



Varuwan Vadivelan Institute of Technology

Dharmapuri – 636 703

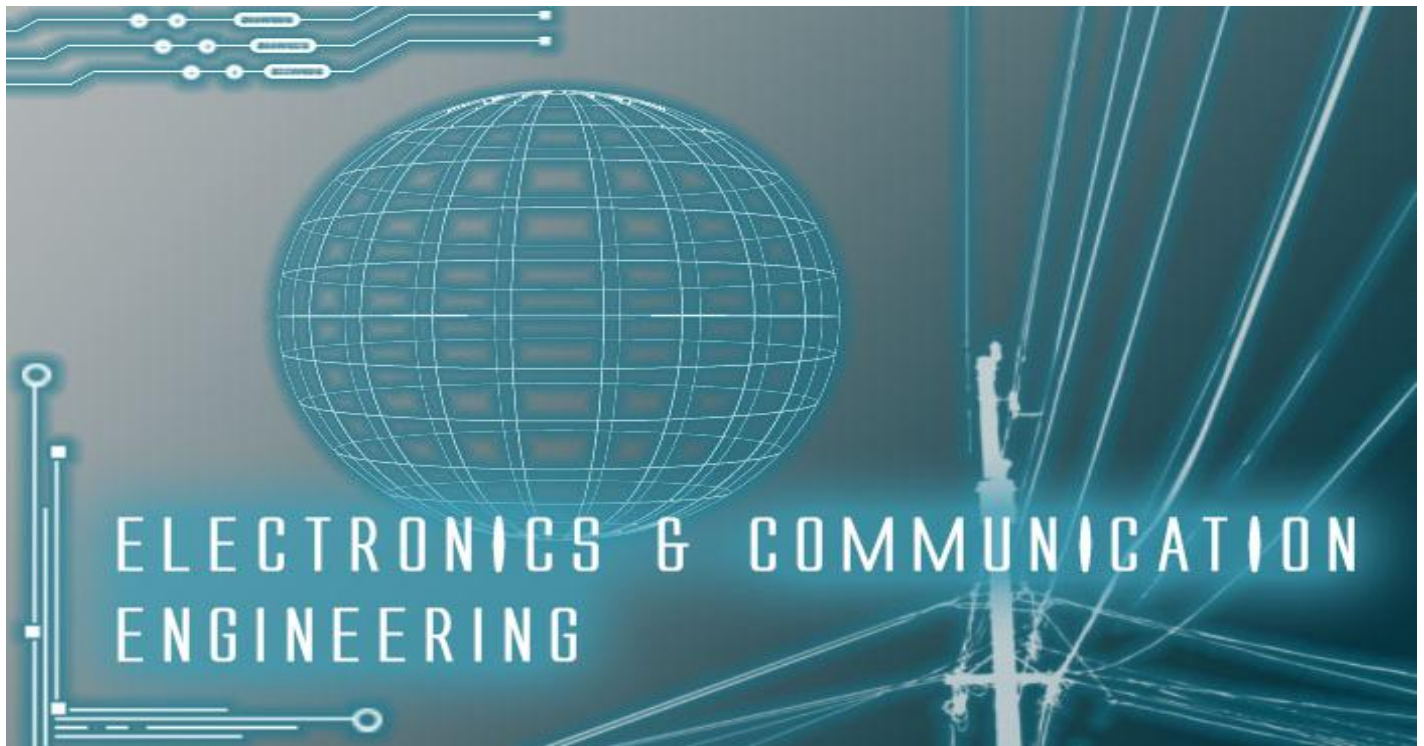
LAB MANUAL

Regulation : 2013

Branch : *B.E.* - ECE

Year & Semester : III Year / VI Semester

EC6612- VLSI DESIGN LABORATORY



ANNA UNIVERSITY CHENNAI**Regulation 2013****EC6612 VLSI DESIGN LAB****FPGA BASED EXPERIMENTS**

1. HDL based design entry and simulation of simple counters, state machines, adders (min 8 bit) and multipliers (4 bit min).
2. Synthesis, P&R and post P&R simulation of the components simulated in (I) above. Critical paths and static timing analysis results to be identified. Identify and verify possible conditions under which the blocks will fail to work correctly.
3. Hardware fusing and testing of each of the blocks simulated in (I). Use of either Chip scope feature (Xilinx) or the signal tap feature (Altera) is a must. Invoke the PLL and demonstrate the use of the PLL module for clock generation in FPGAs.

IC DESIGN EXPERIMENTS: (BASED ON CADENCE / MENTOR GRAPHICS / EQUIVALENT)

4. Design and simulation of a simple 5 transistor differential amplifier. Measure gain, ICMR, and CMRR
5. Layout generation, parasitic extraction and resimulation of the circuit designed in (1)
6. Synthesis and Standard cell based design of an circuits simulated in 1(I) above. Identification of critical paths, power consumption.
7. For expt (c) above, P&R, power and clock routing, and post P&R simulation.
8. Analysis of results of static timing analysis.

TOTAL: 45 PERIODS

INDEX

EXP No.	DATE	LIST OF EXPERIMENT	SIGNATURE OF THE STAFF	REMARKS
1		Simulation of Basic Logic Gates		
2		Simulation of Half adder and Full adder		
3		Simulation of Adder		
4		Simulation of multiplier		
5		Simulation of Binary to Gray code counter		
6		Simulation of Flip Flops		
7		Simulation of Pseudo Random Binary Sequence		
8		Simulation of up-down counter		
9		Simulation of Counter		
10		Design of Sequence Detector		
11		Differential Amplifier		
12		CMOS Invertor		
13		Layout CMOS invertor		
14		Automatic layout generation		

INTRODUCTION

Very-large-scale integration (VLSI) is the process of creating an integrated circuit (IC) by combining thousands of transistors into a single chip. VLSI began in the 1970s when complex semiconductor and communication technologies were being developed.

The microprocessor is a VLSI device. Before the introduction of VLSI technology most ICs had a limited set of functions they could perform. An electronic circuit might consist of a CPU, ROM, RAM and other glue logic. VLSI lets IC designers add all of these into one chip.

During the mid-1920s, several inventors attempted devices that were intended to control current in solid-state diodes and convert them into triodes. Success did not come until after World War II, during which the attempt to improve silicon and germanium crystals for use as radar detectors led to improvements in fabrication and in the understanding of quantum mechanical states of carriers in semiconductors.

Then scientists who had been diverted to radar development returned to solid-state device development. With the invention of transistors at Bell Labs in 1947, the field of electronics shifted from vacuum tubes to solid-state devices. With the small transistor at their hands, electrical engineers of the 1950s saw the possibilities of constructing far more advanced circuits. As the complexity of circuits grew, problems arose.

One problem was the size of the circuit. A complex circuit, like a computer, was dependent on speed. If the components of the computer were too large or the wires interconnecting them too long, the electric signals couldn't travel fast enough through the circuit, thus making the computer too slow to be effective.

Jack Kilby at Texas Instruments found a solution to this problem in 1958. Kilby's idea was to make all the components and the chip out of the same block (monolith) of semiconductor material. Kilby presented his idea to his superiors, and was allowed to build a test version of his circuit.

In September 1958, he had his first integrated circuit ready. Although the first integrated circuit was crude and had some problems, the idea was groundbreaking. By making all the parts out of the same block of material and adding the metal needed to connect them as a layer on top of it, there was no need for discrete components. No more wires and components had to be assembled manually.

The circuits could be made smaller, and the manufacturing process could be automated. From here, the idea of integrating all components on a single silicon wafer came into existence, which led to development in small-scale integration (SSI) in the early 1960s, medium-scale integration (MSI) in the late 1960s, and then large-scale integration (LSI) as well as VLSI in the 1970s and 1980s, with tens of thousands of transistors on a single chip (later hundreds of thousands, then millions, and now billions (10^9)).

The first semiconductor chips held two transistors each. Subsequent advances added more transistors, and as a consequence, more individual functions or systems were integrated over time. The first integrated circuits held only a few devices, perhaps as many as ten diodes, transistors, resistors and capacitors, making it possible to fabricate one or more logic gates on a single device. Now known retrospectively as small-scale integration (SSI), improvements in technique led to devices with hundreds of logic gates, known as medium-scale integration (MSI). Further improvements led to large-scale integration (LSI), i.e. systems with at least a thousand logic gates. Current technology has moved far past this mark and today's microprocessors have many millions of gates and billions of individual transistors.

At one time, there was an effort to name and calibrate various levels of large-scale integration above VLSI. Terms like ultra-large-scale integration (ULSI) were used. But the huge number of gates and transistors available on common devices has rendered such fine distinctions moot. Terms suggesting greater than VLSI levels of integration are no longer in widespread use.

STEP BY STEP PROCESS FOR USING SIMULATION TOOLS
--

AIM:

To study simulation tools using Xilinx software tool.

SOFTWARE REQUIRED:

1. Xilinx ISE Design Suite 14.1

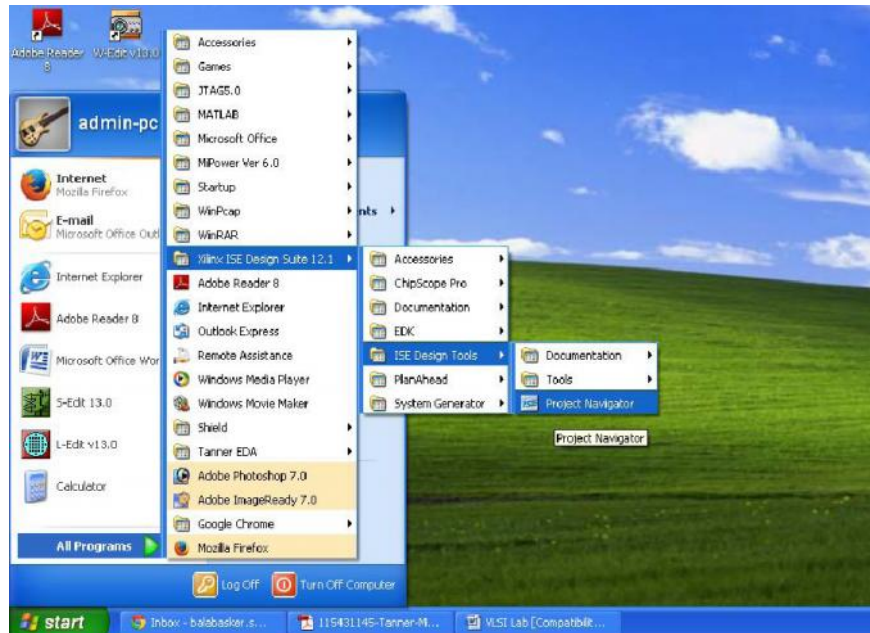
PROCEDURE:

1. Now start the Xilinx ISE Design Suite 14.1
2. Go to file and click new project
3. Enter the project name and click next
4. Select the family name is Spartan 3E, speed is -4 and simulator is verilog click next and click Finish.
5. Click new source.
6. Select verilog module and type file name and click next.
7. Assign input and output port and click next.
8. Finally the report is shown click finish.
9. Type the program save and click synthesis.
10. To see the output wave form change the source from implementation to simulation and click
11. simulator behavior model in ISim simulator.
12. Give values to the input variables and then click run
13. In wave window, click run icon and you can see corresponding output.

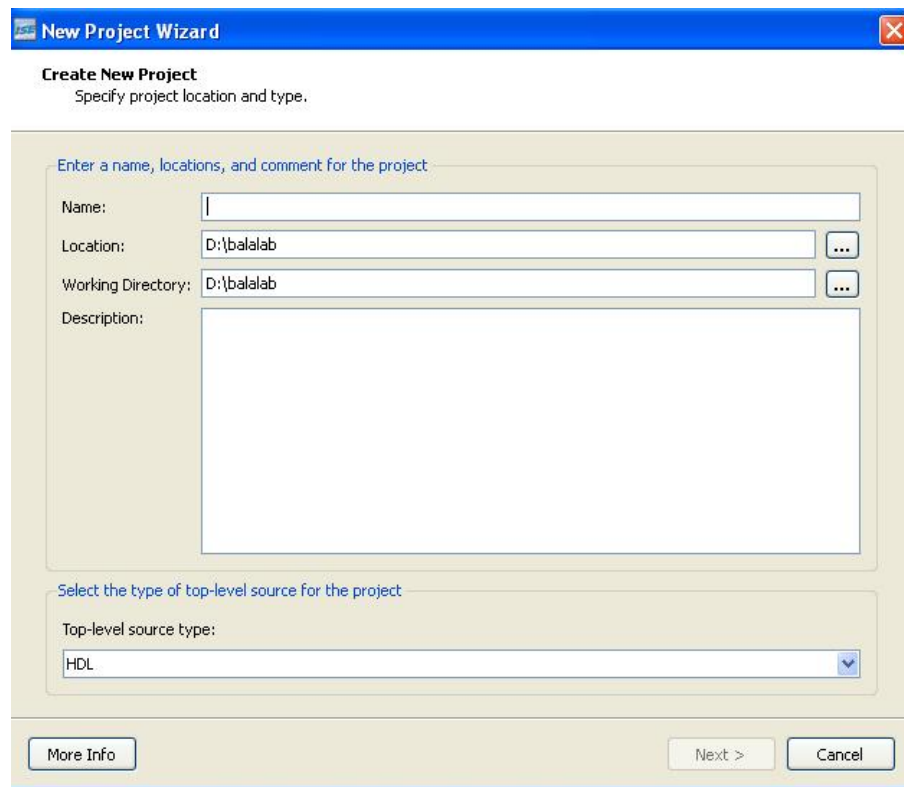
Steps to use Xilinx tool:

Start the Xilinx Project Navigator by using the desktop shortcut or by using the

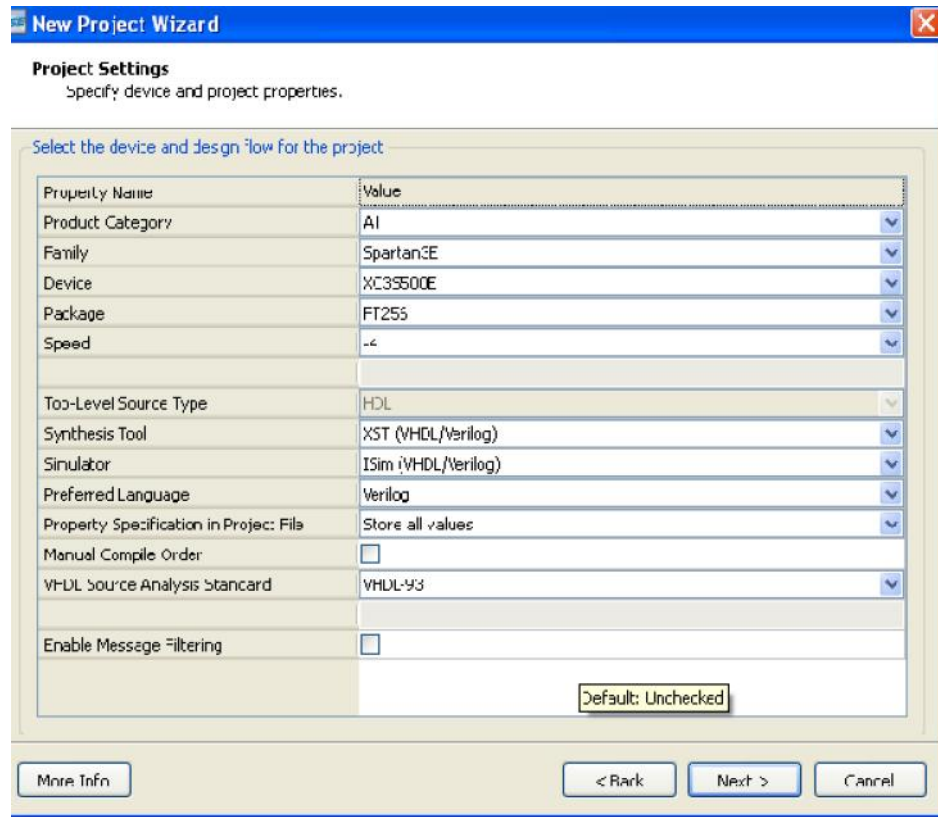
- Start → Programs → Xilinx ISE → Project Navigator.



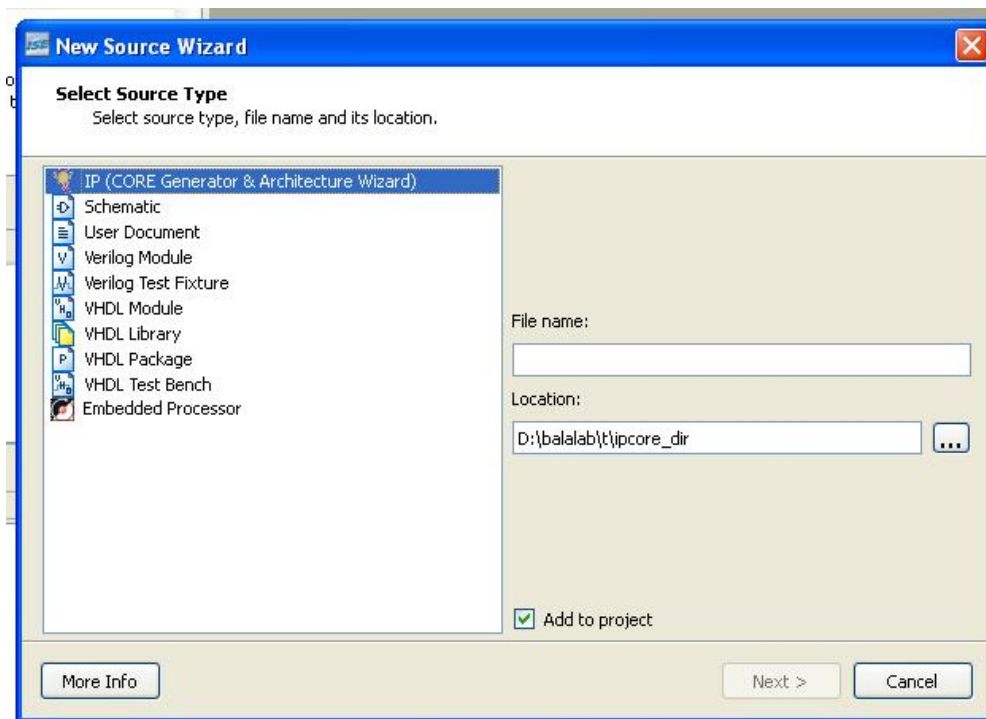
➤ In the Project Navigator window go to FILE →New project→ Click on new source→ verilog module and give the name inverter.v →Define ports→Finish



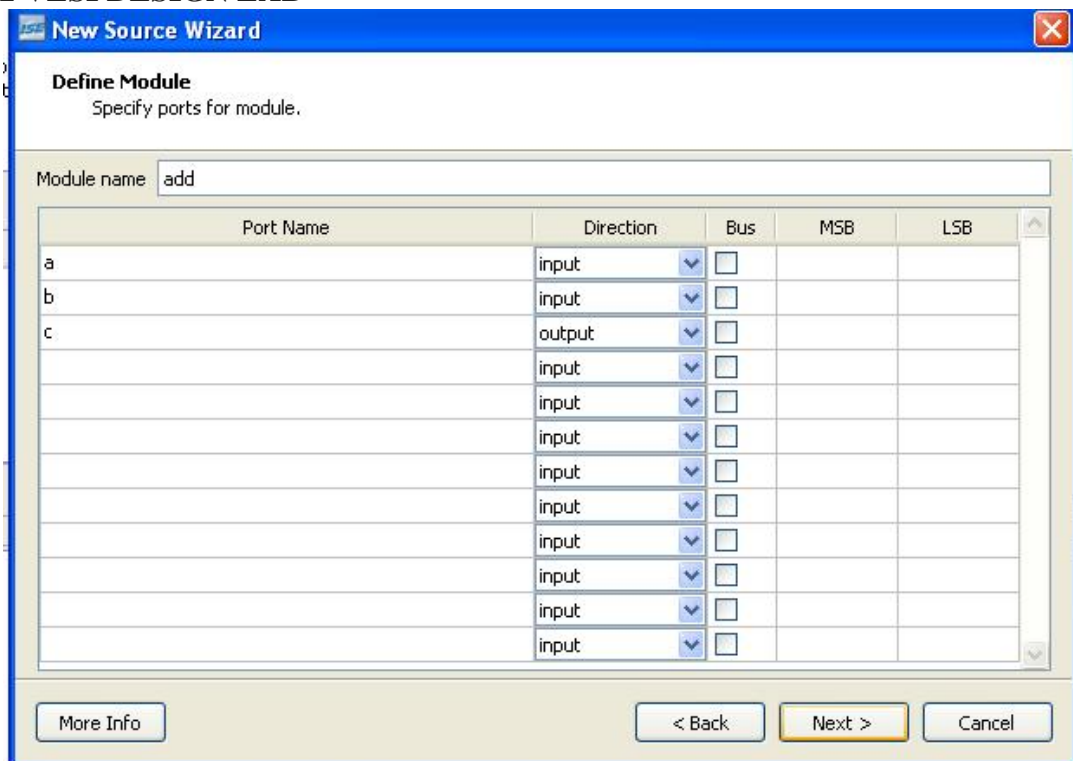
➤ Select devices→General purpose→ Spartan 3→ ISE simulator→verilog



➤ In the create new source window select source type as verilog module give file name



➤ assign inputs and outputs → click next → finish → yes → next → next → finish



- Double click on source file → complete the verilog code for inverter
- Check syntax, and remove errors if present

- Simulate the design using ISE Simulator Highlight **inverter.v** file in the Sources in Project window. To run the Behavioral Simulation, Click on the symbol of FPGA device and then right click → Click on new source → Click on verilog text fixture → Give file name with `_tb` → finish
- Generate test bench file after initial begin assign value for inputs → Click on simulate behavioral model → see the output.

STEP BY STEP PROCESS FOR USING SYNTHESIZE TOOLS

SOFTWARE- Xilinx ISE Design Suite 14.1

Synthesis is an automatic method of converting a higher level abstraction to a lower level abstraction. The synthesis tool convert **Register Transfer Level (RTL)** description to **gate level netlists**.

These gate level netlists consist of interconnected gate level macro cells. These gate level netlists currently can be optimized for **area, speed** etc., The analyzed design is synthesized to a library of components, typically gates, latches, or flipflops. Hierarchical designs are synthesized in bottom up fashion, that is lower level components are synthesized before higher level components. Once the design is synthesized we have a gate level netlist. This gate level netlist can be simulated. Delay for the individual components are available as part of the description of the component libraries. Timing accurate simulation is not possible at this point because the actual timing characteristics are determined by the physical placement of the design within the FPGA chip. However, the functional simulation that is possible at this point is quite a bit more accurate than simulation based on user specified delays. After run the synthesizer in process window then full adder model is converted to **netlist file**.

To convert the RTL to gates, three steps typically occur:

* The RTL description is translated to an unoptimized boolean description usually consisting of primitive gates such as AND and OR gates, flip-flop, and latches. This is a functionally correct but completely unoptimized description.

* Boolean optimization algorithms are executed on this boolean equivalent description to produce an optimized boolean equivalent description.

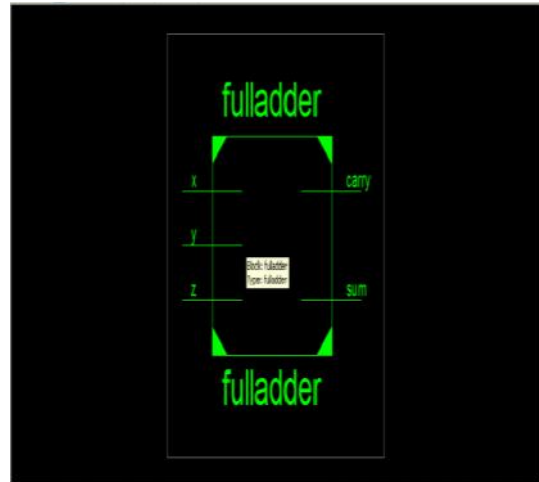
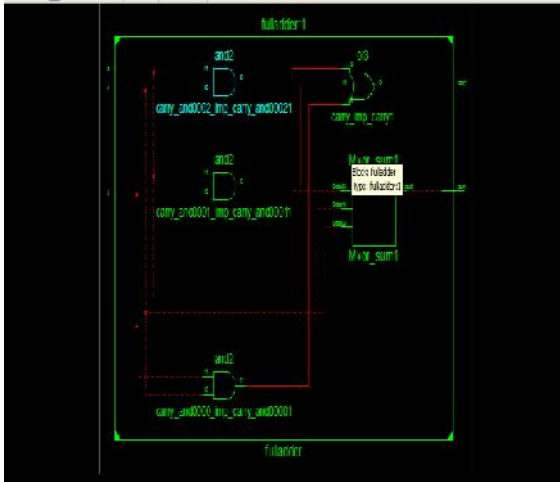
* This optimized boolean equivalent description is mapped to actual logic gate by making use of a technology library of the target process.

PROCEDURE:

1. Now start the Xilinx ISE Design Suite 14.1
2. Go to file and click new project
3. Enter the project name and click next
4. Select the family name is Spartan 3E, speed is -4 and simulator is verilog click next and click Finish.
5. Click new source.
6. Select verilog module and type file name and click next.
7. Assign input and output port and click next.
8. Finally the report is shown click finish.

9. Type the program save and click synthesis.

10. Go to synthesis → View RTL schematic



STEP BY STEP PROCESS FOR USING PLACE AND ROUTE AND BACK ANNOTATION FOR FPGAS

THEORY:

To map this Full adder design onto the FPGA. The primitive hardware elements that are available in Xilinx xc3s500e chip, namely lookup tables and positive-edge-triggered flip-flops are organized as a **two dimensional array of CLBs**. The netlist from synthesizer is composed of **gates, latches, and flipflops**. It is necessary to assign CLB to netlist primitives. This is the process of **mapping** a design. For example gates will be assigned to look-up tables. This process effectively translates the gate level netlist produced by the synthesizer compiler into a netlist of FPGA primitive hardware components. Each element of this new netlist corresponds to a hardware primitive in the FPGA Chip. The mapped design produces and identifies the set of FPGA hardware primitives and their interconnection. The next step is to assign each of the components in the netlist to an equivalent physical primitive on the FPGA chip. Once this assignment or placement is made the interconnection between the components in the netlist must be made within the chip. This will require routing signals through the switch matrix and other interconnect resources available on the FPGA Chip. This Place and route layout was generated from Xilinx ISE **Floor planner**. After place and route the design can be simulated to validate the design. At this point timing is more accurate because the propagation delays along routed signals and through CLBs can be more accurately estimated. This is particularly important for designs that are operating under tight timing tolerance.

To convert the RTL to gates, three steps typically occur:

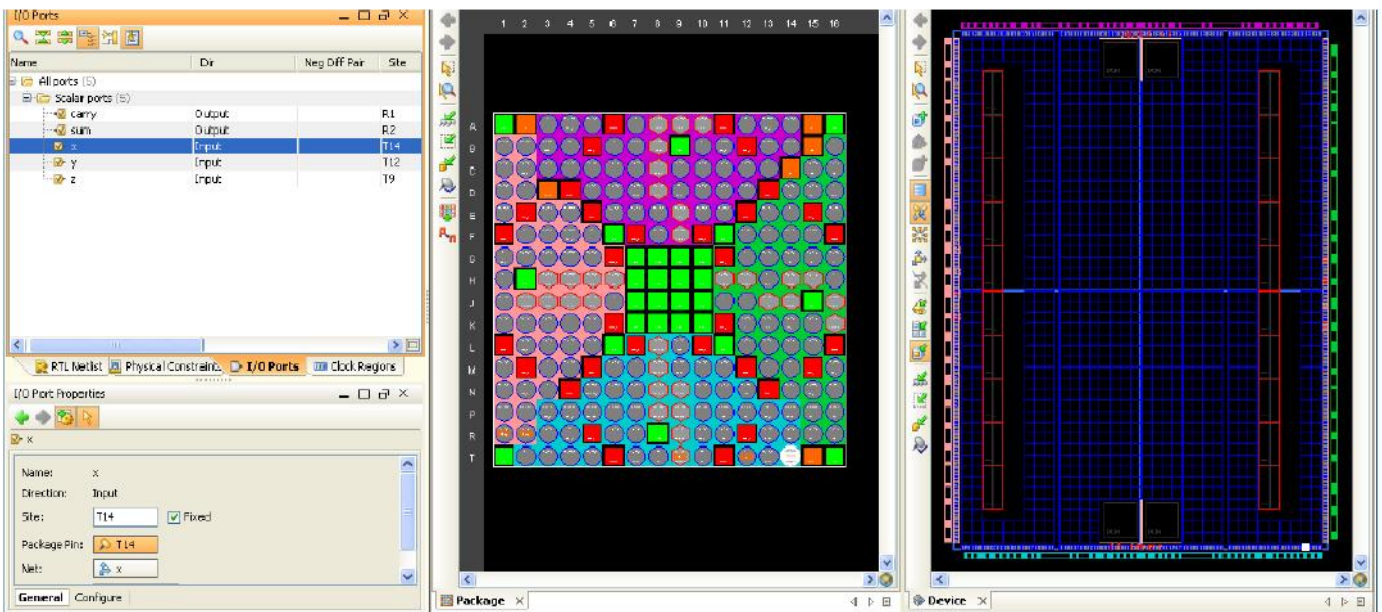
- * The RTL description is translated to an unoptimized boolean description usually consisting of primitive gates such as AND and OR gates, flip-flop, and latches. This is a functionally correct but completely unoptimized description.

- * Boolean optimization algorithms are executed on this boolean equivalent description to produce an optimized boolean equivalent description.

- * This optimized boolean equivalent description is mapped to actual logic gate by making use of a technology library of the target process.

PROCEDURE:

1. Now start the Xilinx ISE Design Suite 14.1
2. Go to file and click new project
3. Enter the project name and click next
4. Select the family name is Spartan 3E, speed is -4 and simulator is verilog click next and click Finish.
5. Click new source.
6. Select verilog module and type file name and click next.
7. Assign input and output port and click next.
8. Finally the report is shown click finish.
9. Type the program save and click synthesis.
10. Choose Implementation → user constraints → I/O pin planning (plan ahead) pre-synthesis, type the input /output port.



PROGRAM:**AND GATE:**

```
module gl(a,b,c);  
input a;  
input b;  
output c;  
and(c,a,b);  
endmodule
```

OR GATE:

```
module gl(a,b,c);  
input a;  
input b;  
output c;  
or(c,a,b);  
endmodule
```

NOT GATE:

```
module gl(a,b);  
input a;  
output b;  
not(b,a);  
endmodule
```

BUFFER GATE:

```
module gl(a,b);  
input a;  
output b;  
buf(b,a);  
endmodule
```

NOR GATE:

```
module gl(a,b,c);  
input a;  
input b;  
output c ;  
nor(c,a,b);  
endmodule
```

XOR GATE:

```
module gl(a,b,c);  
input a;  
input b;  
output c;  
xor(c,a,b);  
endmodule
```

XNOR GATE:

```
module gl(a,b,c);  
input a;  
input b;  
output c;  
xnor(c,a,b);  
endmodule
```

NANDGATE:

```
module gl(a,b,c);  
input a;  
input b;  
output c;  
nand(c,a,b);  
endmodule
```

EXP.NO: 1	SIMULATION OF BASIC LOGIC GATES
DATE:	

AIM:

To write a verilog program for basic logic gates to synthesize and simulate using Xilinx software tool.

SOFTWARE REQUIRED:

1. Xilinx ISE Design Suite 14.1
2. Spartan3E kit

THEORY:**AND GATE:**

The AND gate performs logical multiplication which is most commonly known as the AND junction. The operation of AND gate is such that the output is high only when all its inputs are high and when any one of the inputs is low the output is low.

$$Y = a \& b$$

OR GATE:

The OR gate performs logical addition which is most commonly known as the OR junction. The operation of OR gate is such that the output is high only when any one of its input is high and when both the inputs are low the output is low.

$$Y = a | b$$

NOT GATE:

The Inverter performs a basic logic gate function called Inversion or Complementation. The purpose of an inverter is to change one logic level to opposite level. When a high level is applied to an inverter, the low level will appear at the output and vice versa.

$$Y = \sim a$$

NAND GATE:

The term NAND is derived from the complement of AND. It implies the AND junction with an inverted output. The operation of NAND gate is such that the output is low only when all its inputs are high and when any one of the inputs is low the output is high.

$$Y = \sim(a \& b)$$

NOR GATE:

The term NOR is derived from the complement of OR. It implies the OR junction with an inverted output. The operation of NOR gate is such that the output is high only when all its inputs are low and when any one of the inputs is high the output is low.

$$Y = \sim(a \mid b)$$

EX-OR GATE:

The output is high only when the inputs are at opposite level.

$$Y = a \wedge b$$

EX-NOR GATE:

The output is high only when the inputs are at same level.

$$Y = \sim(a \wedge b)$$

RESULT:

Thus the basic logic gates verilog program for to synthesize and simulate using Xilinx software tool was verified.


```
endmodule
```

PROGRAM:

(SIMULATION OF HALF ADDER AND FULL ADDER)

HALF ADDER

```
module
halfadder(a,b,sum,carry);
input a,b;
output sum,carry;
xor(sum,a,b);
and(carry,a,b);
endmodule
```

FULL ADDER

```
module full_adder
(a,b,c,sum,carry);
output sum,carry ;
input a,b,c ;
assign sum = a ^ b ^ c;
assign carry = (a&b) | (b&c) |
(c&a);
endmodule
```

BEHAVIORAL MODELLING:

```
module full_adder
(a,b,c,sum,carry);
output sum,carry;
reg sum,carry;
input a,b,c;
always @(a,b,c)
begin
sum=a^b^c;
carry=(a&b)|(b&c)|(c&a);
end
```

STRUCURAL MODELLING:

```
module
full_adder(sum,carry,a,b,c);
input a,b,c;
output sum,carry;
wire w1,w2,w3;
xor x1(w1,a,b);
xor x2(sum,w1,c);
and a1(w2,a,b);
and a2(w3,w1,c);
or(carry,w2,w3);
endmodule
```

FULL ADDER USING HALF ADDER

```
module half_adder(x,y,s,c);
input x,y;
output s,c;
xor(s,x,y);
and(c,x,y);
endmodule
module
full_adder(x,y,cin,s,cout);
input x,y,cin;
output s,cout;
wire s1,c1,c2;
half_adder ha1(x,y,s1,c1);
half_adder ha2(cin,s1,s,c2);
or(cout,c1,c2);
endmodule
```

EXP.NO: 2	SIMULATION OF HALF ADDER AND FULL ADDER
DATE:	

AIM:

To write a verilog program for half adder and full adder to synthesize and simulate using Xilinx software tool.

SOFTWARE REQUIRED:

1. Xilinx ISE Design Suite 14.1
2. Spartan3E kit

THEORY:**HALF ADDER:**

The half adder consists of two input variables designated as Augends and Addend bits. Output variables produce the Sum and Carry. The „carry” output is 1 only when both inputs are 1 and „sum” is 1 if any one input is 1. The Boolean expression is given by,

$$\begin{aligned}\text{sum} &= x \wedge y \\ \text{carry} &= x \& y\end{aligned}$$

FULL ADDER:

A Full adder is a combinational circuit that focuses the arithmetic sum of three bits. It consists of 3 inputs and 2 outputs. The third input is the carry from the previous Lower Significant Position. The two outputs are designated as Sum (S) and Carry (C). The binary variable S gives the value of the LSB of the Sum. The output S=1 only if odd number of 1's are present in the input and the output C=1 if two or three inputs are 1.

$$\begin{aligned}\text{sum} &= x \wedge y \wedge z \\ \text{carry} &= (x \& y) \vee (y \& z) \vee (x \& z)\end{aligned}$$

PROCEDURE:

1. Click on the Xilinx ISE Design Suite 14.1 or Xilinx Project navigator icon on the desktop of PC.
2. Write the Verilog code by choosing HDL as top level source module.
3. Check syntax, view RTL schematic and note the device utilization summary by double clicking on the synthesis in the process window.
4. Perform the functional simulation using Xilinx ISE simulator.
5. The output can be observed by using model sim.

RESULT

Thus the for half adder and full adder verilog program for to synthesize and simulate using Xilinx software tool was verified.

PROGRAM:

(SIMULATION OF ADDER)

4-BIT ADDER

```

module adder4(a,b,c,sum,Cout);
input [3:0]a,b;
input c;
output [3:0]sum;
output Cout;
assign {Cout,sum}=a+b+c;
endmodule

```

ADDITION OF TWO 8-BIT NUMBERS

```

module adder(a,b, sum,carry);
input [7:0]a,b;
output [7:0]sum,carry;
assign {carry,sum}=a+b;
endmodule

```

ADDITION OF FOUR 8-BIT NUMBERS

```

module
adder(a,b,c,d,sum,carry);
input [7:0]a,b,c,d;
output [7:0]sum,carry;
assign {carry,sum}=a+b+c+d;
endmodule

```

8-BIT RIPPLE CARRY ADDER

```

module fulladd(a, b, cin, sum,
cout);
input a,b,cin;
output sum, cout;
assign sum=(a^b^cin);
assign
cout=((a&b)|(b&cin)|(a&cin));
endmodule

```

```

module ripplemod(a, b, cin,
sum, cout);
input [7:0] a,b;
input cin;

```

```

output [7:0]sum;
output cout;
wire[6:0] c;
fulladd
a1(a[0],b[0],cin,sum[0],c[0]);
fulladd
a2(a[1],b[1],c[0],sum[1],c[1]);
fulladd
a3(a[2],b[2],c[1],sum[2],c[2]);
fulladd
a4(a[3],b[3],c[2],sum[3],c[3]);
fulladd
a5(a[4],b[4],c[3],sum[4],c[4]);
fulladd
a6(a[5],b[5],c[4],sum[5],c[5]);
fulladd
a7(a[6],b[6],c[5],sum[6],c[6]);
fulladd
a8(a[7],b[7],c[6],sum[7],cout);
endmodule

```

4-BIT RIPPLE CARRY ADDER

```

module fulladd(a, b, cin, sum,
cout);
input a,b,cin;
output sum,cout;
assign sum=(a^b^cin);
assign
cout=((a&b)|(b&cin)|(a&cin));
endmodule
module ripplemod(a, b, cin,
sum, cout);
input [3:0]a,b;
input cin;
output [3:0]sum;
output cout;
wire[2:0] c;
fulladd
a1(a[0],b[0],cin,sum[0],c[0]);
fulladd
a2(a[1],b[1],c[0],sum[1],c[1]);
fulladd
a3(a[2],b[2],c[1],sum[2],c[2]);
fulladd
a4(a[3],b[3],c[2],sum[3],cout);
endmodule

```

EXP.NO: 3	SIMULATION OF ADDER
DATE:	

AIM:

To write a verilog program for Adder to synthesize and simulate using Xilinx software tool.

SOFTWARE REQUIRED:

1. Xilinx ISE Design Suite 14.1
2. Spartan3E kit

THEORY :

When you add large numbers carefully together the addition is done digit by digit. The computer does the same. In the illustration, two 8 –digit binary numbers are being added. The top row contains the first number and the second row the other. Working from the right-hand side, there can be no 'carry' to add to the sum of the first two digits, so a half adder is sufficient. But for the second and subsequent pairs of digits, full adders must be used (any carry' is indicated by a f below the adder). The output will be an 8- bit and if the carry is formed that will be shown in cout output value.

PROCEDURE:

1. Click on the Xilinx ISE Design Suite 14.1 or Xilinx Project navigator icon on the desktop of PC.
2. Write the Verilog code by choosing HDL as top level source module.
3. Check syntax, view RTL schematic and note the device utilization summary by double clicking on the synthesis in the process window.
4. Perform the functional simulation using Xilinx ISE simulator.
5. The output wave form can be observed in model sim.

RESULT:

Thus the for Adder verilog program for to synthesize and simulate using Xilinx software tool was verified.

PROGRAM:

(SIMULATION OF MULTIPLIER)

MULTIPLICATION OF TWO 4-BIT NUMBERS / 4-BIT MULTIPLIER

```

module adder(a,b, out);
input [3:0]a,b;
output [7:0]out;
assign out= a*b;
endmodule

```

BOOTH'S MULTIPLIER (4-BIT MULTIPLIER)

```

module bm(xa,ya,za);
input [3:0]xa;
input [3:0]ya;
output [7:0]za;
integer i;
reg[9:0]temp;
reg[1:0]a;
reg[5:0]y;
reg[5:0]x;
reg[9:0]z;
reg[7:0]za;
always@ (xa or ya)
begin
y[5:0]=6'b000000;
y[5:1]=ya[3:0];
x[4:0]=xa[3:0];
z=9'b000000000;
temp=10'b000000000;
for(i=0;i<=4;i=i+1)
begin
a[0]=y[i];
a[1]=y[i+1];
case(a)
2'b01:temp[4:0]=x[4:0];

```

```

2'b10:temp[4:0]=(~x[4:0])+5'b0001;
2'b11:temp[4:0]=5'b00000;
2'b00:temp[4:0]=5'b00000;
endcase
if(temp[4]==0)
begin
temp[9:5]=5'b00000;
end
else
begin
temp[9:5]=5'b11111;
end
temp=temp<<i;
z=z+temp;
end
za=z;
end
endmodule

```

EXP.NO: 4	SIMULATION OF MULTIPLIER
DATE:	

AIM:

To write a verilog program for multiplier to synthesize and simulate using Xilinx software tool.

SOFTWARE REQUIRED:

1. Xilinx ISE Design Suite 14.1
2. Spartan3E kit.

THEORY:

Multiplication of two elements in the polynomial basis can be accomplished in the normal way of multiplication, but there are a number of ways to speed up multiplication, especially in hardware. In this type the multiplication can done parallel counter and it is generate carry. The multiplication is independent of the carry so we can perform N number of multiplication independent of carry.

PROCEDURE:

1. Click on the Xilinx ISE Design Suite 14.1 or Xilinx Project navigator icon on the desktop of PC.
2. Write the Verilog code by choosing HDL as top level source module.
3. Check syntax, view RTL schematic and note the device utilization summary by double
4. Clicking on the synthesis in the process window.
5. Perform the functional simulation using Xilinx ISE simulator.
6. The output wave form can be observed in model sim.

RESULT:

Thus the for multiplier verilog program for to synthesize and simulate using Xilinx software tool was verified.

SIMULATION OF BINARY TO GRAY CODE CONVERSION:

```
module btog(din,dout);  
output [3:0]dout;  
input [3:0]din;  
assign dout[3]=din[3];  
assign dout[2]=din[3]^din[2];  
assign dout[1]=din[2]^din[1];  
assign dout[0]=din[1]^din[0];  
endmodule
```


EXP.NO: 5	SIMULATION OF BINARY TO GRAY CODE COUNTER
DATE:	

AIM:

To write a verilog program for D-flip flop and D-latch to synthesize and simulate using xilinx software tool.

SOFTWARE REQUIRED:

1. Xilinx ISE Design Suite 14.1
2. Spartan3E kit

THEORY:**CODE CONVERTOR:**

Code is a symbolic representation of discrete information. Codes are of different types. Gray Code is one of the most important codes. It is a non-weighted code which belongs to a class of codes called minimum change codes. In this codes while traversing from one step to another step only one bit in the code group changes. In case of Gray Code two adjacent code numbers differs from each other by only one bit. As this code it is not applicable in any types of arithmetical operations but it has some applications in analog to digital converters and in some input/output devices. Now let us concentrate on the table of Gray Code given below where we can find the difference of binary code from gray code while traversing through the table for their respective decimal numbers.

BINARY TO GRAY CODE CONVERSION:

Binary to gray code conversion is a very simple process. There are several steps to do this types of conversions. Steps given below elaborate on the idea on this type of conversion.

- (1) The M.S.B. of the gray code will be exactly equal to the first bit of the given binary number.
- (2) Now the second bit of the code will be exclusive-or of the first and second bit of the given binary number, i.e if both the bits are same the result will be 0 and if they are different the result will be 1.
- (3) The third bit of gray code will be equal to the exclusive-or of the second and third bit of the given binary number. Thus the Binary to gray code conversion goes on. One example given below can make your idea clear on this type of conversion.

Let (01001) be the given binary number Thus the equivalent gray code is 01101.

Now concentrate on the example where the M.S.B. of the binary is 0 so for it will be 0 for the most significant gray bit. Next, the XOR of the first and the second bit is done. The bits are different so the resultant gray bit will be 1. Again move to the next step, XOR of second and third bit is again 1 as they are different. Next, XOR of third and fourth bit is 0 as both the bits are same. Lastly the XOR of fourth and fifth bit is 1 as they are different. That is how the result of binary to gray code conversion of 01001 is done whose equivalent gray code is 01101.

PROCEDURE:

1. Click on the Xilinx ISE Design Suite 14.1 or Xilinx Project navigator icon on the desktop of PC.
2. Write the Verilog code by choosing HDL as top level source module.
3. Check syntax, view RTL schematic and note the device utilization summary by double clicking on the synthesis in the process window.
4. Perform the functional simulation using Xilinx ISE simulator.
5. The output can be observed using model sim.

RESULT:

Thus the multiplier verilog program for to synthesize and simulate using Xilinx software tool was verified.

PROGRAM:

(SIMULATION OF FLIP FLOPS)

D LATCH:

```

module
dlatch(d,en,b,c,a,q,qbar);
input d,en;
output b,c,a,q,qbar;
wire b,c,a,q,qbar;
not(a,d);
nand(b,d,en);
nand(c,a,en);
nand(q,b,qbar);
nand(qbar,c,q);
endmodule

```

JK FLIP FLOP:

```

module JK_Flip_Flop
(Q,Qbar,clk,reset,J,K);
output Q,Qbar;
input clk, reset,J,K;
reg Q;
always @(posedge clk)
if (reset)
begin
Q <= 0;
end
else if (J == 0 && K == 0)
begin
Q <= Q;
end
else if (J == 0 && K == 1)
begin
Q <= 0;
end
else if (J == 1 && K == 0)
begin
Q <= 1;
end
else if (J == 1 && K == 1)

```

```

begin
Q <= Qbar;
end
assign Qbar = ~Q;
endmodule

```

SR FLIP FLOP:

```

module SR_Flip_Flop
(Q,Qbar,clk,reset,S,R);
output Q,Qbar;
input clk, reset,S,R;
reg Q;
always @(posedge clk)
if (reset)
begin
Q <= 0;
end
else if (S == 0 && R == 0)
begin
Q <= Q;
end
else if (S == 0 && R == 1)
begin
Q <= 0;
end
else if (S == 1 && R == 0)
begin
Q <= 1;
end
else if (S == 1 && R == 1)
begin
Q <= 1'bx;
end
assign Qbar = ~Q;
endmodule

```

EXP.NO: 6	SIMULATION OF FLIP FLOPS
DATE:	

AIM:

To write a verilog program for various flip flops to synthesize and simulate using Xilinx software tool.

SOFTWARE REQUIRED:

1. Xilinx ISE Design Suite 14.1
2. Spartan3E kit

THEORY:**D-FLIP FLOP:**

It has only a single data input. That data input is connected to the S input of RS-flip flop, while the inverse of D is connected to the R input. This prevents that the input combination ever occurs. To allow the flip flop to be in holding state, a D-flip flop has a second input called “clock”. The clock input is AND-ed with the D input, such that when clock=0, the R and S inputs of the RS-flip flop are 0 and the state is held.

D-LATCH:

It has only a single data input. That data input is connected to the S input of RS-flip flop, while the inverse of D is connected to the R input. This prevents that the input combination ever occurs. To allow the flip flop to be in holding state, a D-flip flop has a second input called “enable”. The enable input is AND-ed with the D input, such that when enable=0, the R and S inputs of the RS-flip flop are 0 and the state is held.

D FLIP FLOP:

```
module D_Flip_Flop
(Q,clk,reset,d);
output Q;
input clk,reset,d;
reg Q;
always @(posedge clk)
if (reset)
begin
Q <= 0;
end
else begin
Q <= d;
end
endmodule
```

T FLIP FLOP:

```
module T_Flip_Flop
(Q,clk,reset,t);
output Q;
input clk,reset,t;
reg Q;
always @(posedge clk)
if (reset)
begin
Q <= 0;
end
else if (t == 0)
begin
Q <= Q;
end
else if (t == 1)
begin
Q <= ~Q;
end
endmodule
```

PROCEDURE:

1. Click on the Xilinx ISE Design Suite 14.1 or Xilinx Project navigator icon on the desktop of PC.
2. Write the Verilog code by choosing HDL as top level source module.
3. Check syntax, view RTL schematic and note the device utilization summary by double clicking on the synthesis in the process window.
4. Perform the functional simulation using Xilinx ISE simulator.
5. The output can be observed using model sim

RESULT:

Thus the various flip flops verilog program for to synthesize and simulate using Xilinx software tool was verified.

PROGRAM:**PSEUDO RANDOM BINARY SEQUENCE (4-BIT):**

```

module tff(q,t,c);
output q;
input t,c;
reg q;
initial
begin
q=1'b1;
end
always @ (posedge c)
begin
if (t==1'b0) begin q=q; end
else begin q=~q; end
end
endmodule
module tff1(q,t,c);
output q;
input t,c;
reg q;
initial
begin
q=1'b0;
end
always @ (posedge c)
begin
if (t==1'b0) begin q=q; end
else begin q=~q; end
end
endmodule
module random(o,clk);
output [3:0]o;
input clk;
xor (t0,o[3],o[2]);
assign t1=o[0];
assign t2=o[1];
assign t3=o[2];
tff u1(o[0],t0,clk);
tff1 u2(o[1],t1,clk);
tff1 u3(o[2],t2,clk);
tff1 u4(o[3],t3,clk);
endmodule

```

EXP.NO: 7	SIMULATION OF PSEUDO RANDOM BINARY SEQUENCE
DATE:	

AIM:

To write a verilog program for Pseudo Random Binary Sequence to synthesize and simulate using Xilinx software tool.

SOFTWARE REQUIRED:

1. Xilinx ISE Design Suite 14.1
2. Spartan3E kit

THEORY :**PRBS :**

A PRBS is 'pseudorandom', because, although it is in fact deterministic, it seems to be random in a sense that the value of an element is independent of the values of any of the other elements, similar to real random sequences. A PRBS can be stretched to infinity by repeating it after elements, this in contrast to most random sequences, such as sequences generated by radioactive decay or by white noise, that are 'infinite' by nature. The PRBS is more general than the maximum length sequence, which is a special pseudo-random binary sequence of N bits generated as the output of a linear shift register. A maximum length sequence always has a 1/2 duty cycle and its number of elements .

PROCEDURE:

1. Click on the Xilinx ISE Design Suite 14.1 or Xilinx Project navigator icon on the desktop of PC.
2. Write the Verilog code by choosing HDL as top level source module.
3. Check syntax, view RTL schematic and note the device utilization summary by double clicking on the synthesis in the process window.
4. clicking on the synthesis in the process window.
5. Perform the functional simulation using Xilinx ISE simulator.
6. The output wave form can be observed in model sim.

RESULT:

Thus the various flip flops verilog program for to synthesize and simulate using Xilinx software tool was verified.

PROGRAM:**UP-DOWN COUNTER:**

```
module updowncounterm(clk, clear, updown, q);  
input clk;  
input clear;  
input updown;  
output [3:0] q;  
reg [3:0] q;  
always@(posedge clear or posedge clk)  
begin  
if(clear)  
q <=4'b0000;  
else if(updown)  
q <= q+1'b1;  
else  
q <= q-1'b1;  
end  
endmodule
```

EXP.NO: 8	SIMULATION OF UP-DOWN COUNTER
DATE:	

AIM:

To write a verilog program for Up-Down Counter to synthesize and simulate using Xilinx software tool.

SOFTWARE REQUIRED:

1. Xilinx ISE Design Suite 14.1
2. Spartan3E kit

THEORY :**UP-DOWN COUNTER :**

A counter that can change state in either direction, under the control of an up or down selector input, is known as an up/down counter. When the selector is in the up state, the counter increments its value. When the selector is in the down state, the counter decrements the count. Likewise the counter counts in both the directions continuously until attaining the end of the count. The count is initiated by the positive clock pulse. The counter counts from 0000 to 1111 for up count and 1111 to 0000 for down count.

PROCEDURE:

1. Click on the Xilinx ISE Design Suite 14.1 or Xilinx Project navigator icon on the desktop of PC.
2. Write the Verilog code by choosing HDL as top level source module.
3. Check syntax, view RTL schematic and note the device utilization summary by double clicking on the synthesis in the process window.
4. Perform the functional simulation using Xilinx ISE simulator.
5. The output wave form can be observed in model sim.

RESULT:

Thus the Up-Down Counter verilog program for to synthesize and simulate using Xilinx software tool was verified.

PROGRAM:**MOD 10 COUNTER / DECADE COUNTER:**

```
module counter(clk,rst,enable,counter_output);
input clk,rst,enable;
output reg [3:0]counter_output;
always@ (posedge (clk))
begin
if( rst | counter_output==4'b1001)
counter_output <= 4'b0000;
else if(enable)
counter_output <= counter_output + 1;
else
counter_output <= 4'b0000;
end
endmodule
```

MOD 12 COUNTER:

```
module counter(clk,rst,enable,counter_output);
input clk,rst,enable;
output reg [3:0]counter_output;
always@ (posedge (clk))
begin
if( rst | counter_output==4'b1011)
counter_output <= 4'b0000;
else if(enable)
counter_output <= counter_output + 1;
else
counter_output <= 4'b0000;
end
endmodule
```

EXP.NO: 9	SIMULATION OF COUNTER
DATE:	

AIM:

To write a verilog program for different Counter to synthesize and simulate using Xilinx software tool.

SOFTWARE REQUIRED:

1. Xilinx ISE Design Suite 14.1
2. Spartan3E kit

THEORY :**COUNTER :**

A counter that can change state in either direction, under the control of an up or down selector input, is known as an up/down counter. When the selector is in the up state, the counter increments its value. When the selector is in the down state, the counter decrements the count. Likewise the counter counts in both the directions continuously until attaining the end of the count. The count is initiated by the positive clock pulse. The counter counts from 0000 to 1111 for up count and 1111 to 0000 for down count.

PROCEDURE:

1. Click on the Xilinx ISE Design Suite 14.1 or Xilinx Project navigator icon on the desktop of PC.
2. Write the Verilog code by choosing HDL as top level source module.
3. Check syntax, view RTL schematic and note the device utilization summary by double clicking on the synthesis in the process window.
4. Perform the functional simulation using Xilinx ISE simulator.
5. The output wave form can be observed in model sim.

4-BIT UP COUNTER

```
module counter (clk,clr,q);  
input clk,clr;  
output [3:0]q;  
reg [3:0]tmp;  
always @(posedge clk or posedge clr)  
begin  
if (clr)  
tmp<=4'b0000;  
else  
tmp<=tmp + 1'b1;  
end  
assign q = tmp;  
endmodule
```

RESULT:

Thus the different Counter verilog program for to synthesize and simulate using Xilinx software tool was verified.

PROGRAM:**DESIGN OF SEQUENCE DETECTOR**

```
module me1(x,clk,rst,y);
input x,clk,rst;
output y;
reg[2:0]state;
reg y;
always @ (posedge clk)
begin
if(rst==1)begin
state<=3'b000;
y<=0;
end
else begin
case (state)
3'b000:begin
if (x) begin
state<=3'b001;
y<=0;
end
else begin
state<=3'b000;
y<=0;
end
end
3'b001:begin
if (x) begin
state<=3'b010;
y<=0;
end
else begin
state<=3'b000;
y<=0;
end
end
end
end
```

EXP.NO: 10	DESIGN OF SEQUENCE DETECTOR
DATE:	

AIM:

Designing, synthesizing and implementing a sequence detector for the given sequence using verilog.

SOFTWARE REQUIRED:

1. Xilinx ISE Design Suite 14.1
2. Spartan3E kit

ALGORITHM:

Step 1: Start the program.

Step 2: Declare the input, clk, reset.

Step 3: Declare the output as y.

Step 4: Draw the state diagram whose initial state is reset.

Step 5: If the input matches with a given sequence the present state transits to the next state.

Step 6: If the input does not match with a given sequence the state will be in present state or move to the reset state as per the condition given.

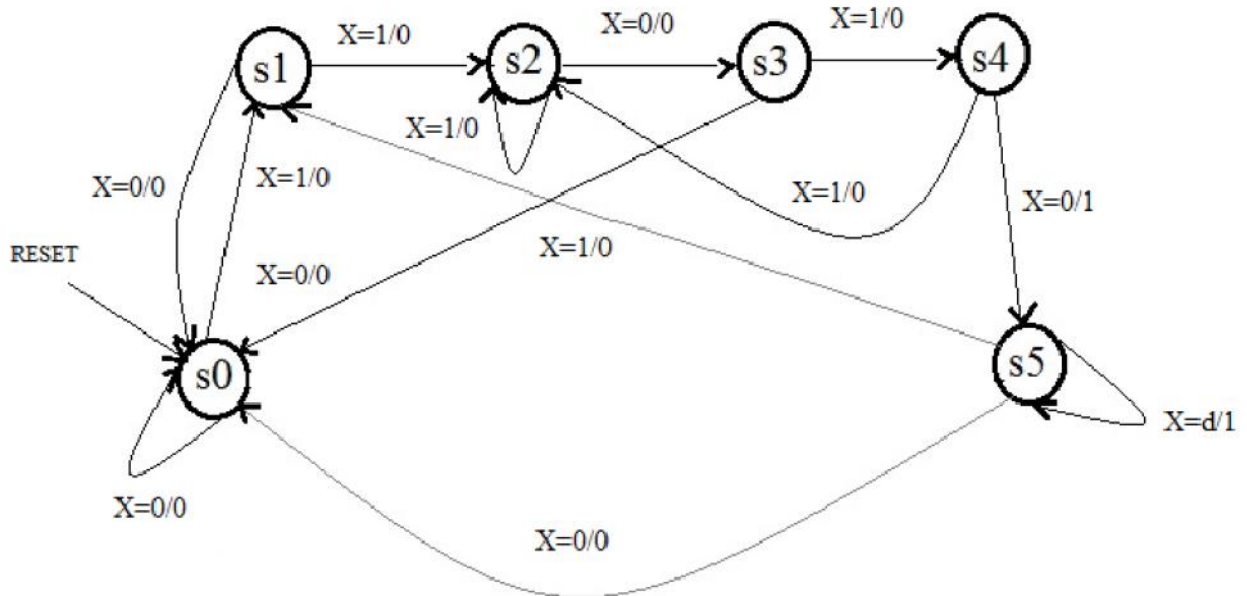
Step 7: The output is always remains “0” until the whole sequence is received.

Step 8: When the sequence is fully received the output changes from “0” to “1”.

THEORY:

A sequence detector is the one which detects the next state when the reset value is given. The transition takes place from the present state to the next state when the value is given or else the present state will be the same state. It can be described using state diagrams. Each state is used to represent the present state or the next state. The five bit sequence 11010 can be detected using this state diagram if reset is given then it goes to the next state s1 and attains the value 1 or else it stays in the same state s0.


```
3'b010:begin
if (x) begin
state<=3'b010;
y<=0;
end
else begin
state<=3'b011;
y<=0;
end
end
3'b011:begin
if (x) begin
state<=3'b100;
y<=0;
end
else begin
state<=3'b000;
y<=0;
end
end
3'b100:begin
if (x) begin
state<=3'b000;
y<=0;
end
else begin
state<=3'b101;
y<=1;
end
end
3'b101:begin
if (x) begin
state<=3'b001;
y<=0;
end
else begin
state<=3'b000;
y<=0;
end
end
endcase
end
end
endmodule
```



STATE TABLE:

PRESENT STATE	NEXT STATE		OUTPUT (Y)	
	X=0	X=1	X=0	X=1
S0	S0	S1	0	0
S1	S0	S2	0	0
S2	S3	S2	0	0
S3	S0	S4	0	0
S4	S5	S0	1	0
S5	S0	S1	0	0

RESULT:

Thus the sequence detector verilog program for to synthesize and simulate using Xilinx software tool was verified.

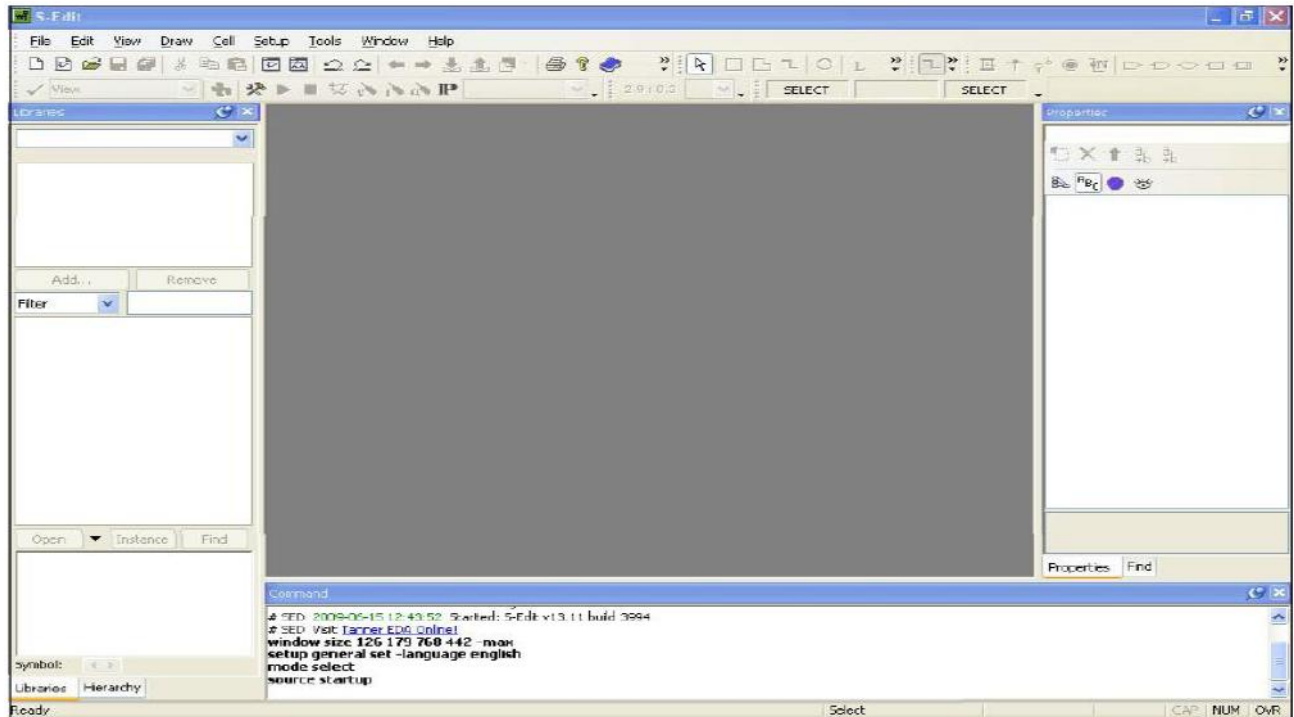
TANNER TOOLS

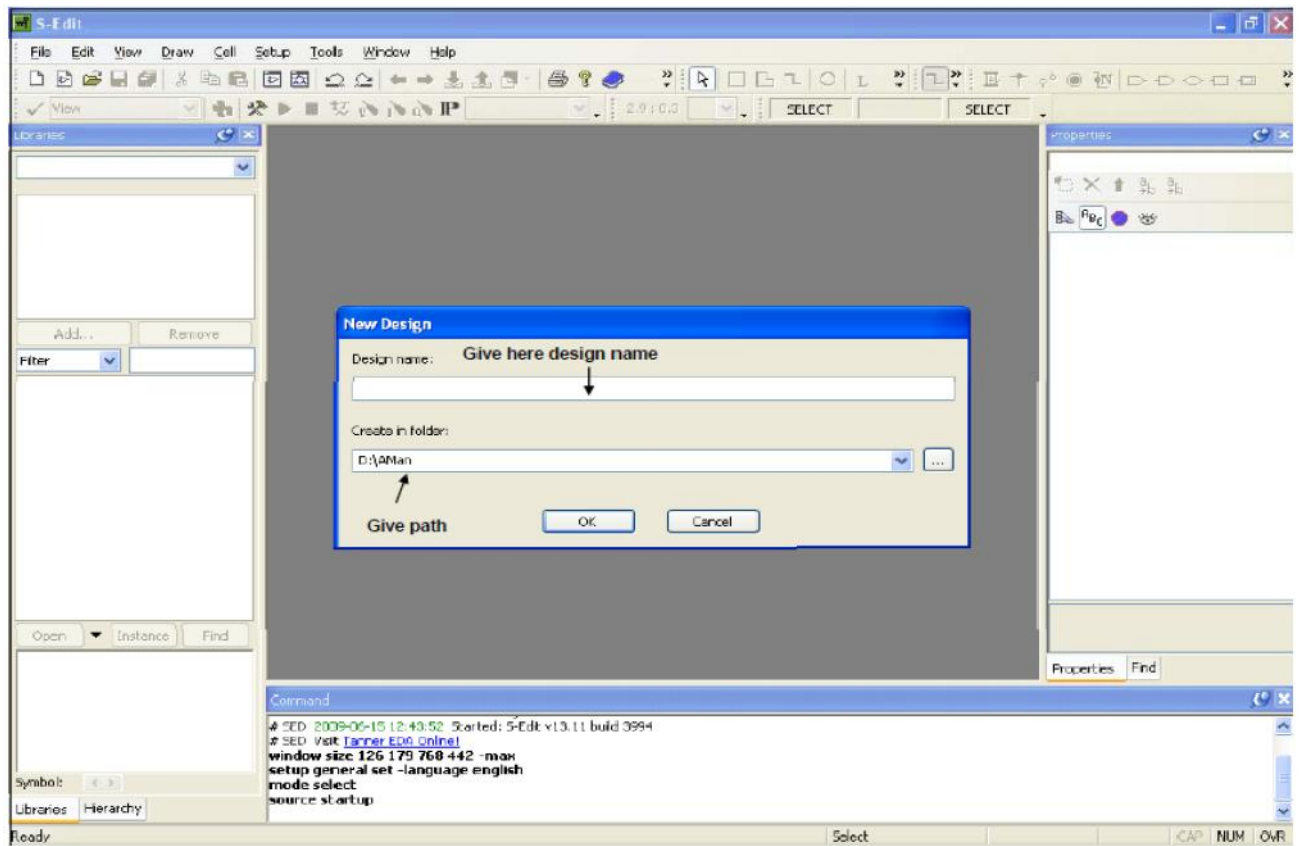
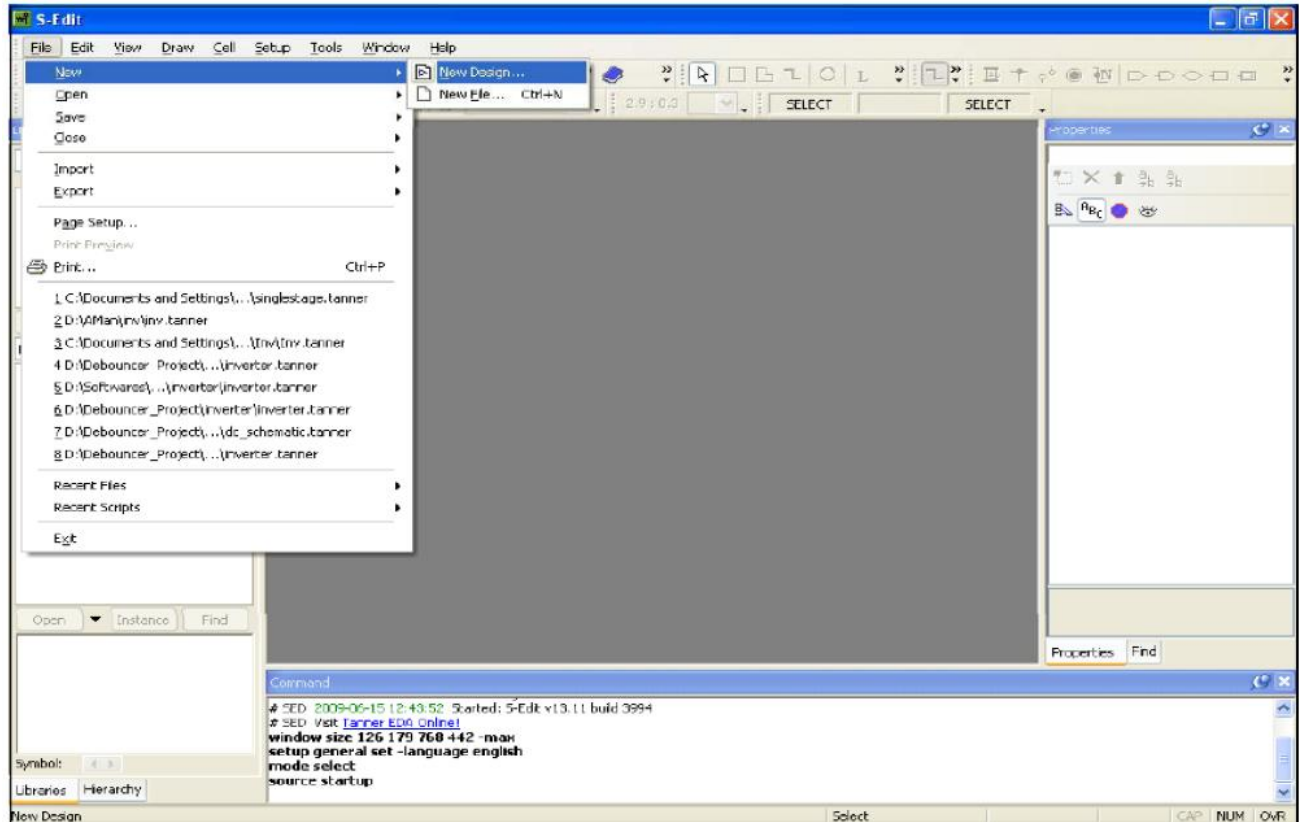
Steps to use Tanner tool:

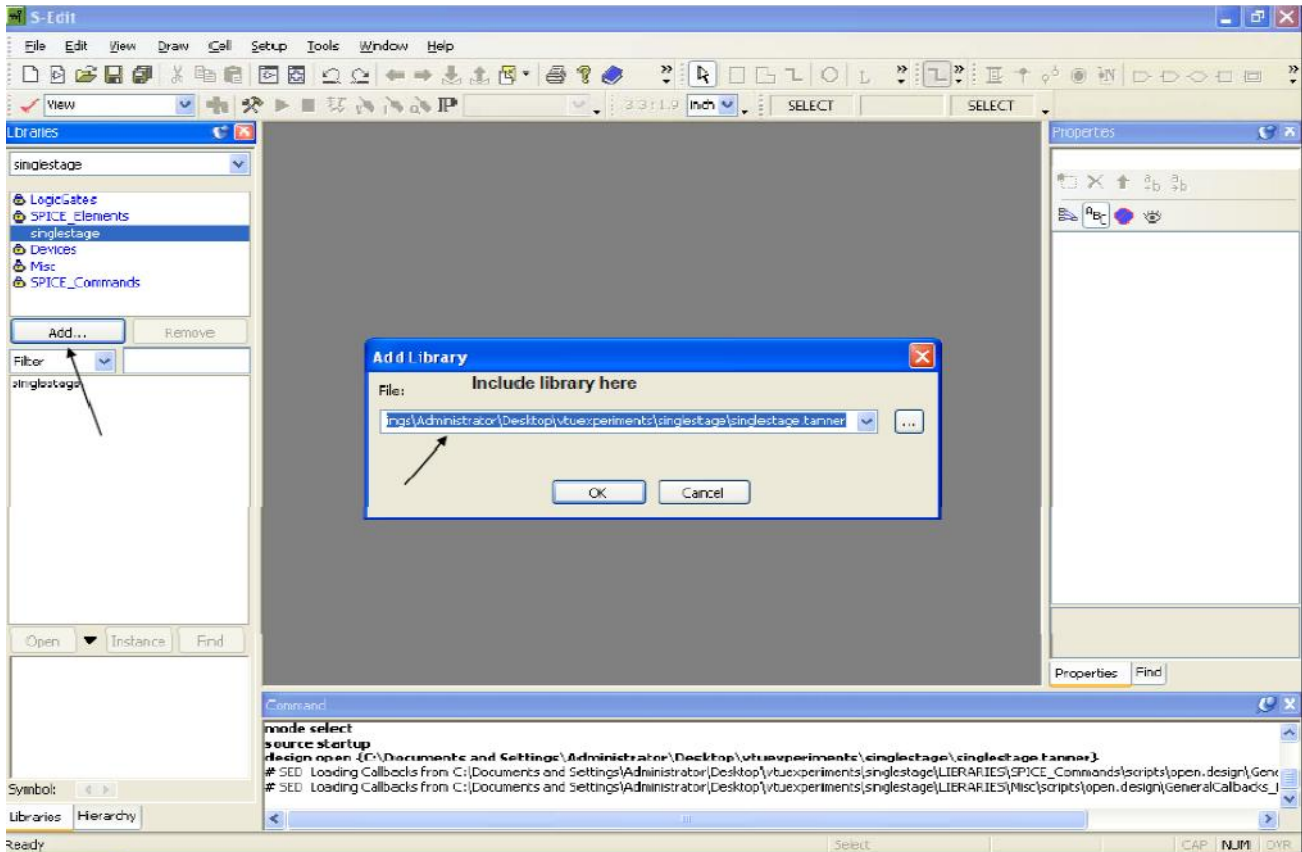
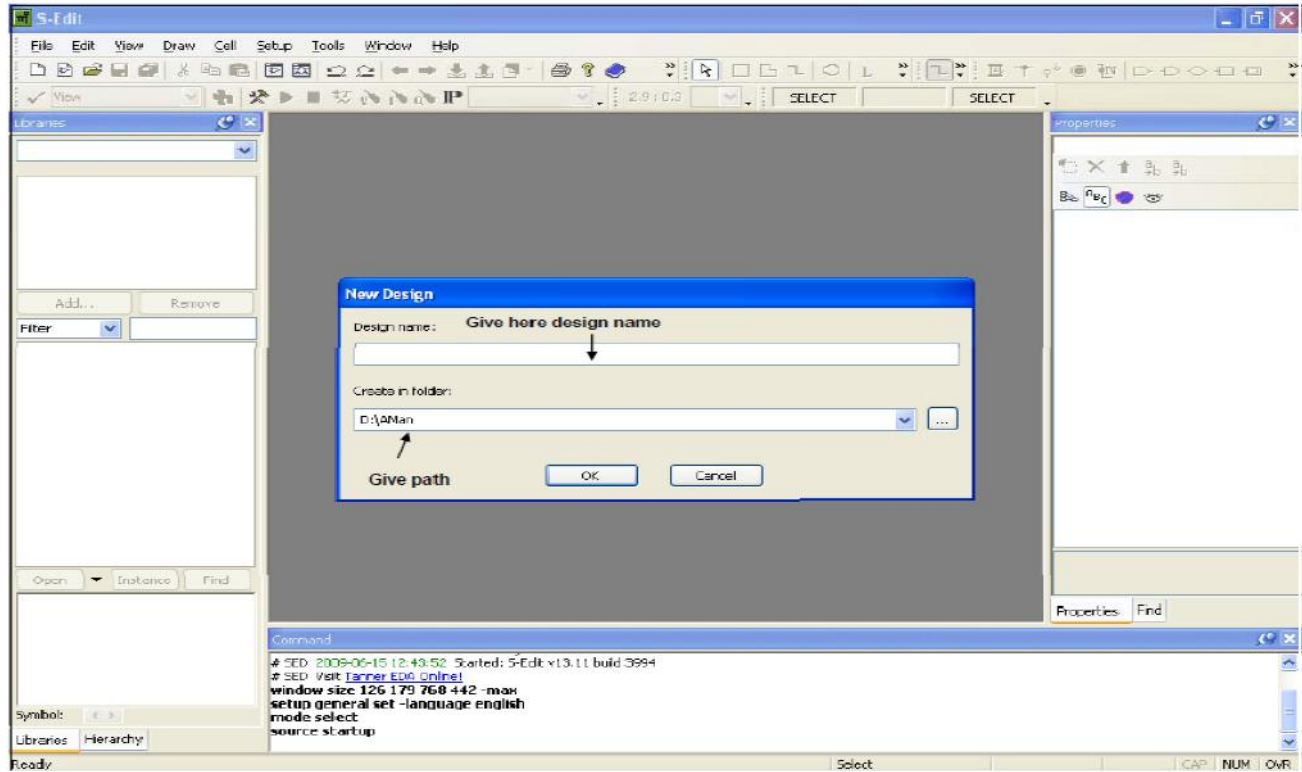
i.SCHEMATIC (S-edit):

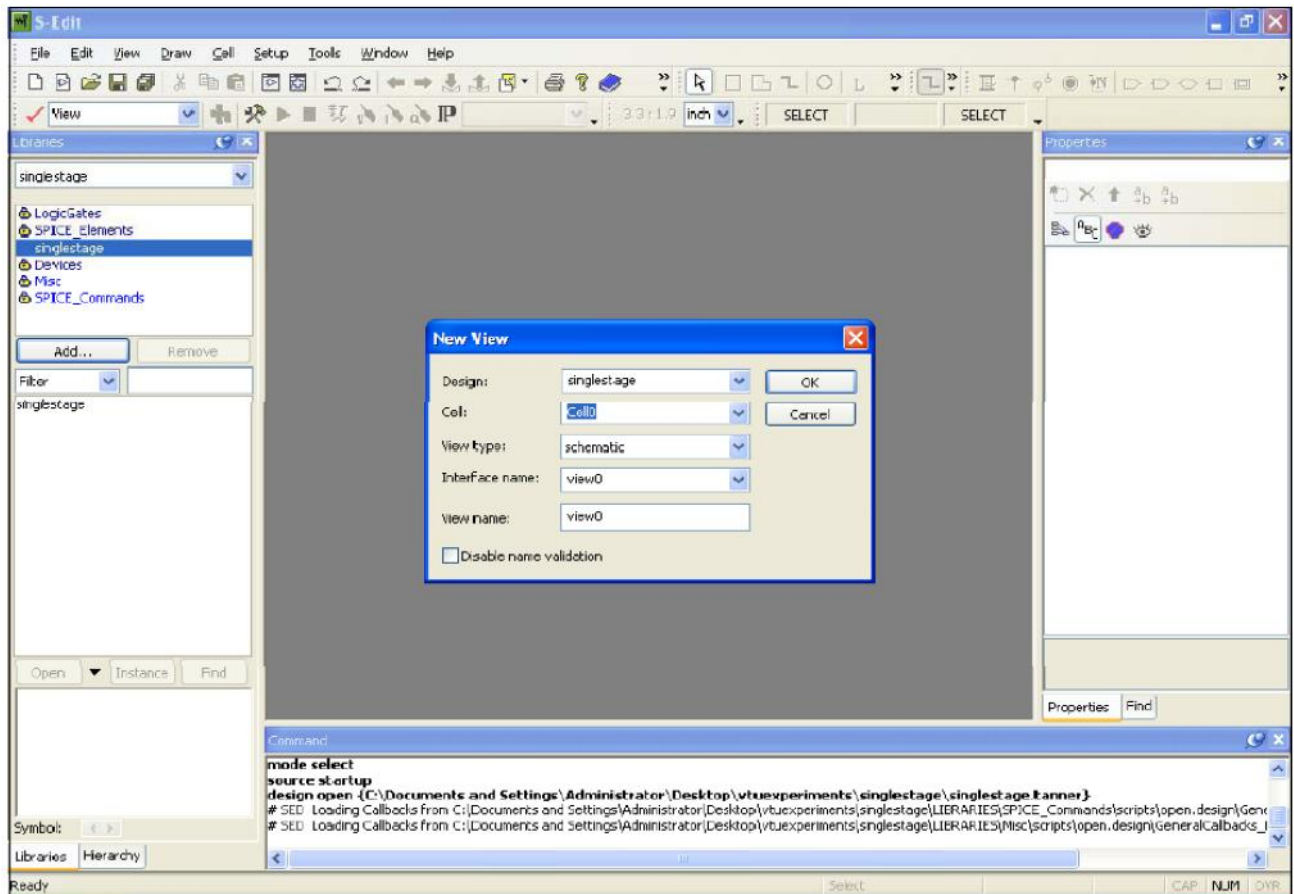
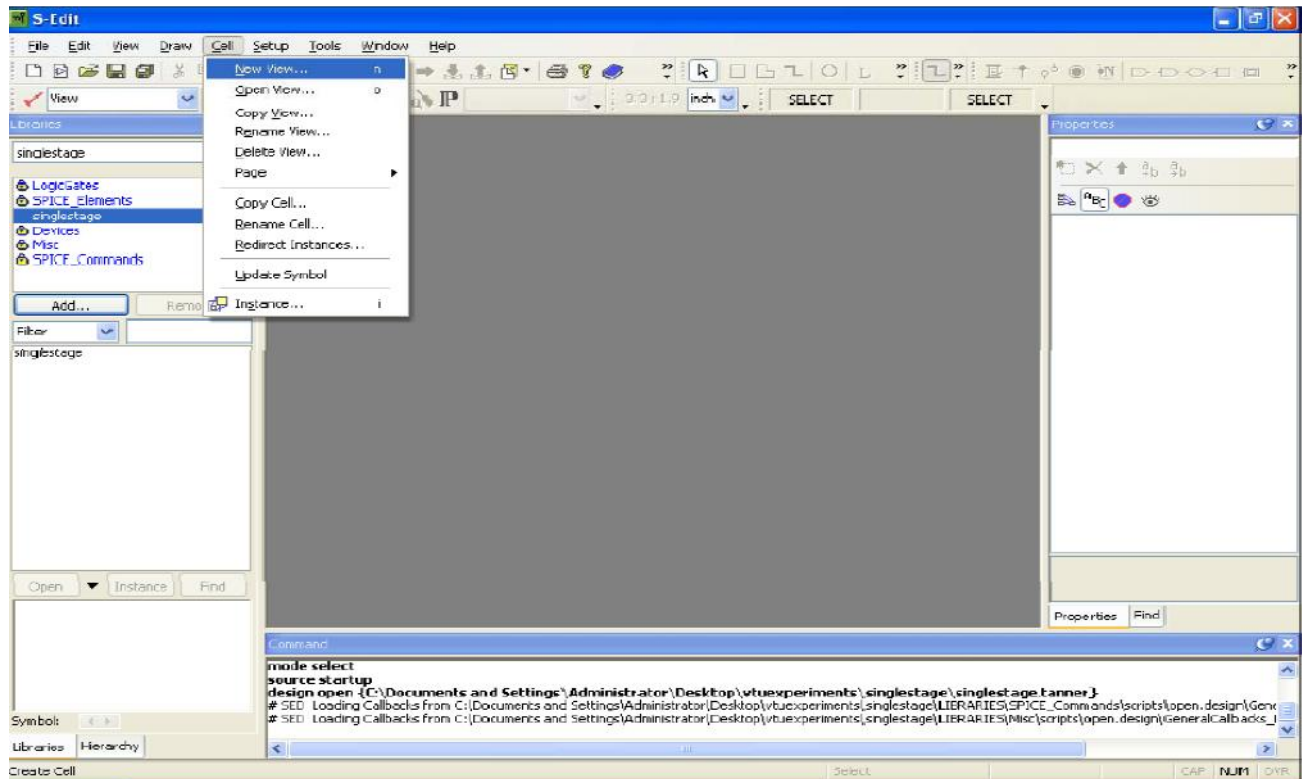
Start the tanner EDA by using the desktop shortcut or by using the

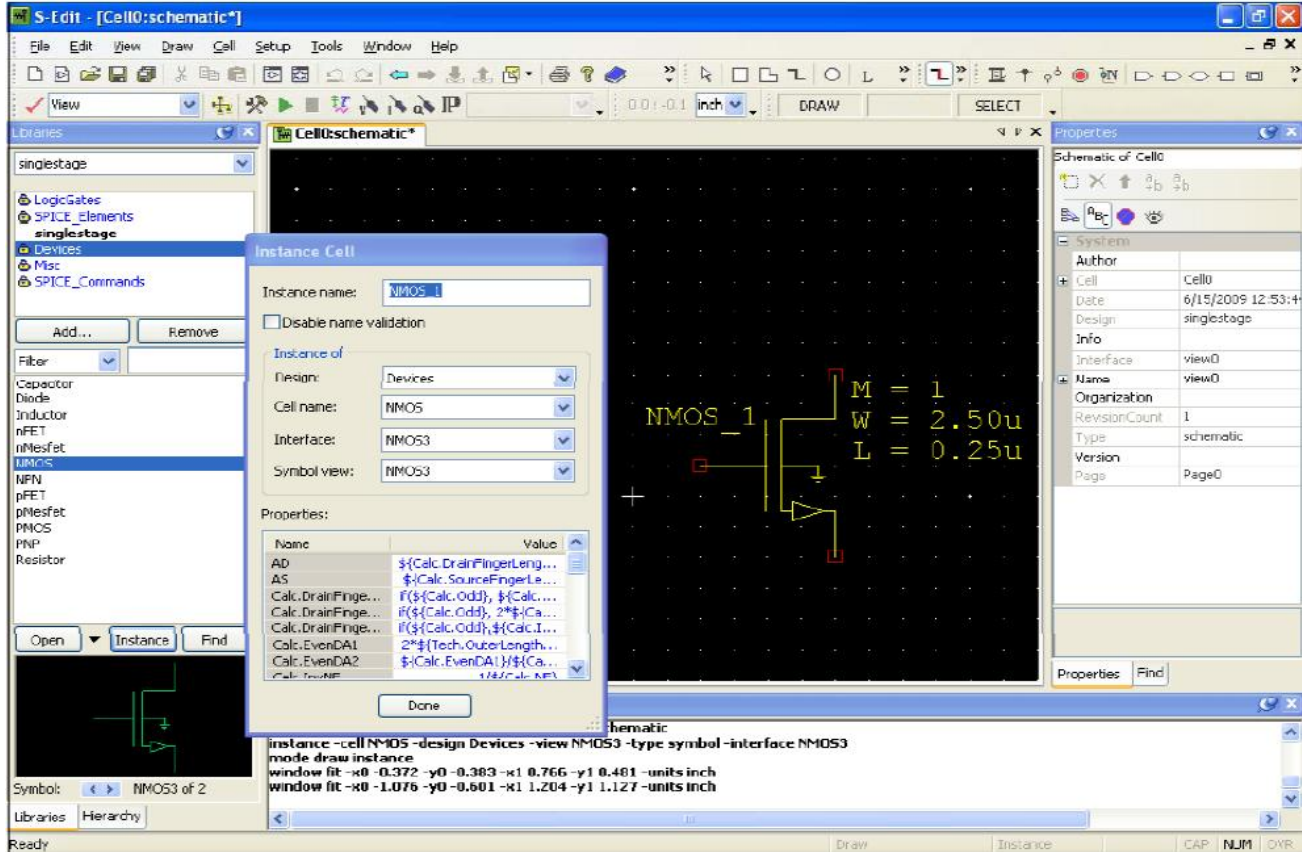
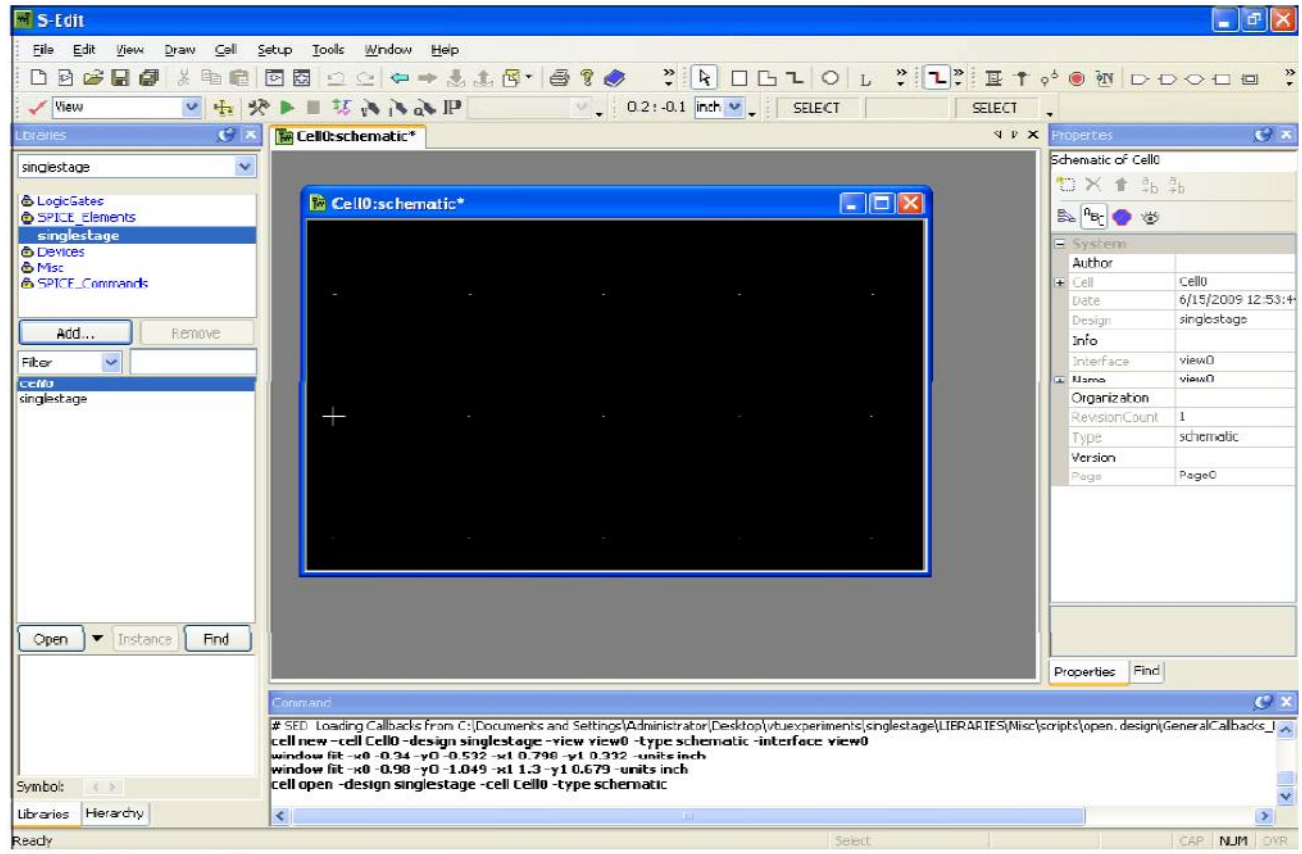
➤ Start → Programs → tanner EDA → tanner tool v13.0 → S-edit.

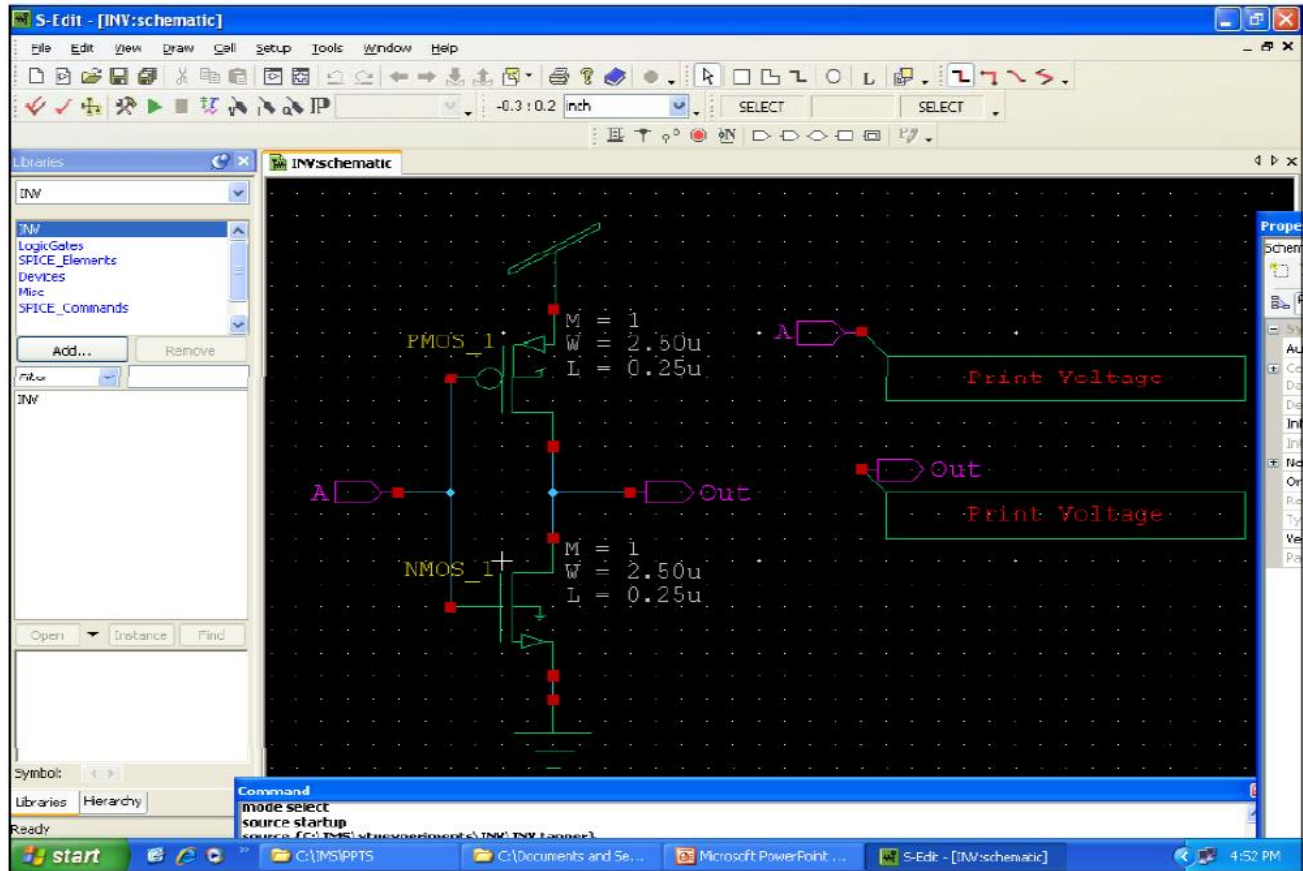




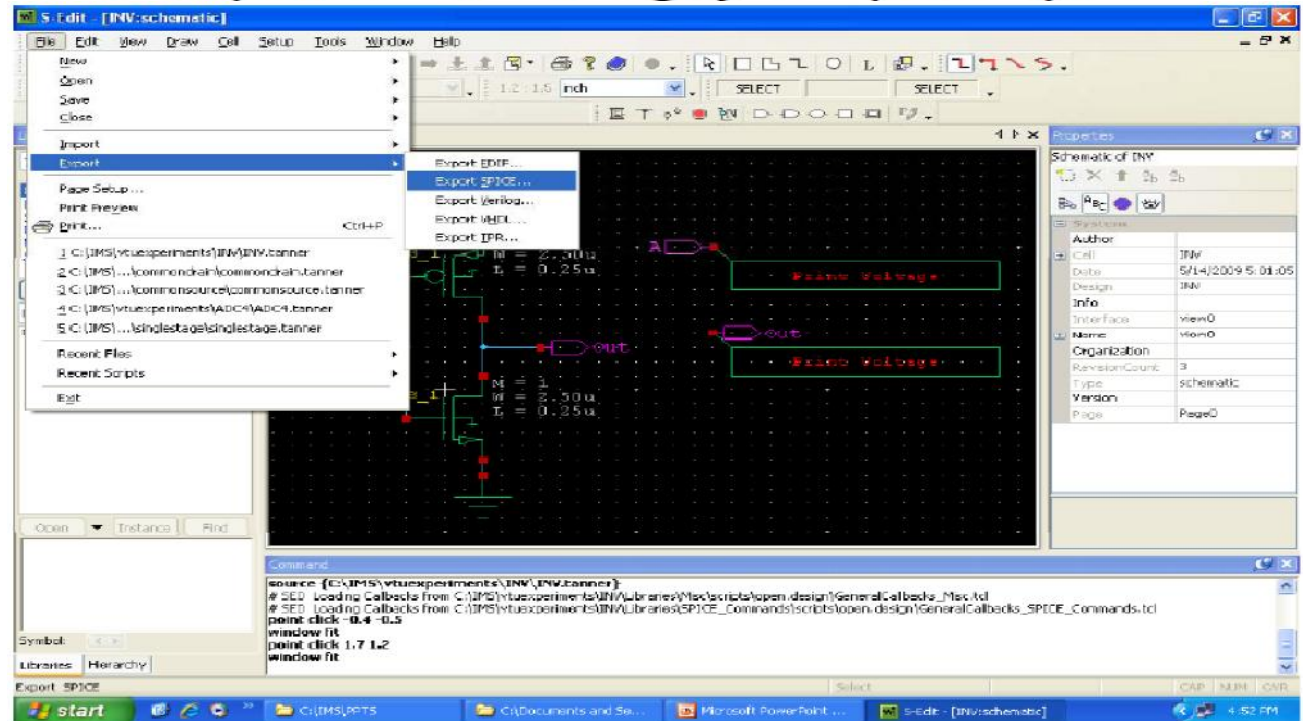


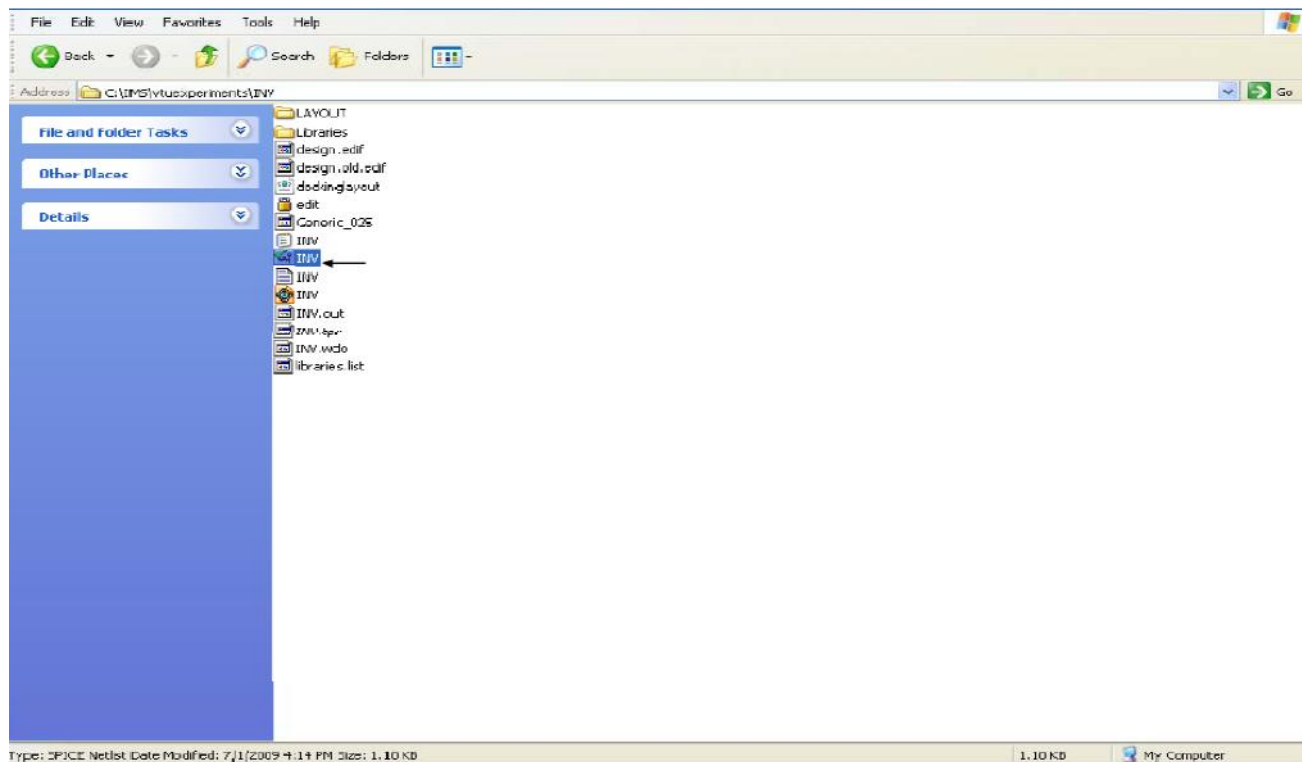
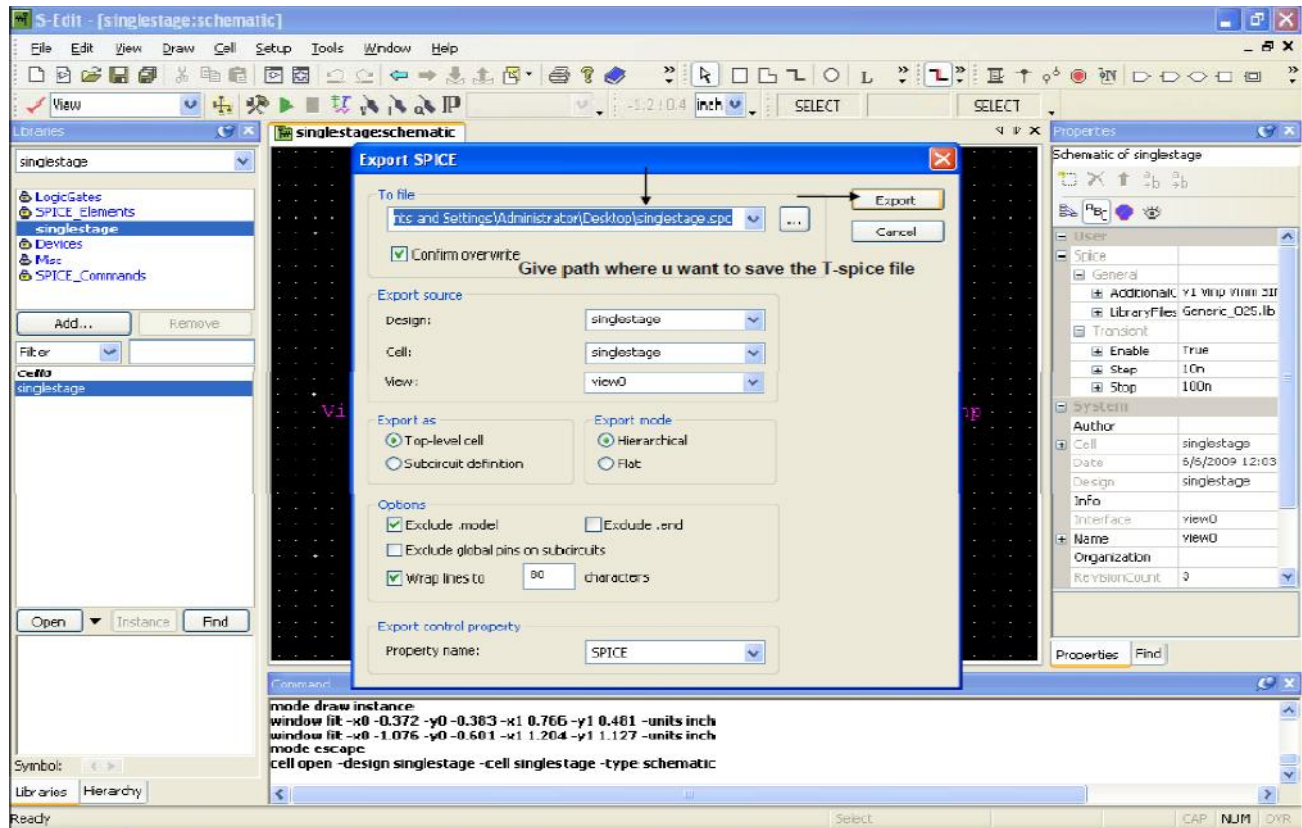


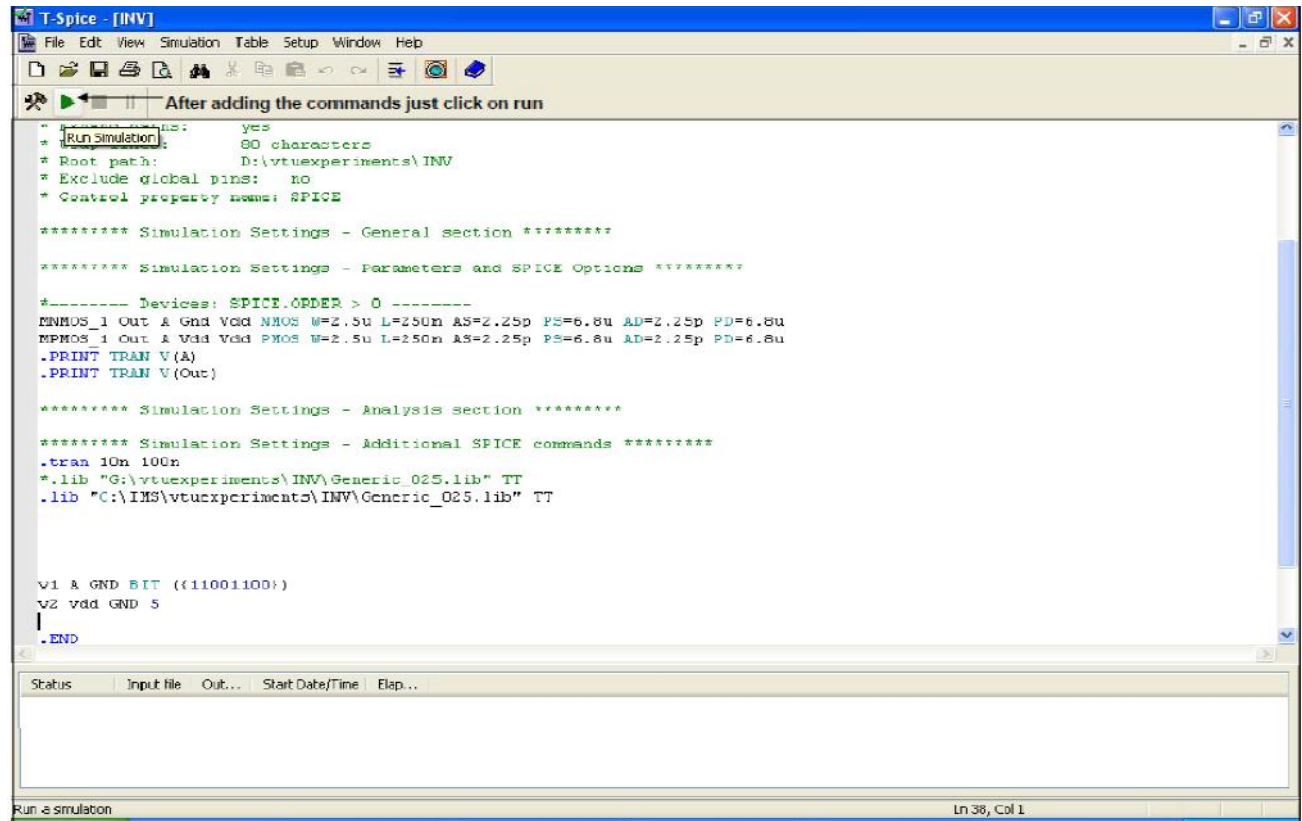
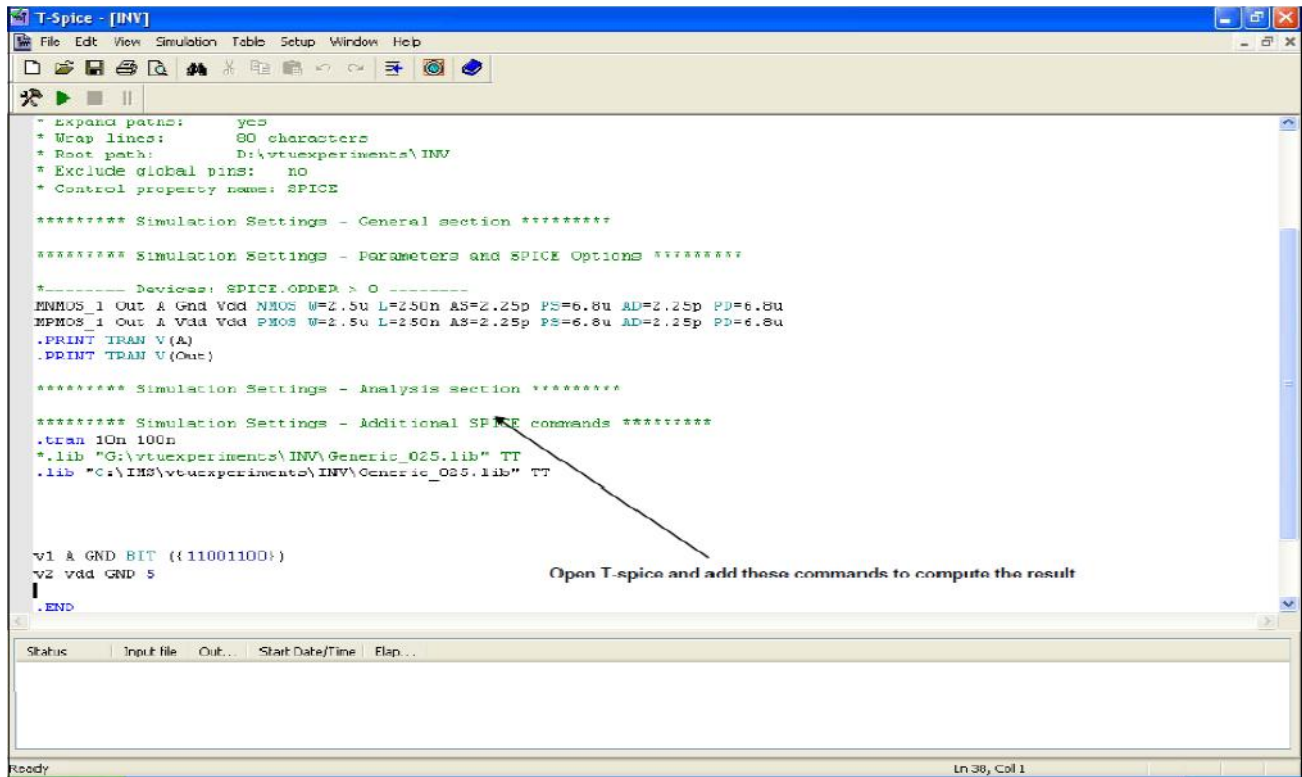


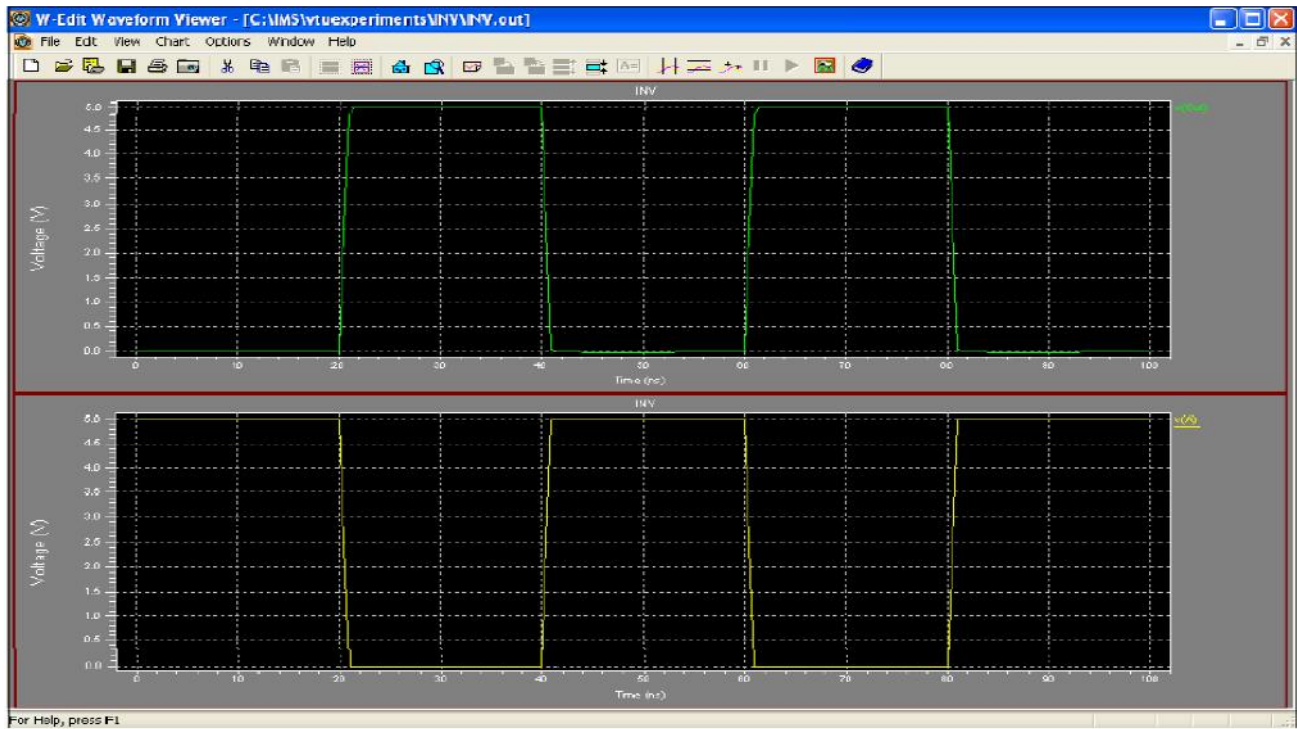


After completion of the design just export T-spice



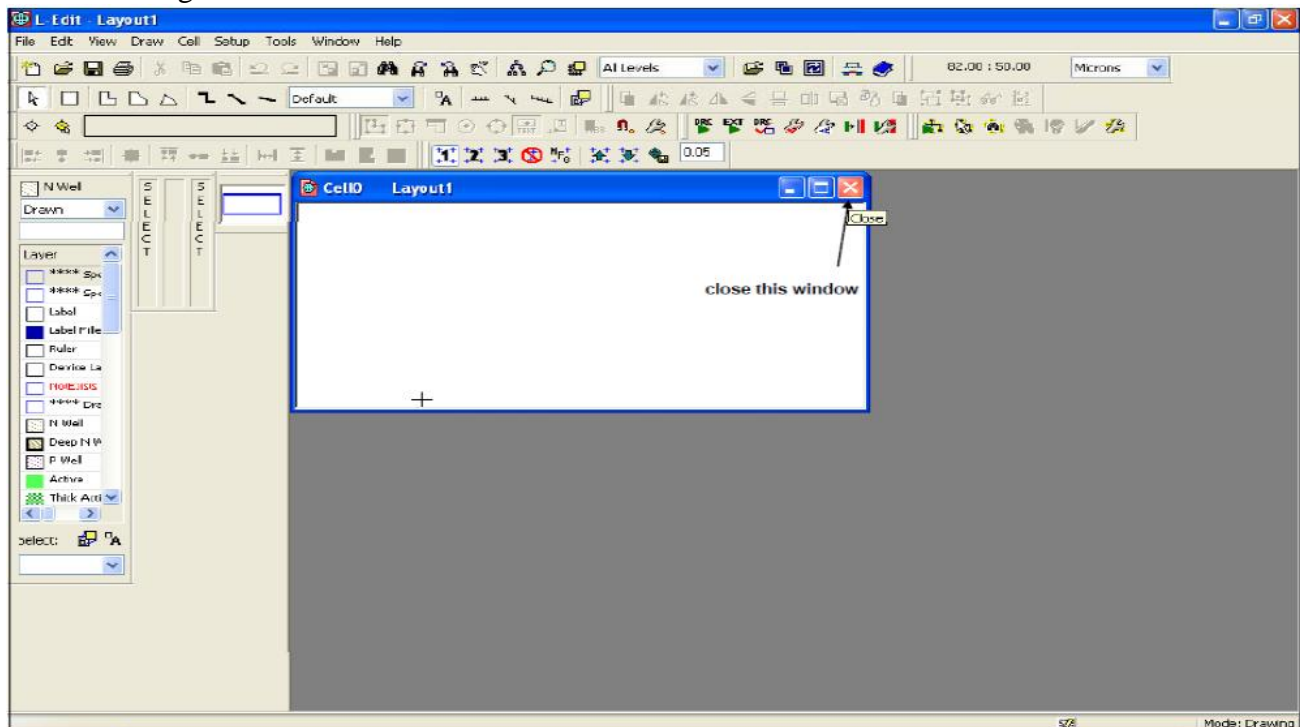


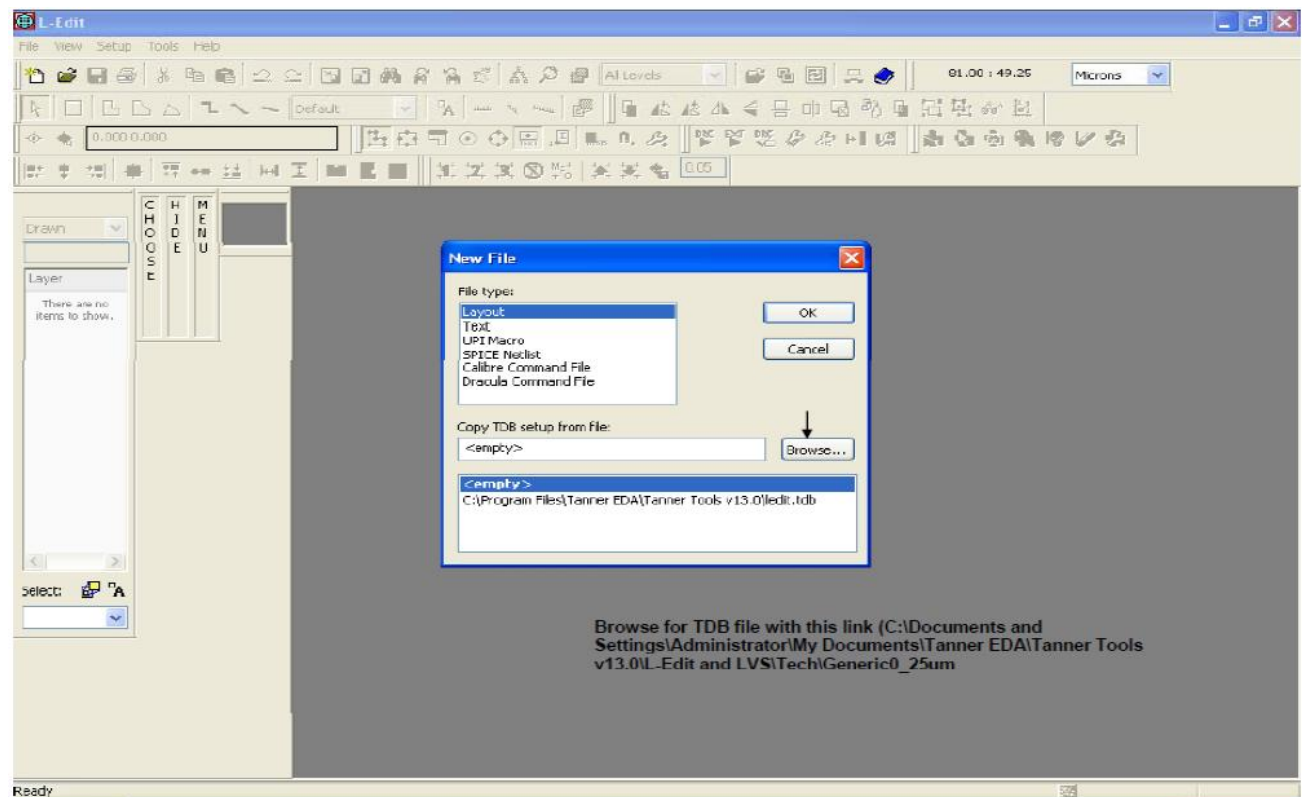
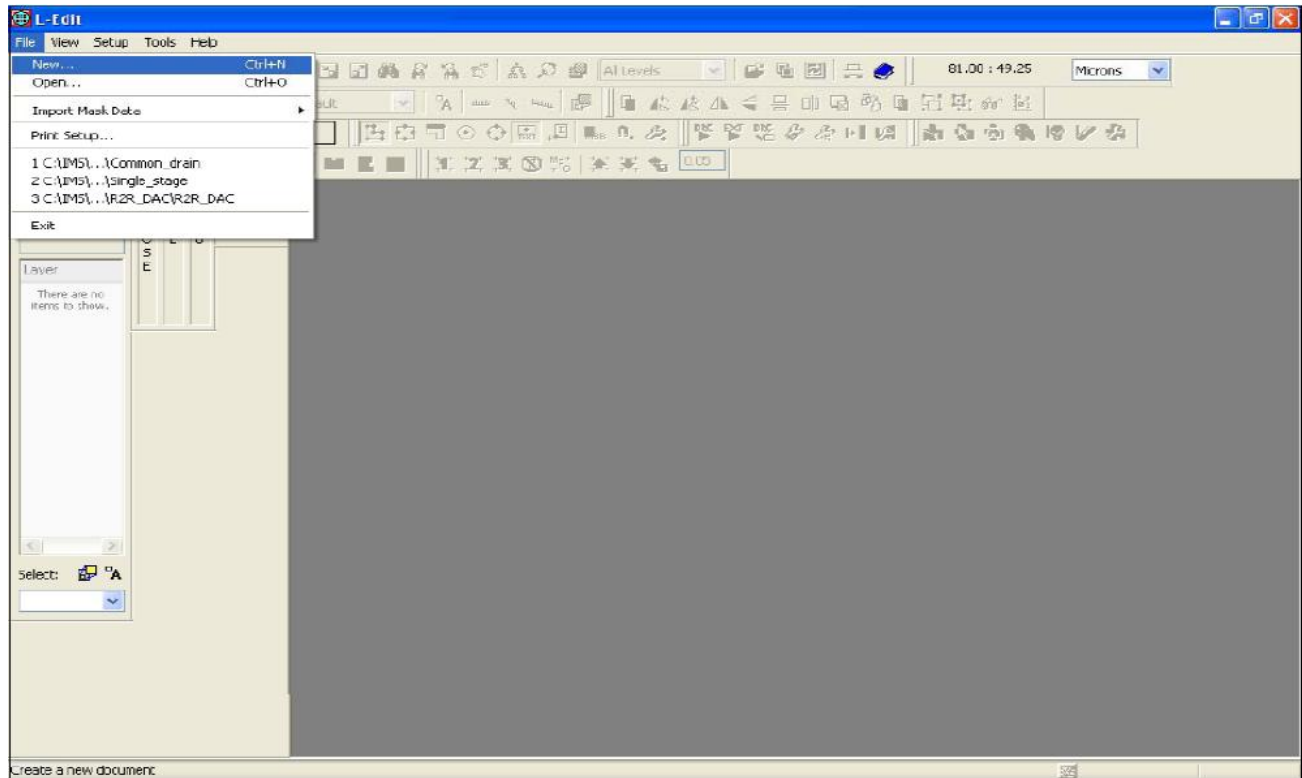


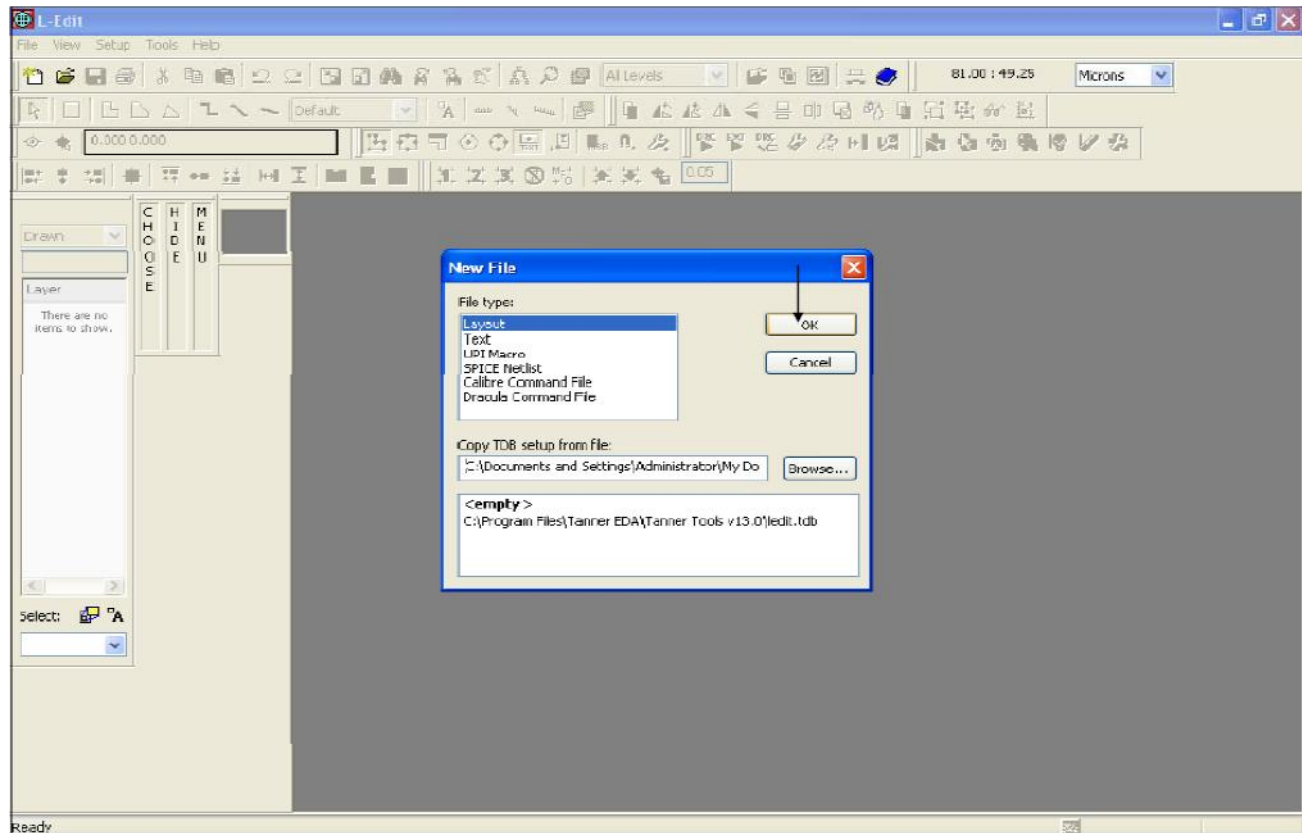
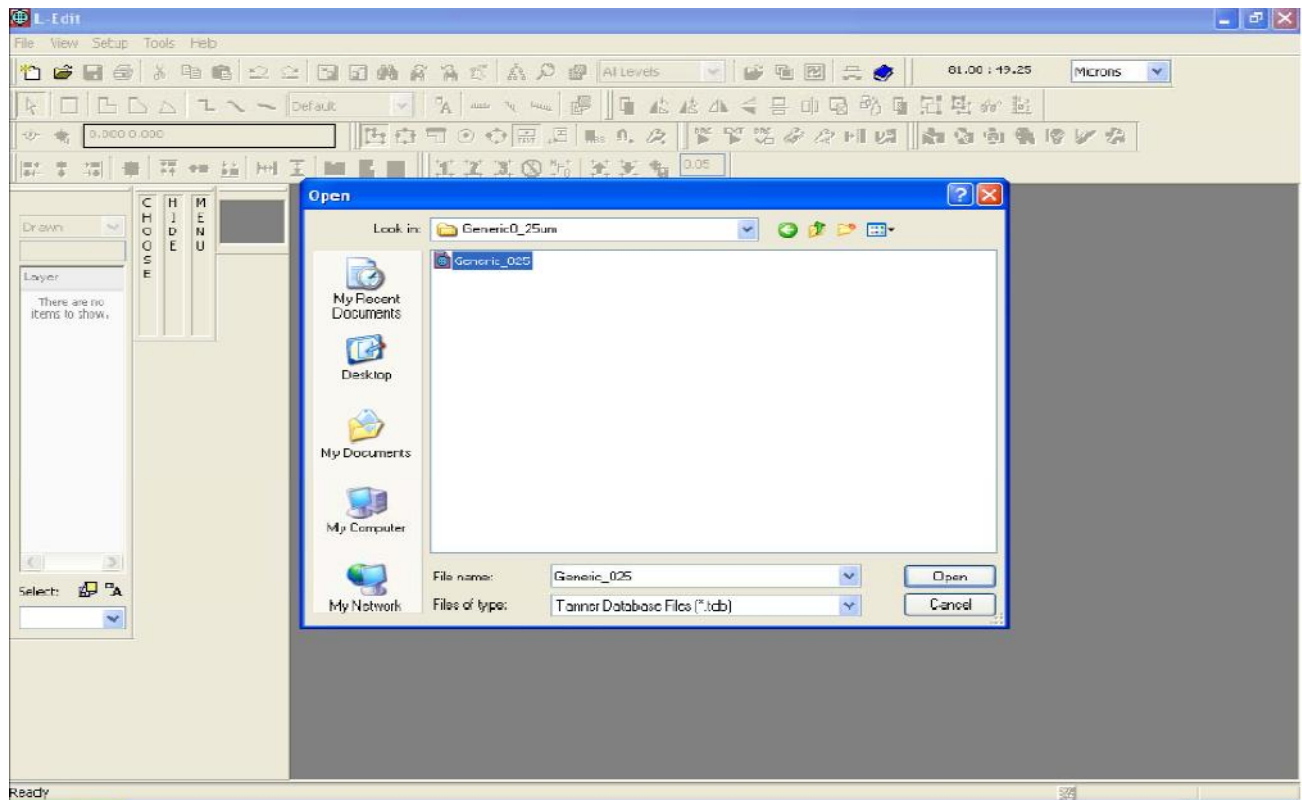


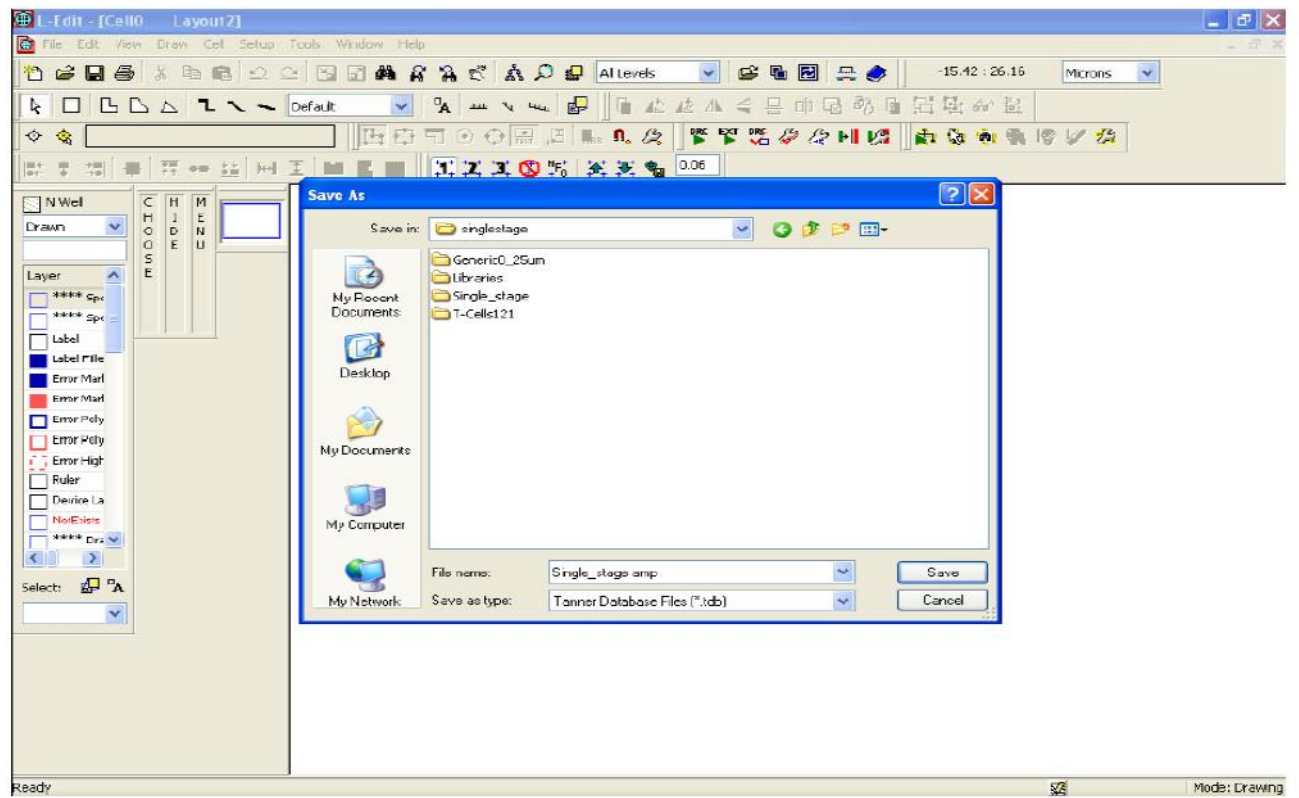
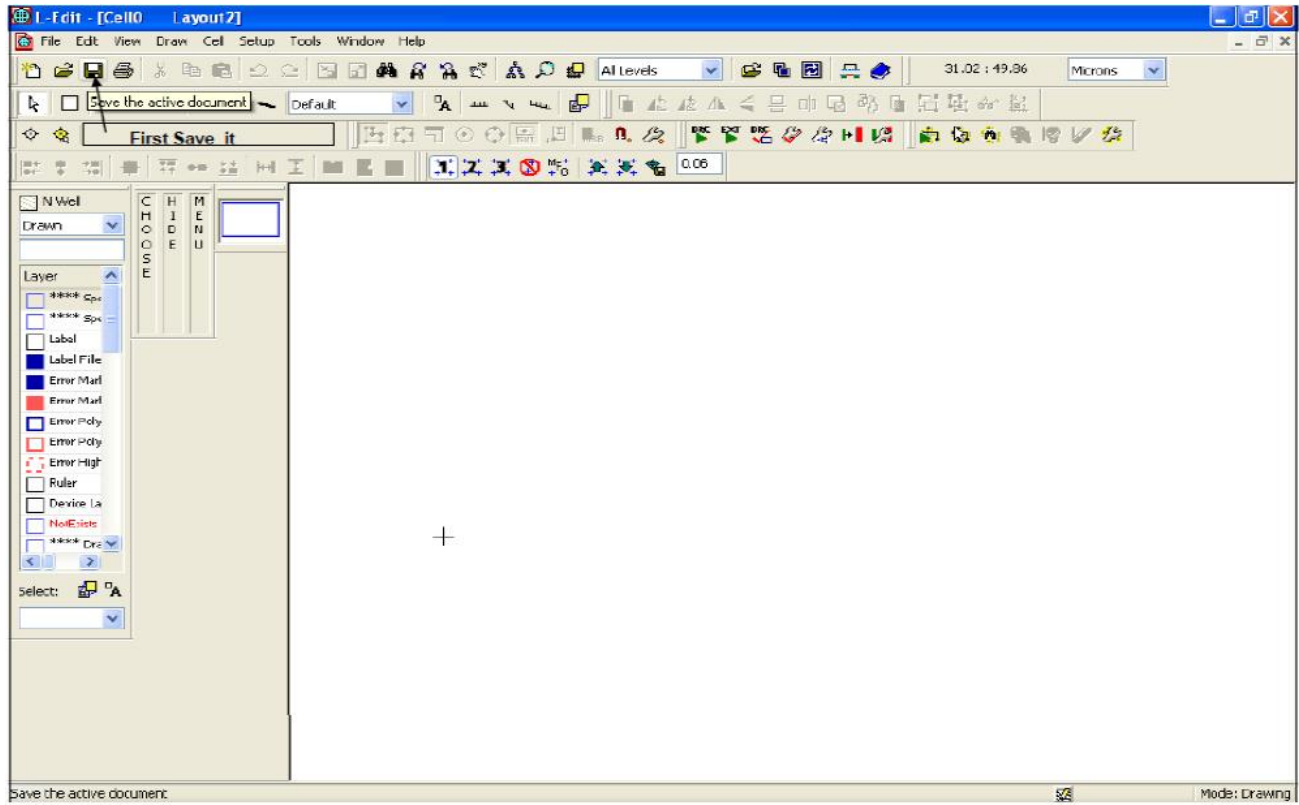
ii) Layout (L-edit):

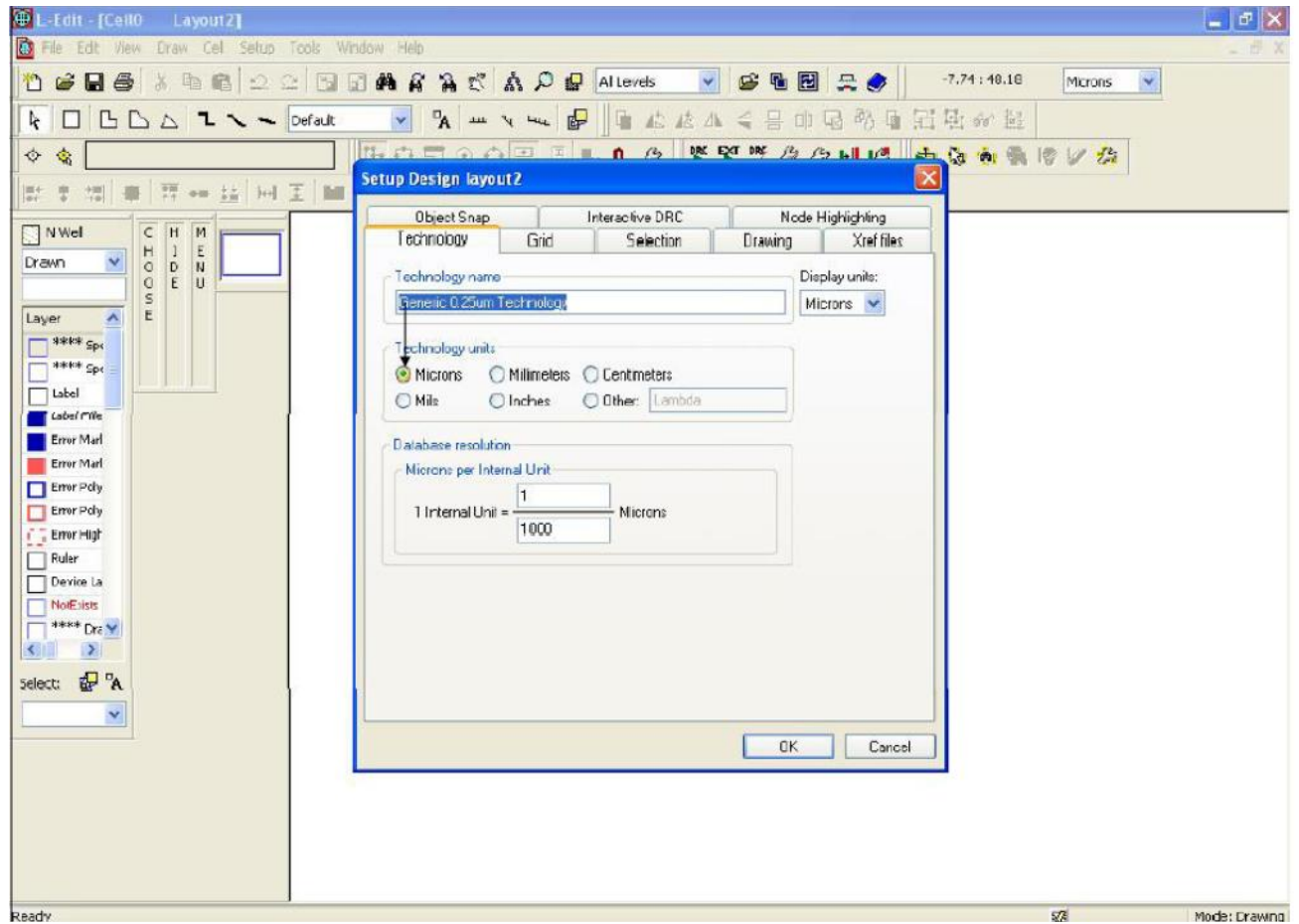
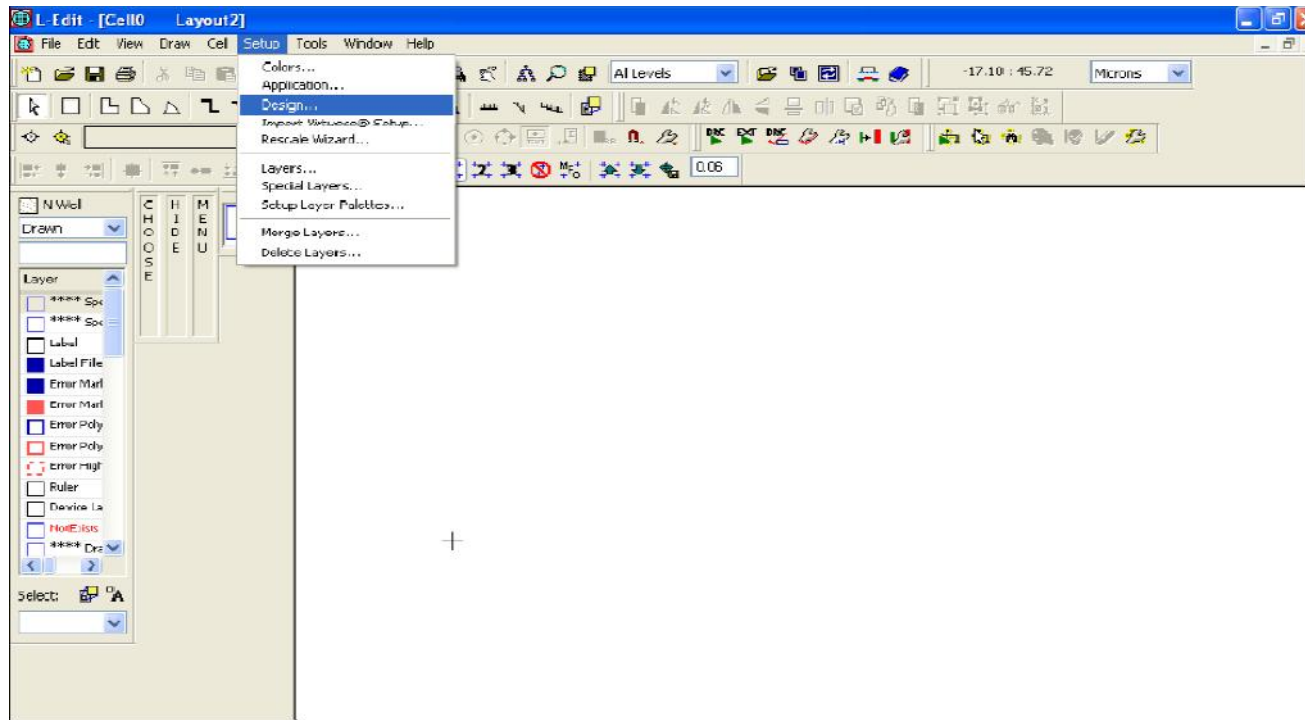
Start → Programs → tanner EDA → tanner tool v13.0 → L-edit

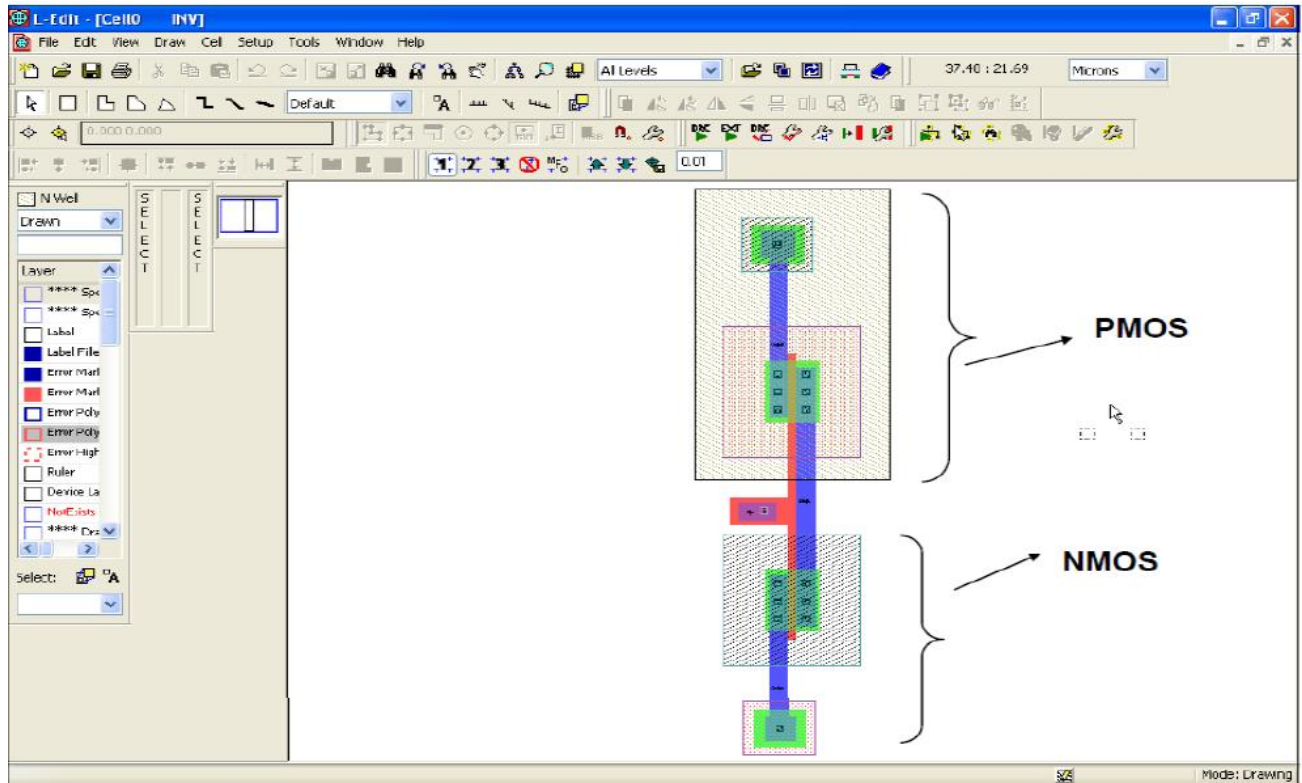
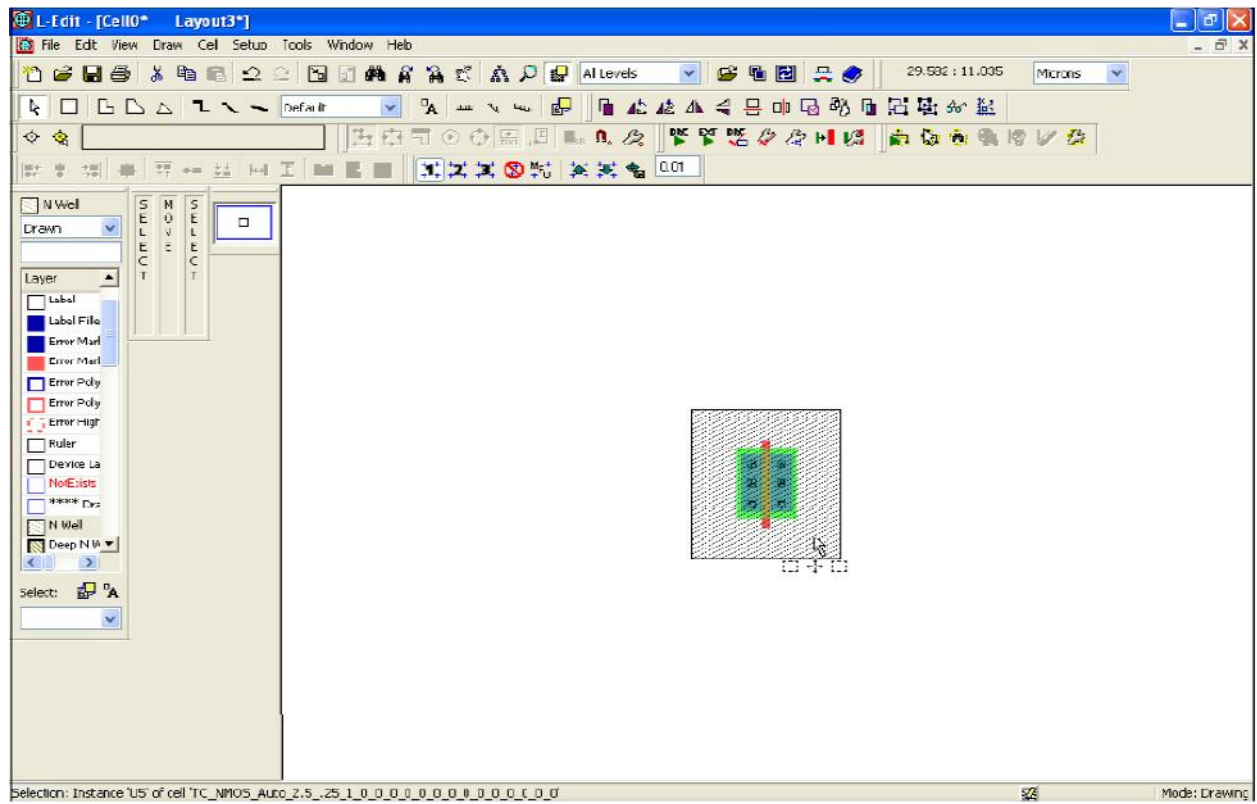


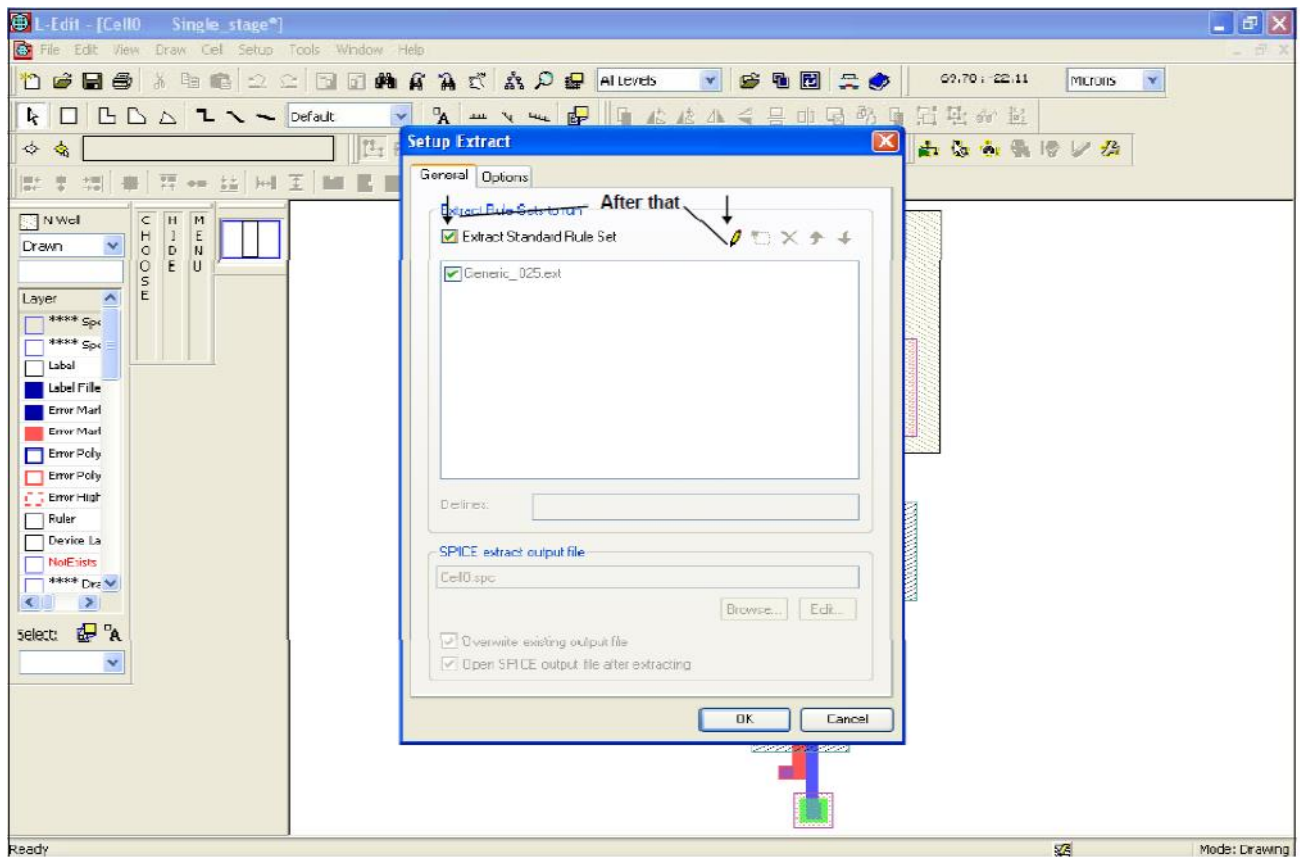
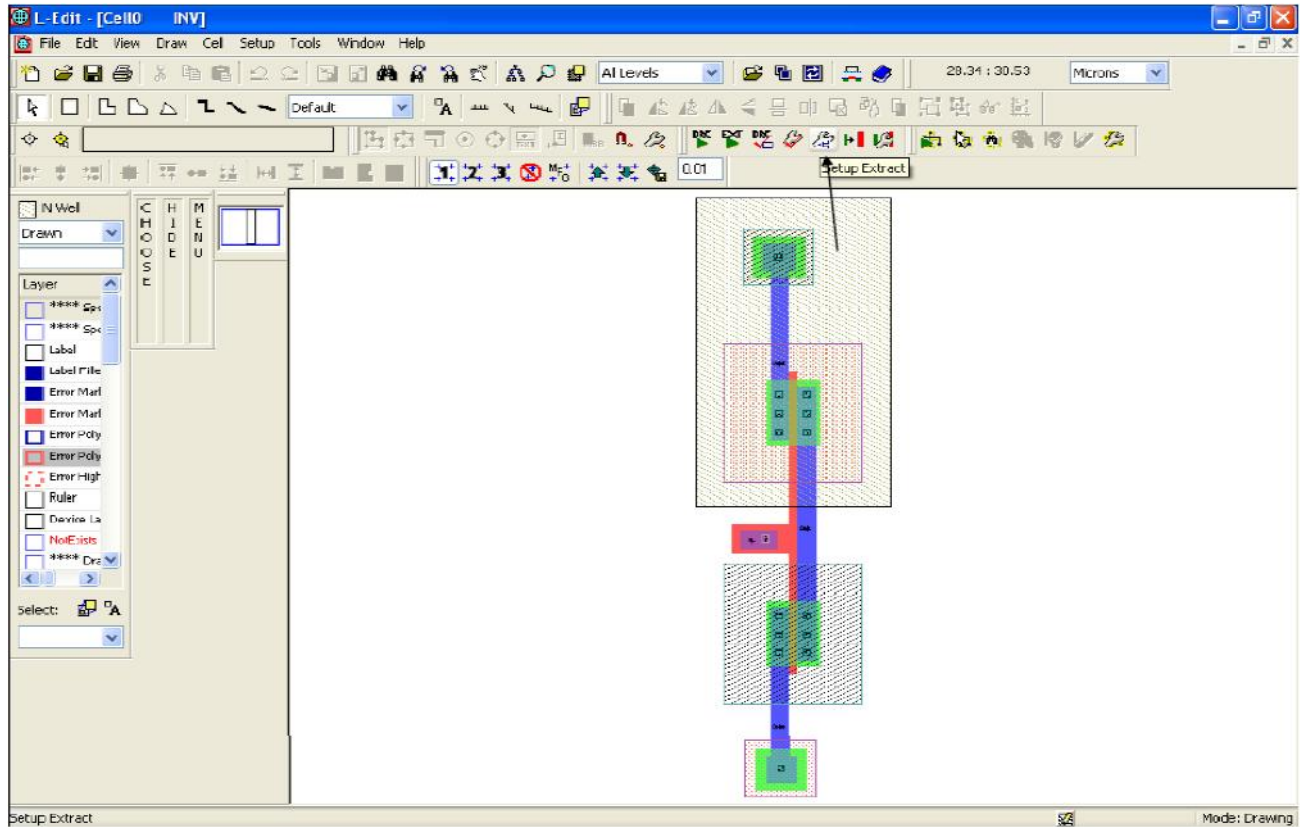


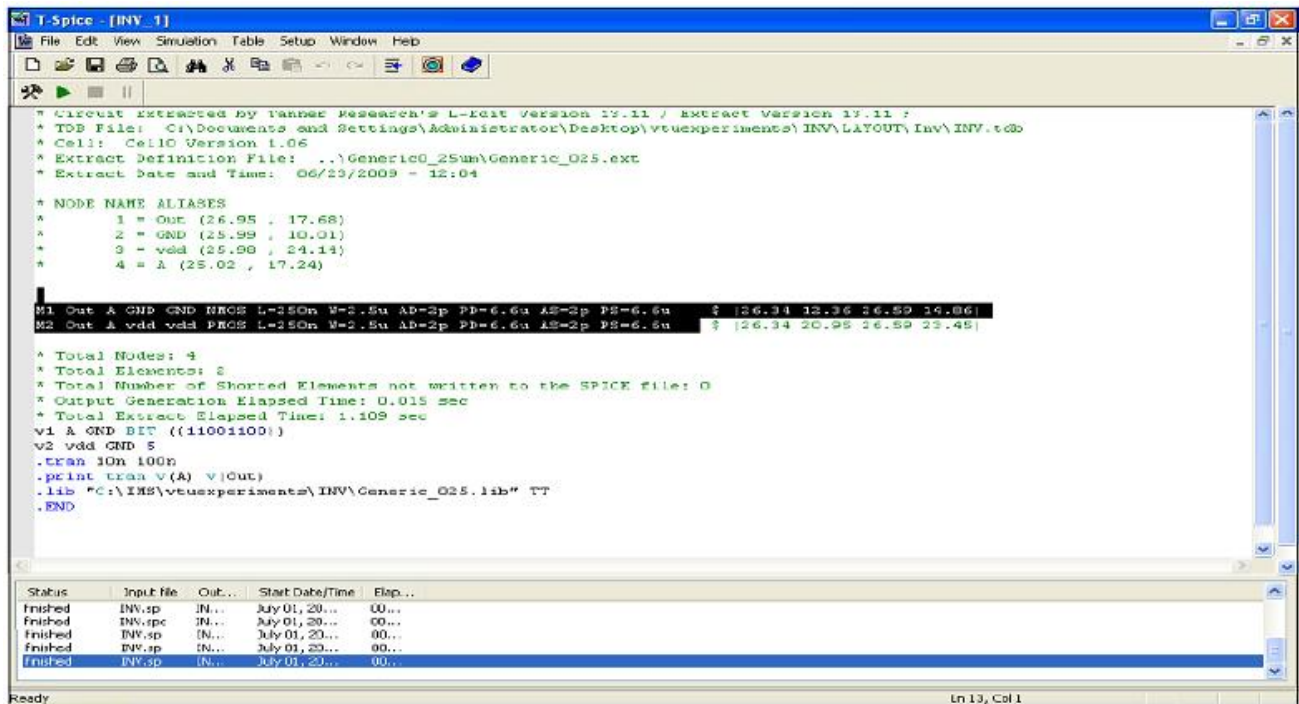
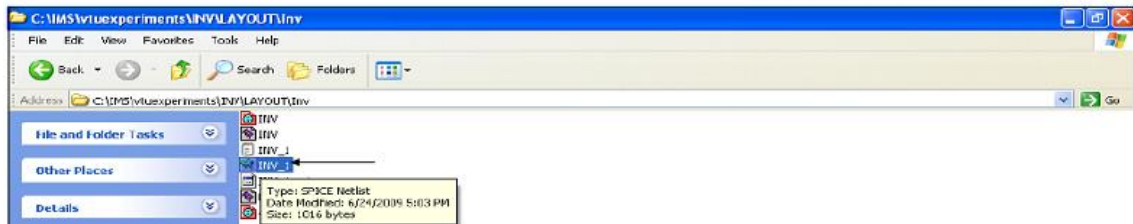
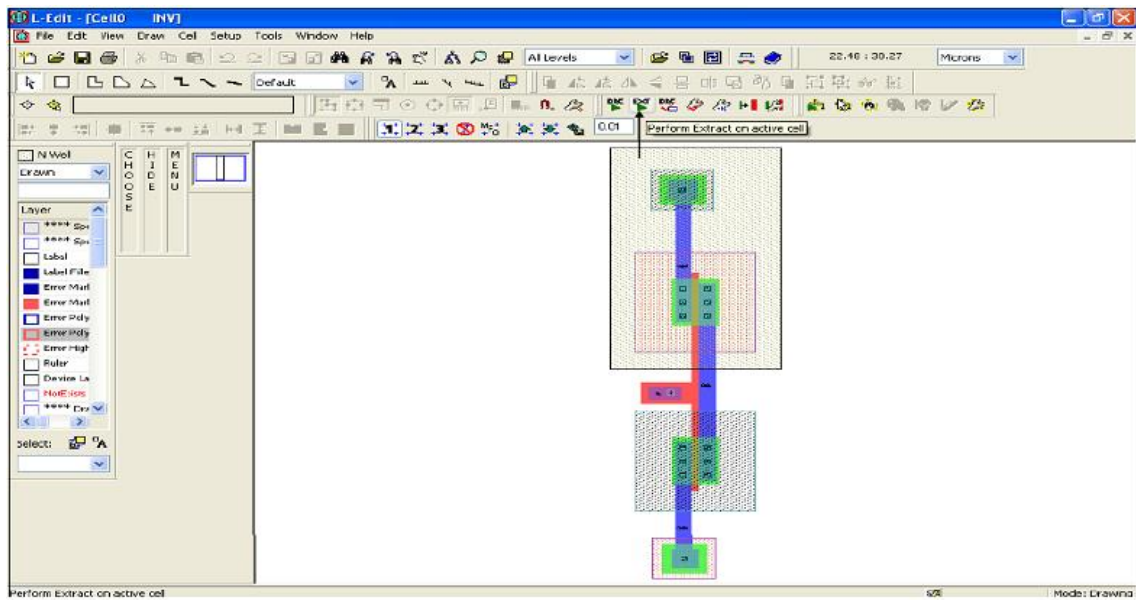


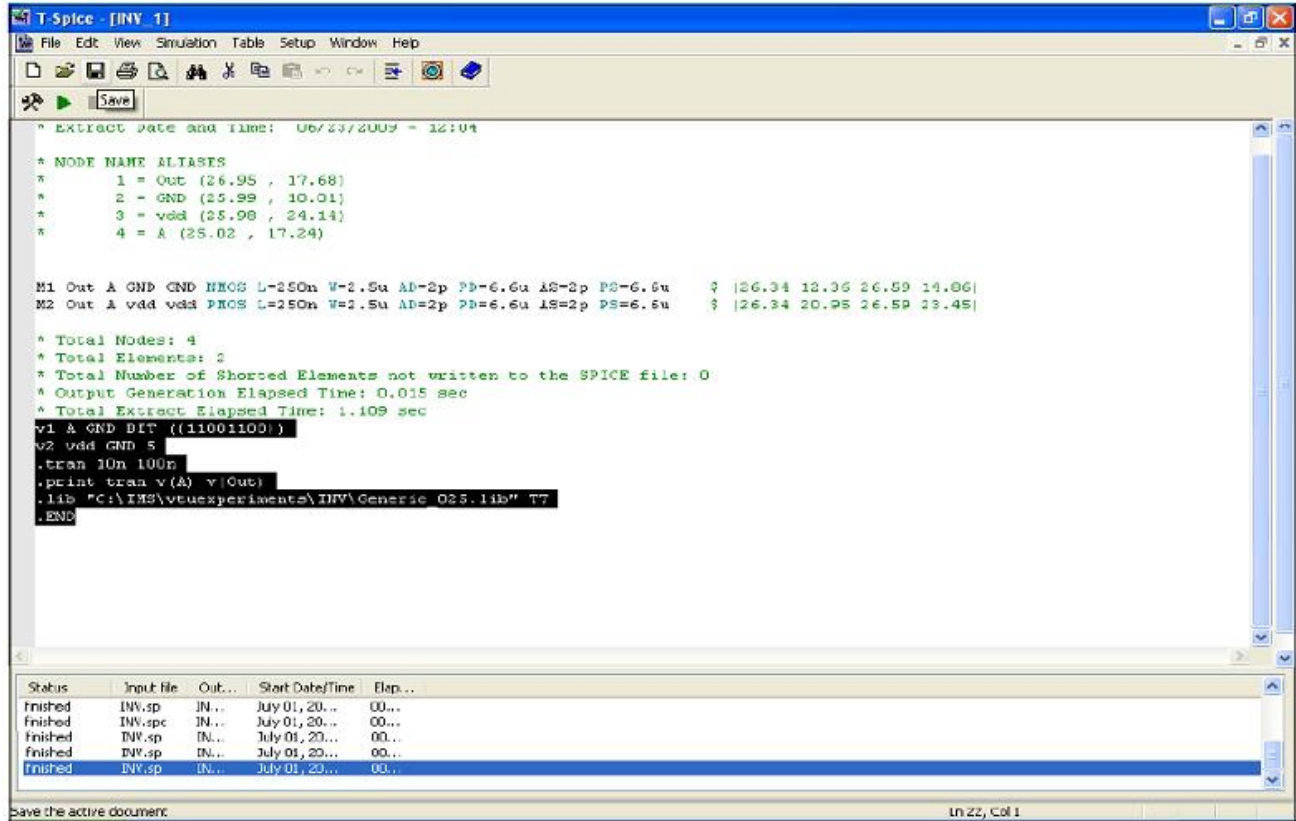


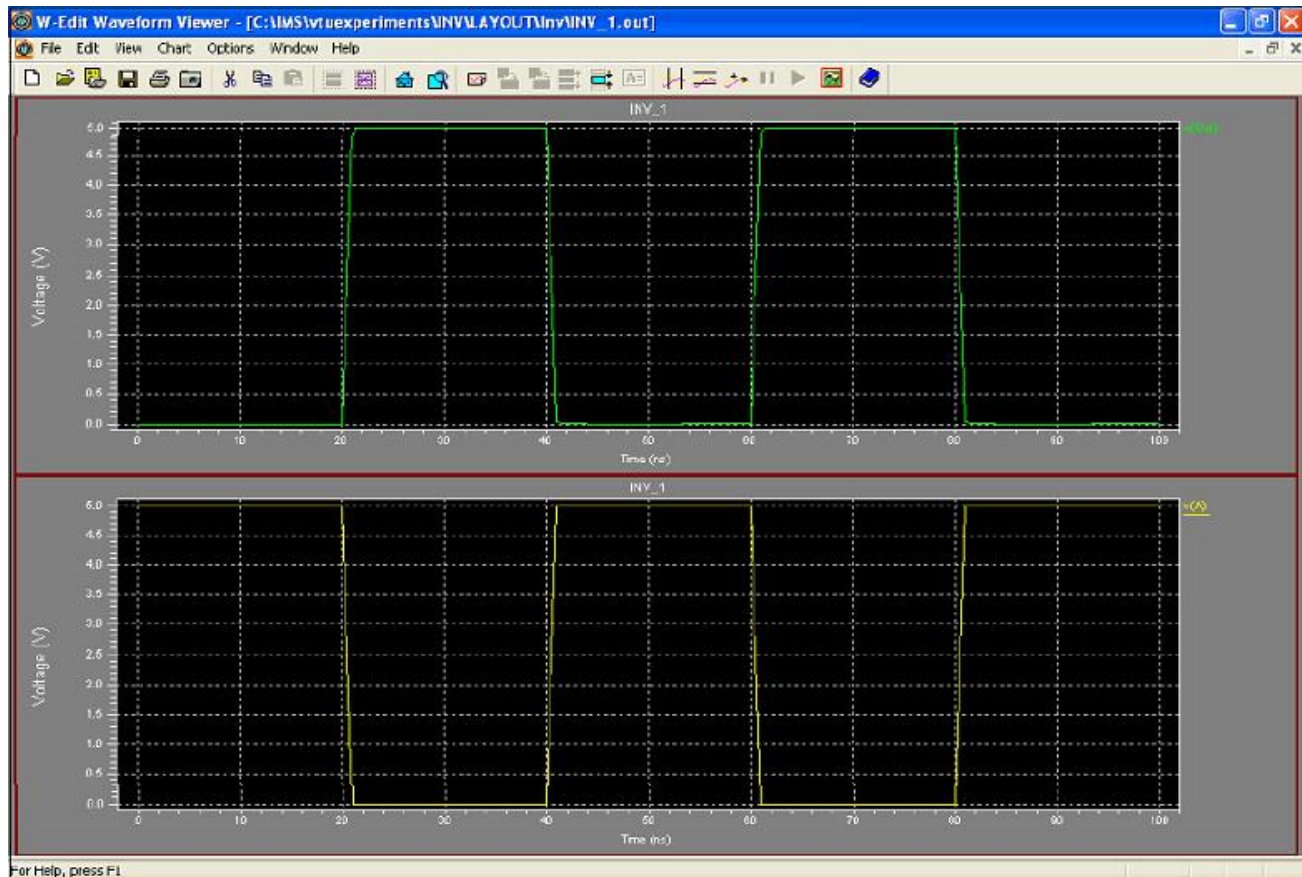
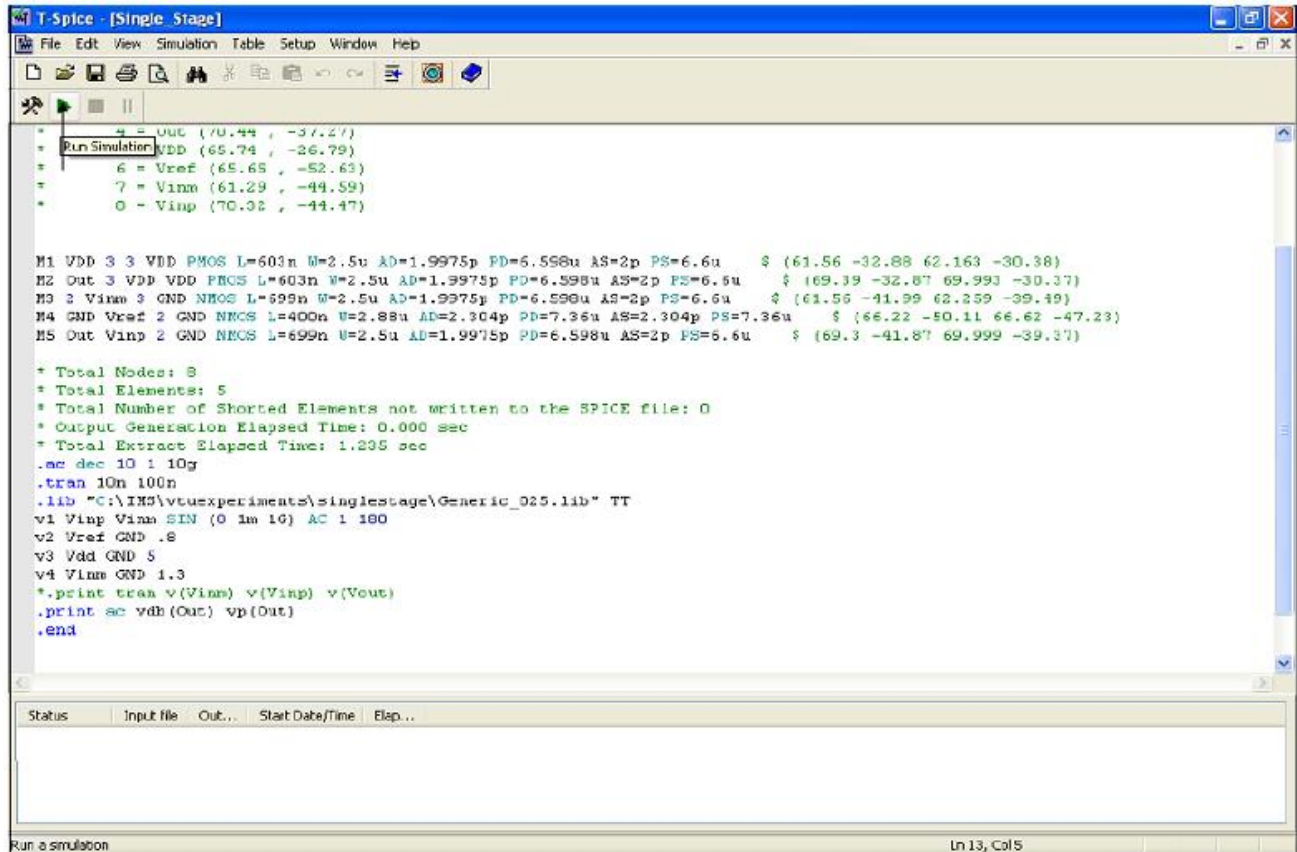


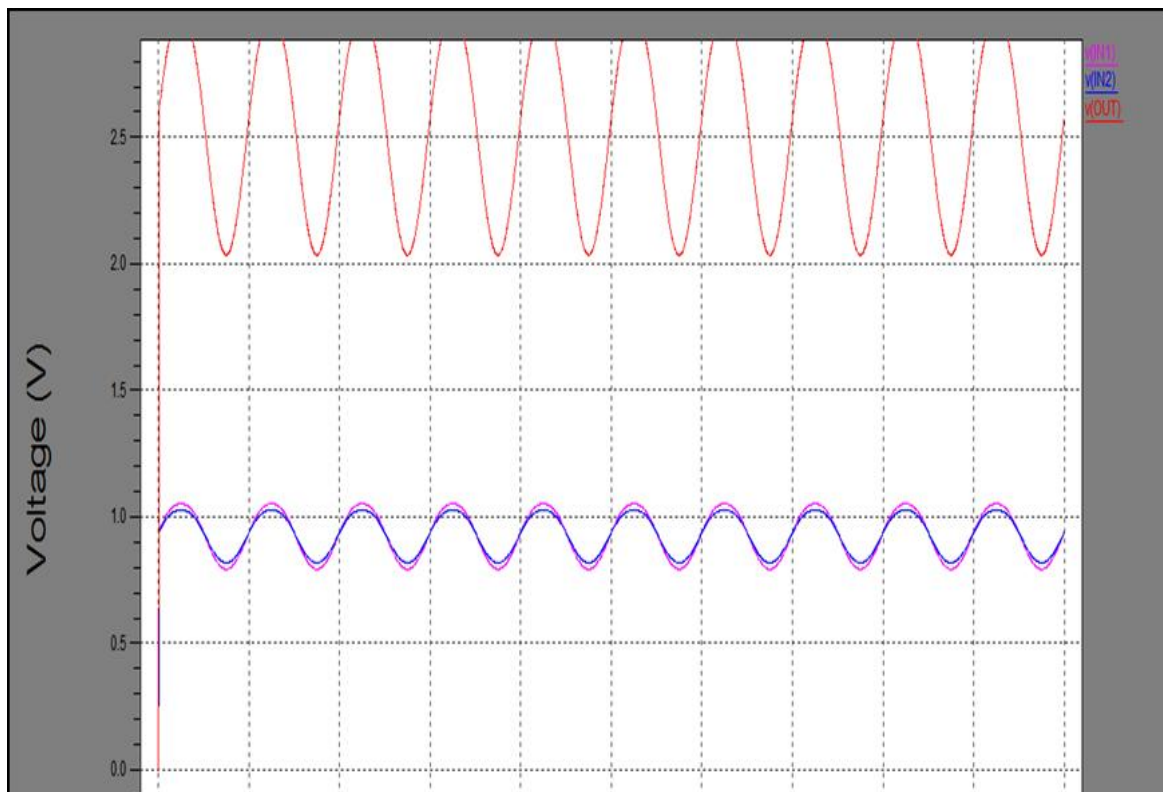
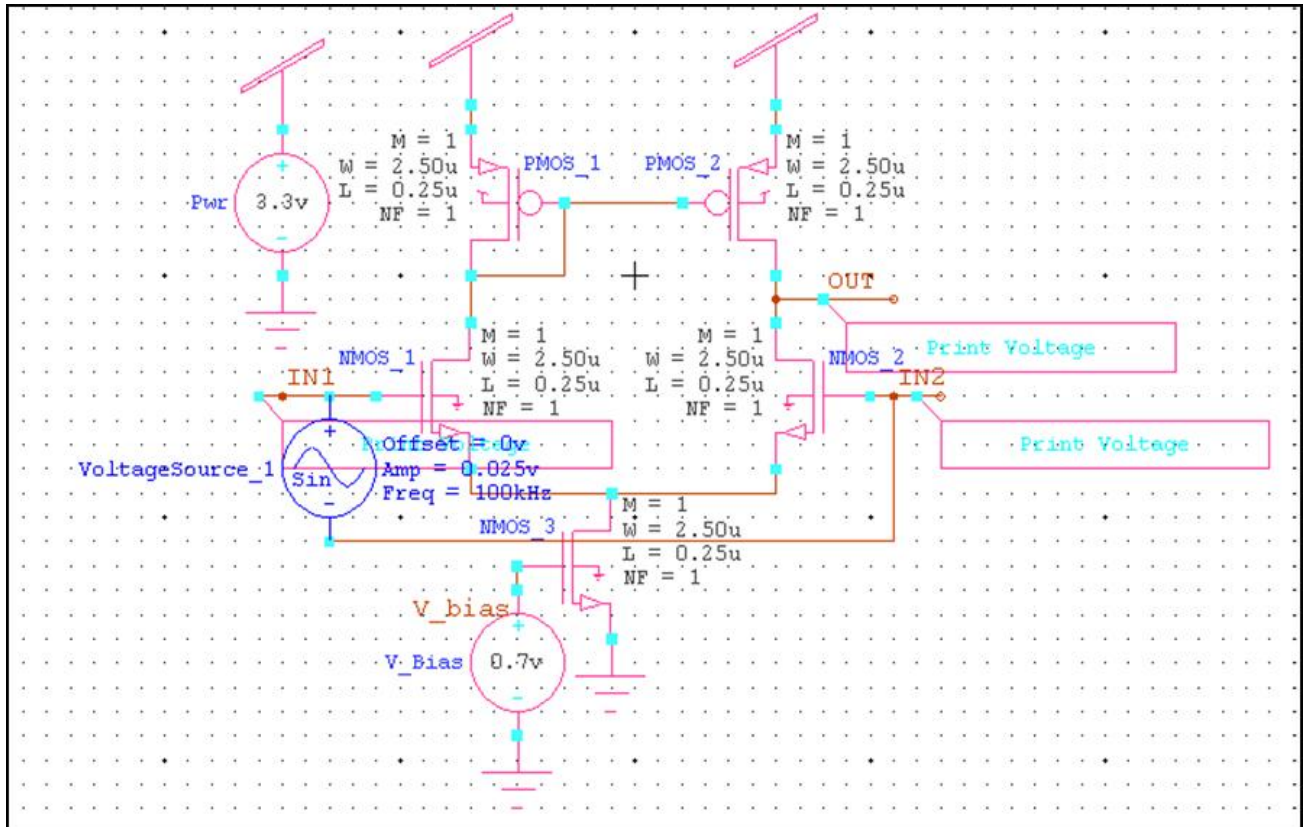












EXP.NO: 11	DIFFERENTIAL AMPLIFIER
DATE:	

AIM:

To calculate the gain, bandwidth and CMRR of a differential amplifier through schematic entry.

FACILITIES REQUIRED:

1. S-Edit using Tanner Tool.

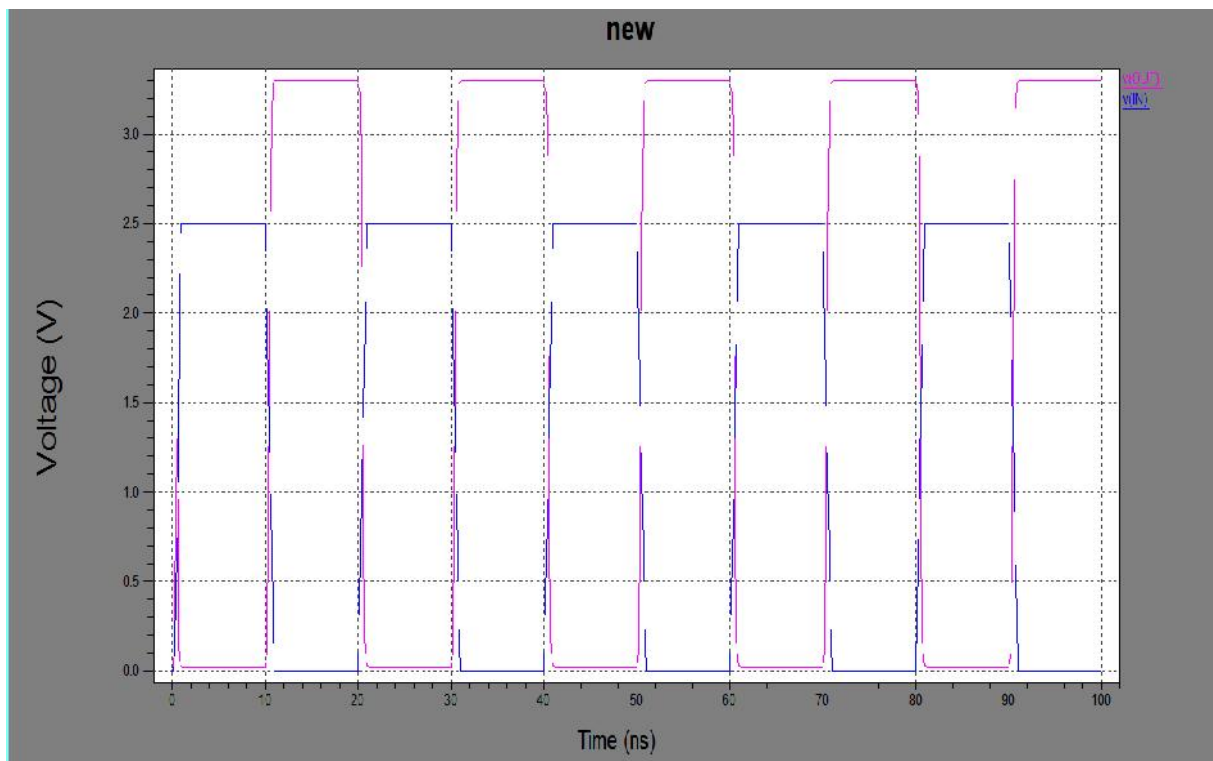
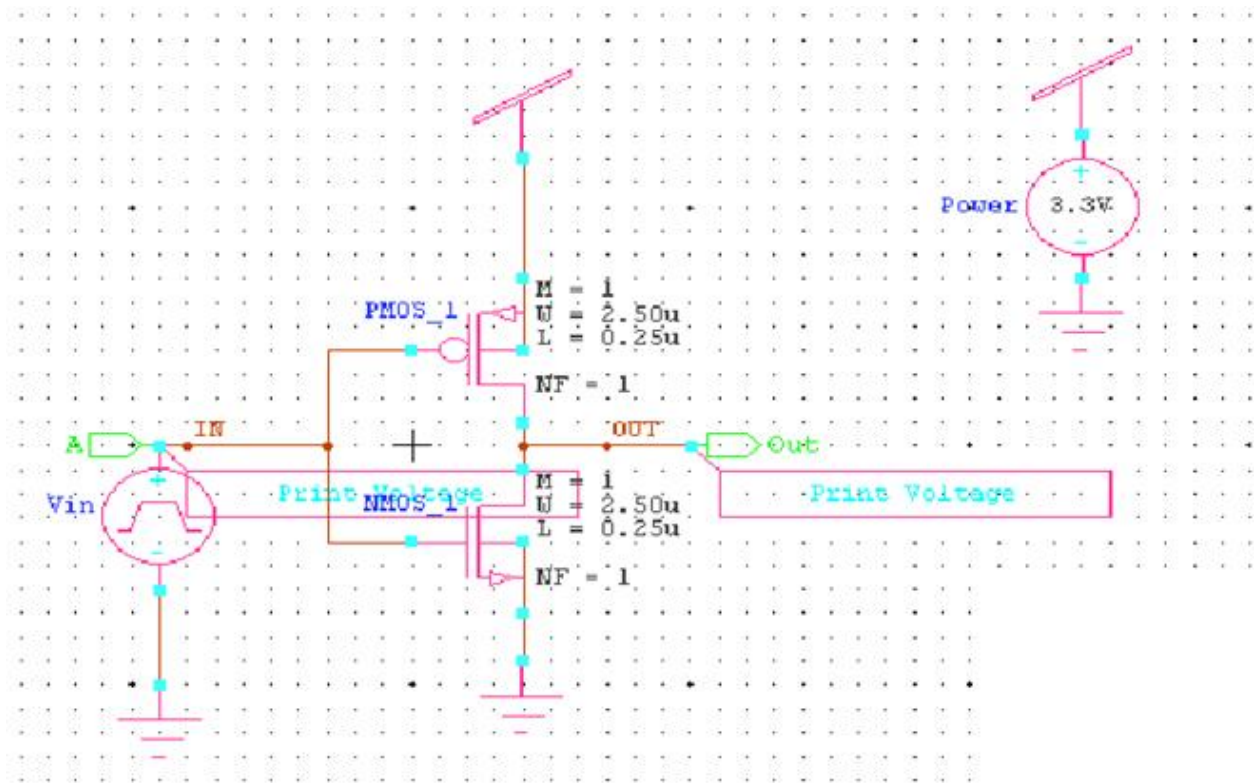
PROCEDURE

1. Draw the schematic of differential amplifier using S-edit and generate the Symbol.
2. Draw the schematic of differential amplifier circuit using the generated Symbol.
3. Perform AC Analysis of the differential amplifier.
4. Obtain the frequency response from W-edit.
5. Obtain the spice code using T-edit.

RESULT

Thus the differential amplifier bandwidth and CMRR are calculated was verified through schematic entry.

SCHEMATIC DIAGRAM: (DIFFERENTIAL AMPLIFIER)



EXP.NO: 12	CMOS INVERTOR
DATE:	

AIM:

To perform the functional verification of the CMOS Inverter through schematic entry.

FACILITIES REQUIRED:

1. S-Edit using Tanner Tool.

PROCEDURE:

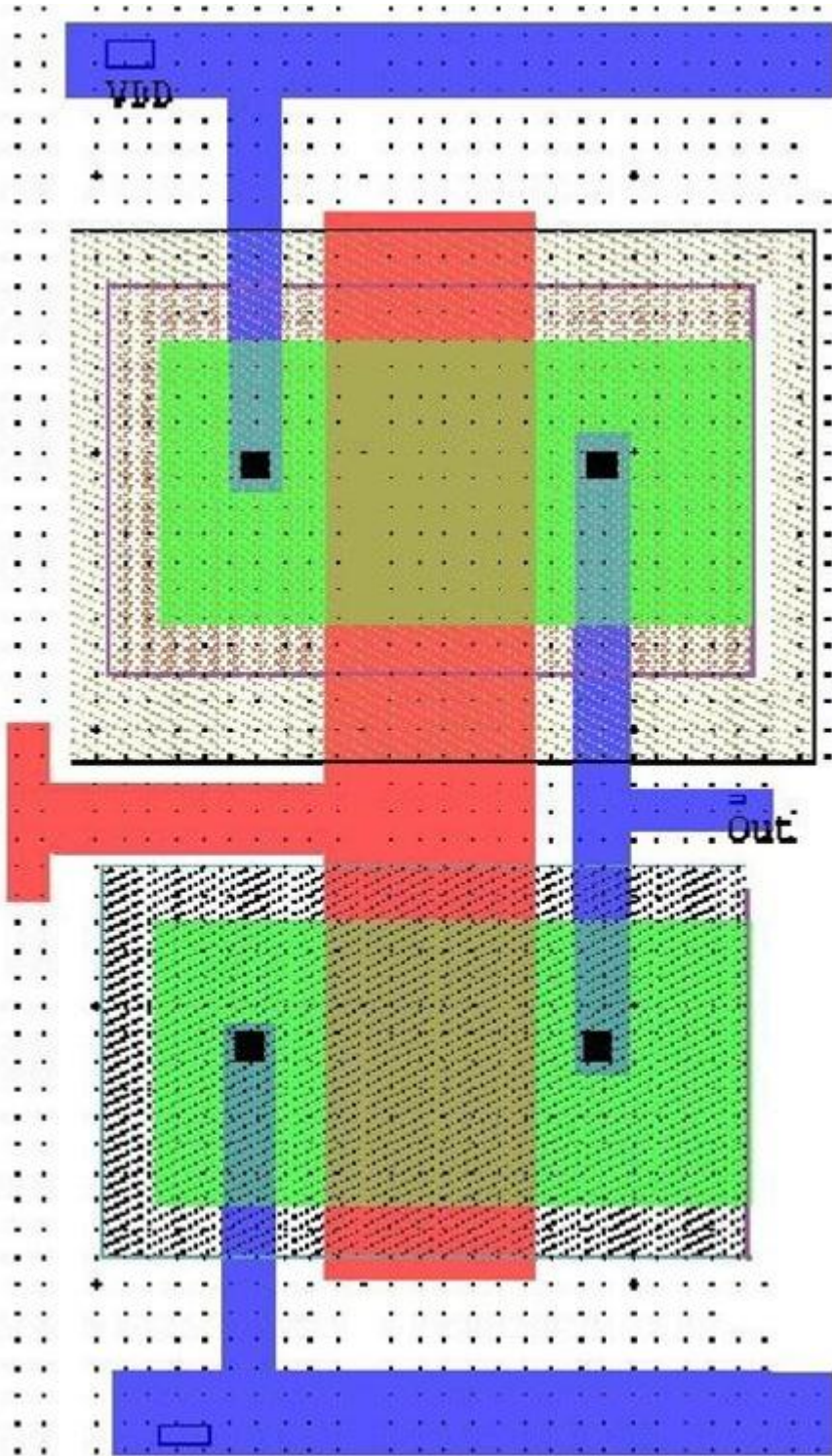
1. Draw the schematic of CMOS Inverter using S-edit
2. Perform Transient Analysis of the CMOS Inverter
3. Obtain the output wave form from W-edit
4. Obtain the spice code using T-edit

THEORY:

Inverter consists of nMOS and pMOS transistor in series connected between VDD and GND. The gate of the two transistors are shorted and connected to the input. When the input to the inverter $A = 0$, nMOS transistor is OFF and pMOS transistor is ON. The output is pull-up to VDD. When the input $A = 1$, nMOS transistor is ON and pMOS transistor is OFF. The Output is Pull-down to GND.

RESULT

Thus the functional verification of the CMOS Inverter was verified through schematic entry.



EXP.NO: 13	LAYOUT CMOS INVERTOR
DATE:	

AIM:

To draw the layout of CMOS Inverter using L-Edit and extract the SPICE code.

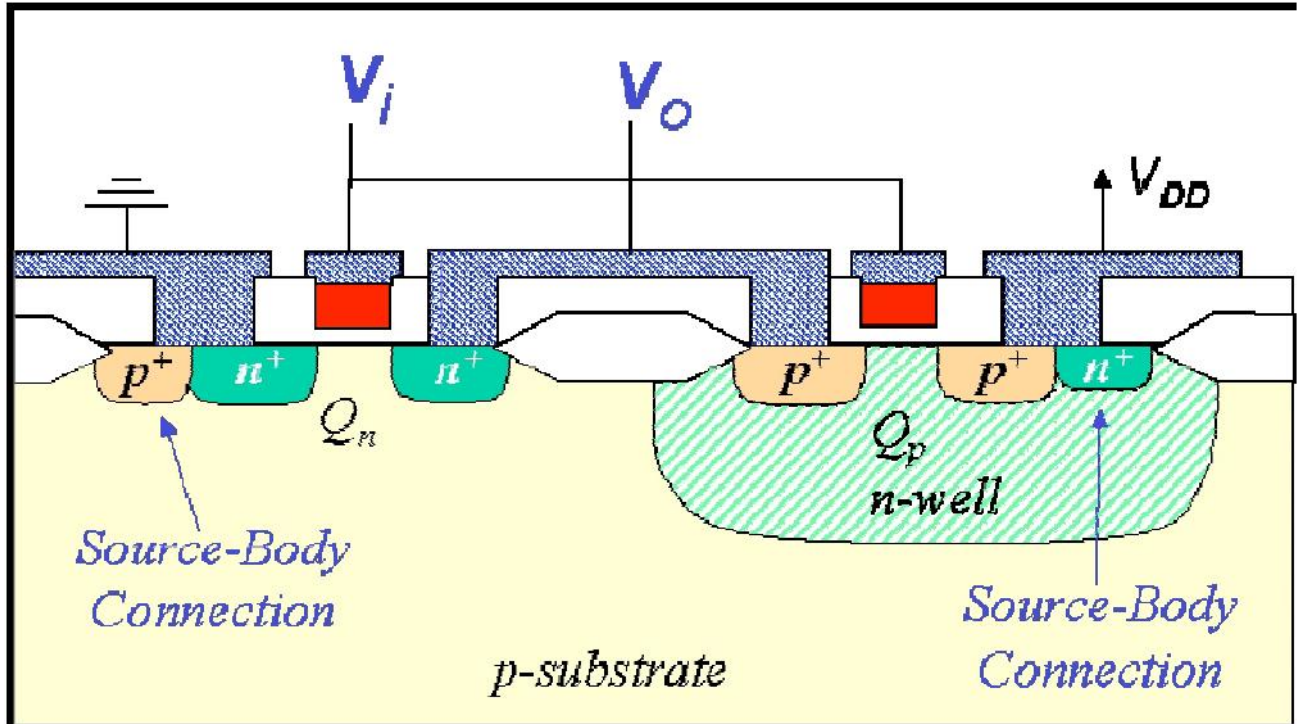
FACILITIES REQUIRED :

1. L-Edit using Tanner Tool.

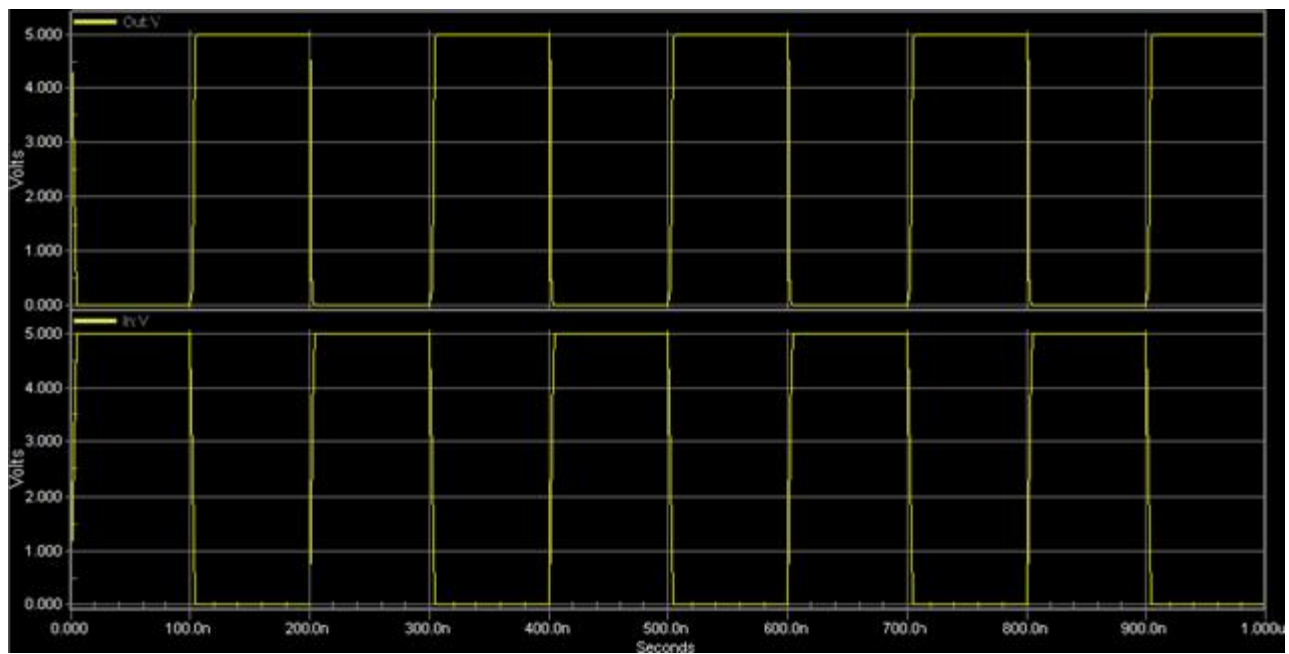
PROCEDURE:

1. Draw the CMOS Inverter layout by obeying the Lamda Rules using L-edit.
2.

i. Poly	-2
ii.Activecontact	-2
iii.ActiveContact – Metal	-1
iv.ActiveContact – Active region	-2
v.ActiveRegion – Pselect	-3
vi.Pselect – nWell	-3
3. Check DRC to verify whether any region violate the Lamda rule
4. Setup the extraction and extract the spice code using T-spice.

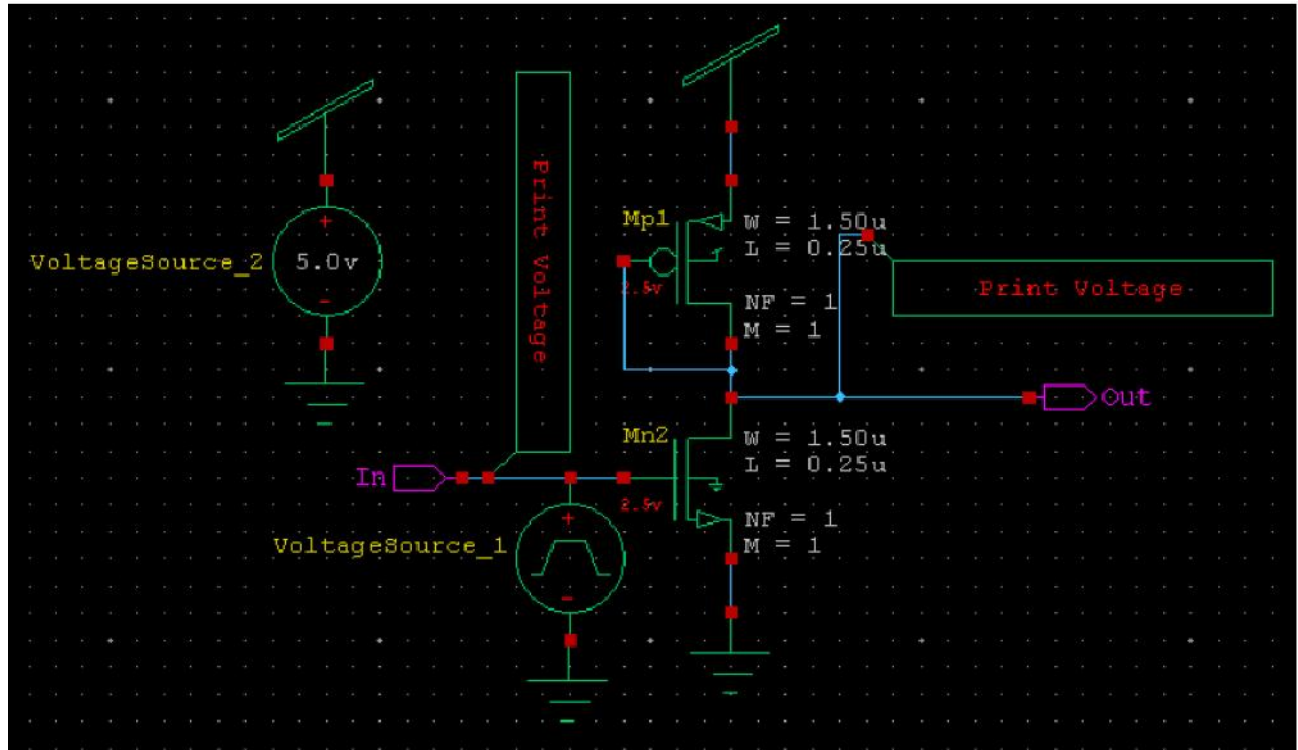


SIMULATED WAVEFORM:



RESULT:

Thus the layout of CMOS Inverter was verified through L-edit.



EXP.NO: 14	AUTOMATIC LAYOUT GENERATION
DATE:	

AIM:

To generate the automatic Layout from the schematic using the Tanner tool and verify the layout using simulation.

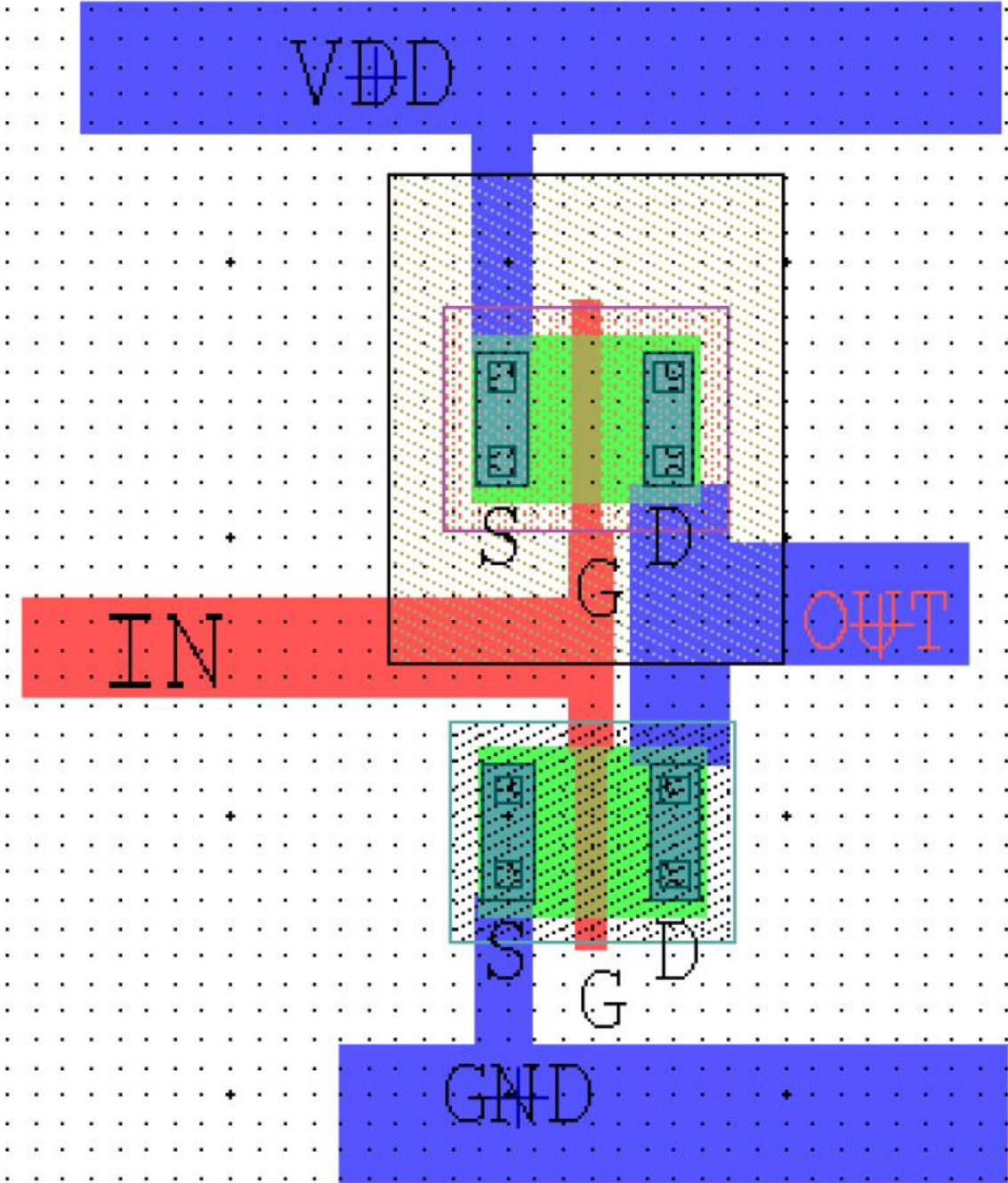
FACILITIES REQUIRED

1. S-Edit and L-Edit using Tanner Tool

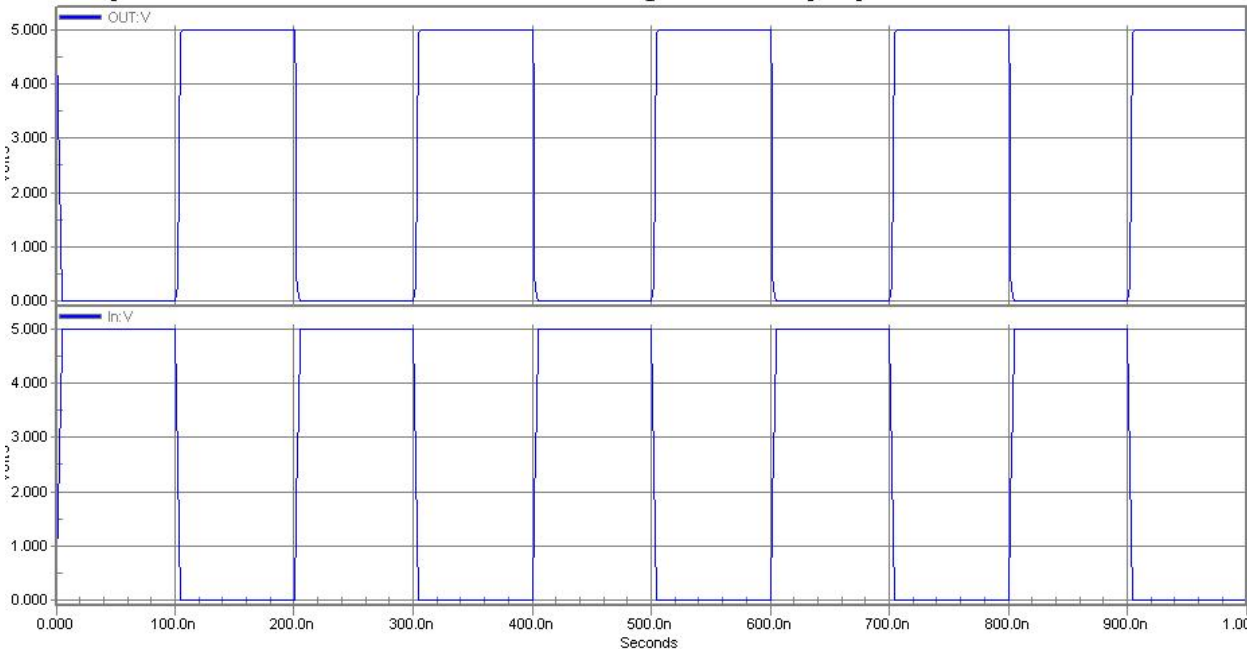
PROCEDURE

1. Draw the schematic using S -Edit and verify the output in W-Edit.
2. Extract the schematic and store it in another location
3. Open the L-Edit, open the design in Ring VCO
4. Create the new cell
5. Open the schematic file(.sdl) using the SDL Navigator
6. Do the necessary connections as per the design.
7. Name the ports and check the design for the DRC Rules
8. Locate the Destination file in the setup Extract window and extract the layout.
9. Include the Library and the print voltage statements in the net list which is obtained.
10. Verify the layout design using W-Edit.

LAYOUT GENERATION:



SIMULATED WAVEFORM:



RESULT:

Thus the automatic Layout was verified through L-edit.