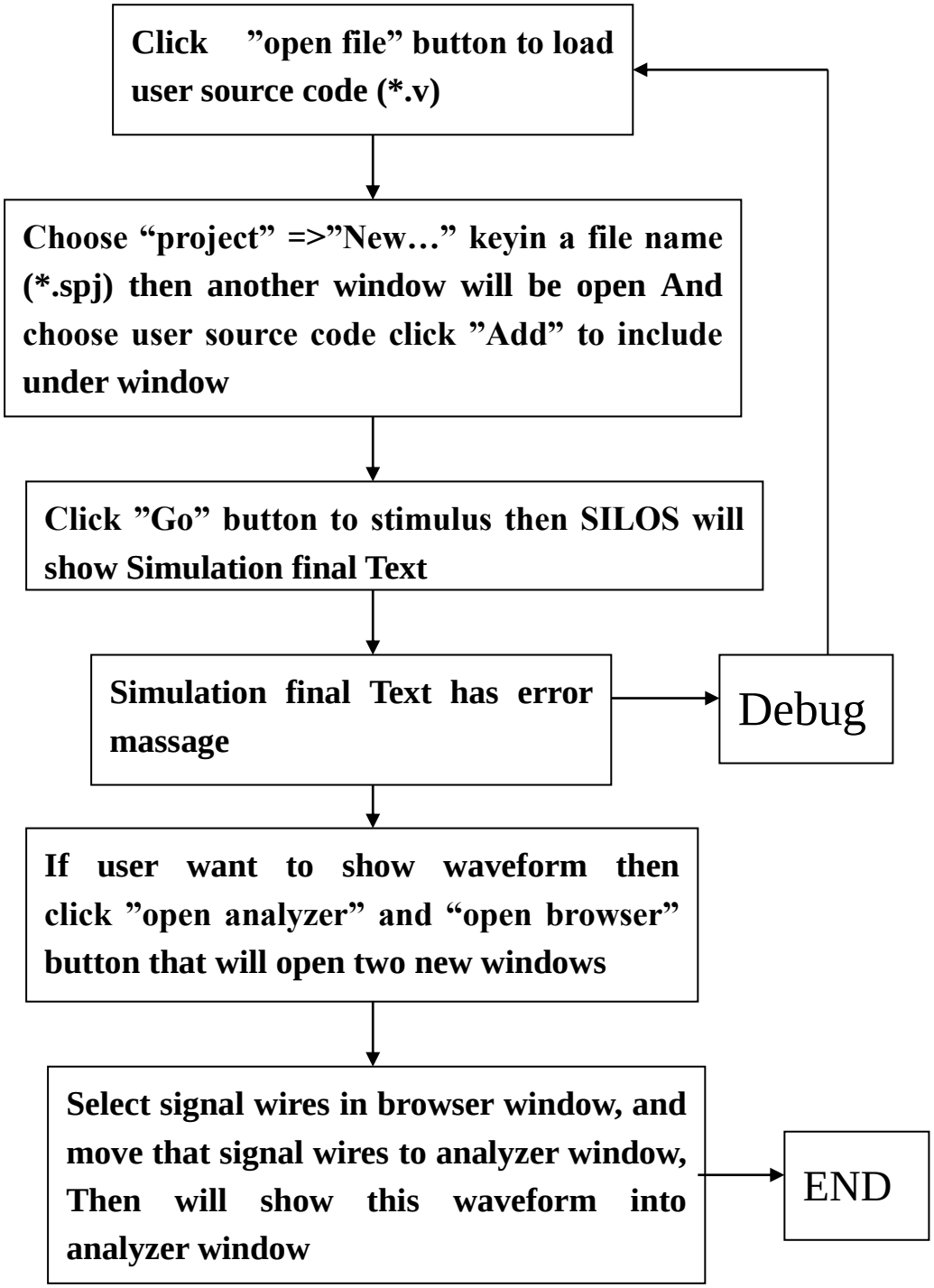
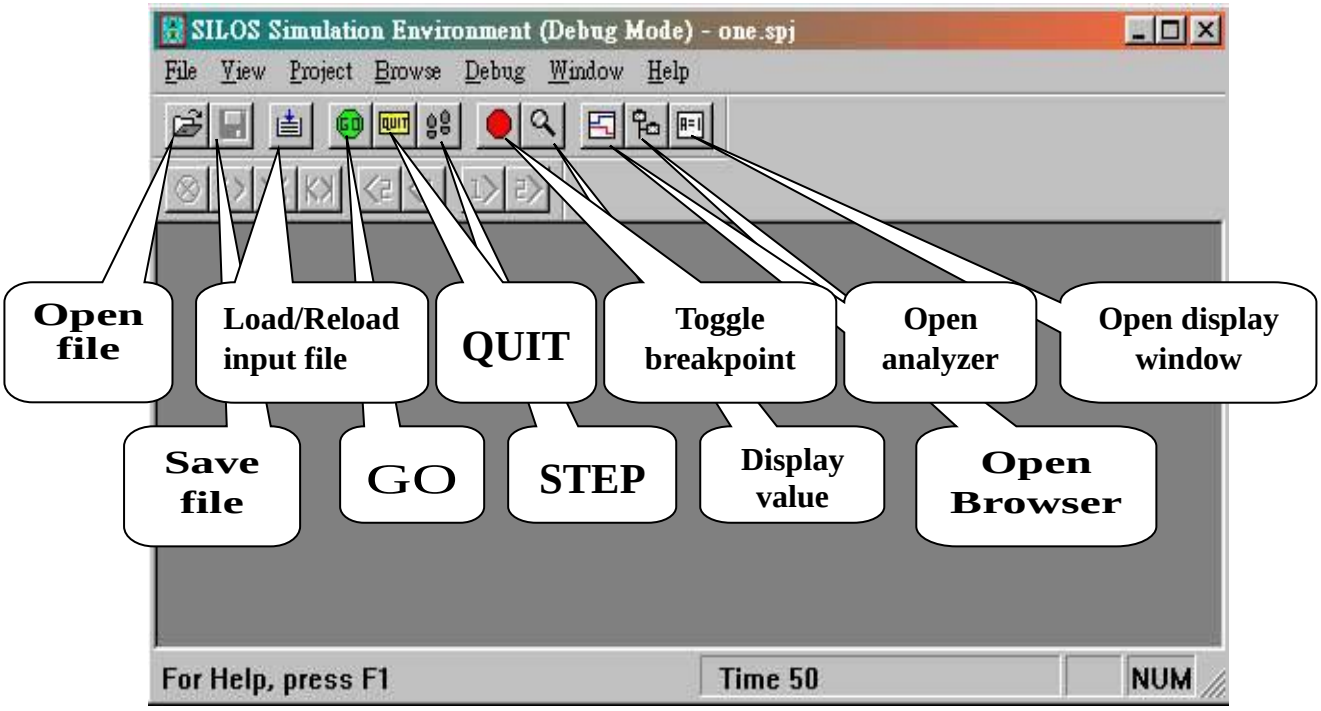
Chapter 1 Verilog Stimulate Environment

1.1 PC Verilog Stimulate Environment

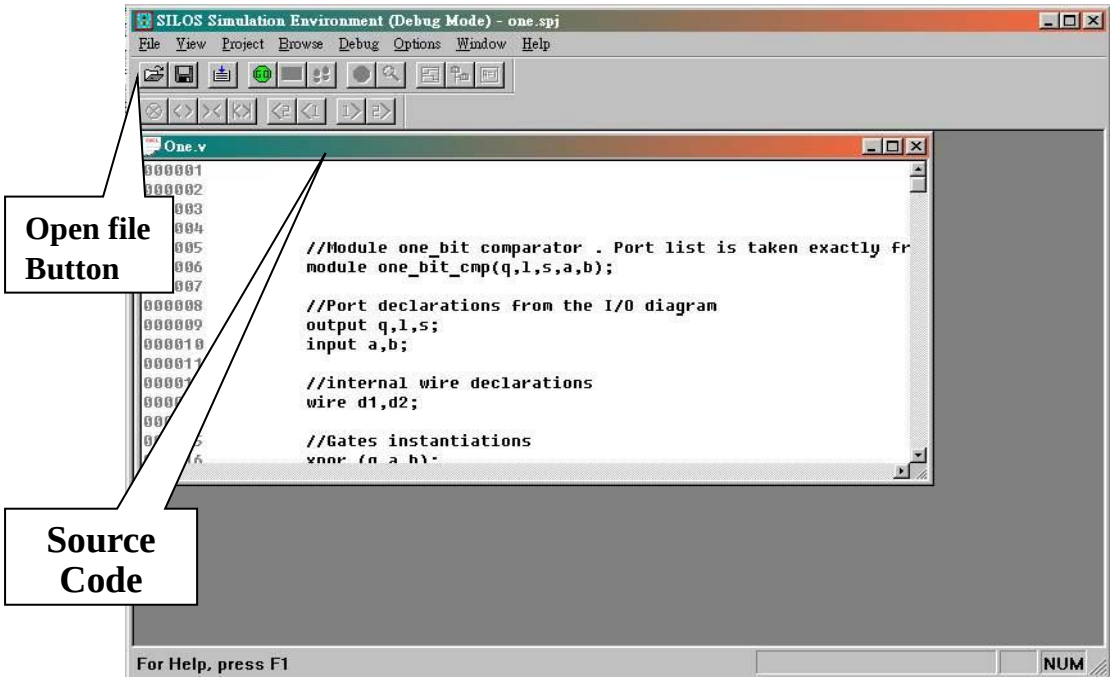
1.1.1 SILOS Environment



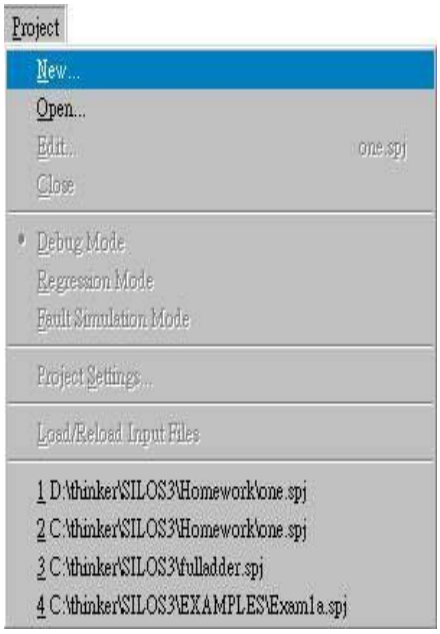
Basic Window



Step 1. Click open file button to open source file

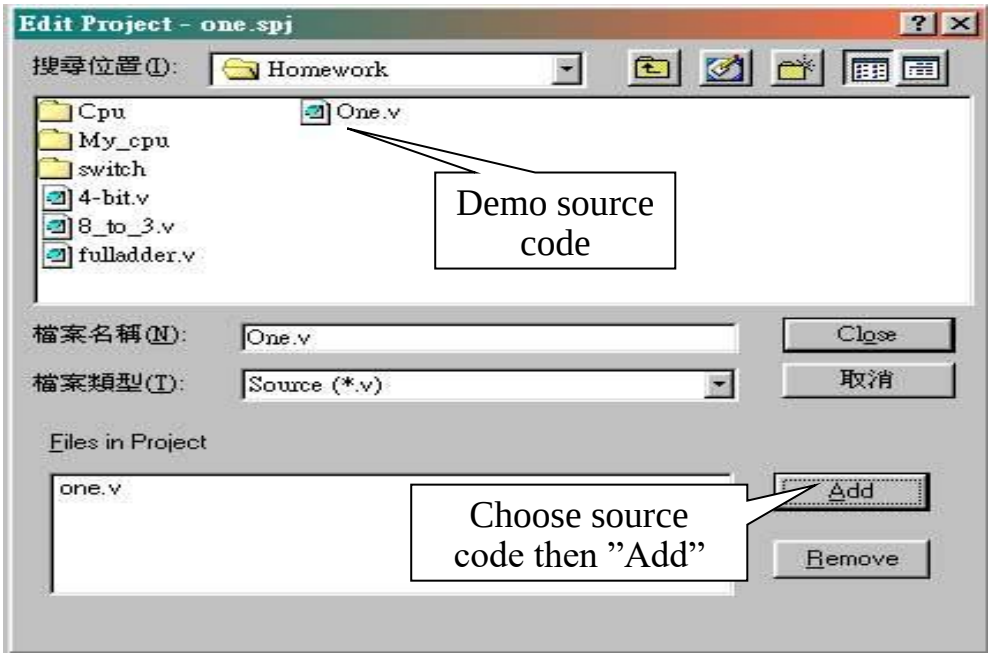


Step2. Choose project => New

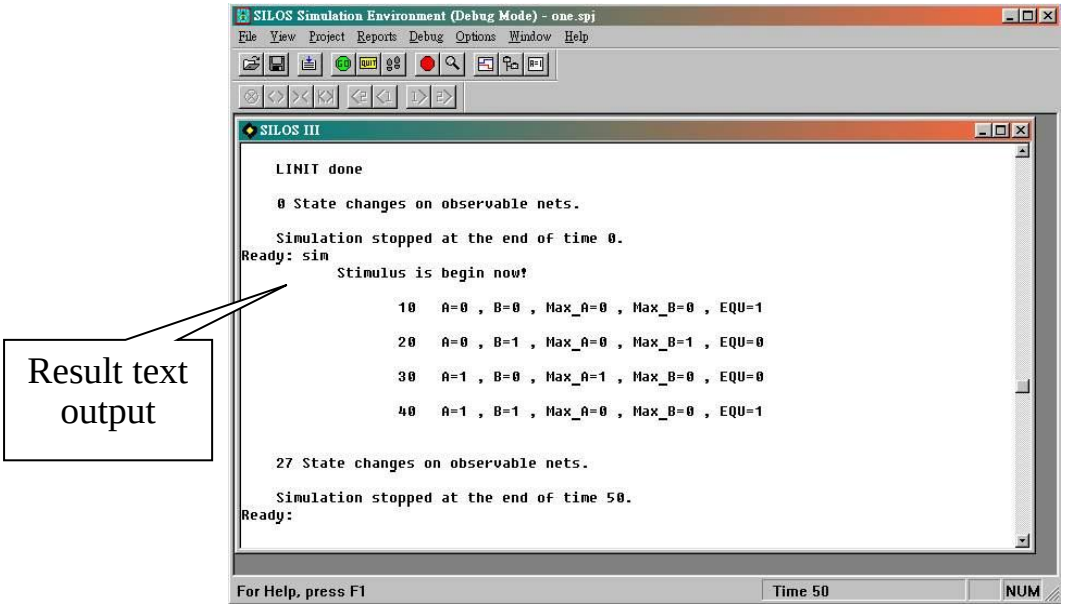


Step3 . keyin file name (*.spj)

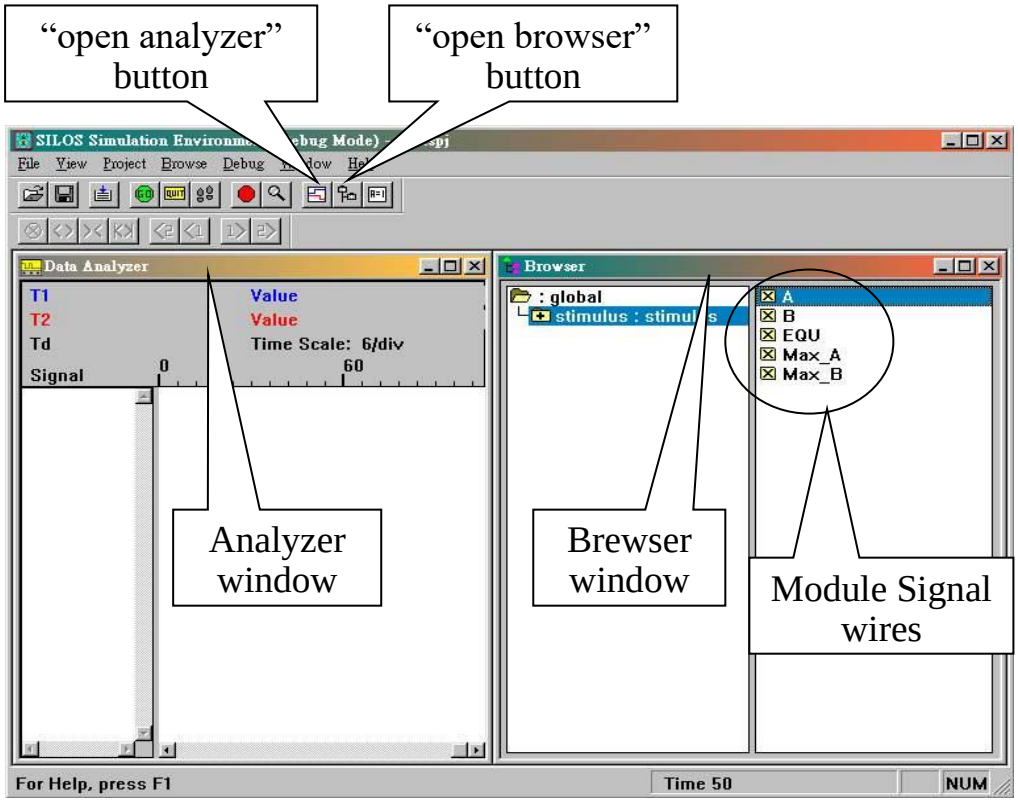
Step4. Choose source code file and Add to under window



Step 5. Click ”go button if user code is Zero error then will show stimulus final Text

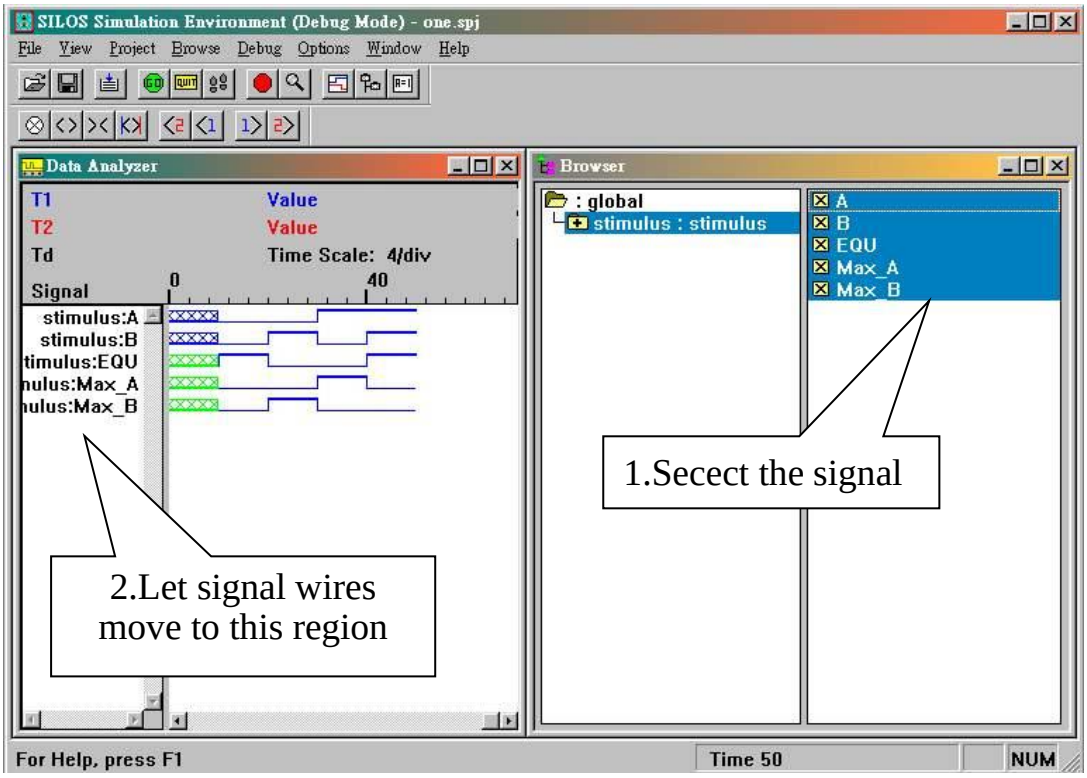


Step 6. Click “open analyzer “ and “open browser” button

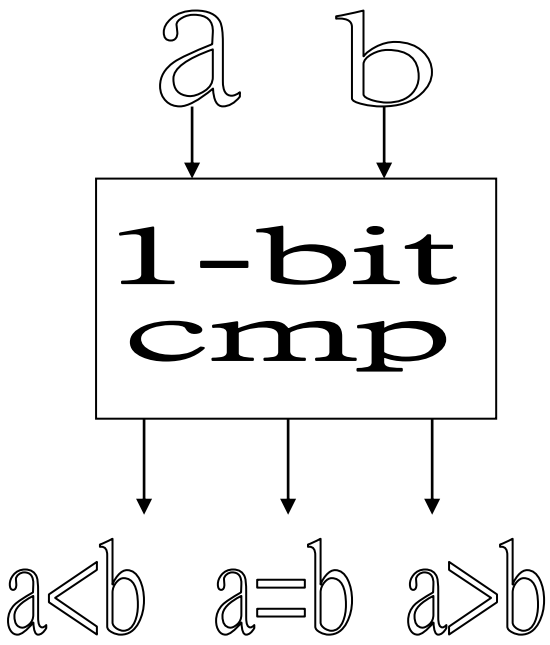


Step 7. Select signal wires and that move to analyzer window

"Signal" region, then waveform will display



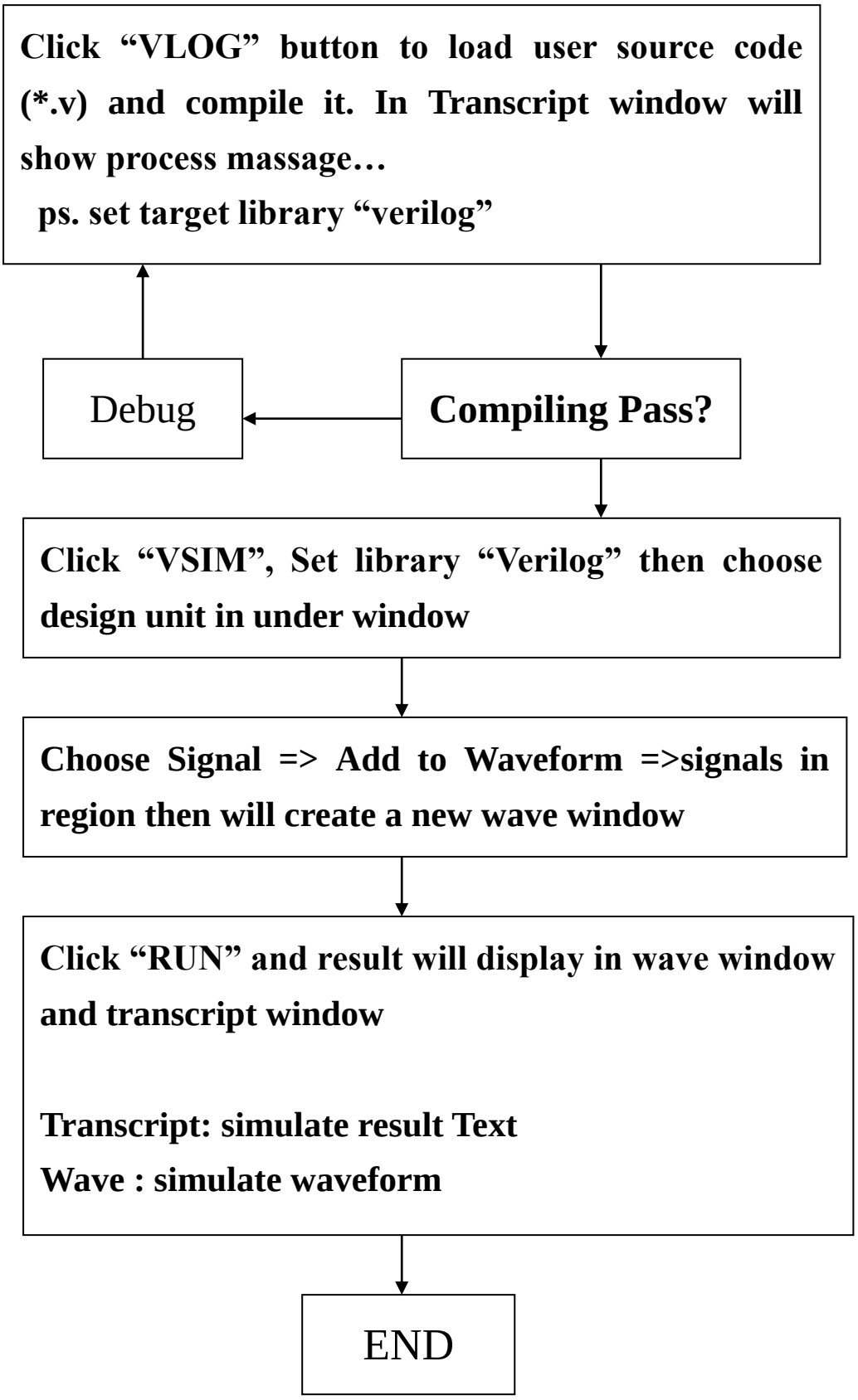
Demo source code : 1-bit comparator



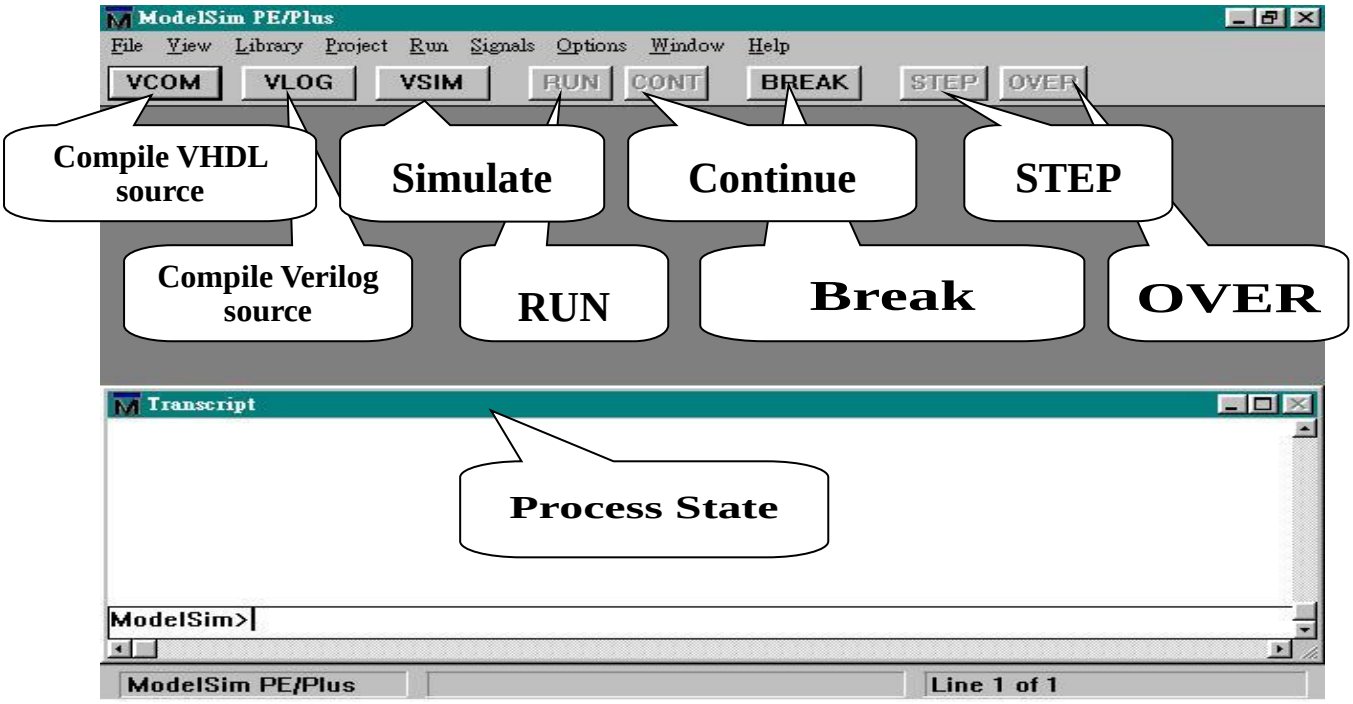

```
//Module one_bit comparator .
module one_bit_cmp(q,l,s,a,b);

//Port declarations from the I/O diagram
output q,l,s;
input a,b;

//internal wire declarations
wire d1,d2;

//Gates instantiations
xnor (q,a,b);
and (l,a,d1);
not (d1,b);
and   (s,b,d2);
not (d2,a);
endmodule
```

```
//Define the stimulus module
module stimulus;

//Declare variable to be connected to inputs
reg A,B;

//Declarate output wire
wire EQU,Max_B,Max_A;

//Instantiate the one-bbit cpmparator
one_bit_cmp    one(EQU,Max_A,Max_B,A,B);

//Display output
initial
begin
$monitor($time," A=%b , B=%b , Max_A=%b , Max_B=%b , EQU=%b \n",
A,B,Max_A,Max_B,EQU);
end
//Stimulus input
initial
begin

#1    $display("Stimulus is begin now!\n");
#9    A=1'b0 ; B=1'b0 ;
#10 A=1'b0 ; B=1'b1 ;
#10 A=1'b1 ; B=1'b0 ;
#10 A=1'b1 ; B=1'b1 ;
#10 $finish;

end

endmodule
```

1.1.2 ModelSim PE/Plus Environment

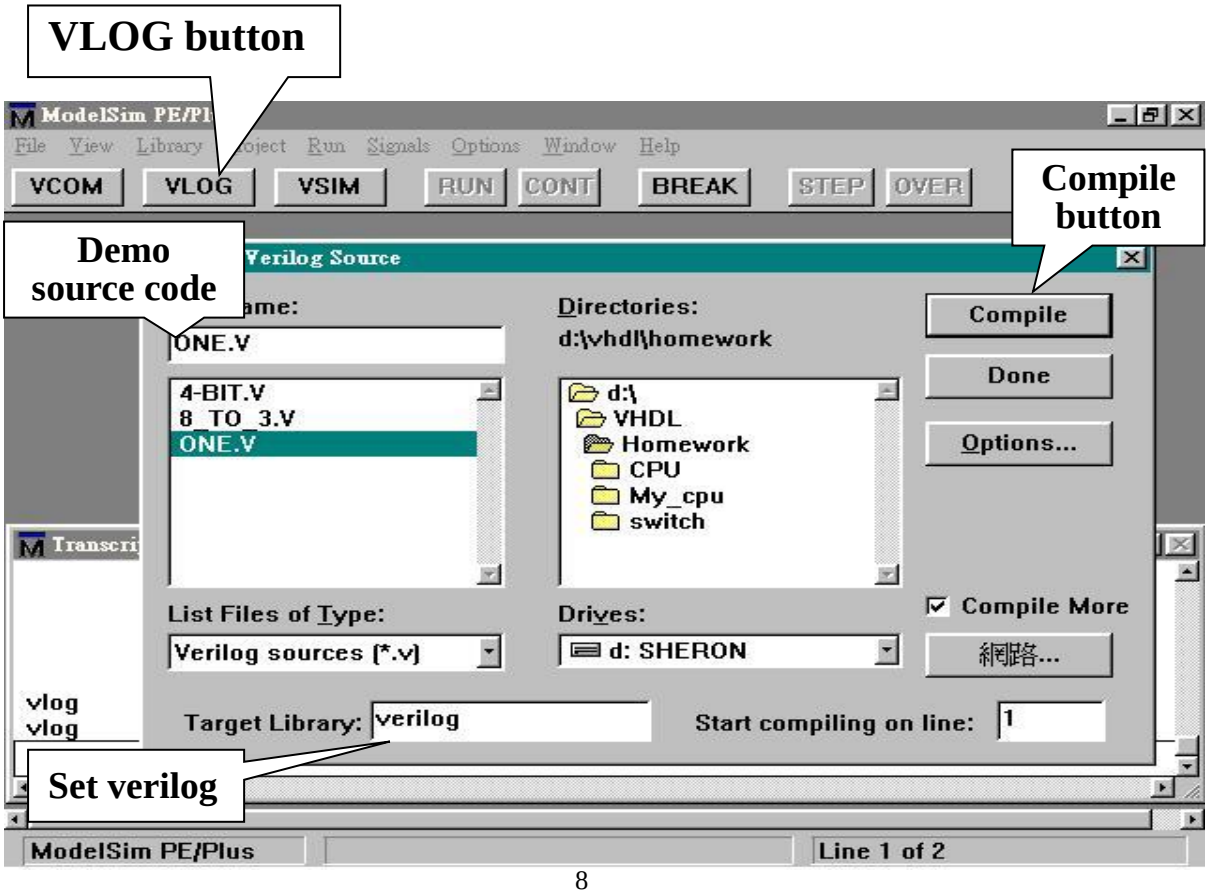


Basic Window

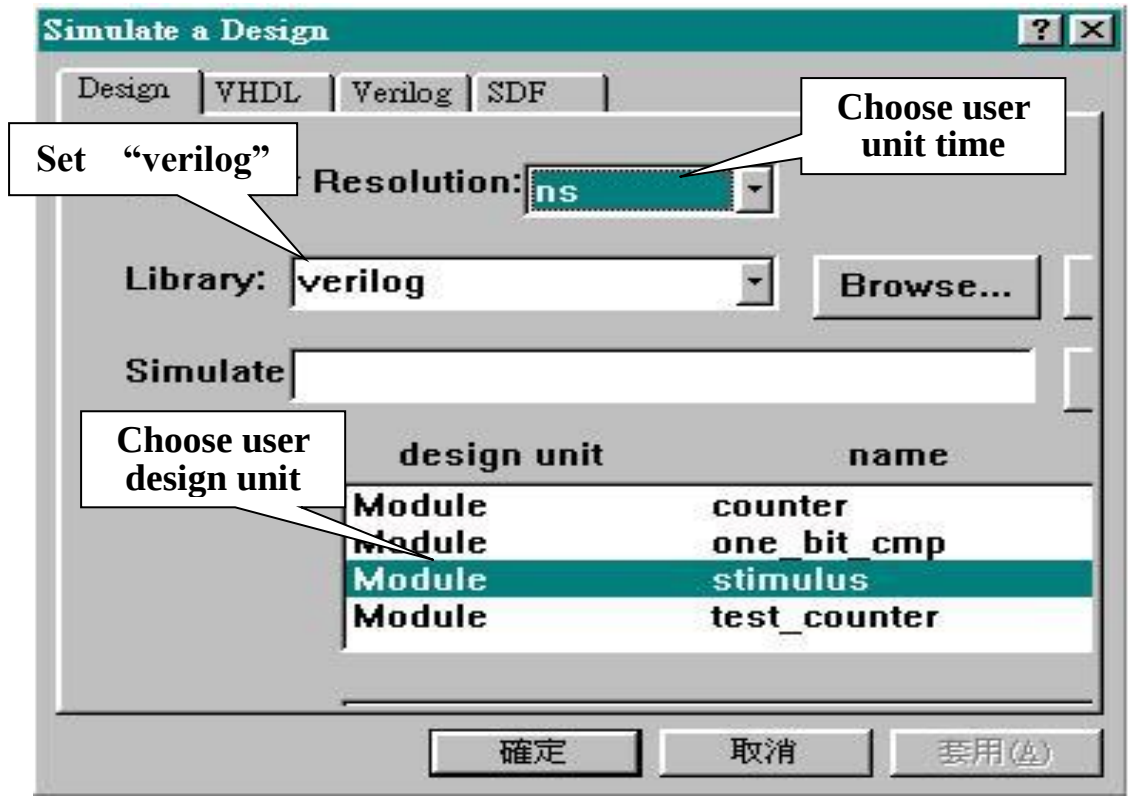


Step1. Click VLOG button and load user source code then Compile

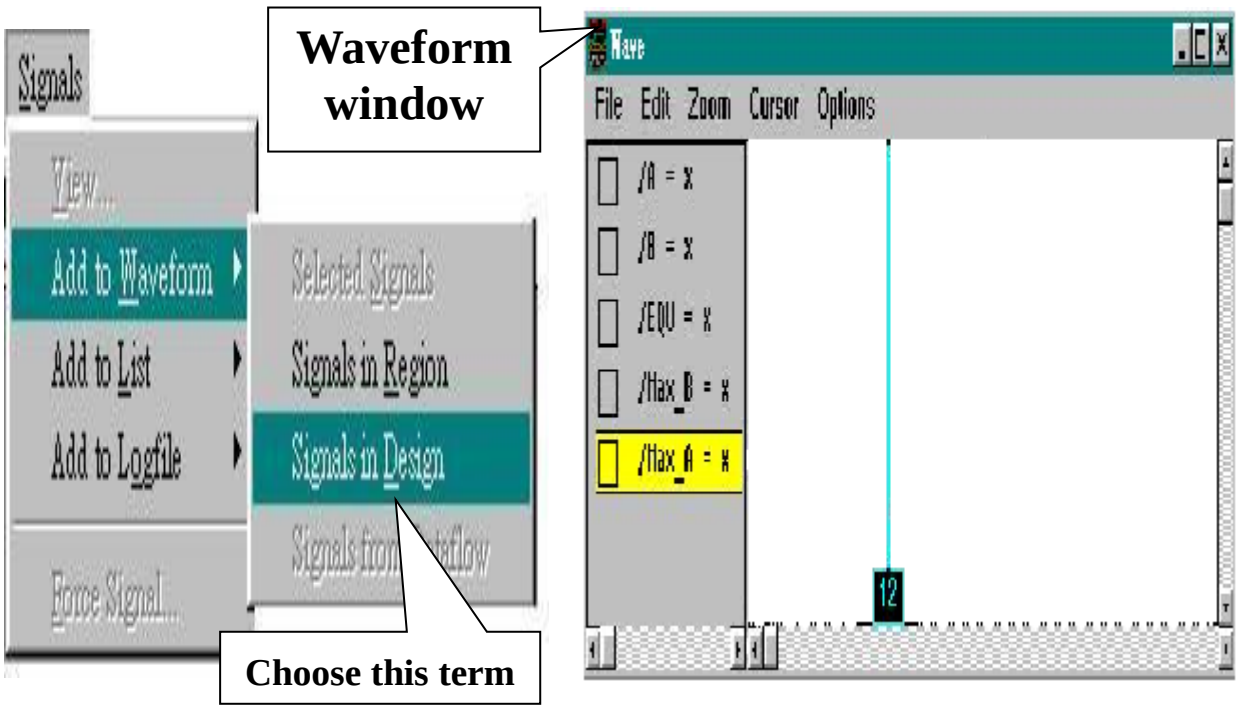
Ps. Set Target Library verilog



Step 2. Click VSIM, set library and choose design unit

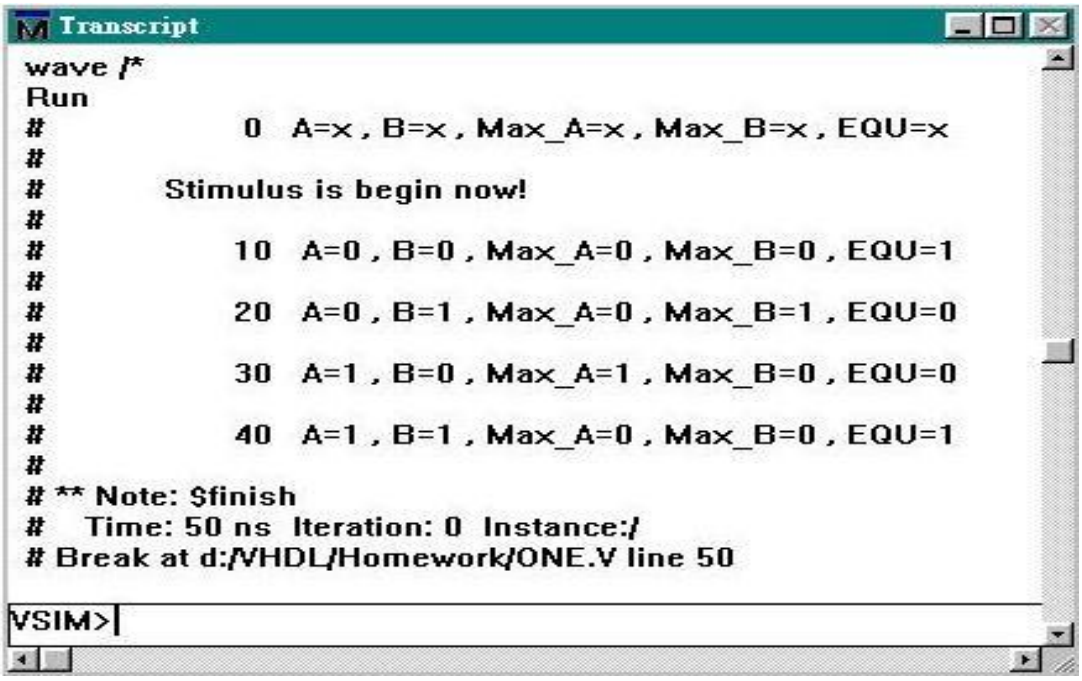


Step 3. Choose signal => Add to Waveform =>Signal in Region then will create a new wave window

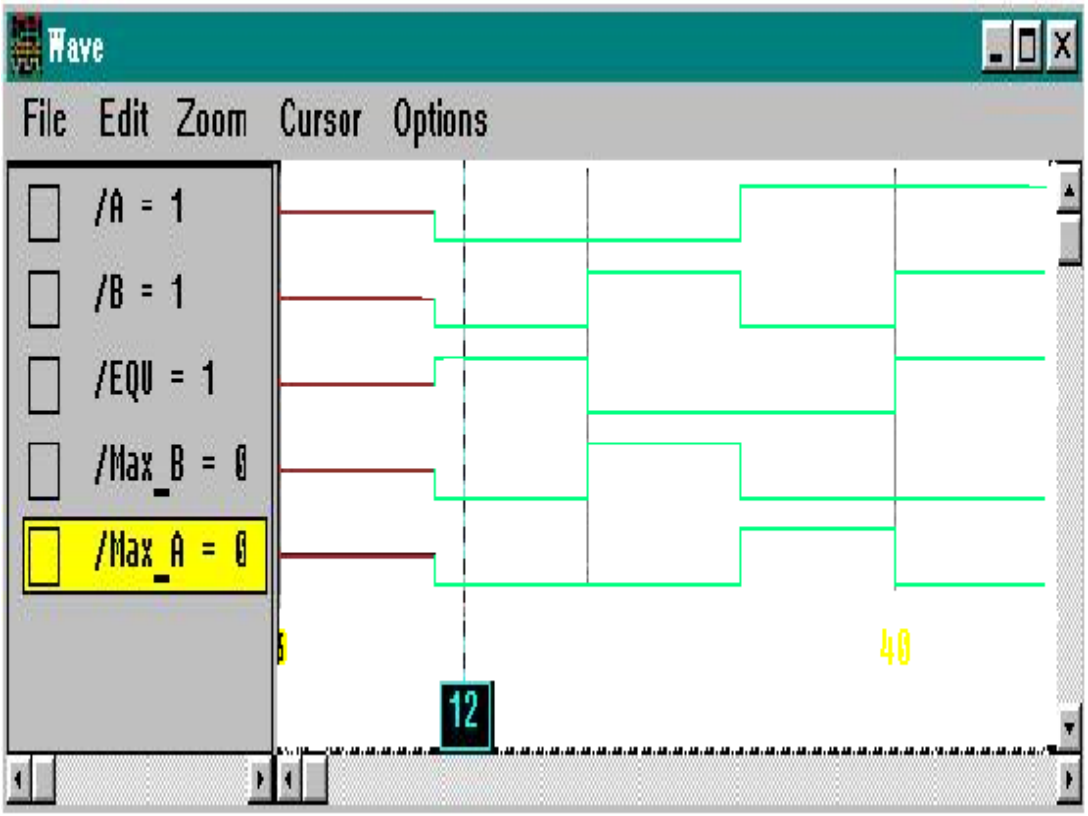


Step 4. Click RUN then create Stimulate Text and Stimulate waveform

(a). Stimulate Text output



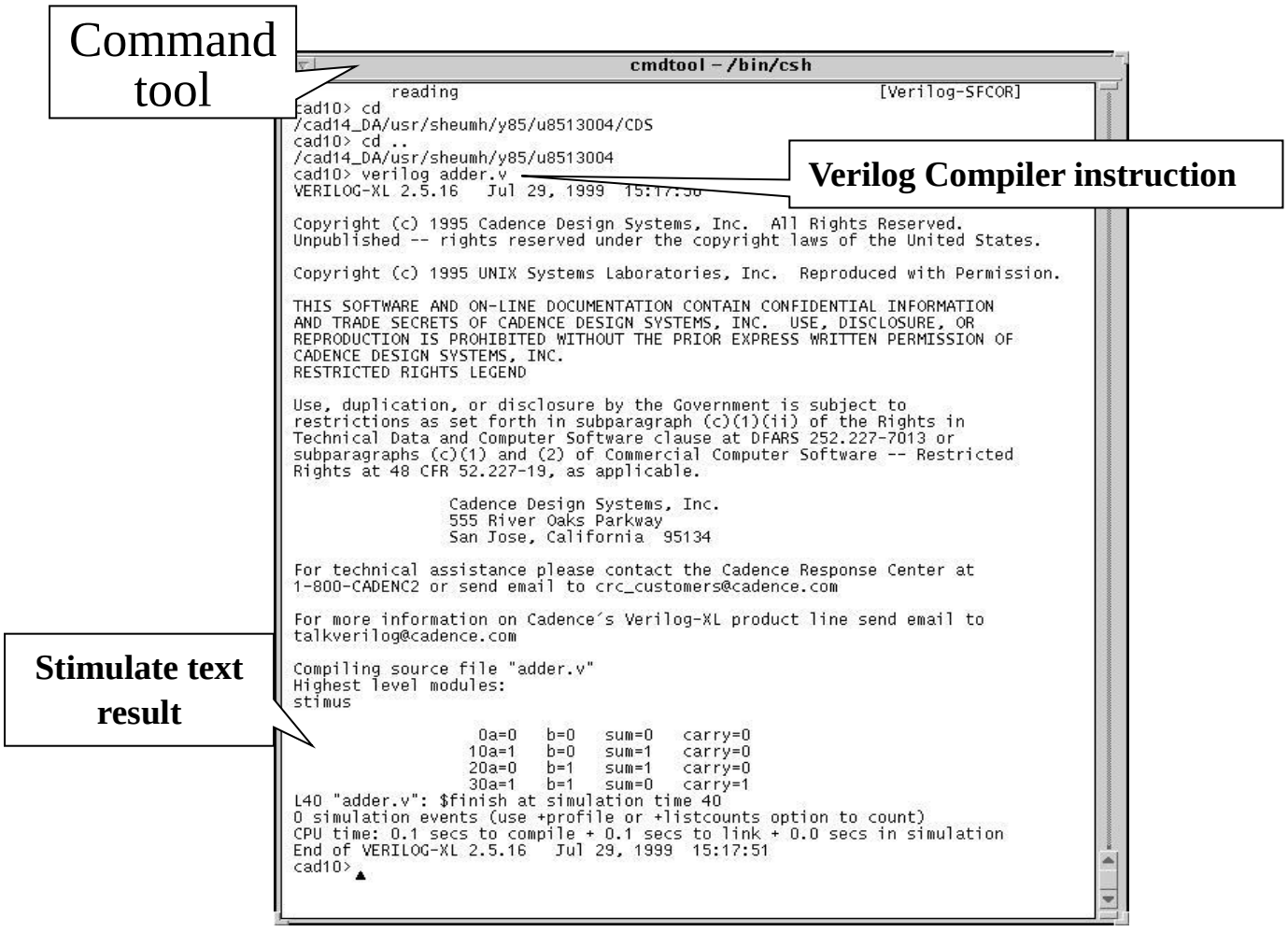
(b). Stimulate Waveform output



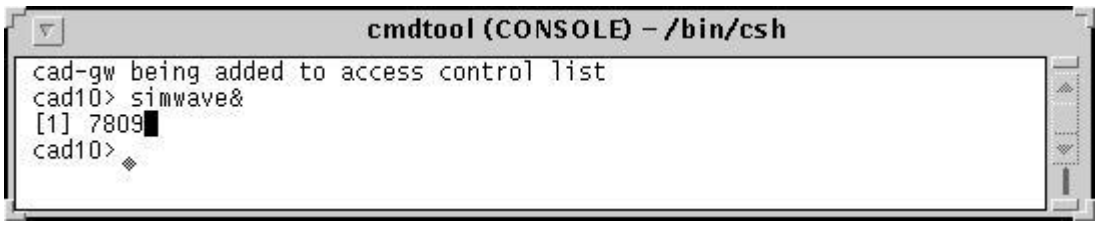
1.2 Workstation CADENCE Environment

Step1. After finishing the verilog source code, keyin the compiler keyword like below.

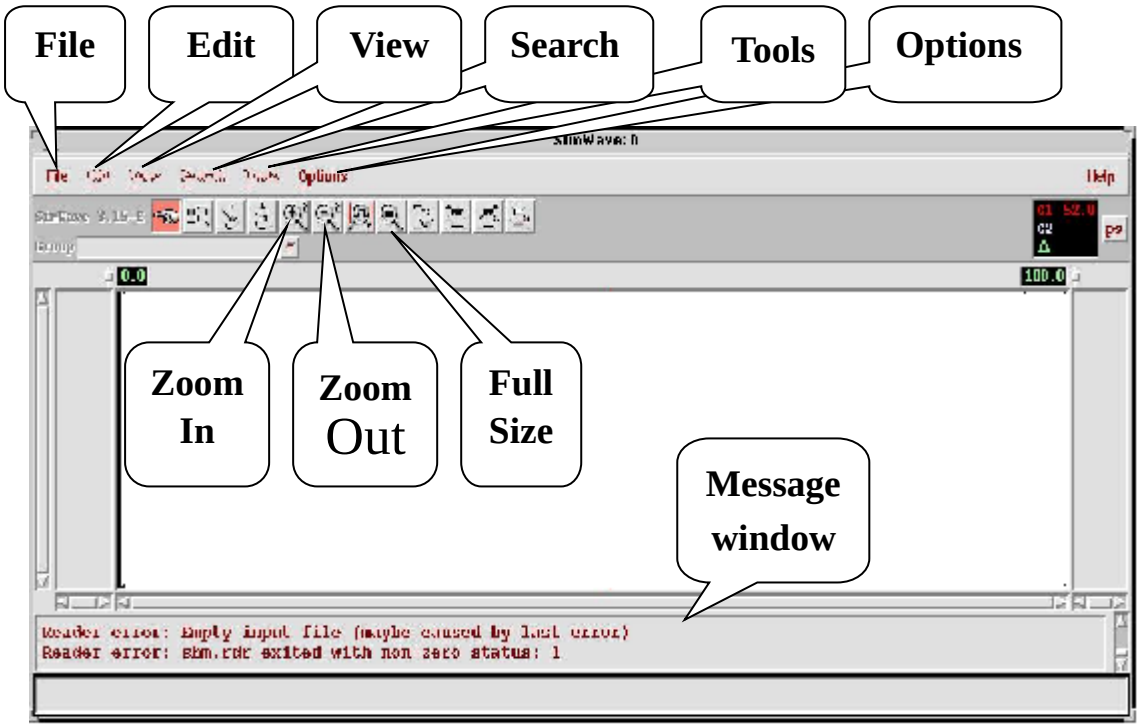
Step2. If user verilog source code is no error, then the system will show stimulate result in command tool window like below.



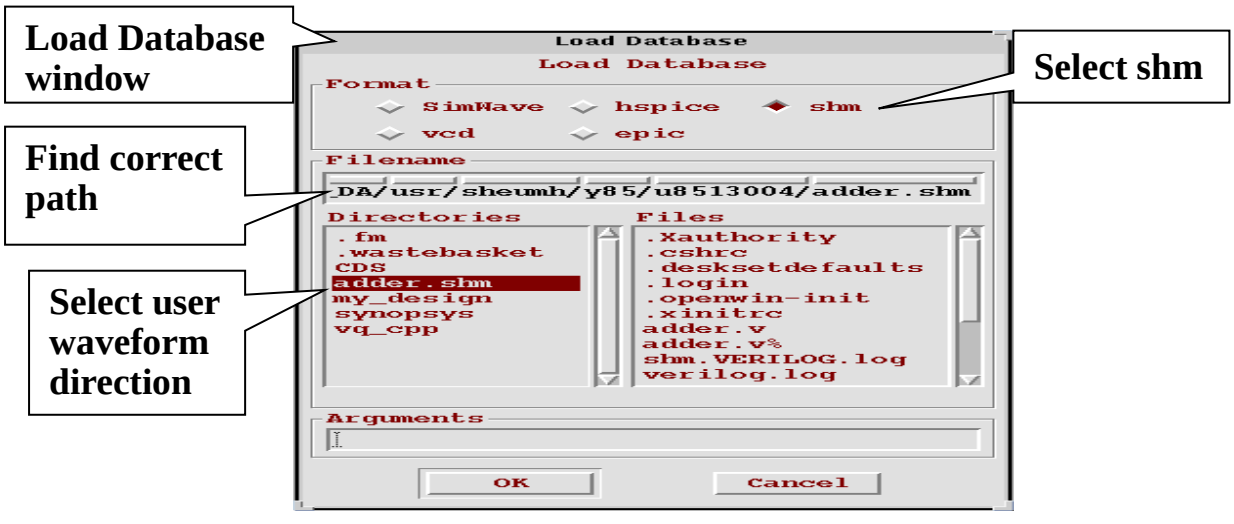
Step3. In Command Tool open the simwave



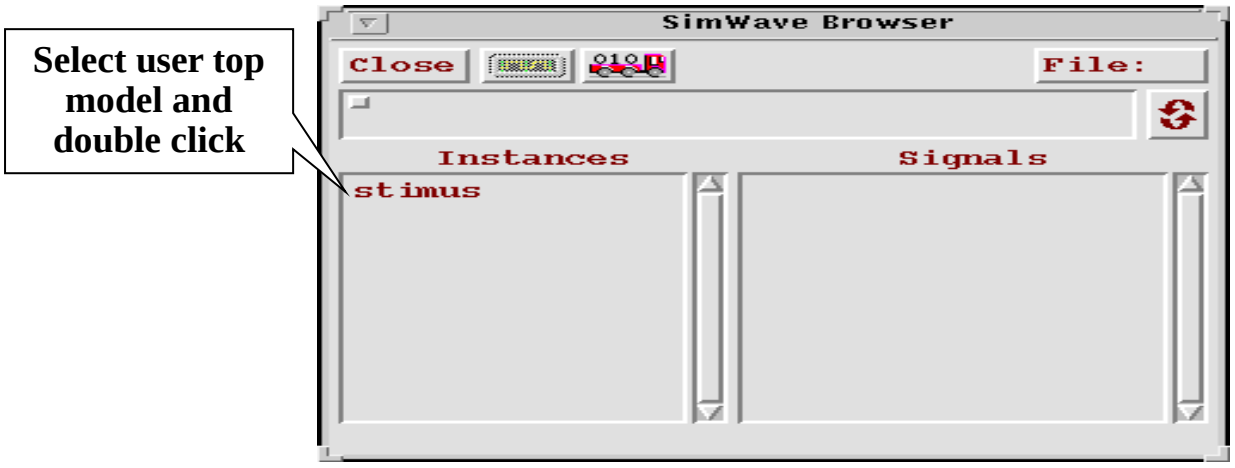
- **The simwave window is shown below**



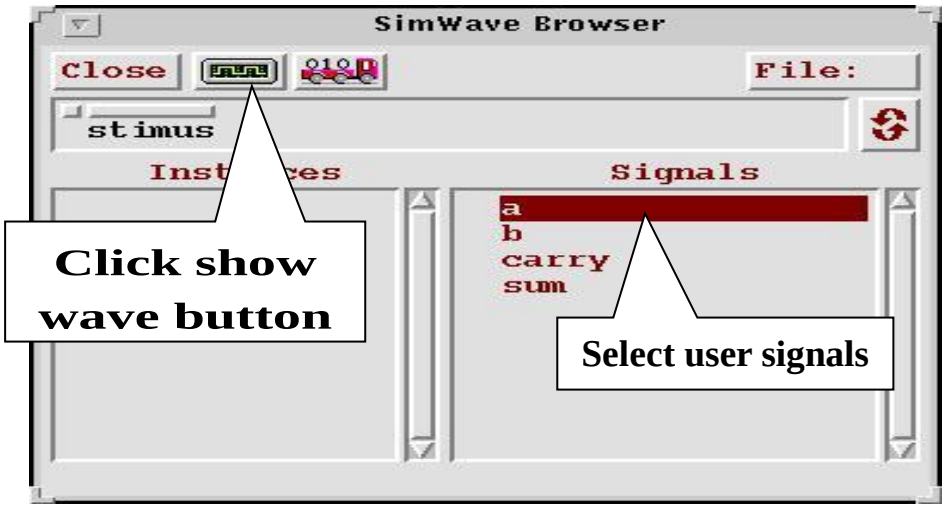
Step4. Click File -> Database -> Load... then open Load window.



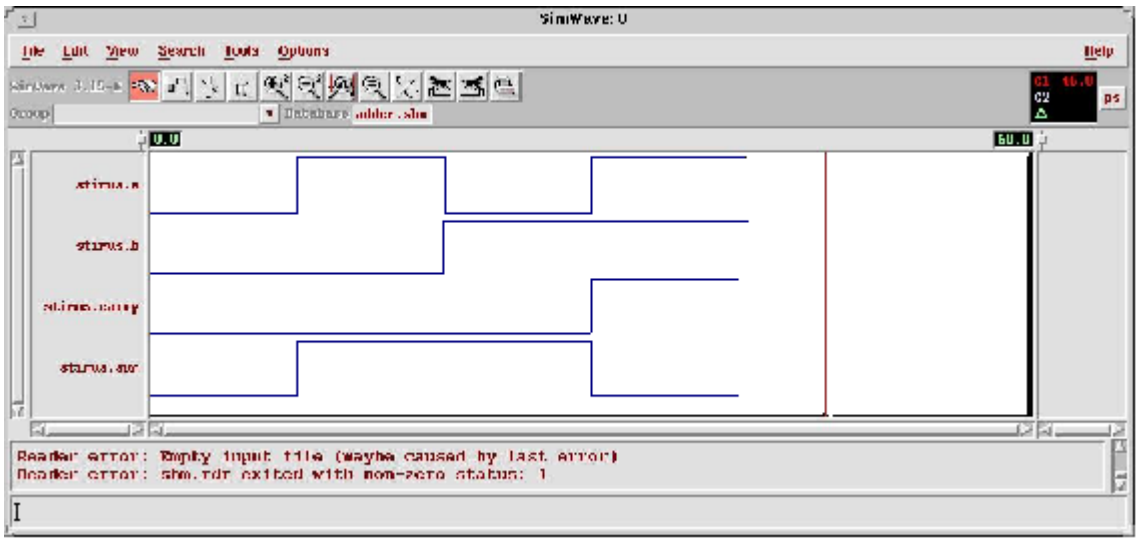
Step5. Click bottom Edit -> Add Signals... then will open browser window

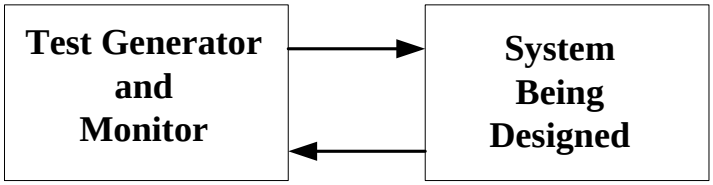


Step6. Select 'a' signal in signals region, then click show wave button



Step7. Repeat step6 to pick up all input and output signals, then the *simwave* window will display those signal waveforms.




Test Generator
and
Monitor
System
Being
Designed