

Vivado Design Suite Tutorial

Power Analysis and Optimization

UG997 (v2014.4) November 19, 2014



Revision History

The following table shows the revision history for this document.

Date	Version	Changes
11/19/2014	2014.4	Updated screen displays throughout document.
10/01/2014	2014.3	<p>Changed tutorial design (in ZIP file). Modified procedures, screen displays, and Tcl commands throughout document to apply to this new design.</p> <p>Updated screen displays throughout document.</p> <p>Added Lab 4: Hardware Power Measurement Using the KC705 Evaluation Board. In this lab, you perform power measurement on the design implemented in a KC705 Evaluation Board, and compare the hardware power numbers with the numbers generated by Vivado Report Power.</p> <p>Changed Software Requirements and Hardware Requirements to add the software and hardware necessary to perform Lab 4: Hardware Power Measurement Using the KC705 Evaluation Board.</p>
06/04/2014	2014.2	<p>Modified procedures to show that the <code>filter_block</code> source is now supplied as a synthesized module.</p> <p>Updated screen displays throughout document.</p>
04/02/2014	2014.1	<p>Updated screen displays throughout document.</p> <p>Added IMPORTANT Note in Lab 1, Step 4: Running Report Power with information about using toggle rate to specify switching activity.</p> <p>Revised procedure in Lab 1, Step 7: Running Functional Simulation with SAIF Output to perform a functional simulation instead of a behavioral simulation.</p> <p>When running Report Power using the data in a SAIF file, added information about observing the Tcl Console to determine the number of matched nets in the design.</p> <p>In Lab 3, Step 1: Set Up to Run Timing Simulation in QuestaSim, added procedure to create new compiled libraries.</p>

Table of Contents

Revision History	2
Power Analysis and Optimization Tutorial	5
Overview	5
Software Requirements	5
Hardware Requirements	6
Locating Tutorial Design Files	6
Lab 1: Power Analysis in the Vivado Tools	8
Introduction	8
Step 1: Creating a New Project	8
Step 2: Synthesizing the Design	14
Step 3: Report Power Settings	14
Step 4: Running Report Power	19
Step 5: Power Properties	21
Step 6: Editing Power Properties and Refining the Power Analysis	22
Step 7: Running Functional Simulation with SAIF Output	25
Step 8: Incorporating SAIF Data into Power Analysis	29
Step 9: Implementing the Design	33
Lab 2: Vivado Simulator Timing Simulation and Power Analysis	35
Introduction	35
Step 1: Open the Implemented Design	35
Step 2: Running Report Power in Vectorless Mode	38
Step 3: Running Report Power with Vivado Simulator SAIF Data	39
Lab 3: QuestaSim Timing Simulation and Power Analysis	44
Introduction	44
Step 1: Set Up to Run Timing Simulation in QuestaSim	45
Step 2: Running Report Power in Vectorless Mode	50
Step 3: Running Report Power with QuestaSim SAIF Data	51
Lab 4: Hardware Power Measurement Using the KC705 Evaluation Board	54
Introduction	54
Step 1: Generate a Bit File from the Implemented Design (Non-Power Optimization)	54

Step 2: Set Up the KC705 Evaluation Board	55
Step 3: Set Up the Fusion Digital Power Designer Software	56
Step 4: Program the Bitstream	57
Step 5: Measure the Hardware Power Rails.....	60
Step 6: Vectorless Power Estimation with Junction Temperature	62
Lab 5: Power Optimization in the Vivado Tools	65
Introduction.....	65
Step 1: Set Up Options to Run Power Optimization.....	65
Step 2: Run <code>report_power_opt</code> to Examine User/Design Specific Power Optimizations.....	68
Step 3: Run <code>report_power</code> to Examine Power Savings.....	70
Step 4: Turn Off Optimizations on Specific Signals and Rerun Implementation	71
Step 5: Run <code>report_power_opt</code> to Examine Tool Optimizations Again	73
Conclusion	74
Legal Notices.....	75
Please Read: Important Legal Notices.....	75

Power Analysis and Optimization Tutorial

Overview

This tutorial introduces the power analysis and optimization use model recommended for use with the Xilinx® Vivado® Integrated Design Environment (IDE). The tutorial describes the basic steps involved in taking a small example design from RTL to implementation, estimating power through the different stages, and using simulation data to enhance the accuracy of the power analysis. It also describes the steps involved in using the power optimization tools in the design.



VIDEO: The [Vivado Design Suite QuickTake Video: Power Estimation and Analysis Using Vivado](#) shows how the Vivado Design Suite can help you to estimate power consumption in your design and reviews best practices for getting the most accurate estimation.



VIDEO: The [Vivado Design Suite QuickTake Video: Power Optimization Using Vivado](#) describes the factors that affect power consumption in an FPGA, shows how the Vivado Design Suite helps to minimize power consumption in your design, and looks at some advanced control and best practices for getting the most out of Vivado power optimization.

Software Requirements

This tutorial requires that the 2014.4 Vivado Design Suite software release or later is installed. For installation instructions and information, see the *Vivado Design Suite User Guide: Release Notes, Installation, and Licensing* ([UG973](#)).

For Hardware Power Measurement, the tutorial requires Texas Instruments Fusion Digital Power Designer software, which can be downloaded from this location:

www.ti.com/tool/fusion_digital_power_designer

Hardware Requirements

Hardware requirements for this Tutorial are:

- Supported Operating Systems to run the Vivado Design Suite, and memory recommendations when using the Vivado tools, are described in the *Vivado Design Suite User Guide: Release Notes, Installation, and Licensing* ([UG973](#)).
- The hardware power measurements in [Lab 4: Hardware Power Measurement Using the KC705 Evaluation Board](#) require a Xilinx Kintex®-7 FPGA KC705 Evaluation Kit. You can find information on the Evaluation Kit at this location:

www.xilinx.com/products/boards-and-kits/EK-K7-KC705-G.htm

- For power measurements through TI Power Regulators (needed in [Lab 4: Hardware Power Measurement Using the KC705 Evaluation Board](#)), use the Texas Instruments USB Interface Adapter. You can find information on the USB Interface Adaptor at this location:

www.ti.com/lit/ml/sllu093/sllu093.pdf

Locating Tutorial Design Files

1. Download the `ug997-vivado-power-analysis-optimization-tutorial.zip` file from the Xilinx website:

<https://secure.xilinx.com/webreg/clickthrough.do?cid=371140>

2. Extract the zip file contents into any write-accessible location.

This tutorial refers to the location of the extracted `ug997-vivado-power-analysis-optimization-tutorial.zip` file contents as `<Extract_Dir>`.



IMPORTANT: You will modify the tutorial design data while working through this tutorial. Use a new copy of the original data each time you start this tutorial.

The following table describes the contents of the zip file containing the tutorial design files, `ug997-vivado-power-analysis-optimization-tutorial.zip`.

Directories/Files	Description
<code>/src</code>	Contains the design HDL and testbench for the functional simulation.
<code>/src/dut_fpga.v</code>	Top module for the design.
<code>/src/bram_tdp.v</code> <code>/src/bram_top.v</code> <code>/src/dut.v</code>	Other design blocks - synthesized module.
<code>dut_fpga_kc705.xdc</code>	Contains clocking and timing constraints for the design.
<code>/src/testbench.v</code>	Testbench for simulating the design.
<code>readme.txt</code>	A readme file about the tutorial design.

Lab 1: Power Analysis in the Vivado Tools

Introduction

In this lab, you will learn about the Power Analysis and Optimization features in the Vivado IDE. The lab will take you through the steps of project creation and power analysis at the synthesis stage, using the Vivado Report Power feature in vectorless mode. It will also demonstrate using the SAIF file generated from behavioral simulation for Vivado Report Power Analysis.

You will analyze power in the Vivado IDE. Then you will examine some of the major features in the Power Report window and closely examine some power specific Tcl commands. You will also learn to create a SAIF file by simulating the design in the timing simulation stage using both the Vivado simulator and QuestaSim.

You will also learn how to invoke Power Optimization after `opt_design` in the Vivado IDE. You will examine the power optimization report and selectively turn power optimizations ON or OFF on specific signals, nets, modules, or hierarchy.

Step 1: Creating a New Project

To create a project, use the New Project wizard to name the project, to add RTL source files and constraints, and to specify the target device.

- On Linux,
 1. Go to the directory where the lab materials are stored:

```
cd <Extract_Dir>/Vivado_Power_Tutorial
```
 2. Launch the Vivado IDE: `vivado`



Figure 1: Vivado IDE – Getting Started Page

- On Windows,
 1. Launch the Vivado IDE:
Start > All Programs > Xilinx Design Tools > Vivado 2014.4 > Vivado 2014.4
 2. As an alternative, click the **Vivado 2014.4** Desktop icon to start the Vivado IDE.
The Vivado IDE Getting Started page contains links to open or create projects and to view documentation.
 3. In the Getting Started page, click **Create New Project** to start the New Project wizard.
 4. Click **Next** to continue to the next screen.

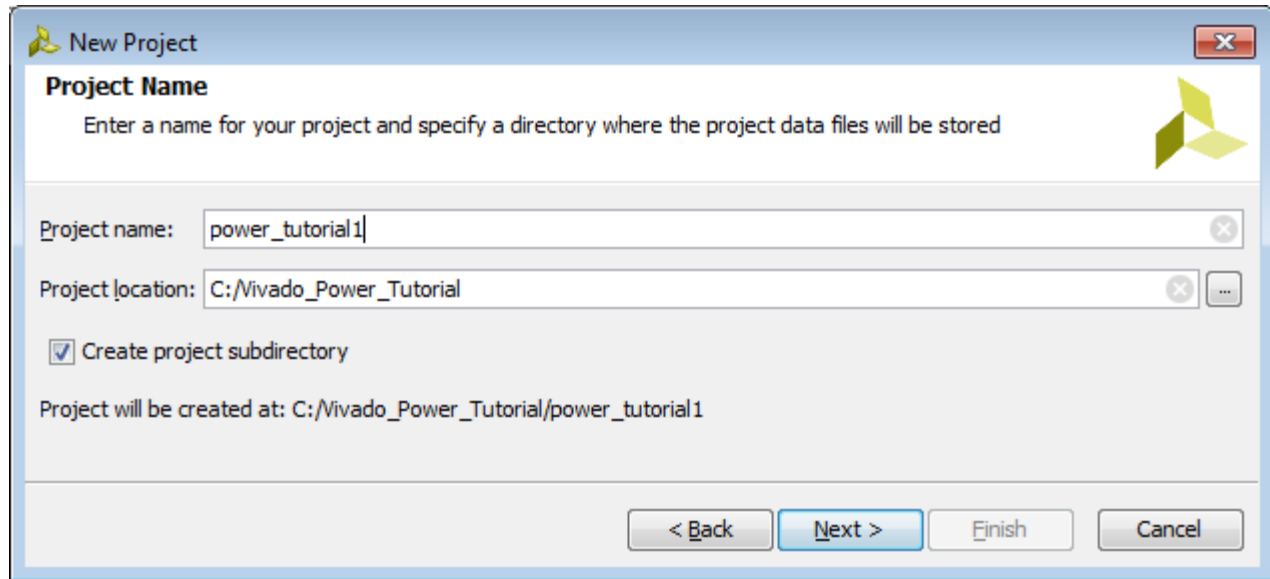


Figure 2: Creating a New Project

5. In the Project Name page, name the new project `power_tutorial1` and enter the project location (`C:\Vivado_Power_Tutorial`). Make sure **Create project subdirectory** is checked and click **Next**.
6. In the Project Type page, specify the **type of project** to create as **RTL Project**, make sure **Do not specify sources at this time** is unchecked, and click **Next**.
7. In the Add Sources page:
 - a. Set **Target Language** to **Verilog** and **Simulator language** to **Mixed**.
 - b. Click the **Add Files** button.
 - c. In the Add Source Files dialog box, navigate to the `<Extract_Dir>/src` directory.
 - d. Select all of the Verilog (`.v`) source files, and click **OK**.
 - e. In the Add Sources page, change the **HDL Source For** the `testbench.v` file to **Simulation Only**.

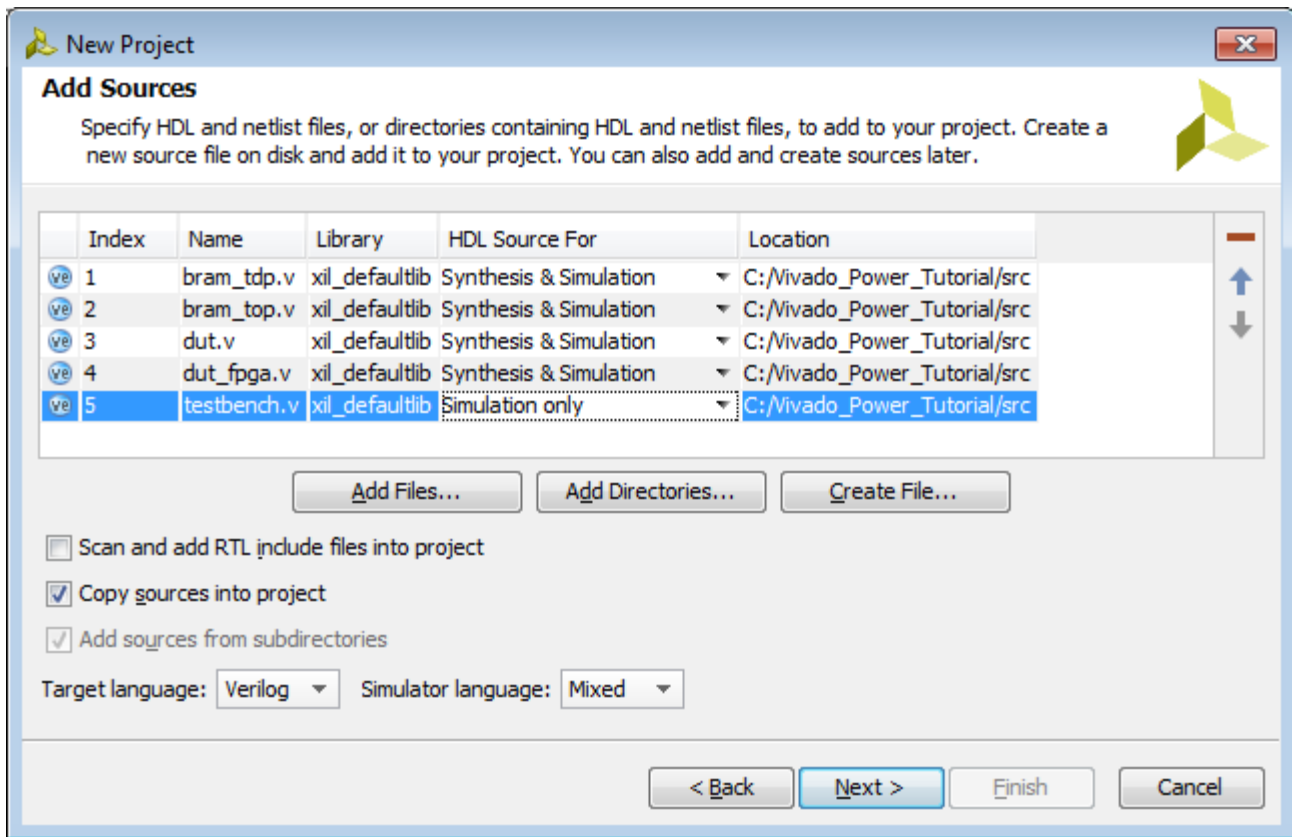


Figure 3: Setting HDL Source Type for testbench.v

- f. Verify that the files are added and **Copy sources into project** is checked. Click **Next**.
8. In the Add Existing IP (optional) dialog box, click **Next**.
9. In the Add Constraints (optional) dialog box, click **Add Files** and select dut_fpga_kc705.xdc in the file browser.
10. Click **Next** to continue.
11. In the Default Part dialog box, click **Boards** to specify the board for the target device and select **Kintex-7 KC705 Evaluation Platform**. Then click **Next**.



TIP: When you specify a **Board**, you are also specifying the part you are targeting for your design, in this case an xc7k325tffg900-2 FPGA.

12. Review the New Project Summary page. Verify that the data appears as expected, per the steps above, and click **Finish**.

Note: It might take a moment for the project to initialize in the Vivado IDE.

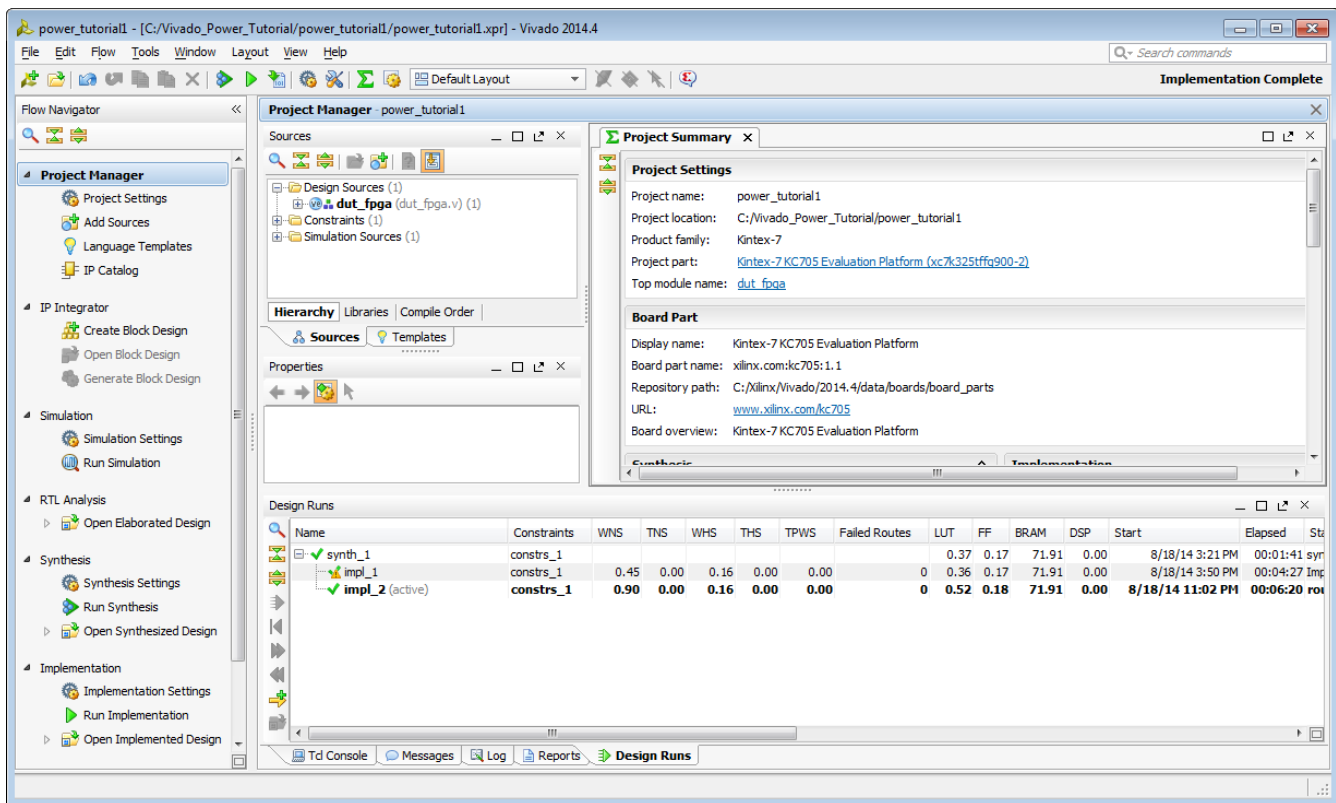


Figure 4: Project in Vivado IDE

- In the Vivado Options dialog box (**Tools > Options > General**), enter the tutorial project directory in the **Specify project directory** box, so that all reports are saved in the tutorial project directory. Then click **OK**.

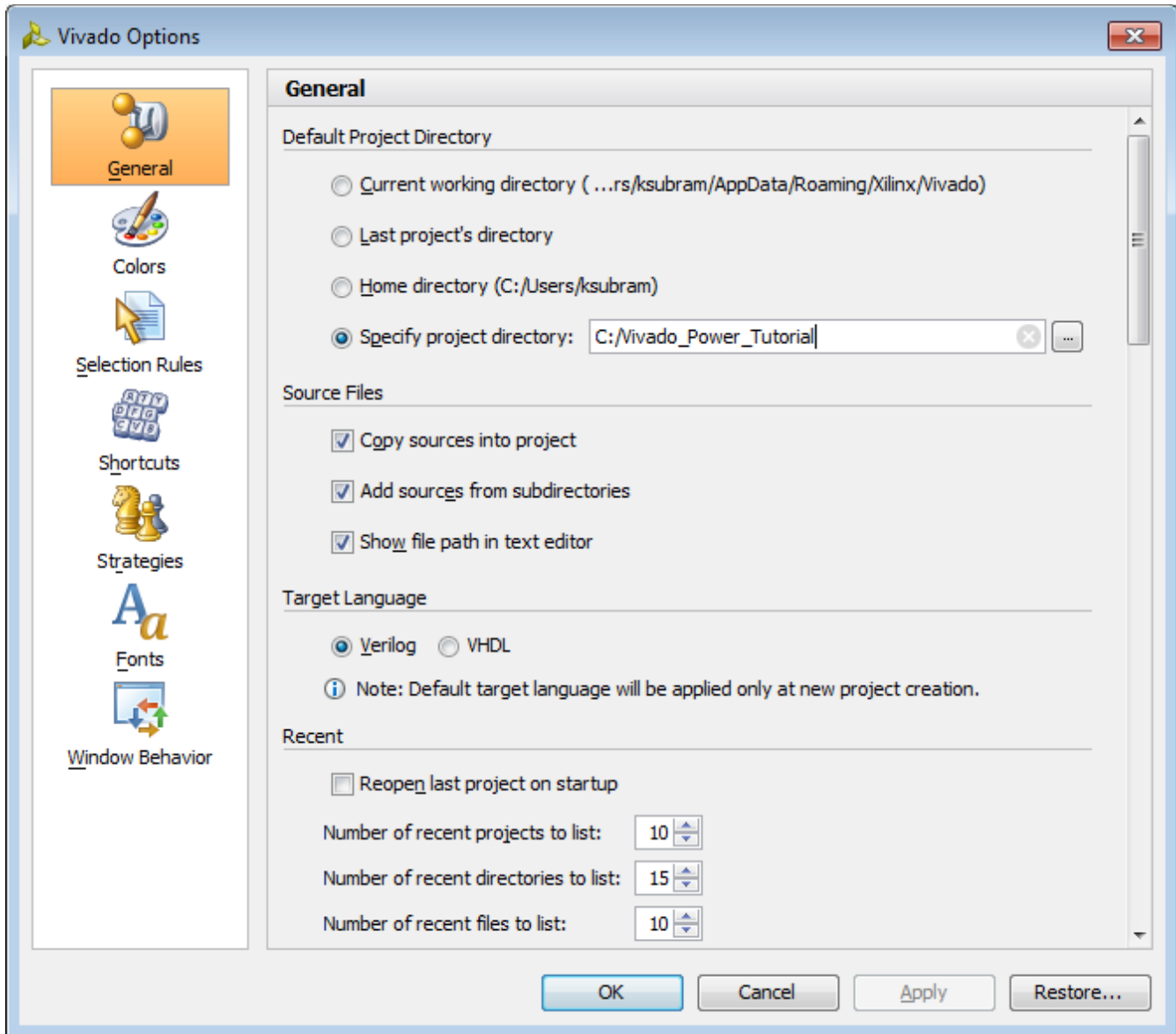


Figure 5: Setting the Project Directory

Now the design is ready for Synthesis.

Step 2: Synthesizing the Design

1. Click **Run Synthesis** in the Flow Navigator.

The Synthesis Completed dialog box appears after synthesis has completed on the design.

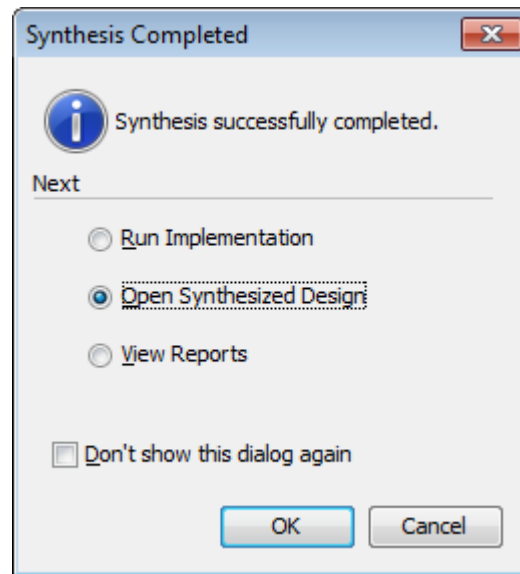


Figure 6: Synthesis Completed Dialog Box

2. Open the synthesized design by selecting **Open Synthesized Design** in the Synthesis Completed dialog box and clicking **OK**.

Step 3: Report Power Settings

The Vivado IDE allows you to specify input data to the Report Power tool to enhance the accuracy of the power analysis.

In the Vivado IDE, you can configure thermal, environmental, and power supply options to mimic the board level settings as closely as possible. For information on setting these options, see the *Vivado Design Suite User Guide: Power Analysis and Optimization* ([UG907](#)).

1. In the main menu bar, select **Tools > Report > Report Power**.
2. In the Report Power dialog box, browse to the project directory and specify the **Output text file** as `power_1.pwr`.
3. Browse to the project directory and specify the **Output XPE file** as `power_1.xpe`. After creating this file when Report Power runs, you can import the file and results into the Xilinx Power Estimator (XPE).

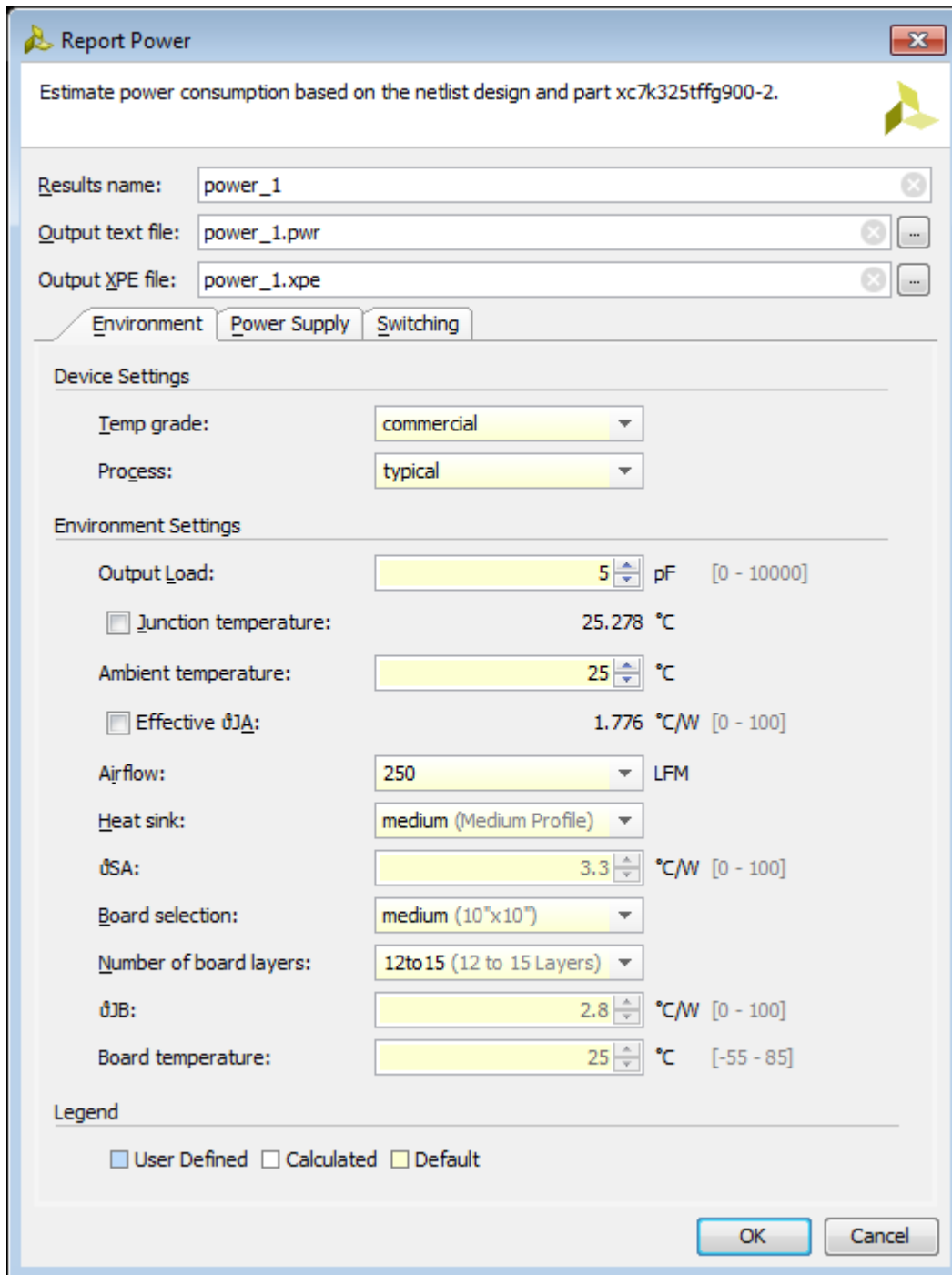


Figure 7: Report Power Dialog Box

4. Examine the **Environment** tab in the Report Power dialog box (Figure 7).
5. In the **Environment** tab, set **Process** to **maximum** for a worst case power analysis.
6. Examine the **Power Supply** tab (Figure 8).



IMPORTANT: By default, Vivado Report Power uses nominal values for voltage supply sources. Voltage is a large factor contributing to both static and dynamic power. For the most accurate analysis, ensure that actual voltage values are entered for each supply. Similarly, ensure temperature and other environmental factors match actual operating conditions.

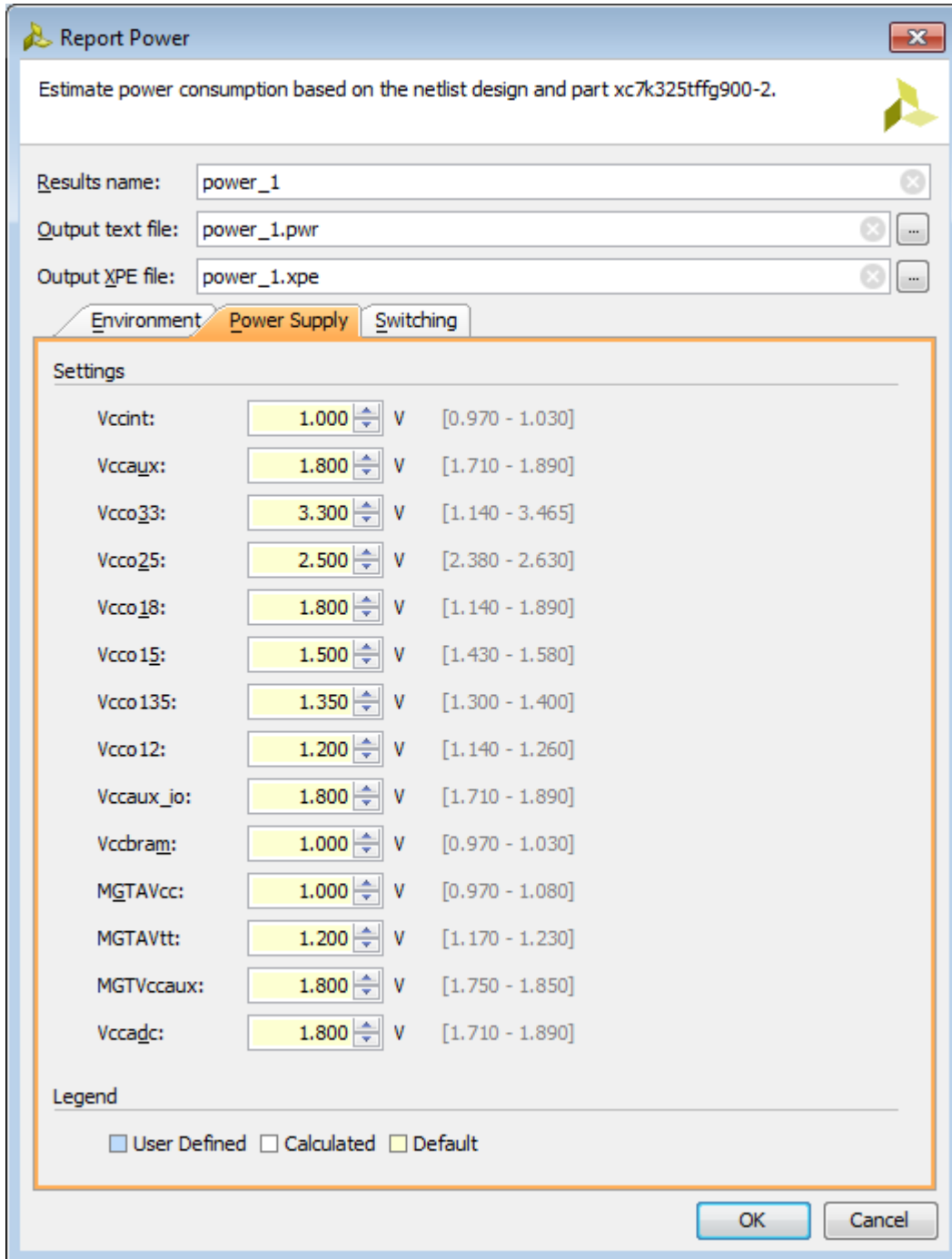


Figure 8: Report Power Dialog Box – Power Supply Tab

7. In the **Switching** tab, click on **Constrained Clocks** and examine the constrained clocks in the design.

IMPORTANT: *Make sure all the relevant clocks in the design are constrained. All the design clocks must be defined using `create_clock` or `create_generated_clock`' XDC constraints, so that Report Power recognizes the clocks.*



Default toggle rate is set to 12.5% and **Default percent high** is set to 0.5. This will be applied to primary ports (non-clock) and block box outputs.

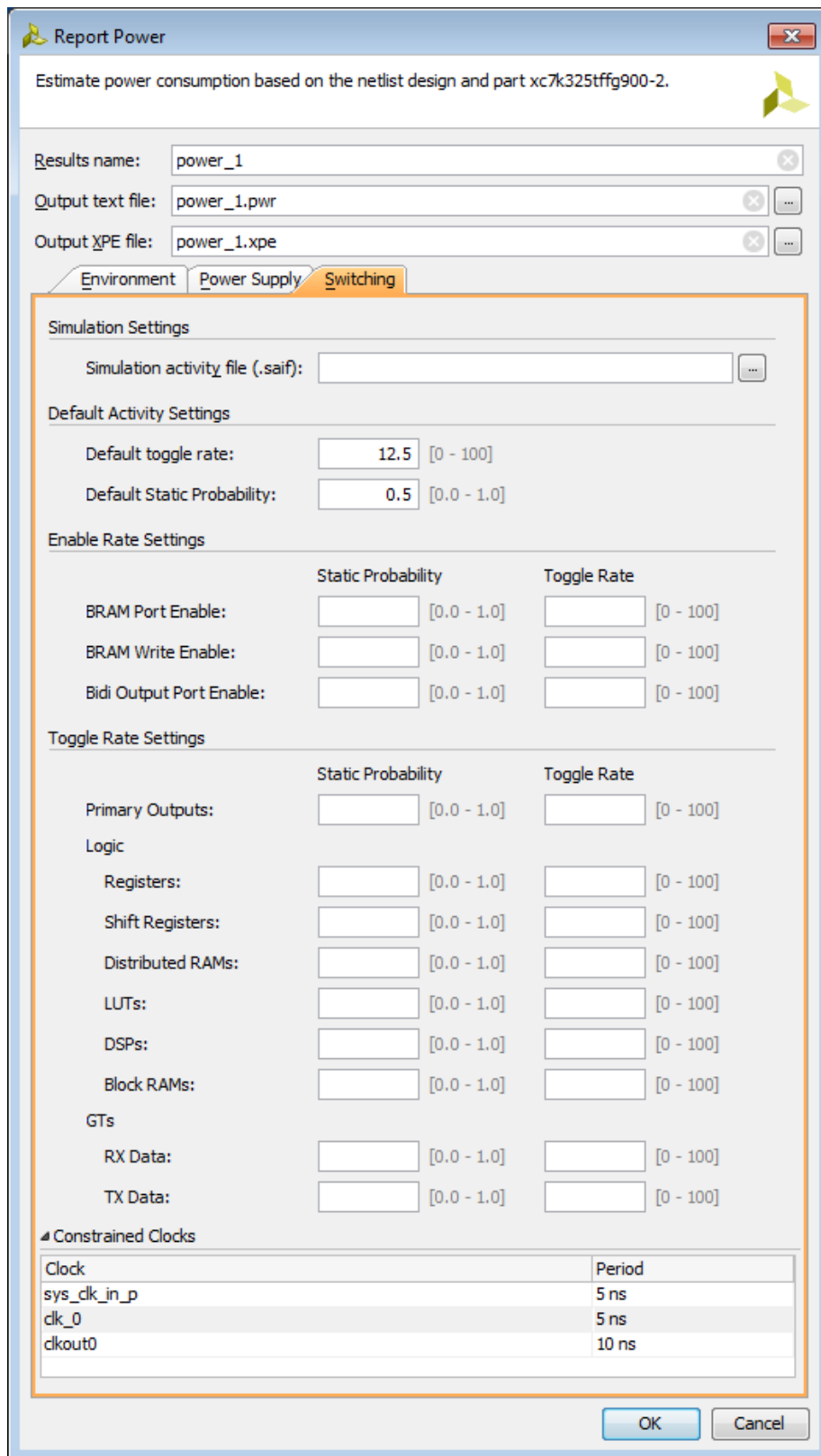


Figure 9: Report Power Dialog Box – Constrained Clocks

Step 4: Running Report Power

1. Click **OK** on the Report Power dialog box.

This runs the `report_power` command.

2. Examine the Power Report **power_1** generated in the Power tab of the results windows area in the Vivado IDE.



TIP: Due to continuous accuracy improvements in the Vivado tools, the actual power numbers you see might be slightly different than the ones that appear in the following figures.

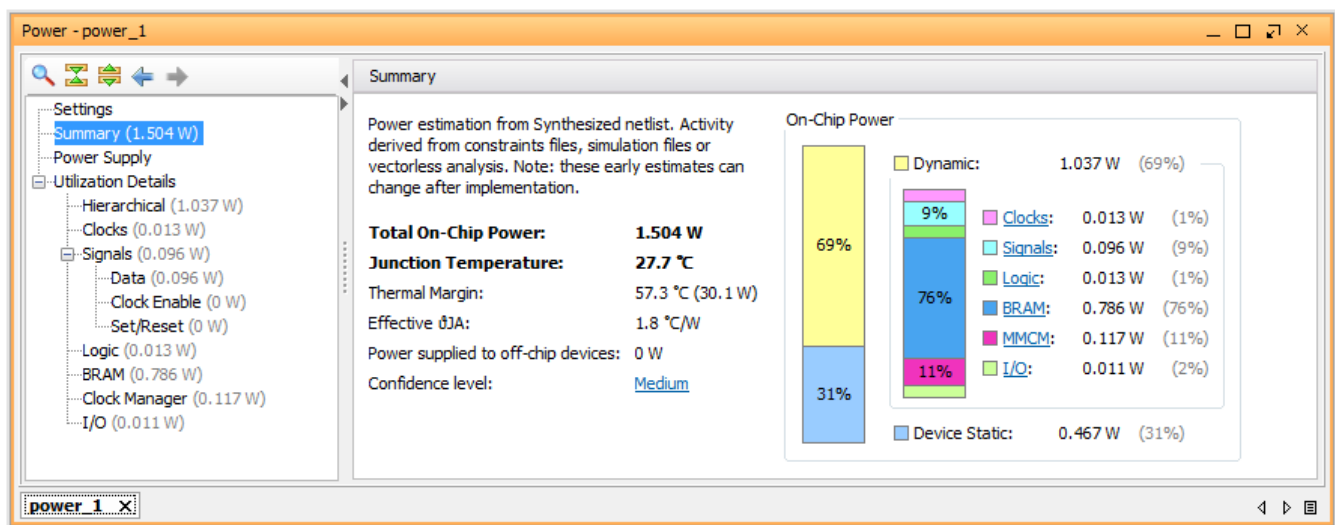


Figure 10: Power Report

3. Examine the power breakdown in the Power Report by block type (**Logic, BRAM, I/O**, etc.).
4. Examine the power supply breakdown in the **Power Supply** view.

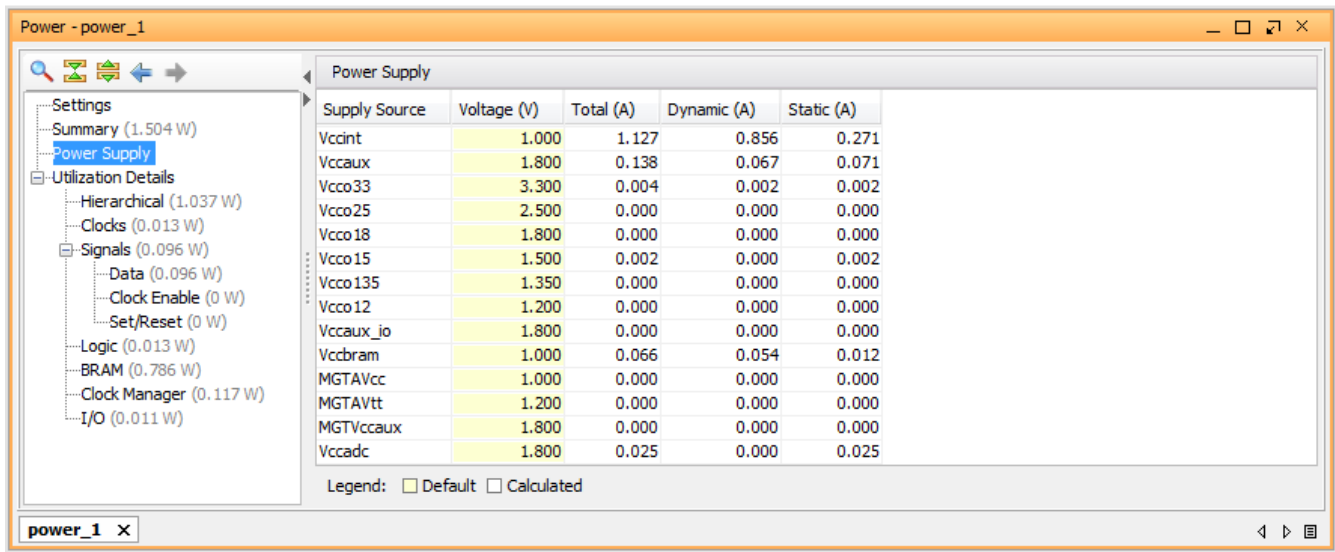


Figure 11: Power Report – Power Supply Display

5. Examine the Hierarchical breakdown of the power in the **Utilization Details > Hierarchical** view.

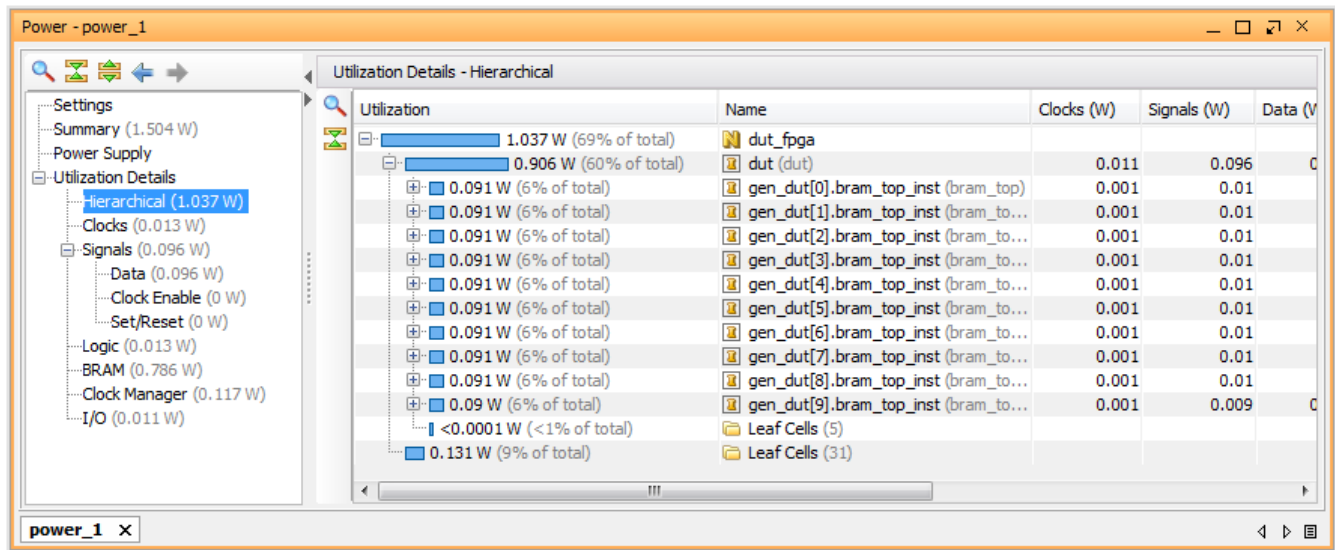


Figure 12: Power Report – Hierarchical Display

6. Examine the **Clocks** view and the various **Signals** views (**Data**, **Clock Enable**, and **Set/Reset**).

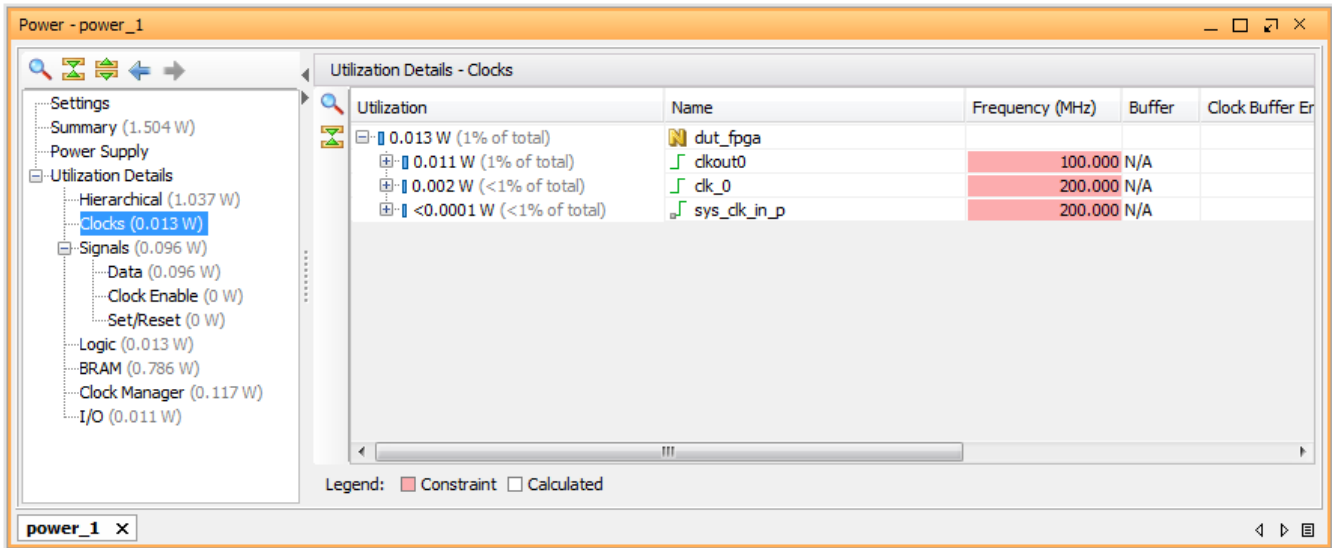


Figure 13: Power Report – Clocks Display

Step 5: Power Properties

1. Note the total power (**Total On-Chip Power**) in the Power Report **Summary** view.
2. Click the **Set/Reset** item in the Power Report.
3. Click on the **dut/reset** signal.

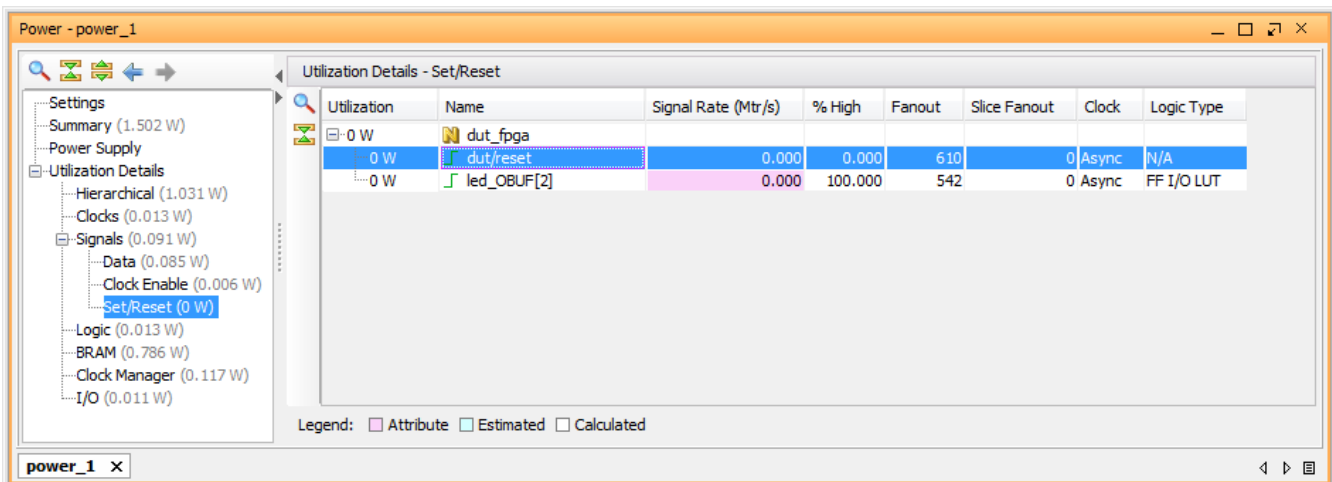


Figure 14: Power Report – Set/Reset Display

- Note that there is a **Power** tab in the Net Properties window, displaying Net Properties for the **dut/reset** signal. Click on **Load Power Properties** to get the power information the first time.

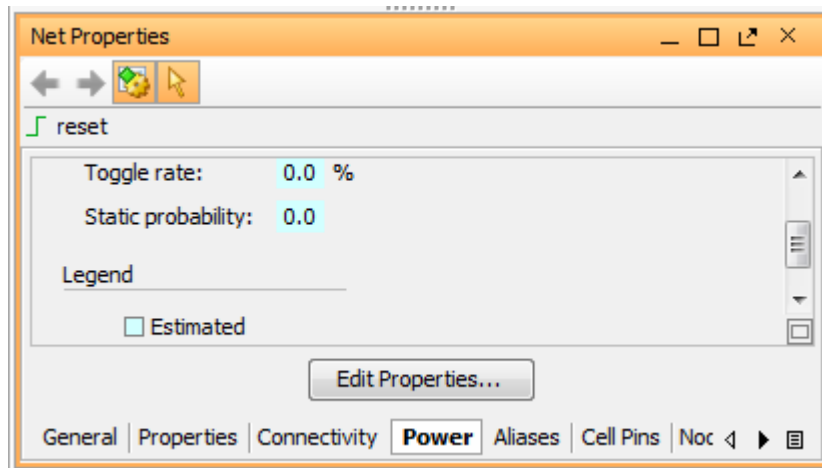


Figure 15: Properties Window – Set/Reset Net Properties

- Note the **Toggle rate** is 0% and the **Static probability** is 0 for the **dut/reset** signal, which indicates that Reset is always de-asserted in the design.

Step 6: Editing Power Properties and Refining the Power Analysis

Assume the Reset is asserted for 10% of the cycles in this design. Switching activity can be set accordingly to re-estimate the power.

- In the Net Properties window, click the **Edit Properties** button.
- In the Edit Power Properties dialog box, change the **Toggle rate** to 4% and the **Static probability** to 0.9.

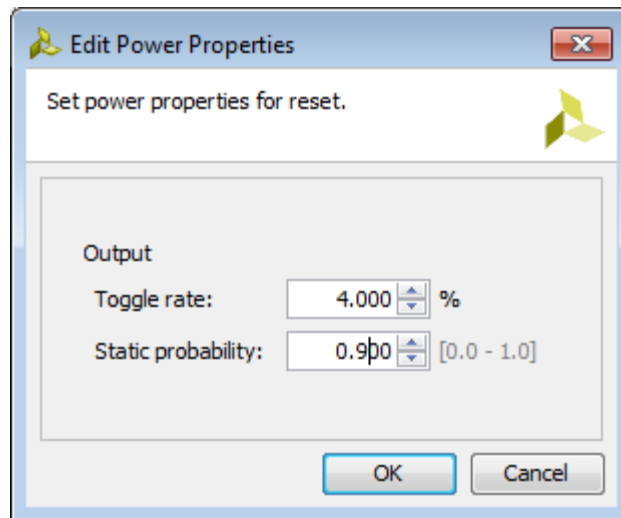


Figure 16: Edit Power Properties Dialog Box

3. Click **OK**. In the Net Properties window, observe that the **Toggle Rate** and **Static Probability** values turn a different color to indicate that they are user defined.

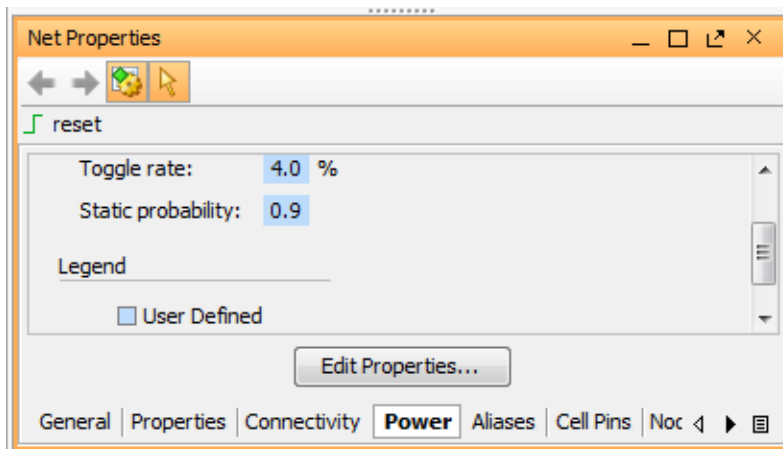


Figure 17: User Defined Color in Net Properties Window

You can also observe the equivalent Tcl command executed in the Tcl Console.

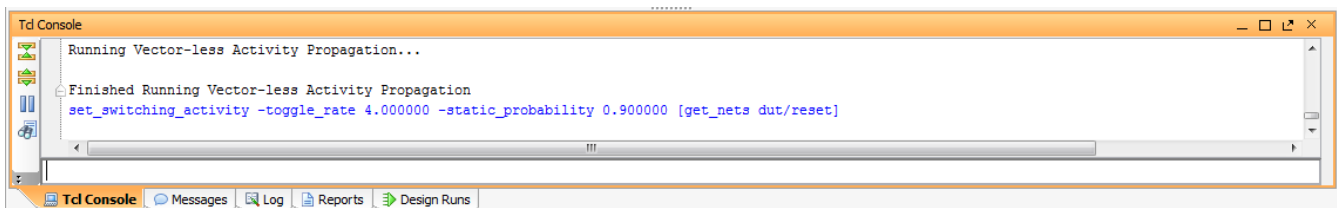


Figure 18: Equivalent Tcl Command for Net Property Update

4. Rerun Report Power (**Tools > Report > Report Power**).
5. Change the **Output text File** and **Output XPE File** to `power_2.pwr` and `power_2.xpe` respectively. Then click **OK**.
6. In the Power window, note the change in total power reported in the **power_2** report compared to the **power_1** report. The total power has *decreased* due to the change in the **Signal Rate** for the **dut_reset** signal. Note that the **Signal Rate** of the **dut_reset** signal is now color coded as being **User Defined** in both the properties window and the Set/Reset view of the Power Report.

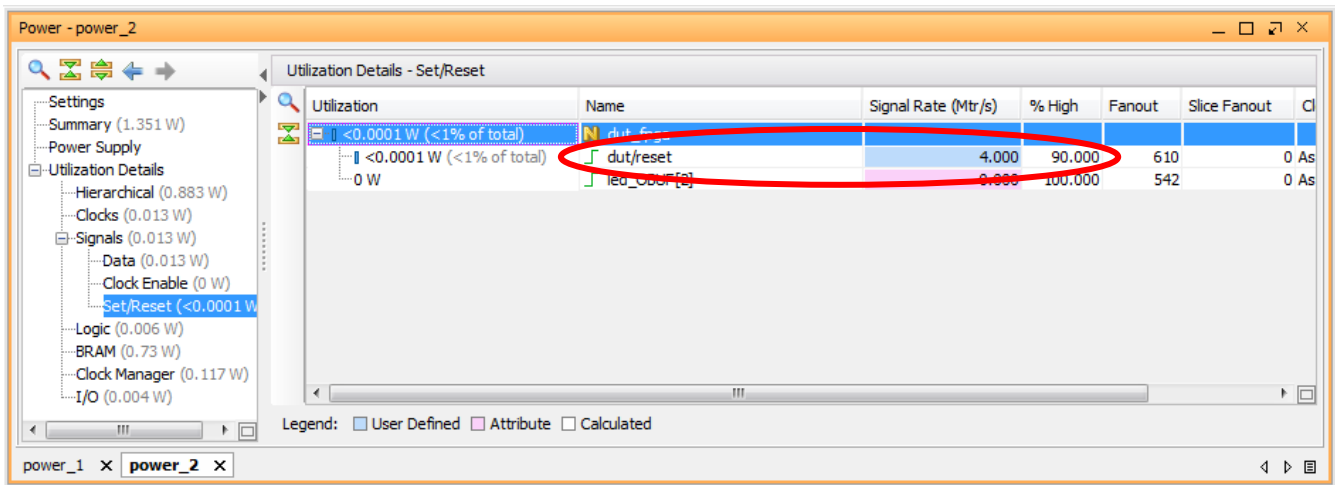


Figure 19: Power Report – User Defined Signal Rate

We recommend using this methodology to double check the signal rates and %high values of high impact I/O ports, control signals (such as resets and clock enables) and high fanout nets. This is an opportunity to guide the Report Power tool to the right estimation scenario.

Refer to the *Vivado Design Suite User Guide: Power Analysis and Optimization* ([UG907](#)) for more information on switching activity.

TIP: In Tcl, use the `set_switching_activity` command to change the signal rate and static probability of signals and use `report_switching_activity` to query the values that were set on the signals.



```
set_switching_activity -signal_rate 4 -static_probability 0.9
[get_nets dut/reset]

report_switching_activity [get_nets dut/reset]
```



IMPORTANT: Switching activity can also be specified in terms of toggle rate. Toggle rate is always associated with a clock. The primary ports can be associated with a specific clock using the `set_input_delay` command. For example:

```
set_input_delay -clock CLK 1 [get_nets dut/reset]
```

If no clock association is found, Report Power will associate the ports with respect to the capturing clock.

Now you can set the toggle rate on the reset net:

```
set_switching_activity -toggle_rate 4 -static_probability 0.9  
[get_nets dut/reset]
```

This is equivalent to a signal rate of 4 MTr/s ($signal_rate = toggle_rate * Freq = 4 * 100MHz$).

Step 7: Running Functional Simulation with SAIF Output

Now that you have created a Vivado Design Suite project for the tutorial design, you can set up and launch the Vivado simulator to run post-synthesis functional simulation. Simulation will generate a switching activity values file (SAIF) that will enable you to do more accurate power estimation on your design.

1. In the Flow Navigator, click **Simulation Settings** to set the behavioral simulation properties.
2. In the Project Settings dialog box, note that the following Simulation defaults are automatically set:

Simulator language: Mixed

Simulation set: sim_1

Simulation top-module name: testbench

- In the Elaboration tab, make sure the **debug_level** is set to **typical**, which is the default value.

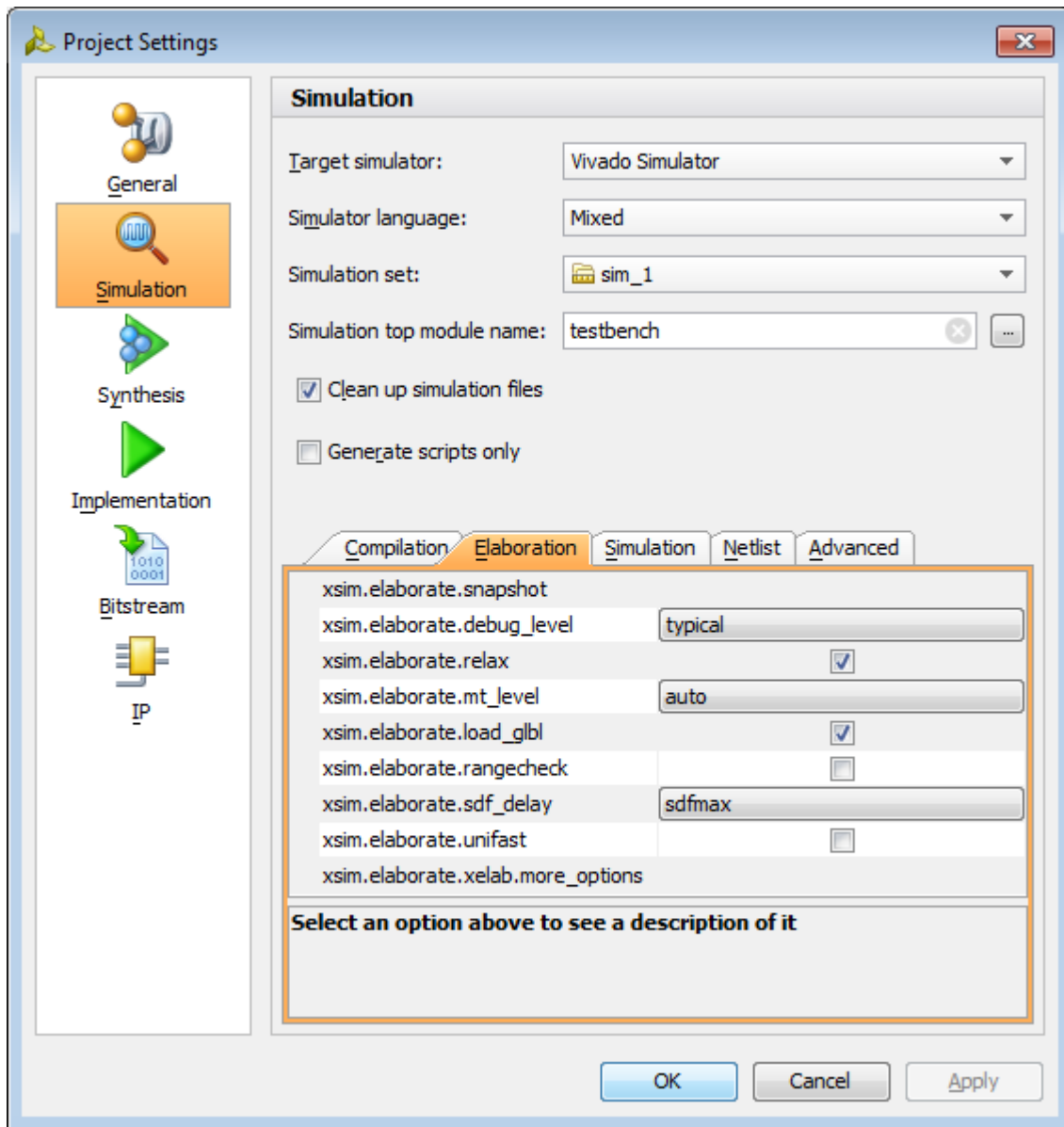


Figure 20: Simulation Settings – Elaboration

- In the Simulation tab set the **xsim.simulate.uut** (design hierarchy) to `testbench/dut_fpga`.
- In the Simulation tab set the **xsim.simulate.saif** (SAIF file name) to `power_tutorial_func.saif`. Observe that the **xsim.simulate.runtime** is 1000ns.
- Click **OK**.

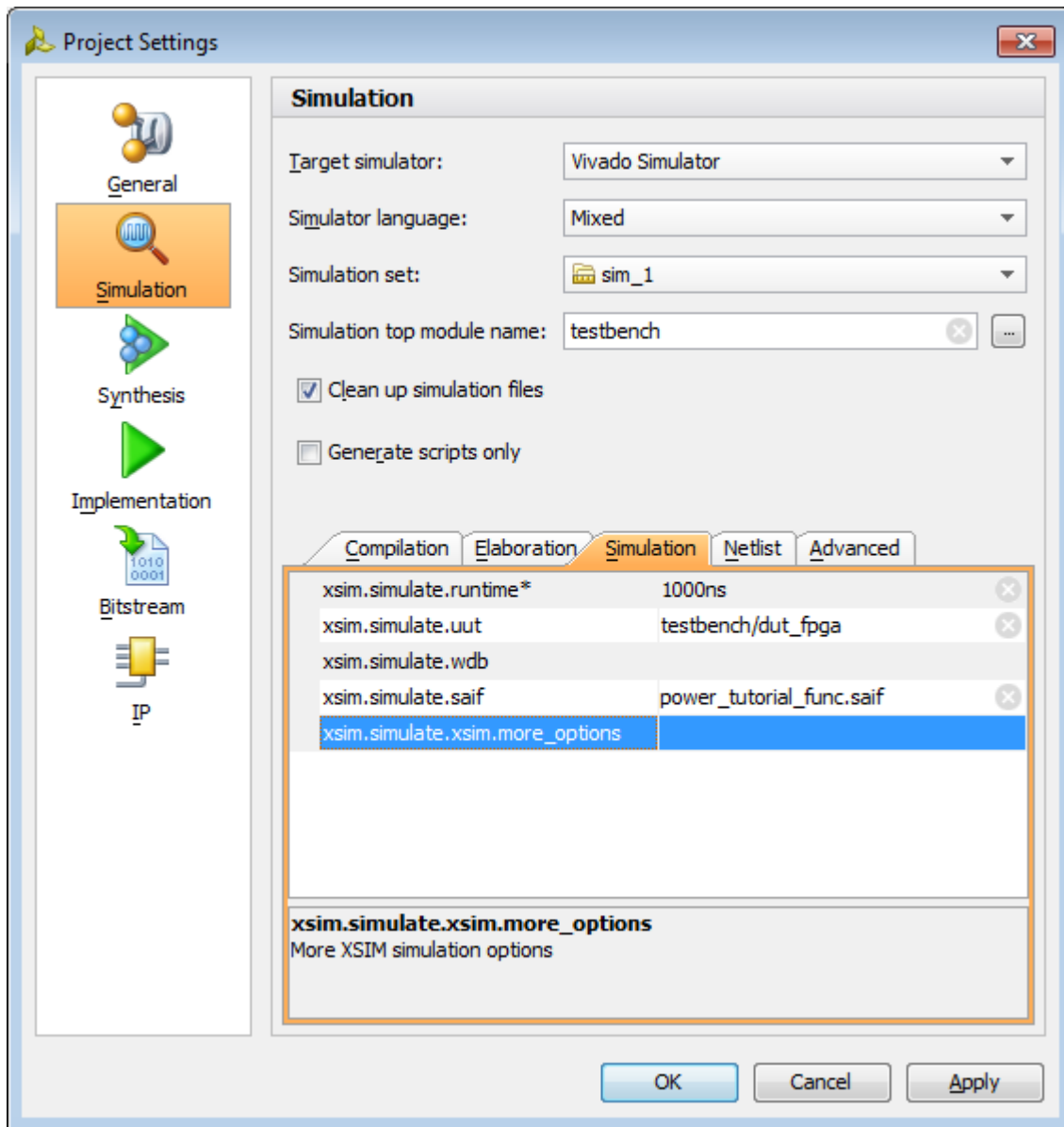


Figure 21: Simulation Settings – Entering SAIF File Setting

With the simulation settings properly configured, you can launch the Vivado simulator to perform a behavioral simulation of the design.

7. In the Flow Navigator, click **Run Simulation > Run Post-Synthesis Functional Simulation**.

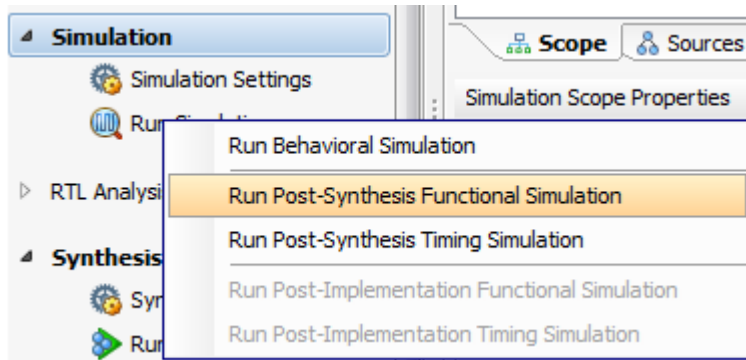


Figure 22: Running Post-Synthesis Functional Simulation from Flow Navigator

When you launch the Run Post-Synthesis Functional Simulation command, the Vivado simulator is invoked to run the simulation.

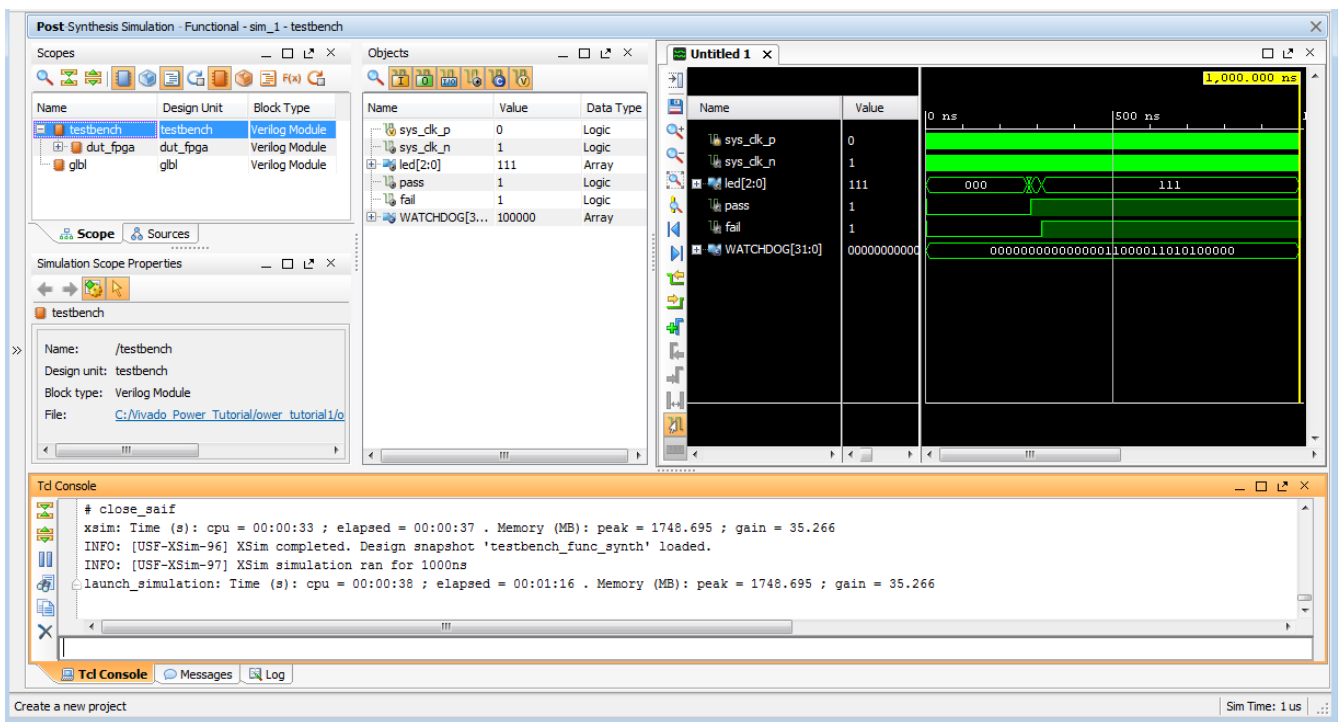


Figure 23: Post-Synthesis Functional Simulation Window in the Vivado IDE

Once the simulation completes, close the simulation window by clicking the **x** at the top right corner of the simulation window.

Step 8: Incorporating SAIF Data into Power Analysis

The SAIF output file requested in the simulation run is generated in the project directory. To further guide the power analysis algorithm we use this SAIF file.

1. Ensure the SAIF file requested is generated. Check to see that the SAIF file requested in the simulation settings prior to running simulation appears in this directory:

```
/<project_directory>/power_tutorial1/power_tutorial1.sim/sim_1/synth/  
func/power_tutorial_func.saif
```

2. In the Flow Navigator window, click on **Synthesized Design**.
3. In the main menu bar, select **Tools > Report > Report Power**.
4. In the Report Power dialog box, make the following changes:
 - o Set the **Results name** to `power_3`
 - o Set the **Output text File** to `power_3.pwr`
 - o Set the **Output XPE File** to `power_3.xpe`
5. In the Report Power dialog box **Environment** tab, set the **Process** (the process corner) to **maximum**.
6. In the Report Power dialog box **Switching** tab, specify the SAIF file location.

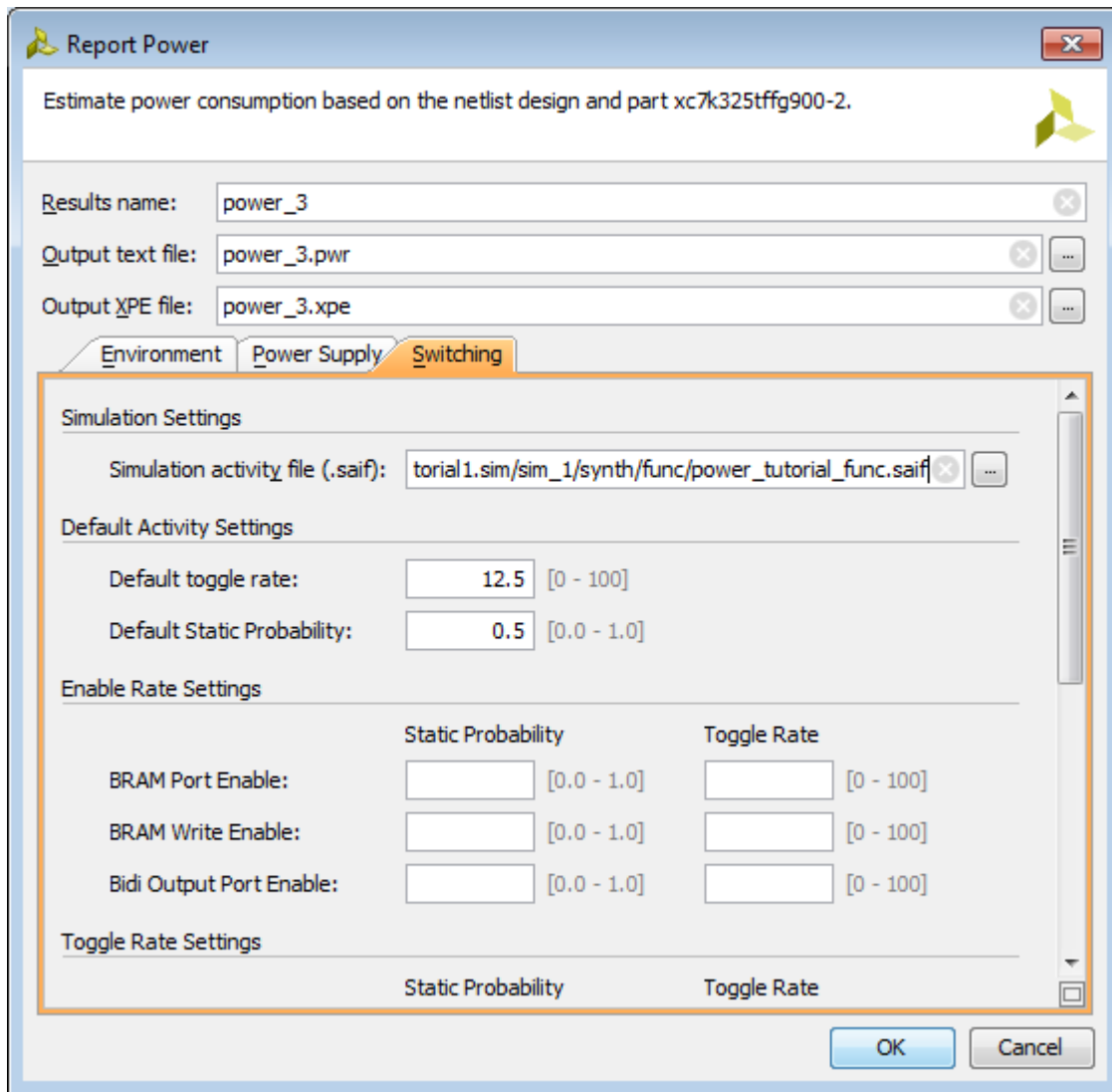


Figure 24: Specifying SAIF File Location

7. Click **OK** in the Report Power dialog box.

The `report_power` command runs, and the Power Report **power_3** is generated in the Power tab of the results windows area.

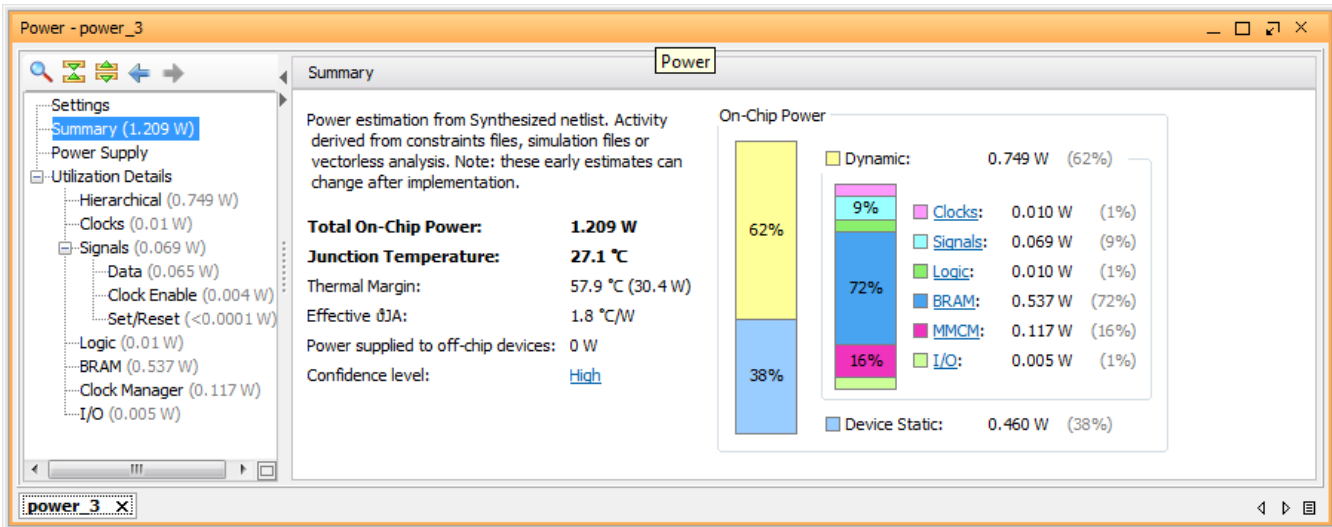


Figure 25: Power Report – Summary

Note that the SAIF annotation results are displayed in the Tcl Console. Make sure all the design nets are matched with simulation nets, to achieve better accuracy by including Simulation data.

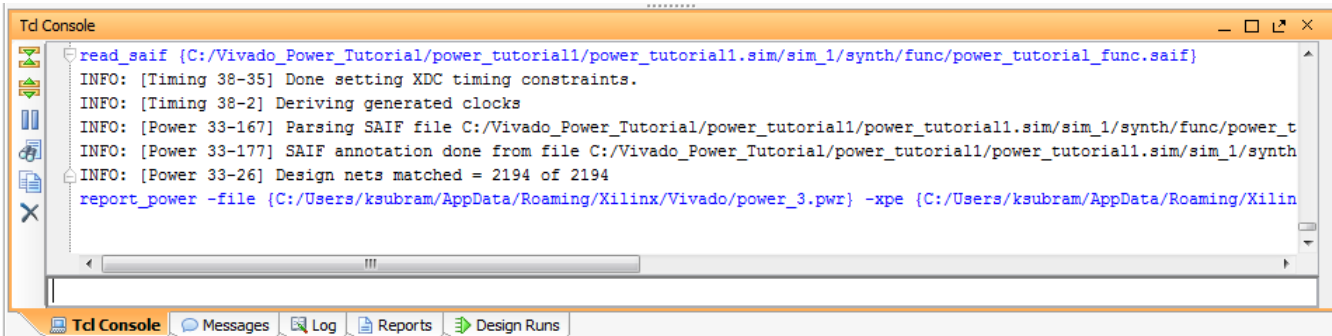


Figure 26: Tcl Console Log – Reading SAIF Data

- Go to the **I/O** view in the Power Report. Note that all the I/O port activity data has been set from simulation data we specified.

The data is color coded to indicate activity rates read from the **Simulation** output file.

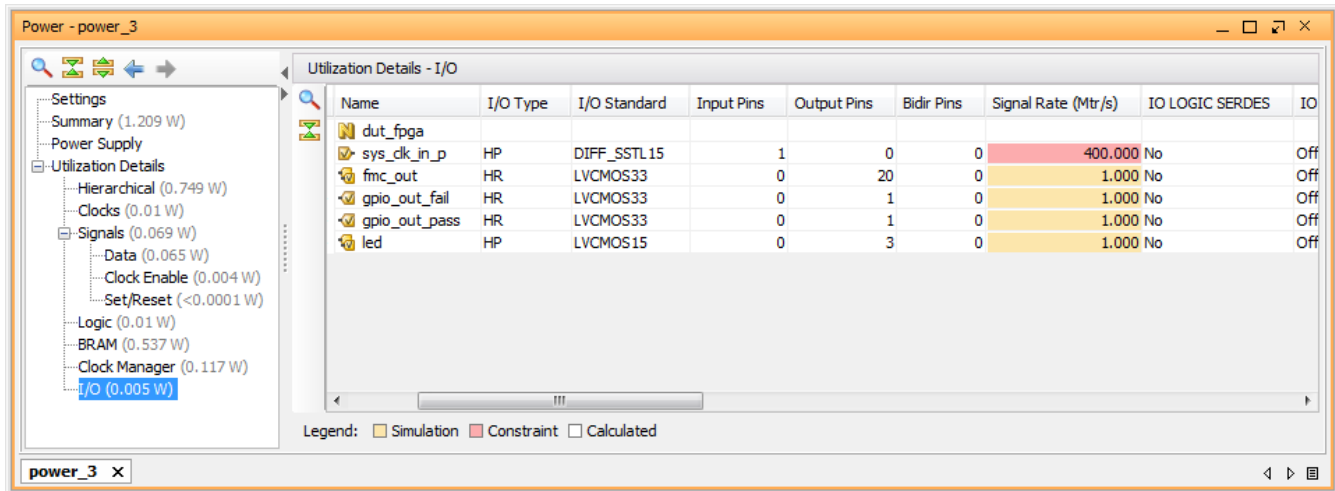


Figure 27: Power Report – I/O Display

- Note the difference in total power numbers (**Total On-Chip Power** in the **Summary** view) between a pure vectorless run in the **power_1** results versus with the behavioral simulation data in the **power_3** results. Also note that the **dut/reset** values are overwritten by simulation SAIF data.

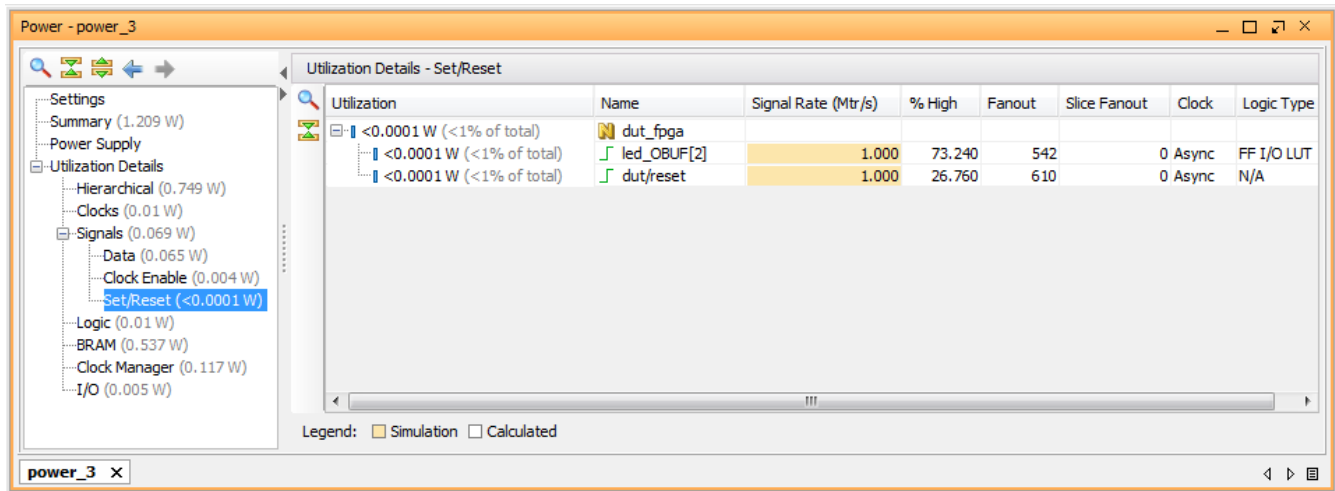


Figure 28: Power Report – Set/Reset Display

Step 9: Implementing the Design

This tutorial helps you understand Power Analysis with and without Power Optimization. In this step, you will run Implementation *without* Power Optimization.

1. In the Flow Navigator, select **Implementation Settings**.
2. In the **Opt Design** settings, select the **NoBramPowerOpt** option for `-directive`.

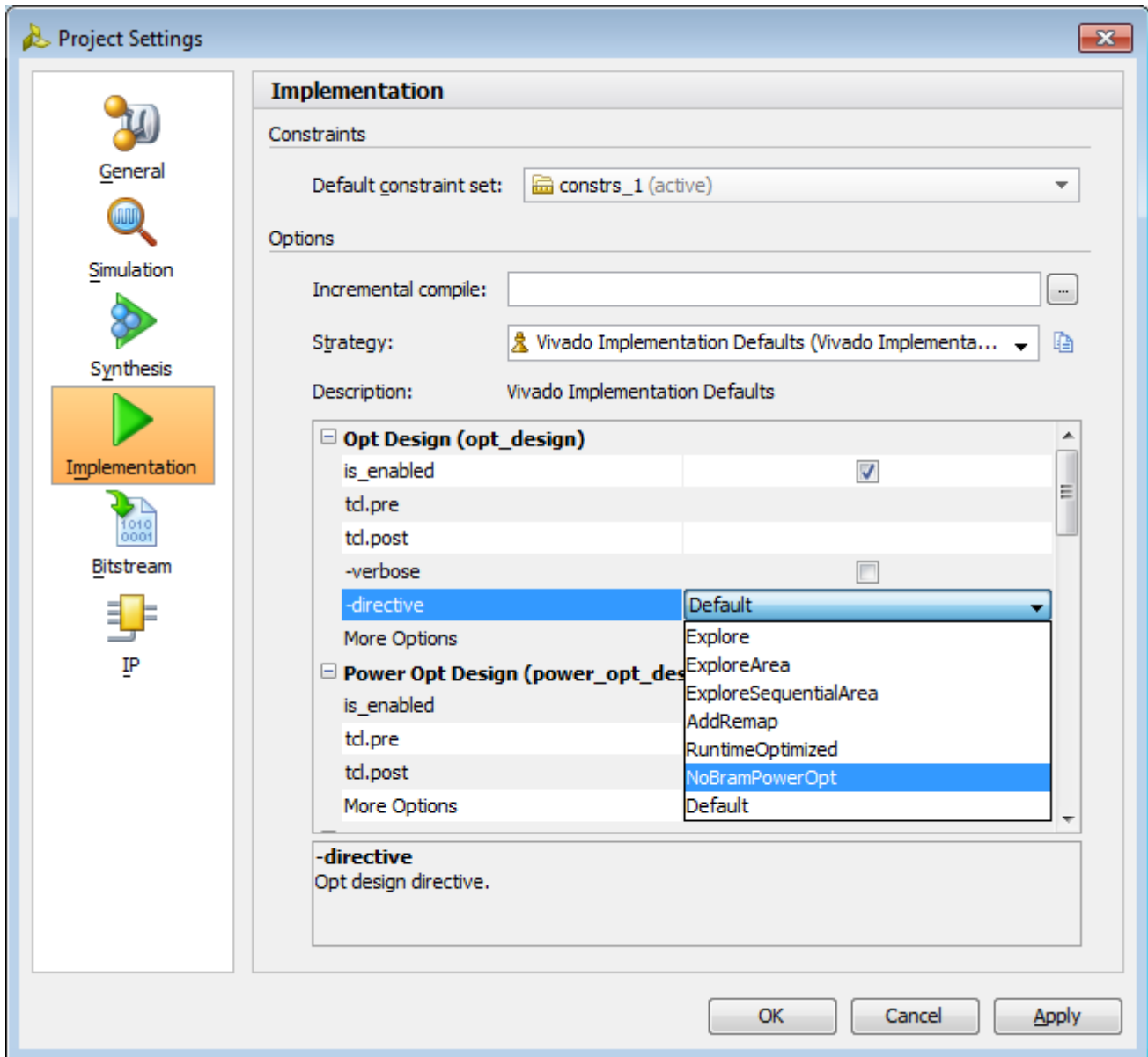


Figure 29: Disabling Power Optimization

3. In the Flow Navigator, click **Run** Implementation.
4. When prompted to **Save project before launching implementation** click **Don't Save**.

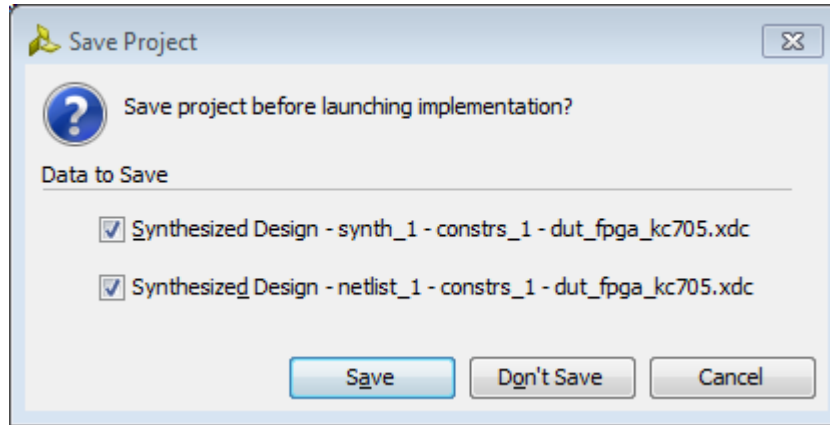


Figure 30: Click "Don't Save" Here

Lab 2: Vivado Simulator Timing Simulation and Power Analysis

Introduction

In this lab, you will learn about generating a SAIF file after running a timing level simulation using the Vivado simulator. The lab will take you through the steps for SAIF file creation, running timing simulation, and estimating power using the SAIF data.

Step 1: Open the Implemented Design

1. In the Implementation Complete dialog box, select **Open Implemented Design** and click **OK** to open the implemented design. When prompted to Save project before launching implementation click **Don't Save**.

Now you are ready to set up and launch the Vivado simulator to run post implementation timing simulation. You will set the timing simulation properties in the Vivado IDE, then run the timing simulation.

2. In the Flow Navigator, click **Simulation Settings** to set the timing simulation properties.
3. In the Project Settings dialog box, note that the following defaults are automatically set:

Simulation set: `sim_1`

Simulation top-module name: `testbench`

4. In the Elaboration tab, make sure that **debug_level** is set to **typical**, which is the default value.
5. In the Simulation tab set the **Design Under Test Instance** to `testbench/dut_fpga` and the **SAIF Filename** to `power_tutorial_timing_xsim.saif`.
6. Observe that the **Simulation Run Time** is 1000ns.
7. Click **OK**.

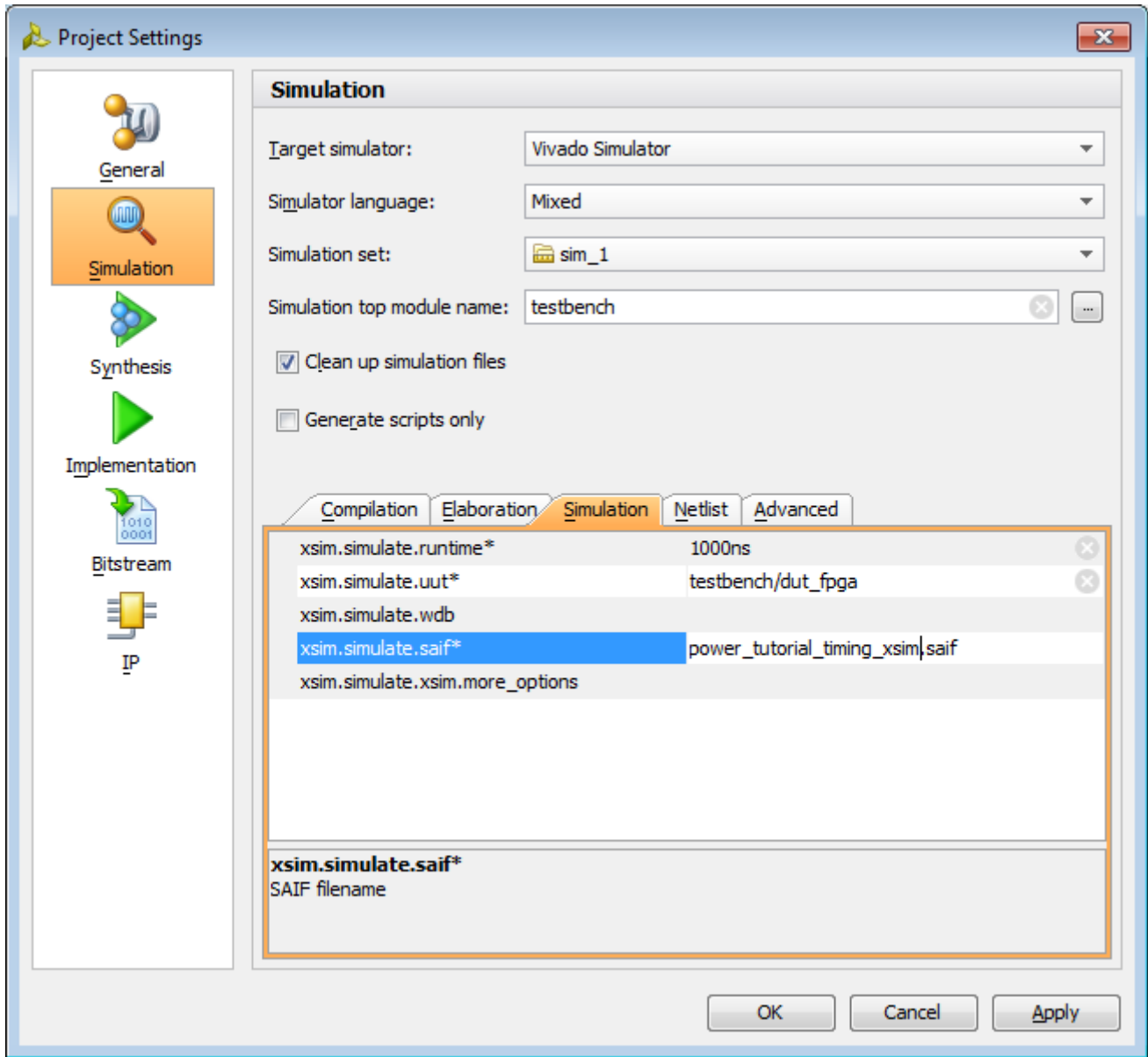


Figure 31: Timing Simulation Settings – Simulation Tab

With the simulation settings properly configured, you can launch the Vivado simulator to perform a timing simulation of the design.

8. In the Flow Navigator, click **Run Simulation >Run Post-Implementation Timing Simulation**.

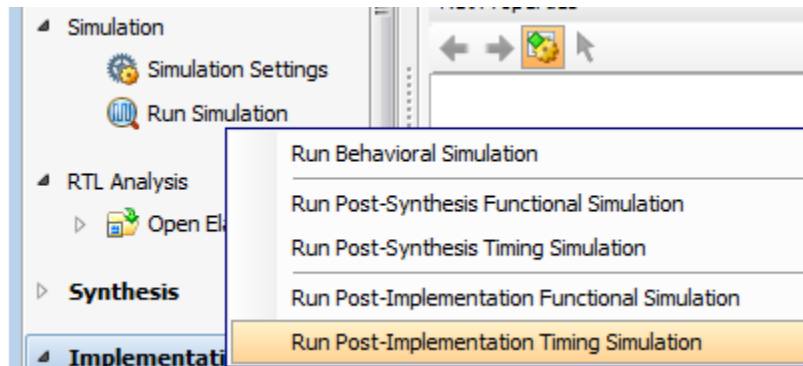


Figure 32: Running Post-Implementation Timing Simulation

9. After the Vivado simulator has finished simulating the design, ensure that the SAIF file requested has been generated. Check to see that the SAIF file requested in the simulation settings prior to running simulation appears in this directory:

```
/<project_directory>/power_tutorial1/power_tutorial_1.sim/  
sim_1/impl/timing/power_tutorial_timing_xsim.saif
```

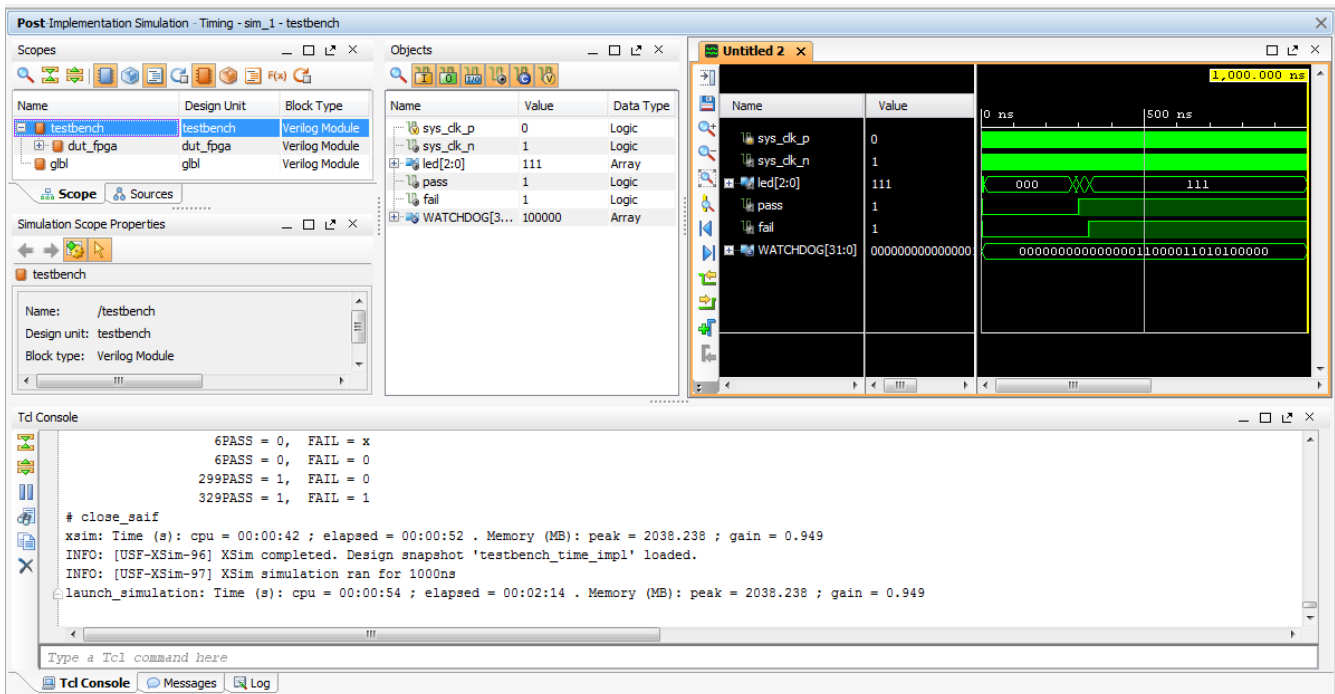


Figure 33: Post-Implementation Timing Simulation using the Vivado Simulator

Step 2: Running Report Power in Vectorless Mode

1. In the Flow Navigator, select **Implemented Design > Report Power** to open the Report Power dialog box.

You can also select **Tools > Report > Report Power** from the main menu bar.

2. In the Report Power dialog box **Environment** tab, make sure the **Process** is set to **maximum** and click **OK**.

The Report Power command creates a Power Report under the **power_1** tab in the results windows area.

3. Note the total power (**Total On-Chip Power**) in the power report **Summary** page (Figure 34).

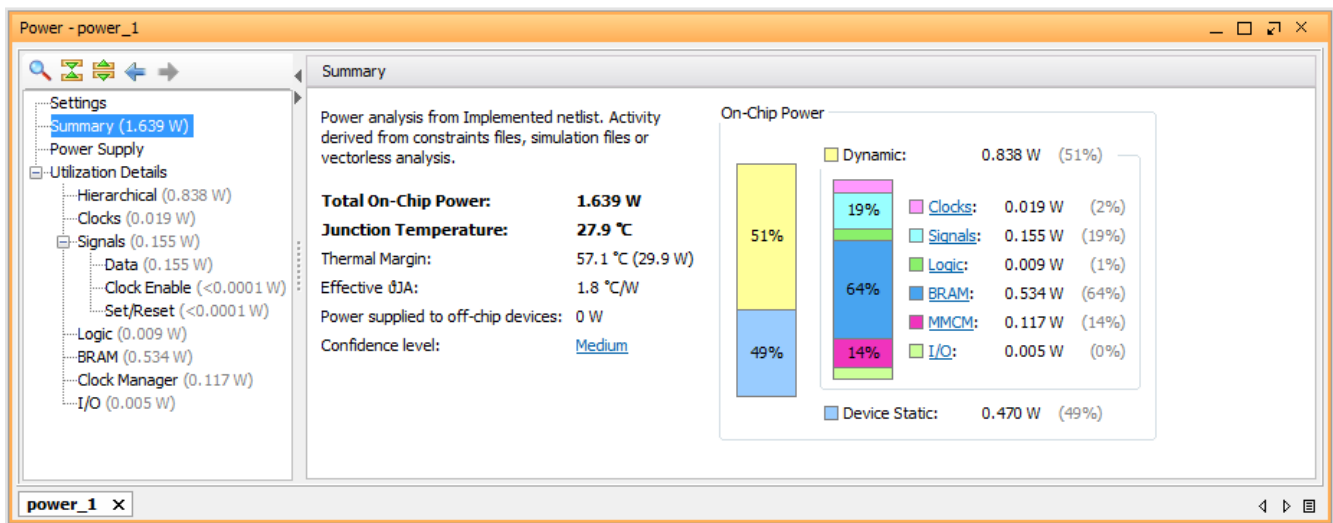


Figure 34: Running Post-Implementation Report Power in Vectorless Mode

Vectorless Analysis is done based on default switching activity specification on the primary ports and the design clocks.

Refer to the *Vivado Design Suite User Guide: Power Analysis and Optimization* ([UG907](#)) for more information on Vectorless Power Analysis.

Step 3: Running Report Power with Vivado Simulator SAIF Data

The project directory contains the SAIF output file requested in the previous timing simulation run. We use this SAIF file – a “Switching Activity Interchange Format” file – to further guide the power analysis algorithm.

1. In the main menu bar, select **Tools > Report > Report Power**.
2. In the Report Power dialog box, specify the SAIF file location in the **Switching** tab.

The SAIF file, which was requested in the simulation settings prior to running timing simulation, should appear here:

```
 /<project_directory>/power_tutorial1/power_tutorial1.sim/  
 sim_1/impl/timing/power_tutorial_timing_xsim.saif
```

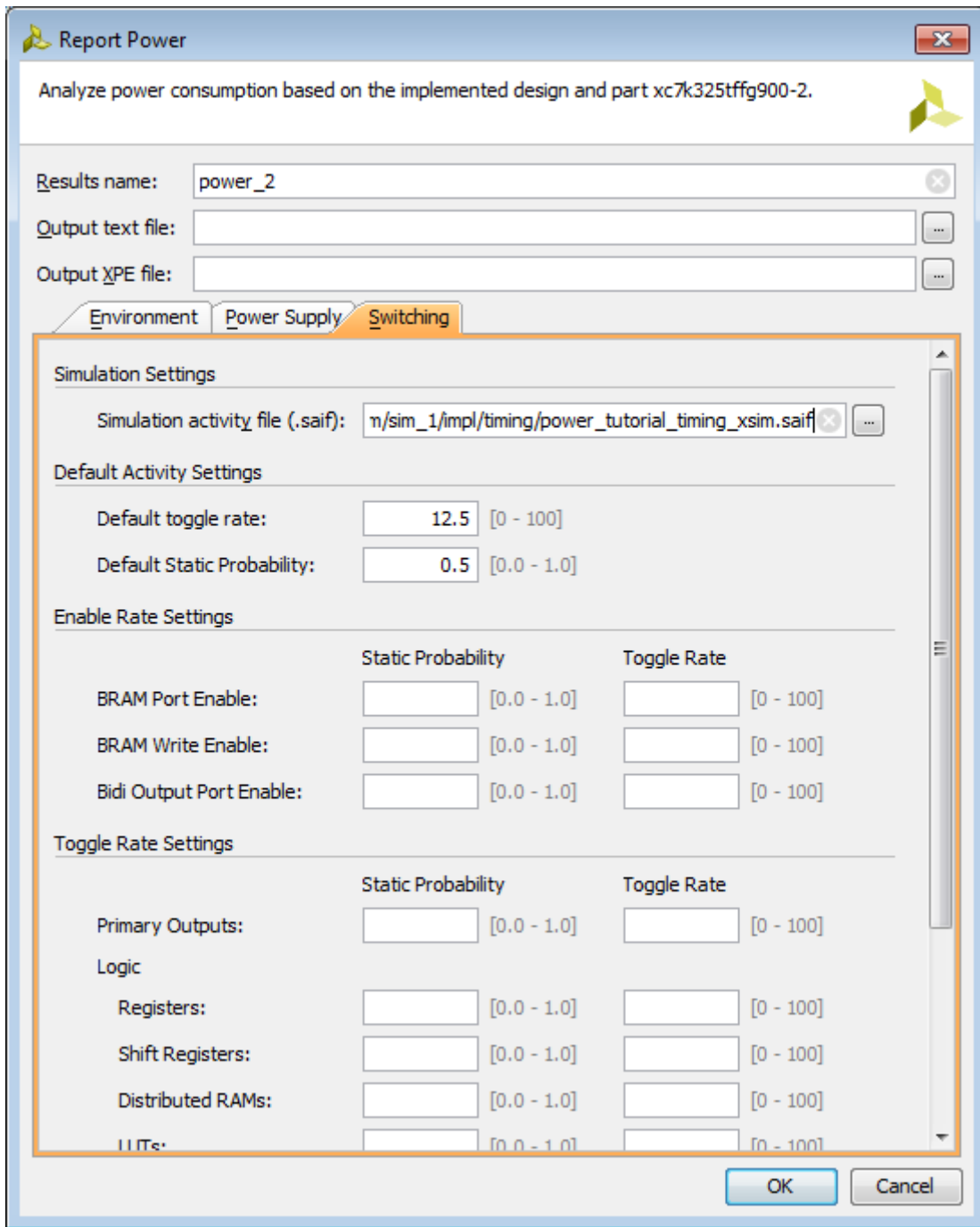


Figure 35: Specifying SAIF File Location

3. Click **OK** in the Report Power dialog box.

After the Report Power command completes, the **Power** tab in the results windows area displays Power Report **power_2**.

In the Tcl console, observe that the SAIF file is read successfully and that 100% of the design nets are matched. This assures you that the generated SAIF file is correct and matched with all design nets.

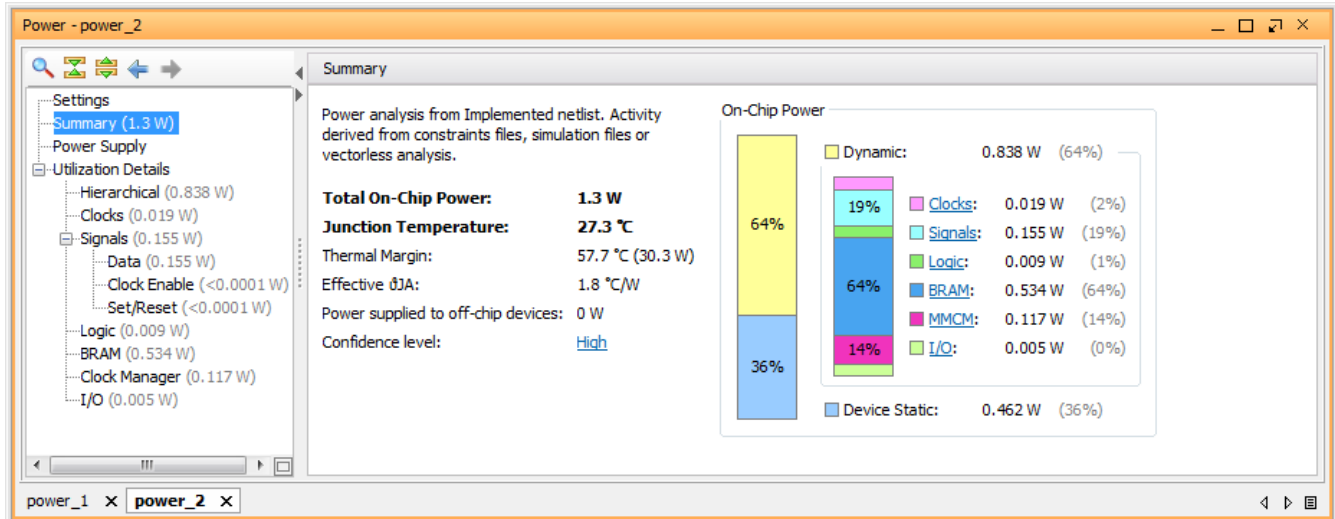


Figure 36: Power Report with SAIF Data

- Note the change in total power (**Total On-Chip Power** in the **Summary** view) in the **power_2** report compared to the **power_1** report. The total power estimated in the report generated with SAIF file data will be different than the total power estimated in the vectorless run (**power_1** results).
- Examine the summary and block level (**On-Chip Power**) power distribution in the **Summary** view of the Power Report.
- Go to the **Utilization Details > Signals > Data** view in the Power Report. Note that all the **Signal Rate** data has been set from simulation data the SAIF file provided.

The data is color coded to indicate activity rates read from the **Simulation** output file.

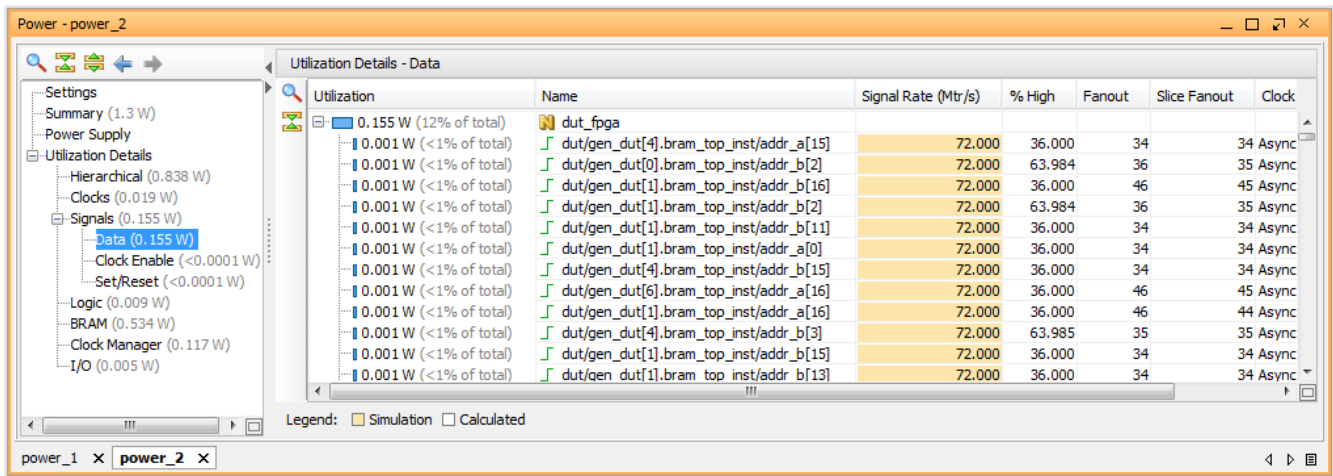


Figure 37: Power Report Signals > Data View - Simulation SAIF Data for Signal Rates

- In the **Summary** view of the **power_1** report (the report generated by the vectorless analysis), click on the Confidence level (**Low** in Figure 38).

The Confidence Level is a measurement of the accuracy and the completeness of the input data Report Power uses as it performs a power analysis.

Notice that for the vectorless analysis the **Confidence Level** is **Low**, since for the **I/O Activity** more than 75% of inputs are missing user specification and for **Internal Activity** less than 25% of internal nodes are user specified.

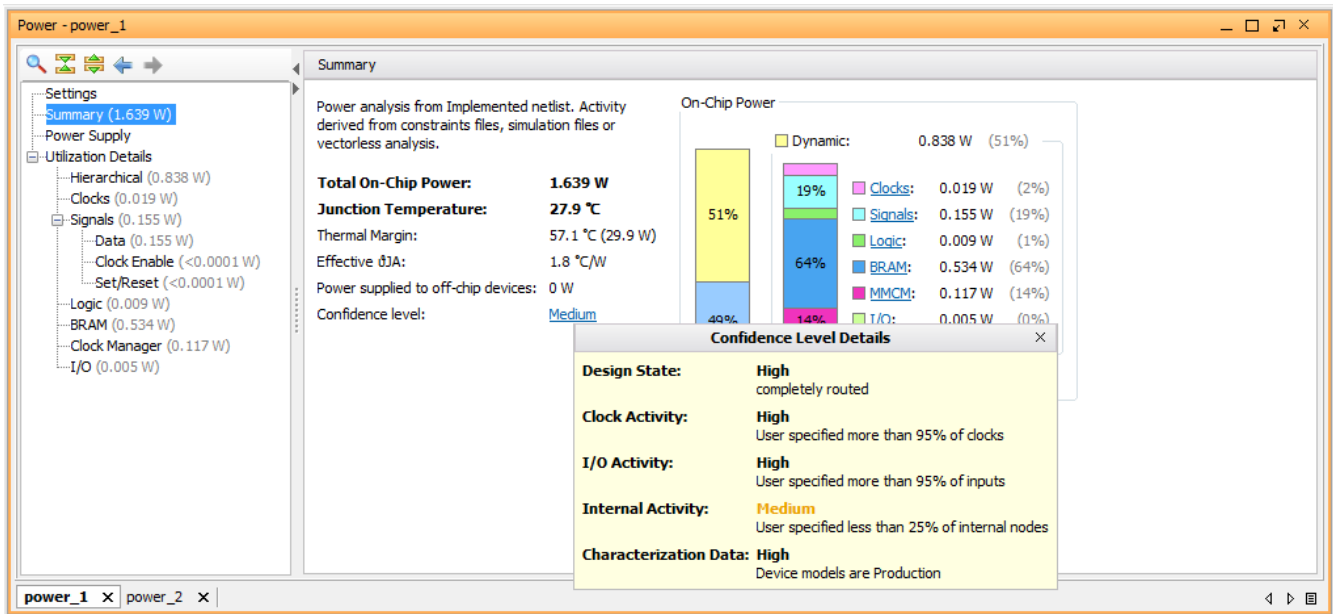


Figure 38: Confidence Level for Vectorless Power Analysis (power_1)

- In the **Summary** view of the **power_2** report (the report generated by the analysis for which you specified a SAIF file as input), click on the Confidence level (**High** in the following figure).

Notice that the **Confidence Level** has increased to **High**, since for the I/O Activity more than 95% of inputs are user specified and for **Internal Activity** more than 25% of internal nodes are user specified.

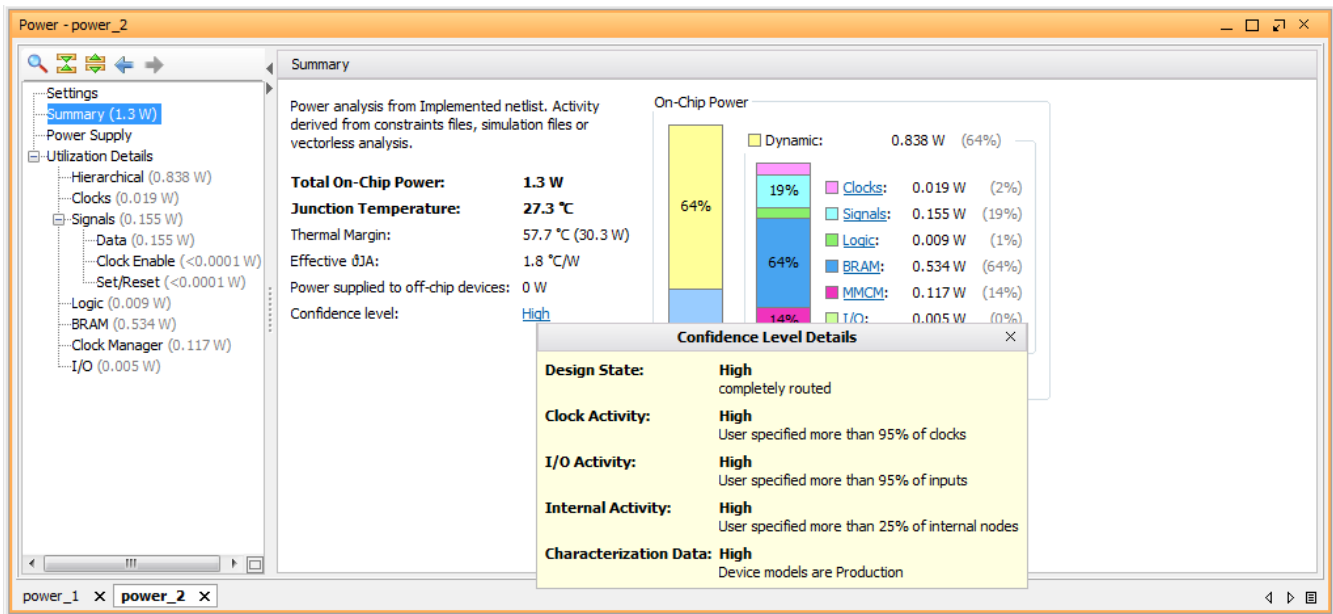


Figure 39: Confidence Level for Power Analysis with SAIF Input (power_2)

Lab 3: QuestaSim Timing Simulation and Power Analysis

Introduction

In this lab, you will learn about generating a SAIF file after running a timing level simulation using a QuestaSim simulator. The lab will take you through the steps for SAIF file creation, running timing simulation, and estimating power using the SAIF data.



IMPORTANT: Make sure the Vivado Design Suite knows where to pick up the QuestaSim tool. You can either:

Manually set the path to ModelSim/QuartaSim using the `$PATH` environment variable

OR

From the **Tools > Options > General** dialog box, define the path to the simulator in the Vivado IDE under the **3rd Party Tools** section: **QuartaSim/ModelSim install path**.

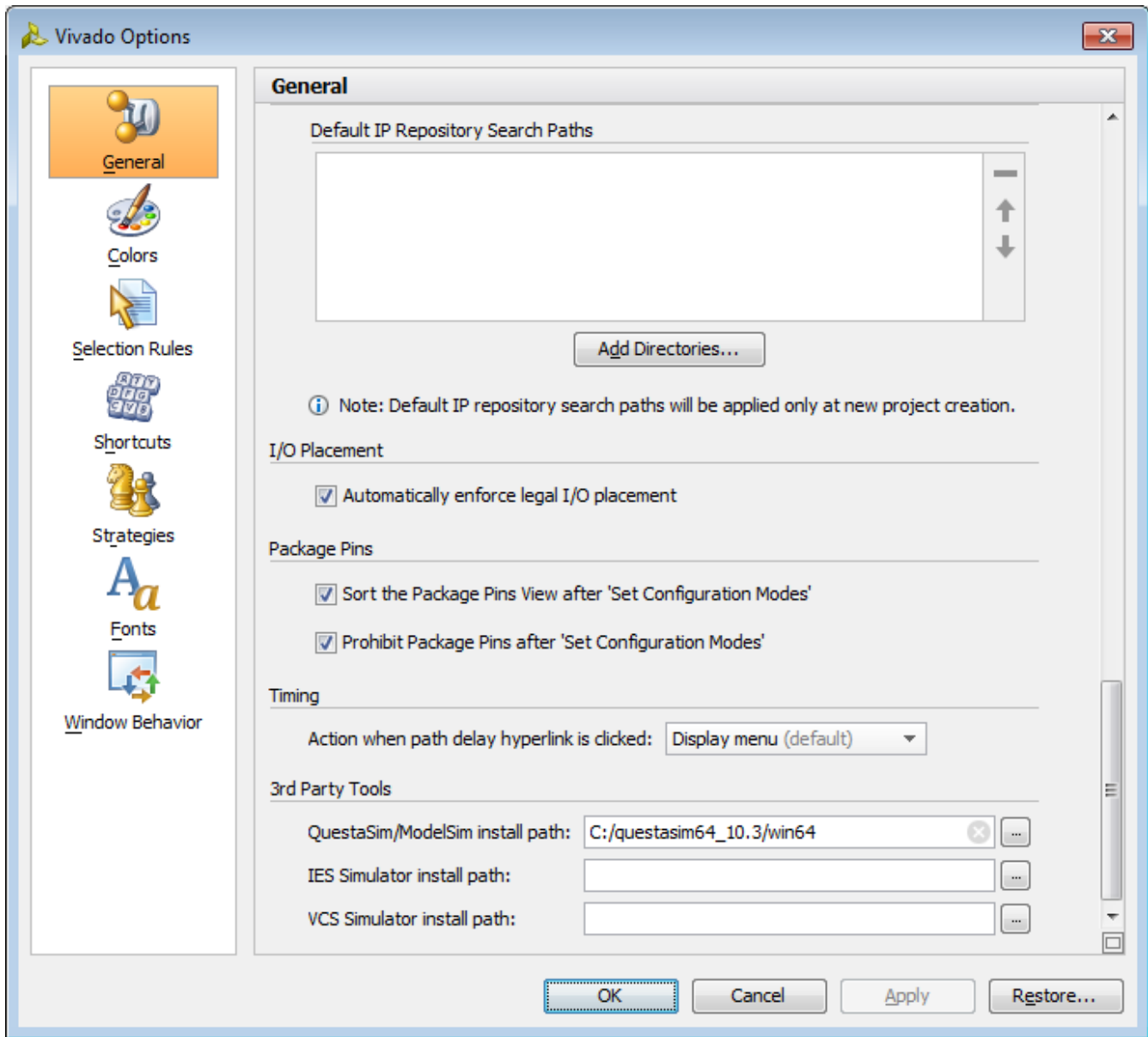


Figure 40: Setting the QuestaSim Installation Path

Step 1: Set Up to Run Timing Simulation in QuestaSim

Now you are ready to set up and launch the QuestaSim simulator to run post-implementation timing simulation. You will set the timing simulation properties in the Vivado IDE, and run the timing simulation.

1. In the Flow Navigator, select **Implemented Design**.
2. In the Flow Navigator, click **Simulation Settings** to set the timing simulation properties.

3. In the Project Settings dialog box, set the **Target simulator** to **QuestaSim/ModelSim**.
4. Click **Yes** to **Ok to change your target simulator to 'QuestaSim/ModelSim'?**

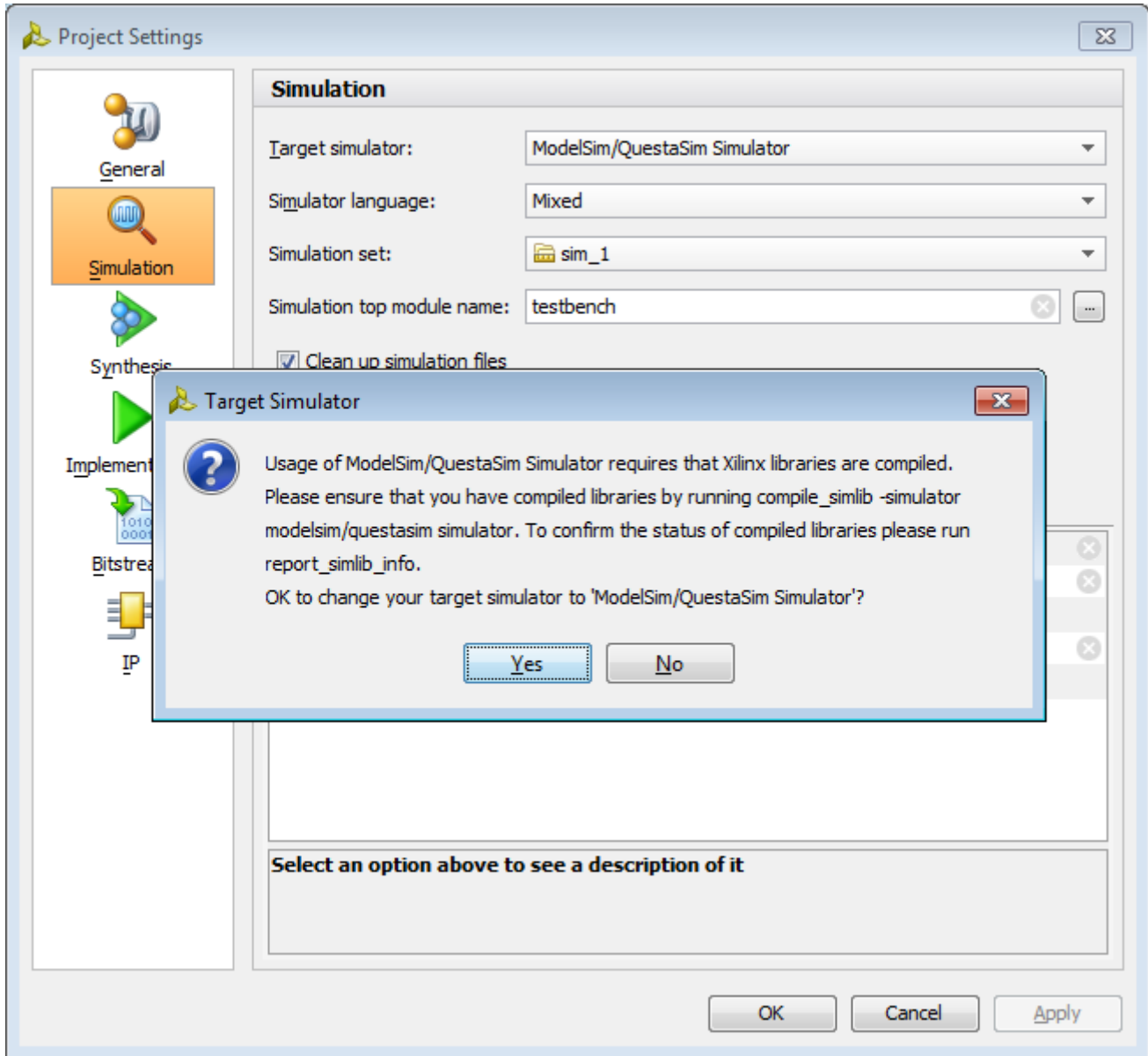


Figure 41: Simulation Settings – Changing the Target Simulator

5. Make sure the **Compiled library location** points to a valid location for the compiled Xilinx simulation libraries.
 - To create new compiled libraries:
 - a. In the main menu, select **Tools > Compile Simulation Libraries**.

- b. In the Compile Simulation Libraries dialog box, set the **Simulator** to **ModelSim** or **QuestaSim**.
- c. Specify the **Compiled library location**.
- d. Click **Compile** to generate compiled libraries.

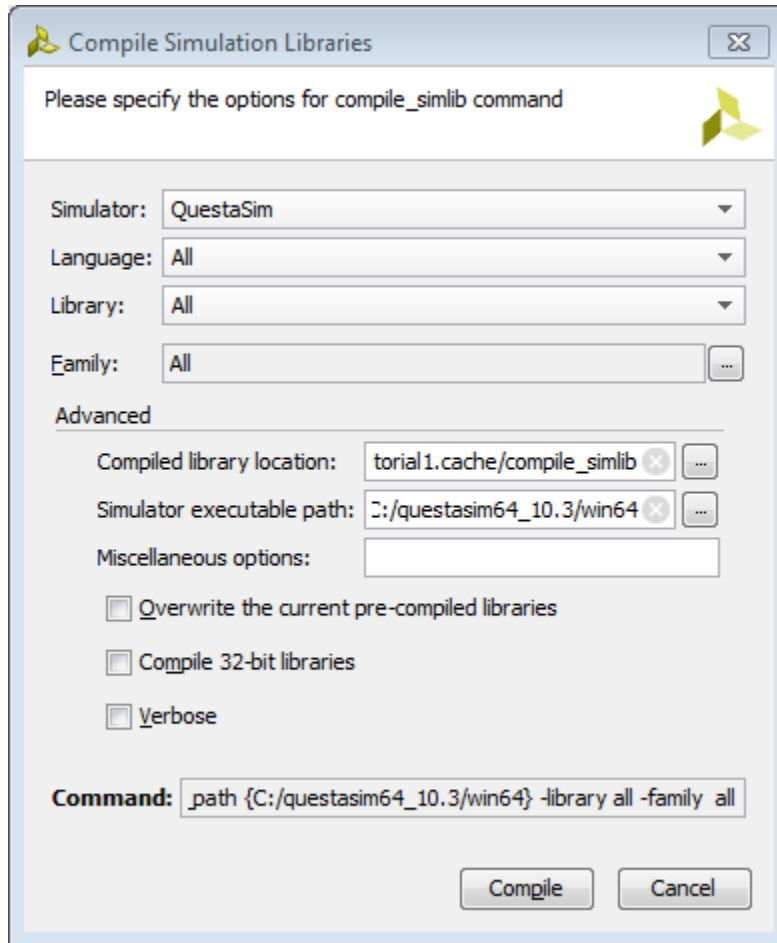


Figure 42: Compiling Simulation Libraries

Use the **Compiled library location** specified during the compiled library generation step and enter it in the **Compiled library location** field in the Project Settings dialog box. It should point to the `compile_simlib` directory.

- 6. In the **Simulation** tab, specify the **Design Under Test Instance** as `testbench/dut_fpga`.
- 7. Set the **SAIF Filename** to `power_tutorial_timing_questasim.saif`.
- 8. Note that the **Simulation Run Time** is 1000ns.

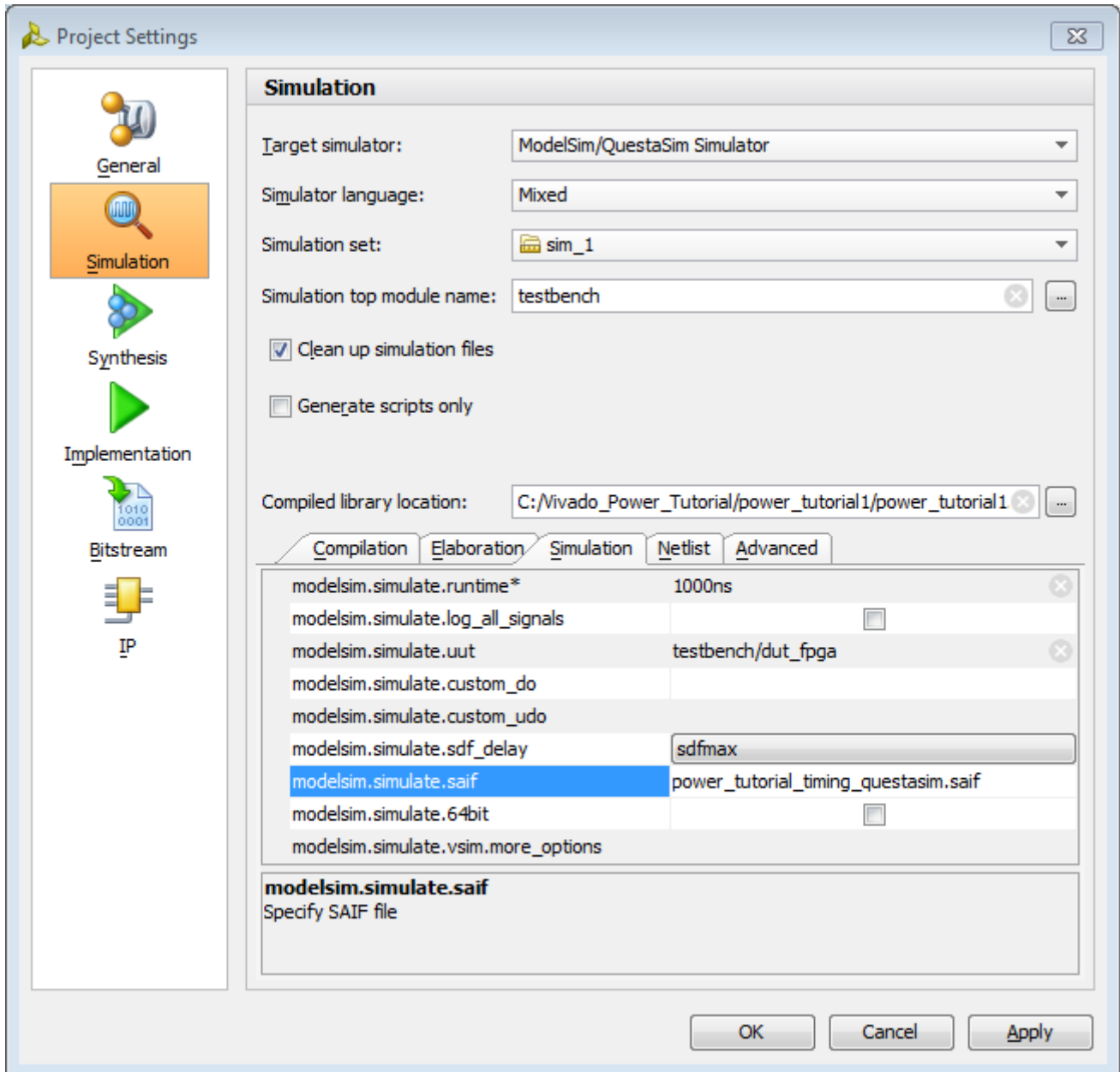


Figure 43: Simulation Settings – Simulation Tab

9. Click **OK**.

With the simulation settings properly configured, you can launch the QuestaSim simulator to perform a timing simulation of the design.

10. In the Flow Navigator, click **Run Simulation > Run Post-Implementation Timing Simulation**.

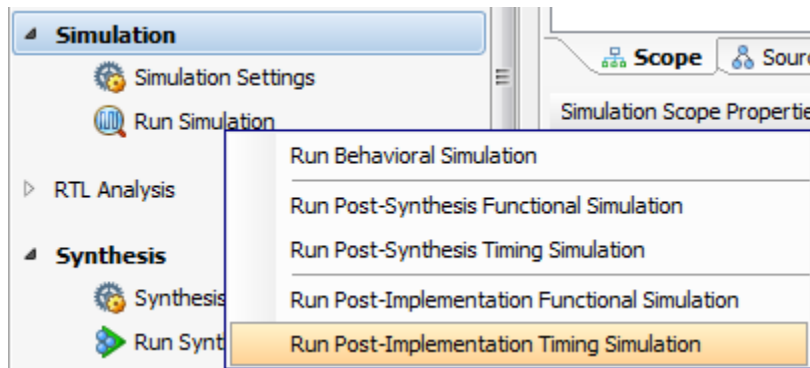


Figure 44: Running Post-Implementation Timing Simulation

A separate QuestaSim GUI opens and starts simulating the design.

11. After the QuestaSim simulator has finished simulating the design, make sure the SAIF file requested has been generated. Check to see that the SAIF file requested in the simulation settings prior to running simulation appears in this directory:

```
/ <project_directory>/power_tutorial1/power_tutorial1.sim/  
sim_1/impl/timing/power_tutorial_timing_questasim.saif
```

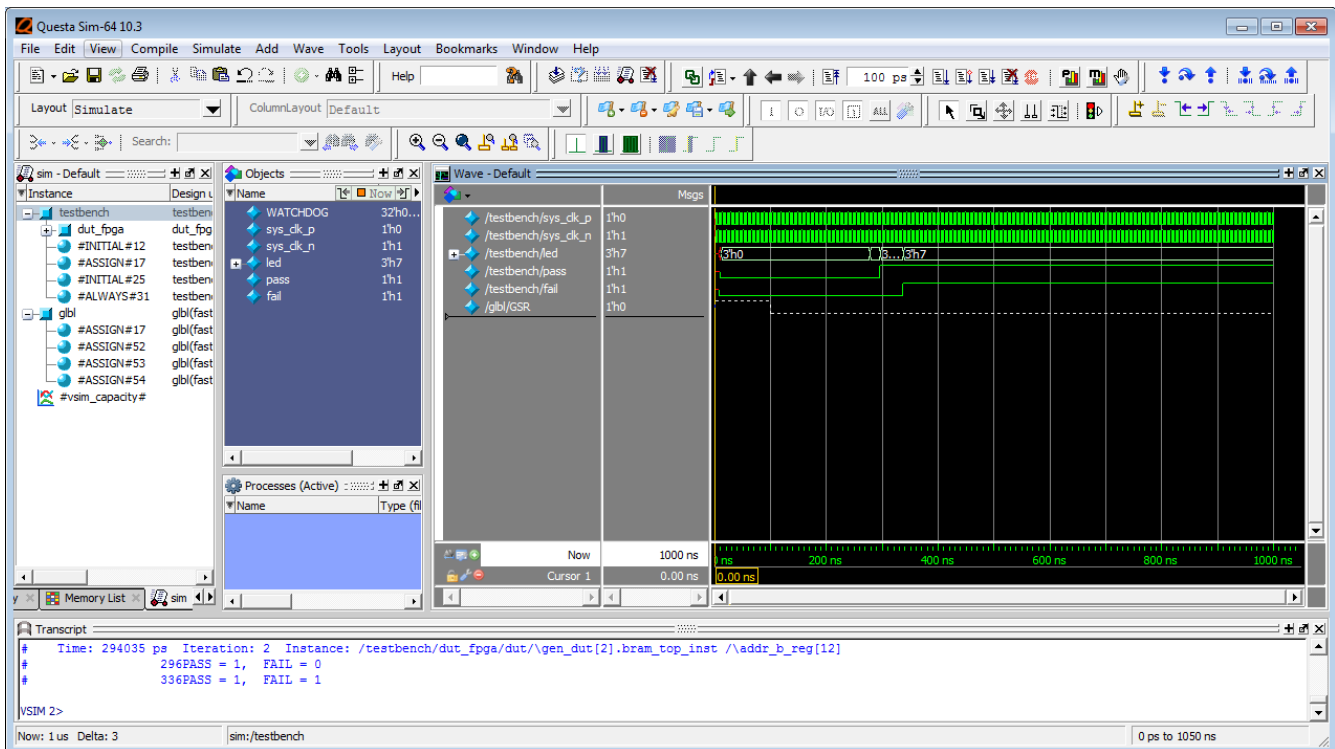


Figure 45: Running Post-Implementation Timing Simulation on QuestaSim

Step 2: Running Report Power in Vectorless Mode



IMPORTANT: If SAIF based report_power has already been run in this session, run the `reset_switching_activity -all` command in the Tcl Console. This will clear the SAIF data in the power engine from the earlier runs.

1. Close any open Report Power views.
2. In the Flow Navigator, select **Implemented Design > Report Power** to open the Report Power dialog box.

Alternatively, select **Tools > Report > Report Power** in the main menu bar.

3. In the Report Power dialog box, make the following settings:
 - o Specify the **Results name** as `power_1`
 - o In the **Environment** tab, Set the **Process** to **maximum**
 - o In the **Switching** tab, leave the **Simulation activity file** empty
4. Verify that all the input settings are correct and click **OK**.

The Report Power command creates a Power Report under the **power_1** tab in the results windows area. Note that the total power for Vectorless Analysis runs with default switching rates.

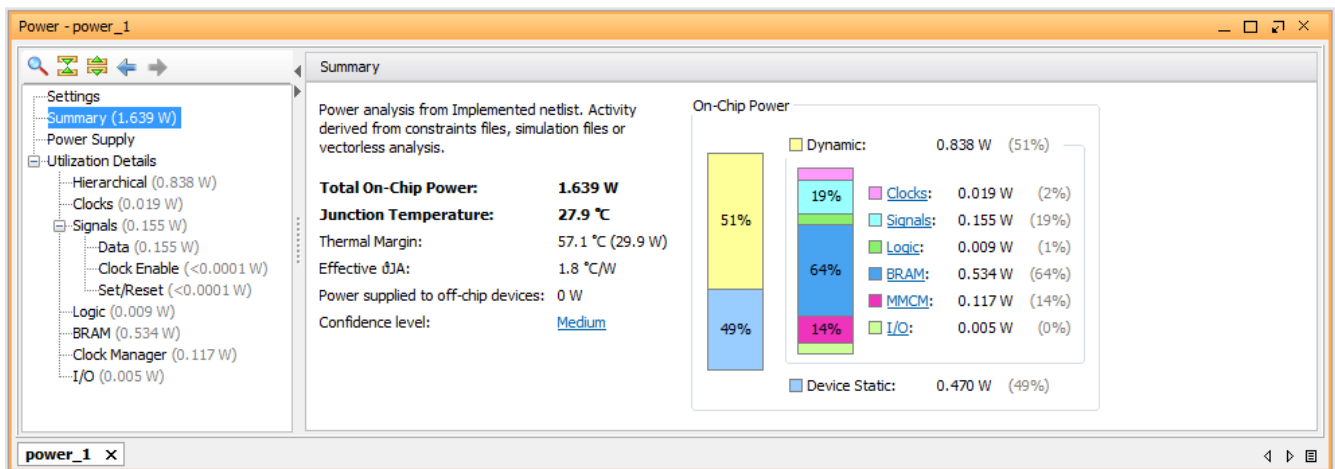


Figure 46: Power Report – Vectorless Mode Run

Step 3: Running Report Power with QuestaSim SAIF Data

The SAIF output file requested in the simulation run has been generated under the project directory. We use this SAIF file – a “Switching Activity Interchange Format” file – to further guide the power estimation algorithm.

1. In the main menu bar, select **Tools > Report > Report Power**.
2. In the Report Power dialog box, specify the SAIF file location in the **Switching** tab.

The SAIF file, which was requested in the simulation settings prior to running simulation, should appear here:

```
 /<project_directory>/power_tutorial1/ power_tutorial1.sim/  
 sim_1/impl/timing/power_tutorial_timing_questasim.saif
```

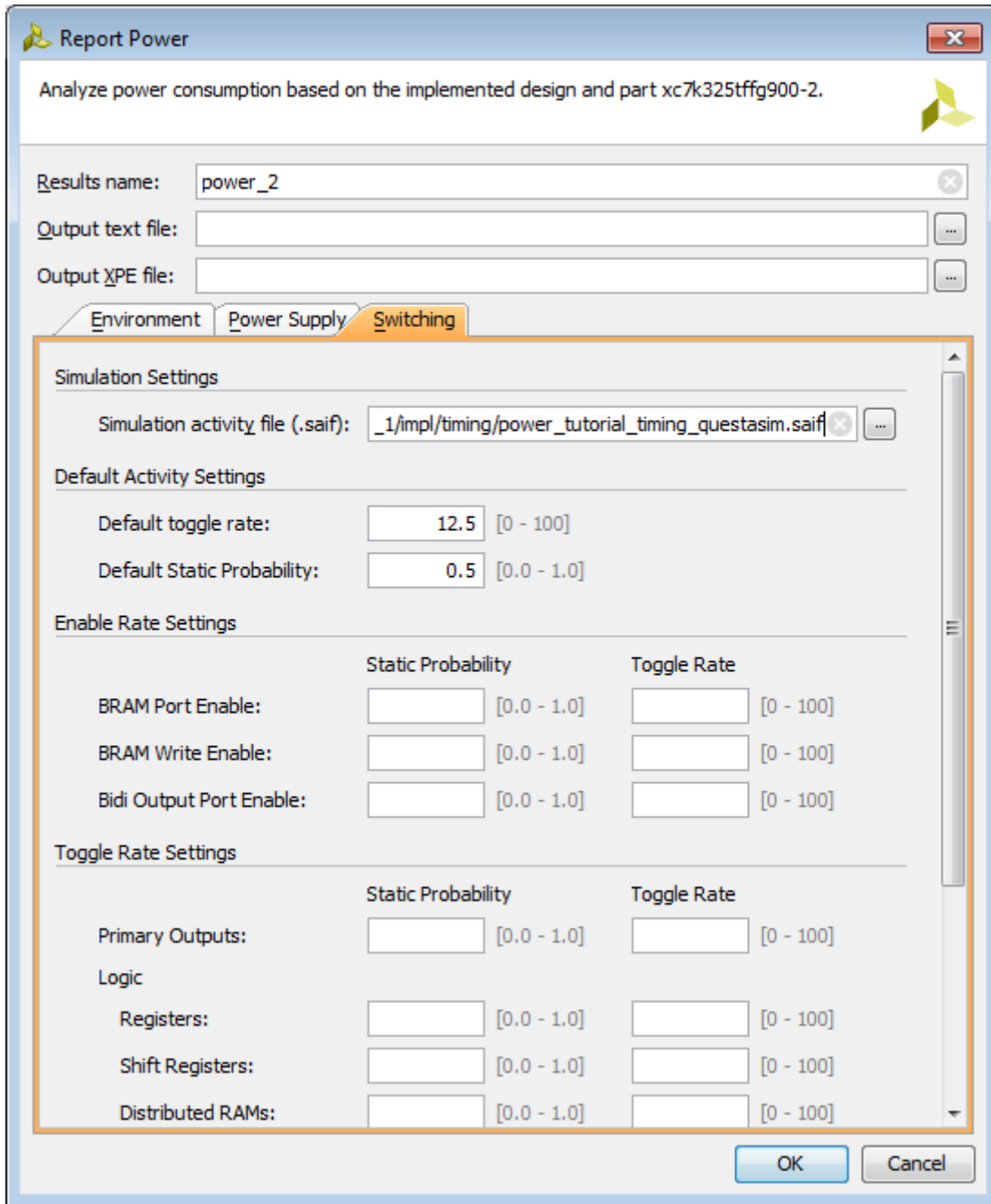


Figure 47: Report Power Dialog Box – Specifying SAIF File

3. Click **OK** in the Report Power dialog box.

The Report Power command runs, and the Power Report **power_2** is generated in the **Power** tab of the results windows area.

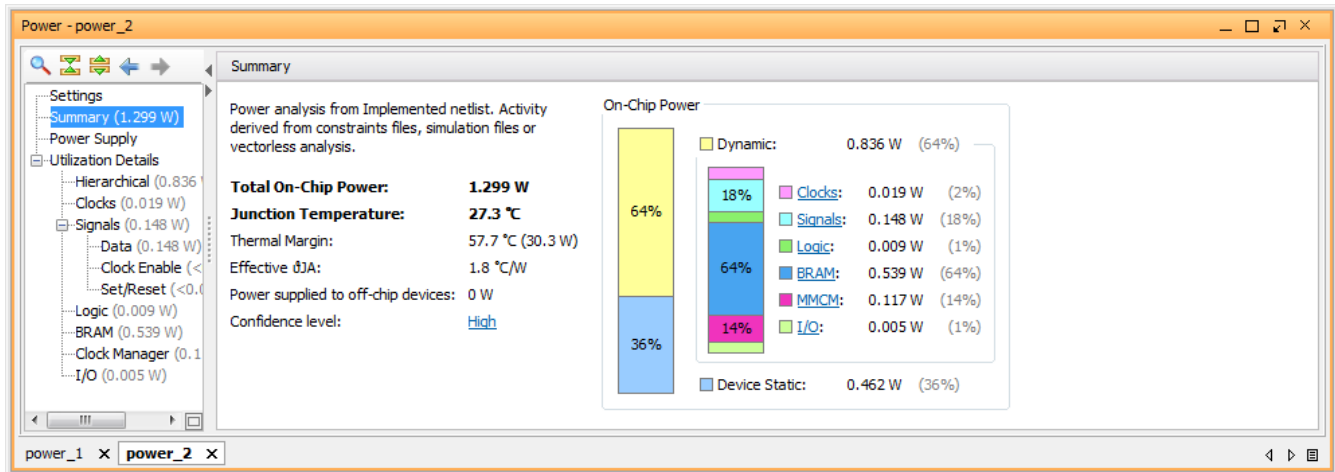


Figure 48: Power Report – With SAIF File Data

- In the Tcl console, observe the `read_saif` results. This shows the percentage of design nets matched with simulation SAIF. This is important for accurate power analysis.
- Go to the **Signals > Data** view in the Power Report and scroll to the right. Note that all the Signal Rate data has been set from simulation SAIF data you provided.

The data is color coded to indicate activity rates read from the **Simulation** output file.

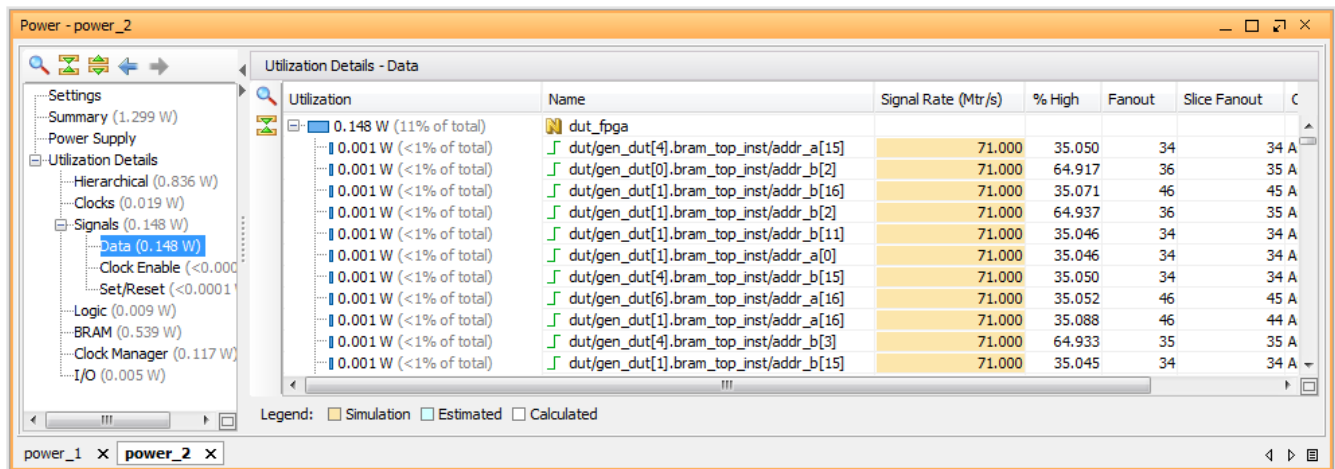


Figure 49: Signals > Data View with SAIF File Data

- Note the change in total power (**Total On-Chip Power** in the **Summary** view) in the **power_2** report compared to the **power_1** report. The total power estimated in the report generated with SAIF file data will be different than the total power estimated in the vectorless run (**power_1** results).

Lab 4: Hardware Power Measurement Using the KC705 Evaluation Board

Introduction

In this lab, you will learn about basic hardware power measurement technique and correlating the hardware power numbers with the numbers generated by Vivado Report Power. The lab will take you through the steps for setting up the hardware measurement, programming a bit file using Vivado Hardware Manager and power measurement through Texas Instruments (TI) Fusion Design Software. It also includes Junction Temperature reading from Vivado System Monitor.

Step 1: Generate a Bit File from the Implemented Design (Non-Power Optimization)

1. In the Vivado Design Suite, open the Implemented design from the previous Lab.
2. In the Flow Navigator, click **Generate Bitstream**.

The Bitstream Generation Completed dialog box appears after the bitstream has been generated.

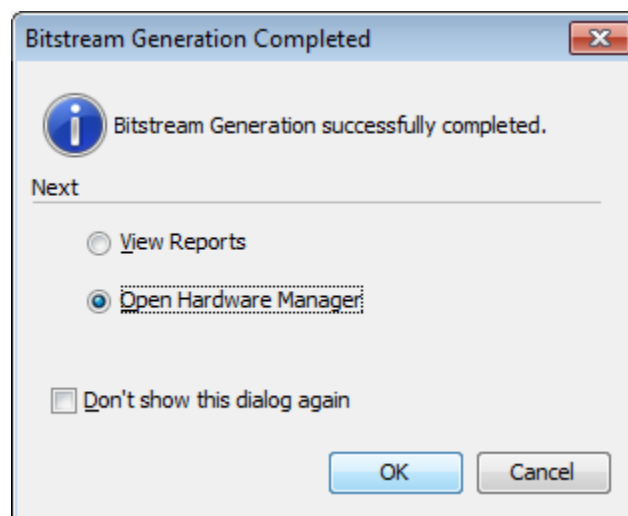


Figure 50: Bitstream Generation Completed Dialog Box

3. Open the Hardware Manager by selecting **Open Hardware Manager** in the Bitstream Generation Completed dialog box and clicking **OK**.

Step 2: Set Up the KC705 Evaluation Board



IMPORTANT: This project is created for the KC705 Rev 1.0 Evaluation Board. The pin constraints are set based on this Evaluation Board. If you are using any other Revisions, update the XDC file `dut_fpga_kc705.xdc` with the correct pin constraints.

1. Connect the Power Cable and Digilent Cable (or JTAG) for Programming.
2. Install the TI Fusion Digital Power Designer software on the PC from this location:
www.ti.com/tool/fusion_digital_power_designer
3. Connect the TI USB Interface Adapter to the PMBus port on the KC705 Evaluation Board.

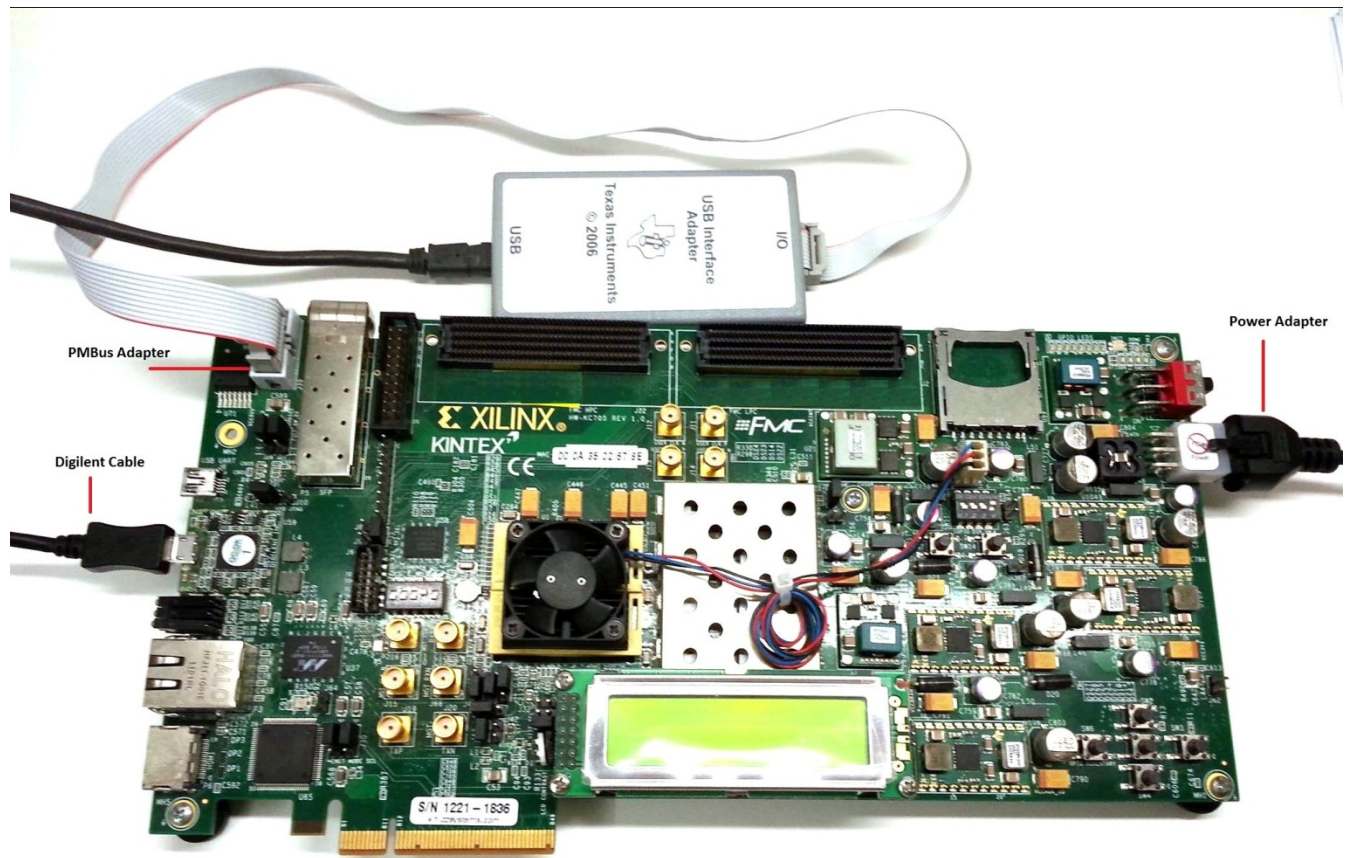


Figure 51: KC705 Hardware Setup

Step 3: Set Up the Fusion Digital Power Designer Software

1. Power ON the KC705 Evaluation Board.
2. Open the Fusion Digital Power Designer.

The software detects the USB adapter and brings up the GUI.

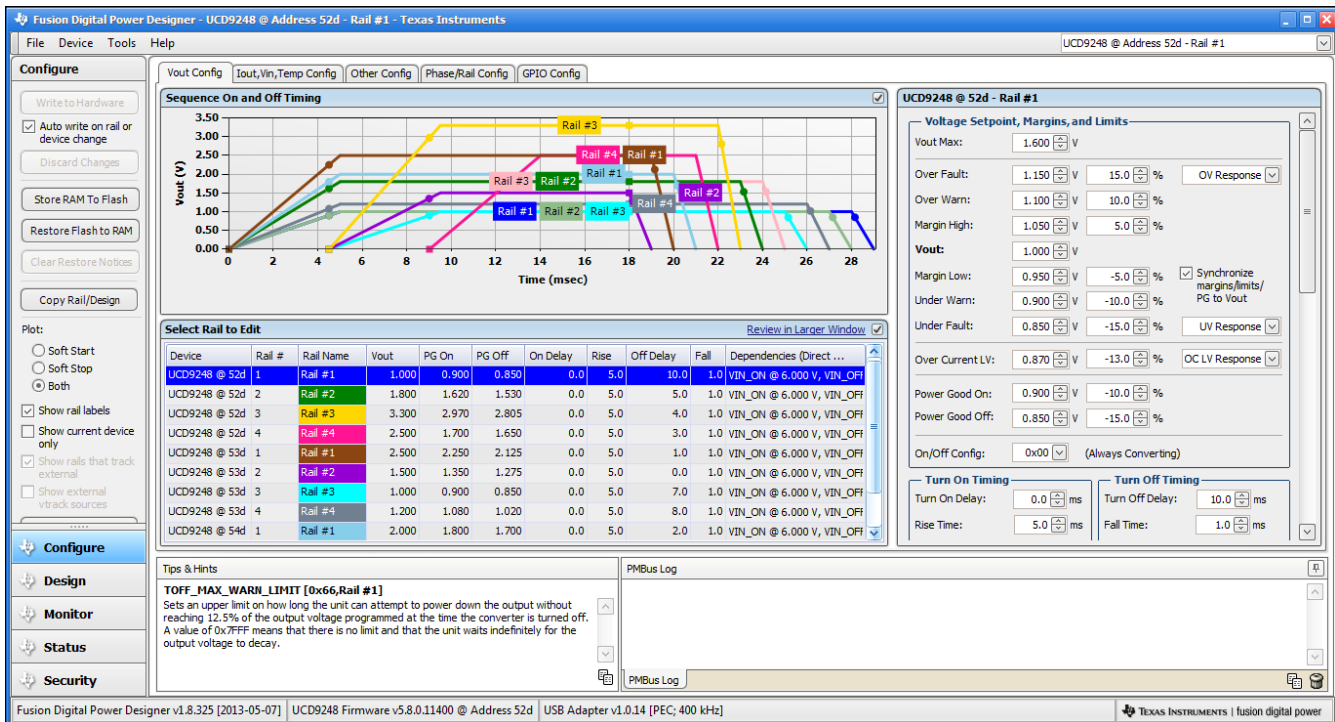


Figure 52: Fusion Digital Power Designer

Step 4: Program the Bitstream

1. Power up the KC705 Evaluation Board.
2. In the Vivado Hardware Manager, click **Open Target** in the green alert bar and select **Open New Target**.

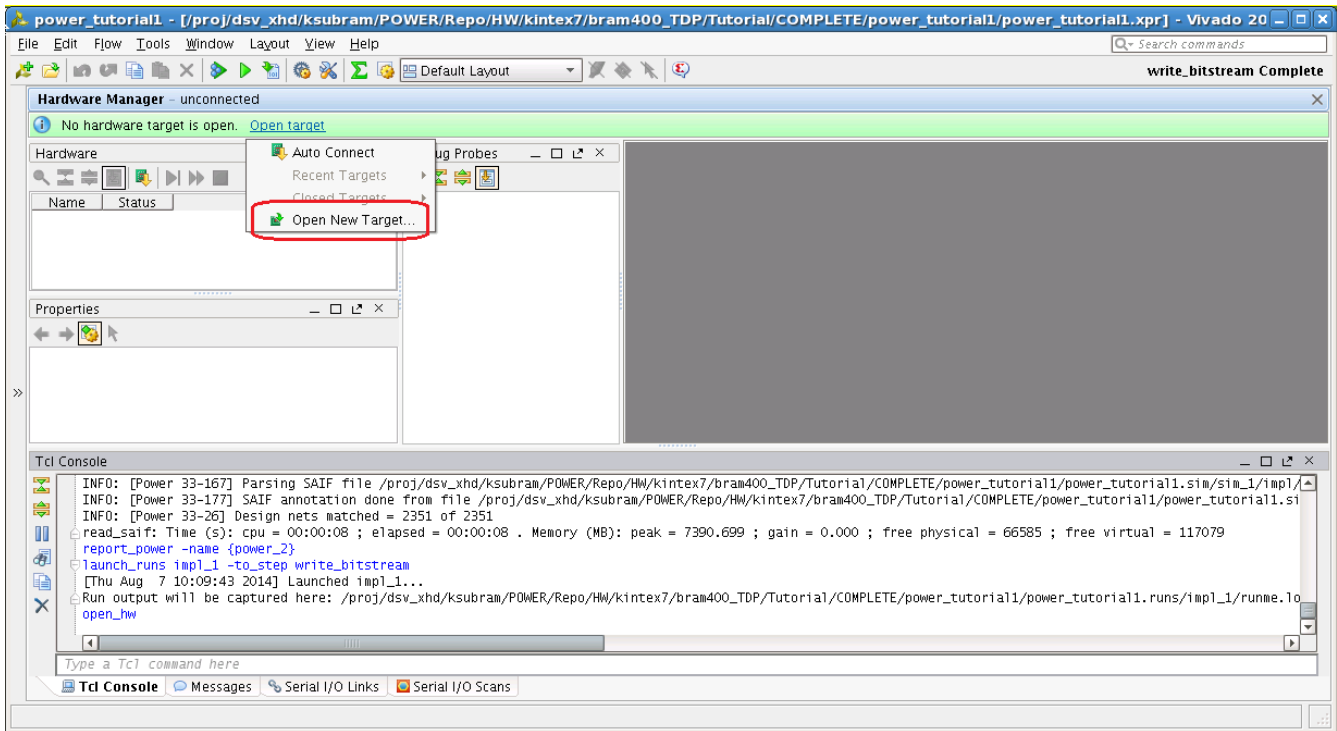


Figure 53: Opening the New Hardware Target

3. In the Open New Hardware Target dialog box, click **Next** to go to the **Hardware Server Settings**.
4. Select the server to which the board is connected.
 - o If the board is connected to the local PC, select **Local Server** and click **Next**.
 - o If you are connecting to a remote server, see Connecting to a Hardware Target Using `hw_server` in the *Vivado Design Suite User Guide: Programming and Debugging* ([UG908](#)).

When the hardware is detected successfully, the part information will be displayed in the Open New Hardware Target dialog box.

5. Verify the part information , then click **Next** and **Finish**.

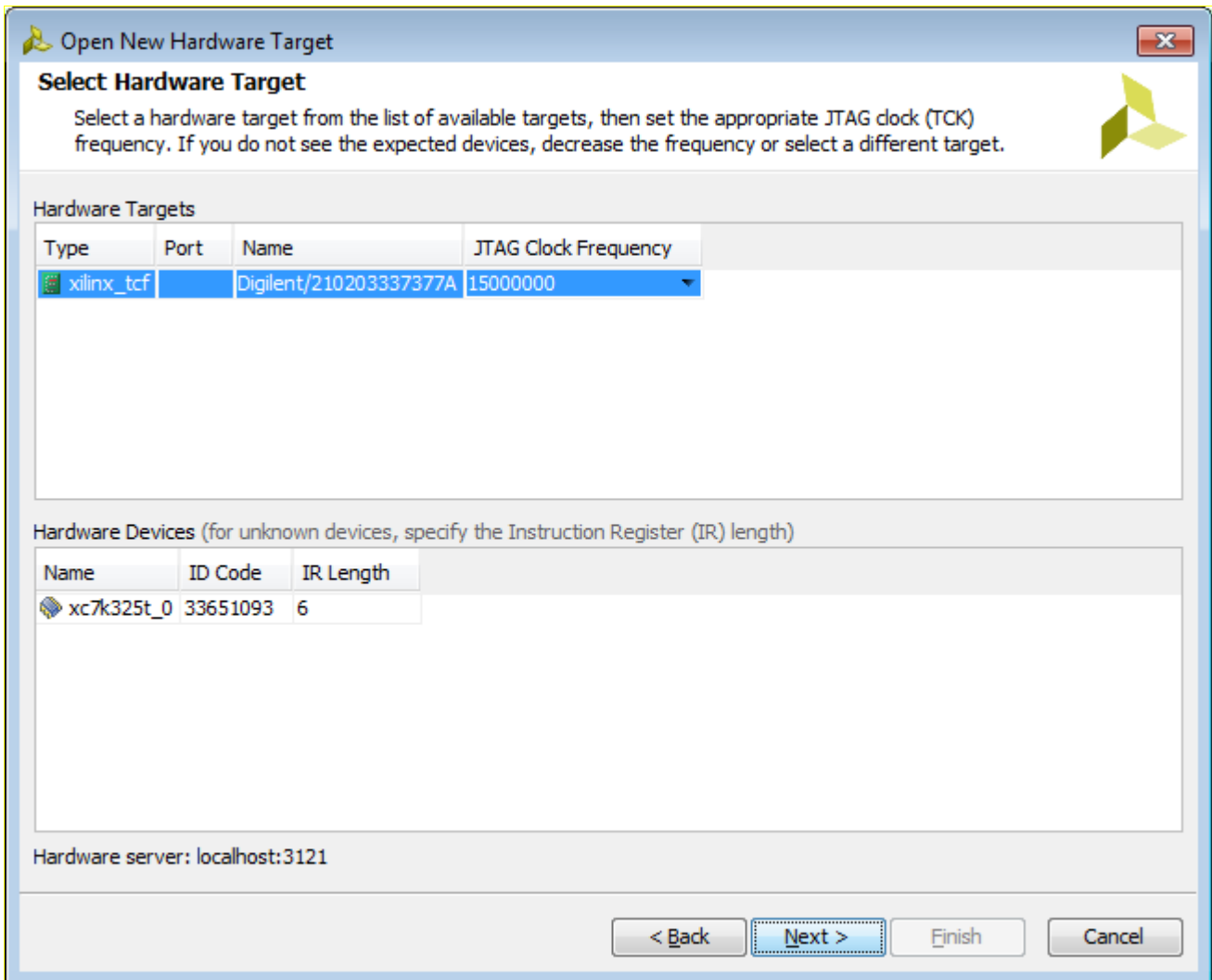


Figure 54: Hardware Target Detected

6. In the Hardware window, right-click the part and select **Program Device**.

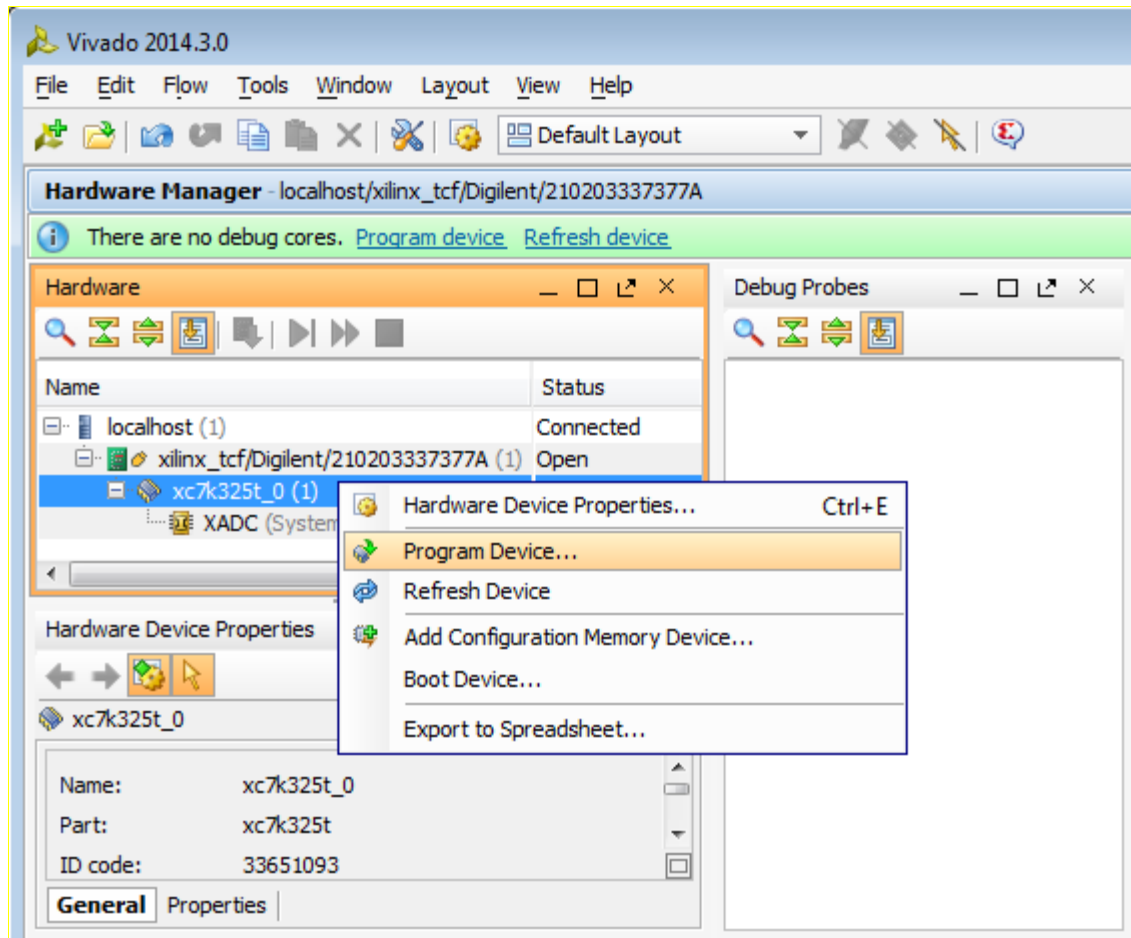


Figure 55: Program Device

7. Select the bit file `<project_dir>/power_tutorial1/power_tutorial1.runs/impl_1/dut_fpga.bit` and click **Program**.
8. Once the programming is done successfully, select **XADC (Sysmon)** in the Hardware window, then right-click and select **Open Hardware Dashboard**.

The System Monitor window opens and plots Die Temperature (Junction Temperature) in the Graph Window.

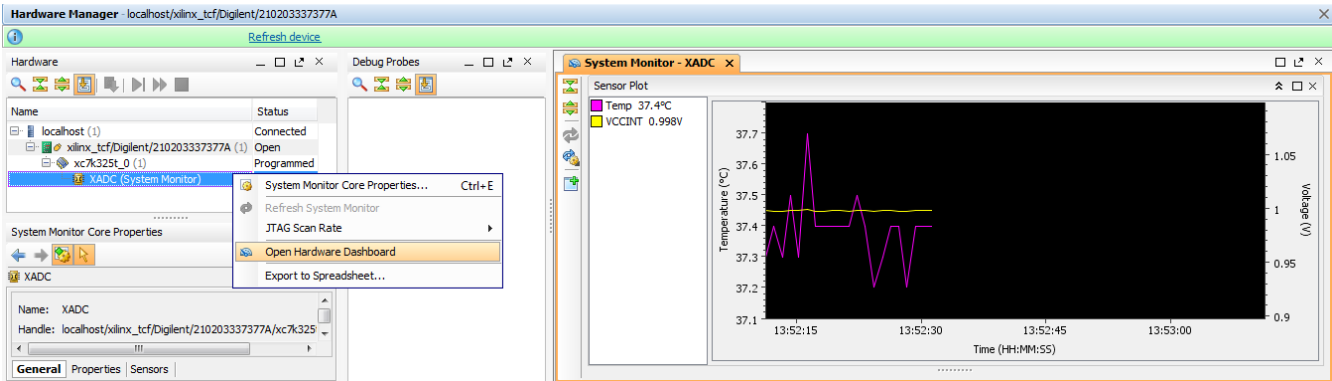


Figure 56: System Monitor – Junction Temperature Plot

Step 5: Measure the Hardware Power Rails

1. In Fusion Digital Power Designer, select a rail in the Configure view and click **Monitor**.

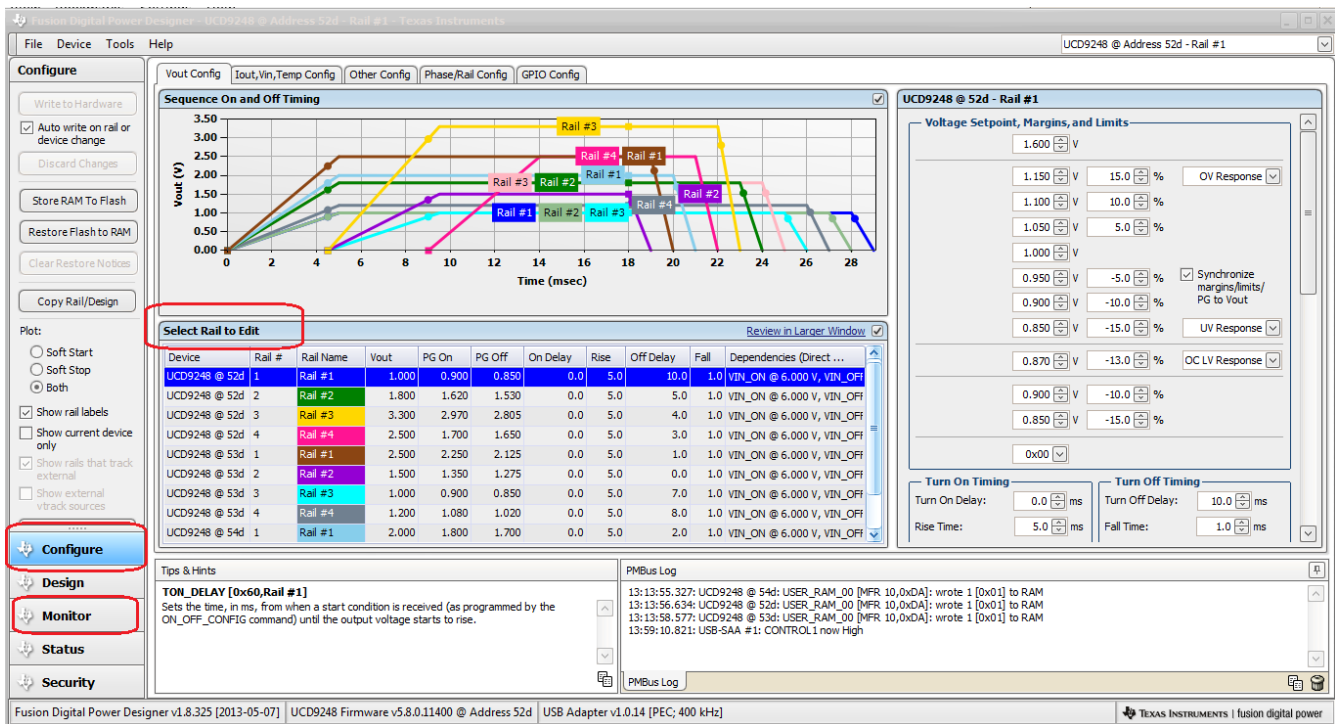


Figure 57: Select a Rail to Monitor

2. Configure the parameters to be monitored.

An Output Power graph will be plotted in the Monitor window.

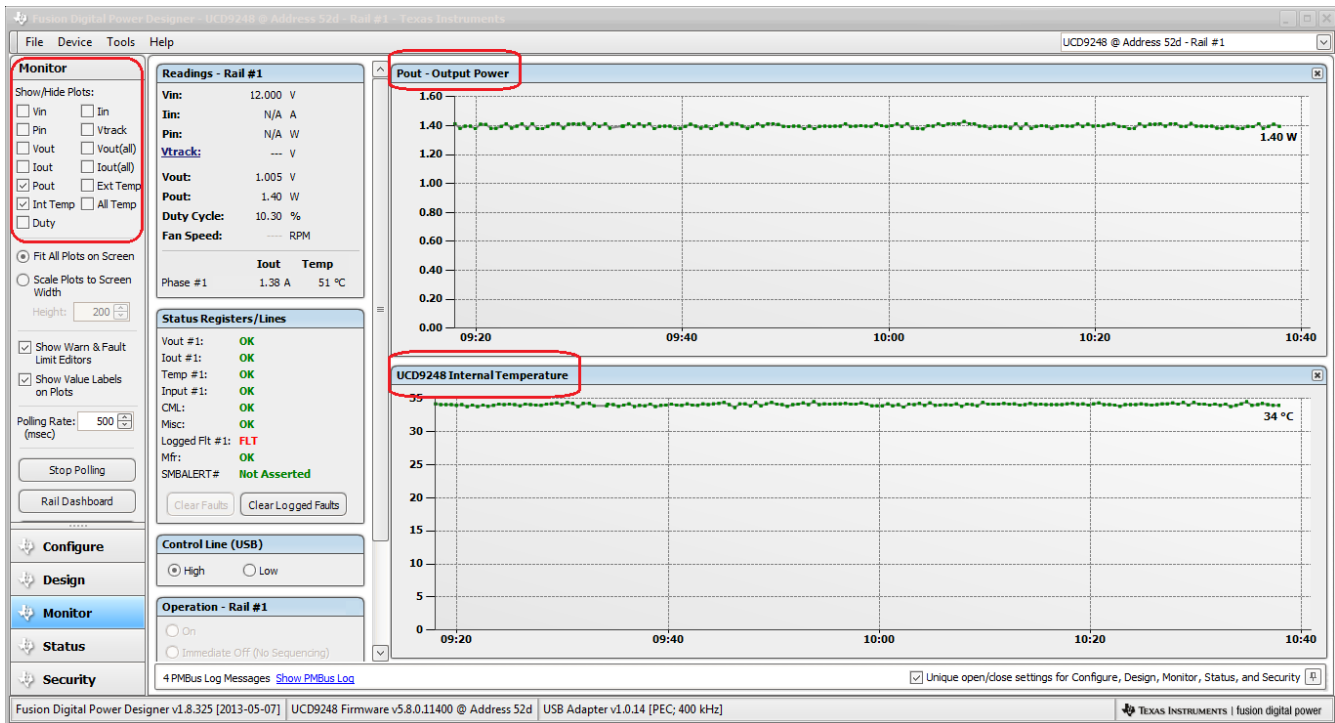


Figure 58: Monitor a Rail – Power and Junction Temperature

- Repeat the steps above to monitor the power information for each rail supplied to the device.
Note that rail information is displayed in terms of Regulator address.

		KC705 (UCD9248)			
device	rail	purpose	voltage	IOUT_CAL_GAIN	IOUT_CAL_OFFSET
52	1	VCCINT	1.0V	0xEBDC	0x8000
	2	VCCAUX	1.8V	0xEBDC	0x8000
	3	VCC3V3	3.3V	0xEBDC	0x8000
	4	VADJ	2.5V	0xEBDC	0x8000
53	1	VCC2V5	2.5V	0xEBDC	0x8000
	2	VCC1V5	1.5V	0xEBDC	0x8000
	3	MGT_AVCC	1.0V	0xEBDC	0x8000
	4	MGT_AVTT	1.2V	0xEBDC	0x8000
54	1	VCCAUX_IO	1.8V	0xEBDC	0x8000
	2	VCCBRAM	1.0V	0xEBDC	0x8000
	3	MGT_VCCAUX	1.8V	0xEBDC	0x8000
	4	N/A	N/A	N/A	N/A

Figure 59: KC705 Rail Address Mapping

4. Note down the Junction Temperature value either from the Vivado Hardware Manager or from Fusion Digital Power Designer.

Step 6: Vectorless Power Estimation with Junction Temperature

For further Power Analysis, you can use the measured Junction Temperature and other thermal settings to feed into Vivado Report Power for better accuracy.

1. In the Vivado Design Suite, open the tutorial project and click **Open Implemented Design** to display the implemented design.
2. In the Tcl Console, run the following command to reset any user defined or SAIF file defined settings.

```
reset_switching_activity -all
```

3. In the main menu bar, select **Tools > Report > Report Power**.
4. In the Report Power dialog box **Environment** tab, enter the **Junction Temperature** value supplied by the hardware power measurement.
5. Set the **Process** to **maximum**.
6. In the **Switching** tab, make sure the **Simulation activity file (saif)** is blank.
7. Click **OK**.

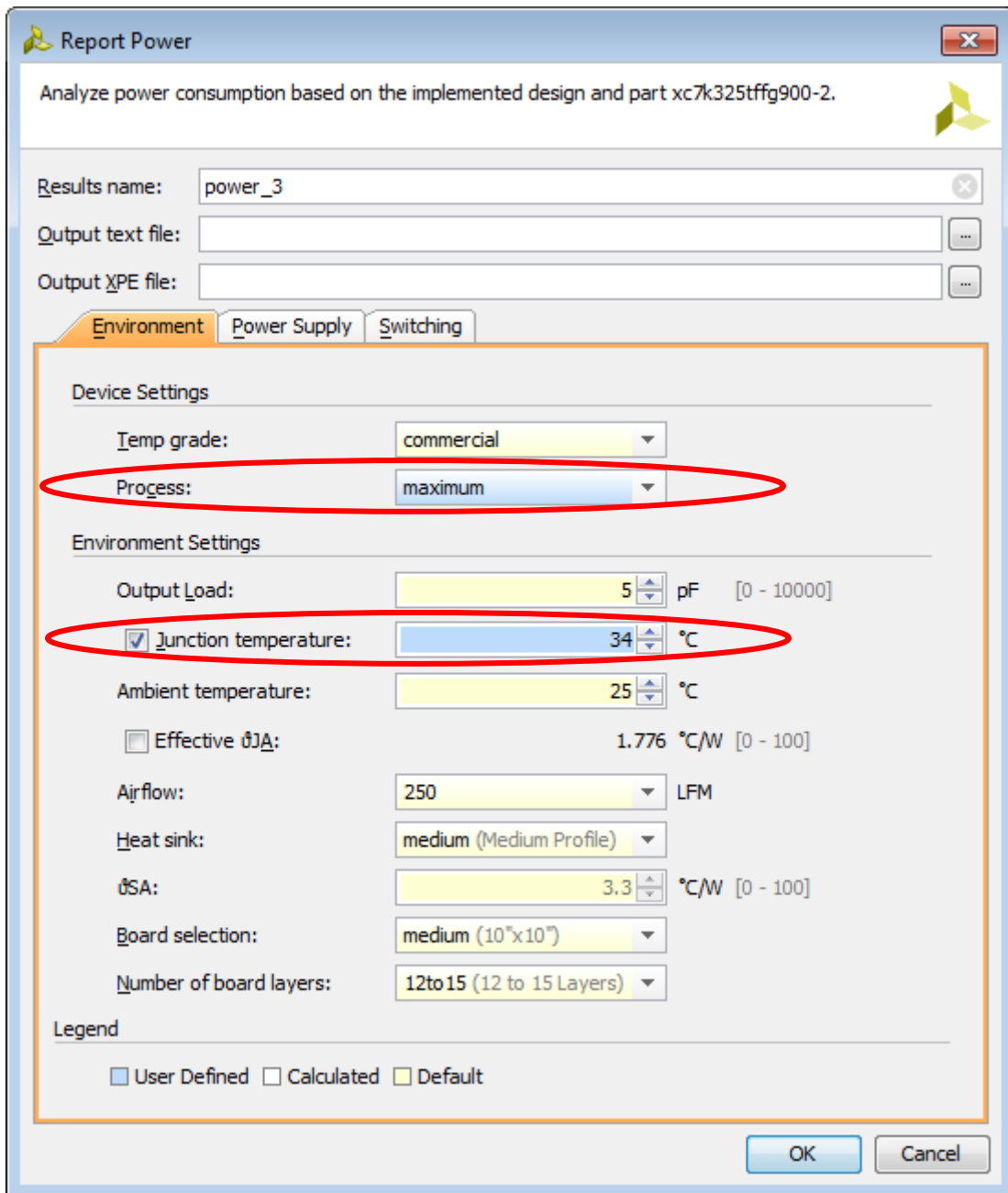


Figure 60: Report Power Settings – Updating Junction Temperature

- In the Power Report, observe that the power numbers increase slightly as compared to the vectorless power analysis using a default **Junction Temperature** value.

Note that the **Junction Temperature** is now color coded as being user defined in the Power Report.

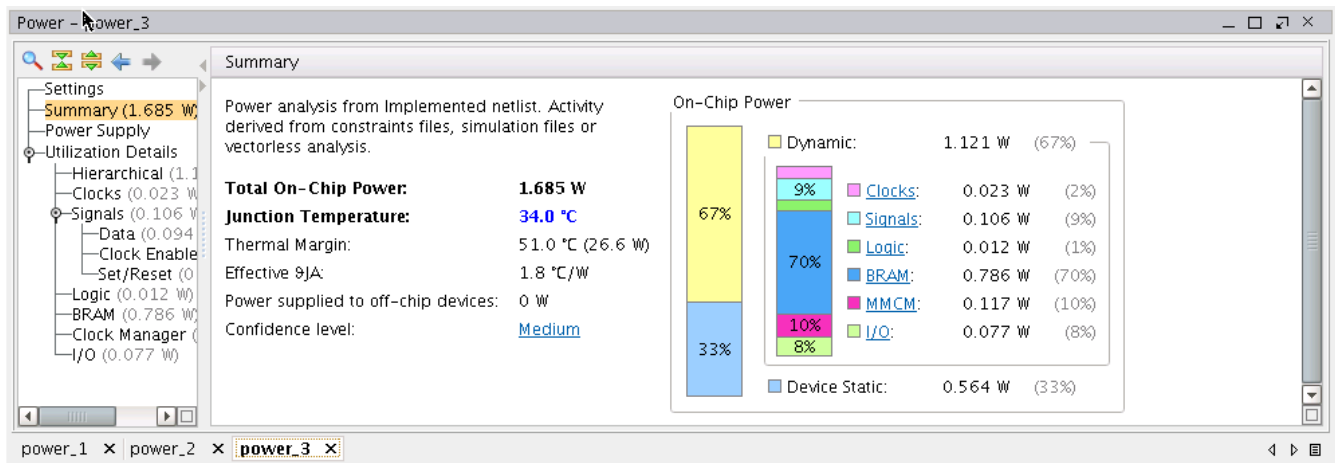


Figure 61: Power Report – Vectorless Analysis with Junction Temperature Supplied

9. Similarly, you can overwrite the Junction Temperature setting and do a SAIF based power analysis.

With this lab, you have completed a Vivado Report Power analysis on post-synthesis and post-implementation netlist designs without Power Optimization. You also experimented with hardware power measurement and with reading Junction Temperature for software analysis.

Note down the power numbers measured and estimated on non-power Optimized design.

Lab 5: Power Optimization in the Vivado Tools

Introduction

In this lab, you will learn about using the Power Optimization features in Vivado. The lab will take you through the steps for invoking Power Optimization after synthesizing the design. It will also guide on using the power optimization report, making decisions and selectively turning off power optimization on signals, blocks, and hierarchies.



TIP: When you run Implementation on your design, the Vivado tools may perform BRAM power optimizations by default during `opt_design`. These optimizations will not affect performance, and will have little impact on area and runtime.

In the previous Lab, the default BRAM power optimization was disabled by setting a `NoBramPowerOpt` directive to `opt_design`.

Step 1: Set Up Options to Run Power Optimization

1. In the Flow Navigator, click **Implementation Settings**.
2. In the Project Settings dialog box, make these settings:
 - o In the **Opt Design** settings, set the `-directive` option to **Default**.
BRAM optimization runs in the **Default** setting for **Opt Design** during Implementation. BRAM optimization was disabled in the previous lab. It is now re-enabled when the design runs Power Optimization.
 - o In the **Power Opt Design** settings, check the **is_enabled** box.
This ensures Power Optimization runs after `opt_design`. Enabling the **Power Opt Design** option prior to `place_design` results in a complete power optimization to be performed. This option yields the best possible power saving from the Vivado tools.

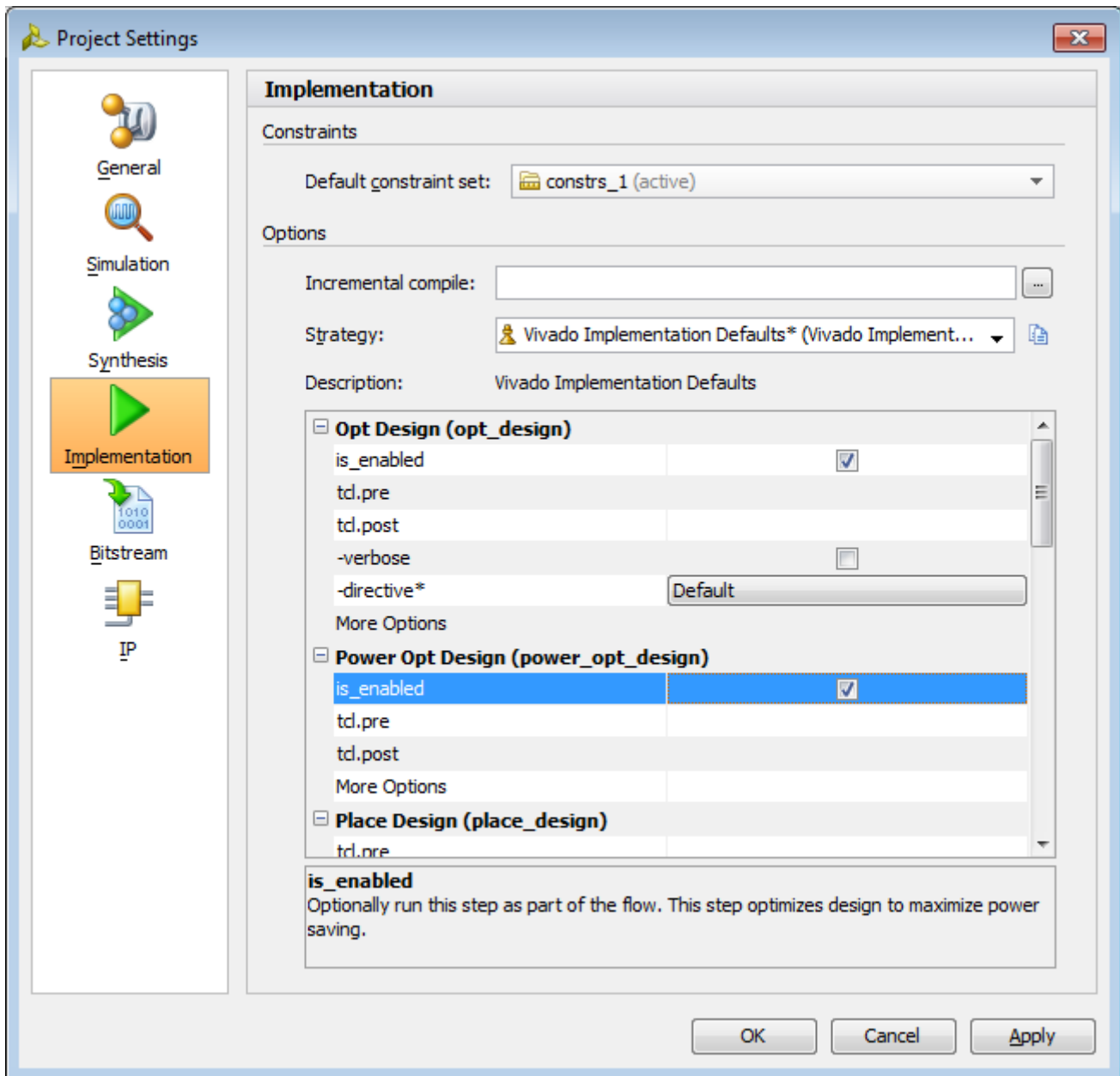


Figure 62: Implementation Settings – Enabling Power Optimization

3. Click **OK**.
4. In the Create New Run dialog box, click **Yes** to “Properties for the completed run ‘impl_1’ have been modified. Do you want to preserve the state of ‘impl_1’ and apply these changes to a new run?”.

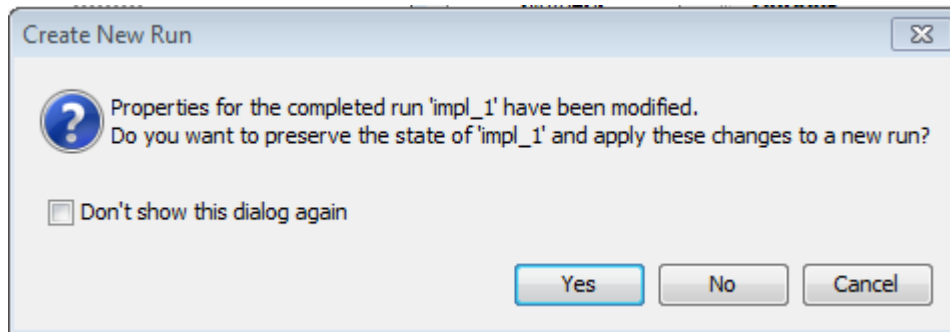


Figure 63: Create New Run Dialog Box

5. In the Create Run dialog box, set the **Run Name** to `impl_2`.
6. Click **OK**.
7. In the Flow Navigator, select **Run Implementation**.

Click **Don't Save** when the Save Project window pops up to save both Synthesis and Implementation constraints.

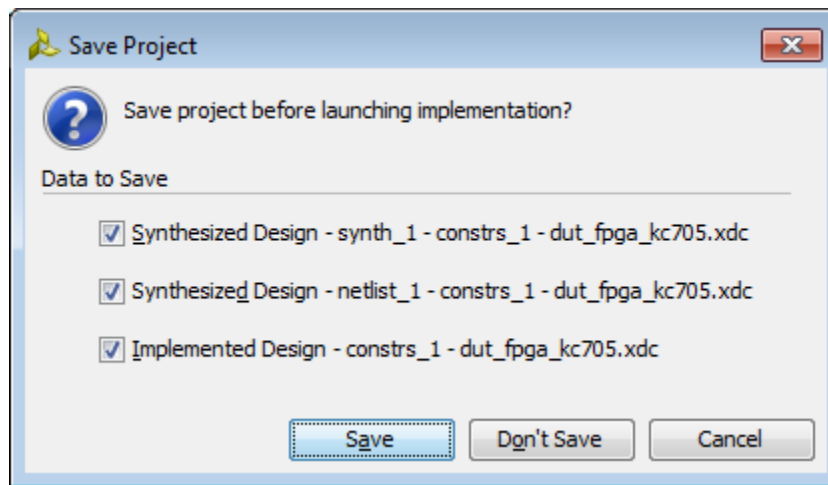


Figure 64: Click "Don't Save" Here

You are running Implementation with Power Optimization turned on.

8. In the Implementation Completed dialog box, select **Open Implemented Design** and click **OK**.
Click **Don't Save** when the Save Project window pops up to save both Synthesis and Implementation constraints.

Step 2: Run `report_power_opt` to Examine User/Design Specific Power Optimizations

1. In the Flow Navigator, select **Implemented Design**.
2. In the main menu bar, select **Tools > Report > Report Power Optimization**.

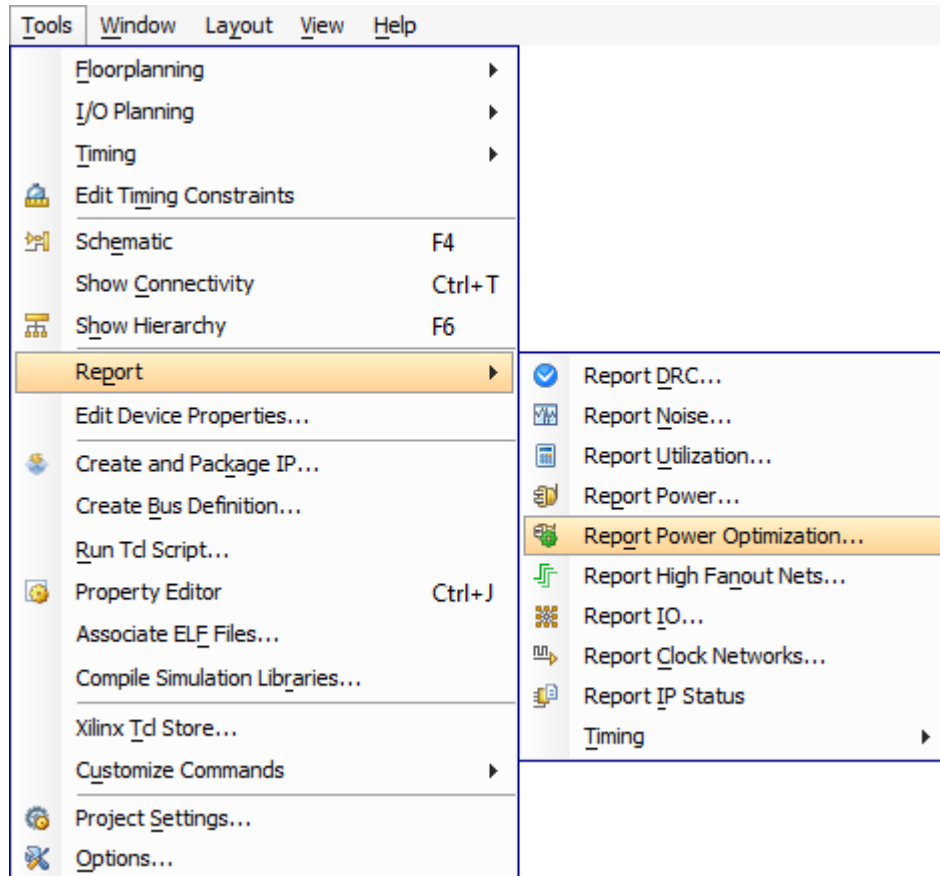


Figure 65: Generating a Power Optimization Report

3. The Report Power Optimization dialog box appears, as shown below.

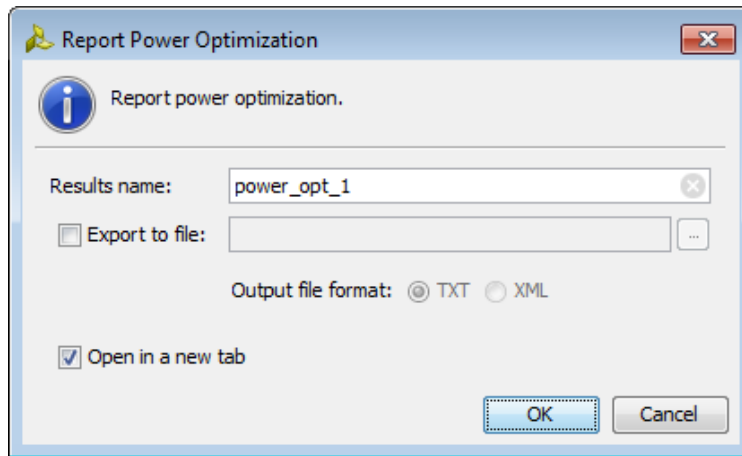


Figure 66: Report Power Optimization Dialog Box

4. Enter `power_opt_1` for the **Results name**.
5. Ensure that the **Open in a new tab** option is checked.
6. Click **OK**.

Alternatively, in the Tcl Console execute this Tcl command:

```
report_power_opt -name power_opt_1
```

7. Observe the report **power_opt1** is generated in the **Power Opt** tab of the results windows area. When the report opens, the **Summary** view is displayed in the report.
8. In the **Summary** view, note that 50% of the BRAMs are clock gated by the tool during Power Optimization.

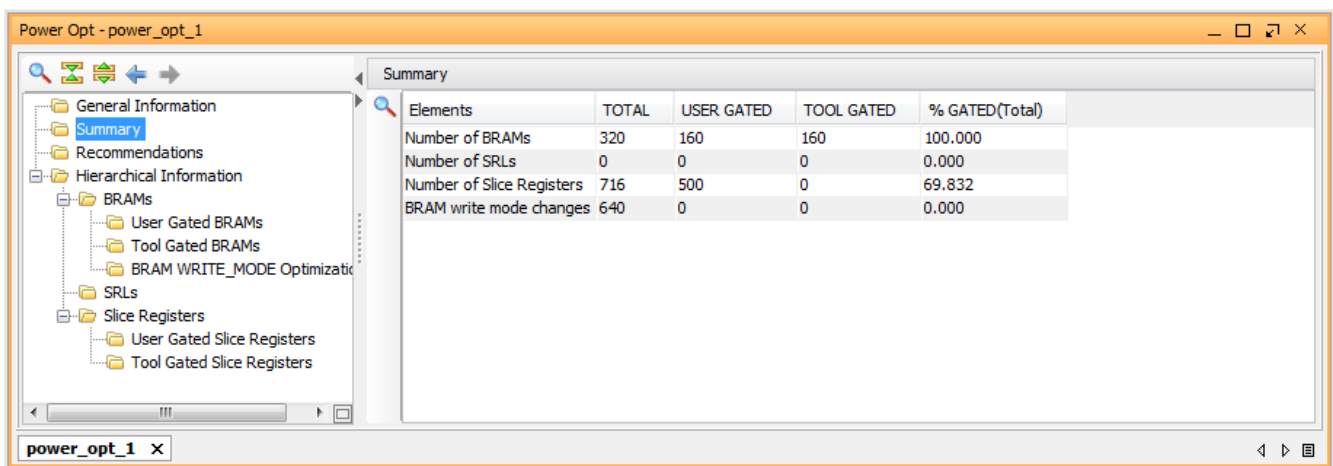


Figure 67: Power Optimization Report – Tool Gated BRAMs

- In the Power Report, select **Hierarchical Information > BRAMs > Tool Gated BRAMs** and observe the detailed information on tool gated BRAM enables.

Step 3: Run report_power to Examine Power Savings

- In the main menu bar, select **Tools > Report > Report Power**.
- In the Report Power dialog box, make the following settings:
 - Specify the **Results name** as `power_1`
 - In the **Environment** tab, make sure the **Process** is set to **maximum**.
- Click **OK**.

Alternatively, in the Tcl Console execute this Tcl command:

```
report_power -name power_1
```

- In the **Summary** view of the Power Report, observe an approximately 100-200mW power savings compared to the non-optimized power run in the previous lab.

You can generate a bitstream to program the hardware and measure its power, to observe the power saving in hardware. See [Lab 4: Hardware Power Measurement Using the KC705 Evaluation Board](#) for hardware power measurement instructions.

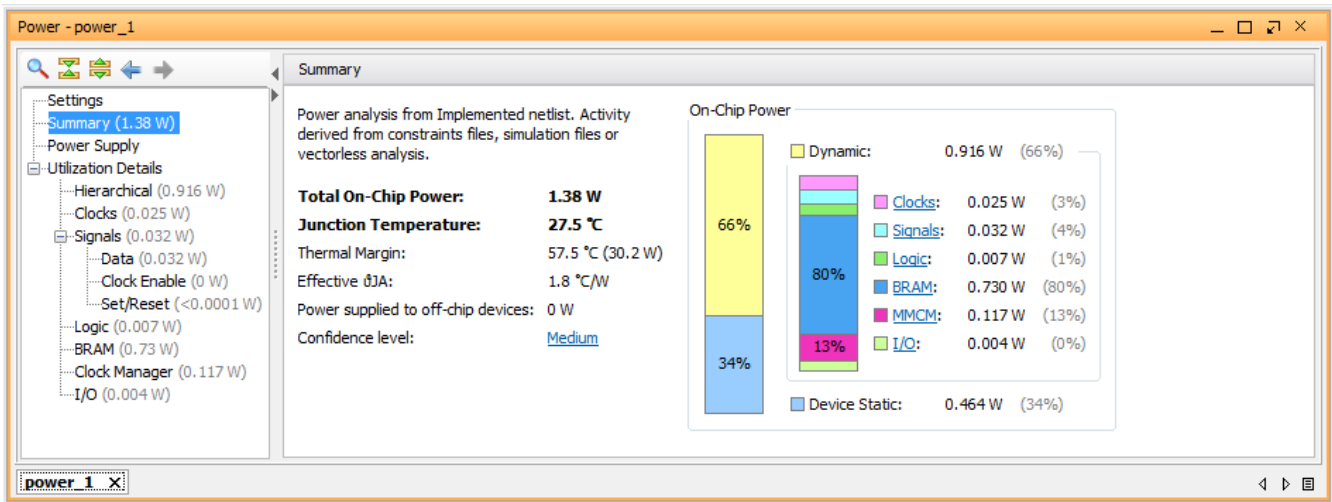


Figure 68: Power Savings with Power Optimization

Step 4: Turn Off Optimizations on Specific Signals and Rerun Implementation

In this step you will learn how to turn off the power optimization on specific BRAMs.



IMPORTANT: Power optimization works to minimize the impact on timing while maximizing power savings. However, in certain cases, if timing degrades after power optimization, you can identify and apply power optimizations only on non-timing critical clock domains or modules using the `set_power_opt` XDC command.

Refer to the *Vivado Design Suite User Guide: Power Analysis and Optimization* ([UG907](#)) for more information on the `set_power_opt` command.

Let us assume that this BRAM is in the critical path:

```
dut/gen_dut[0].bram_top_inst/bram_inst/mem_reg_0_0
```

This step makes sure the tool does not gate this BRAM.

1. In the Tcl Console, type this command:

```
set_power_opt -exclude_cells [get_cells  
dut/gen_dut[0].bram_top_inst/bram_inst/mem_reg_0_0]
```

This will prevent the tool from gating this BRAM.

2. From the Flow Navigator choose **Run Implementation**, which in turn reruns `power_opt_design`.
3. Click **Save** in the Save Project dialog box to save the Synthesized Design and Implemented Design constraints before launching implementation.

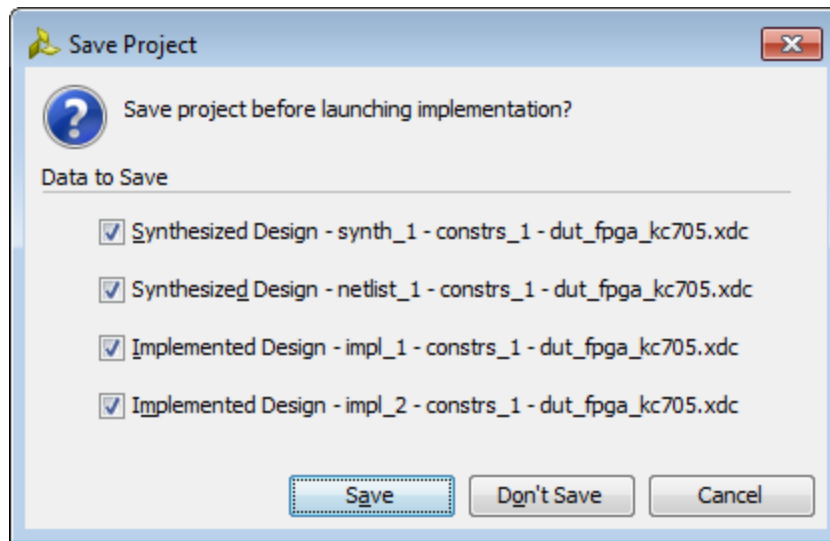


Figure 69: Save Design Constraints Before Rerunning Implementation

Also, select **Implemented Design – impl_2** in the Save Constraints Conflict dialog box to save the changes in constraints from the `set_power_opt` command.

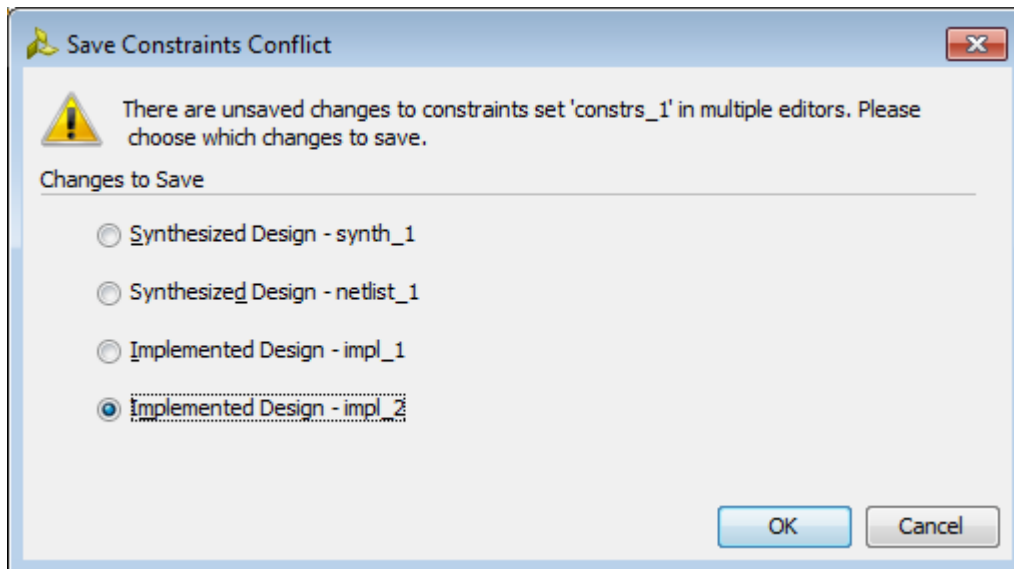


Figure 70: Saving Design Constraints for impl_2

4. In the Implementation Completed dialog box, select **Open Implemented Design** and click **OK**.

Step 5: Run `report_power_opt` to Examine Tool Optimizations Again

1. In the main menu bar, select **Tools > Report > Report Power Optimization**.
2. In the Report Power Optimization dialog box, type in the **Results name** as `power_opt_2`.

Alternatively, execute this Tcl command in the Tcl Console:

```
report_power_opt -name power_opt_2
```

3. In the generated report **power_opt_2** in the Power Opt window, display **Tool Gated BRAMs**.

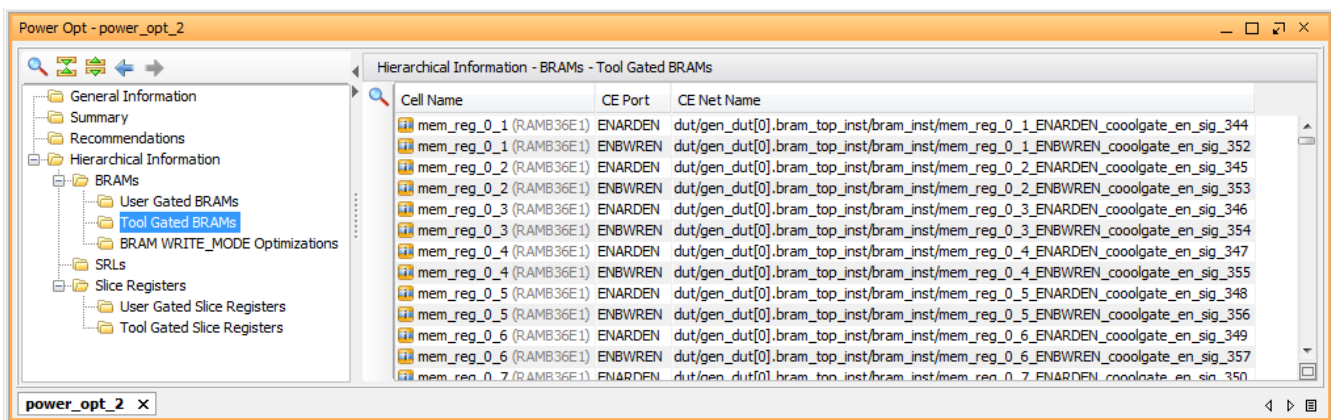


Figure 71: Displaying Tool Gated BRAMs

4. Note that this BRAM is no longer in the list of **Tool Gated BRAMs**:

```
dut/gen_dut[0].bram_top_inst/bram_inst/mem_reg_0_0
```

Conclusion

In this tutorial, you accomplished the following:

- Used the Report Power dialog box to verify and set device, thermal, and environmental conditions that contribute to power estimation.
- Synthesized the design and estimated the power after synthesis.
- Set switching activities on an I/O port and reran Report Power.
- Ran functional simulation using the Vivado simulator and generated a SAIF file that is input to Report Power for a more accurate power analysis.
- Implemented the design, ran post-implementation timing simulation using the Vivado simulator, and generated a SAIF file that is input to report power for a more accurate power analysis.
- Ran QuestaSim post-implementation timing simulation and generated a SAIF file that is input to report power for a more accurate power analysis.
- Performed power measurement on the design implemented in a KC705 Evaluation Board. Compared the hardware power numbers with the numbers generated by Vivado Report Power.
- Learned how to invoke power optimization as part of an implementation run.
- Examined the power optimization report and selectively turned off power optimizations on a cell in the design.

Please Read: Important Legal Notices

The information disclosed to you hereunder (the "Materials") is provided solely for the selection and use of Xilinx products. To the maximum extent permitted by applicable law: (1) Materials are made available "AS IS" and with all faults, Xilinx hereby DISCLAIMS ALL WARRANTIES AND CONDITIONS, EXPRESS, IMPLIED, OR STATUTORY, INCLUDING BUT NOT LIMITED TO WARRANTIES OF MERCHANTABILITY, NON-INFRINGEMENT, OR FITNESS FOR ANY PARTICULAR PURPOSE; and (2) Xilinx shall not be liable (whether in contract or tort, including negligence, or under any other theory of liability) for any loss or damage of any kind or nature related to, arising under, or in connection with, the Materials (including your use of the Materials), including for any direct, indirect, special, incidental, or consequential loss or damage (including loss of data, profits, goodwill, or any type of loss or damage suffered as a result of any action brought by a third party) even if such damage or loss was reasonably foreseeable or Xilinx had been advised of the possibility of the same. Xilinx assumes no obligation to correct any errors contained in the Materials or to notify you of updates to the Materials or to product specifications. You may not reproduce, modify, distribute, or publicly display the Materials without prior written consent. Certain products are subject to the terms and conditions of Xilinx's limited warranty, please refer to Xilinx's Terms of Sale which can be viewed at <http://www.xilinx.com/legal.htm#tos>; IP cores may be subject to warranty and support terms contained in a license issued to you by Xilinx. Xilinx products are not designed or intended to be fail-safe or for use in any application requiring fail-safe performance; you assume sole risk and liability for use of Xilinx products in such critical applications, please refer to Xilinx's Terms of Sale which can be viewed at <http://www.xilinx.com/legal.htm#tos>.

© Copyright 2013-2014 Xilinx, Inc. Xilinx, the Xilinx logo, Artix, ISE, Kintex, Spartan, Virtex, Zynq, and other designated brands included herein are trademarks of Xilinx in the United States and other countries. All other trademarks are the property of their respective owner.