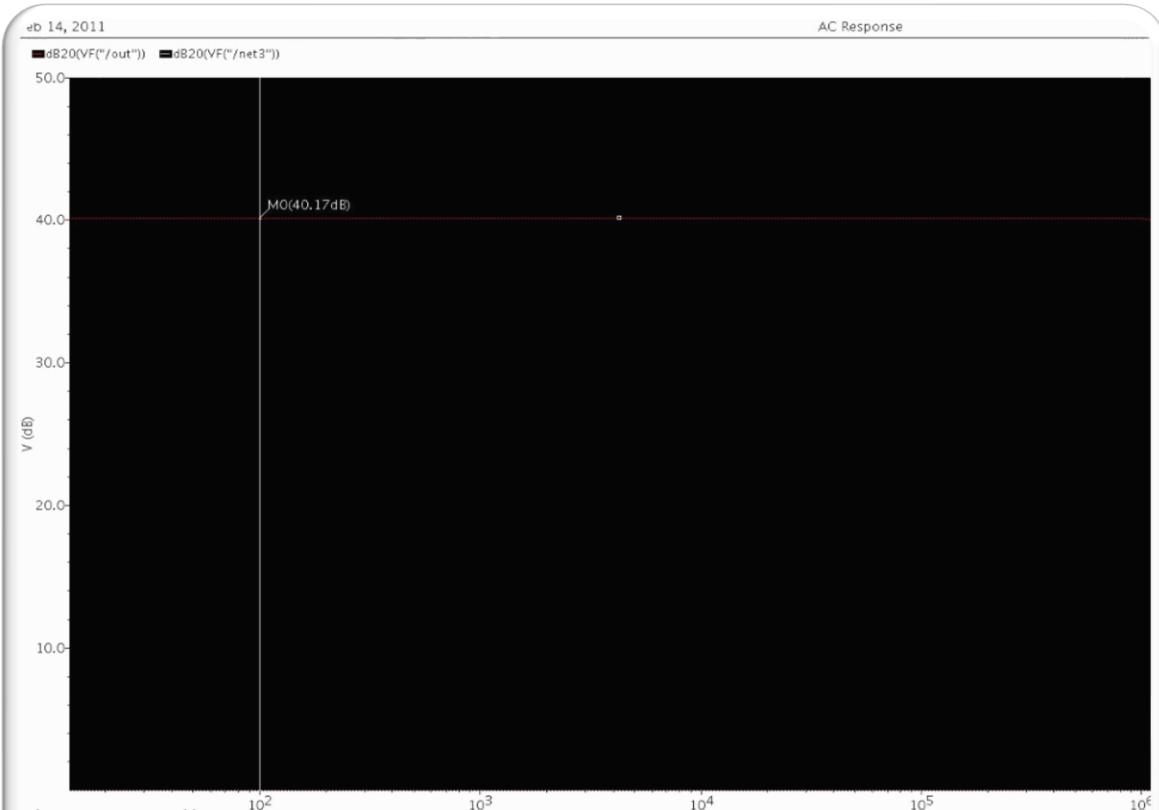


To Calculate the CMRR of the Differential pair:

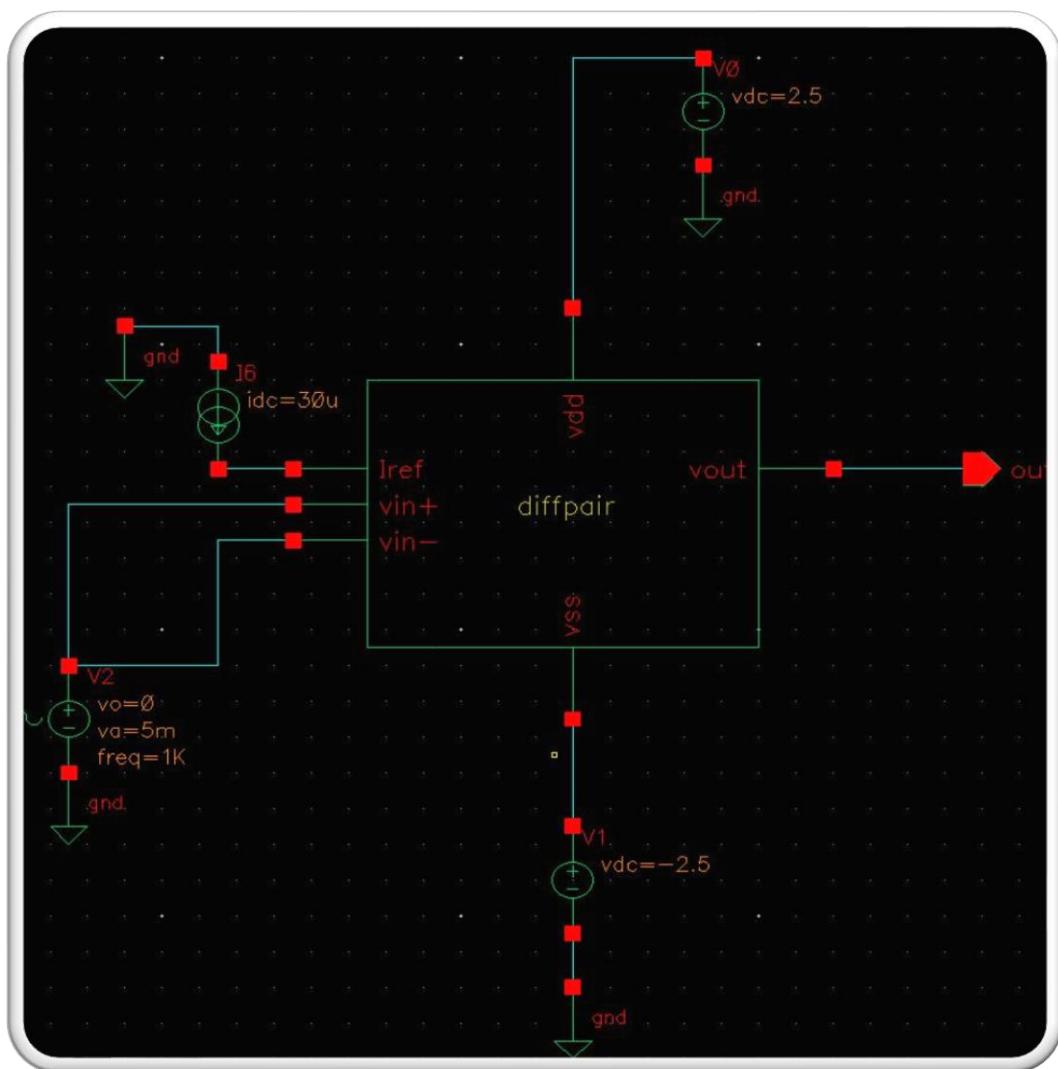
Configure the Differential pair schematic to calculate the differential gain as shown below –



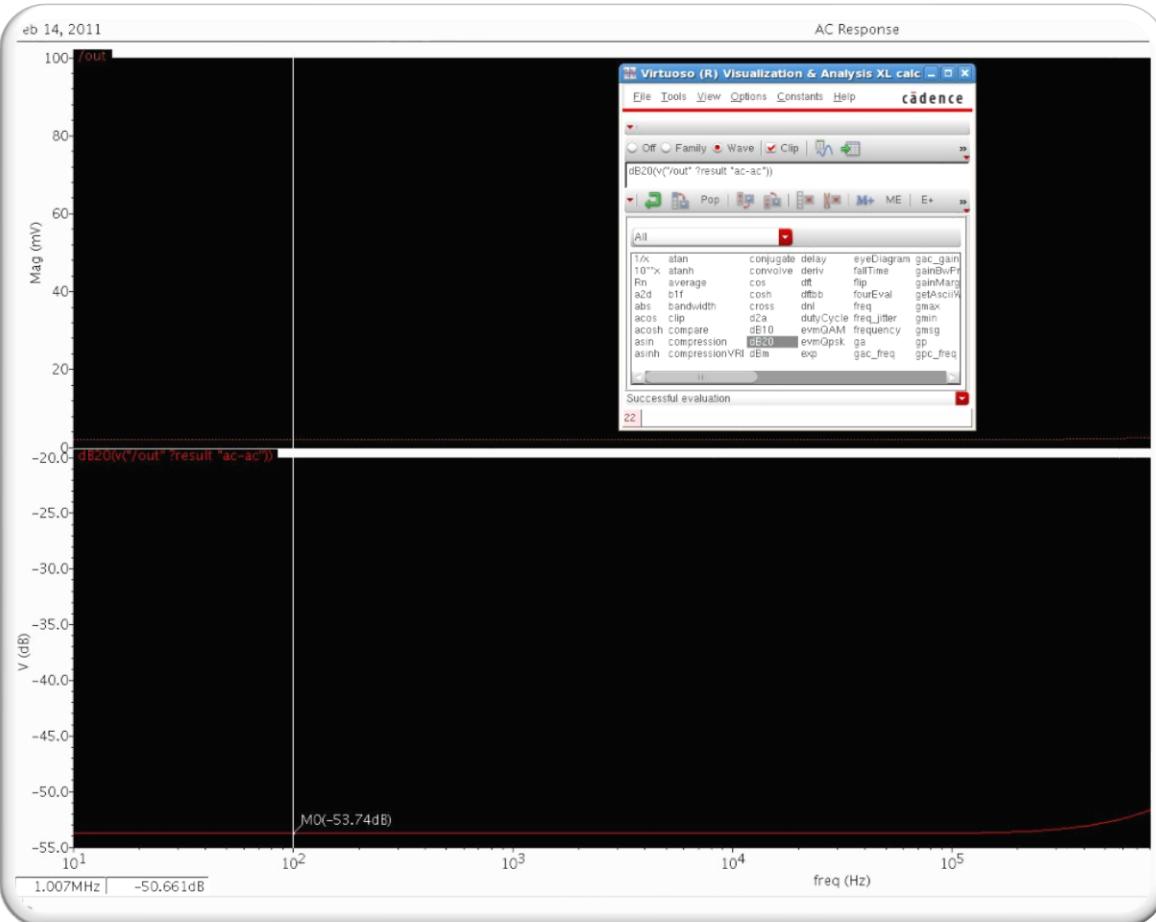
In the ADE L, plot the ac response with gain in dB. Measure the gain at 100hz and at 100Mhz, note down the value of the gain in dB, as shown below –



Configure the Differential pair schematic to calculate the common-mode gain as shown below –



In the ADE L, plot the ac response with gain in dB. Measure the gain at 100hz and at 100Mhz, note down the value of the gain in dB, as shown below –



Calculate the CMRR = $\frac{A_d}{A_c}$, add the gains in dB i.e., $A_d - (-A_c)$. For the output impedance note down the output resistance of the pmos and nmos transistors at the output side, and use the necessary equation like $r_{O1} \parallel r_{O2}$.

Saving the Simulator State

We can save the simulator state, which stores information such as model library file, outputs, analysis, variable etc. This information restores the simulation environment without having to type in all of setting again.

1. In the Simulation window, execute **Session – Save State**.
The Saving State form appears.
2. Set the **Save as** field to **state1_diff_amp** and make sure all options are selected under what to save field. Click **OK** in the saving state form. The Simulator state is saved.

Creating a Layout View of Diff_ Amplifier

1. From the Diff_amplifier schematic window menu execute **Launch – Layout XL**. A **Startup Option** form appears.
2. Select **Create New** option. This gives a **New Cell View Form**
3. Check the Cellname (**Diff_amplifier**), Viewname (**layout**).
4. Click **OK** from the New Cellview form.
LSW and a blank layout window appear along with schematic window.

Adding Components to Layout

1. Execute **Connectivity – Generate – All from Source** or click the icon in the layout editor window, **Generate Layout** form appears. Click **OK** which imports the schematic components in to the Layout window automatically.
2. Re arrange the components with in PR-Boundary as shown in the next page.
3. To rotate a component, Select the component and execute **Edit –Properties**. Now select the degree of rotation from the property edit form.



4. To Move a component, Select the component and execute **Edit -Move** command.

Making interconnection

1. Execute **Connectivity –Nets – Show/Hide selected Incomplete Nets** or click the icon in the Layout Menu.
2. Move the mouse pointer over the device and click **LMB** to get the connectivity information, which shows the guide lines (or flight lines) for the inter connections of the components.
3. From the layout window execute **Create – Shape – Path** or **Create – Shape – Rectangle** (for vdd and gnd bar) and select the appropriate Layers from the **LSW** window and Vias for making the inter connections

**Creating Contacts/Vias**

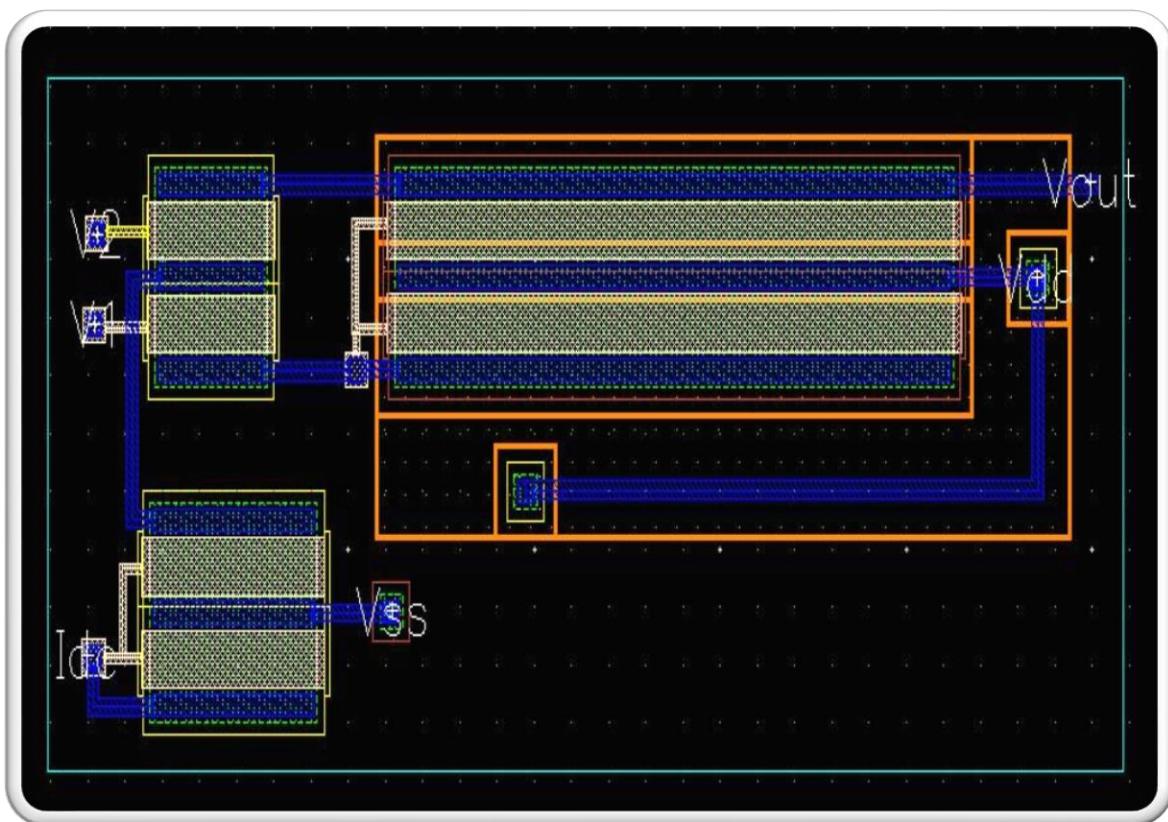
You will use the contacts or vias to make connections between two different layers.

1. Execute **Create — Via** or select command to place different Contacts, as given in below table.

Connection	Contact Type
For Metal1- Poly Connection	Metal1-Poly
For Metal1- Psubstrate Connection	Metal1-Psub
For Metal1- Nwell Connection	Metal1-Nwell

**Saving the design**

1. Save your design by selecting **File — Save** or click to save the layout and layout should appear as below.



Physical Verification

Assura DRC

Running a DRC

1. Open the Differential_Amplifier layout form the CIW or library manger if you have closed that. Press **shift – f** in the layout window to display all the levels.
2. Select **Assura - Run DRC** from layout window.
The DRC form appears. The Library and Cellname are taken from the current design window, but rule file may be missing. Select the Technology as **gpdk180**. This automatically loads the rule file.
3. Click **OK** to start DRC.
4. A Progress form will appears. You can click on the watch log file to see the log file. 5. When DRC finishes, a dialog box appears asking you if you want to view your DRC results, and then click **Yes** to view the results of this run.



6. If there any DRC error exists in the design **View Layer Window** (VLW) and **Error Layer Window** (ELW) appears. Also the errors highlight in the design itself.
7. Click **View – Summary** in the ELW to find the details of errors.
8. You can refer to rule file also for more information, correct all the DRC errors and **Re – run** the DRC.
9. If there are no errors in the layout then a dialog box appears with **No DRC errors found** written in it, click on **close** to terminate the DRC run.



ASSURA LVS

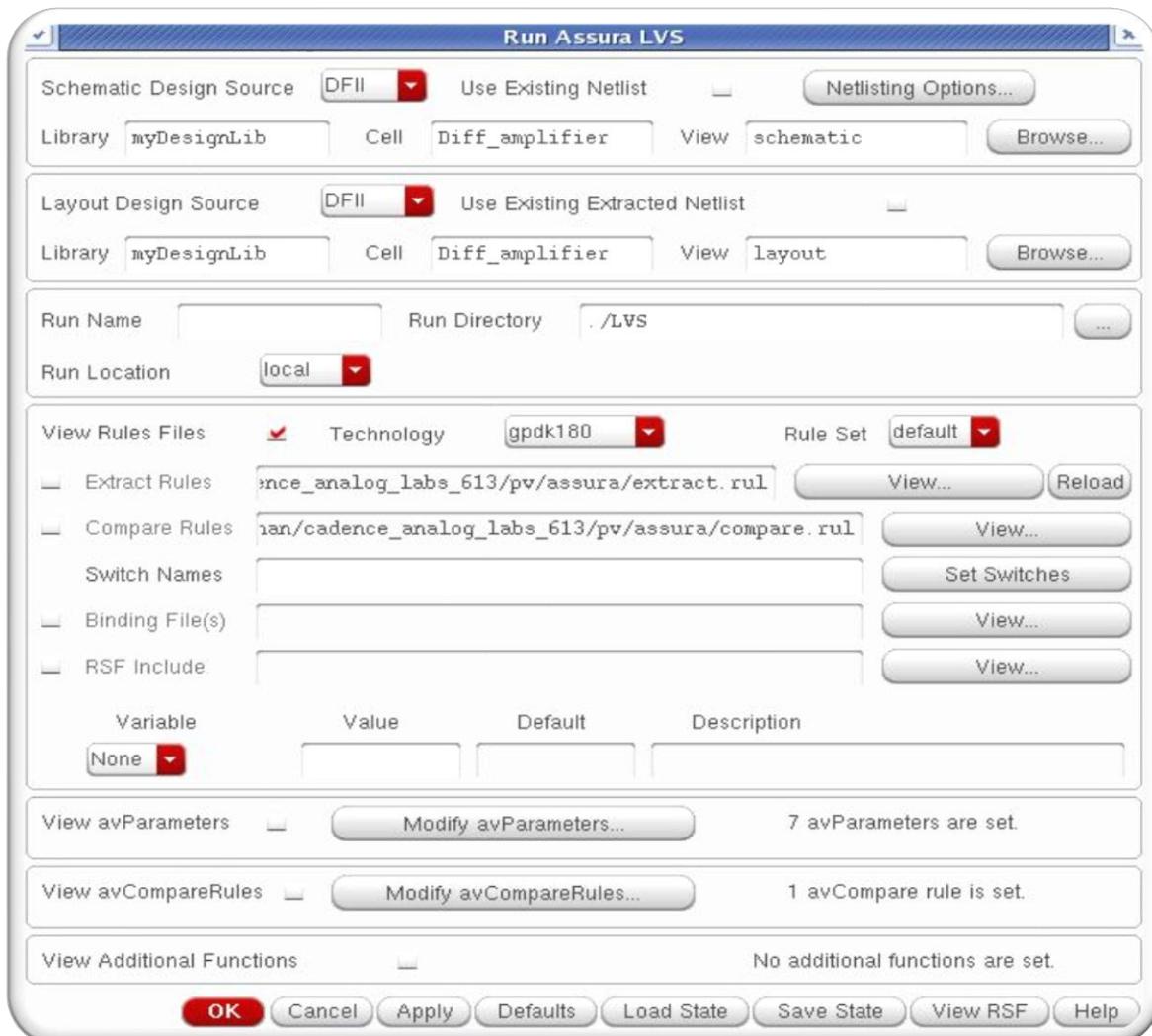
In this section we will perform the LVS check that will compare the schematic netlist and the layout netlist.

Running LVS

1. Select **Assura – Run LVS** from the layout window.

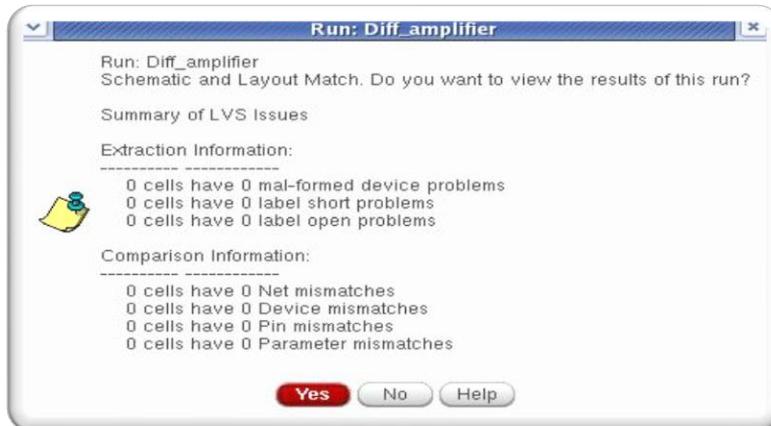
The Assura Run LVS form appears. The layout name is already in the form.
Assura fills in the layout name from the cellview in the layout window.

2. Verify the following in the Run Assura LVS form.

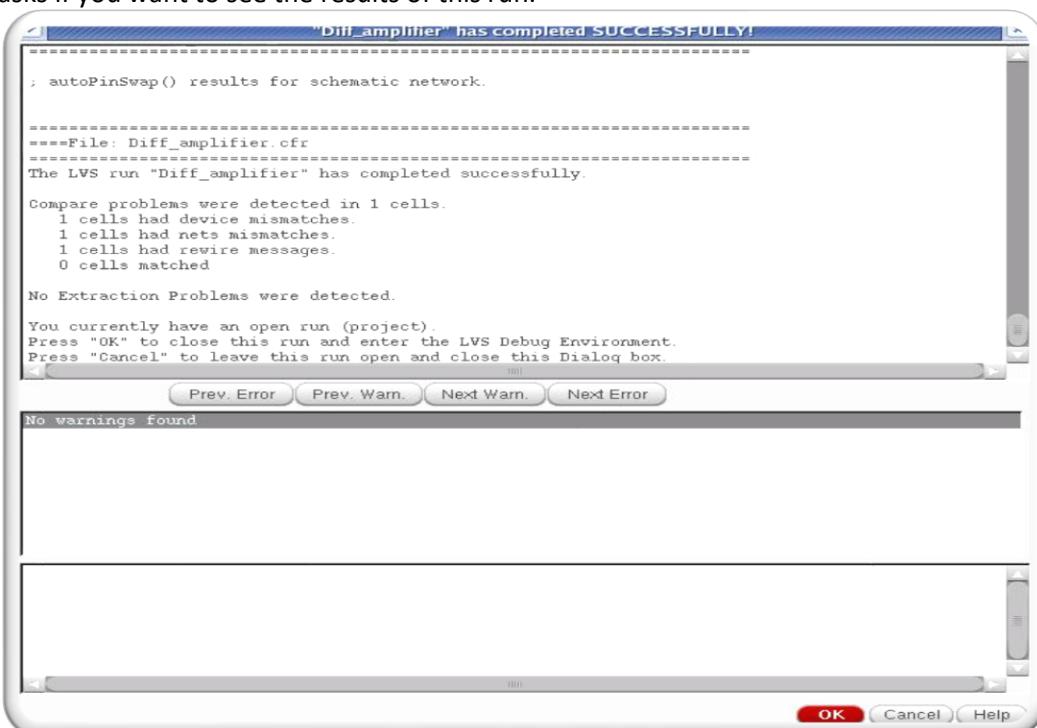


3. The LVS begins and a Progress form appears.

4. If the schematic and layout matches completely, you will get the form displaying **Schematic and Layout Match**.



5. If the schematic and layout do not matches, a form informs that the LVS completed successfully and asks if you want to see the results of this run.



6. Click **Yes** in the form.LVS debug form appears, and you are directed into LVS debug environment.
7. In the **LVS debug form** you can find the details of mismatches and you need to correct all those mismatches and **Re – run** the LVS till you will be able to match the schematic with layout.

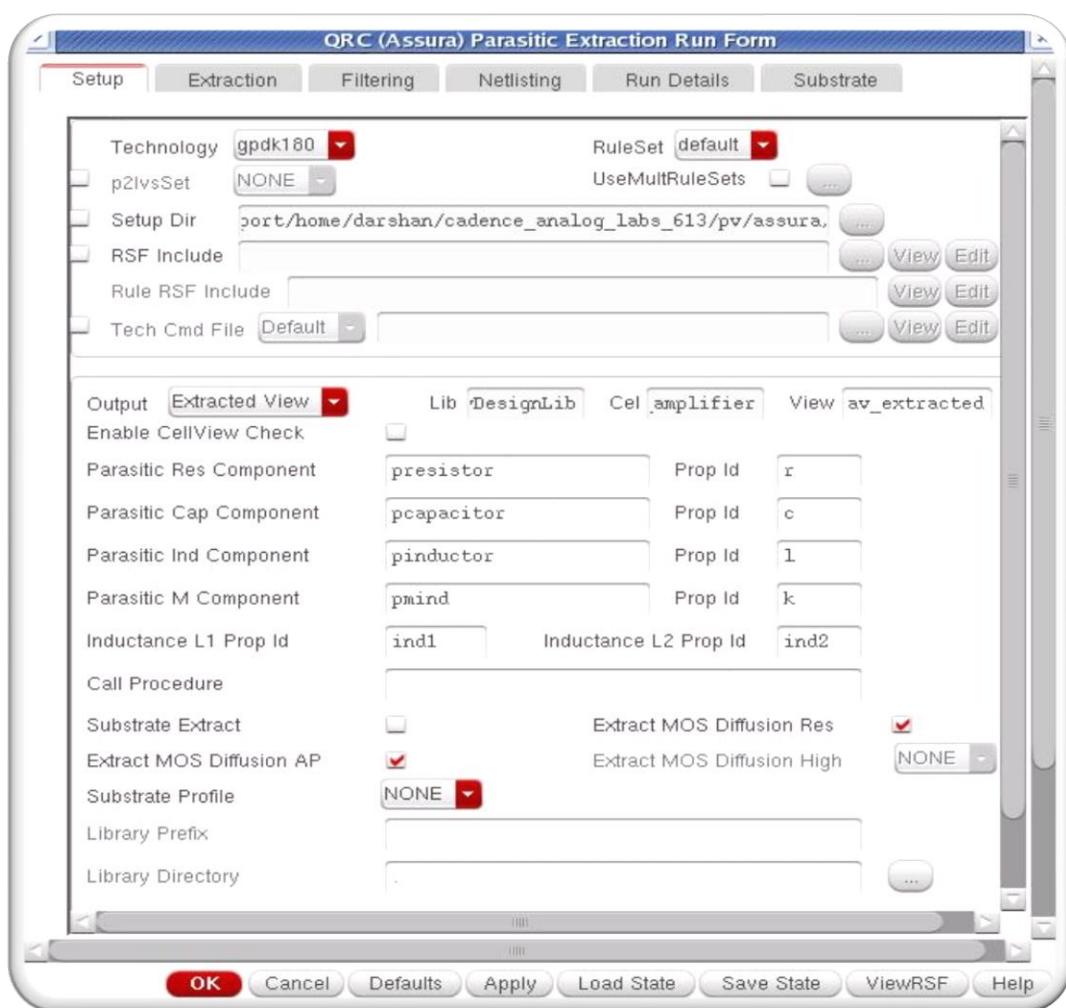
Assura RCX

In this section we will extract the RC values from the layout and perform analog circuit simulation on the designs extracted with RCX.

Before using RCX to extract parasitic devices for simulation, the layout should match with schematic completely to ensure that all parasites will be backannotated to the correct schematic nets.

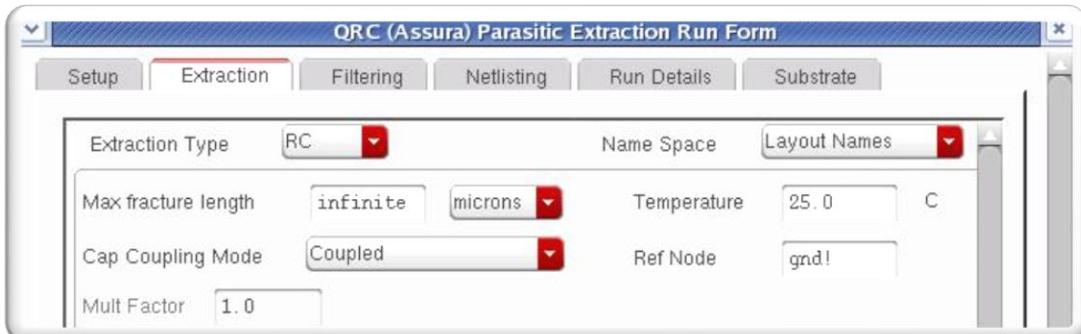
Running RCX

1. From the layout window execute **Assura – Run RCX**.
2. Change the following in the Assura parasitic extraction form. Select **output** type under **Setup** tab of the form.

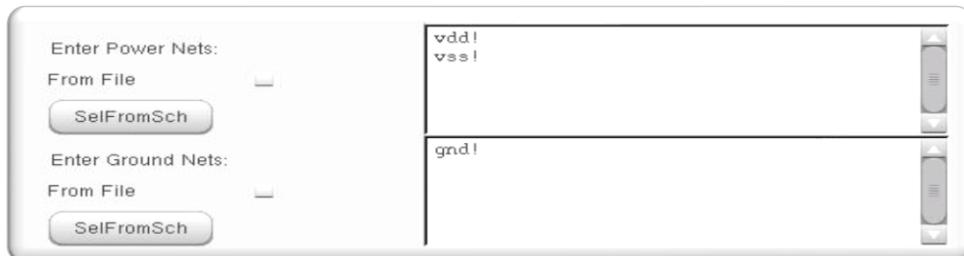


3. In the **Extraction** tab of the form, choose Extraction type, Cap Coupling Mode and

specify the Reference node for extraction.



4. In the **Filtering** tab of the form, Enter Power Nets as **vdd!**, **vss!** and Enter Ground Nets as **gnd!**



5. Click **OK** in the Assura parasitic extraction form when done.

The RCX progress form appears, in the progress form click **Watch log file** to see the output log file.

6. When RCX completes, a dialog box appears, informs you that **Assura RCX run completed successfully**.



7. You can open the **av_extracted** view from the library manager and view the parasitic.



REVA UNIVERSITY

Bengaluru, India

Rukmini Knowledge Park, Kattigenahalli
Yelahanka, Bengaluru - 560 064
Karnataka, India.

Ph: +91- 90211 90211, +91 80 4696 6966
E-mail: admissions@reva.edu.in

www.reva.edu.in

Follow us on

 /revauniversity_official

 /REVAUniversity

 /revauniversity_official

 /@revauniversity_official

 /REVA University

 reva.edu.in

 +91 90211 90211

