

[Close this window](#)

## Compiled Messages

[Print](#)[Save as File](#)**Subject:** Debouncing delay**Topic:** Lab Tutorials and Demonstration**Author:****Date:** November 20, 2009 9:34 PM

Hi All,  
I was wondering how long does the oscillation of the PBs take. I looked at XUPV2P doc, but unfortunately couldn't find anything there. I have done the debouncing, but just want to make it more responsive.  
Cheers

[Reply](#)[Forward](#)**Subject:** Re:Debouncing delay**Topic:** Lab Tutorials and Demonstration**Author:****Date:** November 23, 2009 5:28 PM

Hi,

I think the debouncing delay may vary quite a lot from different boards, but depending on how you use the PBs, I would try looking into things like:

- for a software debouncer, avoid loop-like debouncers because they consumes cpu cycles, fragile to compiler optimization and not portable. Try putting the code into a timer interrupt routine.
- if loop-like debouncer is already in use, try decreasing the delay (a few trial and error tests) to a proper value.
- try using a simple hardware debouncer, there should be quite a few HDL samples for that online.

Cheers,  
Jian

[Reply](#)[Forward](#)**Subject:** Compiler code optimization issue...**Topic:** Lab Tutorials and Demonstration**Author:****Date:** November 20, 2009 2:26 PM

In our code we have a state machine with all the different game modes, and in each mode there is a while loop that loops until the state changes. Inside the loop if the user\_input is num1\_key, num2\_key, or num3\_key, the state changes thus breaking the loop. The user\_input is from the UART interrupt handler and is a global variable. State is also a global variable. The problem we are having is that with code optimization on -O2 the compiler essentially thinks that the state can never change once in the loop, because it doesn't think user\_input can change, and likewise it infinitely loops. The code runs fine with optimization off. Here is the code: #define Main\_Menu 1 #define Readme 2 #define High\_Scores 3 #define Difficulty 4 #define Initialization 5 #define Game\_Play 6 #define Pause 7 #define Dev\_Mode 8 #define Num1\_Key\_UART 0x31 #define Num2\_Key\_UART 0x32 #define Num3\_Key\_UART 0x33 #define Num4\_Key\_UART 0x34 #define Num5\_Key\_UART

```
0x35 #define Enter_Key_UART 0x0D #define Esc_Key_UART 0x1B int state = 0; int user_input = 0;
int main() { switch (state) { case Main_Menu: user_input = 0; while (Main_Menu == state) { //
this is the loop that it is getting stuck in switch (user_input[0]) { // with optimization it doesn't
think user_input ever changes case Num1_Key_UART: state = Readme; break; case
Num2_Key_UART: state = High_Scores; break; case Num3_Key_UART: state = Difficulty; break; }
} break; } } void UartRecvHandler(void *CallBackRef, unsigned int UartCounterNumber) { // Uart
details omitted user_input = whatever key pressed by user; }
```

[Reply](#)[Forward](#)**Subject:** Re:Compiler code optimization issue...**Topic:** Lab Tutorials and Demonstration**Author:****Date:** November 20, 2009 2:32 PM

In our code we have a state machine with all the different game modes, and in each mode there is a while loop that loops until the state changes. Inside the loop if the user\_input is num1\_key, num2\_key, or num3\_key, the state changes thus breaking the loop. The user\_input is from the UART interrupt handler and is a global variable. State is also a global variable. The problem we are having is that with code optimization on -O2 the compiler essentially thinks that the state can never change once in the loop, because it doesn't think user\_input can change, and likewise it infinitely loops. The code runs fine with optimization off.

The example code is attached.

Attachments: [code\\_example.c](#)

[Reply](#)[Forward](#)**Subject:** Re:Compiler code optimization issue...**Topic:** Lab Tutorials and Demonstration**Author:****Date:** November 20, 2009 8:37 PM

Okay. So your code has a case statement (implemented using a "switch" in C), not a state machine (big difference =). I'm sort of missing the problem/question though since your code works. You need to use -O2 for some reason? Or you just want to use some optimization? Or maybe you want to understand why this is happening? Could you clarify what you want to know?

Cheers  
Lesley

[Reply](#)[Forward](#)**Subject:** Re:Compiler code optimization issue...**Topic:** Lab Tutorials and Demonstration**Author:****Date:** November 21, 2009 3:05 AM

My guess would be that you need to mark the variable as volatile, ie. (volatile int user\_input = 0;). The reason being, from the compiler's point of view user\_input doesn't change within main() therefore it can be substituted with a constant value. However, the interrupt can occur at any point during the execution of main and update the value of user\_input, but the compiler has no way of knowing this. With the volatile keyword you're telling the compiler some external agent may modify this memory location so load this location each time it's value is needed.

Eric

Reply

Forward

**Subject:** Re:Compiler code optimization issue...

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** November 22, 2009 2:59 PM

Thanks Eric you nailed it!!

Our audio was starting to cut out when we had to many sprites moving on the screen, but by turning optimization -O2 on it fixed the problem :)

Reply

Forward

**Subject:** Re:Compiler code optimization issue...

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** November 22, 2009 8:13 PM

The keyword volatile is a type qualifier. It tells the compiler that the object of this type can be modified in ways that are not predictable from static analysis of the program and thus forces every reference to the object to be genuine and not of a preloaded (e.g. cached) copy. Recalling encs 351, you should remember that all global variables used in interrupts should be qualified as volatile to ensure that you are using the actual variable (not something stored in cache or preloaded in the register file).

Good job Eric. Sorry I didn't understand your question. However, everyone should be careful to ensure that any global variables they use inside ISRs are declared as volatile. Furthermore, any variables memory mapped to device hardware registers may need to be labelled volatile if you are polling them. (Recall encs 351 =)

Cheers  
Lesley

Reply

Forward

**Subject:** REMINDER: 452/894 Lab Test

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 7, 2009 12:49 PM

Hi everyone,

The lab test/demo starts 4:30pm tomorrow, here are some things you should know:

- Please show up and setup your board at least 10 minutes before your demo time.
- Be sure that your bitstreams are built and ready to be downloaded.
- You MUST demonstrate the results (led blinking, waveforms, XMD/terminal printouts etc) to receive marks.
- Please be prepared to answer questions regarding the implementation and design choices.

Good luck.

Jian

Reply

Forward

**Subject:** [LabTest Info] CoreGen IP Simulation with ISE

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 7, 2009 9:54 AM

Hello Everyone,

An important note to the labtest step 3.

If you are using ISE to simulate your testbench in module06, you need to remove the current instance that is automatically included in the design, and manually add the vhd version of the block memory instance, so that the ISE simulator can recognize the CoreGen IP. Details are as follows:

1. remove the instance (file name ".xco") of the Block-memory IP core from your design
2. add the ".vhd" (same file name as the "\*.xco" but with different extension name), there should be a "\*.v" and "\*.vhd" in the same directory generated along with the "\*.xco" file when you use CoreGen.

This is a bug in ISEv10 for not recognizing the \*.xco format, it's fixed in v11.

Cheers,  
Jian

Reply

Forward

**Subject:** Lab Demo - Part 2

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 6, 2009 7:10 PM

Hi,  
just in case you are having problems with the snoopy counter not behaving like you want it to remember that each time you modify and compile the c code the resulting assembly code is modified. Most importantly, the addresses are usually modified and you need to update the corresponding upper/lower bound in the system assembly view and re-generate the bitstream. Hope this helps.  
-Farzad

Reply

Forward

**Subject:** Re:Lab Demo - Part 2

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 6, 2009 8:47 PM

thx i was stuck for a long time not realising my address completely changed

[Reply](#)  
[Forward](#)

**Subject:** Lab demo part 1  
**Author:**

**Topic:** Lab Tutorials and Demonstration  
**Date:** October 6, 2009 11:59 AM

in Part 1 of Lab demo,  
Could you please tell me that blinking at the rate of once per second means one second on and one second off or 0.5 second on and 0.5 second off?

Thanks  
Fatemeh

[Reply](#)  
[Forward](#)

**Subject:** Re:Lab demo part 1  
**Author:**

**Topic:** Lab Tutorials and Demonstration  
**Date:** October 6, 2009 3:00 PM

Hello,

Sorry, the task should have been made more clear. I think the requirement means the LED toggles every second, meaning 1s on and then 1s off. However, if you have already implemented it as 0.5s on and 0.5 off, it is OK for me too, for you obviously have to know how to change the blinking rate.

Jian

[Reply](#)  
[Forward](#)

**Subject:** Lab Demo step2  
**Author:**

**Topic:** Lab Tutorials and Demonstration  
**Date:** October 5, 2009 3:24 PM

Hi, are we supposed to add snoopy to lab module3 or are we supposed to use lab module5 directly?

John.

[Reply](#)  
[Forward](#)

**Subject:** Re:Lab Demo step2  
**Author:**

**Topic:** Lab Tutorials and Demonstration  
**Date:** October 5, 2009 3:53 PM

either way is fine, as long as you can profile the code required.

Jian

Reply  
Forward

**Subject:** Re:Lab Demo step2

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 5, 2009 5:43 PM

Its better to add it to LAB#3,

thanks,

Nikola Cucuk

Reply  
Forward

**Subject:** Question about interrupt

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 5, 2009 2:33 PM

Hi,

In the lab demo we are asked to stop the blinking when we input something to the hyperterminal, we were thinking of connecting the interrupt port of RS232 to the Intr of the interrupt mean that Intr will have the interrupt signal from both Uart and the timer (we set timer to have a higher priority). The problem is in our C code we can only connect to one them when using XIntc\_Connect and having the XIntc\_Connect in sequence will not work since it will overwrite on it.

Are we missing anything?

Thanks

Reply  
Forward

**Subject:** Re:Question about interrupt

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 5, 2009 2:51 PM

Hi,

Just looked up the docs regarding the XIntc\_Connect, it seems the second parameter "u8 Id" is the thing worth looking into.

"Id contains the ID of the interrupt source and should be in the range of 0 to XPAR\_INTC\_MAX\_NUM\_INTR\_INPUTS - 1 with 0 being the highest priority interrupt".

I guess you didn't change the 2nd parameter when calling XIntc\_Connect in sequence, hence the second one overwrite the first.

Cheers,  
Jian

Reply  
Forward

**Subject:** Lab Demonstration step1 question**Topic:** Lab Tutorials and Demonstration**Author:****Date:** October 3, 2009 4:16 PM

Hi, I cannot figure out how to have the code detect any keyboard input from the hyperterminal. I tried to use scanf to detect any input from the hyperterminal, but xilinx does not support it. I realized that printf is actually xil\_printf, so we tried to look for xil\_scanf, but it does not exist.

Could someone give me a hint?  
Jason.

[Reply](#)[Forward](#)**Subject:** Re:Lab Demonstration step1 question**Topic:** Lab Tutorials and Demonstration**Author:****Date:** October 3, 2009 4:44 PM

I am also having the same problem. we tried using getchar() to read in the keyboard character but i think getchar() also steals some important information in the serial port and hence the counter value is wrong from the start

[Reply](#)[Forward](#)**Subject:** Re:Lab Demonstration step1 question**Topic:** Lab Tutorials and Demonstration**Author:****Date:** October 3, 2009 9:59 PM

Hint: we have configured microblaze to use the RS232-UART as both STDIN and STDOUT (as in the first few steps in module1), so you should be looking into the docs of uart and its drivers.

Cheers,  
Jian

[Reply](#)[Forward](#)**Subject:** Re:Lab Demonstration step1 question**Topic:** Lab Tutorials and Demonstration**Author:****Date:** October 4, 2009 12:49 PM

not sure if we are on the right track, but we done it by checking the Status Register of UART..see if there is incoming data available for receiving. You can find the detail of status register in the UART manual.

Sean

[Reply](#)[Forward](#)**Subject:** Lab Demonstration step3 question**Topic:** Lab Tutorials and Demonstration**Author:****Date:** October 2, 2009 3:55 PM

What is it that we are supposed to test for module6? Are we testing with inputs of different switch combinations and outputs of different LED combinations?

Jason.

Reply

Forward

**Subject:** Re:Lab Demonstration step3 question

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 2, 2009 5:09 PM

Yes, that's correct. Test-benches should cover all possible input cases.

Jian

Reply

Forward

**Subject:** Re:Lab Demonstration step3 question

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 2, 2009 5:58 PM

Thanks Jie

Reply

Forward

**Subject:** Re:Lab Demonstration step3 question

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 5, 2009 7:39 PM

Hi,

I am confused here are we making a testbed in SW/HW more precisely using modelSim or making our own HW to test the module?

Nikola Cucuk

Reply

Forward

**Subject:** Re:Lab Demonstration step3 question

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 5, 2009 10:22 PM

Hi

I'm not sure that I get the question. However, you are supposed to create a testbed for your ISE design from Lab #6 to run in Modelsim. I don't understand the SW/HW question. There's no uBlaze, but this isn't supposed to run on the V2Pro board. Please clarify your question.

Cheers

Lesley

Reply

[Forward](#)**Subject:** Re:Lab Demonstration step3 question**Topic:** Lab Tutorials and Demonstration**Author:****Date:** October 6, 2009 10:49 AM

Hi,

You have answered my question which was badly written as usual hahah.

My question is should we do the testbed in modelSim. The Hardware software part was just asking should we make our own HW to be the testBed or do we do it in SW meaning modelSim. That is all.  
Thanks,

Nik

[Reply](#)[Forward](#)**Subject:** Re:Lab Demonstration step3 question**Topic:** Lab Tutorials and Demonstration**Author:****Date:** October 6, 2009 2:53 PM

Hi  
I want to add that you can use either ISE or ModelSim for the simulation, whichever one you prefer.

Jian

[Reply](#)[Forward](#)**Subject:** Re:Lab Demonstration step3 question**Topic:** Lab Tutorials and Demonstration**Author:****Date:** October 6, 2009 3:13 PM

O ok then so i will just make a simulation in ISE then. Thanks,

Nik

[Reply](#)[Forward](#)**Subject:** Time Slot for Lab Test and Milestone Demo**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 30, 2009 11:35 AM

Hi everyone,

Please go to the course content page and pick a time slot for your lab test and milestone demo, and make sure to sign up the SAME slot with your team partner.

Cheers,  
Jian

Reply  
Forward

**Subject:** Lab 3  
**Author:**

**Topic:** Lab Tutorials and Demonstration  
**Date:** September 26, 2009 5:30 PM

Dear Jian  
In part 8 of Lab.3, we run and then stopped the program and saw where the processor stopped.  
But we do not know how can we Reset the PC to the start address (using rwr)!!!  
rrd could not read from the stop address, it seems it is not register address

Thanks alot,  
fatemeh

Reply  
Forward

**Subject:** Re:Lab 3  
**Author:**

**Topic:** Lab Tutorials and Demonstration  
**Date:** September 26, 2009 5:48 PM

I think you want to do a  
rwr pc 0x0  
to reset the program counter to 0x0.

Hope it helps,  
Kyle.

Reply  
Forward

**Subject:** Re:Lab 3  
**Author:**

**Topic:** Lab Tutorials and Demonstration  
**Date:** September 27, 2009 10:38 PM

Hello Fatemeh,

Kyle is right. And you can also type the "help" within the XMD to check the syntax of the available commands: typing "help" should give you available categories, e.g. "running" and "files", typing "help running" gives you the commands within the "running" categories.

Jian

Reply  
Forward

**Subject:** Module6 setp 15  
**Author:**

**Topic:** Lab Tutorials and Demonstration  
**Date:** September 24, 2009 4:08 PM

Hi,

I am stuck on Module6 setp 15,  
I got the error after i run "implement Design processes"  
and i got error as below:

```
ERROR:ConstraintSystem:59 - Constraint <NET PB_ENTER LOC = AG5;>  
ERROR:ConstraintSystem:59 - Constraint <NET PB_UP LOC = AH4;>  
ERROR:ConstraintSystem:59 - Constraint <NET PB_DOWN LOC = AG3;>  
ERROR:ConstraintSystem:59 - Constraint <NET PB_LEFT LOC = AH1;>  
ERROR:ConstraintSystem:59 - Constraint <NET PB_RIGHT LOC = AH2;>  
ERROR:ConstraintSystem:59 - Constraint <NET PB_ENTER IOSTANDARD = LVTTTL;>  
ERROR:ConstraintSystem:59 - Constraint <NET PB_UP IOSTANDARD = LVTTTL;>  
.....
```

look like there's error regarding push buttons and system clock, BUT not the LED's...I wonder what is the issue?

[Reply](#)[Forward](#)

**Subject:** Re:Module6 setp 15

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** September 24, 2009 5:49 PM

nvm, i think i mapped the wrong names myself, "PB\_ENTER" and such has to be the names declared in example\_verilog.v files~

Sean

[Reply](#)[Forward](#)

**Subject:** Re:Module6 setp 15

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** September 27, 2009 10:51 PM

Hi,

Yes, you should make sure the pin names in the ucf files correspond to the those listed in the "System Assembly View", and preferably to the names in your HDL files too.

Cheers,  
Jian

[Reply](#)[Forward](#)

**Subject:** Re:Module6 setp 15

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** September 24, 2009 7:13 PM

Check to see if you Net or the Name in your .ucf file.  
Cheers

[Reply](#)  
[Forward](#)

**Subject:** Re:Module6 setp 15

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** September 28, 2009 3:20 PM

Hi guys,  
Have you added the system clock ?  
Cheers,  
Piraj

[Reply](#)  
[Forward](#)

**Subject:** Lab5 Step7

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** September 24, 2009 10:07 AM

Hi, if you cannot find Trace\_PC and Trace\_Valid\_Instr ports in MicroBlaze, you need to right click microblaze\_0, Filter Ports, and select All. Then you will see all the ports available.

John.

[Reply](#)  
[Forward](#)

**Subject:** Lab3 adding SW\_1

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** September 22, 2009 5:53 PM

Hi

I noticed somepeople (including my group) have stucked on step 9 for while trying to add SW\_1 to control LED flashing. Just to help clearfy, the "SW\_1" mentioned in the instruction is not the push button SW1, it is the actual "switch" located beside the UP DOWN LEFT RIGHT buttons. (where it label USER INPUT)

Hope this help some people who are also confused by the name.

Cheer,  
Sean

[Reply](#)  
[Forward](#)

**Subject:** Re:Lab3 adding SW\_1

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** September 22, 2009 5:57 PM

ya, thanks for the heads up sean.  
also, switch1 is the second one from the right.

-----

||||

-----  
3 2 1 0

Thanks,  
Jason

Reply

Forward

**Subject:** Lab tutorial3 Step6

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** September 21, 2009 4:35 PM

Hi, I am having problem with running the lab3.c code.

I successfully compiled the project. The debugger is stuck in a infinite loop in the following function in xbasic\_types.c:

```
void XAssert(char *File, int Line)
```

There is a comment instructing me to use debugger to disable the waiting. I can get out of the infinite loop by changing the value of XWaitInAssert from 1 to 0 using the debugger. Am I supposed to do that?

After getting out of the infinite loop, a warning window pops out. The warning says "Program received signal SIGTRAP, Trace/breakpoint trap".

No matter how many times we run, the debugger is stuck at the following line:  
microblaze\_enable\_interrupts();

Does anyone have suggestions?  
Thank you.  
John Ogawa.

Reply

Forward

**Subject:** Re:Lab tutorial3 Step6

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** September 21, 2009 8:30 PM

Hello John,

It's difficult to tell what's wrong based on your post.

My suggestion is to try to read and understand the code and try to fill in the blanks with something that MAKES SENSE, instead of just making the code compile. The code in lab3 is fairly well written and shouldn't take much time.

The purpose of lab3 is to guide you through looking up the EDK driver references, so it is not a good idea to start patching the holes that the debugger indicates (which might well be meaningless) before you know why.

Jian

[Reply](#)[Forward](#)**Subject:** Re:Lab tutorial3 Step6**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 22, 2009 7:04 AM

Hello

Also, note that the debugger has multiple different ways of "stepping" through the code: next, step, continue (to breakpoint). It's probably not your problem, but trying different methods of moving through the code might help you figure out what your problem is.

Cheers

Lesley

[Reply](#)[Forward](#)**Subject:** Lab 2 Step 16**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 19, 2009 1:07 PM

what does it mean by "The GPIO data sheet states that the GPIO DATA register is at offset 0x00 and the GPIO

TRI register is at 0x04. Previously, you set the base address of the GPIO to 0x80040000. Add these numbers to determine the register addresses."

and i was unable to write to the address and the value is 00000001 is that normal? thanks

[Reply](#)[Forward](#)**Subject:** Re:Lab 2 Step 16**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 19, 2009 1:15 PM

ohh never mind i found out that the tri-state register is the only one that is writable using XMD

[Reply](#)[Forward](#)**Subject:** Re:Lab 2 Step 16**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 19, 2009 6:59 PM

I think you can actually write to both, but only when the data is set to an output.

Kevan

[Reply](#)[Forward](#)**Subject:** Re:Lab 2 Step 16**Topic:** Lab Tutorials and Demonstration

**Author:****Date:** September 19, 2009 8:52 PM

You were unable to write to the data register because you were not doing it right.

I think you should re-read the GPIO data sheet, especially the part concerning the register settings.  
Hint: on page 11 to 12.

Jian

[Reply](#)[Forward](#)**Subject:** EDK & ISE tools in Lab-1A**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 17, 2009 4:00 PM

Hello everyone,

We are currently installing and testing the latest version 10.1.03 of EDK and ISE tools on the computers in LAB-1A, hopefully they will be ready for use by tomorrow.

But if you are going to start your tutorial labs in LAB-1A anyway, please be advised that the tools version are still 10.1.02, they should work fine with the tutorial labs, as the labs are prepared in this version. However, since the peripheral libraries and documentation on Xilinx website have all been updated to 10.1.03, the MIGHT be some compatibility issues.

Also, for those who are going to install a copy of the tools on their personal computers, please drop by at ASB8803.1 and sign the Xilinx Tools Agreement. The size of the tools is about 7.5GB, so remember to bring a large drive.

Cheers,  
Jian

[Reply](#)[Forward](#)**Subject:** Re:EDK & ISE tools**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 17, 2009 11:25 AM

Hello all,

I'm sorry to tell you that there has been some delay installing the tools on the lab-1a computers, as we have to wait for confirmation that all the tools (including those used in other courses) required have been installed properly on the source machine before cloning it to other work stations.

Pang, our lab technician, is currently working on the cloning process, normally it takes 4-5 hours. So if everything goes fine, we should have the tools ready for your first lab session by 4:30pm. See you then.

Jian

[Reply](#)[Forward](#)**Subject:** Re:EDK & ISE tools**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 17, 2009 4:00 PM

Hello everyone,

Pang has finished installing the latest version of tools in the lab-1a computers. =)

a reminder: for those who haven't had a LAB-NET account (the account for you to logon to the lab-1a computers), please go to Pang's office (ASB9878) and follow the instructions on his door.

Cheers,  
Jian

[Reply](#)[Forward](#)**Subject:** Installation of EDK/ISE Tools**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 15, 2009 2:21 PM

Hello

For those who are going to install the tools on their personal computers, here are a few notes for installation:

- the folder/package you copied contains three subfolders: edk, ise and Linux\_Packs, you ONLY need the edk and ise folder (assuming most of you are using windows, and it's compatible with the computers in lab-1a)
- read the installation document "ise\_install\_guide.pdf" in the edk subfolder
- install edk first and then ise
- the serial for registration is in the "edk" subfolder with the name "Entitlement\_Center\_Reg\_info.pdf"
- this is the installation for version 10.1, so after the installation, click check for updates and download ALL the patches. There are about 2GB+ of download, so if you have a slow connection at home, it is better to download the patches within the SFU network (assuming you have a laptop =)
- check the version from the menu Help->About

Please post in this thread if you have problems installing the tools.

Cheers,  
Jian

[Reply](#)[Forward](#)

**Subject:** Re:Installation of EDK/ISE Tools**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 16, 2009 3:34 PM

Hi I have tried to install the edk but it seem that the setup up file doesn't run.

It seem that some people experienced the same problem, is it possible the file is corruptd? becuase ISE setup file actualy run properly.

Thanks,  
Sean

[Reply](#)[Forward](#)**Subject:** Re:Installation of EDK/ISE Tools**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 16, 2009 10:17 PM

Hi Sean,

I just to run the installer in xp mode in windows virtual PC. (in windows 7)  
I got the installer wizard pop up. I didn't go on installing but it did run.

Seems like the issue is from the OS side. Are you using Vista or Windows7?

Thanks,  
Jason Tsai

[Reply](#)[Forward](#)**Subject:** Re:Installation of EDK/ISE Tools**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 16, 2009 10:54 PM

Thanks Jason.

I should have added that the EDK/ISE tools are tested under Windows XP, running as a native OS. If anyone has experience/info installing the tools on Windows Vista / Windows 7, it would be appreciated if you could share it in this thread.

I would not recommend installing the tools in a virtualized OS, because

(i) some parts of the tool flow are resource intensive so you are likely to observe performance downgrade within the virtual OS, and

(ii) you would have to consider the interfacing issues from the guest OS to the host OS and then to the board, should you encounter any connection problems.

Jian

[Reply](#)

[Forward](#)**Subject:** Re:Installation of EDK/ISE Tools**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 16, 2009 10:36 PM

Hi Sean,

Could you be more specific about how the installer "didn't run"? Were there any errors/msg relating to that?

It might worthwhile checking that you are running the installer under the folder "edk\edk\setup.exe", instead of the other one under folder "edk\edk\bin\nt\setup.exe".

Jian

[Reply](#)[Forward](#)**Subject:** Re:Installation of EDK/ISE Tools**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 16, 2009 11:18 PM

Alas, version 10.1 of the tools doesn't run on Vista. Sorry guys. I don't use it, so I didn't think to mention it. Hopefully most of you have XP or can partition your disk space to install it. I don't know how virtual xp will work, but Jian is right - it will be slower.

Cheers  
Lesley

[Reply](#)[Forward](#)**Subject:** Re:Installation of EDK/ISE Tools**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 17, 2009 2:03 AM

Hi Jian

The installer "didn't run" as nothing happen after I double clicked the setup.exe in edk\edk\.

After testing with Jason and confirmation from prof, I guess I will try to install it under windows XP instead.

Thanks for help!  
Sean

[Reply](#)[Forward](#)**Subject:** Re:Installation of EDK/ISE Tools**Topic:** Lab Tutorials and Demonstration**Author:****Date:** October 12, 2009 2:53 PM

I was able to install it on my computer running Windows server 2008 R2 (windows 7 server) but

since this is only 64bit the driver for the board doesn't seem to work properly. Everything else works but I can't upload the bitstream to the board :-(  
Also I tried to run it with XP in Virtualbox but it was very unstable and it BSOD'ed my computer randomly when the board was connected.

You can actually get it to install under vista/windows 7 as long as you are running a 32bit version. All you need to do is run the installer file (setup.exe) directly from the "edk/bin/nt" folder.

-Farzad

Reply

Forward

**Subject:** Re:Installation of EDK/ISE Tools

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 15, 2009 1:20 AM

I managed to get the Xilinx tools to work flawlessly under Windows 7 32bit, which confirms the above.

The only difficulties I ran into were with the Xilinx updater - it kept freezing up. However, with a little patients, and use of "windows XP compatibility mode" when running the updater, it eventually works.

Sonca

Reply

Forward

**Subject:** Re:Installation of EDK/ISE Tools

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 15, 2009 1:21 AM

oops typo; i meant to say patience, not patients. =)

Sonca

Reply

Forward

**Subject:** Re:Installation of EDK/ISE Tools

**Topic:** Lab Tutorials and Demonstration

**Author:**

**Date:** October 19, 2009 12:31 AM

yeah the updater looks like it gets stuck (mine stayed at 10% for about half an hour) but it's actually working. Just give it plenty of time and remember that it's downloading ~2GB worth of updates. If you are unsure if the updater is stuck take a look in one of the ".xinstall" folders (there are three of these folders: in the base Xilinx directory, in the EDK dir and the ISE dir). This should be where the updater is downloading the files.

-Farzad

Reply

[Forward](#)**Subject:** Info about Lab1**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 15, 2009 12:16 PM

Hi,

I'll be posting information relating to the tutorial lab 1 here, so if you have questions or useful information to share about lab 1, please try to post in this thread.

- Peripheral Library

The tools (10.1.02 and 10.1.03 ) seem to have some compatible issues with the peripheral lib that comes with the CD (lib\_rev\_1.1.zip), so please go to the xupv2p website and download the latest version.

- Where to place your project files

Please put your modules in the local hard drive (e.g.. C:\temp) instead of the net work drive (H:), because there used to be permission problems for the H: drive and it slows down the tools for its limited read/write speed.

Please feel free to reply to this thread if you find any other issues regarding lab1. thanks

Cheers,  
Jian

[Reply](#)[Forward](#)**Subject:** About the Tutorial Labs**Topic:** Lab Tutorials and Demonstration**Author:****Date:** September 15, 2009 11:22 AM

Hello,

The files and docs about the tutorial labs could be found on the course website

<http://www.ensc.sfu.ca/~lshannon/courses/ensc452/index.html>  
under "Lab Information".

Please down load both the pdf and the zip files for the tutorial labs.

Cheers,  
Jian

[Reply](#)[Forward](#)[Close this window](#)