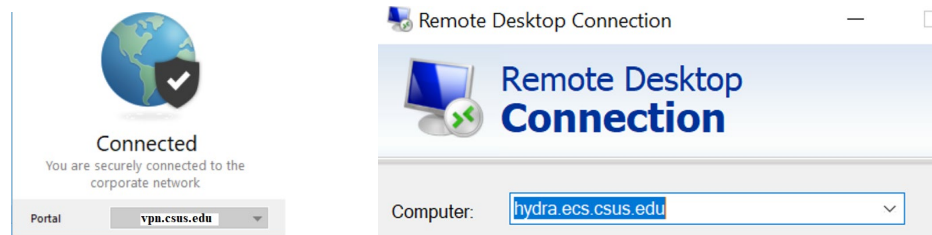


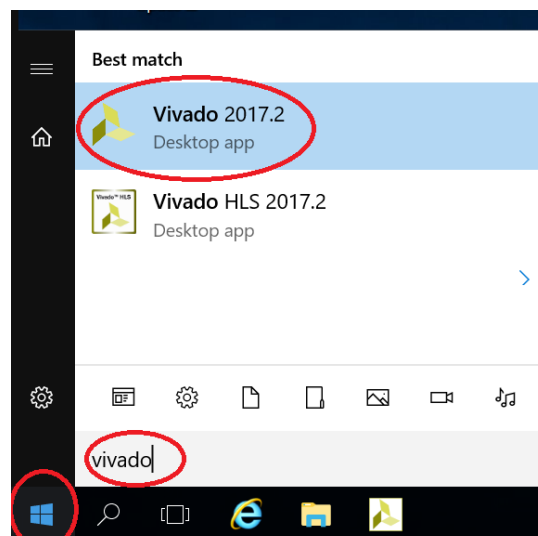
Vivado Simulation Tutorial
By: Prof. Pang

1. Global VPN:
<https://www.csus.edu/college/engineering-computer-science/computing-services/internal/documents/globalprotect-vpn.pdf>
2. Hydra Remote Desktop:
<https://www.csus.edu/college/engineering-computer-science/computing-services/internal/documents/hydra-remote-desktop.pdf>

In order to use the Vivado software tool, you must first establish a connection with the global VPN (Portal: vpn.csus.edu), and then use Hydra Remote Desktop on your computer by connecting with "hydra.ecs.csus.edu".



3. After you log into hydra.ecs.csus.edu, click on the window icon at the left corner of the screen, and then type "Vivado". You'll see the Vivado icon, and then click on it to open the Vivado software tool.



4. After you wait for some time, you will first see the following Vivado window.



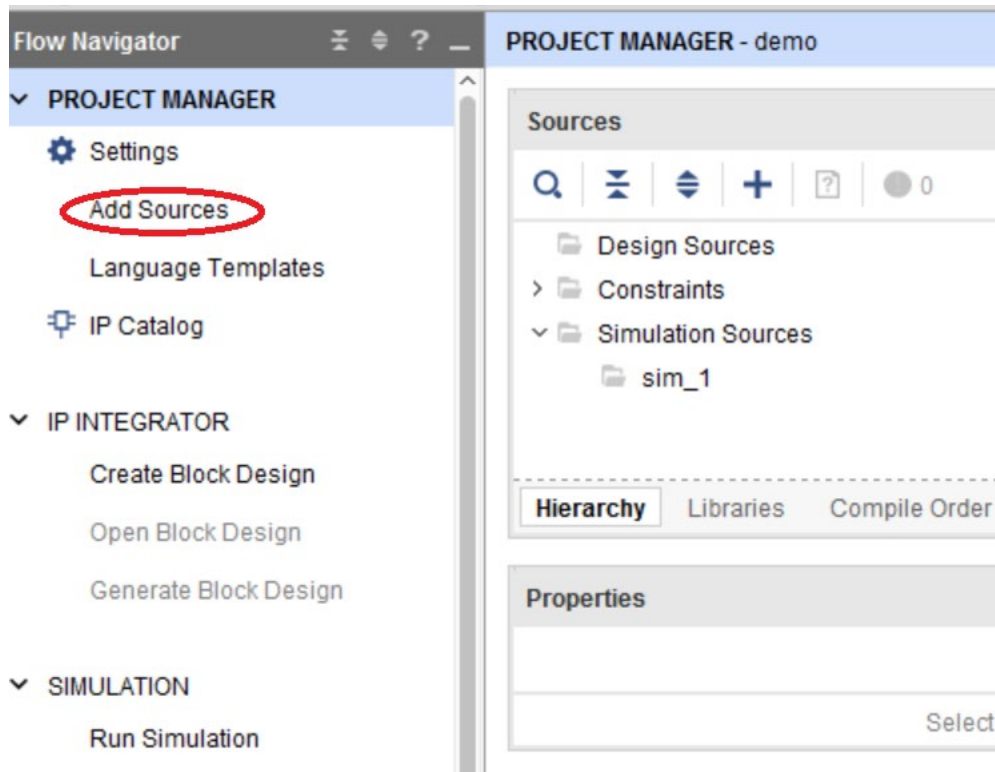
5. Next, after some time, you will see the following window. Then, double click on the “create project” icon.



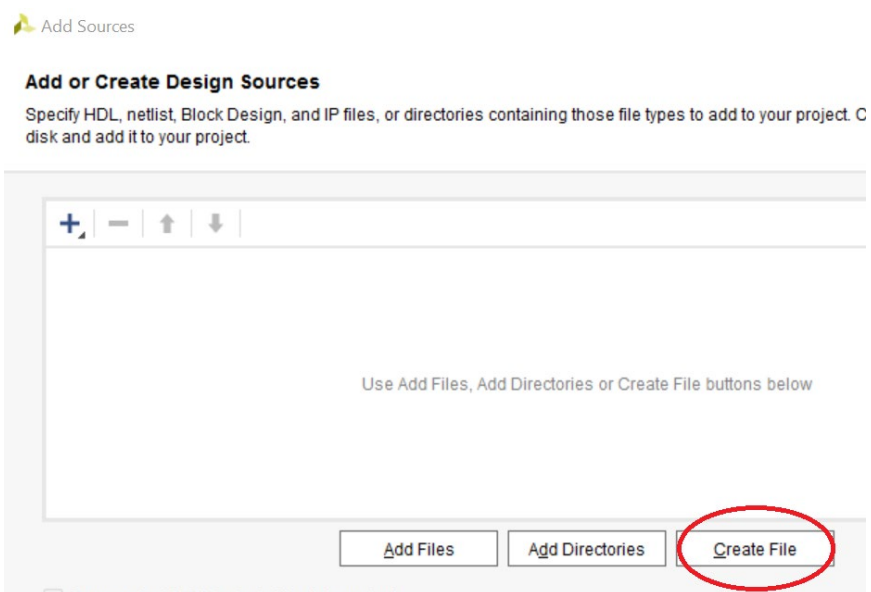
Other Procedures are shown below:

1. Start VIVADO Design Tool.
2. Quick Start -> Create Project -> Create a New Vivado Project
3. Set Project Location to be under your user directory.
4. Type “demo” after “Project Name:” -> click on next, and then select “RTL Project”.
Note: You can type different names for different projects.
5. Do not specify sources and constraints. Just click on “next”.
6. You can keep the default part for the Project settings and then click on “next”.
7. Click on “finish”.

8. Now, click on “Add Sources” -> “Add Or create Design Sources”-> Next.

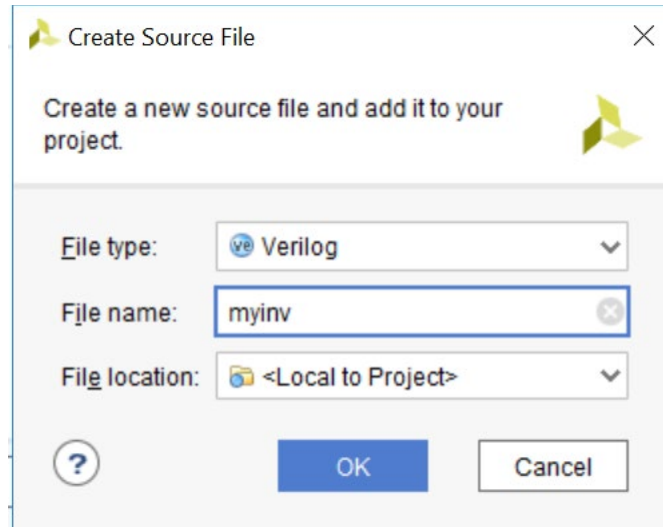


9. Click on “Create File”.

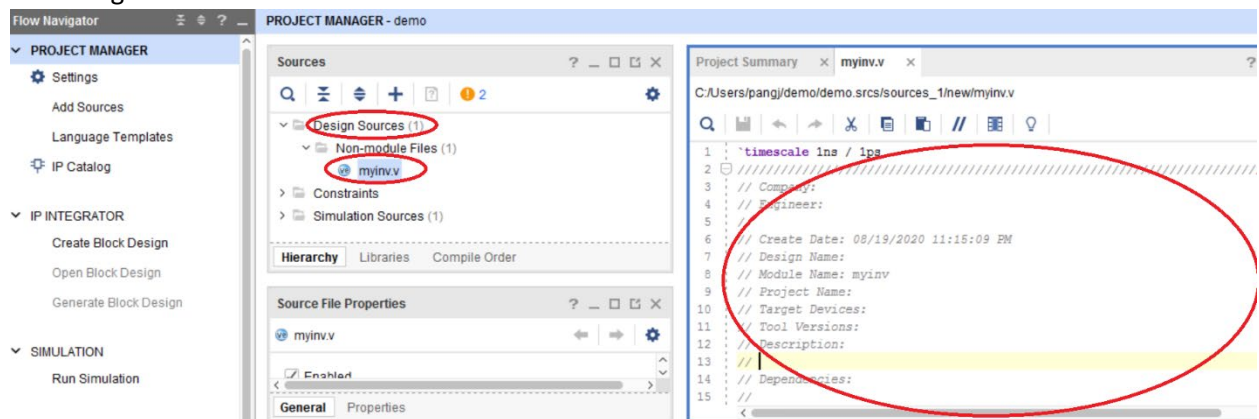


10. Enter “myinv” as file name. Make sure “Verilog” is the file type-> ok-> finish->ok.

You’ll need to select “VHDL” as file type if you work on VHDL projects in the future.



11. Find “Design Sources” located in the the “Sources” window. Click on “Non-Module Files” and then you will see the “myinv.v” icon. Double click on the “myinv.v” icon to find source code on the right.

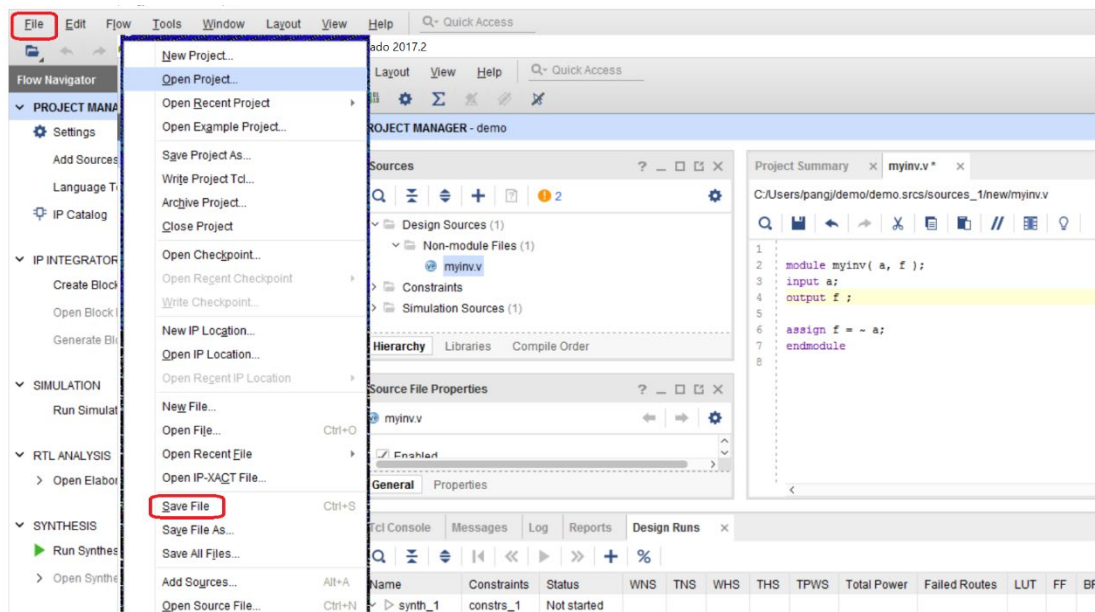


12. Now you can edit the “myinv.v” source code as following:

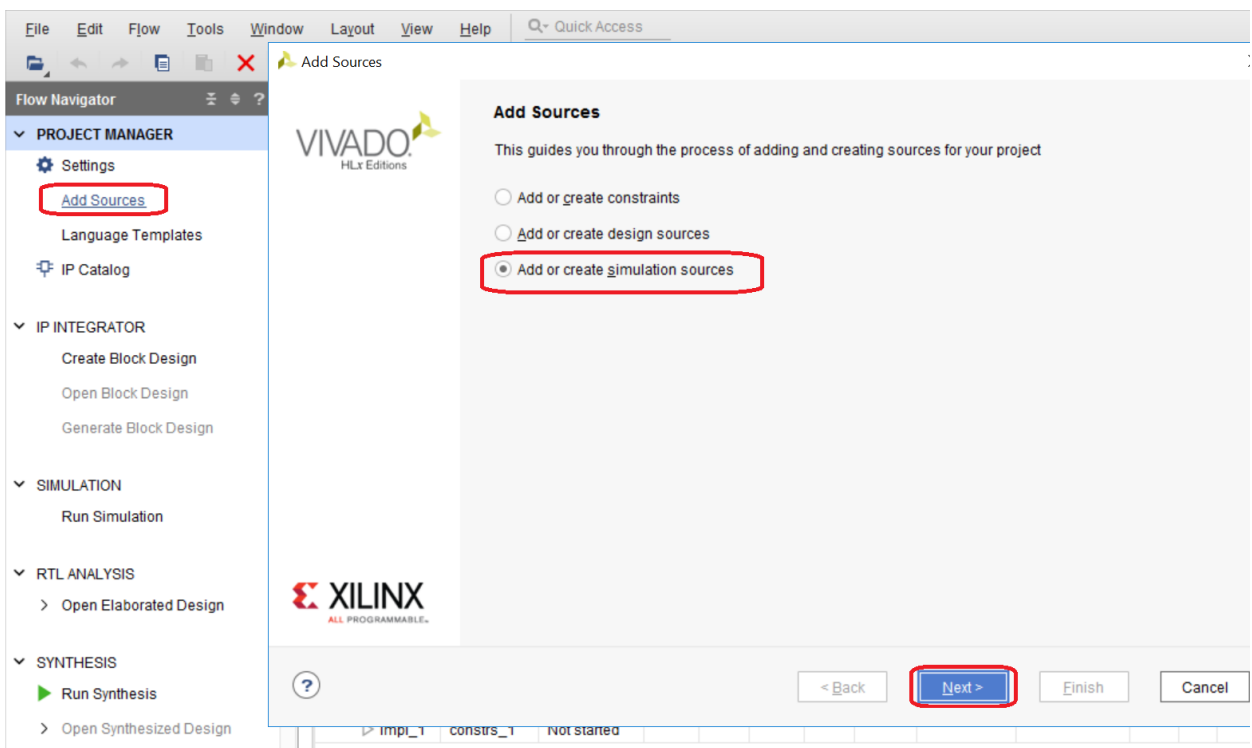
```
module myinv( a, f );
input a;
output f;

assign f = ~ a;
endmodule
```

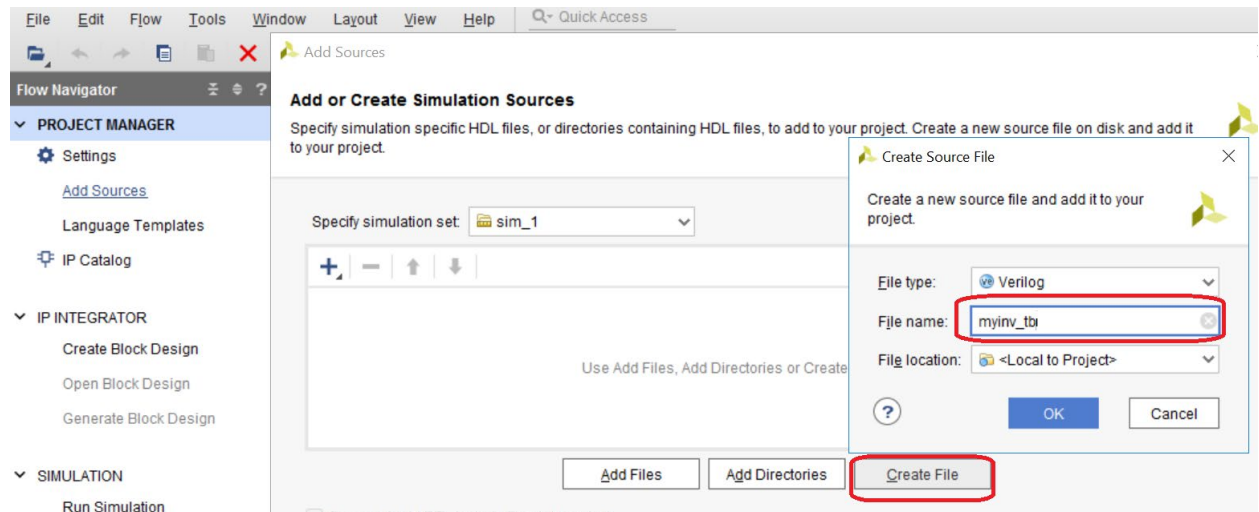
13. Make sure you save the above “myinv.v” file. Click on “File” button on the top of the tool menu -> “Save File”.



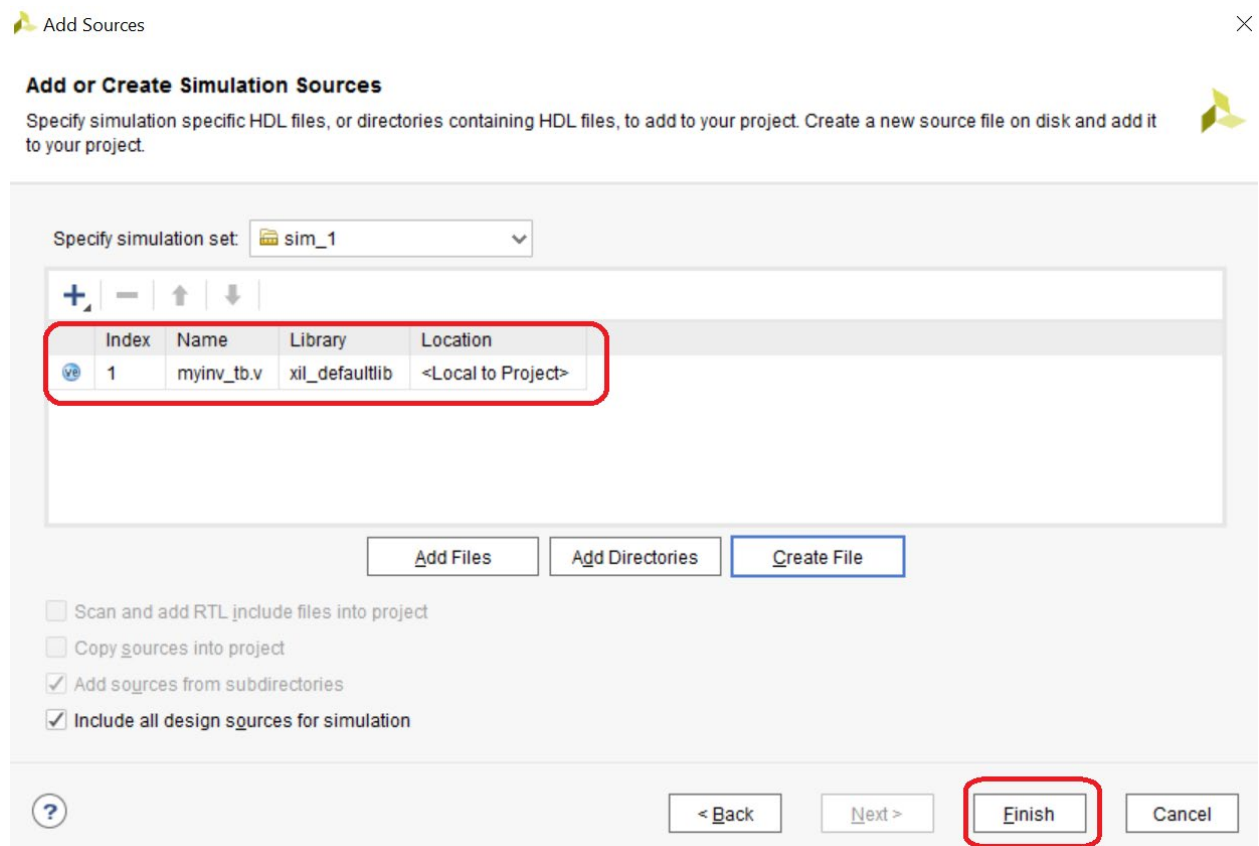
14. Next, you will need to create a simulation file by following the diagram below.
“Add Sources” -> “Add or create simulation sources” -> “Next”.



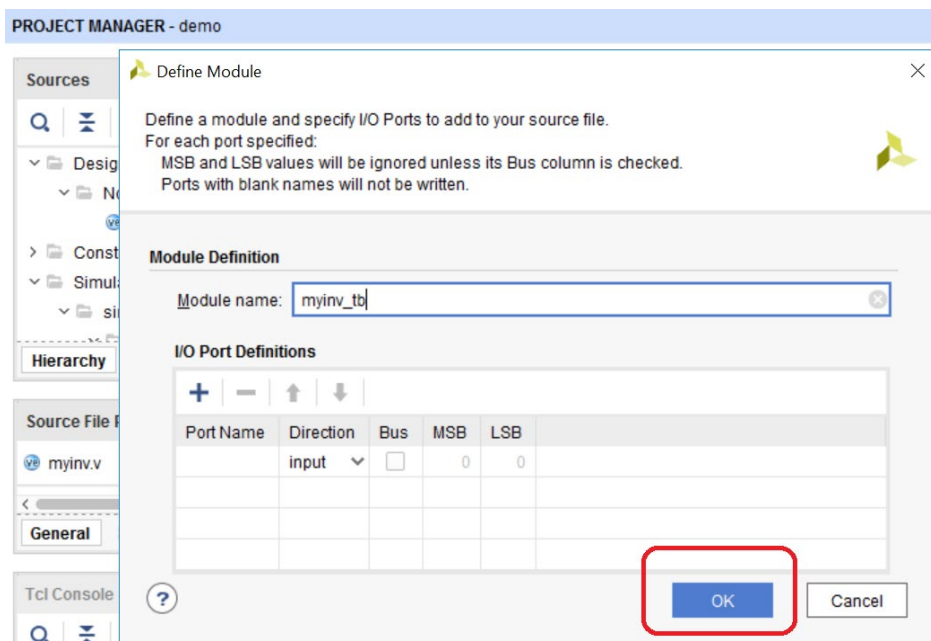
15. In the following popup window, click on “Create File” button, and then enter “myinv_tb” in the “File name” part of the “Create Source File” window shown below -> “OK”.



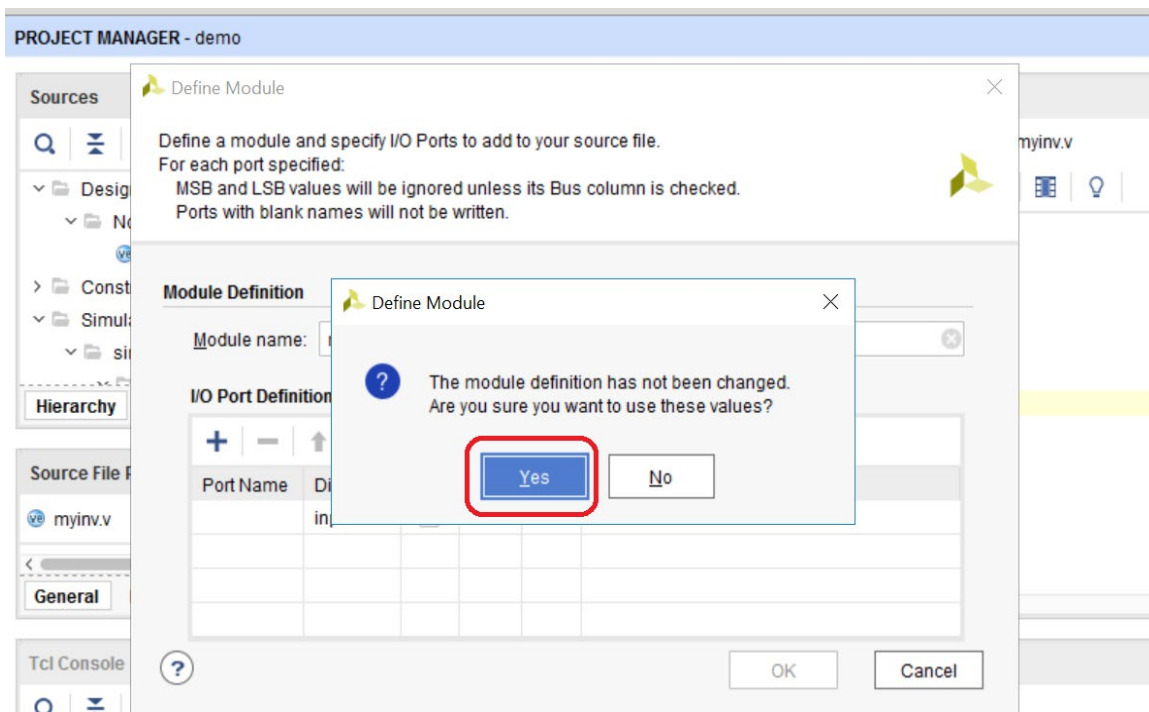
16. In the following window, you’ll see the name of “myinv_tb.v” is added. Next, click on the “Finish” button.



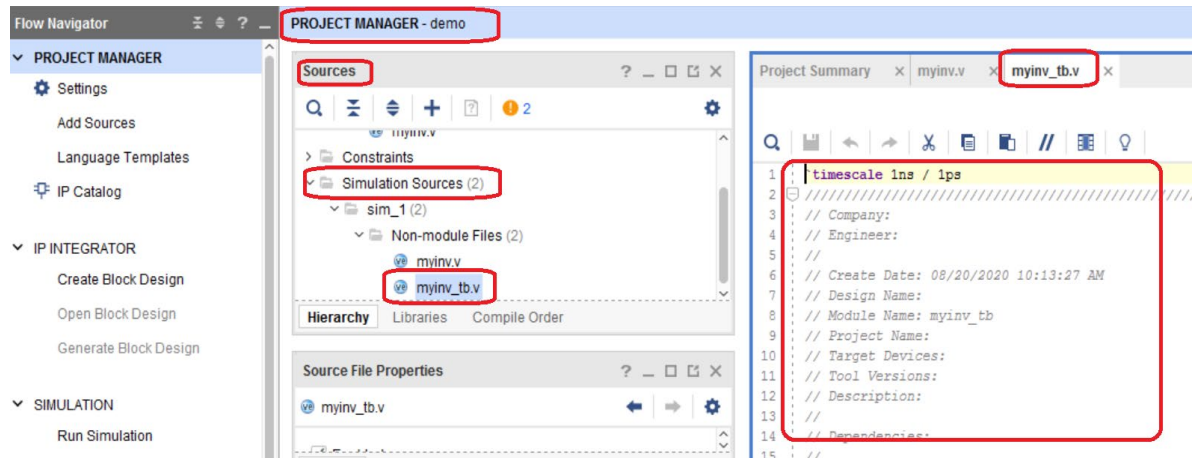
17. In the following popup window, click on “OK” button.



18. In the following popup window, click on “Yes” button.



19. Find the new “myinv_tb.v” shown in the diagram below.



20. Now you can edit the “myinv_tb.v” source code as following:

```
`timescale 1ns / 1ps
module myinv_tb;
reg a;
wire f;
```

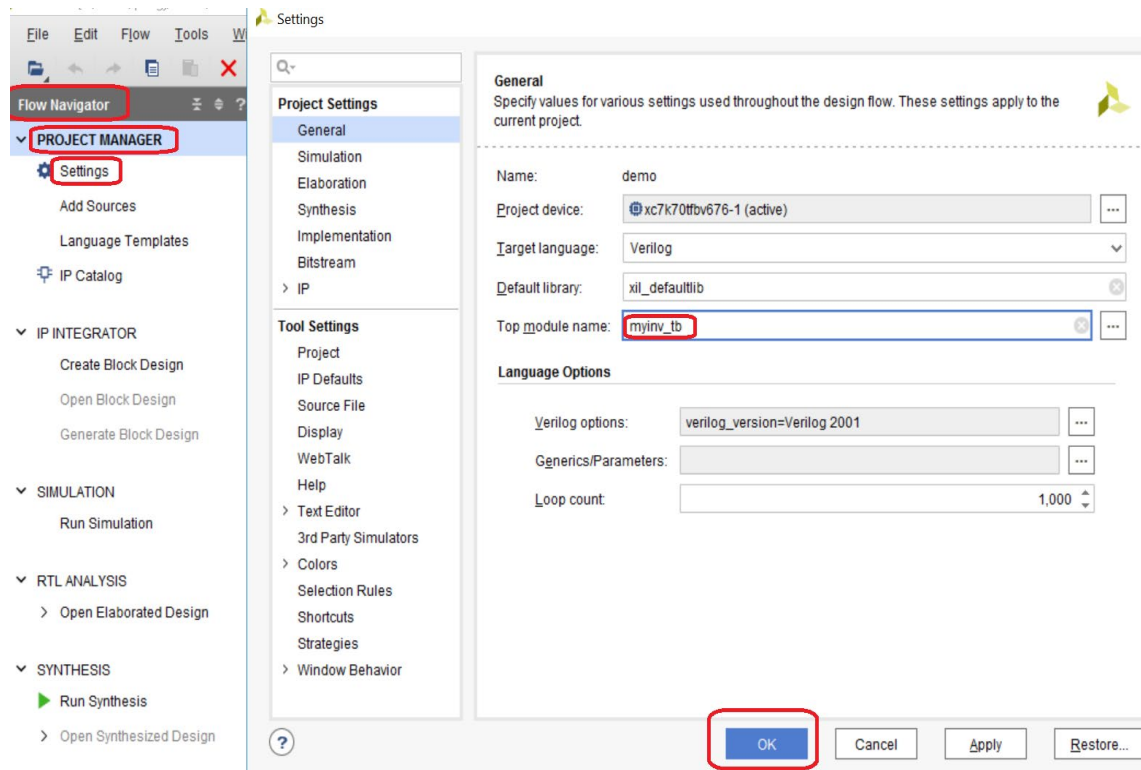
```
myinv uut (a, f);
```

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

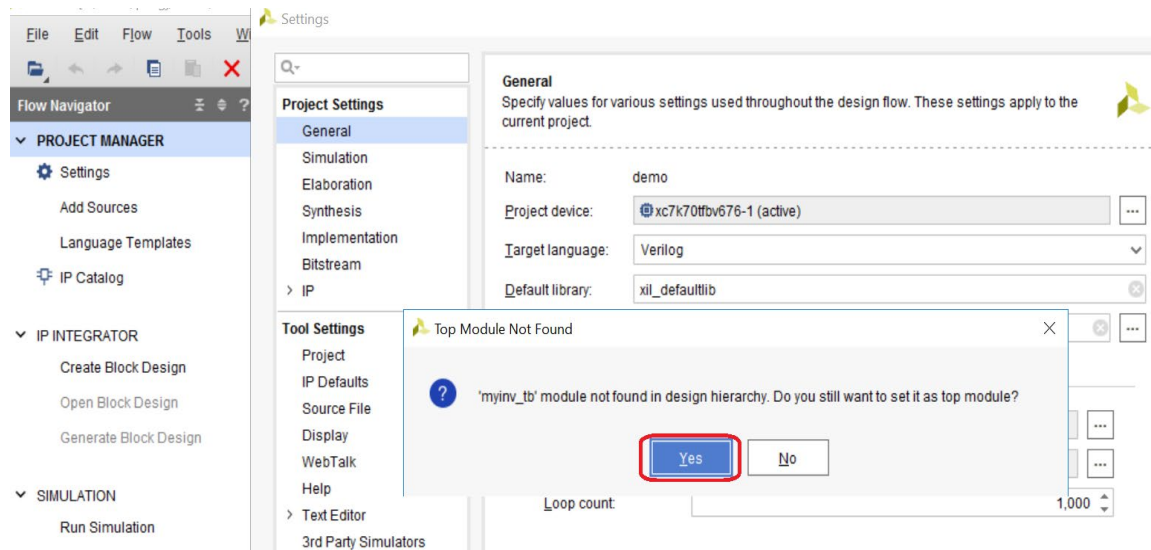
21. Make sure you save the above file after you finish editing the file.

Note, if you make some changes to your code at any other time, you will need to save the file again.

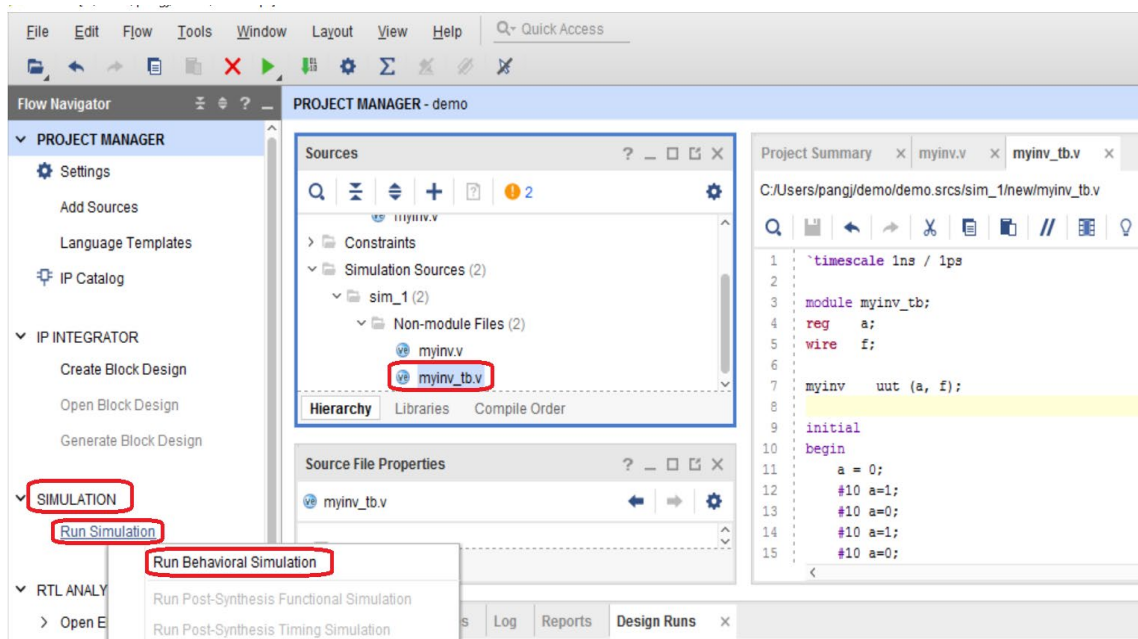
22. In the “Flow Navigator” window, click on “Settings” under “Project Manager”. Next, in the new “Settings” popup window, enter “myinv_tb” as “Top module name” and then click on the “OK” button.



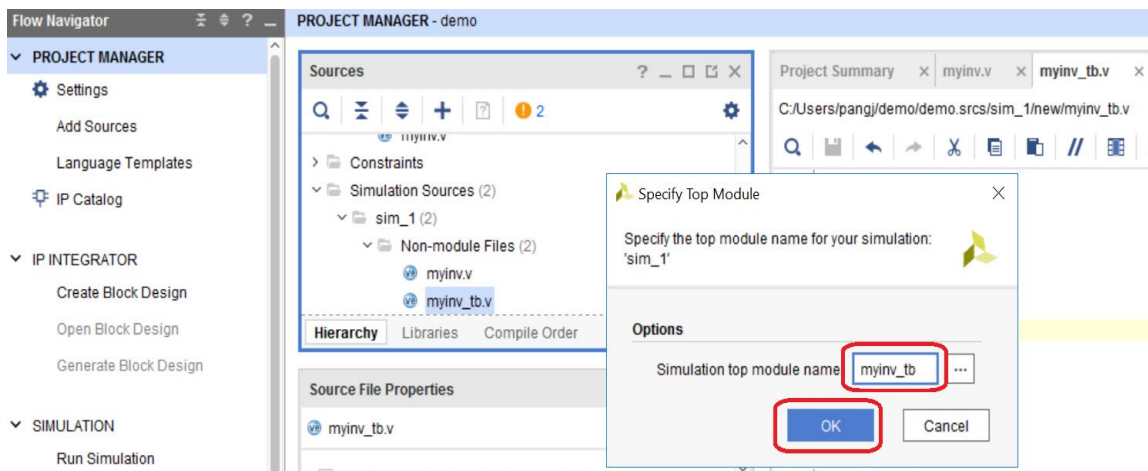
23. In the following popup window, click on “Yes”.



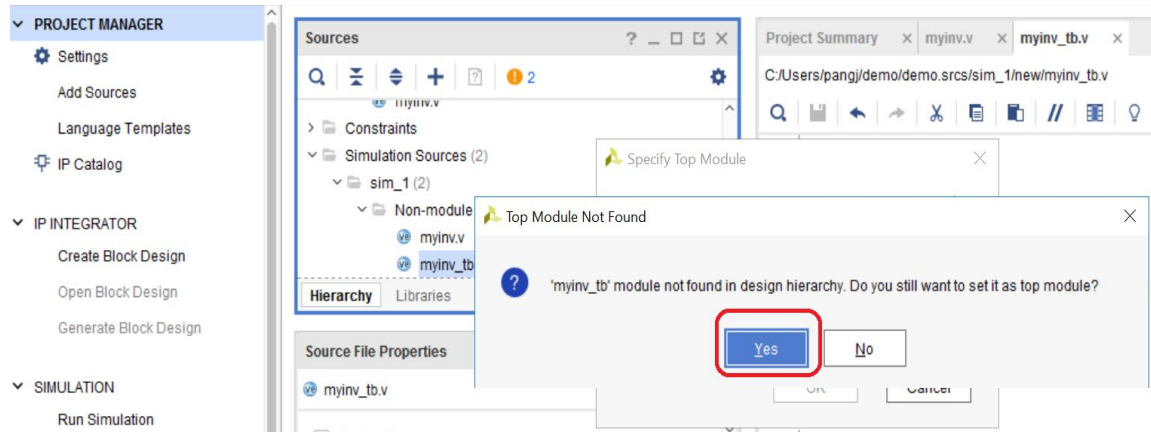
24. Select “myinv_tb.v” located in the Sources window, and then in the “Project Manager” window, click on “Run Simulation” -> “Run Behavioral Simulation”.



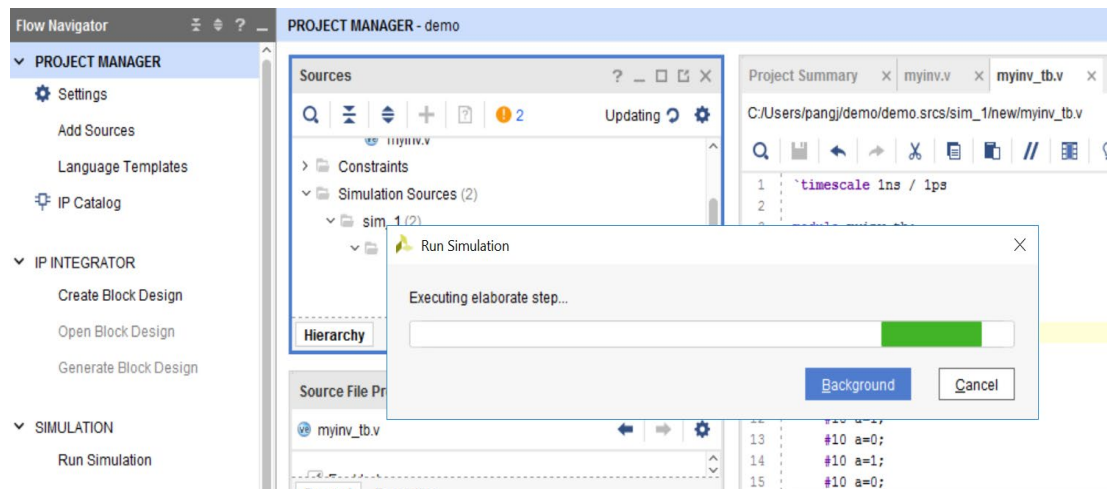
25. Type “myinv_tb” in the “Simulation top module name” part of the popup window, and then click on “OK”.



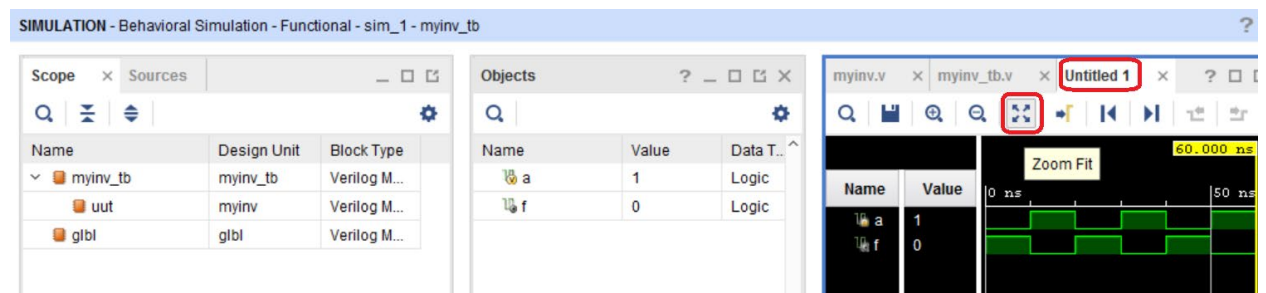
26. Select “Yes” in the following diagram.



27. Next, you will see “Run Simulation” in progress.



28. Click on “Untitled_1” icon shown below, and then click on the “Zoom Fit” button. At last, you will be able to see the simulation waveform shown below.



29. Congratulation! You have successfully completed the simulation!