Atlas EDA

User's Guide v.2025.3

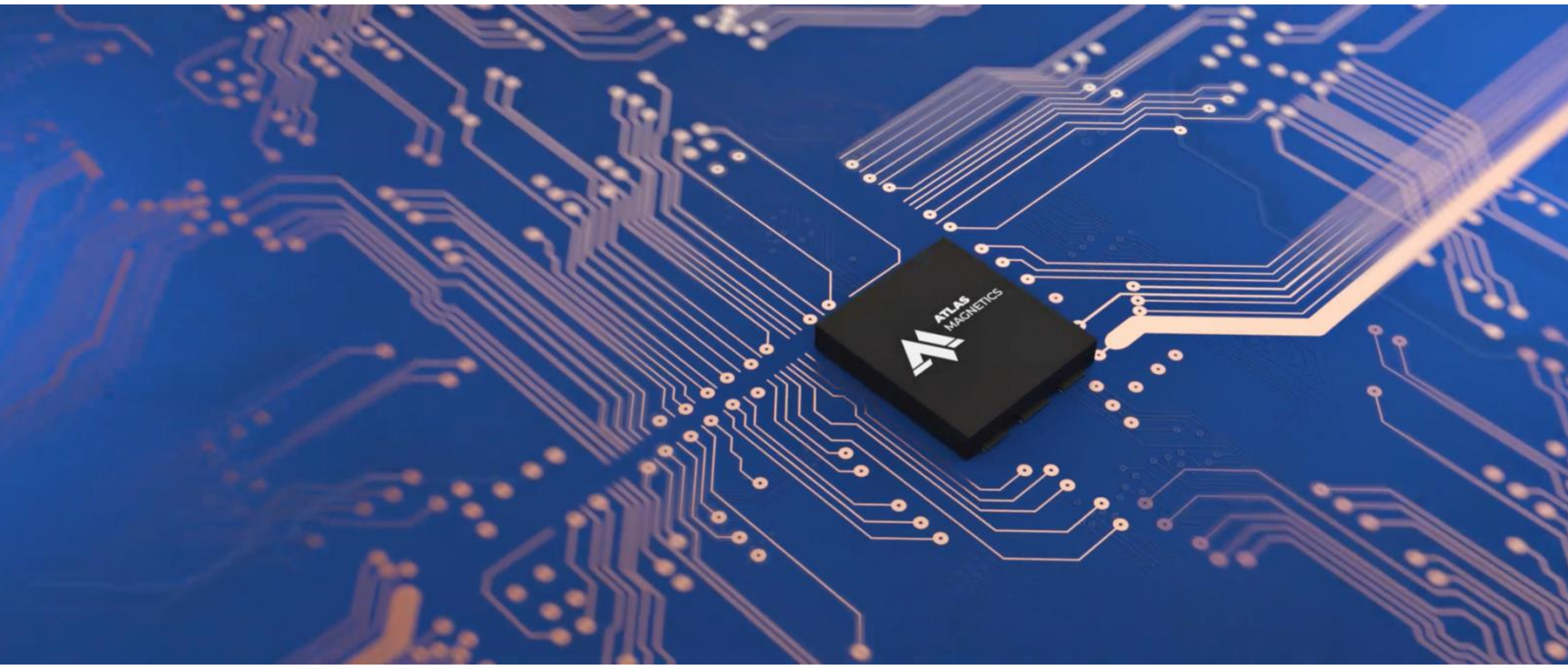
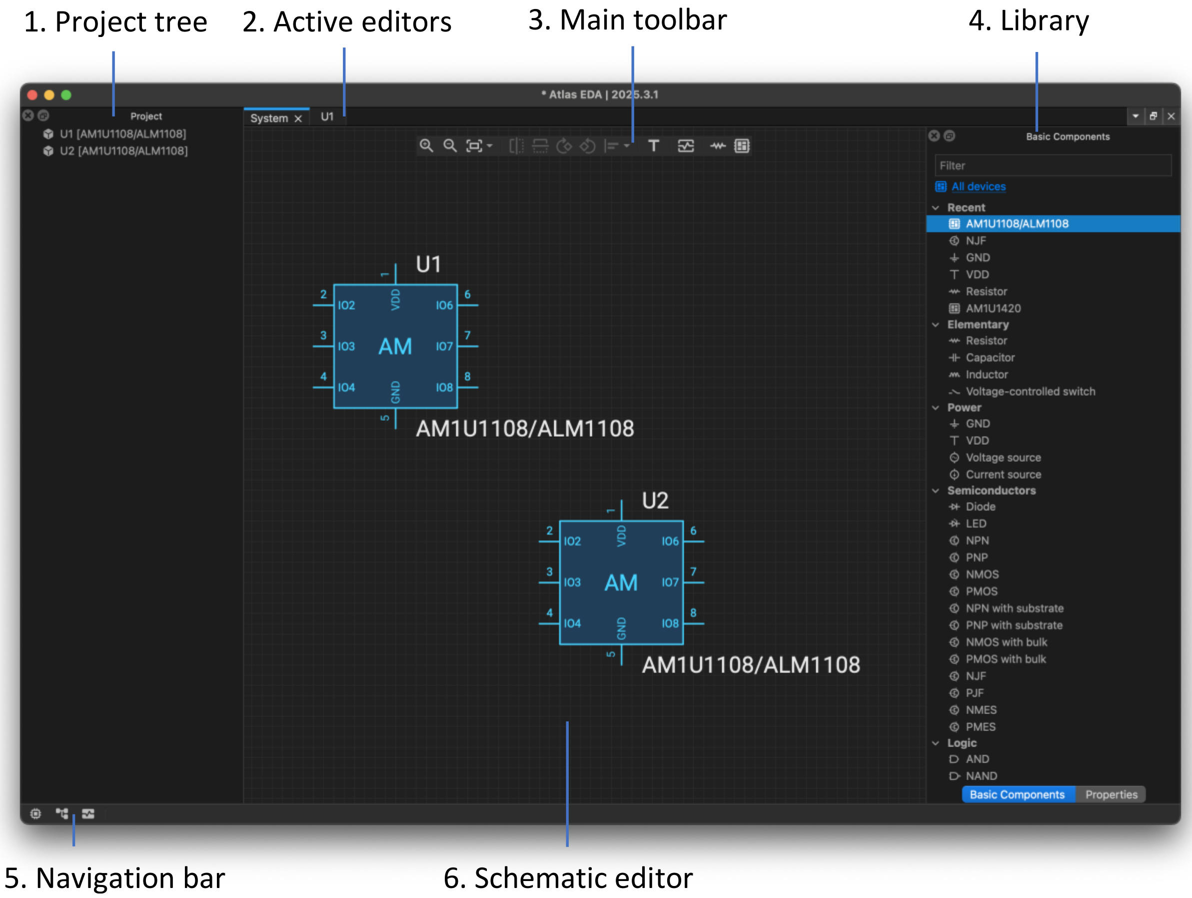


Table of Contents

Main window structure



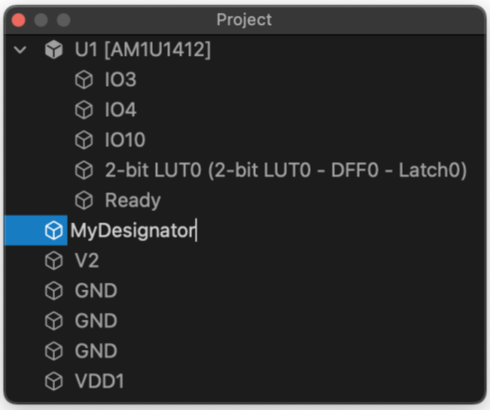
Atlas EDA is a multi-window schematic capturing tool. Blank project starts from *'System'* level editor (**2. Active editors**), which main purpose is to capture the complete schematic of your device, that can include the wide variety of components available in the **built-in Library (4)**.

Most of the components contain configurable parameters and will be available for configuration via Properties panel (double-click on component). Some components with high level of configurability, like Atlas Magnetics uASICs, support configuration via separate visual editors. You can open such editors and work in parallel.

Project tree

Project tree (1) is a main navigation tool for the project. Double-click on any specific item in the list to locate it on schematic.

Component designator can be changed using **[ENTER]** (for macOS) or **[F2]** (for Windows):

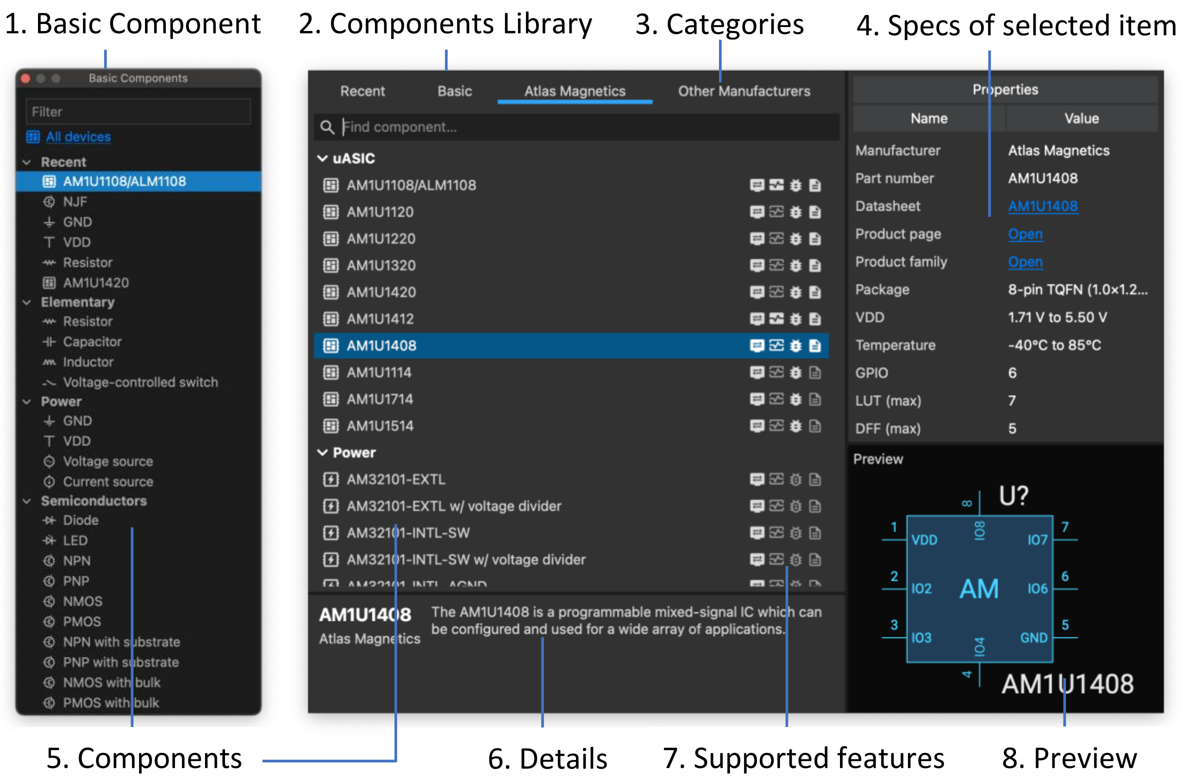


Components Library

Atlas EDA provides multiple Libraries of components, based on a design level.

System level Library

Basic Components (1) library (on a toolbar) and **Components library (2)** (on a toolbar or **[A]**) are available on a *System* level (*Basic Components* is a subset of *Components library*, represented in a separate dock panel for quick access).



Components library contains items like resistor, capacitor, inductor, as well configurable ICs (like uASIC, Power devices from Atlas Magnetics), separated by categories

Components in specific category are grouped by product family and show the list of supported features:

Visual editor – basically means there is a dedicated editing tool for this component. It could be schematic editor, structure visualizer, etc.
Simulation – if simulation model is available for this component, then you will be able to simulate schematic, which contains it.
Hardware Debug – dedicated debug tool, which requires specific hardware, like Ferro Board.
Datasheet – datasheet or other documentation for specific component.

Adding components from Basic Components library, Components library and Macrocells library:

1. Double-click on specific component you want to add
2. By default, continuously add as many components of selected type as you need, using left mouse button **(LMB)**. Use right mouse button click **(RMB)** or **[ESC]** to stop adding items.
3. (Optional) press and hold **[SHIFT]** for opposite behavior – add single item, select and open Properties panel
   a. You can redefine default behavior in Setting->Schematic (*Default object placement method*)

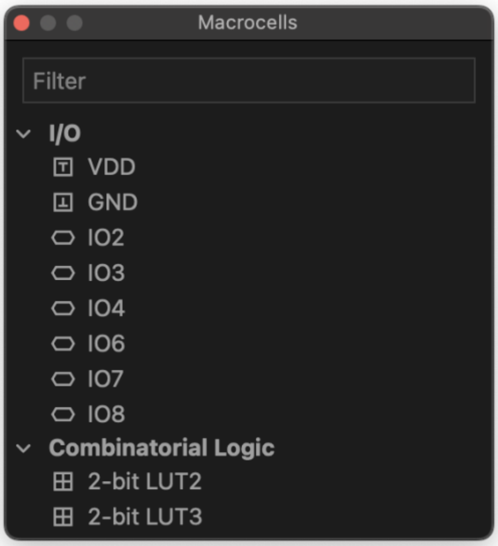
Note: Drag-and-drop is supported on Basic Components

Secret keys, before you drop the component:

a. **[X]**, **[Y]**: flip left-to-right and top-to-bottom
b. **[R]**, **[L]**: rotate clockwise and counter-clockwise
c. **[SPACE]**: change the macrocell function (works for AM uASICs Multifunctional category macrocells. E.g.: LUT->DFF->Latch)
d. **[SHIFT]**: alternate component adding behavior to the opposite style

Component level Library (uASIC)

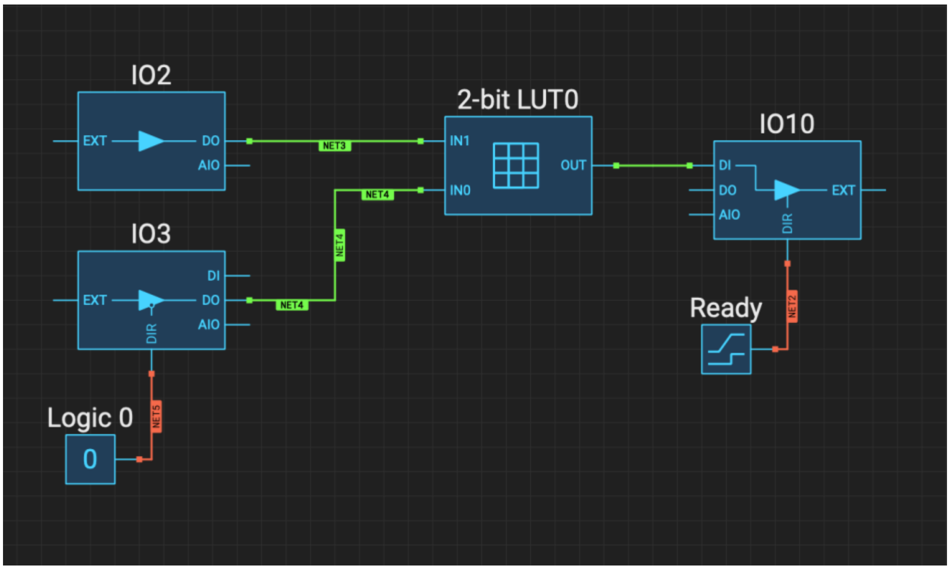
Components like uASICs, which support visual/schematic editing, introduce a list of macrocells (internal resources of uASIC), represented in Library form – **Macrocells** library.



The procedure of adding components from the Macrocells library is the same as for Basic Components (see previous chapter).

Schematic editor

Schematic editor essentially is a tool for capturing the schematic of your design. In addition, some components (like uASIC) support visual editing, in particular the schematic editing. You can open as many editors as you need and work in parallel. All the changes will be synchronized within your project.



Adding connections:

1. Click on the first node (source)
2. (Optional) click on the work area to introduce the custom point in the connection path
3. Click on the second node (destination)

Secret keys, while adding new connection:

a. **[ESC]** or **RMB** to cancel
b. **[SPACE]** switches the routing mode: Auto -> 'L' corner -> '7' corner -> direct line
c. **RMB** declines the latest custom point in the path or cancels the path if no custom points were left
d. Hold **[SHIFT]** for continuous adding;
e. Hold **[CTRL]** or **[COMMAND]** to force create Net Label (described in the following chapters);

Editing connections:

Click on connection to select, then:

a. Drag segment of connection or corner point to change position
b. Double-click on segment to create new segment
c. Double-click on the corner point to reset connection path
d. Or use context menu. Note: context menu of a corner point has additional actions (e.g. **Remove** point)

Removing items:

1. Select item you want to remove (component, connection, text item, etc.)
2. Press **[DELETE]** or use **Delete** action in context menu.

Connection types:

a. Regular connection (green): just a normal connection, works as you may expect
b. System connection (red): special restricted/protected connection, usually autogenerated, illustrates a specific configuration (e.g.: AM1U1108 IO4 macrocell to operate properly as 'Digital out' requires the IO4 DIR node to be connected to Logic 1 OUT node. Corresponding system connection will be added as soon as you configure IO4 as 'Digital out')
c. Shared connection (purple): it is basically a Regular connection (green) but shared with multiple nodes. Usually appears in uASICs, where multiple input nodes have the same address, which makes a connection to one of them shared with all.

'System' level editor has some extra features in comparison to *'Component'* level:

Copy objects:

1. Select objects
2. **[CTRL]** or **[COMMAND]**+**[C]** or Context menu → Copy
3. Click **LMB** on the area where to put the copied objects

Export/import memory (AM products):

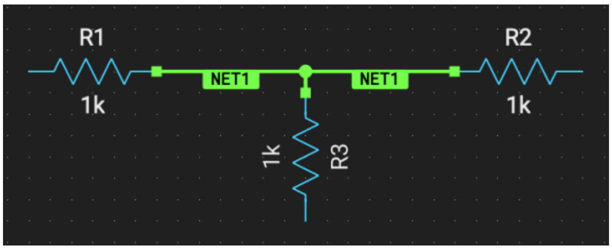
1. Call context menu on AM device → Import or Export
2. Pick file location

Connection network

Connections on schematic are grouped as networks (**connection networks**). Each *connection network* contains exclusively those connections which distribute the same signal.

Each *connection network* has unique name (e.g. NET1, … etc., by default), it can be changed via context menu of specific connection.

Wires display network name. You can enable/disable this label from context menu*:*

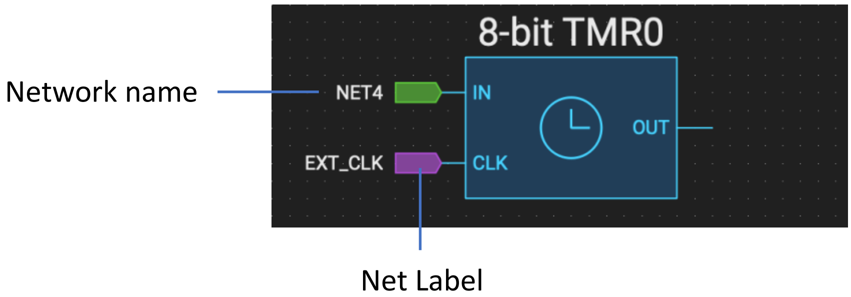


Operations supported on the Connection network:

a. Select
b. Highlight
c. Rename
d. Add/Show in Simulation Viewer

Net Labels

Net Label is an alternative form of connection representation, used to reduce schematic complexity. Any connection can be represented as a wire or *Net Label.* Corresponding controls are in a context menu of wire and *Net Label*.



Creating Net Label:

Call context menu on a wire → Turn into Net Label (or Turn network into Net Label for entire network conversion)

Secret keys:

a. Hold **[CTRL]** or **[COMMAND]** button while creating connection to add *Net Label*
b. Hold **[SHIFT]** for continuous adding

Operations supported on the *Net Labels:*

a. Convert Net Label (or entire network) to wire(s) and back
b. Change network name
c. Selection (single Net Label or network)

Memory map

Memory map (on the toolbar) is a tool to display the configuration register value of AM uASIC. Every bit corresponds to specific feature of device, can be easily navigated, and checked for current value.

Depending on the memory structure, you will see two representation types.

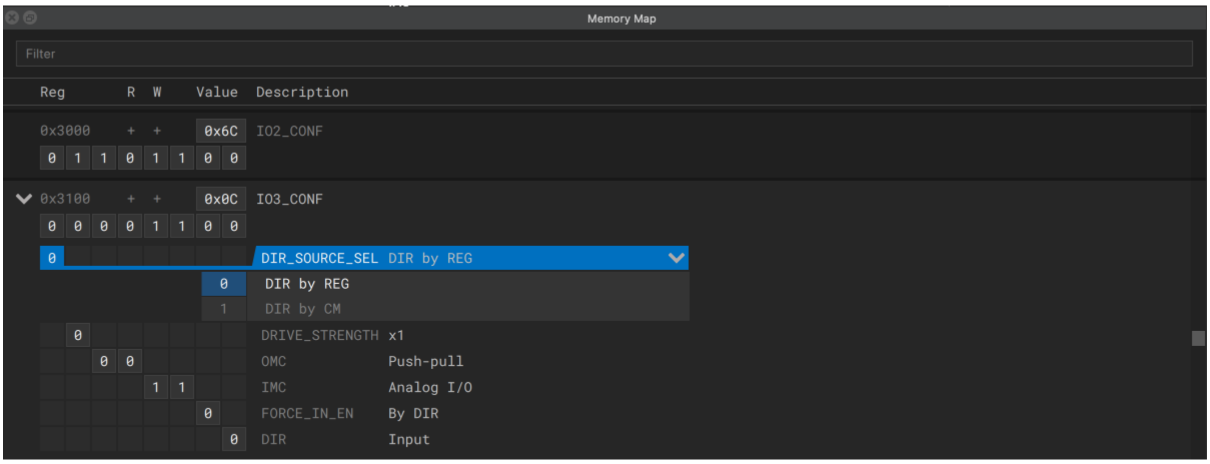
Representation A:

To see the details about specific register – click on it. In case the register is a part of multi-bit configuration, you will see the group of bits containing the one you've been interested in.

Color scheme explanation:

a. Blue: selected bit and containing multi-bit configuration
b. Grey: no changes after the last action
c. Green: value changed 0 -> 1 after the last action
d. Red: value changed 1 -> 0 after the last action

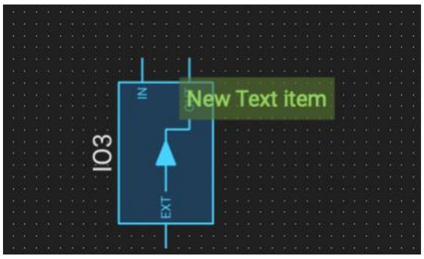
Representation B:



In this representation mode memory is arranged as bytes, where each byte can be expanded to show extra details on the properties structure and available values.

Text item

Text item can be used for adding simple comments on your schematic.

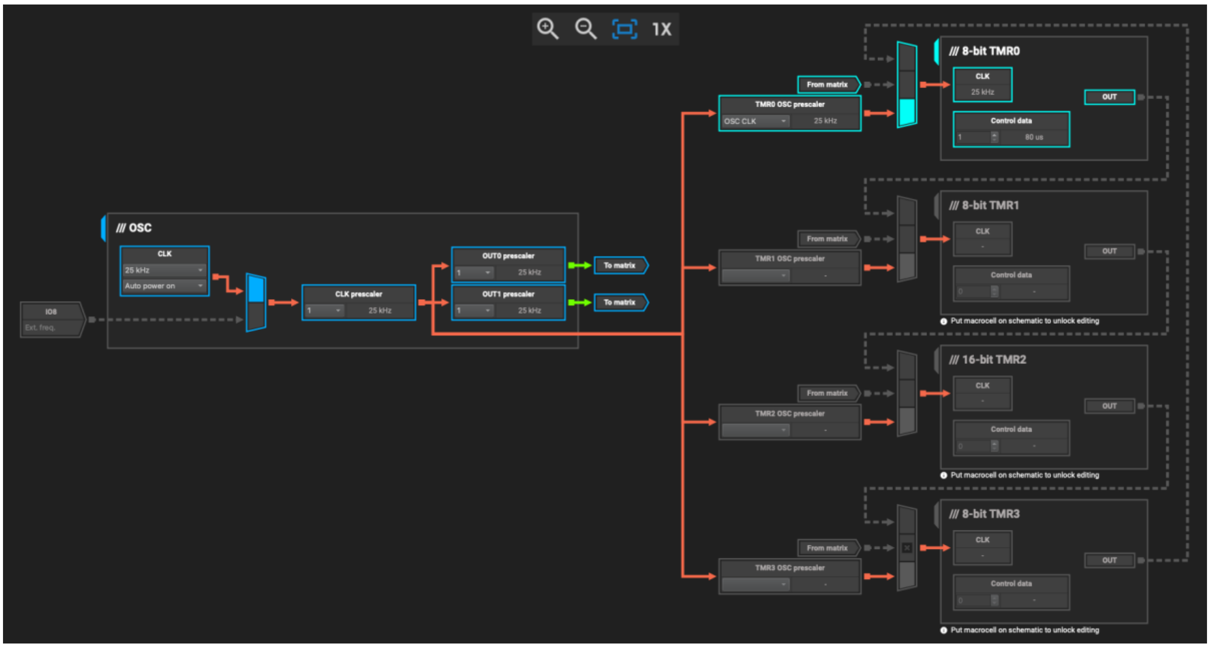


Adding new Text item:

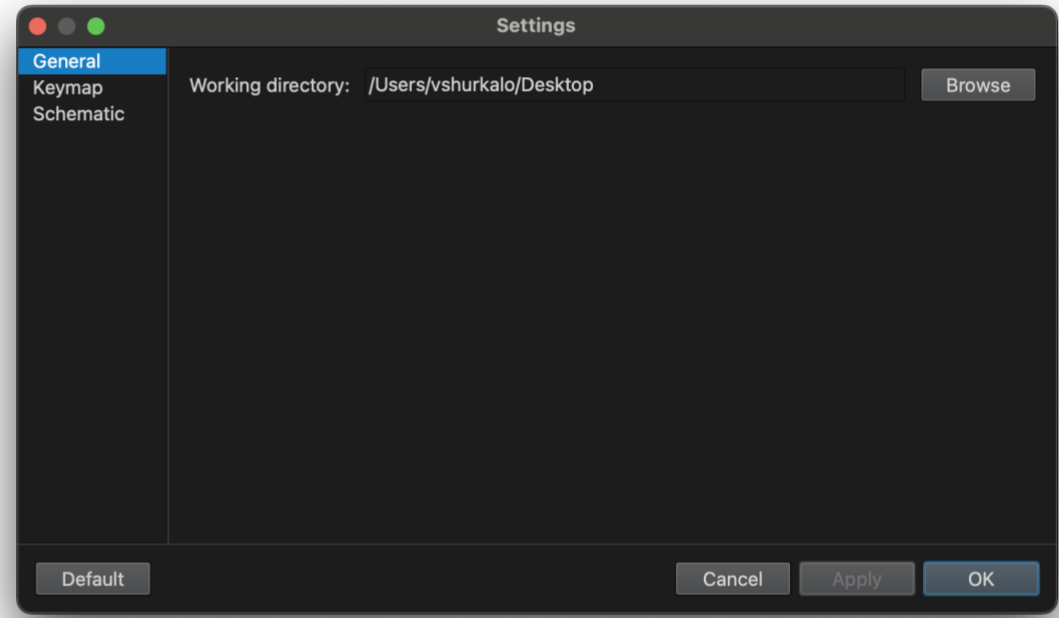
1. Click on the  button on the toolbar or **[CTRL]** / **[COMMAND]** + **[T]** on a keyboard
2. Place the mouse on specific point of your schematic and **LMB** to drop the Text item
3. (Optional) continue clicking **LMB** to continuously adding new Text items
4. (Optional) press and hold **[SHIFT]** to alternate default adding behavior – add one item and select.
    a. Default adding behavior can be redefined in Settings → Schematic (*Default object placement mode*)
5. **[ESC]** or **RMB** to finish
6. Double-click on Text item (or Context menu->Edit) to edit text

Clock Tree

Clock tree ( on the toolbar) is specific for AM uASICs tool that represents the IC's clock diagram, structure and dependencies between clock sources and synchronous macrocells, state of this subsystem, it's current configuration. It is also allows to change it's configuration.



Settings



Settings listing by page

General:

a. *Working directory:* sets default path for file open/save/import dialogs for faster navigation

Keymap:

a. *Keymap table:* set shortcuts for available actions

Schematic:

a. *Default scale:* sets default scale factor on schematic editors (defines relative scale of x1 zoom)
b. *Show grid on a schematic:* show/hide grid
c. *Grid style:* sets grid style
d. *Default connection type:*
    a. *Select automatically:* select between wire and Net Label depending on source type and/or already created connections
    b. *Wire, Net Label:* always create connection of selected type
e. *Show network name on a wires by default:* show or not the network name on a wire. Change applies only on newly added wires
f. *Show external connections on component level:* on the component level (e.g. uASIC) show text label near IOs with a name of external network, connected to it on a System level
g. *Default way of adding objects on schematic:* sets components adding style

Simulation

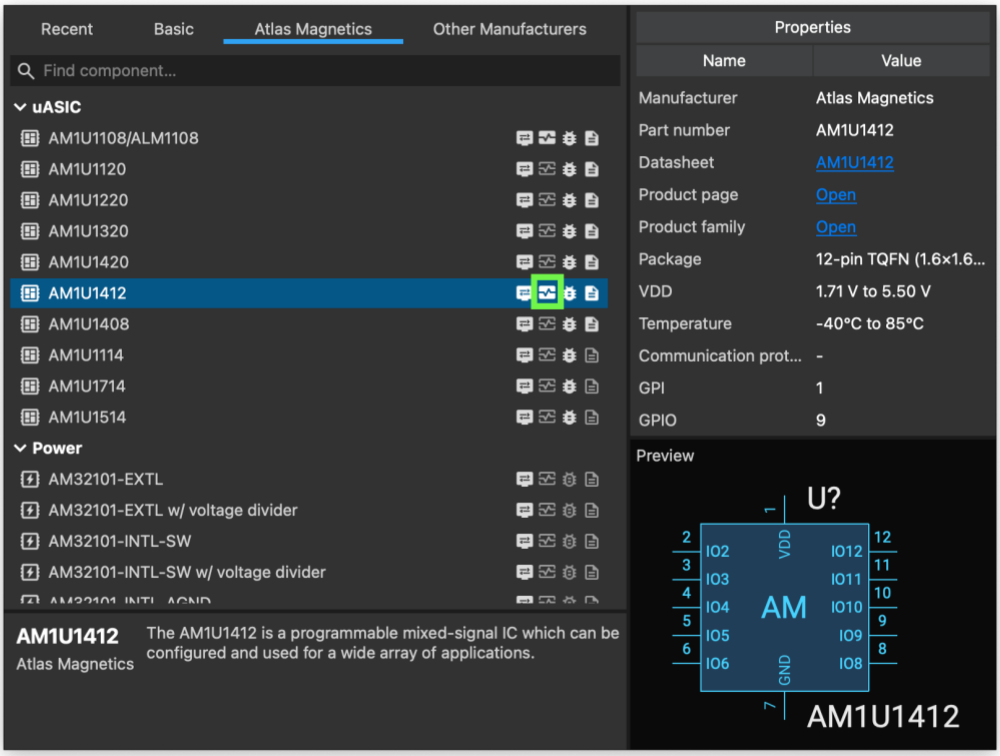
Atlas EDA incorporates NGSPICE simulation engine, which enables several analyses of your schematic: *Transient, AC, DC, Operating Point.*

Here is a step-by-step example of *Transient* analysis for simple uASIC, to guide through the main steps.

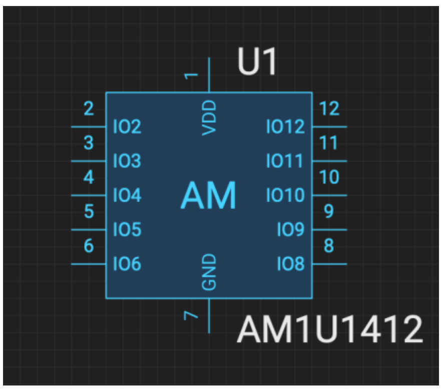
Desired function: *POWER_ON = AND (BUTTON_A, BUTTON_B)*

Let's use uASIC for logic implementation, which for sure would be an overkill, but why not.

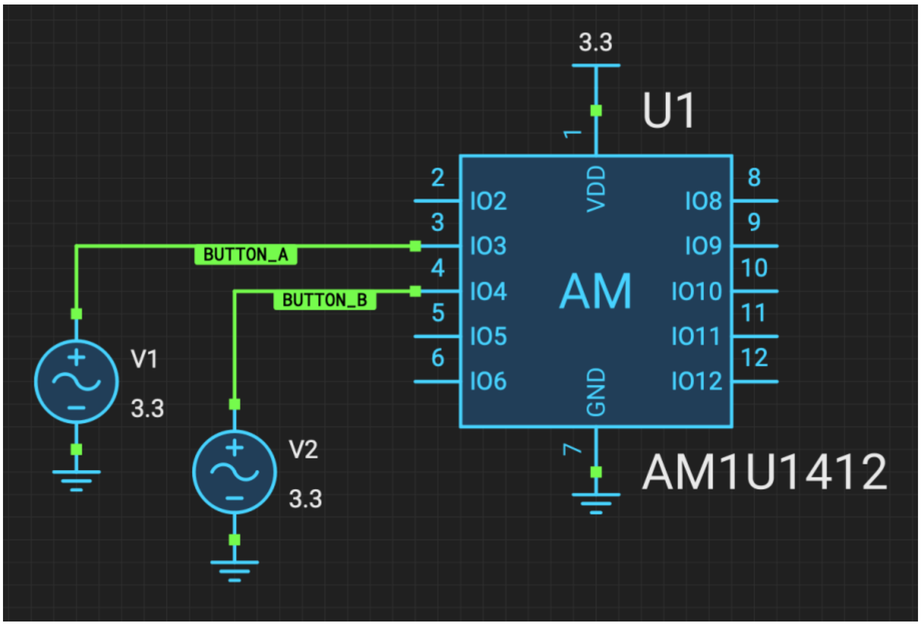
For successful simulation we have to make sure all the components we are going to use in schematic have simulation models. Let's start with uASIC. We're looking for the uASIC with enabled Simulation icon, like AM1U1412:



Placing uASIC on the schematic:

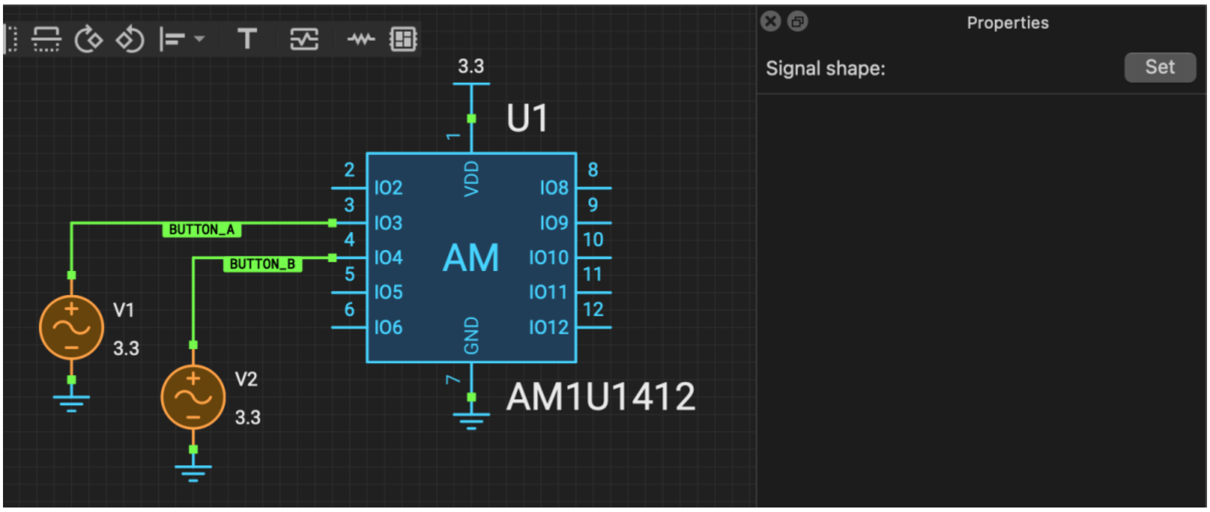


Let's use *IO3* as the *BUTTON_A* input, *IO4* as the *BUTTON_B* input, *IO10* as the *RESULT* output:



Basic components, like *Voltage source, VDD, GND* can be found in the *Library* in Basic category. When connection is done, let's configure the input signals by setting waveforms.

To see both V1 and V2 waveforms at the same time – select both components and click on *Set* button on the *Properties* panel:

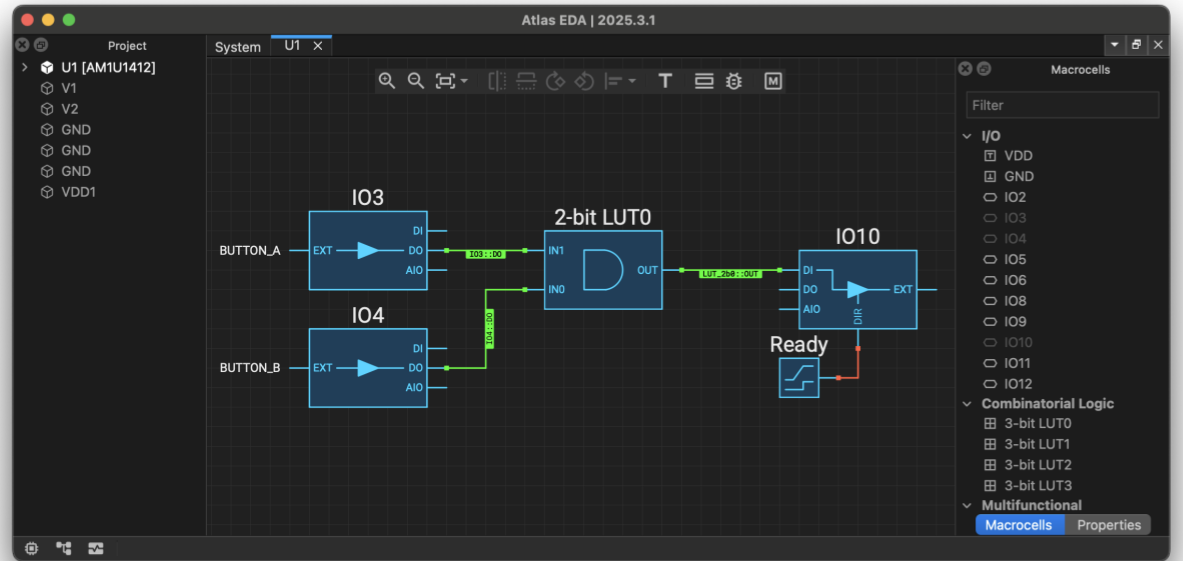


Let's set both waveforms *Type* as a *Pulse train*, defining the following shapes:

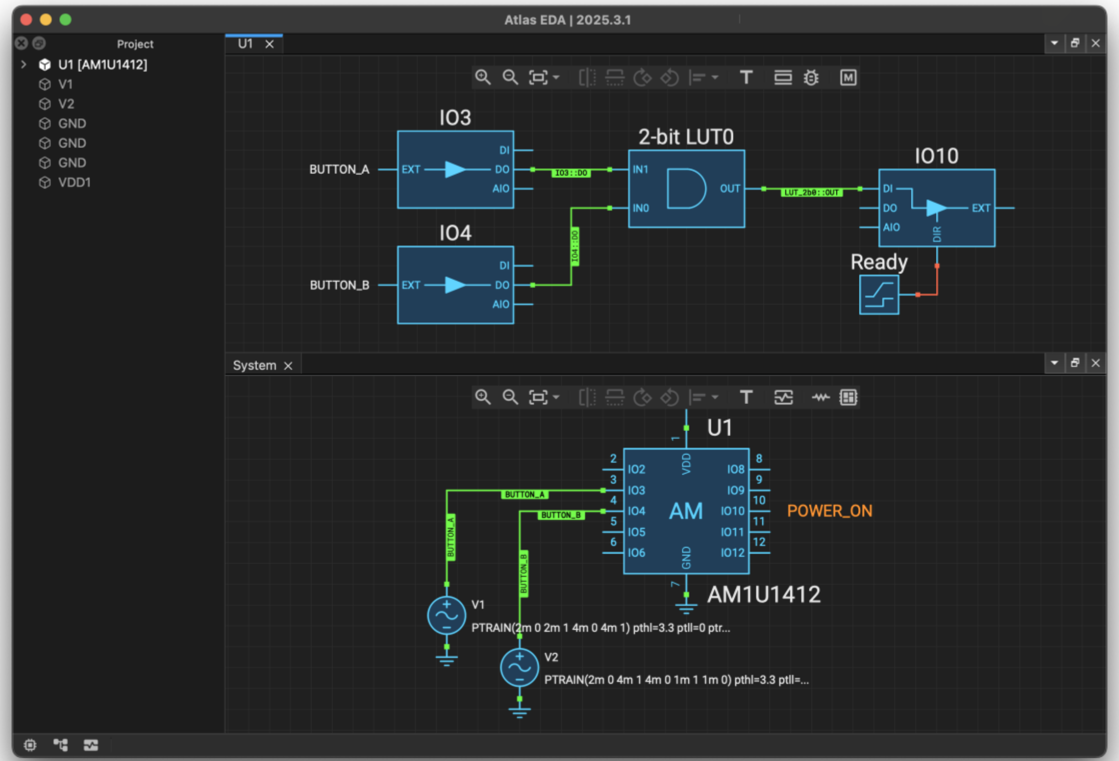
Next important step is to configure the desired logic inside uASIC. So, we need to open uASIC schematic editor (double click on the U1). Remember, the desired function should translate to:

IO10 = AND (IO3, IO4)

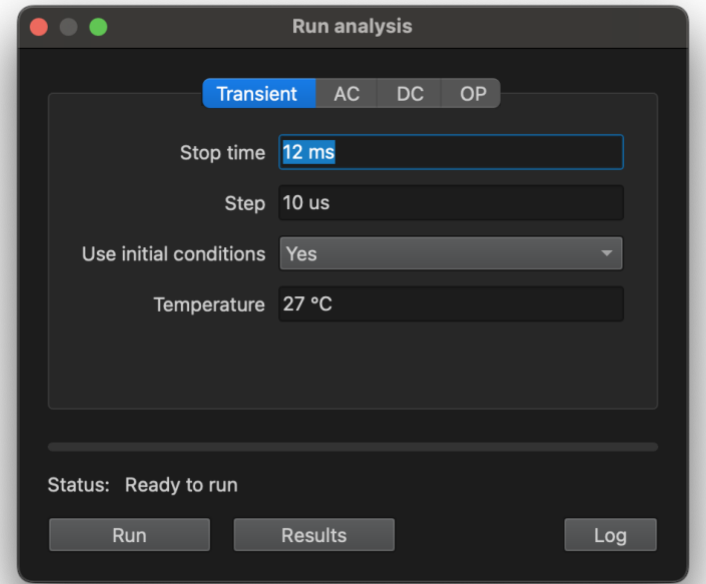
Let's make corresponding schematic:



Let's review entire solution:

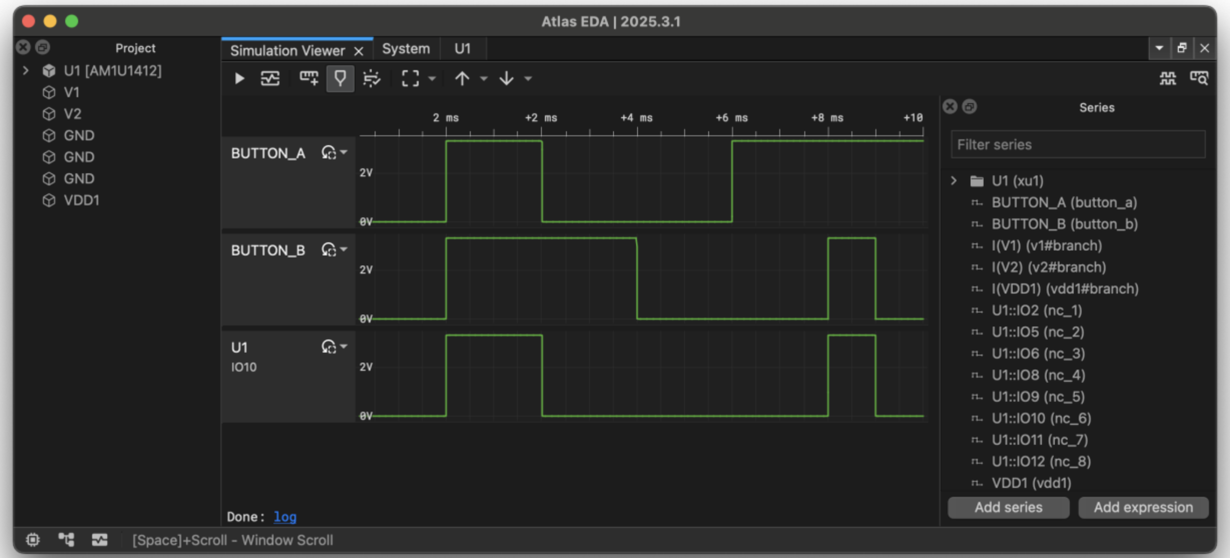


Everything looks good. Let's run the simulation. Open Analysis dialog (). Select Transient analysis, and set *Stop time* to cover desired timeframe (in our example – 12ms):

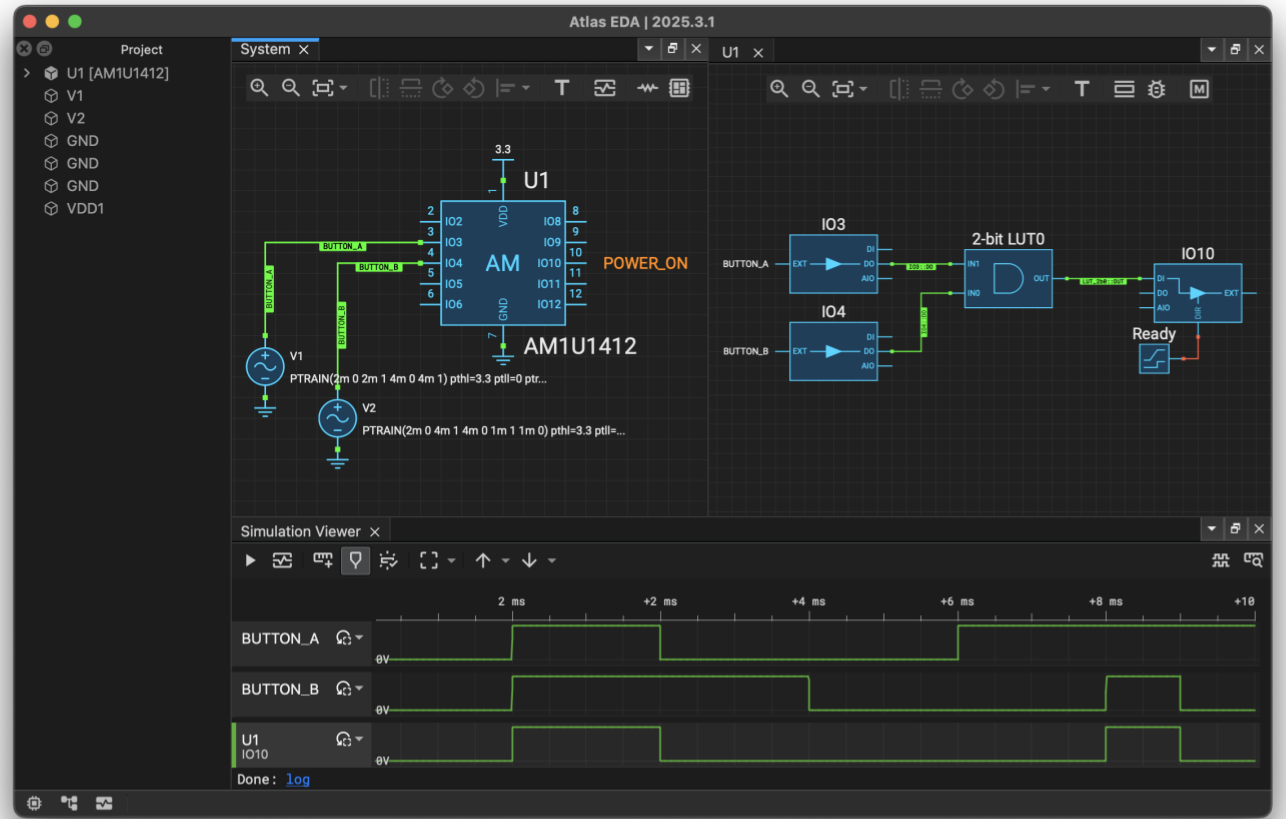


Click *Run* button to start simulation. As soon as the simulation is started, the simulation results window will appear. You can see the simulation data arriving in real time.

Let's put the following waveforms on screen to check the result: *BUTTON_A, BUTTON_B, U1::IO10.*



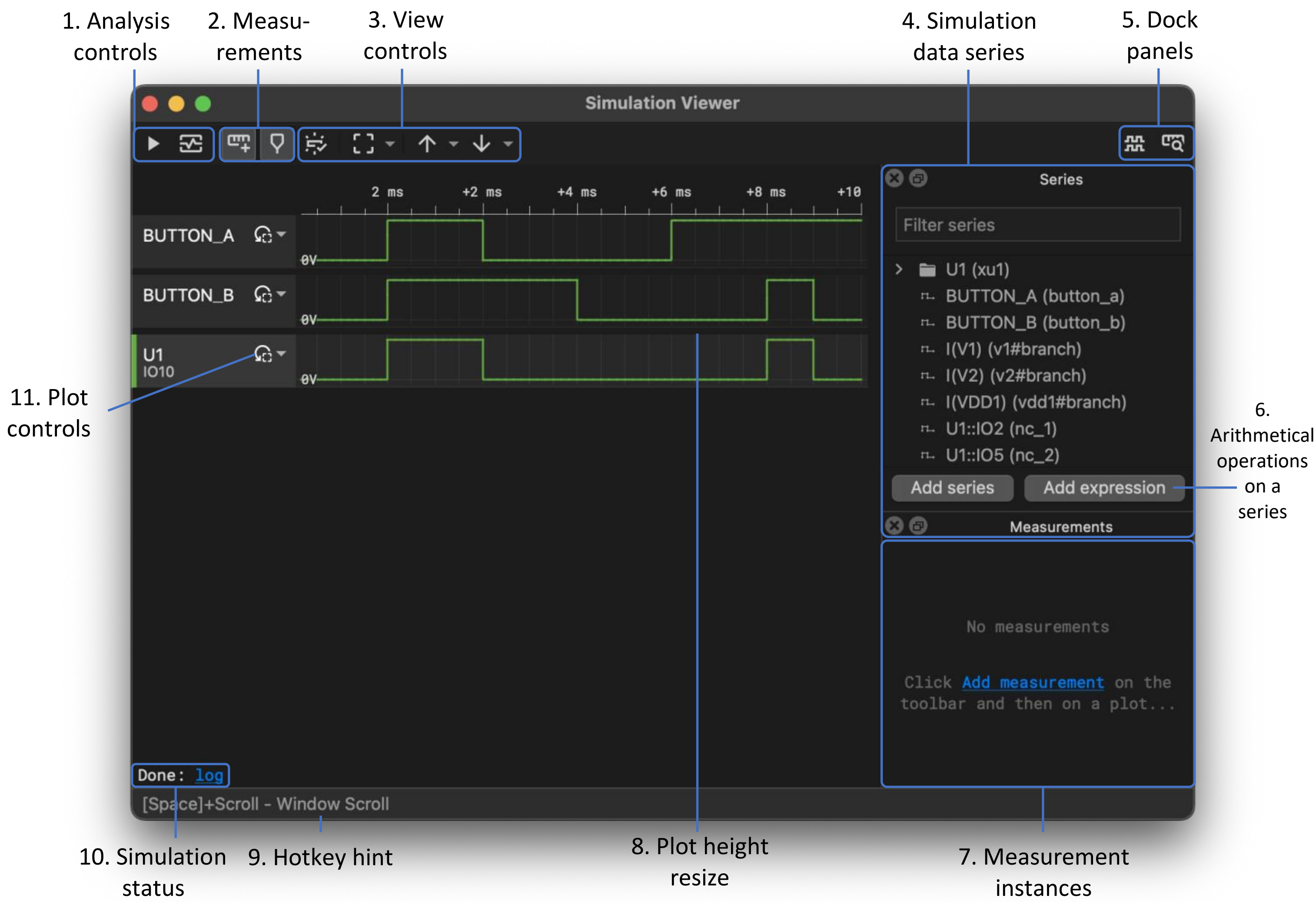
We can also rearrange the window layout to make System, U1 and Simulation Viewer tabs be visible at the same time, to review entire solution:



Simulations Viewer

Simulation Viewer gathers and displays the results from the most recent simulation analysis.

In many ways *Simulation Viewer* has similar controls with other waveform managers across the application, like *Source Editor* in *Hardware Debug* and *Source Editor* in *Simulation.*



Once the simulation is started – *Simulation Viewer* pops up, showing the simulation progress (**Simulation status (10)**) and simulation data collection (**Simulation data series (4)**).

Simulation can be interrupted using Stop button, or the *Analysis window* can be recalled (for instance to look at the NGSPICE output in the Log), using **Analysis controls (1)**.

While the simulation is still running, you can start adding data series to the view area. *You will see how data points arriving in real time.* This allows to make decision either to stop or continue simulation as early as possible.

To add data series to the view area – find specific series in the **Simulation data series (4)** window, double-click (or select and click *Add series* button).

Scrolling, scaling, resizing waveforms

! Window scroll is working only when you press and hold [SPACE] button.

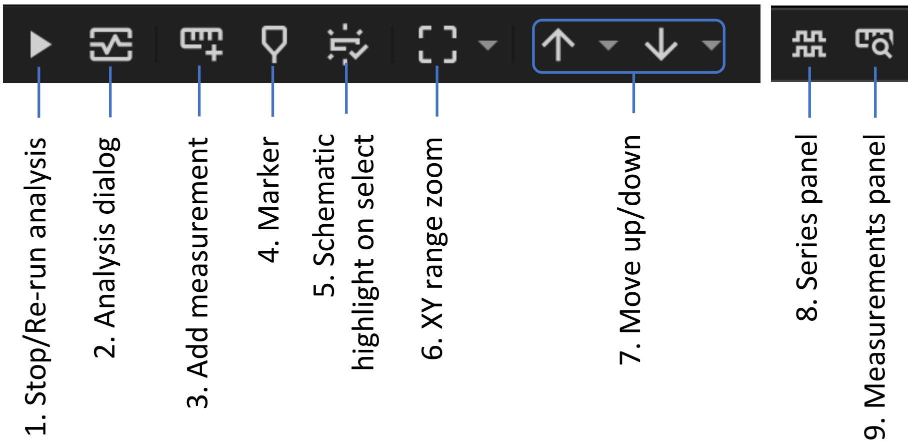
This limitation was made intentionally to avoid interference of window vertical scrolling and waveform X zoom. To have a reminder on this – please look at the **Hotkey hint (9)**, *which is listening to key modifiers like [SHIFT], [CTRL]/[COMMAND], [ALT]/[OPTION], to suggest available hotkeys.*

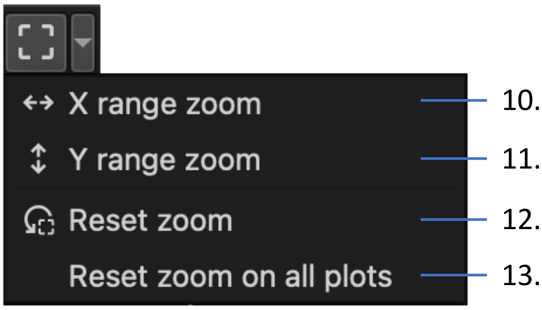
Here is the list of supported waveform controls:

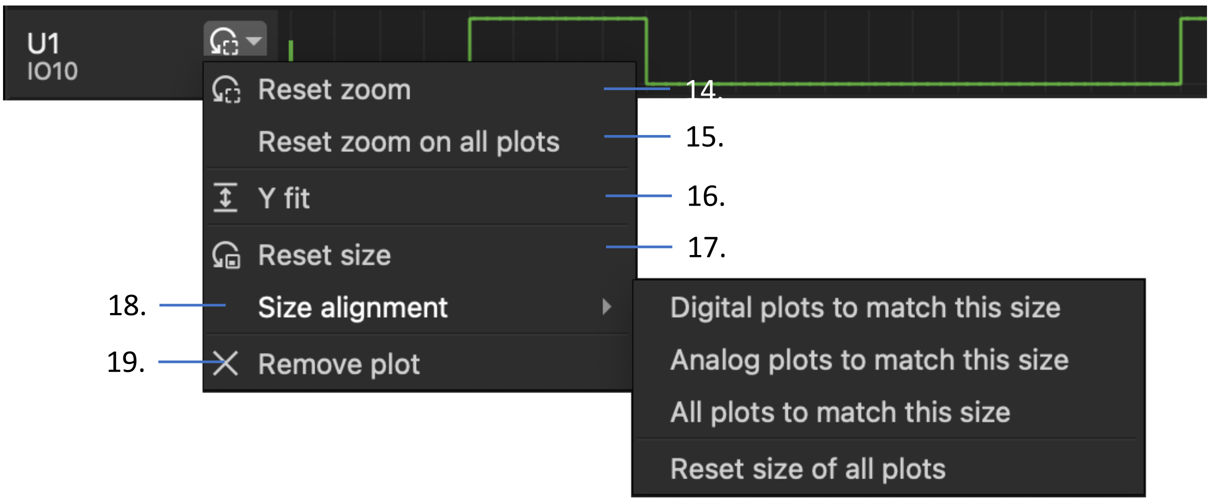
ACTION	MOUSE	TOUCHPAD
	SCROLL	
X SCROLL	Drag-n-drop	Horizontal scroll
Y SCROLL	Drag-n-drop	
	ZOOM	
X ZOOM	Wheel	Vertical scroll
	[SHIFT] + Wheel	[SHIFT] + Vertical scroll
		[SHIFT] + Horizontal scroll
Y ZOOM	[CTRL]/[CMD] + Wheel	[CTRL]/[CMD] + Vertical scroll
		[CTRL]/[CMD] + Horizontal scroll
XY ZOOM	[CTRL]/[CMD] + [SHIFT] + Wheel	[CTRL]/[CMD] + [SHIFT] + Vertical scroll
		[CTRL]/[CMD] + [SHIFT] + Horizontal scroll
	RANGE ZOOM	
X RANGE ZOOM	[SHIFT] + Rect selection	
Y RANGE ZOOM	[CTRL]/[CMD] + Rect selection	
XY RANGE ZOOM	[ALT]/[OPTION] + Rect selection	
	[CTRL]/[CMD] + [SHIFT] + Rect selection	
RESET ZOOM	Range zoom rect from right to left	

Note: notation like “[CTRL]/[CMD]” means [CTRL] for Windows, [CMD] for macOS.

Toolbar and plot actions





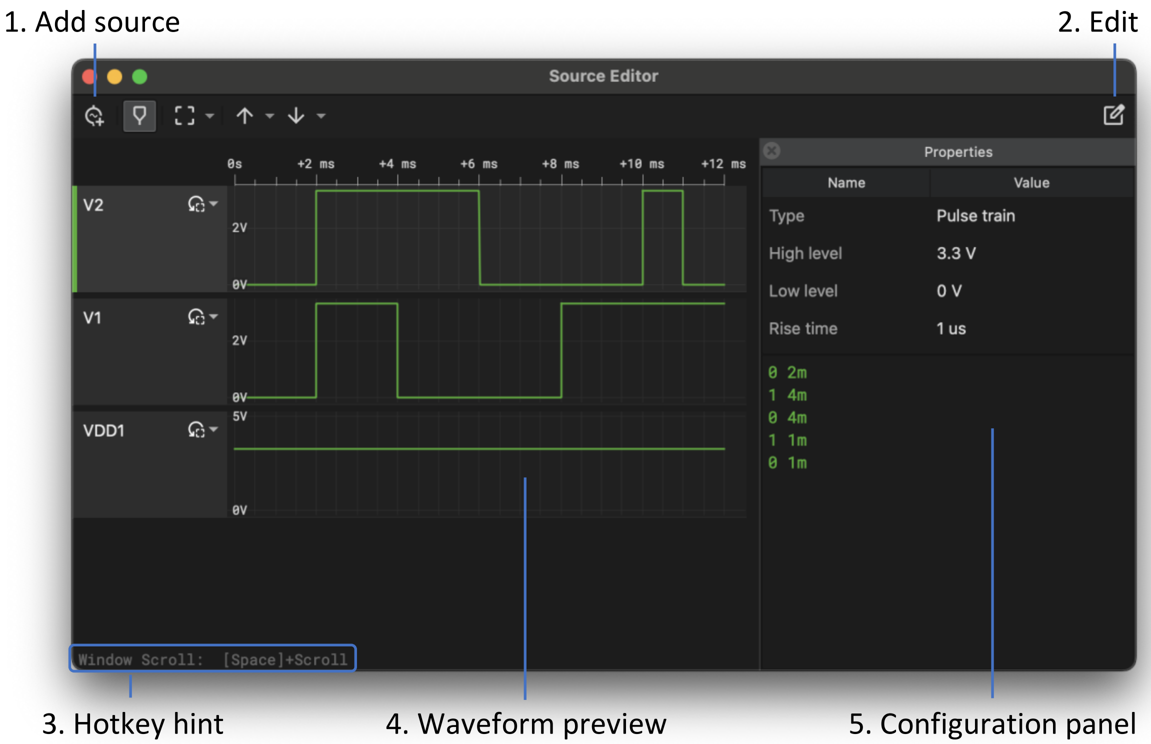


Description:

a. **Stop/Re-run analysis (1)** – stop the simulation or restarts simulation with previous configuration
b. **Analysis dialog (2)** – open analysis dialog
c. **Add measurement (3)** – click on the action and then on the waveform to set beginning and ending of a measurement box. Measurement item will appear in Measurements panel
d. **Marker (4)** – on/off marker. Maker hints the waveform value in particular point

e. **Schematic highlight on select (5)** – on/off automatic highlight of source schematic for selected plot. Turn it on, select plot and look on schematic – source network will be highlighted
f. **XY range zoom (6)** (action with menu) – activates XY range zoom mode, when you can press on the plot and drag a rect area to select which area you would like to scale.
    i. **X range zoom (10)** – does only X zoom, keeping Y zoom the same
    ii. **Y range zoom (11)** – does only Y zoom, keeping X zoom the same
    iii. **Note:** range zoom works when you drag the rect from left to right. If you drag from right to left – zoom level resets
g. **Move up/down (7)** (actions with submenu) – move selected waveform up or down
h. **Series (8)** – open Series panel
i. **Measurements (9)** – open Measurements panel
j. **Reset zoom (12, 14)** – reset zoom on selected waveform
k. **Y fit (16)** – set Y scale to fit a visible at the current moment part of a waveform
l. **Reset size (17)** – reset plot height to the default one
m. **Size alignment (18)** – set of options to align plot size
n. **Remove plot (19)** – remove selected plot from the view

Source Editor (Simulation)



Source Editor's main function is to provide visual representation of selected source waveforms on the single timeline with ability to change their configuration.

Toolbar description:

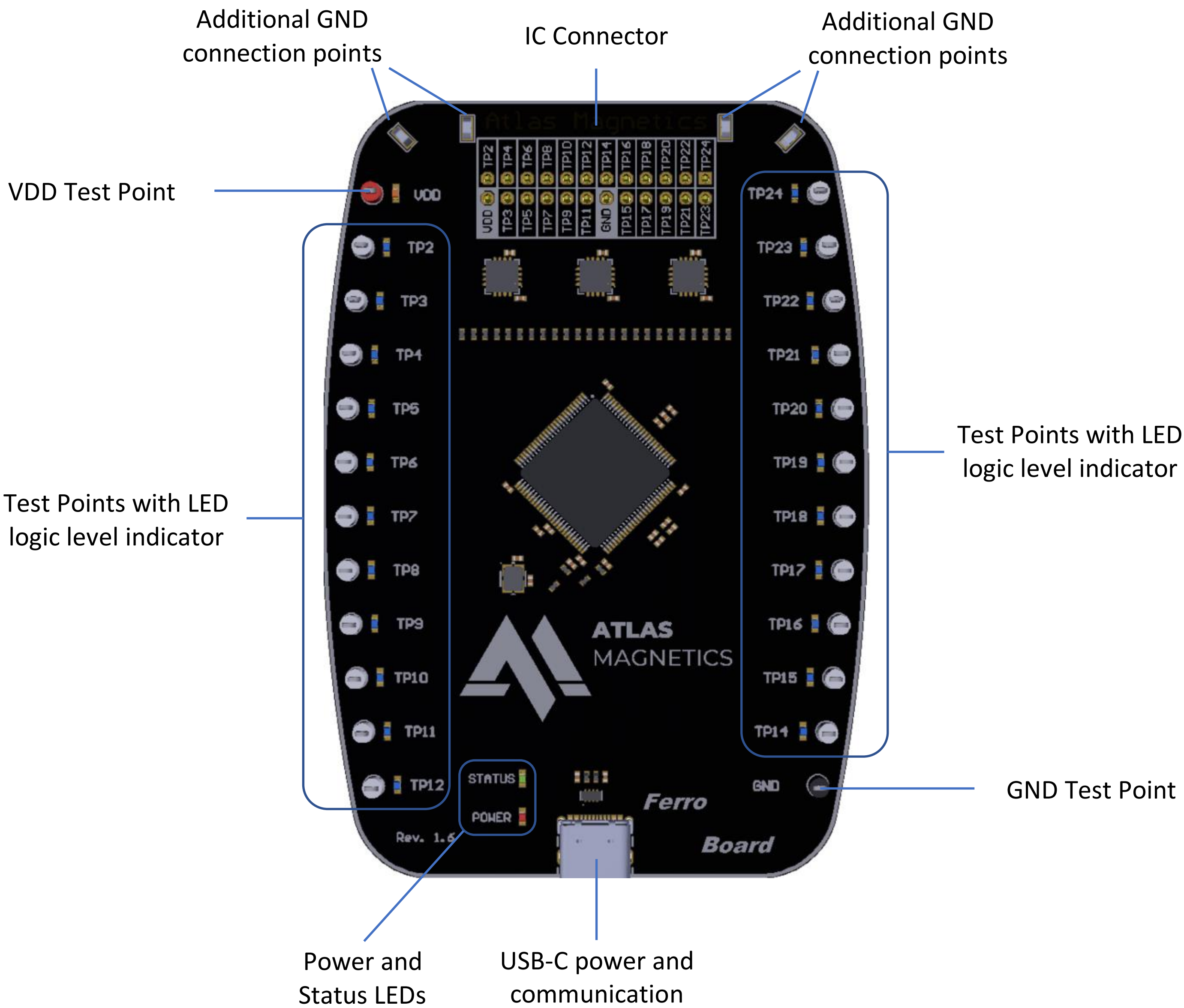
a. **Add source (1)** – shows a list of voltage and current sources, used on the schematic, giving you the option to add one to the view for configuration

b. **Edit (2)** – opens configuration panel (or double-click **LMB** on the **waveform preview (4)**)
c. **Note:** rest of the controls operate in the same way it is described in *Simulation Viewer*

Hardware Debug. Ferro Board

Hardware debug ( on the toolbar) is another specific for AM uASICs tool, allowing to debug and evaluate your project using real IC. Special hardware required – Ferro Board – it should be connected to your computer with corresponding IC inside the Socket Board.

Here is the Ferro Board hardware overview.

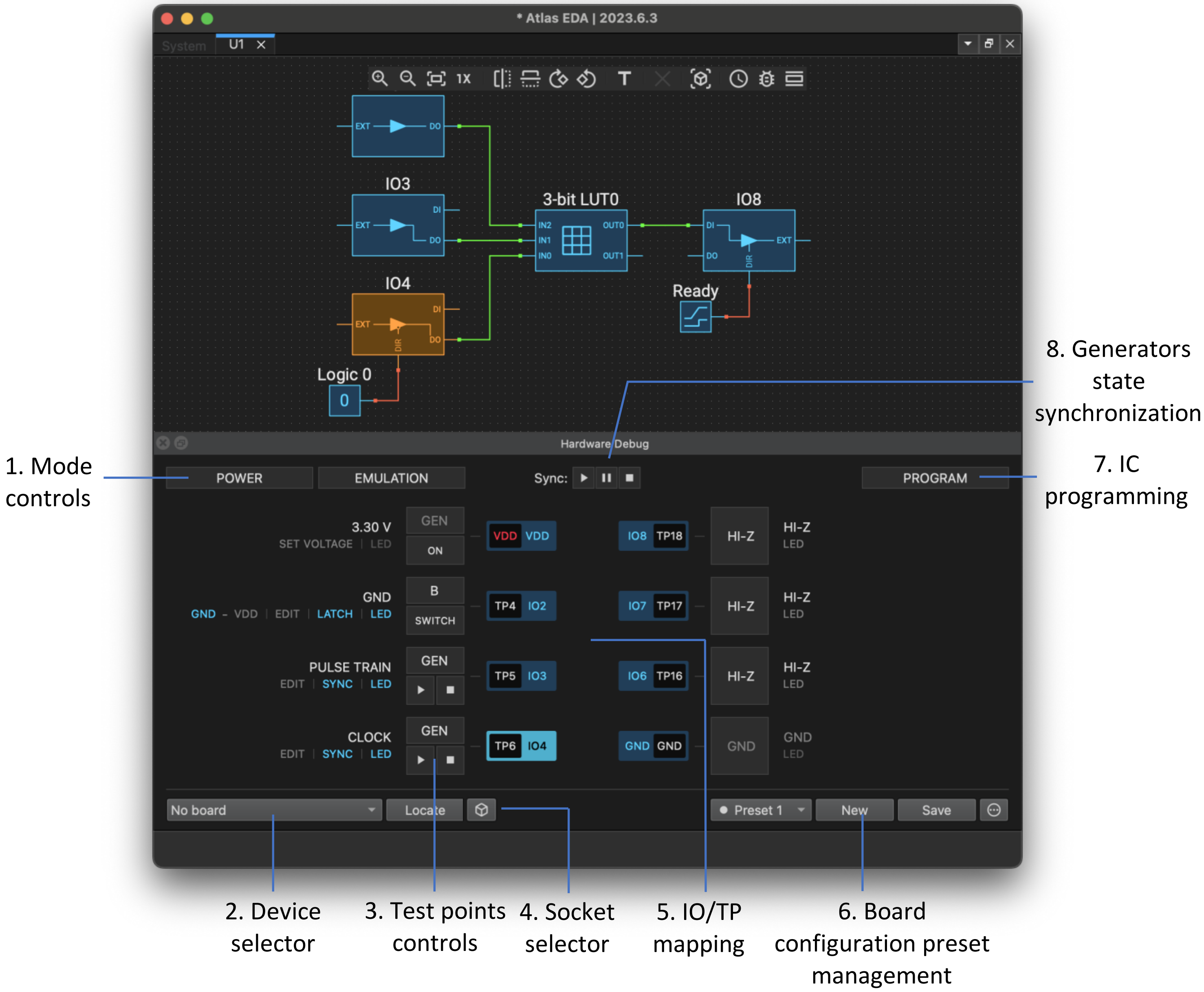


Here are a couple important things to know before you start:

a. Before applying an external signal to the test point, put it in a high-impedance state
b. The external signal applied to the test point should not have a voltage higher than the chip's power supply voltage
c. External chip power cannot be applied through the VDD test point

d. Applying external power to the chip while VDD or discharge is on can cause the chip and the Ferro Board to fail
e. Interrupting the firmware update process may cause the motherboard to fail
f. Physical damage to the Ferro Board or its components can cause it to fail
g. Direct contact of electrically conductive liquids on the ferro board can lead to its failure

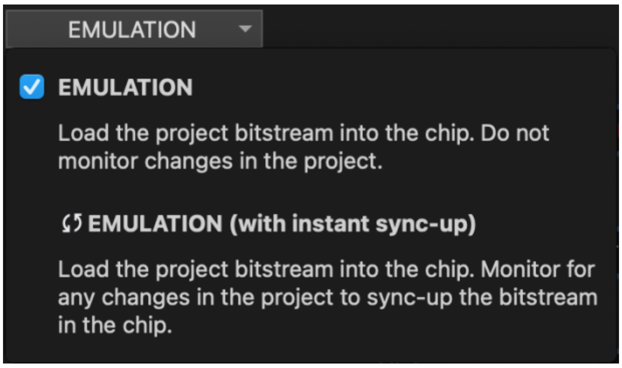
Software GUI controls overview.



Before starting any debug operations, please make sure you've your device connected with proper IC inside Socket Board. In case multiple devices connected to your computer – use **Device selector (2)** to connect with correct one. Make sure to select proper socket board using **Socket selector (4).** Button **Locate** will blink the LED on the selected board, so you can visually identify it.

Emulation mode will load the IC with your current bitstream, allowing to test it without programming the part. Please keep in mind that emulated bitstream is kept in the IC memory while it is powered on.

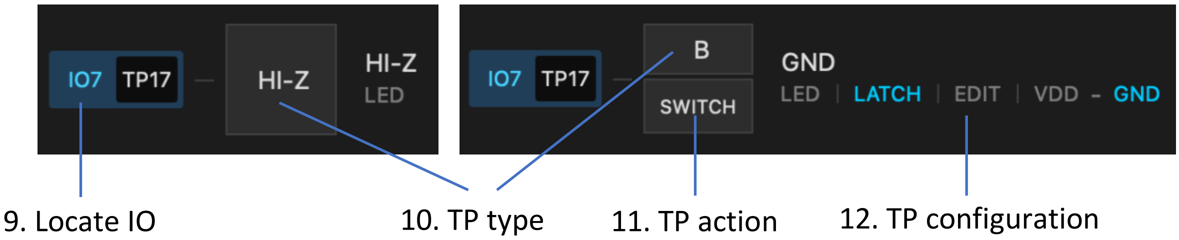
Some ICs can update their bitstream during emulation. This function is incorporated within **Emulation** modes, available for selection in the dropdown menu of the Emulation button:



Once the **Emulation with instant sync-up** is enabled and **Emulation** is running – each change on the schematic will be immediately transferred to corresponding IC, updating it's bitstream.

Use **Test points controls (3)** to configure preferred signal which should be applied to specific test point and as the result – applied to specific IO of your IC.

To select signal type – click on **test point type button (10)** and select preferred one from the list.



Depending on type, test point can support specific configuration. Use buttons in **TP configuration (12)** section to setup the one you need.

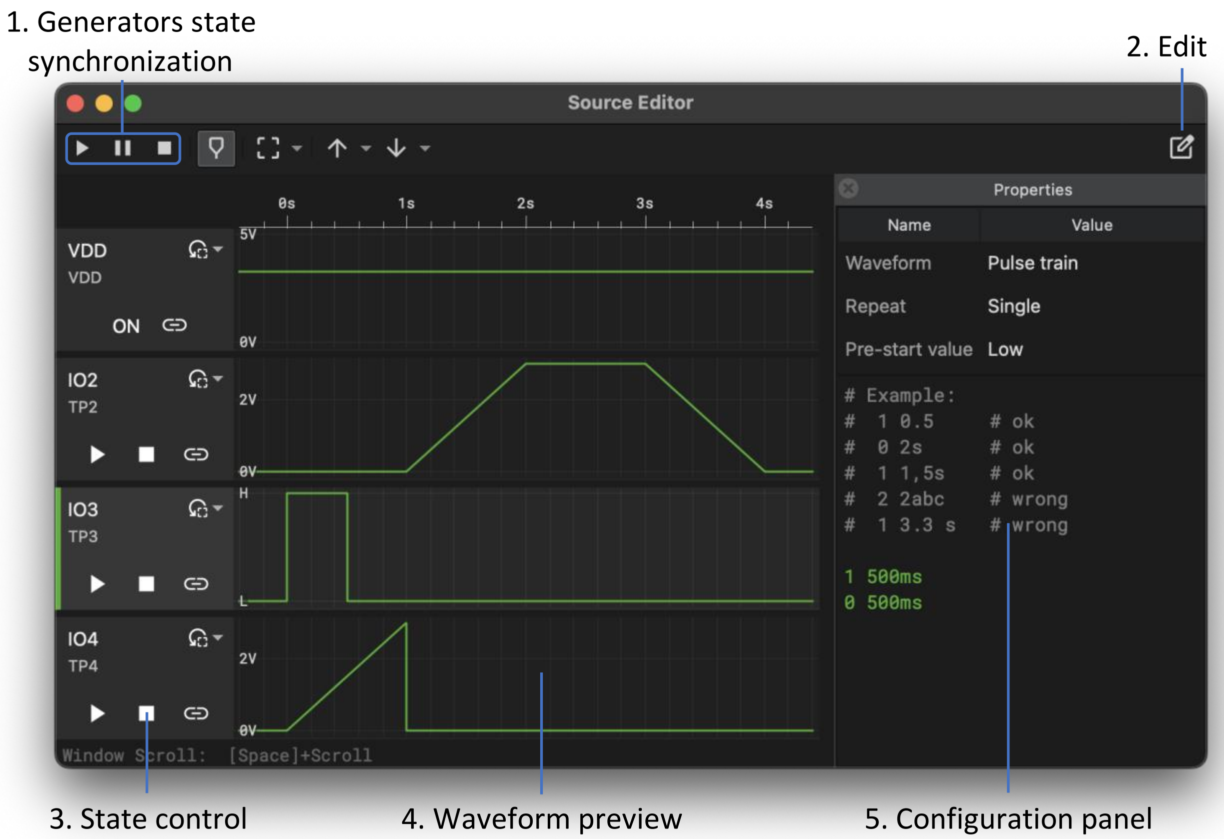
Another type-depended control is **TP action button(s) (11)**, which usually represent one or few main actions supported by selected type of TP. For instance, On/Off state for Constant voltage generator, Switch – for Button or Start/Pause/Stop for waveform generators.

Waveform generators support state synchronization (see **Generators state synchronization (8)**), which allow synchronous start/pause/stop generators which have **SYNC** option enabled on their **TP configuration (12)** section. For more information about waveform editing, please refer to the **Source Editor**.

If board is Powered On (see **Mode Controls (1)**), every state change you do immediately reflects in the Ferro Board configuration.

There is also feature to save the board configuration as a preset (**Board configuration preset management (7)**) and then restore it any time you need, picking it from the list of saved presets. Preset is basically a snapshot of current configuration.

Source Editor (Hardware Debug)



Source Editor's (Hardware Debug) main function is basically the same as for the Simulation's *Source Editor* analogue – provide visual representation of the source waveforms on the single timeline with ability to change the configuration of any represented waveform.

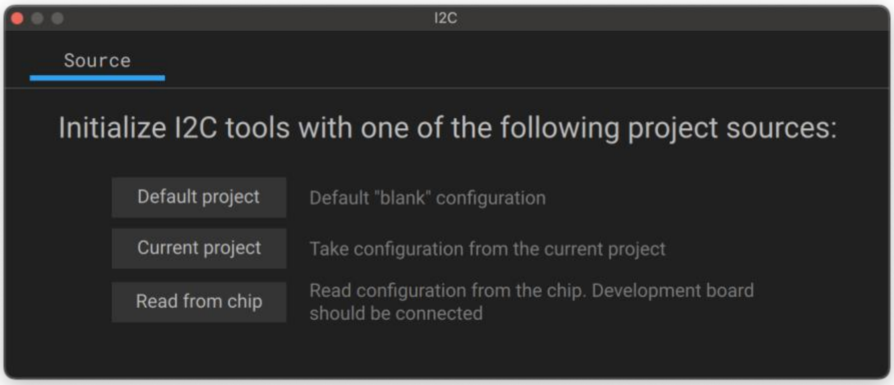
Toolbar description:

d. **Generators state synchronization (1)** – starts/pauses/stops all generators which have SYNC option enable
e. **Edit (2)** – opens configuration panel (or double-click **LMB** on the **waveform preview (4)**)
f. **Note:** rest of the controls operate in the same way it is described in *Simulation Viewer*

I2C tools

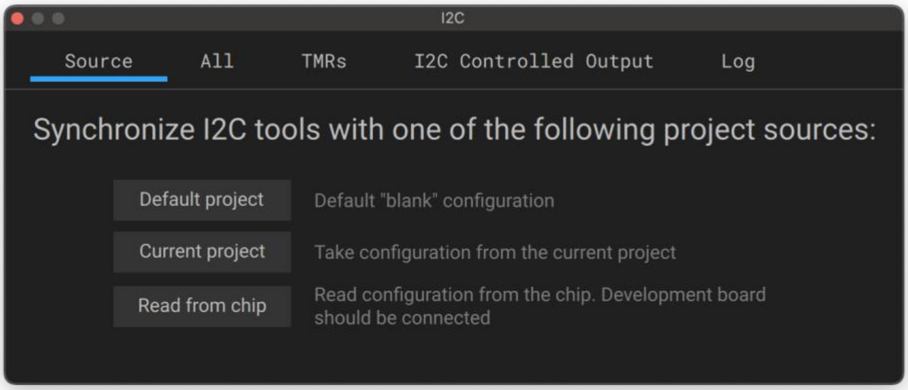
uASICs with I2C interface can benefit from the flexibility of changing and/or reading register values while chip is operating. Atlas EDA provides I2C tools to handle I2C communication. You can find **I2C** button in **Hardware Debug** window if specific uASIC supports it.

On initial start of the I2C tools you will be asked to initialize them with the sequence on your choice.



Why do you need that step? You can think about I2C tools as a data buffer. You change something, then you write the change to the chip. You can write the whole data buffer to the chip as well. So, if your initial data is blank and you write it all – no surprise the chip gets "cleared". This make the initial data so important.

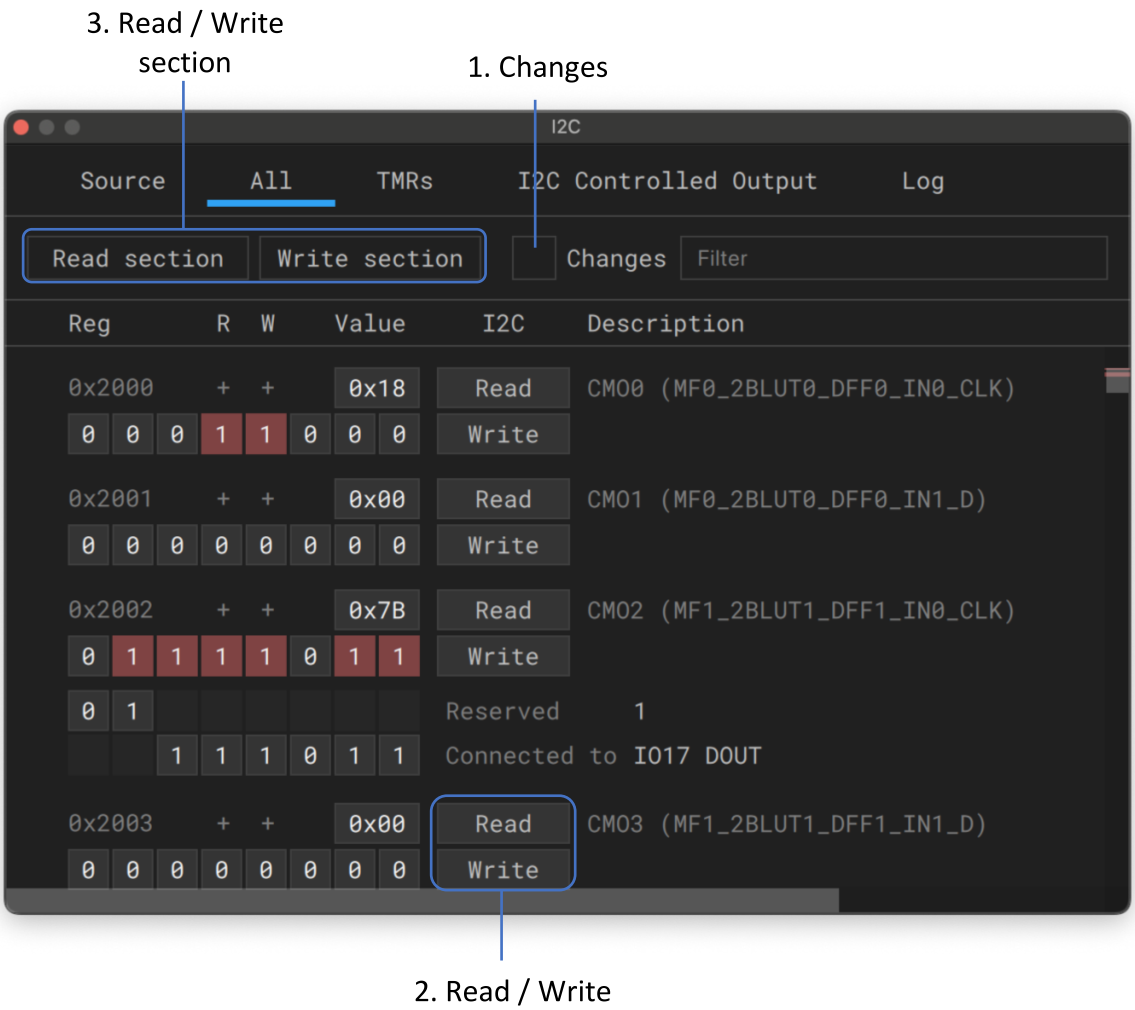
Once initialized, you can always go back and sync the data again with another source.



After initialization, you also will be able to see the data tabs, like: All, TMRs, I2C Controlled Output, etc. Their goal is to provide convenient representation of the chip memory whether you want to see the whole memory or for instance only TMR macrocells;

Data editing and transactions

Memory representation is very similar to Memory map, except here you can edit register values by clicking on the value you want to change.



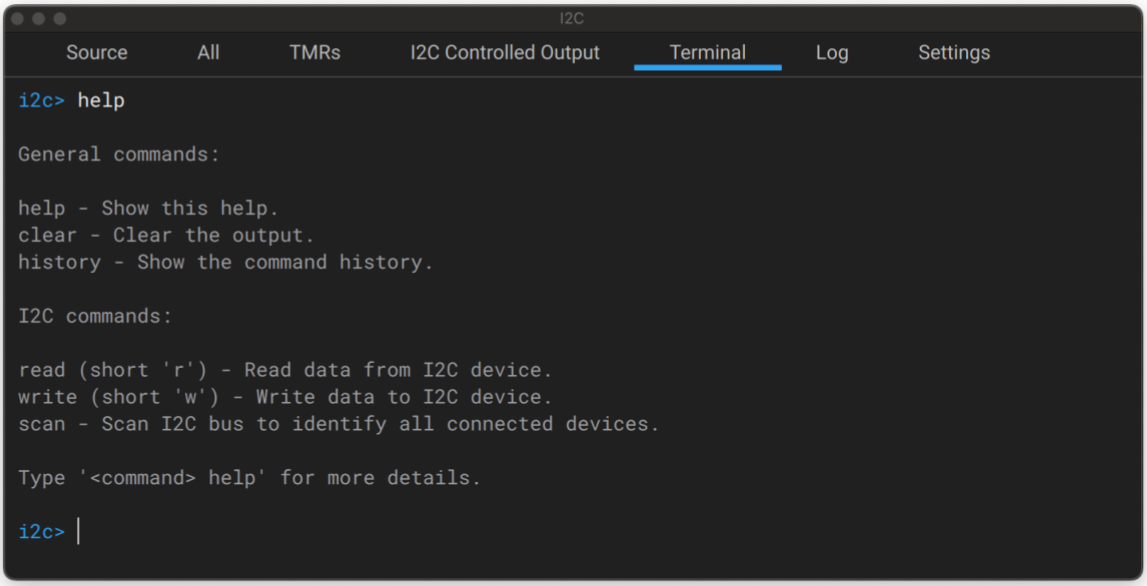
Main controls:

a. **Changes (1)** checkbox will hide all unchanged bytes from the screen;
b. **Read / Write (2)** buttons do the corresponding transaction with specific byte;
c. **Read / Write section (3)** do the corresponding transaction with all visible bytes on the tab;

Terminal

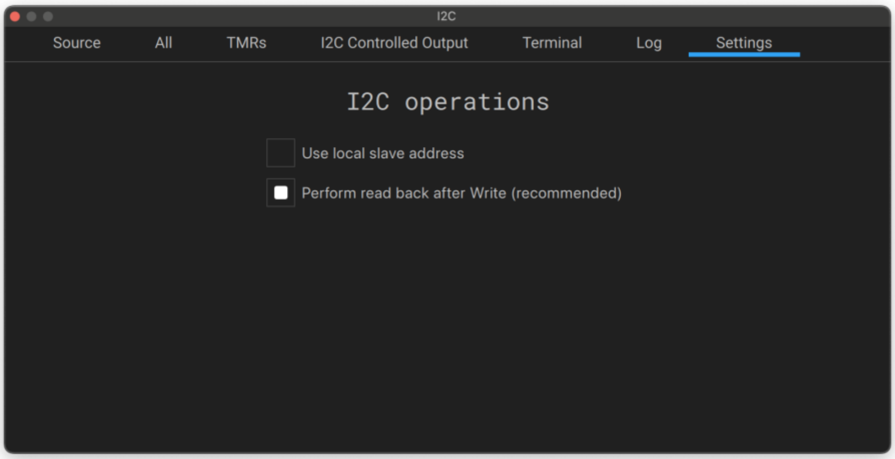
Terminal allows to execute a command prompt.

Open **Terminal** tab and enter "help" command to see the list of supported commands and their usage. To see more details on specific command, enter "<command_name> help" (e.g. "read help").

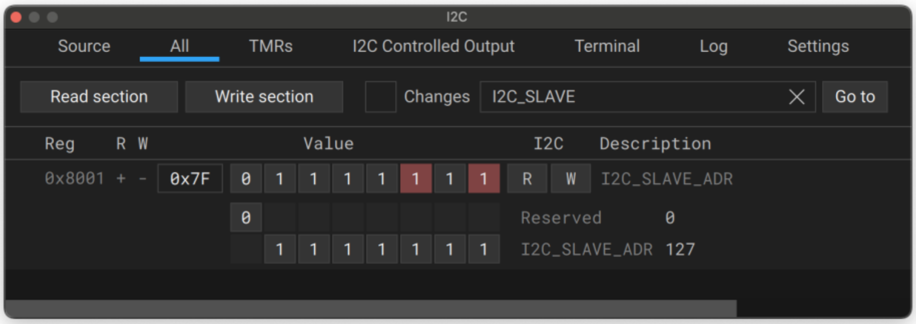


Settings

Settings tab contains I2C related options.



Use local slave address: forces I2C tool to use I2C slave address which is set in bitstream (check out I2C_SLAVE_ADR on the tab All) instead of scanning I2C bus to detect the slave address.



Perform read back after Write: proceed with I2C read after write command to check what has written. It is recommended to keep this option enabled unless you intentionally want to clean up the I2C bus transactions as much as possible for debug purpose.

Legal Statement and Contact information

1. ATLAS MAGNETICS (AM) AND ITS SUBSIDIARIES MAKE NO WARRANTY OF ANY KIND, EXPRESS OR IMPLIED, WITH REGARDS TO ANY INFORMATION CONTAINED IN THIS DOCUMENT, INCLUDING, BUT NOT LIMITED TO, THE IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT OF THIRD PARTY INTELLECTUAL PROPERTY RIGHTS (AND THEIR EQUIVALENTS UNDER THE LAWS OF ANY JURISDICTION).

2. The Information contained herein is for informational purpose only and is provided only to illustrate the operation of AM's products described herein and application examples. AM does not assume any liability arising out of the application or use of this document or any product described herein. This document is intended for skilled and technically trained engineering customers and users who design with AM's products. AM's products may be used to facilitate safety-related applications; however, in all instances customers and users are responsible for (a) selecting the appropriate AM products for their applications, (b) evaluating the suitability of AM's products for their intended applications, (c) ensuring their applications, which incorporate AM's products, comply the applicable legal and regulatory requirements as well as safety and functional-safety related standards, and (d) ensuring they design with appropriate safeguards (including testing, validation, quality control techniques, redundancy, malfunction prevention, and appropriate treatment for aging degradation) to minimize the risks associated with their applications.

3. AM assumes no liability for any application-related information, support, assistance or feedback that may be provided by AM from time to time. Any customer or user of this document or products described herein will assume all risks and liabilities associated with such use, and will hold AM and all companies whose products are represented herein or on AM's websites, harmless against all damages and liabilities.

4. Products described herein may be covered by one or more United States, international or foreign patents and pending patent applications. Product names and markings noted herein may also be covered by one or more United States, international or foreign trademarks and trademark applications. AM does not convey any license under any of its intellectual property rights or the rights of any third parties (including third parties whose products and services may be described in this document or on AM's website) under this document.

5. AM does not warrant or accept any liability whatsoever in respect of any products purchased through unauthorized sales channel.

6. AM's products and technology may not be used for or incorporated into any products or systems whose manufacture, use or sale is prohibited under any applicable laws and regulations. Should customers or users use AM's products in contravention of any applicable laws or regulations, or for any unintended or unauthorized application, customers and users will (a) be solely responsible for any damages, losses or penalties arising in connection therewith or as a result thereof, and (b) indemnify and hold AM and its representatives and agents harmless against any and all claims, damages, expenses, and attorney fees arising out of, directly or indirectly, any claim relating to any noncompliance with the applicable laws and regulations, as well as any unintended or unauthorized application.

7. While efforts have been made to ensure the information contained in this document is accurate, complete and current, it may contain technical inaccuracies, omissions and typographical errors. AM does not warrant that information contained in this document is error-free and AM is under no obligation to update or otherwise correct this information. Notwithstanding the foregoing, AM reserves the right to make modifications, enhancements, improvements, corrections or other changes without further notice to this document and any product described herein. This document is written in English but may be translated into multiple languages for reference. Only the English version of this document is the final and determinative format released by AM.

8. Any unauthorized copying, modification, distribution, transmission, display or other use of this document (or any portion hereof) is prohibited. AM assumes no responsibility for any losses incurred by the customers or users or any third parties arising from any such unauthorized use.

9. This Notice may be periodically updated.
The Atlas Magnetics logo is a trademark of Atlas Magnetics Co in the United States and other countries. All other trademarks are the property of their respective owners.

Atlas Magnetics,Co
900 Douglas Fir Drive, 2, Reno, Nevada
89511, United States of America

Contact Information:
To access more details, kindly proceed to our website:
https://atlasmagnetics.com/

Atlas EDA technical support:
atlas-eda-support@atlasmagnetics.com

Sales:
crm@atlasmagnetics.com