

Title: EmPowerSoc flow for Active Power Estimation of SOC

EmPowerSoC is an open-source EDA tool for power estimation of SoC. It has been developed alongside the existing Qflow open-source tool chain which is currently being used for implementation of RTL to GDSII flow.

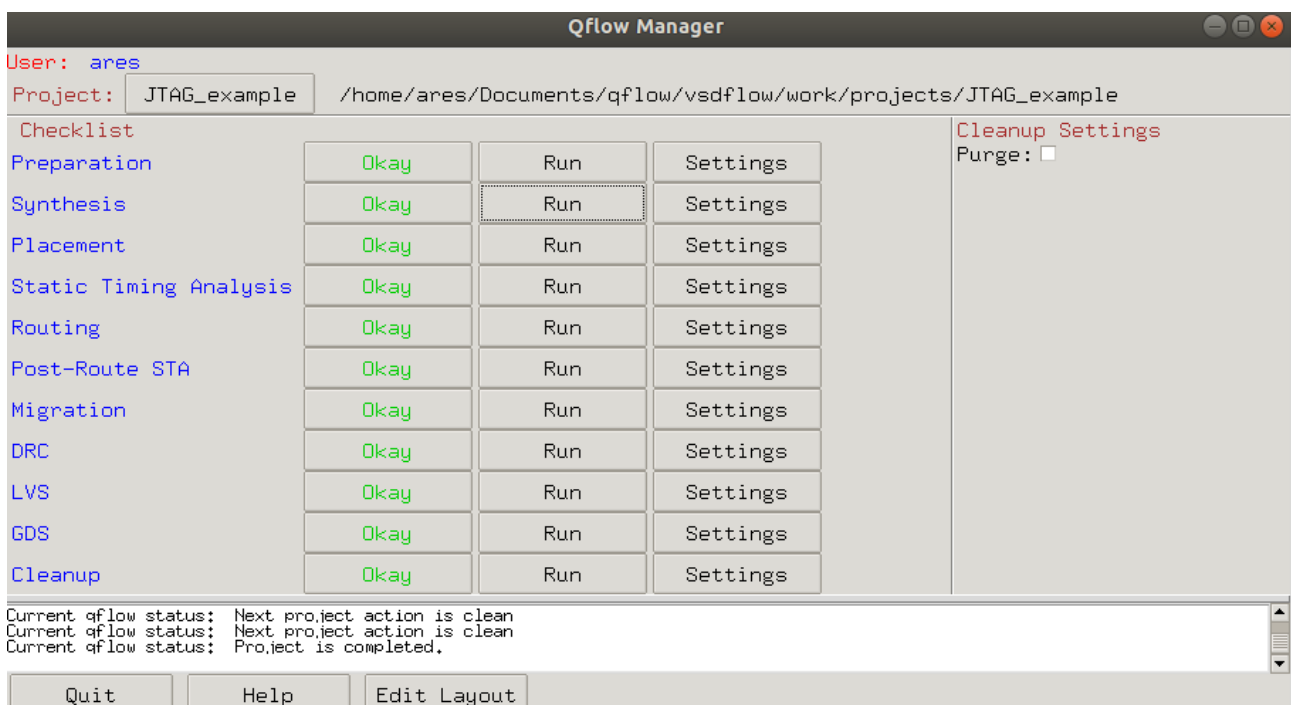
The development work was carried out by Mr. Akshat Jain, Mr. Nimesh Shahdadpuri and Mr. Naveen Dugar, bonafide students of B.E. (ECE) at NSUT under the guidance of Dr. Kunwar Singh, Assistant Professor, Department of ECE, Netaji Subhas University of Technology.

Needless to say copyright of this work lies with VLSI Design Lab, NSUT and distribution of this work in any form for any purpose whatsoever is strictly prohibited.

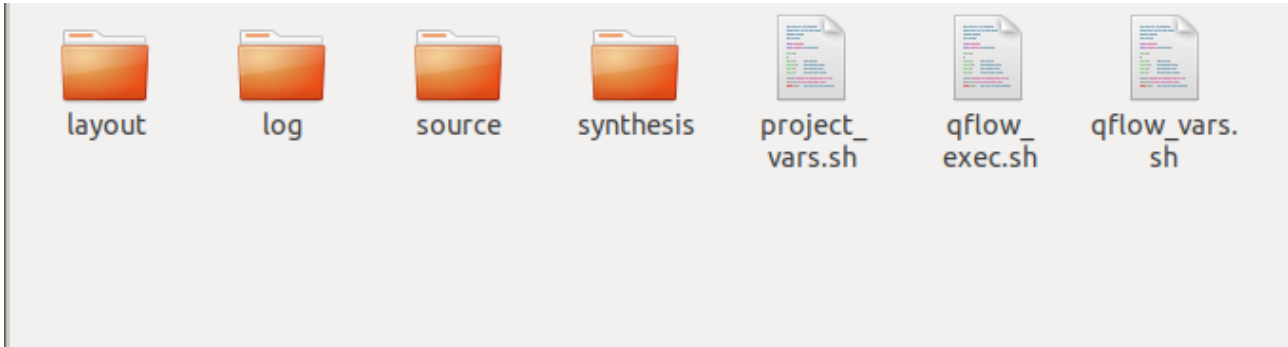
Copyright © 2020, NSUT

All rights reserved.

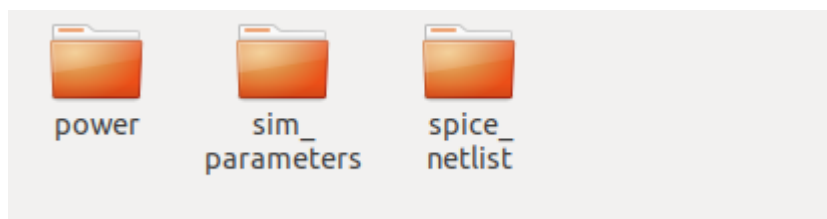
- ➔ Download the provided example source code verilog files for Jtag.
- ➔ Make project directory “**JTAG_Example**” and “**source**” directory inside JTAG_Example. Paste the JTAG verilog files inside source folder.
Use the following commands-
 - mkdir JTAG_Example
 - cd JTAG_Example
 - mkdir source
 - mv <source_location> ./source/tap_top.v
 - mv <source_location> ./source/tap_defines.v
- ➔ In the **JTAG_Example** directory, open the qflow gui using terminal window and run the RTL2GDS flow by selecting required parameters.



- ➔ After completion of RTL2GDS, these below mentioned folders should be created in the project directory.



- ➔ Now, inside the project directory Jtag_example, right click and select open in terminal. Run the command “empower” in terminal to start EmPowerSoc.
- ➔ EmPowerSoc directory will be created in your project directory. It will contain 3 sub directories-
 - ➔ Power – Contains the calculated power values.
 - ➔ Spice Netlist - Contains the spice netlists for all the standard cells
 - ➔ Sim_Parameters – Contains the saved project simulation parameters which can be loaded directly in the future.



- ➔ In the GUI, by changing tabs you can switch between Standby Power estimation and Active Power estimation.

The screenshot displays the EmPowerSOC GUI window titled "EmPowerSOC : tap_top". It features a menu bar with "File" and "Help". Below the menu bar are two tabs: "Active Power Consumption" and "Standby Power Consumption". A blue arrow points to the "Standby Power Consumption" tab, indicating the current active state. The main interface is divided into two columns. The left column, under the "tms_pad_i" dropdown, contains a "Pulse" radio button (selected) and a "DC Input" radio button (unselected). Below these are input fields for "Initial Value [V]", "Pulsed Value", "Delay", "Rise Time", "Fall Time", "Pulse Width", and "Period", each followed by an "Add Pulse" button. Below the "DC Input" section is a "Value [V]" input field followed by an "Add Input" button. The right column, under the "tdo_pad_o" dropdown, contains a "Value" input field followed by an "Add Load" button. Below this are input fields for "Supply Value [V]", "Stop Time", and "Time Step Size". At the bottom of the right column is an "Add Model file" button. A large empty rectangular area occupies the bottom center of the window. At the very bottom, there are two orange buttons: "Review Simulation Parameters" and "Run Simulation".

➔ Apply input pulses to all the required ports and add DC inputs wherever required.

The screenshot shows the 'EmPowerSOC : tap_top' application window. It has a menu bar with 'File' and 'Help'. Below the menu bar are two tabs: 'Active Power Consumption' (selected) and 'Standby Power Consumption'. The main area is divided into two columns. The left column is for 'tms_pad_i' and the right column is for 'tdo_pad_o'. In the left column, the 'Pulse' radio button is selected, and various pulse parameters are set: Initial Value [V], Pulsed Value, Delay (0p), Rise Time (10p), Fall Time (10p), Pulse Width (1n), and Period (2n). There is an 'Add Pulse' button. Below that, the 'DC Input' radio button is unselected, and there is a 'Value [V]' field with an 'Add Input' button. A blue arrow points to the 'DC Input' radio button. The right column has a 'Value' field with an 'Add Load' button. Below that, there are fields for 'Supply Value [V]', 'Stop Time', and 'Time Step Size'. At the bottom of the right column is an 'Add Model file' button. At the very bottom of the window are two large orange buttons: 'Review Simulation Parameters' and 'Run Simulation'.

EmPowerSOC : tap_top

File Help

Active Power Consumption Standby Power Consumption

tms_pad_i tdo_pad_o

☒ Pulse

Initial Value [V]

Pulsed Value

Delay 0p

Rise Time 10p

Fall Time 10p

Pulse Width 1n

Period 2n

Add Pulse

☐ DC Input

Value [V]

Add Input

Add Load

Supply Value [V]

Stop Time

Time Step Size

Add Model file

Review Simulation Parameters Run Simulation

- Specify the load capacitances for the output nodes. And add the Supply Value, simulation time and time step for simulation.

EmPowerSOC : tap_top

File Help

Active Power Consumption Standby Power Consumption

tms_pad_i

☒ Pulse

Initial Value [V]

Pulsed Value

Delay

Rise Time

Fall Time

Pulse Width

Period

Add Pulse

☐ DC Input

Value [V]

Add Input

tdo_pad_o

Value 10pf

Add Load

Supply Value [V] 1.8

Stop Time 40n

Time Step Size 10p

Add Model file s:/Documents/qflow/vsdfow/work/scripts/SampleLib.mod

Review Simulation Parameters Run Simulation

- Add the model file for NMOS and PMOS. We have provided a sample model file for reference.

Note: If you want to use your own model parameters, kindly ensure the library name and model name are same as the one in sample model file provided.

➔ Review the simulation parameters. Make any changes if necessary.

EmPowerSOC : tap_top

File Help

Active Power Consumption Standby Power Consumption

tms_pad_i tdo_pad_o

☒ Pulse

Initial Value [V]

Pulsed Value

Delay 0p

Rise Time 10p

Fall Time 10p

Pulse Width 1n

Period 2n

Add Pulse

☐ DC Input

Value [V]

Add Input

Value 10pf

Add Load

Supply Value [V] 1.8

Stop Time 40n

Time Step Size 10p

Add Model file s:/Documents/qflow/vsdfow/work/scripts/SampleLib.mod

Supply 1.8
Stop Time 40n
Step Size 10p

Input DC bs_chain_tdi_i 0V
Input DC mbist_tdi_i 0V
Input DC debug_tdi_i 0V
Input Pulse tms_pad_i Initial_Value 0 Pulsed_Value 1.8 Delay 0p Rise_Time 10p Fall_Time 10p Pulse_Width 1n Period 2n
Input Pulse tck_pad_i Initial_Value 0 Pulsed_Value 1.8 Delay 0p Rise_Time 10p Fall_Time 10p Pulse_Width 1n Period 2n
Input Pulse trst_pad_i Initial_Value 0 Pulsed_Value 1.8 Delay 0p Rise_Time 10p Fall_Time 10p Pulse_Width 2n Period 20n
Input Pulse tdi_pad_i Initial_Value 0 Pulsed_Value 1.8 Delay 0p Rise_Time 10p Fall_Time 10p Pulse_Width 1n Period 2n
Load tdo_pad_o 10pf

Review Simulation Parameters Run Simulation

- ➔ Finally, run the simulator. You will be asked to provide a name for your project in order to save it.

EmPowerSOC : tap_top

File Help

Active Power Consumption Standby Power Consumption

tms_pad_i tdo_pad_o

☒ Pulse

Initial Value [V]

Pulsed Value

Delay 0p

Rise Time 10p

Fall Time 10p

Pulse Width 1n

Period 2n

Add Pulse

☐ DC Input

Value [V]

Add Input

Value 10pf

Add Load

Supply Value [V] 1.8

Stop Time 40n

Step Size 10p

Add Model file /Documents/qflow/vsdfow/work/scripts/SampleLib.mod

Input

Enter file name

TestProject

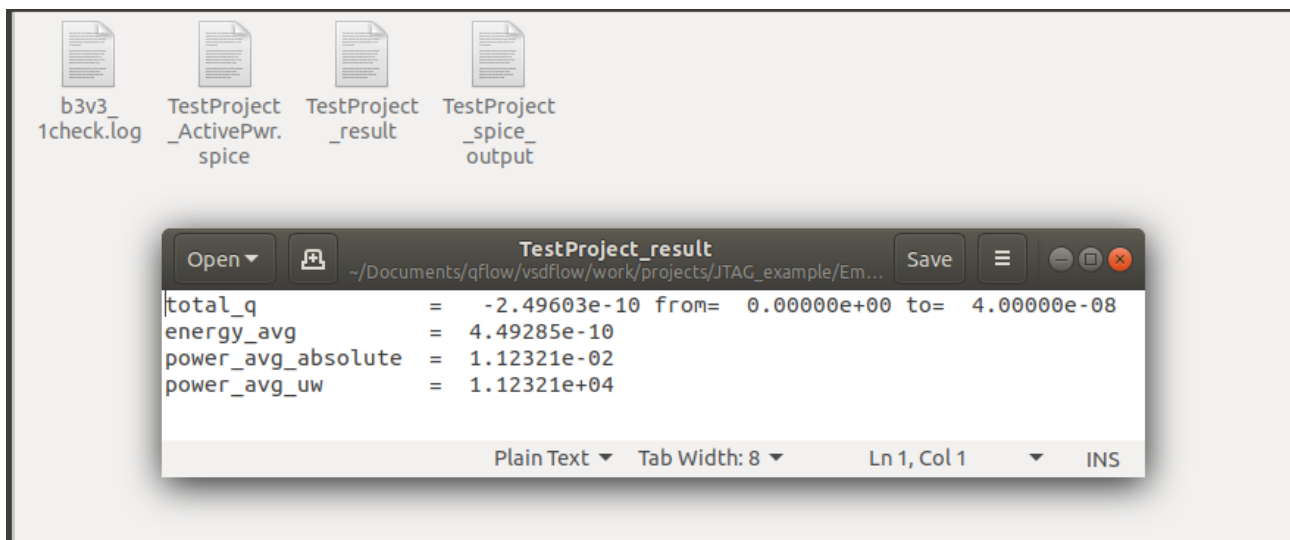
Cancel OK

Supply 1.8
Stop Time 40n
Step Size 10p

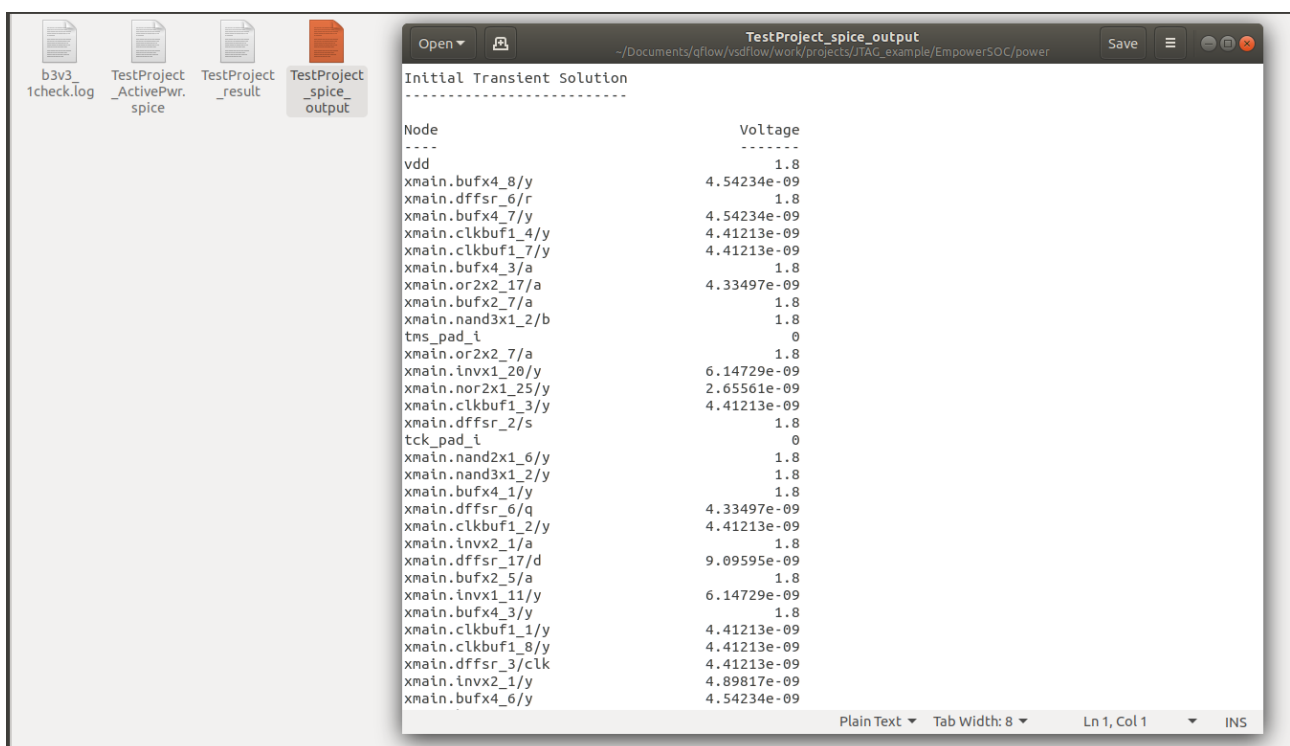
Input DC bs_chain_tdi_i 0V
Input DC mbist_tdi_i 0V
Input DC debug_tdi_i 0V
Input Pulse tms_pad_i Initial_Value 0 Pulsed_Value 1.8 Delay 0p Rise_Time 10p Fall_Time 10p Pulse_Width 1n Period 2n
Input Pulse tck_pad_i Initial_Value 0 Pulsed_Value 1.8 Delay 0p Rise_Time 10p Fall_Time 10p Pulse_Width 1n Period 2n
Input Pulse trst_pad_i Initial_Value 0 Pulsed_Value 1.8 Delay 0p Rise_Time 10p Fall_Time 10p Pulse_Width 2n Period 20n
Input Pulse tdi_pad_i Initial_Value 0 Pulsed_Value 1.8 Delay 0p Rise_Time 10p Fall_Time 10p Pulse_Width 1n Period 2n
Load tdo_pad_o 10pf

Review Simulation Parameters Run Simulation

- The Estimated Power will be saved as a text file TestProject_result in the EmpowerSoc>Power directory.



- The complete ngspice output will also be saved.



- ➔ In future, you can load these parameters directly into the tool by going into File>Open, and then selecting the simulation parameters file for that project.

