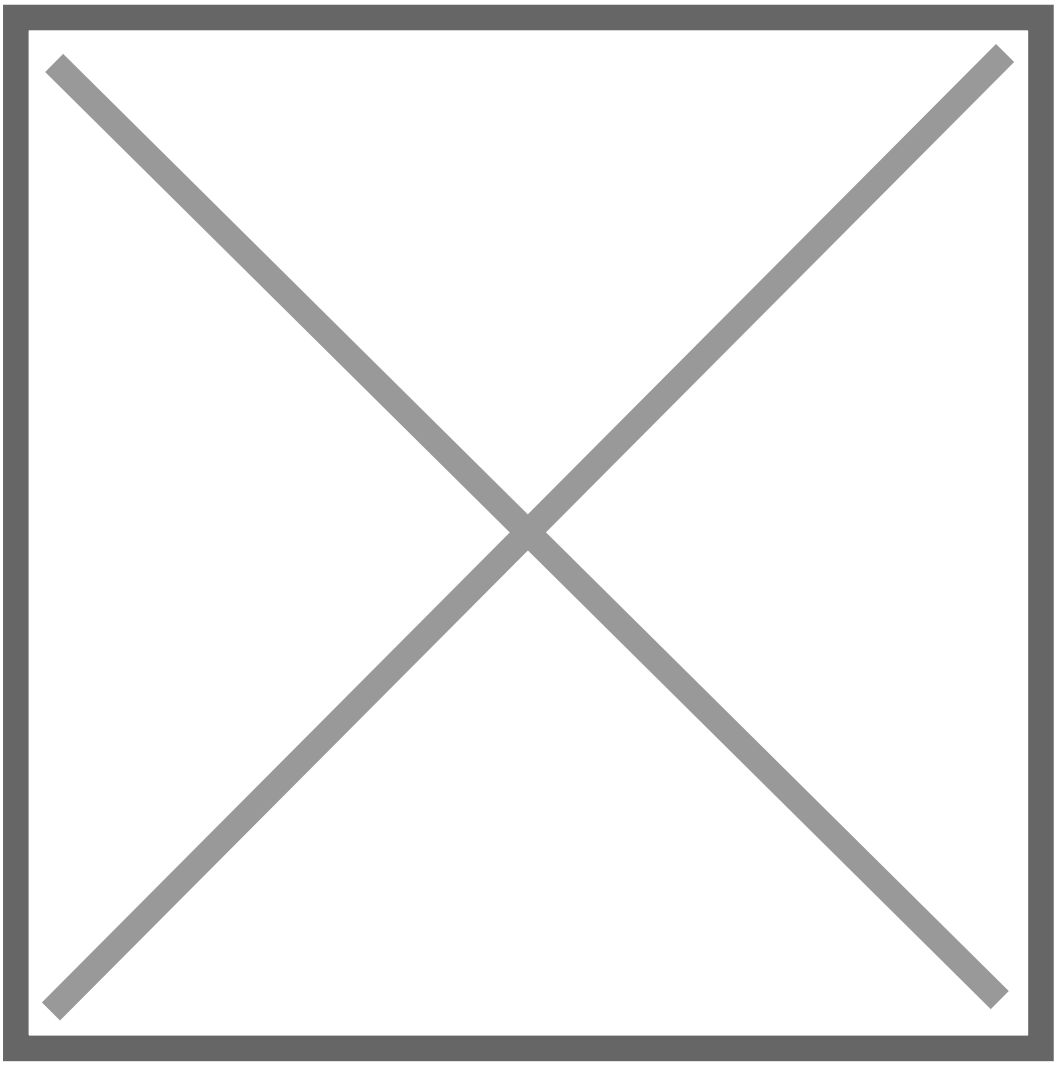
Getting Started with ChipInventor

- Creating a New Project
- Building the Circuit in the Blocks Page
- Exploring the Files Page
- Simulating the Project
- Synthesizing the Circuit
- Viewing the Project in 3D
- Reviewing Results (Results Tab)
- Wrapping Up

Creating a New Project

In this tutorial, you will create your first project and learn about the ChipInventor design flow. As a practical example, we'll create a basic AND gate circuit. To further reinforce your understanding, you'll be challenged to implement additional logic gates, such as OR and NOT, on your own.

1. **Click New Project on the top navigation bar or underneath it**



2. **Fill the project details:**

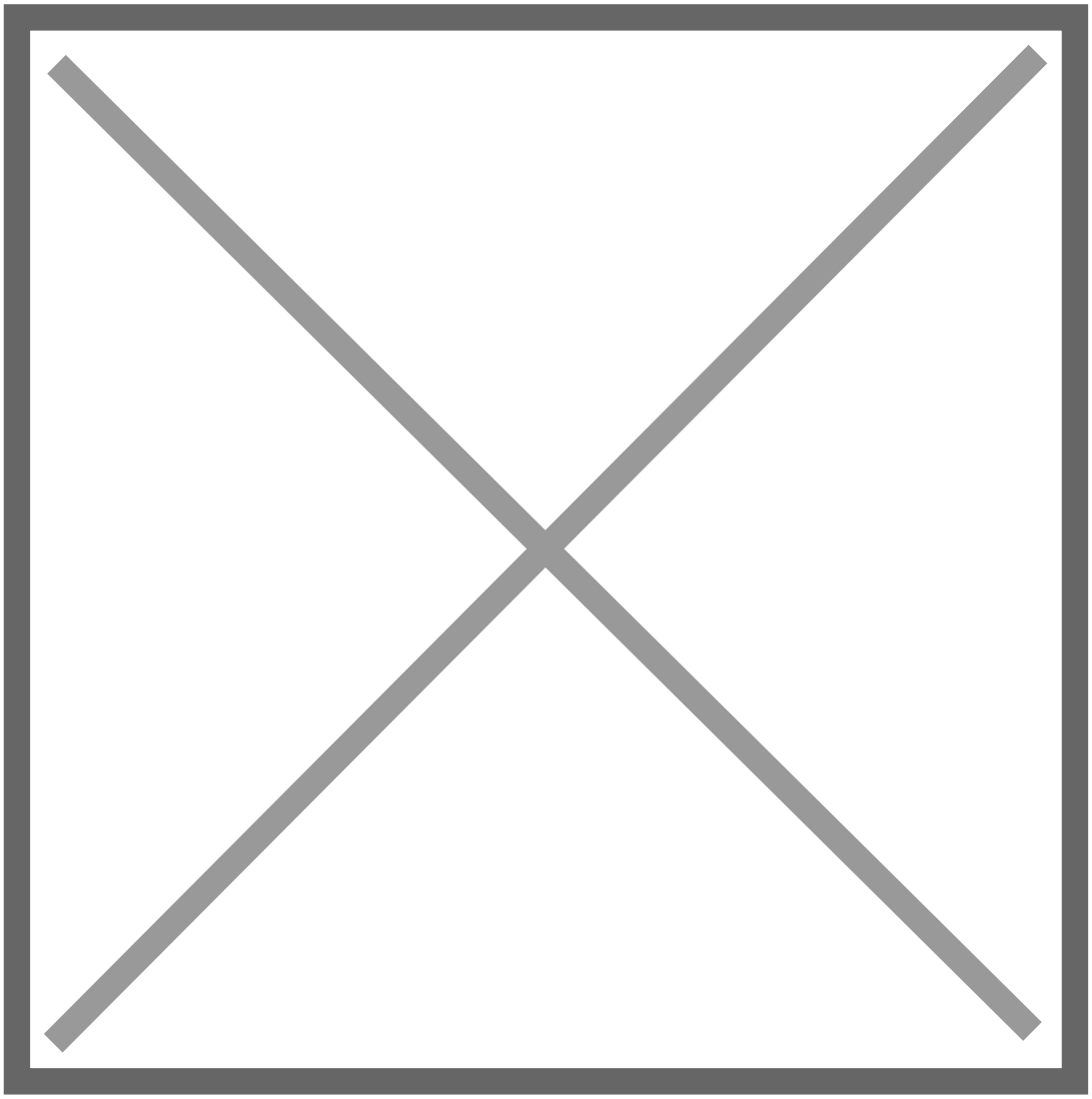
- Name: AND Gate.
- Description: Basic example of an AND logic gate.
- Type: OpenLane SKY360.

3. **At the bottom of the page, click Create Chip to start. You'll be redirected to the Blocks Page.**

Building the Circuit in the Blocks Page

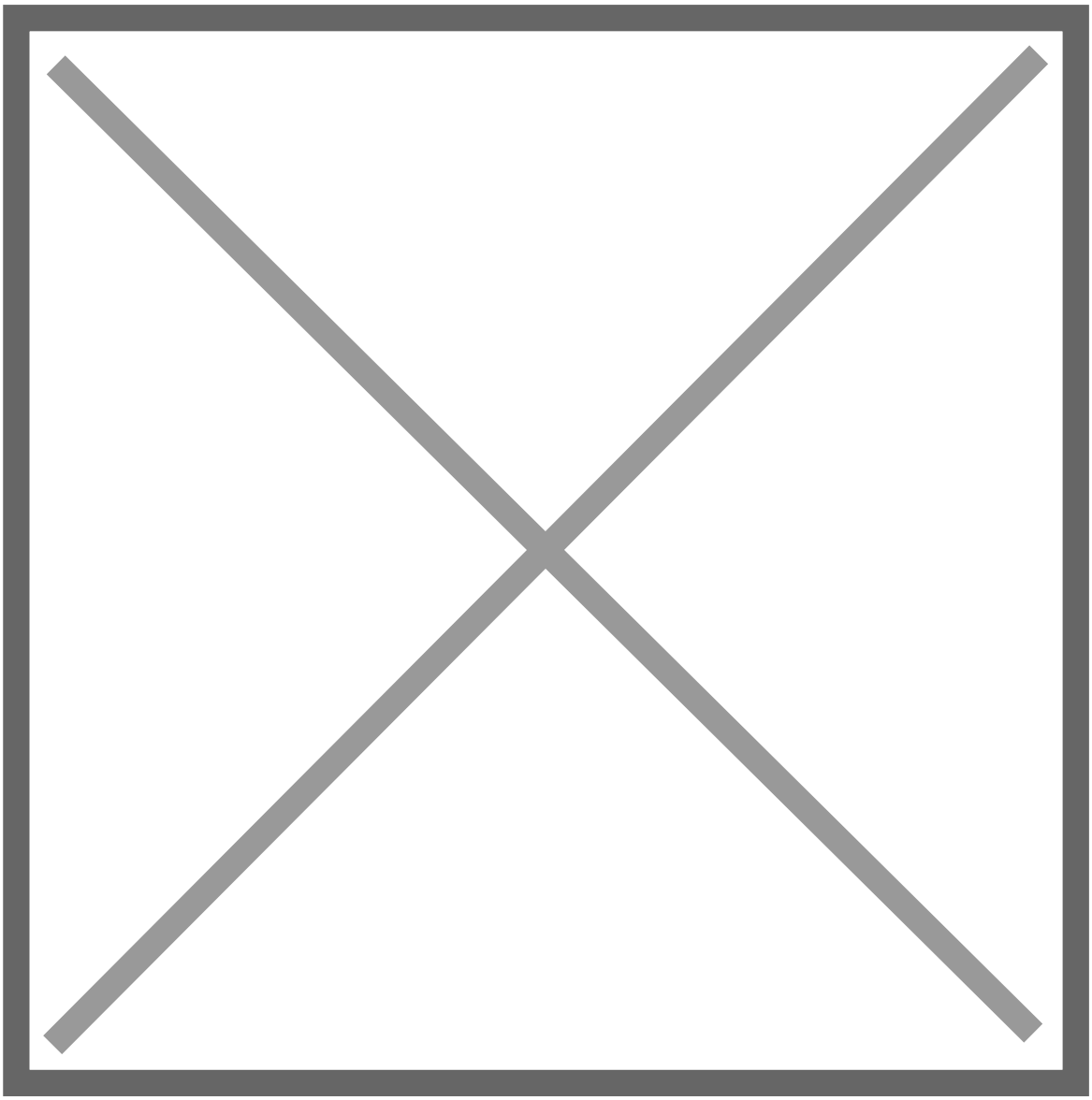
1. **From the left menu, drag the following blocks to the workspace:**

- **Input:** Add two blocks to represent the gate’s inputs.
- **Output:** Add one block to represent the gate’s output.
- **AND:** Use the search bar to locate this block and drag it to the workspace.



2. **Connect the blocks:**

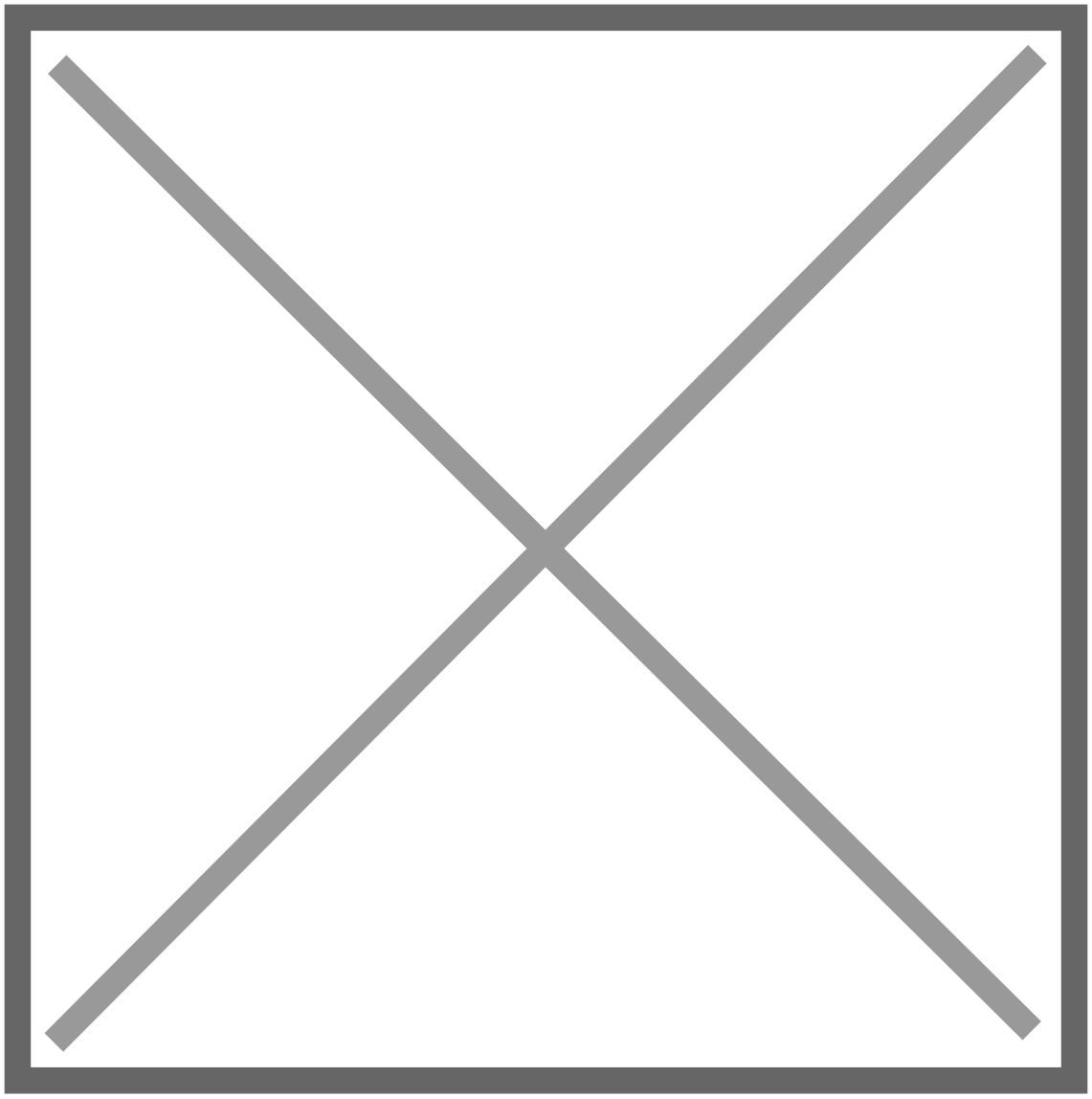
- Click on the output pin of each Input block and drag to the input pins of the AND block.
- Connect the output pin of the AND block to the Output block.



3. **Name the variables:**

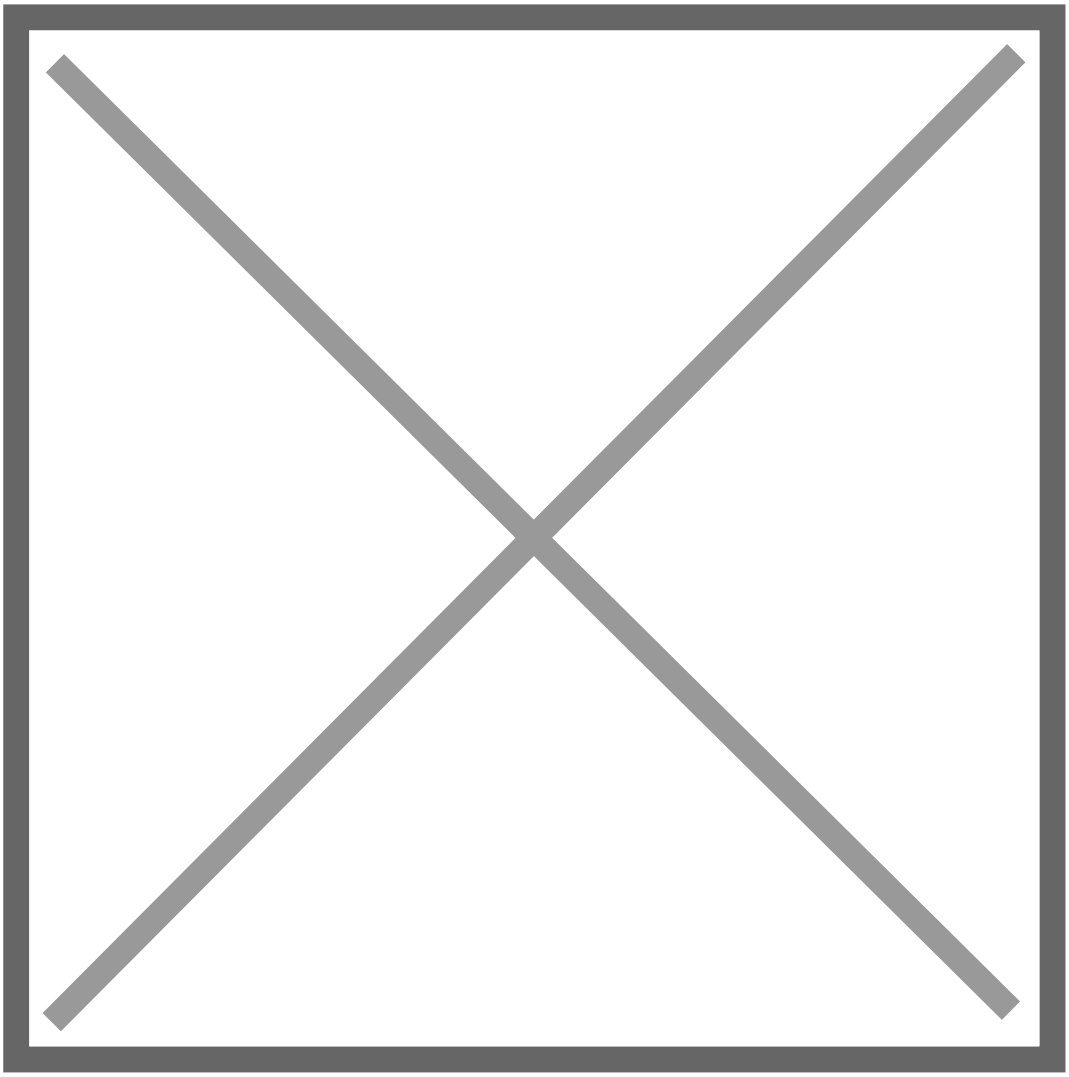
- **Input 1:** a.
- **Input 2:** b.
- **Output:** out.

4. **At the top center of the page, click Save to save your project. It will appear a pop-up confirming the operation.**



Exploring the Files Page

1. **Now, it’s time to automatically generate the Verilog code for your project. On the top navigation bar, click Files. Similar to the Save function, a pop-up will appear to confirm this operation.**



> In addition to the Verilog code, the system creates all the files necessary to synthesize your project.

2. **In the Explore section on the left, browse your project files:**

- Right-click on files to open options.
- Double-click on the main file to open it in the editor.

3. **Review the Verilog code generated from the Blocks tab diagram.**

- This code translates the blocks into hardware description language.
- For beginners, it’s recommended not to modify the code directly.

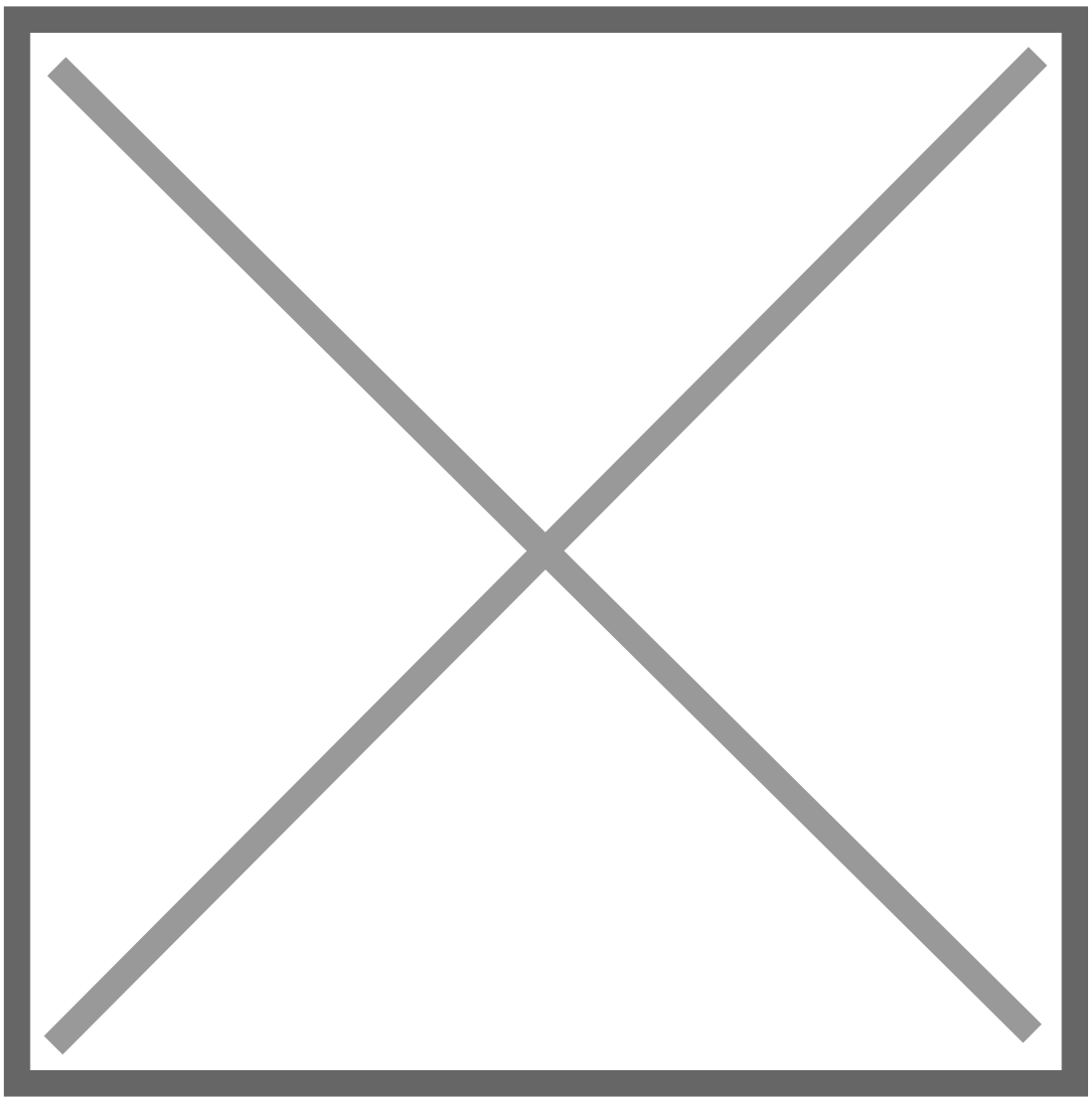
Simulating the Project

1. On the top navigation bar, go to the Simulate tab and select Dynamic Simulation.

2. The Verilog code will load automatically.

3. Click Run to execute the simulation:

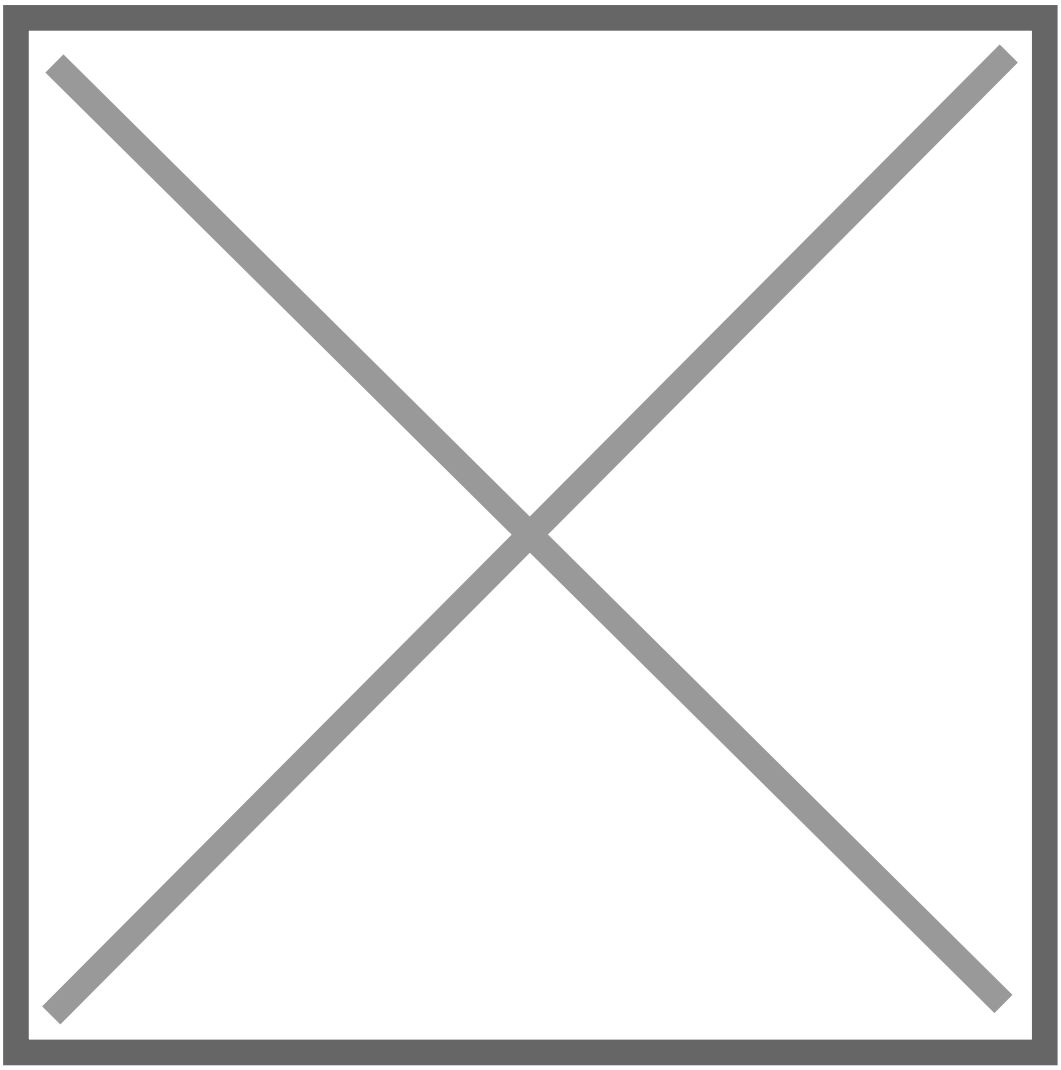
- Use the interactive menu to change the input values (a and b).
- Observe the output (out) to verify the behavior of the AND gate.



4. Inspecting Signals

- Hover your cursor over a wire in the circuit, and a magnifying glass icon will appear.
- Click on the wire to visualize the signal waveform for that specific connection.

- To compare signals, click on another wire. The tool will display both signals in the same graph for side-by-side analysis.

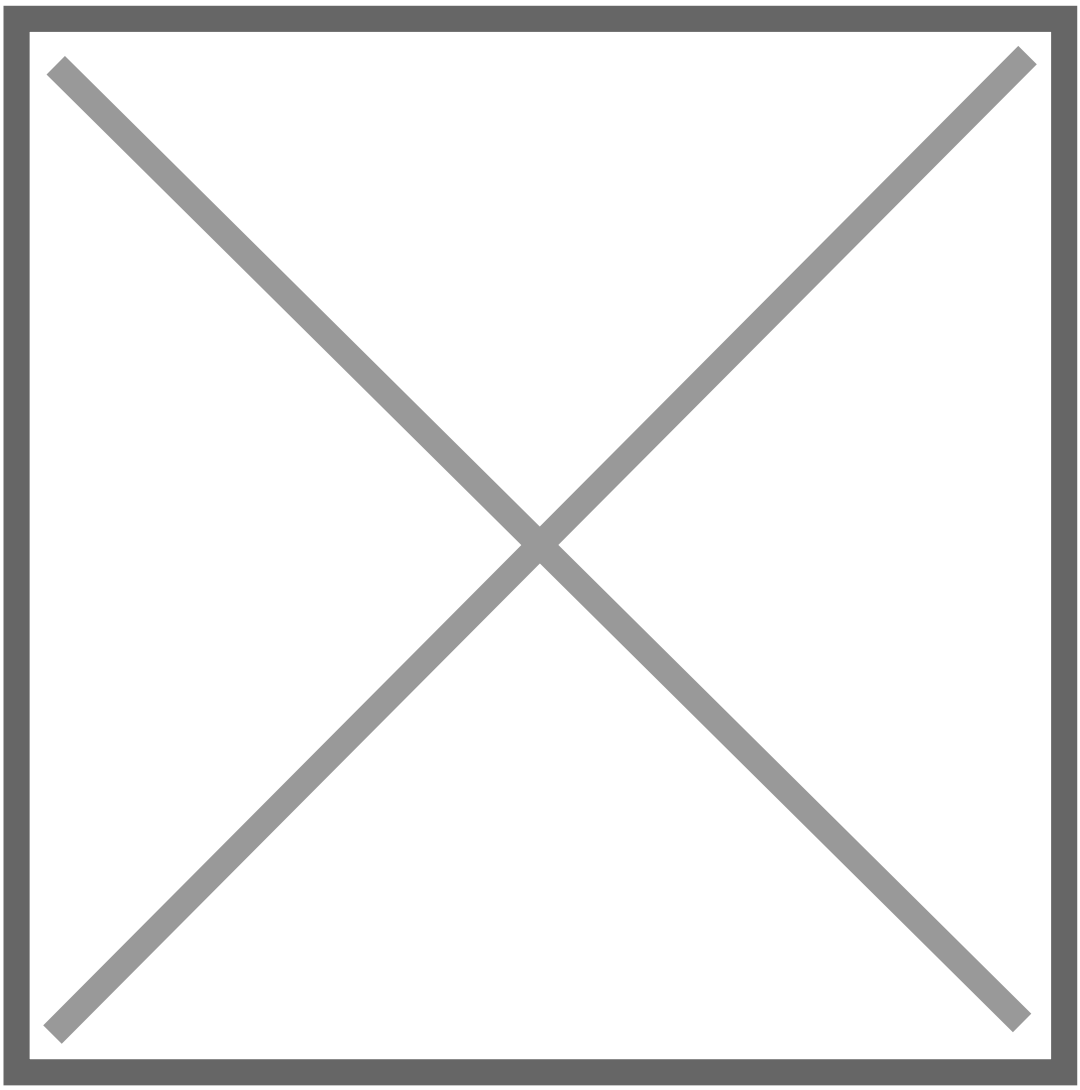


5. Adjusting Graphs You can use the graph controls in the lower menu to adjust the speed and position of the waveform display. You can adjust this menu by clicking and dragging on its top edge.

  - **Scale:** Modify the simulation speed to slow down or speed up the signal playback, enabling detailed analysis of transitions and behaviors.
  - **Range:** Scroll through the time axis to focus on specific parts of the curve, making it easier to study signal changes and timing.

6. Advanced Simulate

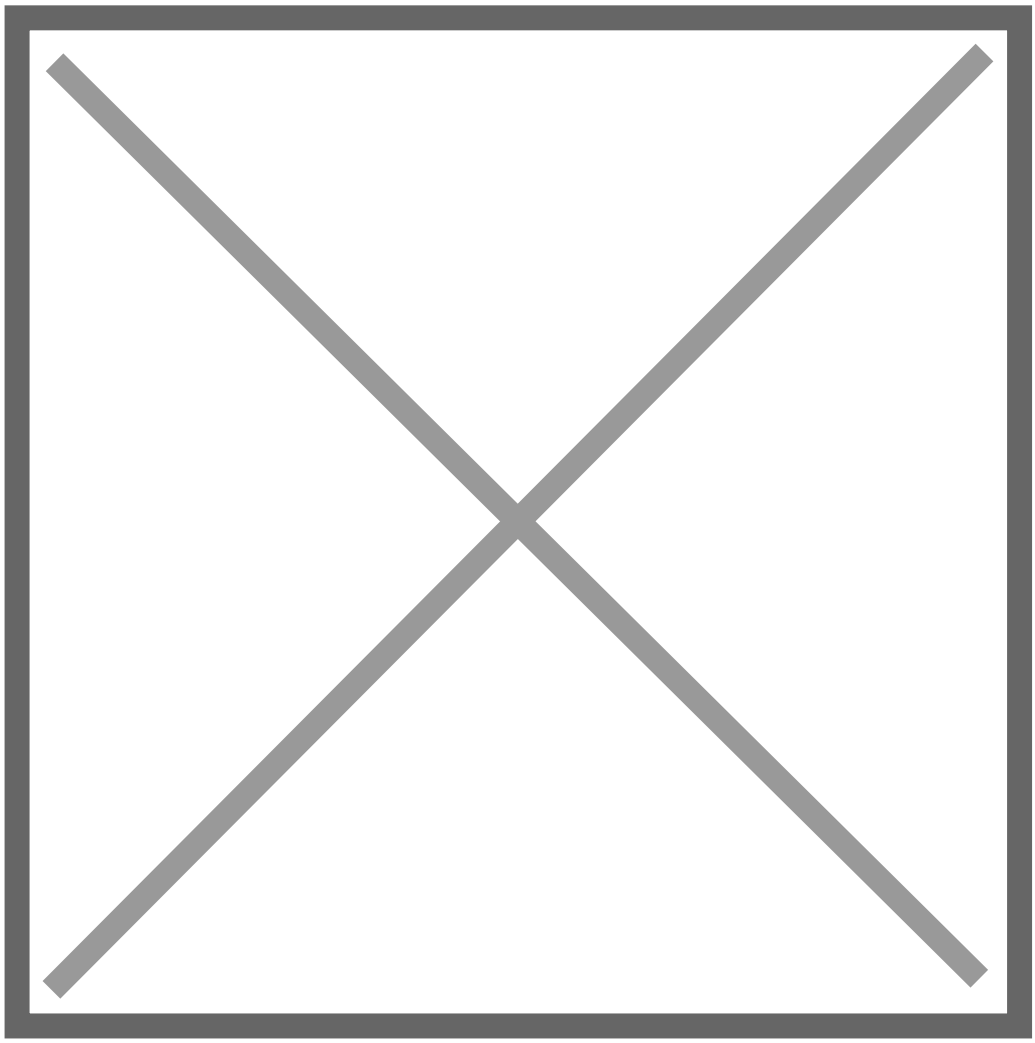
For further exploration, select Advanced Simulate and use tools like Run Iverilog to check for code errors.



The Advanced Simulate option provides additional tools for in-depth circuit analysis, including error checking, detailed debugging, and timing verification. These advanced features will be covered in future tutorials, where we'll explore their functionality in more detail.

Synthesizing the Circuit

1. On the top navigation bar, click the Synthesize tab to transform your Verilog hardware description into a physical layout in GDSII format.



2. The synthesizer integrates various tools to perform the main chip design tasks, including:

- RTL Synthesis
- Equivalence Checking
- Floorplanning
- Placement
- Routing
- Timing Analysis
- Signoff Checks
- GDSII Generation
- Creates a 3d view of your project

> **Note:** The synthesis may take a time. **Be patient and wait to finish.**

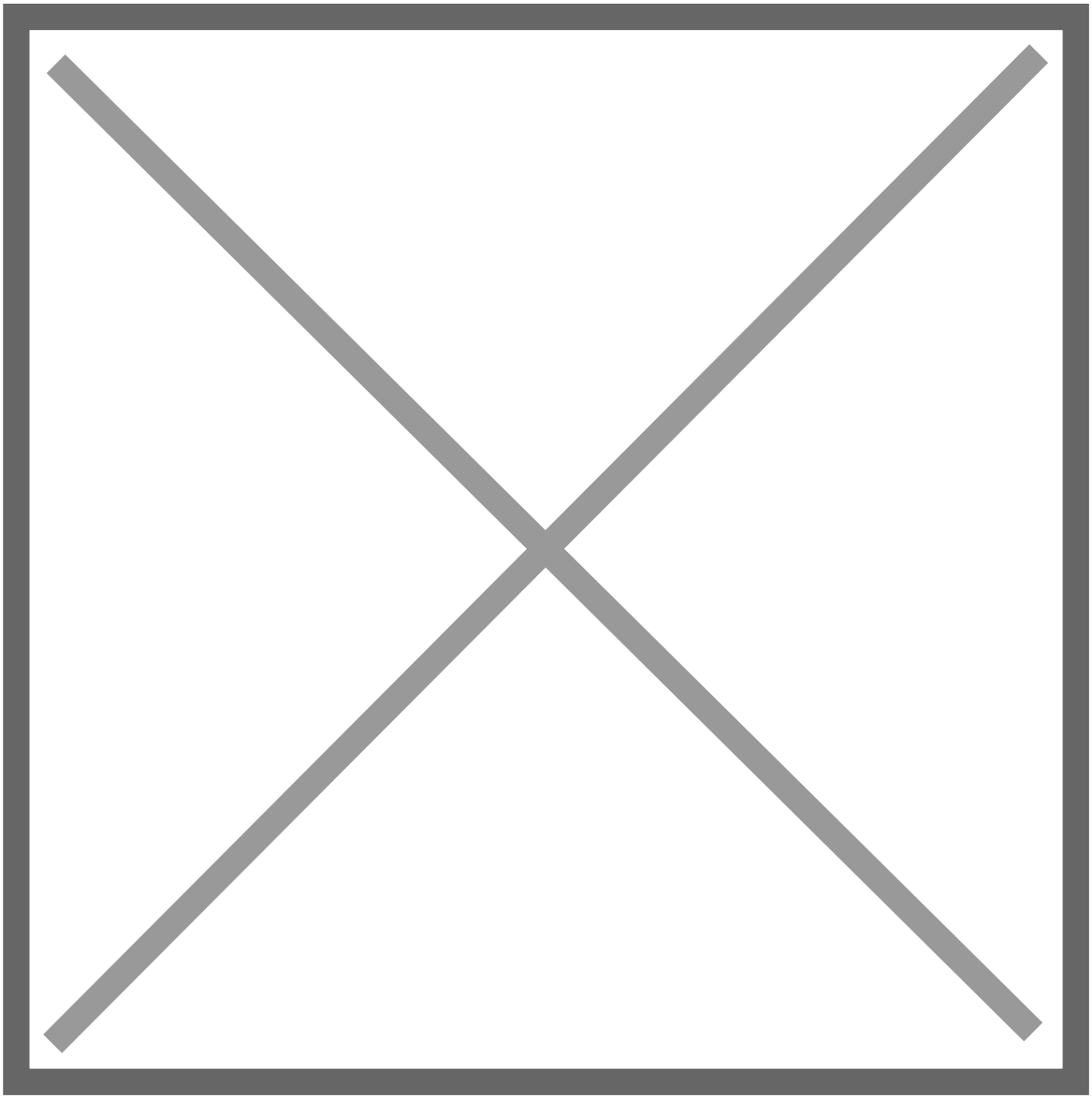
3. When the synthesis finishes, you can view the synthesis log by clicking on the “Green Tick”.

Viewing the Project in 3D

1. On the top navigation bar, navigate to the View tab and select Interactive 3D Layout.

2. Use the controls to explore the layout:

- **Rotate:** Click and drag with the left mouse button.
- **Zoom:** Use the mouse scroll wheel.
- **Pan:** Press Ctrl + left mouse button or right mouse button to reposition the layout.



3. Filter Layout Elements
The layout includes several visualization options to help focus on specific elements:

- Toggle Filler Cells: Hide non-functional elements like filler cells.

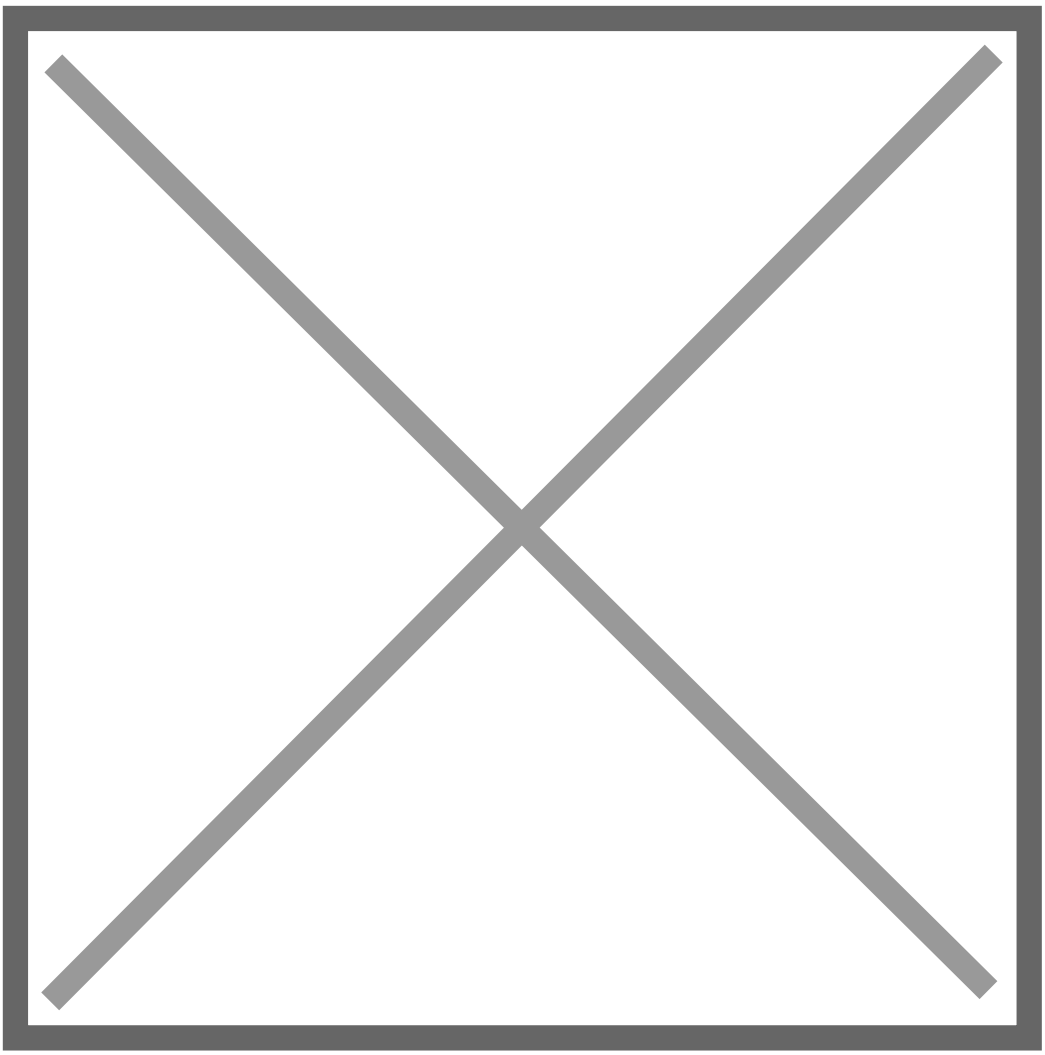
- Toggle TopCellGeometry: Remove power and ground buses from the view.
- Toggle Instance Labels: Show or hide labels for individual components.
- Toggle Net Connections: Highlight the connections between components.

> These options allow for a cleaner and more focused view of the circuit, depending on what you wish to analyze.

4. Isolating Specific Cells

To focus on a particular component, use this feature to analyze specific parts of the circuit in detail:

- Hover the mouse over the desired cell, and then:
- The first option is to click on it to isolate the component, filtering out unrelated elements, in the right bar, using the filter layout elements.
- The second option is to click on the buttons 1, 2 and 3 on your keyboard, with the mouse over the desired cell. Click 3 again to visualize the whole chip again.



Reviewing Results (Results Tab)

1. Open the Results tab to access a summary of the synthesis process.

2. The results table provides essential metrics, such as:

- **Performance:** Insights into timing and speed.
- **Area Utilization:** The size of the circuit on the chip.
- **Power Consumption:** Estimates based on the design.

3. 2D Visualization
The Results tab also includes a 2D layout view:

- This allows you to inspect the circuit’s placement and routing in a simplified two-dimensional format.
- Use this tool to verify the design’s integrity and ensure proper alignment of components.

> Both the 3D and 2D views provide complementary insights into your project, making them valuable for design validation and optimization.

Wrapping Up

Return to the tool's dashboard by clicking Home on the top menu. Your project will appear under Recently Open Projects, ready to be reopened and continued.

- **Final Challenge:** Create and test OR and NOT gates on the platform. Follow the steps learned in this tutorial, from circuit design in the Blocks tab to 3D visualization in the View tab.

> **Tip: Revisit this tutorial whenever you have questions. Practice frequently to master ChipInventor’s tools. Good luck!**