

## iCE40HX8K-EVB and iCE40HX1K HOW-TO GUIDE FOR WINDOWS

This document describes the different steps to write Verilog code and program the Olimex iCE40HX8K-EVB and iCE40HX1K-EVB boards under Windows environment. We also describe the changes needed to operate the iCE40HX1K demo programs on the iCE40HX8K.

Written by Benoit BOUCHEZ (aka BEB) - (email : benoit -at- imodular-synth -dot- com)  
Document version 1 - August 2017

Even if we tried to make this document error free, you may find errors or missing informations. Don't hesitate to report any issue to us, so we can update the document.

Note that we can not provide support for the Olimex boards, iCECube software, various utilities, etc... so any question sent to the previous email will **not** be answered. Please use Olimex forum for any question related to the support.

### **Step 1 : Installation of iCEcube2**

First of all, you will need to download two files from Lattice website :

- iCEcube2 installation program
- iCEcube2 license file


iCEcube2 is not open source and it requires a license to operate (the good thing is that you can get the license for free)

Before you can download anything from Lattice website, you will need to create a user account (it's free too). Click on "Register" on top bar of Lattice's website and follow all instructions. Note that you will need to click on the link provided in the confirmation email, so you need to use a valid email address.

Once you are registered on Lattice website, go to the following address :

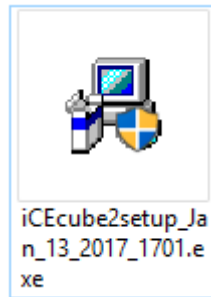
<http://www.latticesemi.com/Products/DesignSoftwareAndIP/FPGAandLDS/iECube2.aspx>

Scroll down the whole page to reach the following table

| Quick Reference  |   | Information Resources  |           | Downloads  |          |          |
|--|---|--|-----------|--|----------|----------|
| <a href="#">Application Note</a><br><a href="#">Installation Guides</a><br><a href="#">Technical Briefs</a><br><a href="#">User Manual</a> |   | <a href="#">Product Brochure</a><br><a href="#">Release Notes</a><br><a href="#">Tutorials</a> |           | Downloadable Software  |          |          |
|   | TITLE ▾                                     | NUMBER ▾   | VERSION ▾ | DATE ▾   | FORMAT ▾ | SIZE     |
| <input type="checkbox"/>   | <a href="#">iECube2 2017-01 for Linux</a>   |  | 2017-01   | 2/21/2017  | TGZ      | 361 MB   |
| <input type="checkbox"/>   | <a href="#">iECube2 2017-01 for Windows</a> |  | 2017-01   | 2/21/2017  | ZIP      | 727.8 MB |
| <input type="checkbox"/>   | <input type="checkbox"/> Select All         | <input type="button" value="Notify Me of Changes"/>  |           | <input type="button" value="Download Selected as Zip File"/> |          |          |

Click on the link "iECube2 xxxxx for Windows" (the numbers in the middle are the version code and may change). You will then reach the license agreement screen. Check the box under the license text and click "Download". The download of the installation program will then start (warning : the zip file is quite big – more than 700MB !)

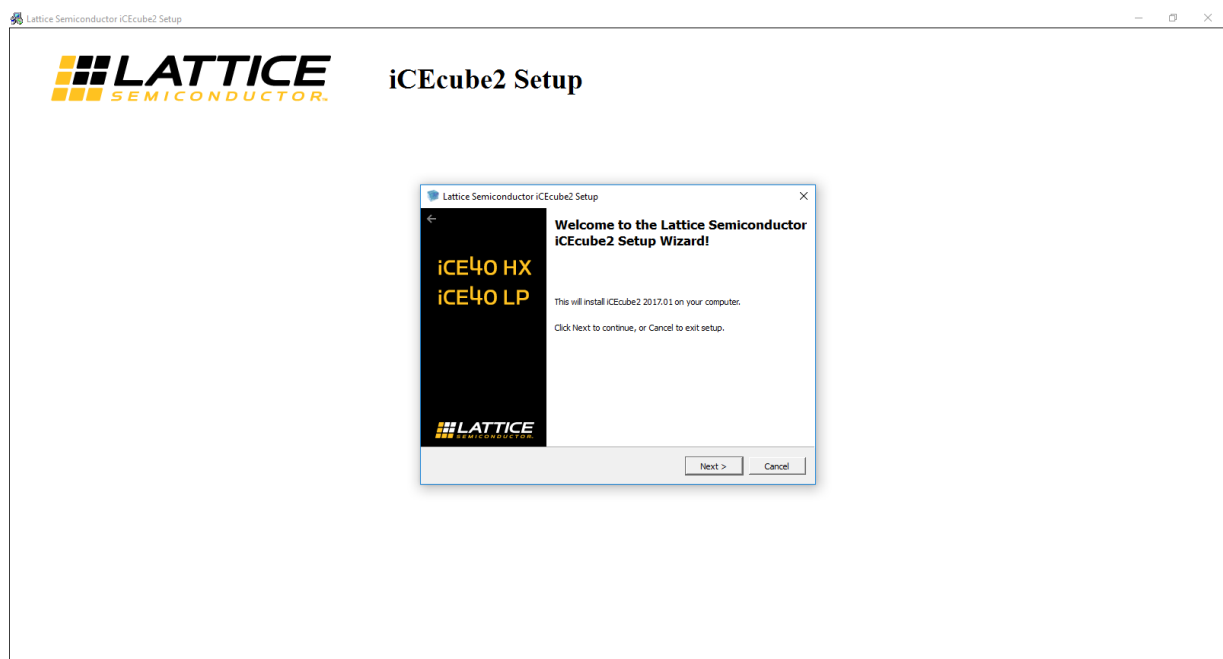
Once the download is finished, you will have the installer ready.



Note that Lattice indicates that iCEcube2 is designed to run under Windows XP and Windows 7. We use it under Windows 10 without any problems, so it can be assumed that it runs also correctly under Windows 8, but we did not test it.

Start the installer by double-clicking the icon and follow the instructions.

IMPORTANT : the iCEcube2 installer must be executed with administrator's rights.



Once the installation is finished, you should have the iCEcube2 icon on your desktop window.



DO NOT START iCEcube2 for now! Since you don't have yet a license, the program will abort.

The license is physically attached to your computer and uses the network card MAC address to identify it. So the first thing to do is to retrieve the MAC address of the machine on which the software will run.

Press and hold the Windows key, and press the "R" key. In the window that opens, enter "cmd" (without the quotes) and press Enter. A shell window (sometimes called "MS-DOS window", but it is **not** a MS-DOS window at all...) will appear.

Type "ipconfig /all" command at the prompt

```
C:\WINDOWS\system32\cmd.exe
Microsoft Windows [version 10.0.15063]
(c) 2017 Microsoft Corporation. Tous droits réservés.

C:\Users\Benoit>ipconfig /all
```

A list of all the network adapters available on your computer will appear. We recommend you to choose the MAC address of either the WiFi adapter or the physical network card.

It is not recommended to use virtual network cards addresses (from VMWare or Virtual Box for example) except if you know exactly what you. Since these addresses can be modified at any moment, they may invalidate the iCEcube2 license file (and you may loose countless hours to understand why the license file is not working anymore)

The MAC address is displayed on the line "Physical address", it is formed of 6 digits, separated by hyphens (sorry, my computer is in French... and yes I have hidden the three last digits of my address).

```
Carte Ethernet Ethernet :

Suffixe DNS propre à la connexion. . . :
Description. . . . . : Realtek PCIe GBE Family Controller
Adresse physique . . . . . : 30-5A-3A-
DHCP activé. . . . . : Non
```

Go back to Lattice website and locate the "Request license" section on the iCEcube2 download page.

## Licensing

### iCECube2 Software Free License

To request a license you will need the following:

- Lattice website user account
- Physical MAC address (12-digit hexadecimal value)

[Click here to request your license.](#)



## Software Downloads & Documentation

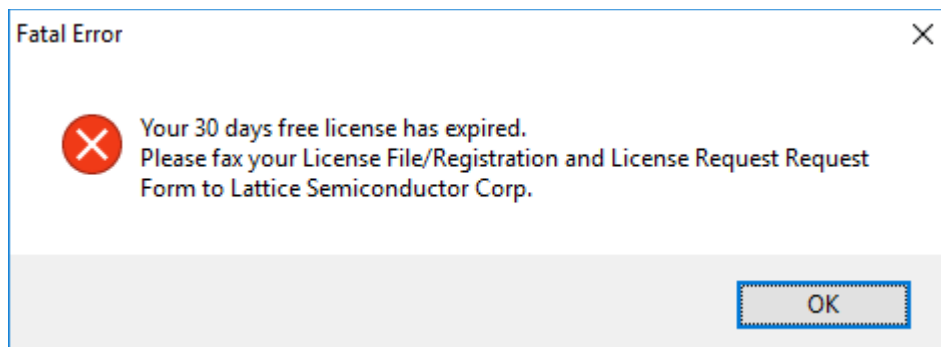
| Quick Reference                     | Information Resources            | Downloads                             |
|-------------------------------------|----------------------------------|---------------------------------------|
| <a href="#">Application Note</a>    | <a href="#">Product Brochure</a> | <a href="#">Downloadable Software</a> |
| <a href="#">Installation Guides</a> | <a href="#">Release Notes</a>    |                                       |

Click on "Click here to request your license". On the next page, check that you have provided your Lattice's account details and enter the MAC address you have retrieved on the computer.

Do not forget to check the "I am not an employee of Cadence, etc, etc..." and click on "Generate license"

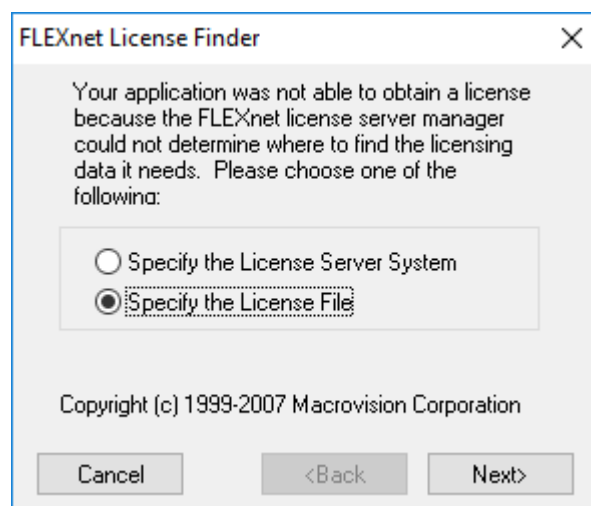
The license file will be sent to the email address you gave in your Lattice account (that's why you need a valid email address). The file is called license.dat. Save it on your computer hard disk, we will need it very shortly.

Here we are finally, we can start iCEcube2... and get a funny window, just to be scared...

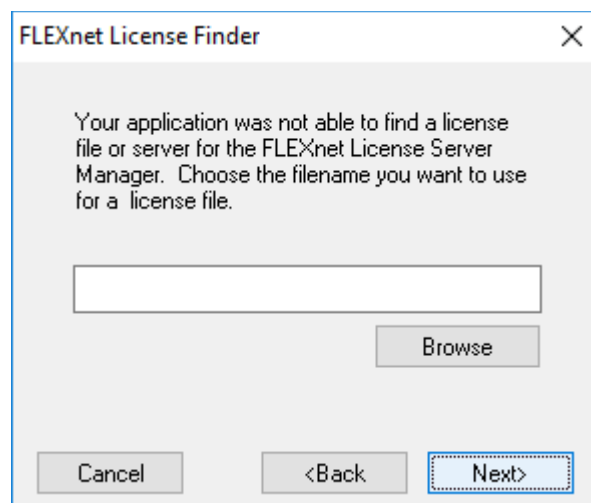


Don't worry, this window is displayed almost each time iCEcube2 is started, but it shall be ignored.

Click on the OK button to close the window, you will then get the following window :

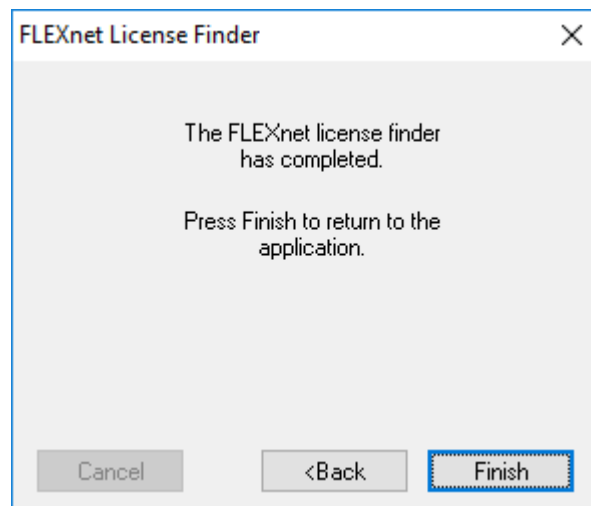


Choose "Specify the license file" and click on "Next"

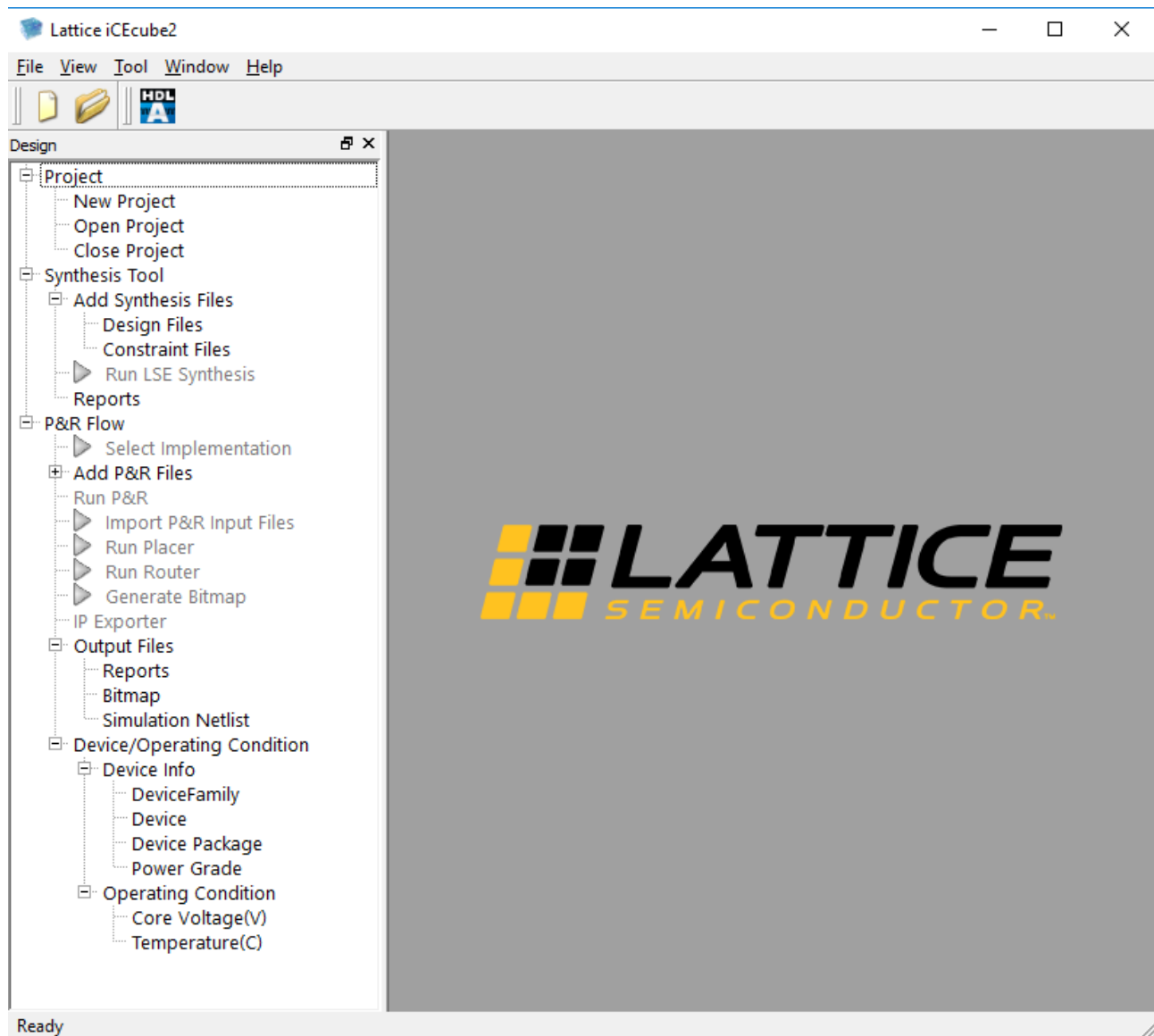


Click on "Browse" and locate the license you got by email just before. Once the license file

location is entered, click on Next, and you should get the following window, which confirms that your license file has been accepted.



Click "Finish" to close the window, iCEcube2 will start automatically.



## **Step 2 : Compilation of your first project**

We will now see how to generate the FPGA bitmap (the file to load in the Flash on the Olimex iCE40HX8K or iCE40HX1K boards)

In iCEcube2, double click on "New project" on the left tree window (single click will not work!)

On the window that opens, enter :

- the name of the project you want to create (spaces are forbidden !)
- the location of the project on your hard disk.
- iCE40 for the device family

If you work with the iCE40HX8K-EVB board, select :

- H8K for the device
- CT256 for the package

If you work with the iCE40HX1K-EVB board, select :

- H1K for the device
- VQ100 for the package

**Let all the other parameters as they are by default!**

New Project

Project

Project Name: OlimexBlinkDemo

Project Directory: D:\FPGA\Test\_Olimex

Device

Device Family: iCE40

Device: HX8K

Device Package: CT256

Operating Condition

Junction Temperature (in degrees Celsius)

Range: Commercial Best: 0 Typical: 25 Worst: 85

Core Voltage(V)

Voltage Tolerance Range: +/-5%(datasheet defau Best: 1.26 Typical: 1.2 Worst: 1.14

IOBank Voltage(V)

topBank 2.5 bottomBank 2.5

leftBank 2.5 rightBank 2.5

Perform timing analysis based on

Best Typical Worst

Start From Synthesis

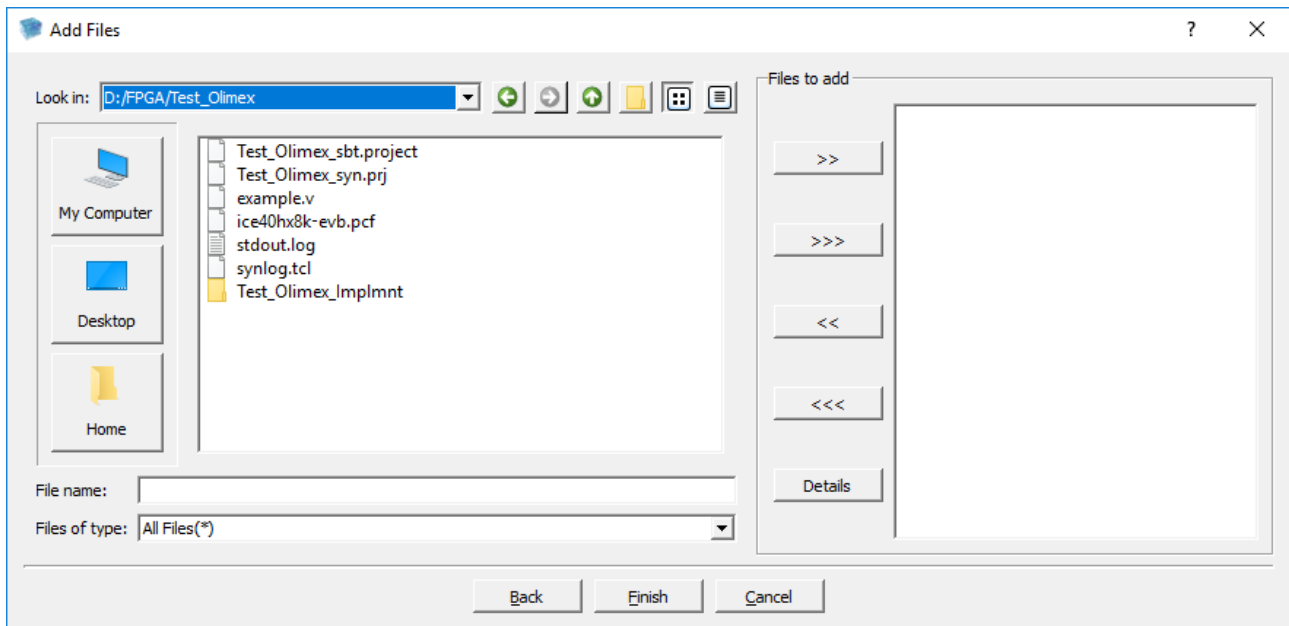
Start From BackEnd

IP Generation

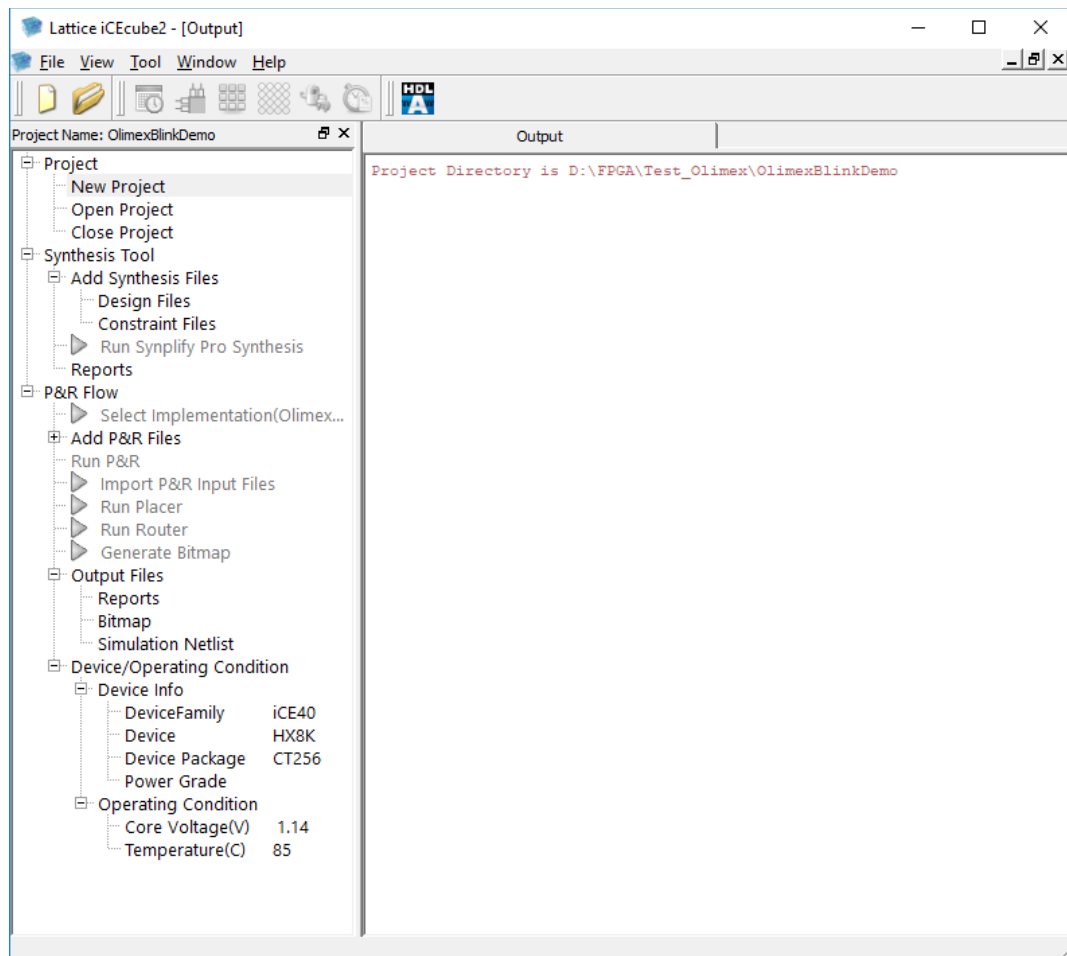
Next Cancel

**(Note that all of our screen shots are showing the H8K, since this is the board we use. The screen shots will be slightly different if you work with the H1K)**

Click on the next button to display the following window



For now, click on "Finish" button, we will add the project files later. You will then return to iCEcube2 main window.



Download the two following files from Olimex Github repository (use them even if you have a iCE40HX8K board) :

- example.v
- ice40hx1k-evb.pcf

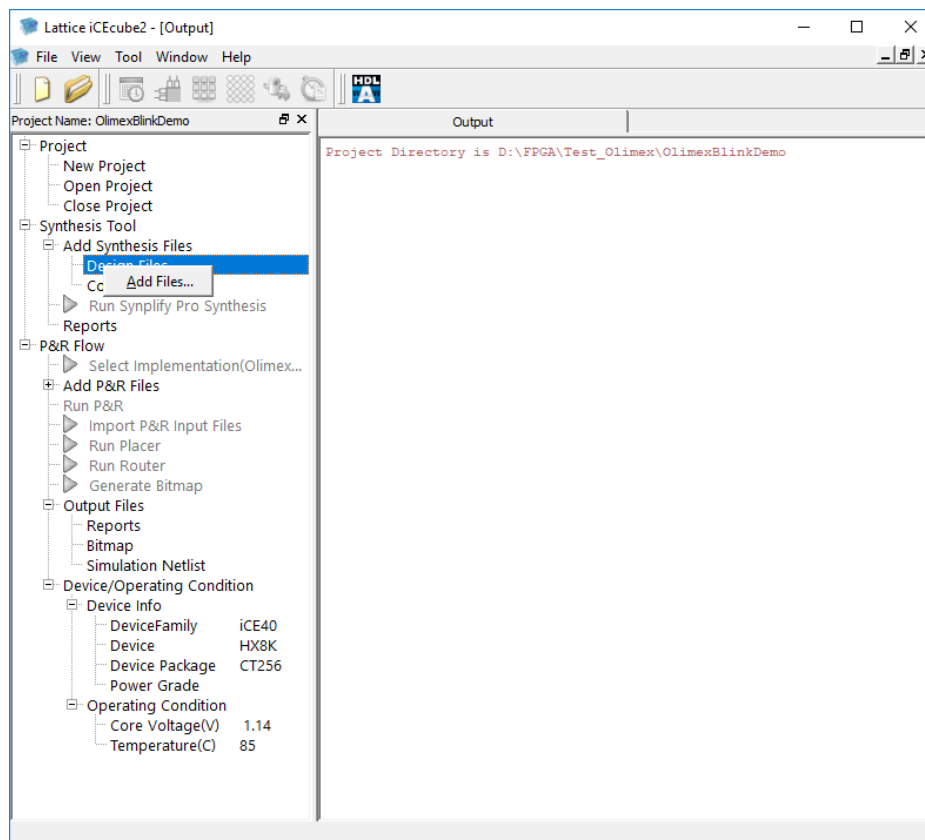
These files are available here :

<https://github.com/OLIMEX/iCE40HX1K-EVB/tree/master/demo/ice40hx1k-evb>

Save the files in your project directory (the exact location does not really matter, but it's better to keep everything organized)

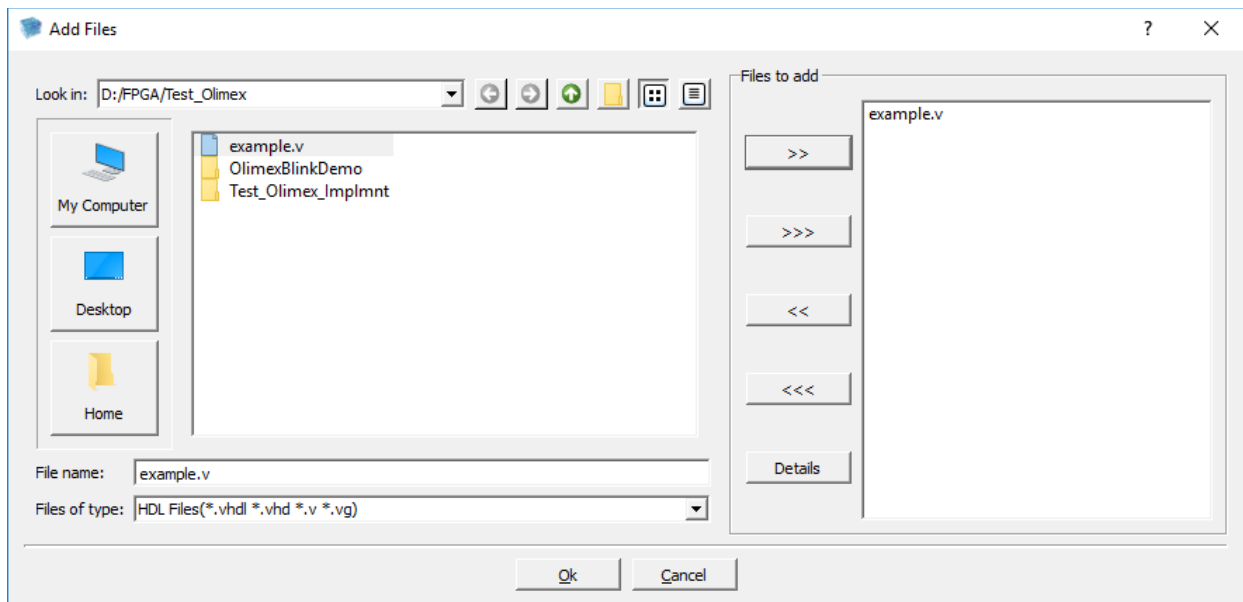
If you have a iCE40HX8K board, make a copy of the ice40hx1k-evb.pcf and save it under ice40hx8k-evb.pcf.

Back in iCEcube2, go to "Synthesis tool" / "Add synthesis files" and right-click on "Design files"



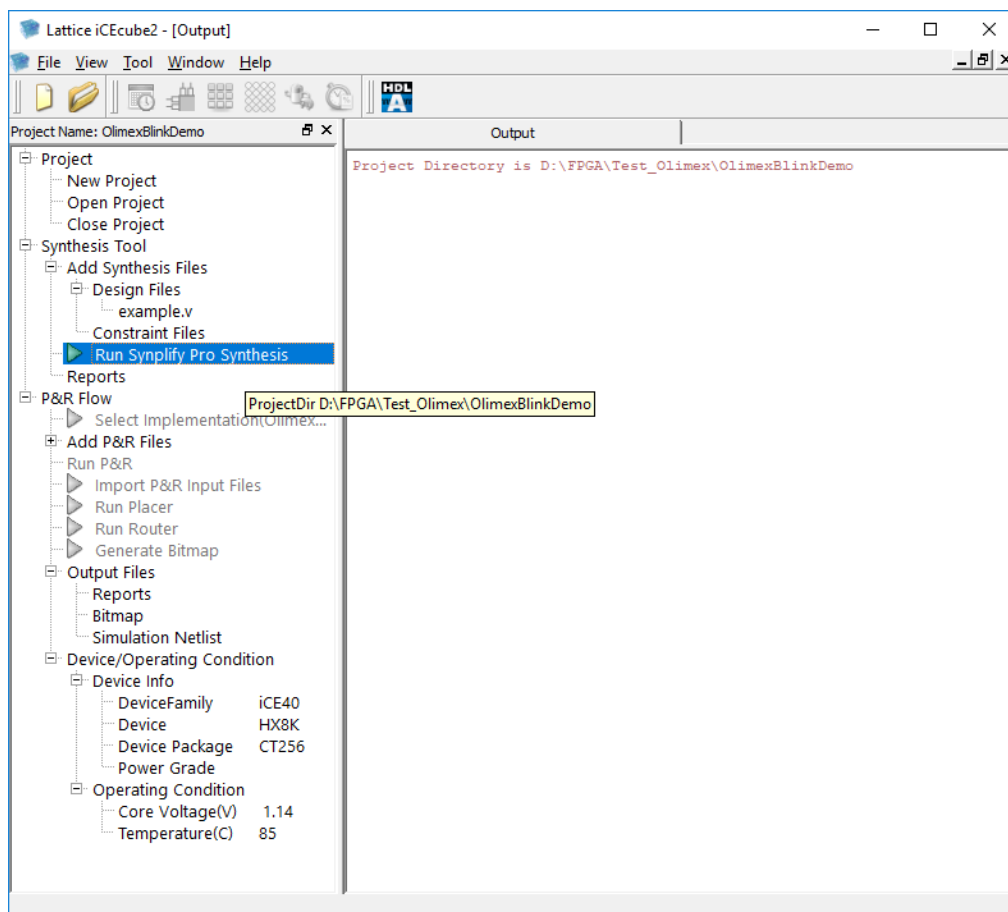
Click on "Add files" and locate the verilog source file you have downloaded just before. Click on the file name (example.v) and click on the ">>" button to make it appear in the right pane.





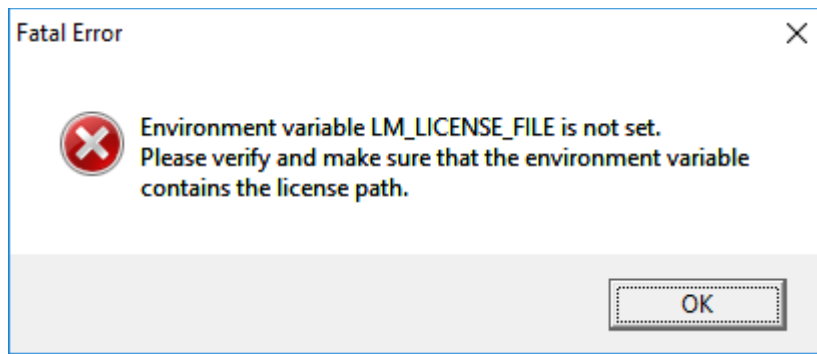
Click on "OK" to close the window. The file will now appear in the left tree under "Design files" entry. Note that a green arrow has appeared now on "Run Synplify Pro Synthesis".

Double click on the line with the green arrow.



This will run the synthesis engine (it "compiles" the Verilog source code).

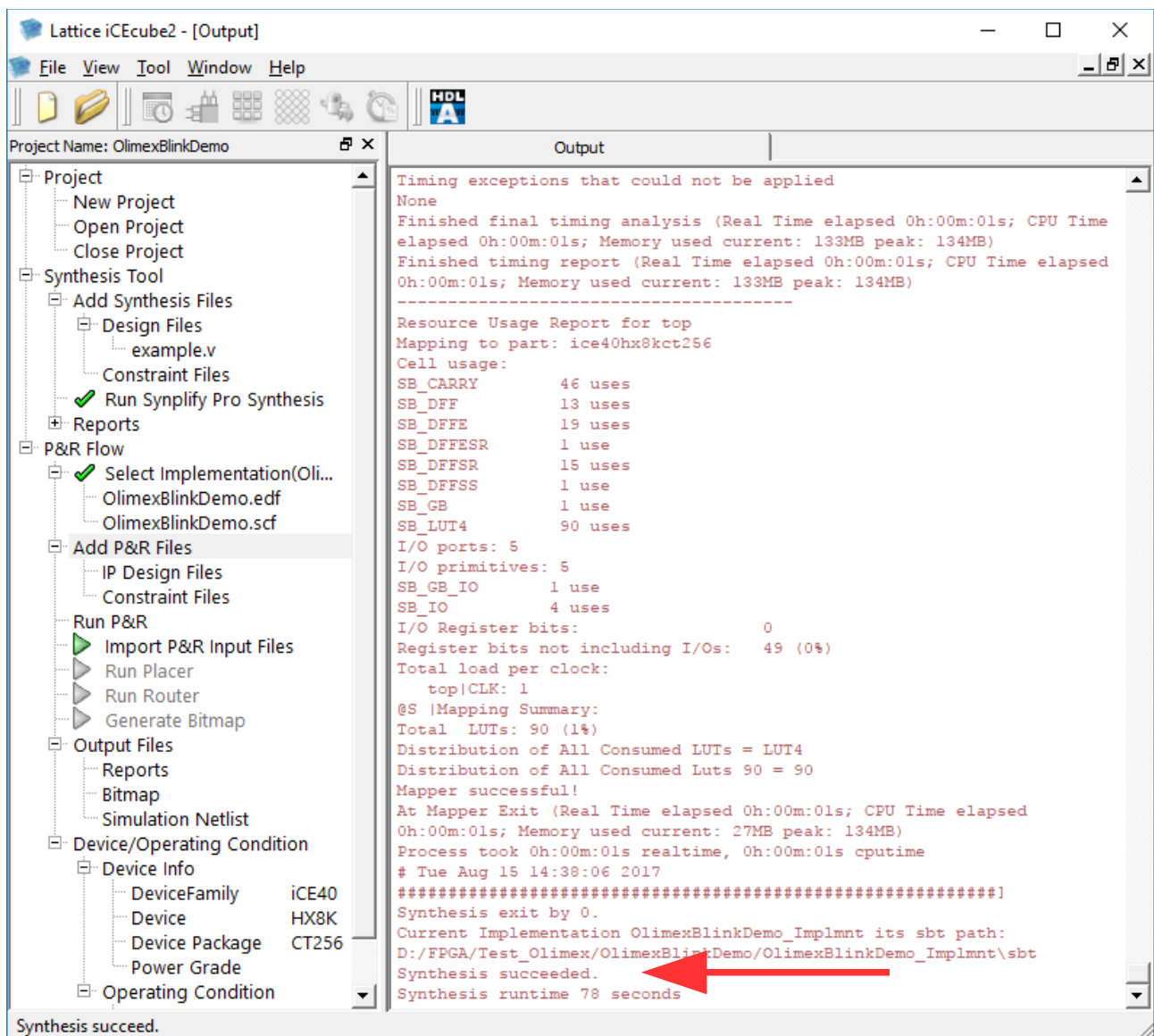
If you get a alert box telling that an environment variable LM\_LICENSE\_FILE is not set, just click on OK. It's not a fatal error at all !



Let the synthesis engine run (it can take a while to complete, especially on slow machines).

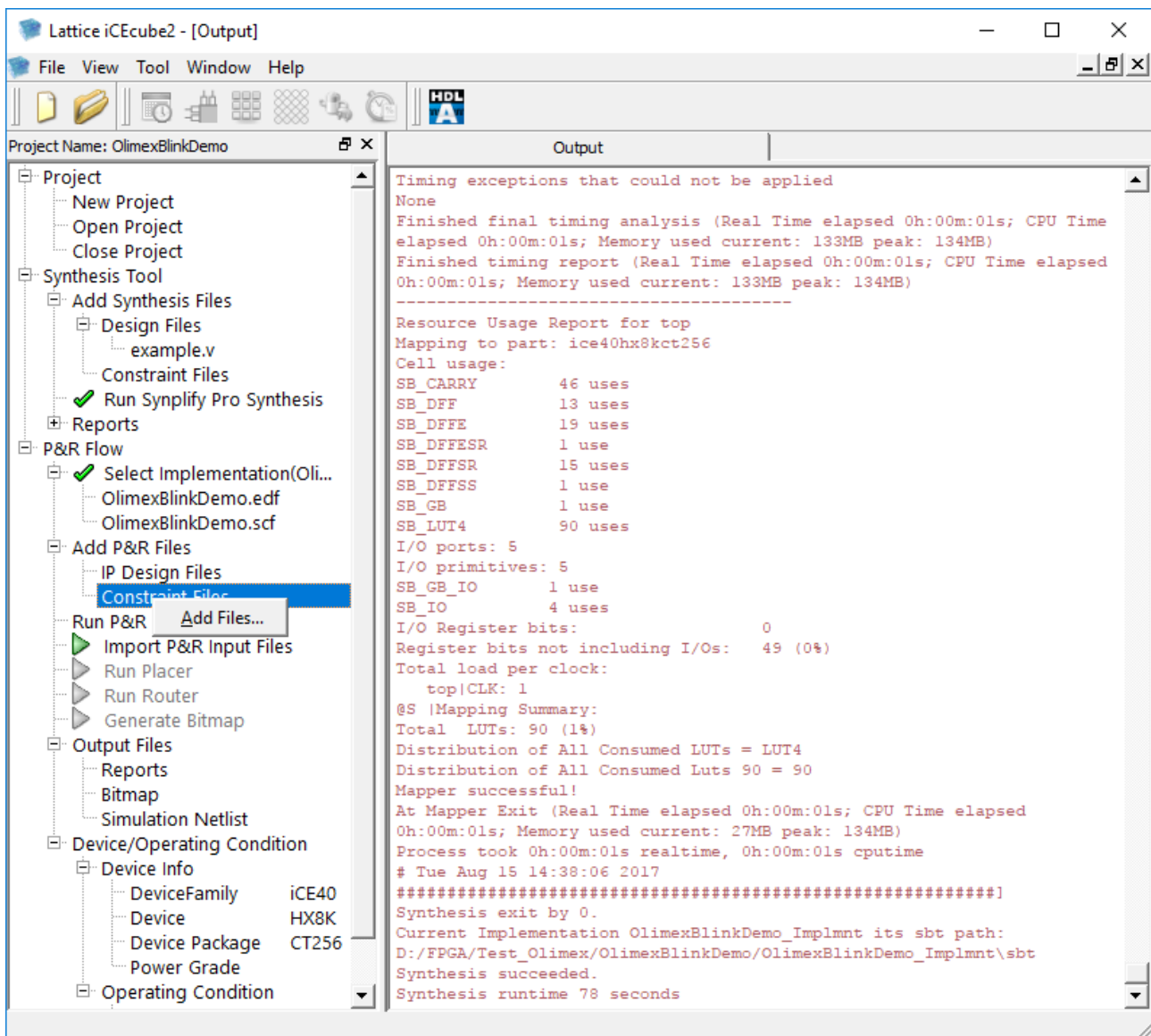
When the synthesis is finished, you will get a complete(!) report being displayed on the right pane...

The important thing is to make sure that you see "Synthesis succeeded" at the end of the report, otherwise it means that your Verilog code is corrupted!



Note also the green check marks, indicating that synthesis has been performed correctly.

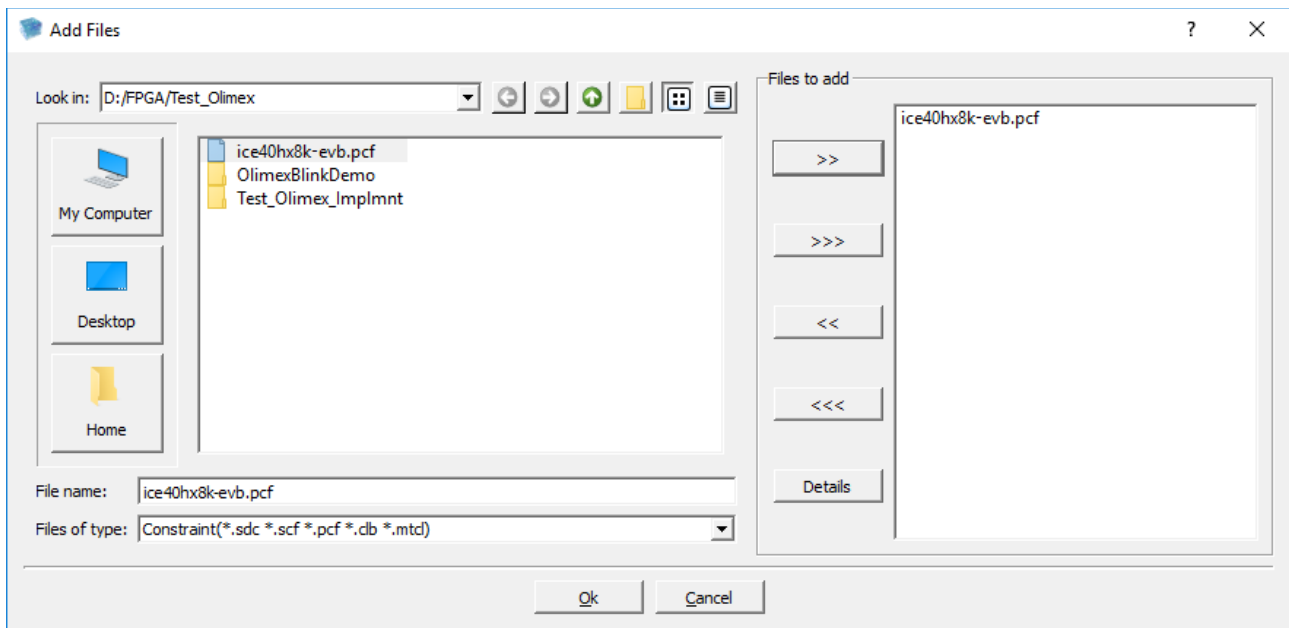
Go now to "P&R flow" / "Add P&R Files" and right click on "Constraint files", then click on "Add files..."



If you have the iCE40HX1K board, select the ice40hx1k-evb.pcf file.

If you have the iCE40HX8K board, select the ice40hx8k-evb.pcf file.

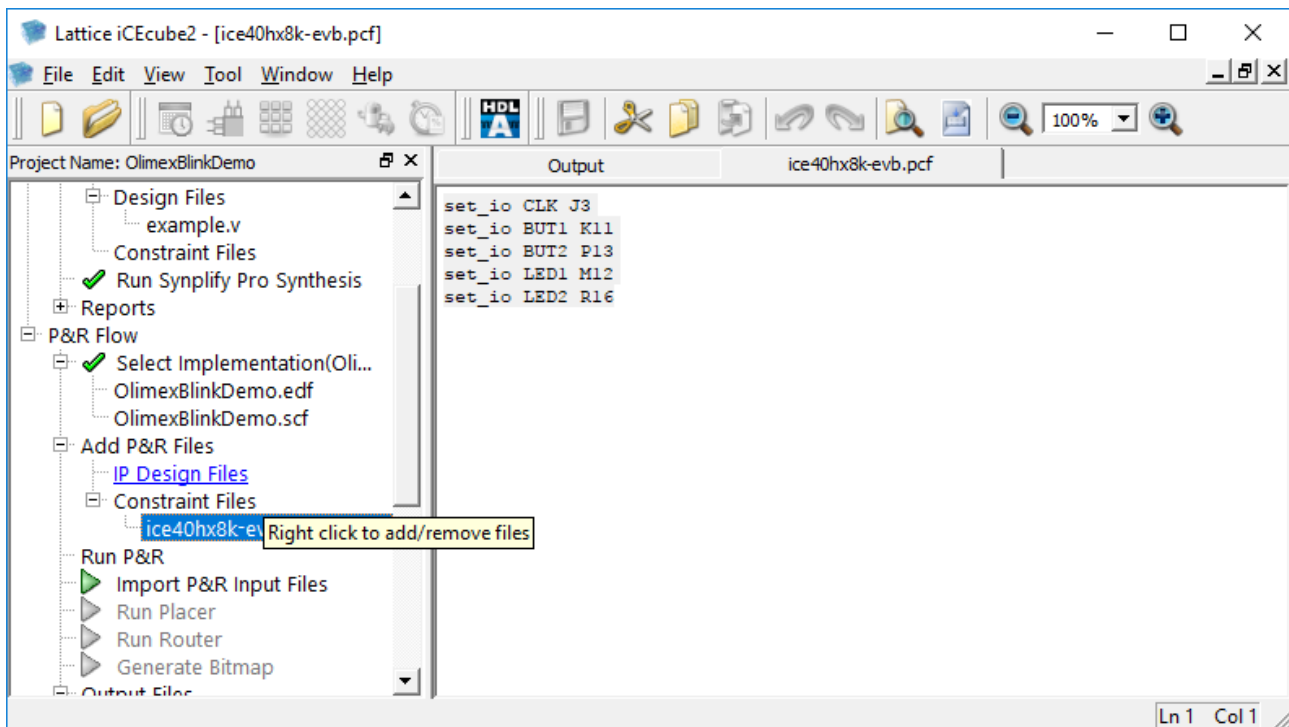
Click then on ">>" button to add the file in the project.



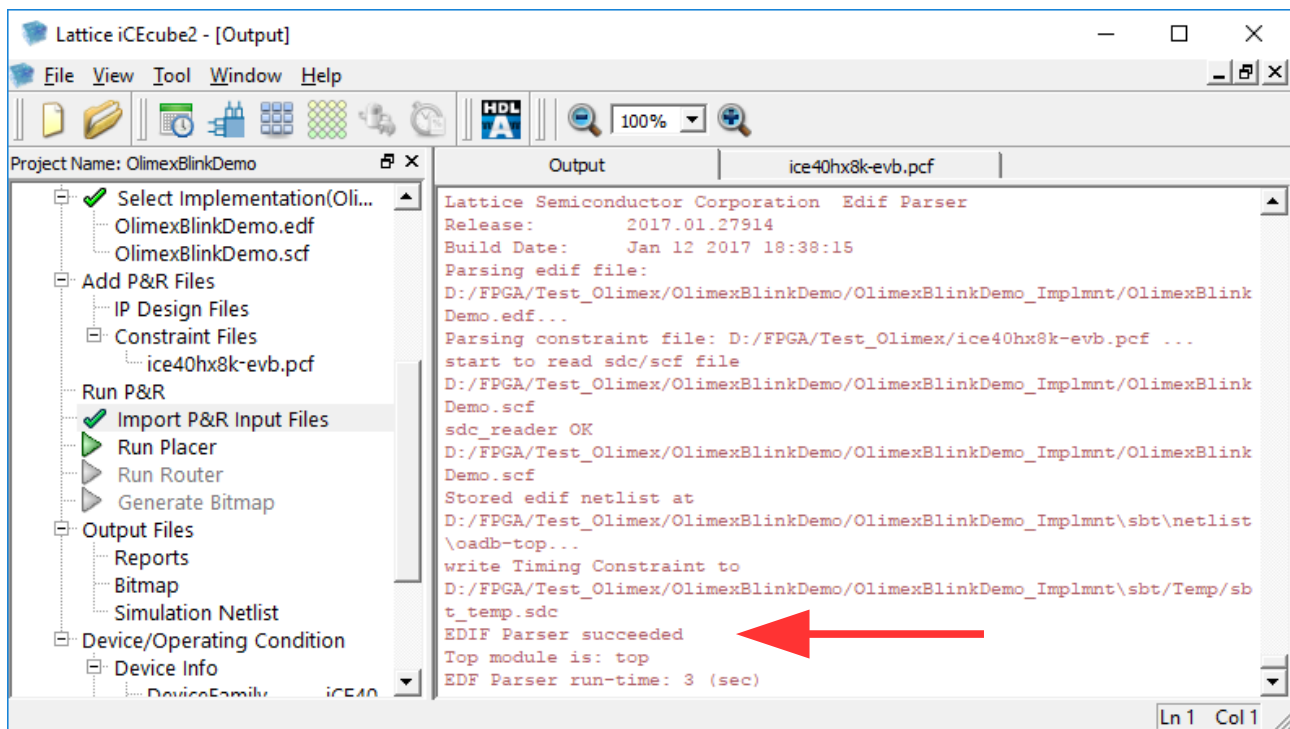
Click OK to close the window.

**IMPORTANT** : if you have the iCE40HX8K-EVB board, you **must** edit the file content (since it is still a copy of the H1K file for now). This can be done directly from iCEcube2. Double click on ice40hx8k-evb.pcf file in the "Constraint files" section and replace the file content by the following lines

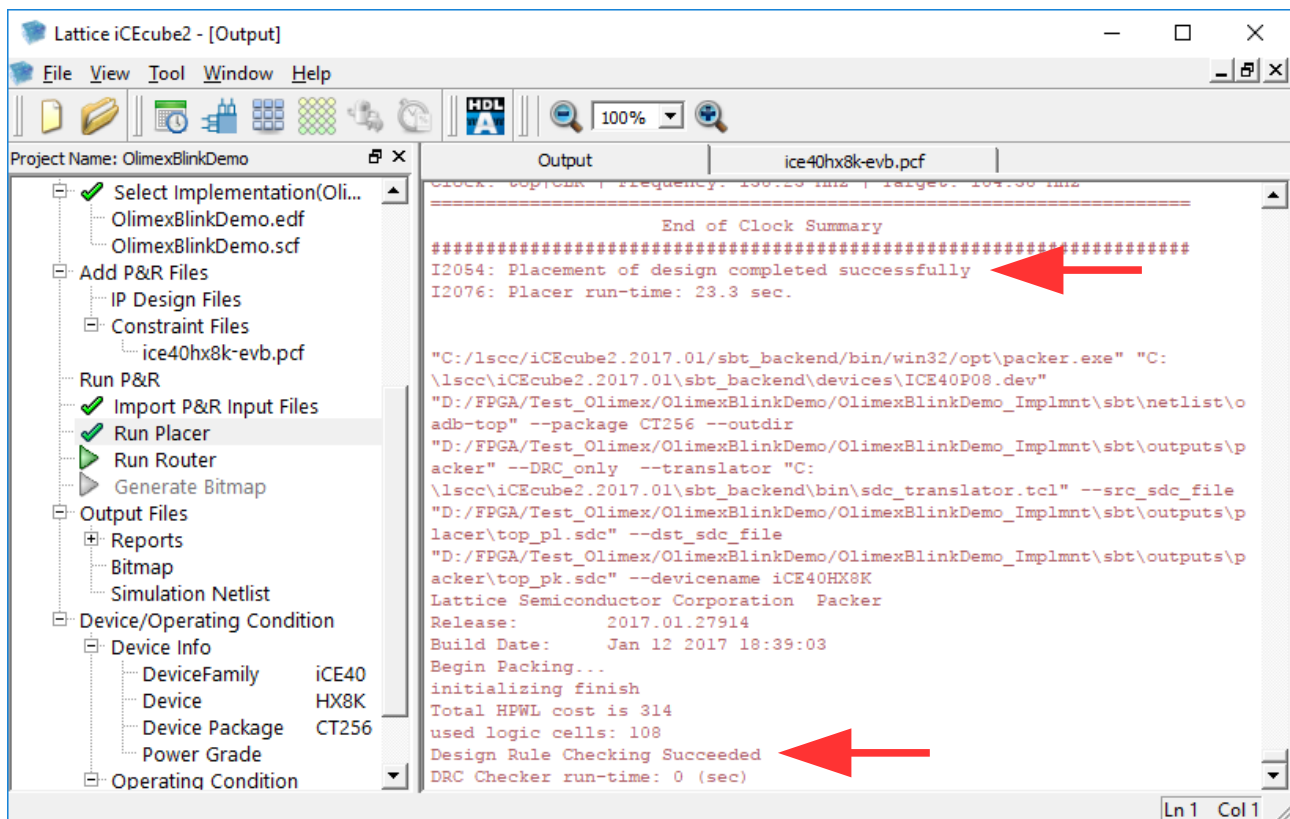
```
set_io CLK J3
set_io BUT1 K11
set_io BUT2 P13
set_io LED1 M12
set_io LED2 R16
```



Double click on "Import P&R Input Files" and let the mapper run. Make sure that you get the "EDIF Parser succeeded" line at the end of the report.

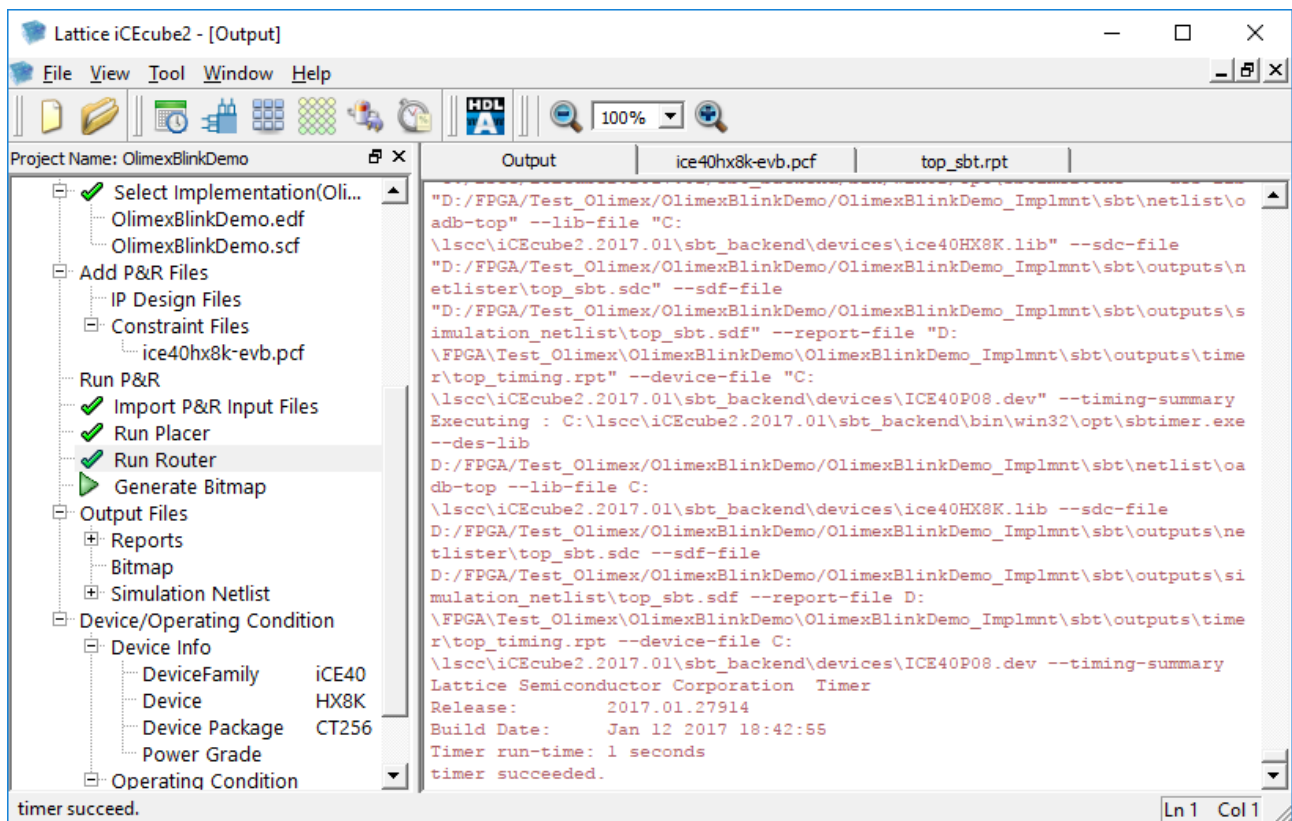


Double click on "Run Placer" (don't worry, this step will take some time to complete)

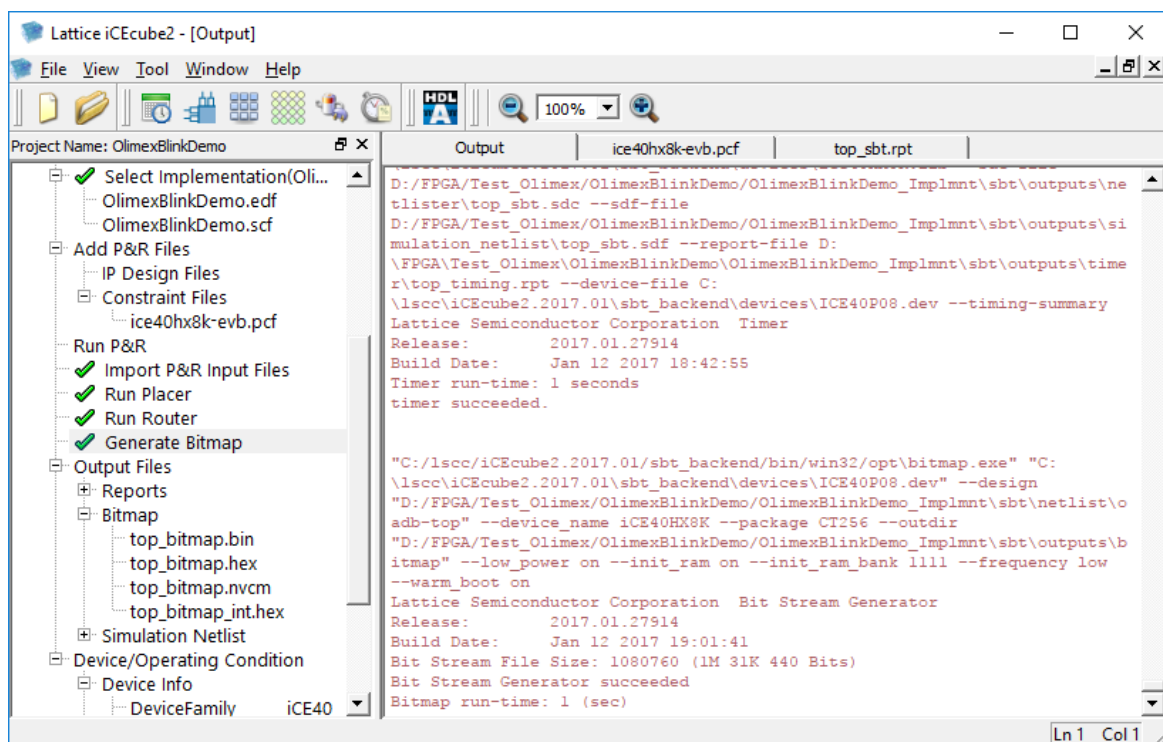


Check that you get the "Placement of design completed successfully" and "Design Rule Checking Succeeded" lines in the report (confirmed by the green check mark in the tree)

Double click on "Run router". Check that you get a green check on "Run Router" to confirm that everything went fine.



And... finally.... double click on "Generate bitmap"! This last step generates the file that will be uploaded in the Flash memory on the EVB board.



The file can be seen in "Output Files" / "Bitmap" (the file we need is the .bin file)

CONGRATULATION : you have generated your FPGA binary file!



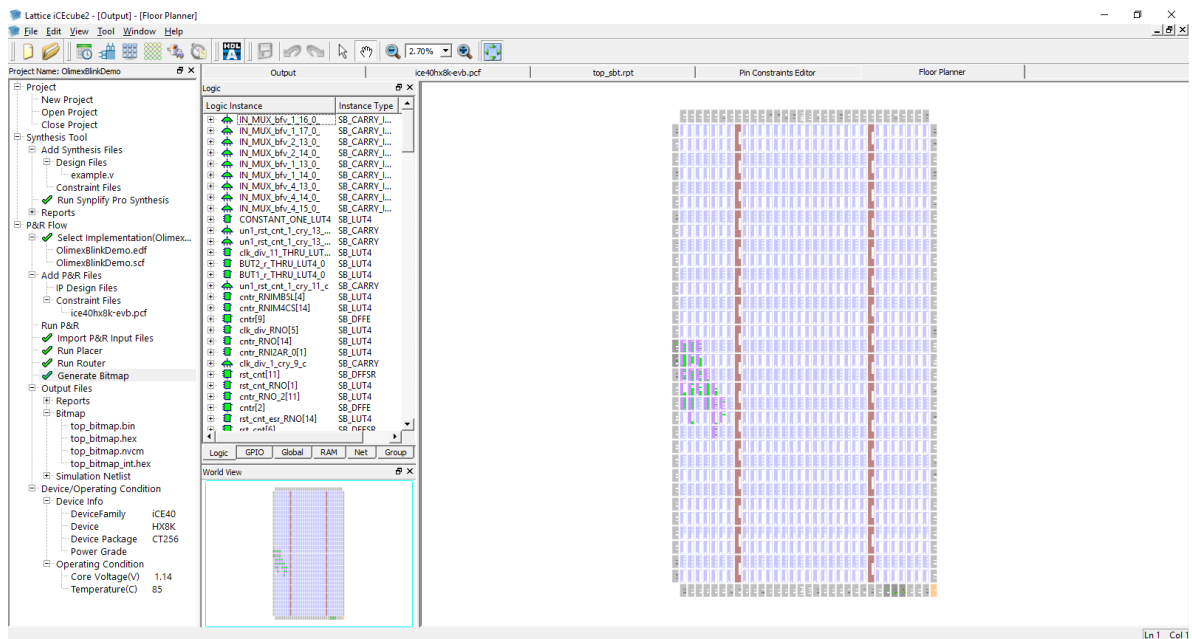
### Step 3 : For the fun...

Once the bitmap file has been generated, there are two screens which are interesting to consult.

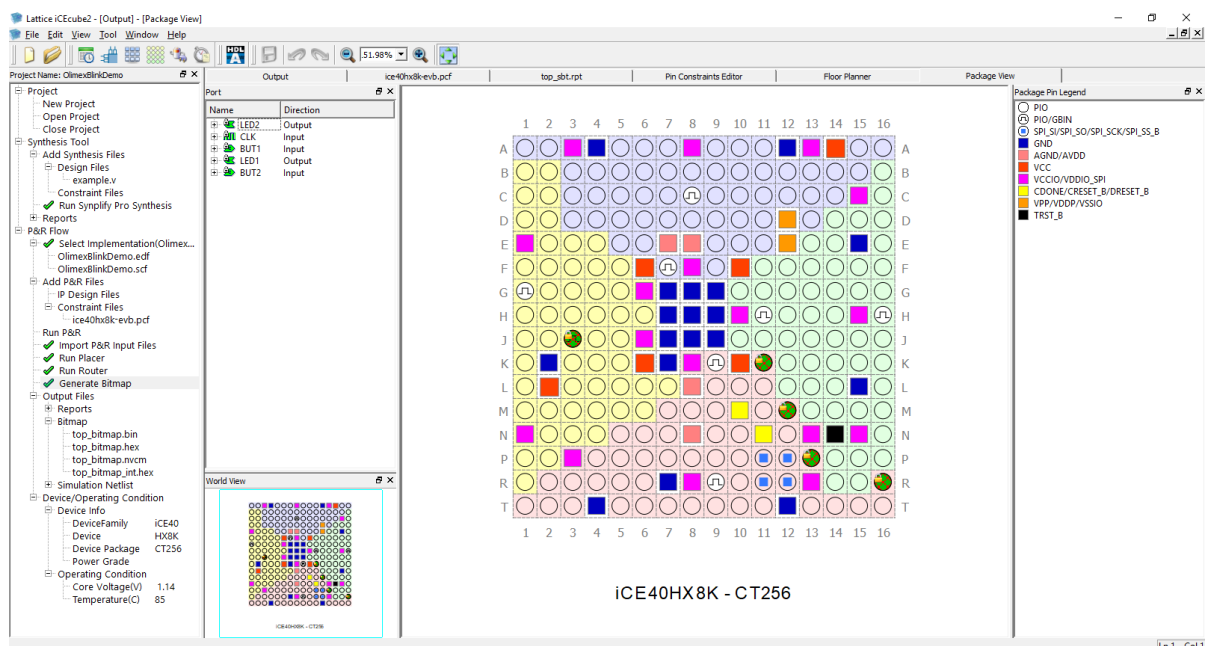
The first one is called "Floor Planner", it shows the resource used on your FPGA. When you work with big Verilog or VHDL projects, this screen is very important to know when you reach the limits of the chip you use.

Click on "Tools" / "Floor planner" menu to show the following window (it can be reached also on the button bar on the top)

Detailing the floor planner functionality is out of the scope of this starter manual. Please refer to Lattice manuals for details about this tool.



The second useful tool is the Package View, which shows which pin is used for which function. This window is reached by "Tool" / "Package View" menu.



#### **Step 4 : Let's go in the real world**

Now that you have the FPGA binary file (the "bitmap"), you can upload it in the Olimex board.

The easiest solution is to use the Olimexino 32U4 board, loaded with the iceprogduino sketch. You will find all details about the configuration of the board on Olimex website.

You can also use another Arduino (we use here a Intel Galileo Gen2) loaded with the sketch. Note that the iceprogduino sketch is designed to run natively on Olimexino 32U4 boards only, it needs to be modified to run on Arudino Zero, Arduino Due and Intel Galileo boards. If you use a Galileo board, we can provide you the modified sketch.

TAKE AN EXTREME CARE ABOUT THE ARDUINO VOLTAGE! Do not use a +5V Arduino to program the iCE40 EVB boards, this would destroy immediately the components (the EVB runs on 3.3V only)

On Windows side, there are two programs that can be used : winiceprogduino, which has been compiled using GCC or WinProgICE, which has been compiled under Visual Studio. We prefer the second one (not only because we wrote it, but because we have enhanced the user interface). The two programs are used in the same way (they use the same basic source code).

Connection details between the Olimexino and the iCE40 boards via UEXT connector are given on Olimex website. We highly recommend to connect the iCE40 board and the programmer with power supply disconnected.

Once the programmer and the iCE40 board are connected and powered, you can start the programming software. Check the serial port number used by the Olimexino (using Windows control panel or Arduino IDE), you will need it for the programming tool.

You can easily test the connection between the two boards using the following command (depending on the software you want to use, and replace xx with Arduino serial port number)

```
winprogice -ICOMxx -t  
or  
winiceprogduino -ICOMxx -t
```

If you see "Winbond W25Q16" being reported, then you are sure that communication is ok between the two boards

To program a bitmap file for the FPGA, the command syntax is simple

```
winprogice -ICOMxx -v bitmapfile.bin  
or  
winiceporgduino -ICOMxx bitmapfile.bin
```

This command will erase automatically the Flash and will transfer the bitmap file to the Flash.

In all cases, check that you get the "Bye" message at the end. If you see the program stopping or if you see an error message, the Flash will not be programmed completely and the FPGA will not be programmed correctly and may act strangely.