

Dharmapuri – 636 703

1

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.

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

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

VVIT

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

<u>AIM:</u>

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 \rightarrow Programs \rightarrow Xilinx ISE \rightarrow Project Navigator.

INABEL WARE VIEW	Accessories	•					
	m Games	•					
	JTAG5.0	۲	-				
admin-pc	matlab	۲				1995	-
	microsoft Office	٠				100	1.1
	MiPower Ver 6.0	•	-				
Mozila Firefox	🛅 Startup						
- E-mail	🛅 WinPcap		nts	*			
Microsoft Office Out	im WinRAR			-	-		
0	💼 Xilinx ISE Design Suite 12.1	Þ		Accessories	•		
Internet Explorer	💹 Adobe Reader 8		0	ChipScope Pro			
A dela Provide of	🥭 Internet Explorer		0	Documentation	•		
Adobe Reader B	🗐 Outlook Express		0	EDK			
Microsoft Office Wor	🔔 Remote Assistance		-	ISE Design Tools		Cocumentation	
	🕑 Windows Media Player		1	PlanAhead		m Tools +	
5-Edt 13.0	Windows Movie Maker		0	System Generator	••	Project Navigator	and a state of the
	m Shield			Contraction of the			
L-Edit v13.0	Tanner EDA					Project Navigator	
	Adobe Photoshop 7.0						
Calculator	🔞 Adobe ImageReady 7.0			and in the second		Contract of the local division of the	
	Google Chrome	,		Statistics.			
All Programs 🌔	🧶 Mozilla Firefox			Color Color		Service - The Party of the Part	
	-	_		through the			
	Log Off 🔟 Turn Off						

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

🖾 New Project Wiza	rd 🛛 🔀
Create New Project Specify project loc	ation and type.
Enter a name, locati	ons, and comment for the project
Name:	
Location:	D:\balalab
Working Directory:	D:\balalab
Description:	
Select the type of to	p-level source for the project e:
HDL	
More Info	Next > Cancel

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

elect the device and design flow for the	project	
Property Name	Value	
Product Category	AI	V
Family	SpartanGE	~
Device	XC35500E	~
Package	FT255	~
Speed	-4	¥
Too-Level Source Type	HDL	~
Synthesis Tool	XST (VHDL/Verilog)	~
Sinulator	ISim (VHDL/Verilog)	~
Preferred Language	Verilog	~
Property Specification in Projec: File	Store all values	~
Manual Compile Order		
VHDL Source Analysis Stancard	VHDL-93	~
Enable Message Filtering		

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

icore_dir
je

> assign inputs and outputs \rightarrow click next \rightarrow finish \rightarrow yes \rightarrow next \rightarrow next \rightarrow finish

Define Module Specify ports for module.						
odule name add						
Port Name	Direction		Bus	MSB	LSB	10
1	input	~				
0	input	~				
	output	~				
	input	~				
	input	~				
	input	~				
	input	~				
	input	~				
	input	~				
	input	~				
	input	~				
	input	~				123
	Indexe.	0000				

- \blacktriangleright Double click on source file \rightarrow 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 \rightarrow Click on new source \rightarrow Click on verilog text fixture \rightarrow Give file name with _tb \rightarrow finish

> Generate test bench file after initial begin assign value for inputs \rightarrow Click on simulate behavioral model \rightarrow 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 synthesize 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 ROOT 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 synthesize 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 produce by the synthesize compiler into a netlist of FPGA primitive hardware components. Each elements of this new netlist corresponds to a hardware primitive in the FPGA Chip. The mapped design produces identifies the set of FPGA hardware primitives and their interconnection. The next step is to assign each of the components in the netlist to a equivalent physical primitives 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 inter connect resources available on 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 \rightarrow user constraints \rightarrow I/O pin planning (plan ahead) pre- ynthesis, 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	
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.

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.

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 top an inverter, the low level will appear at the output and vice versa.

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.

$$\mathbf{Y} = \mathbf{a} \, \mathbf{\&} \, \mathbf{b}$$

VVIT DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

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.

$$\mathbf{Y} = \mathbf{a} \mid \mathbf{b}$$

EX-OR GATE:

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

$$\mathbf{Y} = \mathbf{a} \wedge \mathbf{b}$$

EX-NOR GATE:

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

 $\mathbf{Y} = \mathbf{\sim}(\mathbf{a} \ \mathbf{^b})$

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 hal(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,

> sum = x ^ y carry = x & y

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.

sum = x ^ y ^ z carry= (x & y) | (y & z) | (x & z)

PROCEDURE:

1. Click on the Xilinx ISE Design Suite 14.1or 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.

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

<u>8-BIT RIPPLE CARRY ADDER</u>

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

<u>4-BIT RIPPLE CARRY ADDER</u>

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	
DATE:	SIMULATION OF ADDER

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 comwter 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 acd 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 usee (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 14.1or 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 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

<u>BOOTHS MULTIPLIER (4-BIT</u> 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'b00000000;temp=10'b00000000; for(i=0;i<=4;i=i+1)</pre> 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'b00 01; 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;</pre> z=z+temp;end za=z; end endmodule

EXP.NO: 4 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 14.1or 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.

EC6612 VLSI DESIGN LAB PROGRAM:

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

DATE:

SIMULATION OF BINARY TO GRAY CODE COUNTER

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:

- Click on the Xilinx ISE Design Suite 14.1or 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 for 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 $Q \leq Q;$ 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;</pre> 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 $Q \leq Q;$ 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 $Q \ll 1'bx;$ end assign Qbar = $\sim Q;$ endmodule

EXP.NO: 6	
DATE:	SIMULATION OF FLIP FLOPS

<u>AIM:</u>

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</pre>

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 \ll Q;
end
else if (t == 1)
begin
Q <= ~Q;
end
endmodule
```

PROCEDURE:

- 1. Click on the Xilinx ISE Design Suite 14.1or 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=\sim 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 $q=\sim q$; 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	
DATE:	SIMULATION OF PSEUDO RANDOM BINARY SEQUENCE

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 14.1or 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 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 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 14.1or 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.

VVIT DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

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	
DATE:	SIMULATION OF COUNTER

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:

- Click on the Xilinx ISE Design Suite 14.1or 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;</pre> y<=0; end else begin case (state) 3'b000:begin if (x) begin state<=3'b001;</pre> y<=0; end else begin state<=3'b000;</pre> y<=0; end end 3'b001:begin if (x) begin state<=3'b010;</pre> y<=0; end else begin state<=3'b000;</pre> y<=0; end end

EXP.NO: 10	
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 s1and attains the value 1or else it stays in the same state s0.

EC6612 VLSI DESIGN LAB 3'b010:begin if (x) begin state<=3'b010;</pre> y<=0; end else begin state<=3'b011;</pre> y<=0; end end 3'b011:begin if (x) begin state<=3'b100;</pre> y<=0; end else begin state<=3'b000;</pre> y<=0; end end 3'b100:begin if (x) begin state<=3'b000;</pre>

y<=0; end else begin state<=3'b101;</pre> y<=1; end end 3'b101:begin if (x) begin state<=3'b001;</pre> y<=0; end else begin state<=3'b000;</pre> y<=0; end end endcase end end endmodule

EC6612 VLSI DESIGN LAB <u>STATE DIAGRAM:</u>



STATE TABLE:

PRESENT	NEXT	STATE	OUTPUT (Y)		
SIAIE	X=0	X=1	X=0	X=1	
S 0	S 0	S1	0	0	
S1	S 0	S2	0	0	
S2	S 3	S2	0	0	
\$3	S 0	<u>\$4</u>	0	0	
84	85	80	1	0	
85	S 0	<mark>S1</mark>	0	0	

RESULT:

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

TANNER TOOLS

VVIT DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

Steps to use Tanner tool:

i.SCHEMATIC (S-edit):

Start the tanner EDA by using the desktop shortcut or by using the \rightarrow Start \rightarrow Programs \rightarrow tanner EDA \rightarrow tanner tool v13.0 \rightarrow S-edit.



🖬 S-Edjt				
Eile Edit Yiew Draw Cell	Setup Iools <u>Window</u> Help			
Vow pen Save Coso Import Export	Kew Ele			
Page Setup Print Preview Brint	Ctrl+P			
1 C ADocuments and Settings)\ 2 D AManiprovino tamer 2 C ADocuments and Settings)\ 4 D ADobouncer Project,\invert 5 D ADobouncer Project,\invert 6 D ADobouncer_Project,\invert 7 D ADobouncer_Project,\invert 8 D ADobouncer_Project,\invert Recent Files Recent Scripts Egt	singleskage, tanner Inv(Inv. tanner tør, tanner tør, tanner Inverter, tanner ter, tanner ter, tanner			
Open 💌 Instance Find				Properties Find
	Command			.0×
Symbols	# SED 2009-06-15 (2:43:52 Started: # SED Vist <u>Tarner EDA Online)</u> window size 126 179 768 442 -ma setup general set -language engli mode celect	5-Edity13.11 bund 3994 н sh		
Ubraries Hierarchy	source startup			
New Design	1		Select	CAP NUM OVR
S-Edit				- # X

Eilo Edit Yiow Draw Coll	Setup Tools Window Help	
DDCC		
View View	🛠 🕨 🖽 🗞 🏷 🖓 📭 🚽 🚽 🖉 🖓 🖓 🖓 👘 😌 🔤 😵 😵 😵 😵 😵 😵	
Loranes 🥝 🖻		Properties 🥩 🗙
~		
		(二) × (1) 35 35
		Be fec 🗢 🤕
Add Remove	New Design	
Fíter 🔽	Design name: Give here design name	
	Create in folder:	
	D:\AMan 💌 🛄	
	Give path OK Cancel	
Open - Instance Find		
		Properties Find
	Correspond	/0 ×
	# SED Visit Tarrer EDA Online) window size 126 179 768 442 -may	
Symbol:	setup general set -language english	
Libraries Herarchy	source startup	
Beady	Educt	CAR NUM CAR

🖬 S-Edit			đΧ
Eile Edit Yiew Draw Cell	Setup Icols Window Help		
		·* • • • • • • • • • • • • • • • • • • •	• *
Vion Vion	を M 切 心 IP 2910.0 9、 SELECT SELECT SELECT	•	
Libraries 🥑 🕷		Properties.	×
×		to X t B. B.	
		8% PBr 🔿 105	
Add Remove	New Design		
Fiter	Design name: Give here design name		
	Create in folder:		
	D:\AMan V		
	1		
	Give nath		
Open - Instance Find			
		Properties Find	
	Cormand		.(9 ×
	# SED_2009-06-15 12:43:52_Started; 5-Edt v13.11 buid 3994		~
	window size 126 179 768 442 -max setun general set -language english		
Symbol:	mode select		1
Ubraries Herarchy			×
ricady	Sect	CAP I NUM	M OWR
S. Felix			-1 ×
Eile Edit Bau Desu Call S	dana Taula Wadayi Hala		
		A constant and an and an	>>
		* • • • • • • • • • • • • • • • • • • •	
			-
Luranes C 🖸		riuperces	0.0
singlestage		* × + a _b a _b	
& LogicSates		P. A	
singlestage			
Devices Misc			
SPICE_Commands			
Add Remove			
Filter	Add Library		
singlescage	File: Include library here		
	Ings\Administrator\Desktop\vtuexperiments\singlestage\singlestage tanner 🕑 📖		
	1		
1 1			
Open Instance Find			
		Properties Find	
	Constand		Øx
	mode select		~
	design open (C) design open (C) Documents and Settings\Administrator\Decktop\vtuevperiments\singlestage\singlestage 4 SED Loading Cultures from C (Documents and Cattions) designs to Decktop vtuevperiments) in state of the term	tanner). E Commenderstellerer der	A Care
Symbol:	# SED_Loading Callbacks from C:(Documents and Settings)Administrator(Desktop)/tuexperiments(singlestage)LIERARIES(SP)C # SED_Loading Callbacks from C:(Documents and Settings)Administrator(Desktop)/tuexperiments(singlestage)LIERARIES(Misc)	onnianos(scripts(open.design) (scripts(open.design)(GeneralCalba)	acks_I
Libraries Hierarchy	<		>
Ready	Sebit	CAP NUP	M OVR

VVIT DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

VVIT

S-Edit		
Eile Edit View Draw	Cell Setup Tools Window Help	
🗅 🖻 🐸 🔚 🖉 🐇 🖣	<u>Now Viewaan na aka 18 18 18 18 19 19 19 19 19 19 19 19 19 19 19 19 19 </u>	
🖌 View 🕑	Open Yew	
Lorance	Rename View	Topartes 😡 🛪
singlestage	Delete View	En NZ A JA JA
	Page +	
SPICE_Elements singlestage	Copy Cell	Ba ^H BC ♥ ♥
Devices	Bename Cell Redirect Instances	
SPICE_Commands	 Update Symbol	
Add Remo	P Instance i	
Filter		
Open V Instance F	Find 1	Properties Find
	Command .	
Symbol:	mode select mode select source startup design open {C:\Documents and Settings\Administrator\Desktop\vtuexperiments\singlestage\singlestage ta # SED Loading Callbacks from C:\Documents and Settings\Administrator\Desktop\vtuexperiments\singlestage\LIERARIES\EP\CE_d # SED Loading Callbacks from C:\Documents and Settings\Administrator\Desktop\vtuexperiments\singlestage\LIERARIES\EP\CE_d	nner] Commands\scripts\open.design\Genc ipts\open.design\GeneralCalbacks_
Libraries Hierarchy		2
Create Cell	Select	CAP NUM OVR

M S-Edit									_ 5	X
Eile Edit View Draw Cell S	etup <u>T</u> ools <u>W</u> indow (Heip								
		1. t. 🗗 · e	8 ? 🧼 🔋 🛛	4	201	*	: II 1 0	5 @ 7m 1		»
View View		Ib		nch 🗸 🚦	SELECT		SELECT			
Lbraries 9 ×						í de la compañía de la		Properties	5	9 *
single stage										_
								* × +	ab ab	
LogicGates SPICE Elements								Ba Par 🔷	*	
singlestage										-
SPICE_Commands										
Ned Down		New View			E State	3				
Add Kenove			-	120						
singlestage		Design:	singlescage	×	OK					
		Cel:	2610	~	Cancel					
		View type:	schematic	~						
		Interface name:	view0	~						
		View name:	view0							
		Disable name va	alidation							
Longe La Constanti de la Consta										
Open Tinstance Find										
								Properties	Find	
	Command								\$	9 ×
	mode select									^
	design open {C:\Docum	ents and Settings	\Administrator\De	sktop\vtu	experiments	singlestage	singlestage t	anner}	winter and a stand of	
Symbol:	# SED Loading Calibacks fr	on C: Documents an	d Settings (Administrat	or(Desktop))	/tuexperiments	singlestage\LIE	RARIES(MISC)S	cripts\open.de	signi,GeneralCalback	s_
Libraries Hierarchy	<									2
Ready	Marca				Selec	E.			CAP NUM	OVR

DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING





After completion of the design just export T-spice



DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

🖷 S-Edit - (singlestage:schema	utic]								💶 🗗 🗙
Eile Edit View Draw Cell	Setup Tools	<u>W</u> indow <u>H</u> elp							_ & X
0000000000000		⊆ ⇔ → & t. 6	1. 61) 🧶 🤌 🧍	062	О L ? 1	》 臣十。	6 🖲 🖭 🕞 🖯	
🗸 View 🔍 🕂 🖇	V 🛛 🖉	A SAP	V	-1:2:0.4 in	ch 🔍 🚬 SEI	LECT	SELECT		
Lbraries 🕑 🛪	singlest	age:schematic					X II P	Properties	9 ×
singlestage 🗸 🗸		Export SPICE					1000	Schematic of singlest	tage
	ana na an							10 X 1 36 3	łb
LogicGates SPICE Elaments		To file		÷		Export	10100		
singlestage		nts and Settings\Adr	ninistrator(C	Tesktop\singlestage.	SDC 🖌 🛄	Carcel			
Devices		Confirm overwrite						Srice	-
SPICE_Commands	2019 (B. 16)		Give pat	h where u wan	it to save the	e T-spice file	•0.00 to 15	General	
The second s		Export source						🗄 AdditionalC	VI VIND VIND 31
Add Remove		Design:	[singlestage	~		•10 • 10 • 10	LibraryFiles	Generic_025.lb
			1					Transient	Think
Filter		Cell:	1	singlestage	~			Enable	10p
singlestage		View:		view0	~			E Stop	100n
	Vi						181 2 2	System	
		Export as		Export mode				Author	
	10.00	Top-level cell		() Hierarchical			101 101 12	⊕ ⊂ell	singlestage
		Subcircuit definition	n	◯ Flat				Date	6/6/2009 12:03
		1 h-						Design	singlestage
	400 A 40	Options					10104	Into	
		Exclude .model	1	Exdude .end				Incerrace	VIEWD
		Exclude global pins	s on subcircu	uits				Omanization	Nevio
	+x+	Vitran lines to	80 (tharacters				RevisionCount	3
	5.5 J 5						10703-0		
Open V Instance Find		Export control property	v						
		Property name:	1	SDICE	120			Deres New Stand	
		Troporcy number	-	Drice				Properdes Fillo	
	Command								<u> </u>
	mode draw	instance	1 0 765	10.001					~
	window fit -	-x0 -0.372 -y0 -0.303 -x	1 0.700 -y. 1 1.204 -y	1 0.401 -units inci 1 1.127 -units inci					
	mode escap	pe Instan displaytant, call (
Symbol:	Len open -o	iesiyii siriyiestage -cell s	singlestag	e -cype schemaci	5				*
Libraries Hierarchy	5								2
Ready						Select			CAP NUM DYR



I I-Spice - [INV]		
File Edit View Simulation Table Setup Window Help		_ & ×
🗅 🛎 🖬 🚳 📐 🛤 者 🖻 💼 이 이 📑 🔞	۸	
\$ ▶ ■		
" Expand patho: yes		~
* Wrap lines: 80 characters		
* Root path: D:\vtuexperiments\INV		
* Exclude global pins: no		
* Control property name: SPICE		
********** Simulation Settings - Ceneral se	ection *********	
******** Simulation Settings - Parameters	and SPICE Options ********	
* Devices: SPICE.OPDER > 0		
MNMOS 1 Out A Gnd Vdd NMOS W=2.5u L=250n .	AS=2.25p PS=6.8u AD=2.25p PD=6.8u	
MPMOS_1 Out A Vdd Vdd PMOS W=2.5u L=250n .	AS=2.25p PS=6.8u AD=2.25p PD=6.8u	
.PRINT TRAN V (A)		
. PRINT TRAN V (Out)		
******** Simulation Settings - Analysis :	section *********	
******* Simulation Settings - Additiona	1 SP KE commands ********	
.tran 10n 100n		
*.lib "G:\vtuexperiments\INV\Generic 025	Lib" TT	
.lib "C:\IMS\vtucxperiments\INV\Ceneric_O	25.1ib" TT	
V1 & GND BIT ((11001100))		
VZ Vdd GND 5	Open T-spice and add these commands to compute the result	
	······································	
END		~
		2
		1
Status input file Out Start Date/Time Elap		
arty	In 38, Col 1	
And I	Erroby Cort	





ii) Layout (L-edit): Start \rightarrow Programs \rightarrow tanner EDA \rightarrow tanner tool v13.0 \rightarrow L-edit

₩L-Edit - Layout1	
File Edit View Draw Cell Satup Tools Window Help	
10 📽 🖬 番 永 階画 🛍 🕰 🔄 🗃 🦛 森 森 武 森 戸 🔐 Altevels 💽 😅 🖥 🗃 🚍 🛷 🕴 82.00:50.00 Microns	~
▶□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□	
◆ � 匹亞司 ⊘ ⊕ 픧 旦 № № ♥ \$\$ Ø Ø H Ø ♠ \$ Ø ♥ \$\$	
話 ま 湖 書 蒋 ++ 益 HI 玉 III II	
Nwel 5 5 Cello Layoutt	
Drawn V L L	
- **** 50	
Close this window	
Label File	
Device La	
P 1994	
2011 Thick Anti	
beer: 59 %	
	Mode: Drawing

File View Setup Tools Heb New Cirl+N Cirl+N Cirl+N Cirl+N Cirl+N Open Cirl+O Cirl+O Cirl+O Cirl+O	
New Ctrl+N Open Ctrl+O	
Import Mask Deta	
Print Setup	
1 C.11M5)\Common_drain 2 C.11M5)\Single_stage 3 C.11M5\\R2R_DAC(R2R_DAC	
Create a new document 374	



🚇 L-Edit	- 2 🔀
File View Setup Tools Heb	
10 🛎 🖬 🚳 🖄 🖻 🎕 🖄 🖆 🖓 🗃 🆓 🎝 🖓 🖉 Altevels 💽 💕 🖷 🖾 🚍 🚸 🕴 81.00; 19.25	Microns
F □ B B B I - Default R - N - N - N - B B B B B B B B B B B B B	
◆★ 0,000,000 西中司の今日四周の 2 学習法のな时期 たちもも	18 1 2
本語 幸 蒋 寺 益 田王 ■ 王田 近江 文 ② 術 天文 名 ⁰⁰⁵	
С Н И Ореп ?Х	
Drawn 😒 💆 🗗 🕺 Look in: 🍙 GenericD_25un 🕑 🥥 🌮 📰 •	
Layer There are no items to show,	
Desklop	
Ny Documents	
My Computer	
File name: Genetic 025 V Open	
Select: 20 Tanner Database Files (*tdb) V Cencel	
Ready	22

55



D-Fdit - [Cell0 Layout2]	2
🛅 File Edit View Draw Cell Setup Tools Window Help	- 8 ×
🞦 🖨 📮 🚭 🕺 🖻 🛍 🗠 🗠 🖂 🖬 🖨 🎧 📽 🥼 💭 🕵 Altevels 🔽 🥵 🖥 📴 🚍 🥏 🛛 31.02:149.86 🛛 Micronis 🔽 🗖	
▶ □ Seve the active document Default PA · · · · · · · · · · · · · · · · · ·	
◆ 冬 First Save it 国際司の登録用 № 0. 次 弊緊張 ダ 次 H 12 前 30 前 18 18 12 23	
	Weder Presson
Save the active document Save	Mode: Drawing



DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

EC6612 VLSI DESIGN LAB	57
🗑 L-Edit - [Cell0 Layout2]	
🔯 File Edit View Draw Cel Setup Tools Window Help	_ 8
○ ご ご ご ご ご こ こ こ こ こ こ こ こ こ こ こ こ こ こ	
Original Wathanese® Solution: Original III IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII	
Rectale Witzerd Image Layers Schup Leyer Polebies Merge Layers Detec Layers	

🕀 L-Edit - [Cell0 🛛 Layou12]		- B X
File Edit View Draw Cel Setup Tools W	isdow Help	- 11 X
Image: Second Secon	Image: Second	
Ready	578	Mode: Drawing

VVIT DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

🗑 L-Edit - [Cell0* Layout3*]	
🛅 File Edit View Draw Cel Setuo Tcols Window Heb	- 6 ×
🛅 🖨 🔚 🎒 👗 😫 🛍 🕰 🖆 🔂 🕼 🏟 🧌 🎇 🖏 🖏 🎝 🥵 Allevels 🔽 💕 🖬 🔂 🚍 \$	<u>~</u>
▶ □ Б Б Δ 飞 ヽ > Defail > N = ヽ い ┏ ┗ & & A 4 4 8 0 B B B か &	
◆ � [] 四空司 ⊙ ↔ झ 8 16. 22 🚏 📽 🖉 22 년 16 18 18 19 12 23	
NWeel Drawn V Layer Layer Layer Layer C T T Layer C T T Layer C T T C T T	
Selection: Instance 'US' of cell 'TC_NMOS_Auto 2.5, .25 1 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0 0	Mode: Drawinn





59



DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING







🗑 T Spice	-[INV_1]		wani		
🕍 File Ed	t View Simu	liation Ta	ble Setup Winde	s Help	- 6 3
0 20	166	44 X	B	E 🔘 🥏	
-	Save	77			
2C 🕨	Date				
" EXU	act pate	and Ti	me: U6/23/2	JU9 - 12:01	
* NODE	NAME ALT	ASES			
*	1 = Out	(26.9	5 , 17.681		
	2 - GND	(25.9	9 , 10.01)		
*	3 = vdd	1 (25.9	8 , 24.14)		
	4 = 8 ((25.02	, 17.24)		
M1 Out	A GND CN	D NHOS	L-250n V-2.	Su AD-2p PD-6.6u AS-2p PS-6.6u 9 126.34 12.36 26.59 14.86	
M2 Out	A vdd vd	A PROS	L=250n ¥=2.	5u AD=2p PD=6.6u 18=2p PS=6.6u \$ [26.34 20.P5 26.5P 23.45]	
* Tots	1 Nodes:	4			
* Tota	1 Element	a: 2	and Reserves		
A Outs	ut Genera	or and	langed Times	D 015 gen	
* Tota	I Extract	Elans	ed Time: 1.1	0.010 Bec	
VI A C	ND BET ((110011	001)		
v2 vdd	GND 5				
.tran	10n 100n				
.print	tran v(A	4) v Ou	(t)		
.lib	C:/INS/ve	uexper	iments/INV/C	eneric_025.1ib" T7	
- END					
*					
Status	Input file	Out	Start Date/Time	Elap	^
Inished	INV.SD	IN	Juy 01, 20	00	
finished	TWA-shc	IN	July 01, 20	00	
finished finished	DVV.SD				
finished finished finished	INV.sp INV.sp	DN	July 01, 20	90. /.	
finished finished finished finished	DNV.sp DNV.sp DNV.sp	IN IN	July 01, 20 July 01, 20	00 00	
finished finished finished finished	INV.sp INV.sp INV.sp	IN.,. IN.,.	July 01, 20 July 01, 20	00 00	

COULT A LEI DESIGN LAD	02
A T-Spice - [Single_Stage]	
📓 File Edit View Simulation Table Setup Window Help	- 6 >
🗅 🖆 🖬 🍜 🔃 構 🖇 階 略 🕫 🗢 🖙 🦉 🛷	
* • = =	
<pre>4 = Out (70.44, -37.27) RunSimulation(ybp) (65.74, -26.79) 6 = Vref (65.65, -52.63) 7 = 7 = Vinm (61.29, -44.59) 0 = Vinp (70.32, -44.47)</pre>	~
M1 VDD 3 3 VDD PMOS L=603n W=2.5u AD=1.9975p PD=6.598u AS=2p PS=6.6u \$ (61.56 -32.88 62.163 -30.38) M2 Out 3 VDD VDD PMOS L=603n W=2.5u AD=1.9975p PD=6.598u AS=2p PS=6.6u \$ (69.39 -32.87 69.993 -30.37) M3 2 Vinm 3 OND NMOS L=699n W=2.5u AD=1.9975p PD=6.598u AS=2p PS=6.6u \$ (61.56 -41.99 62.259 -39.49) M4 GND Vne± 2 GND NMOS L=400n W=2.88u AD=2.304p PD=7.36u AS=2.304p PS=7.36u \$ (66.22 -50.11 66.62 -47.23) M5 Out Vinp 2 GND NMOS L=699n W=2.5u AD=1.9975p PD=6.598u AS=2p PS=6.6u \$ (69.3 -41.87 69.999 -39.37)	
* Total Nodes: B	
* Total Elements: 5	
 Total Number of Shorted Elements not written to the SPICE file: 0 Output Generation Flement These 0 000 sec 	
* Total Extract Elaport Time: 1.205 pec	
.ar dec 10 1 10g	
tran 10n 100n	
.11b "C:\INS\vtuexperiments\singlestage\Generic_025.11b" TT	
vi Ving Vina SIN (0 Im IG) AC 1 180	
V3 Vad GND S	
v4 Vinne GND 1.3	
*.print tran v(Vinm) v(Vinp) v(Vout)	
.print ac vdh(Out) vp(Out)	
.ena	
	~
	13.
Status input me Out., StattOater/ime Elep	
un a smulation Ln 13, Col 5	





DEPARTMENT OF ELECTRONICS AND COMMUNICATION ENGINEERING

EC6612 VLSI DESIGN LAB SCHEMATIC DIAGRAM:





EXP.NO: 11	
DATE:	DIFFERENTIAL AMPLIFER

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 AMPLIFER)

VVIT

Time (ns)

EXP.NO: 12	CMOS INVEDTOD	CMOS INVEDTOD	
DATE:	CINOS INVERTOR		

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.

<u>RESULT</u>

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

EC6612 VLSI DESIGN LAB CMOS INVERTER:



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:



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

EC6612 VLSI DESIGN LAB SCHEMATIC DIAGRAM:



EXP.NO: 14 DATE:

AUTOMATIC LAYOUT GENERATION

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.
EC6612 VLSI DESIGN LAB

LAYOUT GENERATION:



VVIT

EC6612 VLSI DESIGN LAB

SIMULATED WAVEFORM:



RESULT:

Thus the automatic Layout was verified through L-edit.