GreenPAK Designer User Guide

Dialog Semiconductor
Corporate Headquarters
2560 Mission College Blvd
Santa Clara, CA 95054
USA
Phone: +1 408 845 8500
http://www.renesas.com

Copyrights

Contents

1. Introduction

This document describes the installation and usage of Dialog Go Configure® Software Hub software. This software can be used as a stand-alone application for the firmware development and for chips programming. If the information in this guide is not sufficient to resolve issues experienced with software tools, refer to the resources listed under the Support section.

Features that are common for GreenPAK, AnalogPAK, HVPAK, PowerPAK, AutomotivePAK Designers and ForgeFPGA Workshop are described in chapters with a GreenPAK Designer general name. The differences are described in separate chapters.

1.1 Application Overview

Each GreenPAK Designer is a full-featured integrated development environment (IDE) that allows you to specify exactly how you want the device to be configured. This provides you direct access to all GreenPAK device features and complete control over the routing and configuration options. GreenPAK Designer will be used as a general name for GreenPAK 3-6 Designers.

GreenPAK Designer has an integrated programming tool that allows you to program configured design into your GreenPAK chip. With this tool you can also read an already programmed chip and export its data to the Designer. Designer will generate a project, which has the same configuration as chip.

To start working with GreenPAK Designer please take the following steps:

- Download and install Go Configure Software Hub software;
- Select what components you need;
- Interconnect and configure components;
- Specify the pinout;
- Test your design with the Debug Tool;
- Use appropriate GreenPAK development platform to program your project into GreenPAK chip. You can find your kit on Dialog's webstore.

1.2 System requirements

PC System Configuration
Minimum System Requirements for Go Configure Software Hub:
CPU: 2500MHz
System Memory (RAM): 4GB
Graphics Card: 512MB
Free Hard Disk Space: 2GB
Operating System: Windows 7/8.1/10, MAC OS X (v10.8 or higher), Ubuntu 18.04 (32, 64-bit), Debian 11 (32, 64-bit).

1.3 Support

Free support for Go Configure Software Hub is available online at http://www.renesas.com
Also click Help- > Social in software tools and get access to Facebook, Twitter, LinkedIn and Dialog TV.

For software updates, please go to the Software page on our website.
You can find all these resources in the **Help** menu of software tools.

1.4 Acronyms

These are the acronyms used in the User Guide.

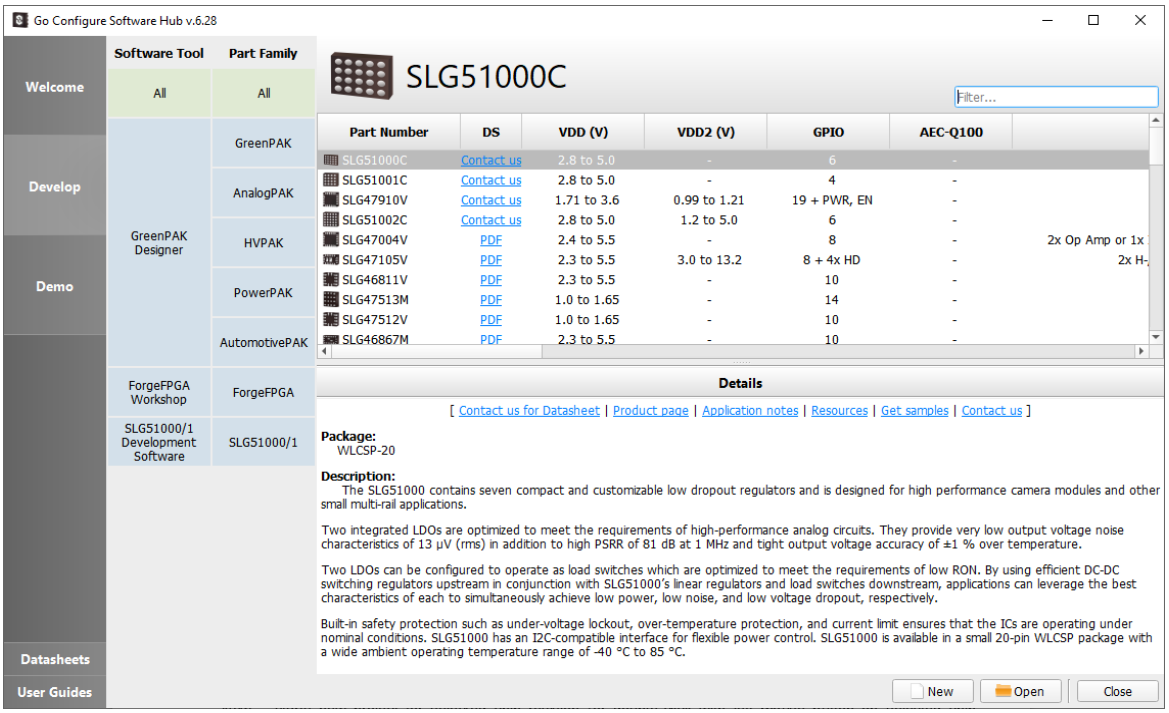
Table 1-1. Acronyms

Acronym	**Description**
GPD	GreenPAK Designer
GPP	GreenPAK Programmer
IDE	Integrated Development Environment
I/O	Input/Output
IC	Integrated Circuit
OE	Output Enable
USB	Universal Serial Bus
GPIO	General Purpose Input / Output
GPI	General Purpose Input
GPO	General Purpose Output
NMOS	N-channel MOSFET (metal-oxide-semiconductor field-effect transistor)
PMOS	P-channel MOSFET (metal-oxide-semiconductor field-effect transistor)
ASM	Asynchronous State Machine

2. Go Configure Software Hub

This section describes Go Configure Software Hub application and its features.

Figure 2-1. Go Configure Software Hub User Interface



Go Configure Software Hub:

- Welcome – welcome page with short information and tips for new users.
- Develop – on this page user can select chip revision to start new project for required revision: SLG51000C, SLG51001C, SLG47910V, SLG51002C, SLG47004V, SLG47115V, SLG47105V, SLG46811V, SLG47513M, SLG47512V, SLG46867M, SLG46857-AP, SLG46855V/-AP, SLG46827-AG, SLG46826V/G, SLG46824V/G, SLG46881V, SLG46880V, SLG46880-AP, SLG46585M, SLG46583V, SLG46582V, SLG46580V, SLG46538-AP, SLG46538V/M, SLG46517M, SLG46537V/M, SLG46536V, SLG46535V, SLG46534V, SLG46533V/M, SLG46532V, SLG46531V, SLG46721V, SLG46722V, SLG46108V, SLG46110V, SLG46116V, SLG46117V, SLG46120V/P, SLG46121V, SLG46125M, SLG46127M, SLG46169V, SLG46170V, SLG46621V, SLG46620V/G/-AG, SLG46625-AP, SLG46140V.
- Demo – on this page user can start software tools in Demo mode for selected Demo project and read information about it.
- Recovery Files – page with restored files after crash or freeze. Files was saved with Autosave feature in predetermined time intervals.

New – starts new project for selected chip revision (or double-click with left mouse button on selected chip revision icon);
Open – opens existing project, automatically selects chip revision;
Close – close Go Configure Software Hub.

3. GreenPAK Designer

This section describes GreenPAK Designer application and its features.

3.1 GreenPAK Designer Interface Overview

GreenPAK Designer consists of: main menu, toolbar, main work area, output window, properties panel and components list (see Figure 3-1, Figure 3-2).

Figure 3-1. GreenPAK User Interface

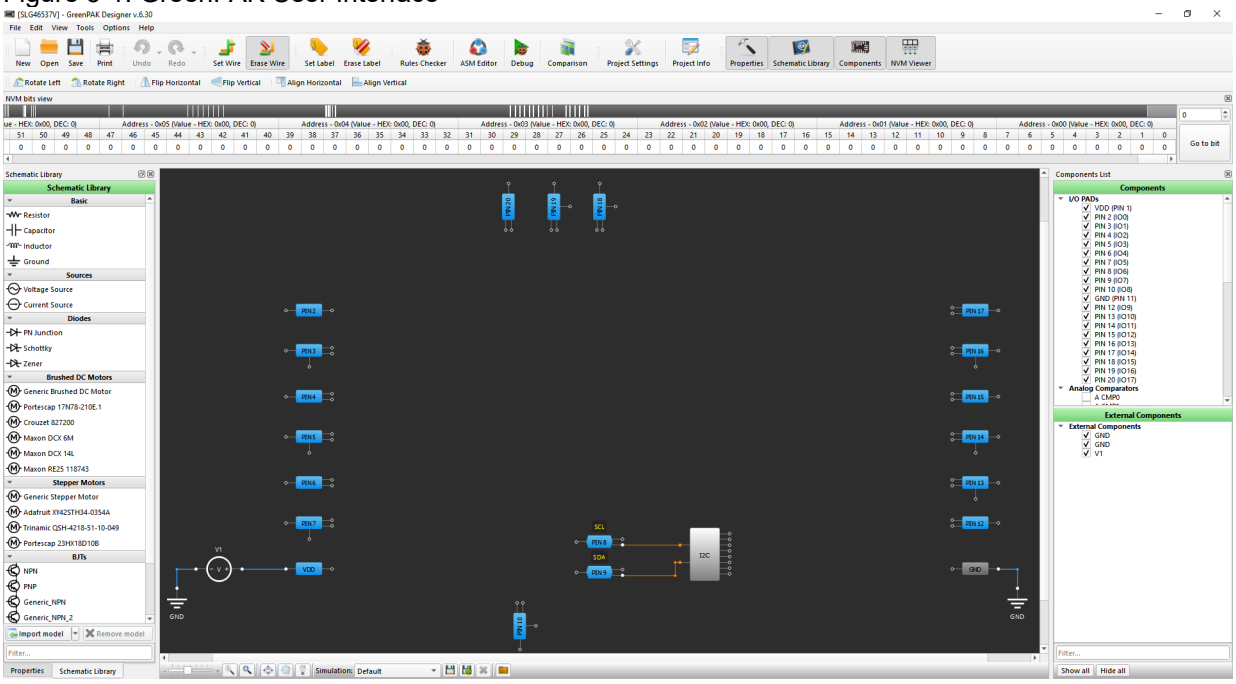
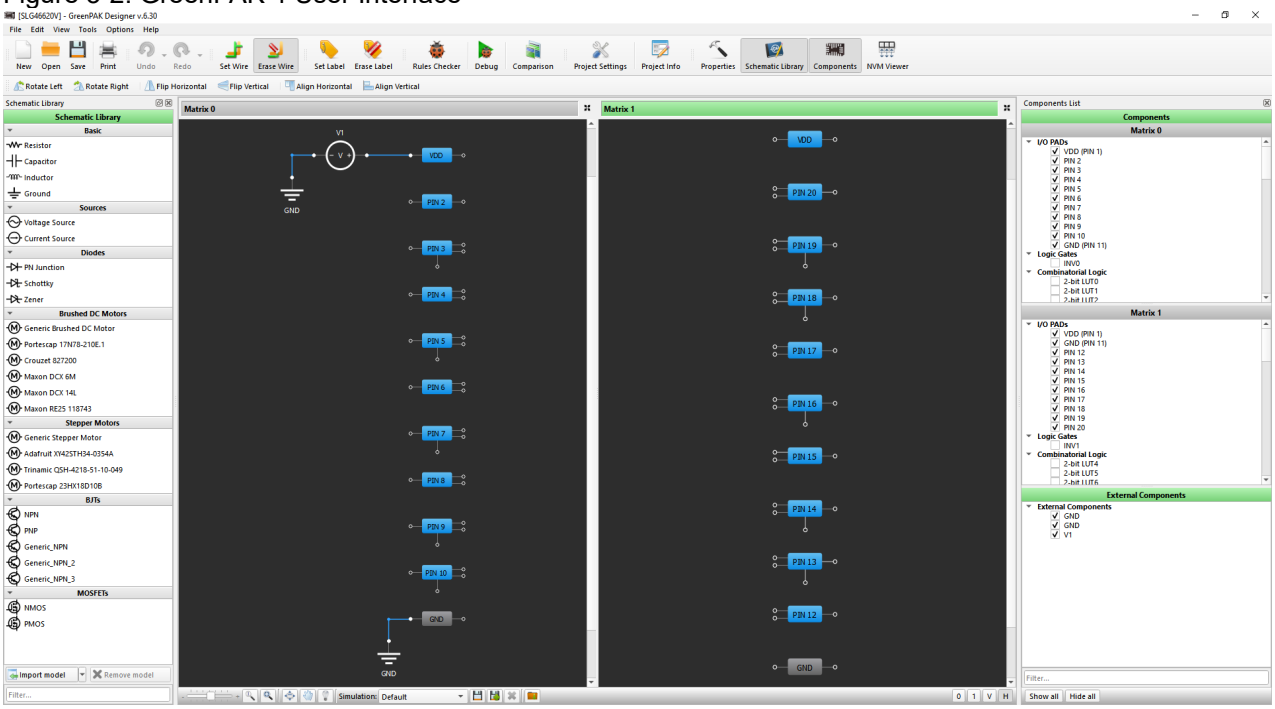


Figure 3-2. GreenPAK 4 User Interface



3.1.1. Main Menu

Main menu contains controls described below:

- **File**
  - New – start new or open existing project from Go Configure Software Hub;
  - Open – open existing project in software tools;
  - Save – save current project;
  - Save as – save current project in specified location;
  - Import NVM bits – load configuration bits from text file;
  - Export NVM bits – save configuration bits to text file;
  - Print – start Print Editor referring item 4;
  - Print Sch (Beta) – simple print feature without detailed block information;
  - Application Notes – opens examples web page;
  - Exit program – close software tools;
- **Edit**
  - Rotate Left – rotate a selected block counterclockwise;
  - Rotate Right – rotate a selected block clockwise;
  - Flip Horizontal – horizontal reflection of a selected block
  - Flip Vertical – vertical reflection of a selected block
  - Align Horizontal – horizontal alignment of selected blocks
  - Align Vertical – vertical alignment of selected blocks
  - Set Label – creating a text label for selected blocks
  - Erase Label – erasing text labels near selected blocks
  - Set Wire – enable wire creating mode;
  - Erase Wire – enable wire erase mode;
- **View**
  - Zoom in – increase the work area scale;
  - Zoom out – decrease the work area scale;
  - Fit work area – tune scale to show all blocks visible in project;
  - Zoom 1:1 – set default scale;
  - Full-screen mode – switch to full-screen mode
  - Pan mode – enable/disable scene move in pan mode;
  - Show hints – enable/disable hints for blocks on the scene;
  - Properties – show/hide Properties panel;
  - Schematic Library – list of external components for Software Simulation;
  - Components – show/hide software tools blocks list;
  - NVM Viewer – show/hide NVM bits viewer;
  - Rules Checker Output;

- **Tools**
  - Debug – this tool is included for convenient project testing;
  - Rules Checker – checks current design for correct settings;
  - Comparison – compares bits of two projects;
  - ASM Editor – allows to configure the State Machine using state diagram and set the output configuration for SM Output block;
  - I2C Tools – enhanced I2C tools with I2C snapshot Reconfigurator (only in GreenPAK 5-6 Designer);
- **Options**
  - Settings – default projects folder, autosave, toolbars position, recovery, shortcuts and update options;
- **Help**
  - Help – show help window;
  - User Guides – open User guides web page;
  - Legend box – show the color legend box;
  - Dialog web site – open Dialog official web site;
  - Software and documentation – open Software & Dock web page;
  - Dialog web store – open Dialog chip store;
  - Design support – web page with training courses and videos;
  - Contact Us – web form with request;
  - Social – Dialog Semiconductor in social networks;
  - Application Notes – open examples web page;
  - Datasheet – open documentation web page;
  - Updater – open software update tool;
  - About GreenPAK Designer – show information about software tools versions modification.

3.1.2. Tool-bars

Toolbar provides a quick access to frequently used functions. There are 8 tool-bars:

- **File**
  - New;
  - Open;
  - Save;
  - Print;
- **Undo**
  - Undo;
  - Redo;
- **Wire**
  - Set wire;
  - Erase Wire;
- **Label**
  - Set Label;
  - Erase Label;

- **Item editor**
  - Rotate Left;
  - Rotate Right;
  - Flip Horizontal;
  - Flip Vertical;
  - Align Horizontal;
  - Align Vertical;
- **Tools**
  - Rules Checker;
  - Debug;
  - ASM Editor (only in GreenPAK 5-6 Designer);
  - Comparison;
  - Project Settings;
- **Panel switcher**
  - Properties;
  - Schematic Library;
  - Components;
  - NVM Viewer;
- **Navigation**
  - Zoom slider – adjust scale;
  - Zoom 1:1;
  - Fit work area;
  - Full screen mode;
  - Pan mode;
  - Show item hint;

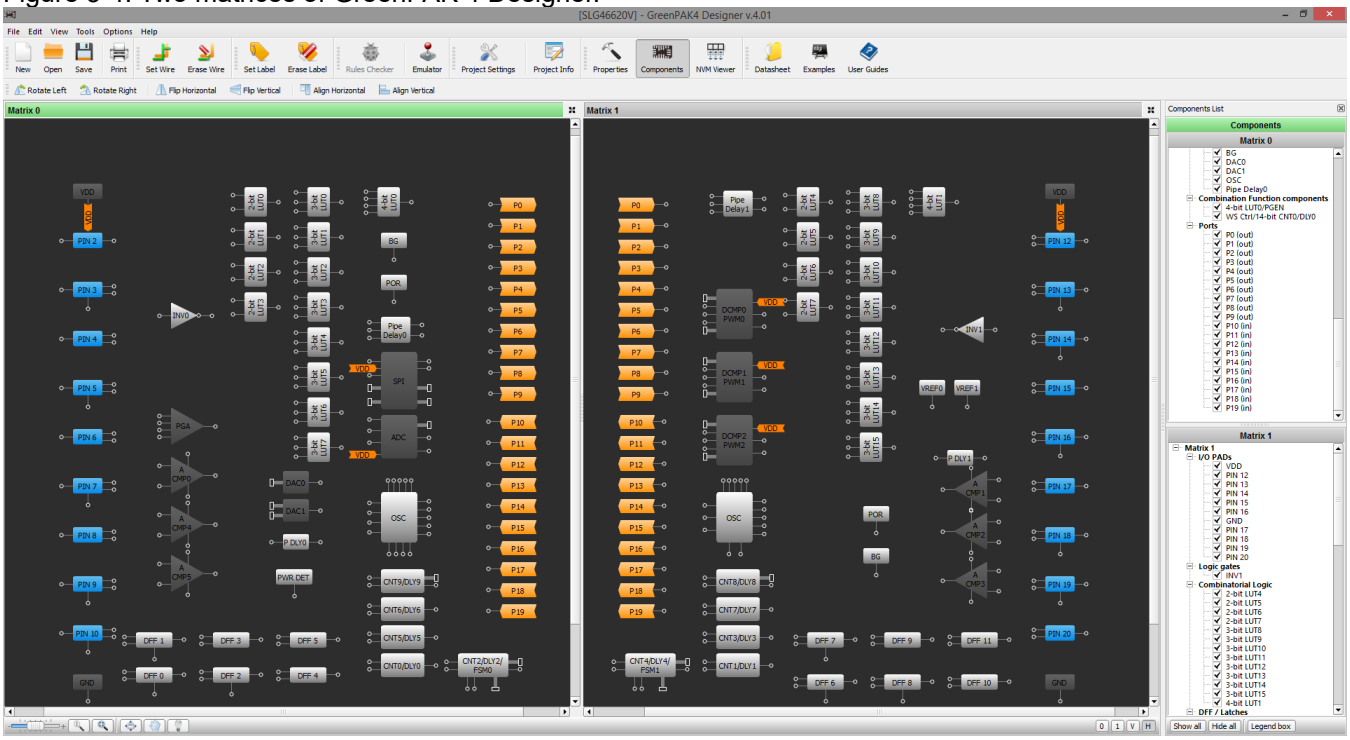
3.1.3. Work Area

Work area contains all blocks available in GreenPAK chip and their connections. In GreenPAK 4 Designer (SLG46620V and SLG46621V) work area consist of 2 matrices: Matrix0 and Matrix1(Figure 3-4). The components of each matrix can be interconnected through 10 input and 10 output ports.

Matrices window placement define buttons: 

- Matrix 0 (full screen);
- Matrix 1 (full screen);
- Vertical placement (2 matrices);
- Horizontal placement (2 matrices);
- One of matrices in separate window or monitor.

Figure 3-4. Two matrices of GreenPAK 4 Designer.



Three types of components connection:

- Connectivity matrix connections (green) – user can connect any output to any input through wiring tool;
- Settings defined connections (orange) – these connections are predefined and depend on block settings;
- Buses (wide orange line) – buses also depend on block settings. All buses are 8-bit wide.

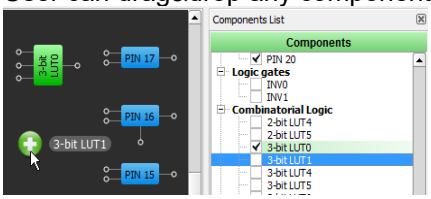
All blocks can be moved using mouse or keyboard (Ctrl+Arrow Keys or Alt+Arrow Keys) and rotated. You can move a few blocks at the same time by using multiple select option. Rotation, flipping and alignment is also available for more than one block at a time.

3.1.4. Properties Panel

Properties panel contains all settings available for selected chip component. The panel is divided in two partitions: **Properties** and **Connections**. Properties division contains settings and parameters that could be specified for a selected block. Connection division contains settings which control the predefined connections to the selected block. Last division could not be present in some blocks. Some parameters and settings are common for a few blocks. There are 2 types of controlling elements **Edit Box** and **Drop List.** To change settings in Drop List you can click and select action or place the mouse pointer over selected list and scroll by mouse wheel. To enter value into the Edit Box you can use keyboard, mouse scroll or buttons at the right. After finishing all configurations press **Apply** button to confirm changes. If you want to discard changes you can press **Reset** button with options: reset settings to default or reset connections to default.

3.1.5. Components List

The Components list is an instrument that contains all blocks available in chip. It provides user with the possibility to show/hide unused blocks. You cannot hide blocks that are connected by any type of lines. In the GreenPAK chip there are connections which are beyond the connectivity matrix. They are controlled by settings of proper components and cannot be fully disconnected. That's why there are some blocks that cannot be hidden. Hidden blocks retain their configuration. For this reason, be sure to configure hidden components properly. You can show/hide selected blocks by using the check/un-check feature on the list. In order to show a group of blocks, double-click on the check box of the desired group. In order to hide a group use a single click.
There are two buttons at the bottom of the components list – Show all (shows all blocks) and Hide all (hides all blocks which are not connected to a circuit). Also user can use filter to find required components.
User can drag&drop any component from Component List to the workarea to the right place:



3.1.6 Color Scheme

Components:

Mode		Description
Normal	Selected	
		Turned on
		Turned off
		I/O PAD

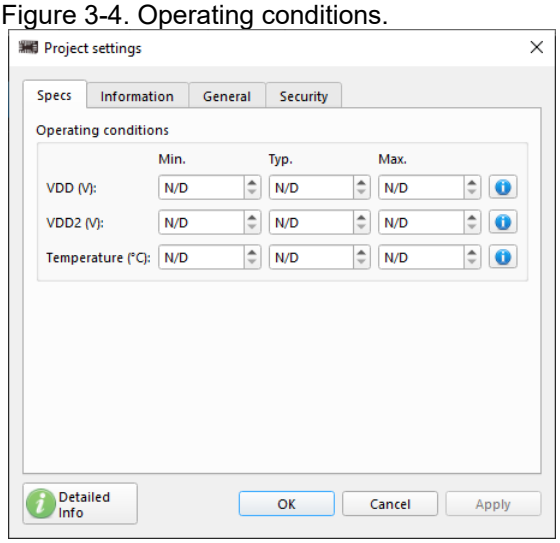
Components Pin Tips:

Color	Value
	User can connect wire to this pin
	Pin has already been connected (and there cannot be any other connections to this pin)
	User can connect wire to this pin only after changing component option
	Inner connection, user cannot connect wire to this pin
	External I/O Pin (I/O from chip)

3.2 Creating a Project

To create a new GreenPAK project start Go Configure Software Hub or go to **File->New** or click the "New" icon on the toolbar. While creating new project in GreenPAK Designer specify operating conditions – VDD and Temperature.

Figure 3-4. Operating conditions.



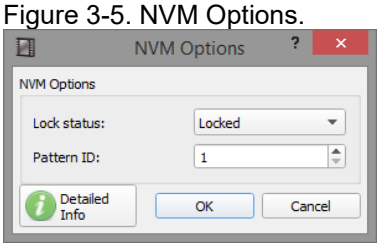
A new project will be created in current window and all unsaved changes will be lost. Also you can start a new GreenPAK Designer copy and it will be loaded with the blank project. By default the project is configured for minimal power consumption and some components are disabled. All disabled components are darker and colored in red after selection. GreenPAK 3-6 projects use [.aap,.hvpak,.gp3,.gp4,.gp5,.gp6] file extension. It contains information about position, rotation/flipping and configuration of chip blocks, all wire connections, and bit file sequence settings of test mode, etc. Interface settings will not be saved in the Project file.

3.2.1 Updating Existing Projects

If you load an existing project created by a previous version of GreenPAK Designer and want to save changes, it will be saved in the updated file format.

3.2.2 Lock NVM Window

Figure 3-5. NVM Options.

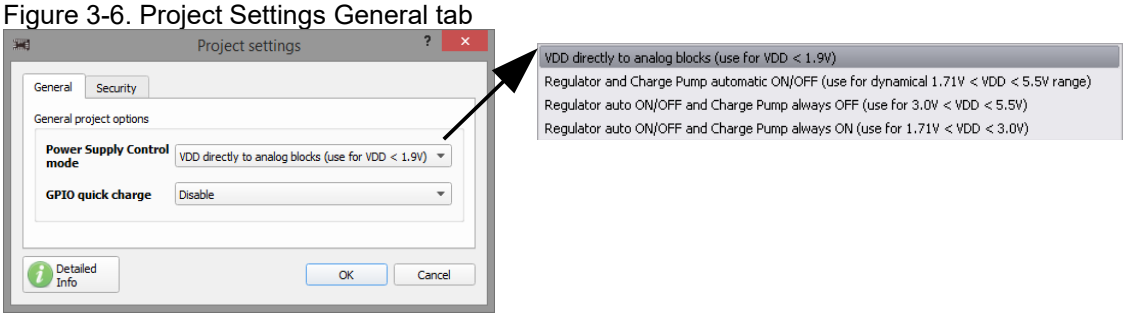


Lock status – blocks NVM reading. A programmed project becomes unavailable for chip reading. Though chip is still applicable for the emulation.

Pattern ID – gives an ID (1-255) to the project. The ID will be put in the chip after programming, and also will be read back during “chip reading” operation.

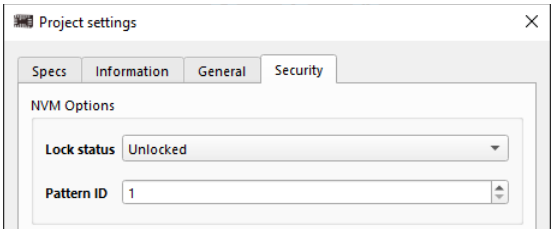
3.2.3 Project Settings Window

Figure 3-6. Project Settings General tab



GPIO quick charge – this option will temporarily enable 2k resistor for ~1µs duration in parallel to internal pull-up/downs during power on, before reset signal is released to internal blocks. For example, this option should be used to ensure the internal pull-up rise-time is fast enough to be detected as high level during power-on. Otherwise, a rising edge with weak internal pull-up can be quite slow, and miss detection as a high-level because of too slow of risetime.

Figure 3-7. Project Settings Security tab

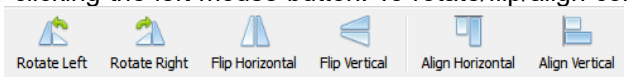


Lock status – blocks NVM reading/writing. A programmed project becomes unavailable for chip reading or for chip writing via I2C Tools for chips with I2C Serial Communication (see chip datasheet). The chip is still applicable for emulation.

Pattern ID – gives an ID (1-254/65534) to the project. The ID will be put in the chip after programming, and also will be read while “chip reading” operates.

3.3 Configuring Chip Components

3.3.1 Placing Components

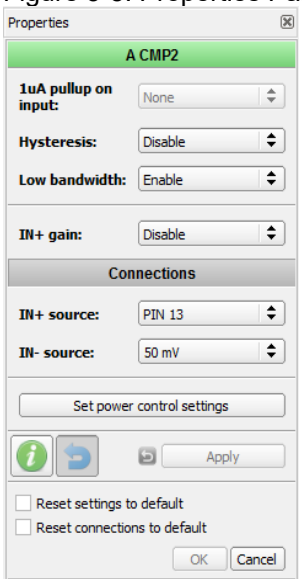
When you open GreenPAK Designer it will start with a blank project. A blank project contains pins and blocks which cannot be hidden. Components can be moved, rotated, flipped and aligned. In order to move a component, simply drag it where you want by clicking the left mouse button. To rotate/flip/align component select it and press the "Rotate/Flip/Align" buttons 

on the toolbar or select Rotate/Flip/Align in the main menu.

3.3.2 Setting Chip Components Parameters

Each chip component has different parameters. Some components have parameters that are shared with other components. Changes in one block cause changes in other blocks. Component settings are available at component **Properties** panel (Figure 3-8) which appears after double-clicking on the component. **Properties** panel consists of three parts: Properties, Connections, and Information. Properties section contains all settings of a selected component. Connections section allows you to configure connections that couldn't be made using wiring tool. Information section contains short information about parameters of selected component. After making changes in **Properties** panel click the **"Apply"** button to save changes. If you do not click the **"Apply"** button and select another block, a save changes message box will appear.

Figure 3-8. Properties Panel



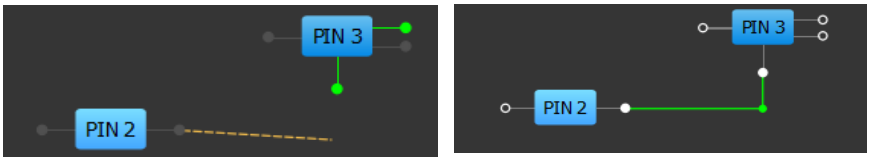
Reset connections and/or settings to default: this option allows to reset NVM bits, components properties, wire connections from/to component.

3.4 Specifying Interconnections

You can interconnect chip components to achieve the necessary functionality. To make a connection please select **Set wire** 

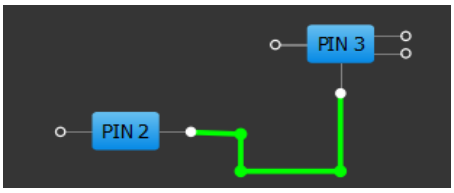
on the **Wire** toolbar or from the main menu. Next, click the first and second pins that you want to connect. After selecting the first pin, GPD highlights allowed connections in green. If you click the first pin and then decide to exit line creating mode press **Esc** or the right mouse button.

Figure 3-9.



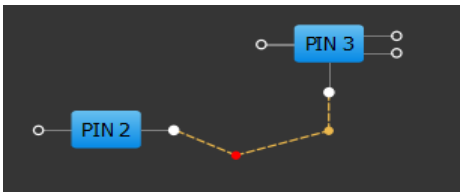
Also you can manually correct the created wires.
You can move horizontal lines up and down, vertical lines left and right (Figure 3-10).

Figure 3-10.



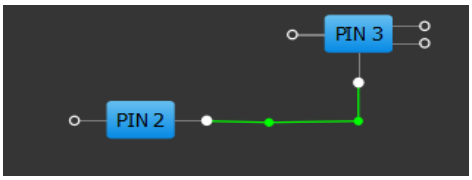
You can move points on the wire (Figure 3-11).

Figure 3-11.



In order to create additional points on the line use the double click (Figure 3-12).

Figure 3-12.



Only the green color pins can be connected Using Wire Creating tool. Some components have pins that are not allowed to be connected using wiring tool. Connections between such pins (orange dotted line and violet pin color) and buses can be made only by changing settings in **Connections** section of the **Properties** panel of proper components. In this case violet pins can change color to green and user can connect them using wiring tool. Orange wires will be automatically generated. Orange wires also can be modified by user. Input pins without connections are considered to be tied to ground.

In order to delete wire please select **Erase wire** 

at the **Wire** tool-bar and click on the selected wire. Only green wires can be deleted.

Additional controls for add/remove wires:
Hold button to force wire mode:

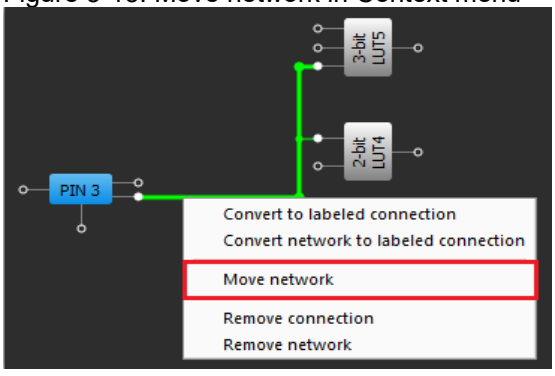
- Shift: for Set Wire;
- Alt: for Erase Wire;

Action with multiple wires:

- Hold Ctrl+Shift and click on pin: add multiple wires from the same source pin;
- Hold Ctrl+Alt and click on wire: remove all wires from source pin;
- Hold Ctrl: works as Ctrl+Shift or Ctrl+Alt based on current wire mode;

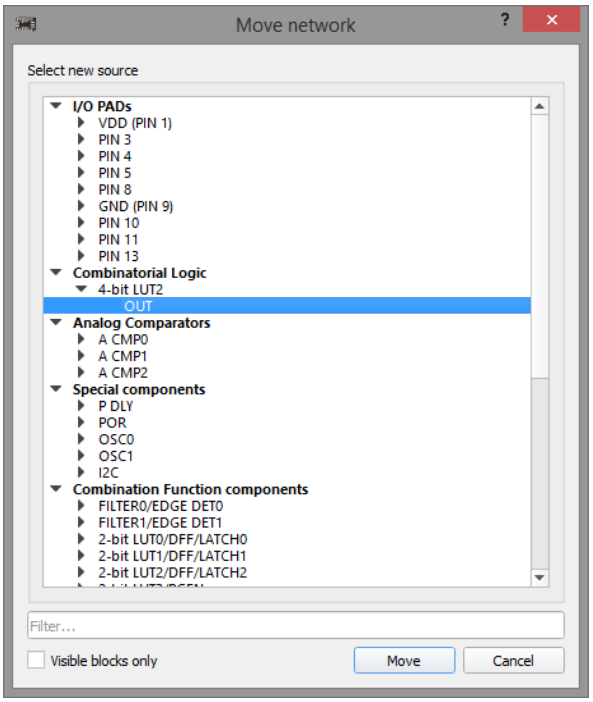
Move network
Move network feature provides the fastest way to reconnect all matrix wires from any pin to another. Simply click on wire with right mouse button and select Move network in Context menu

Figure 3-13. Move network in Context menu



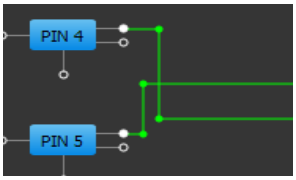
Select new source from list in Move network window. User can select new source only from list of visible blocks or from list of all blocks.

Figure 3-14. Move network window



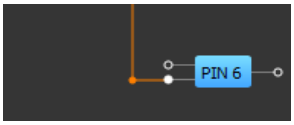
3.4.1 Wire Types

Figure 3-15. Green Line



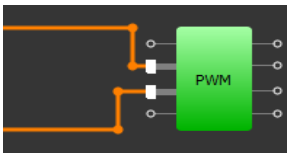
Green lines in GreenPAK Designer software tools are used to mark manual wires. Using them you can manually connect necessary blocks to operate in the desired way. You can connect block output to multiple inputs, but wiring of different outputs to one input is impossible.

Figure 3-16. Orange Line



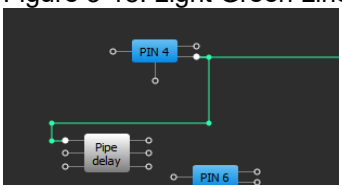
Orange lines are used to mark the internal functional bounds of the chip blocks. They do not have the impact on chip operation until the proper function is used. These lines can't be erased.

Figure 3-17. Bold Orange Line



Bold orange lines (like the orange lines) mark the internal bounds. The difference is that the bold orange lines mark 8-bit parallel data buses. These lines also cannot be erased. They do not have an impact on chip operation until the proper function is used and the proper option is set.

Figure 3-18. Light Green Line

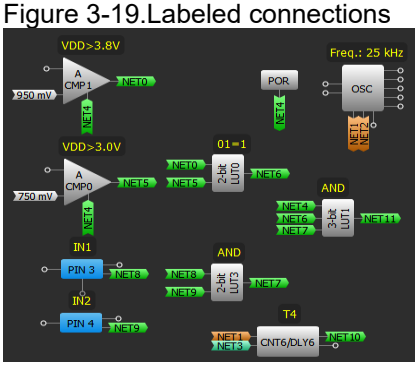


Light green lines are used to mark the shared connections. Their behavior is the same as the green lines.

Replacing wires by labels

This option converts wired connection to 2 labels (for output and input pins) and back (Figure 3-19). Name of the label will be generated automatically: NETx, where x – random number. If output was connected to few inputs all of them should have the same name. For changing the connection type use the context menu of the block, line or label(NET).

Figure 3-19.Labeled connections



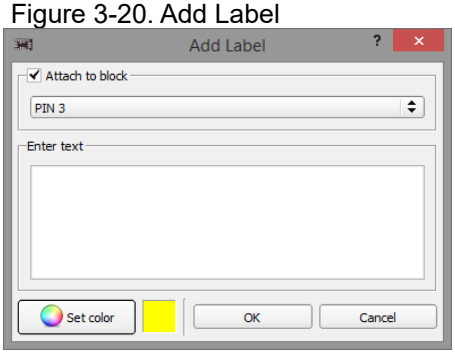
Available options for wire (context menu):
- Convert to labeled connection;

Available options for label (context menu):
- Convert to wired connection;
- Rename network;
- Remove connection.

3.4.2 Set/Erase Label

Using Set/Erase Label the user can add/delete text label. The Set Label tool adds a text label to the selected component or without connecting them to the specific component. The user can Attach label to component or Detach label(s) from component(s). If no component is selected, then the user can select a component from the list offered by the Set Label tool. The user can also choose text color. If the selected component already has a label, Set Label tool can edit label text. If the user selects more than one component, it is possible to change the text color without changing text in all components at once. If the user changes the text while more than one component is selected, it will be changed on all selected components at once as well. Erase Label deletes text label.

Figure 3-20. Add Label



3.5 Specifying the Pinout

3.5.1 Port Connections

Pin blocks can be connected just like any other blocks using the Wiring Tool.

3.5.2 Port Drive Modes

GreenPAK chips have GPIO, GPO and GPI pin components. These components can be configured to work in the following modes:

- Digital in with Schmitt trigger;
- Digital in without Schmitt trigger;
- Low voltage digital in;
- 1x push pull;
- 2x push pull;
- 1x open drain NMOS;
- 2x open drain NMOS;
- 4x open drain NMOS;
- 1x open drain PMOS;
- 2x open drain PMOS;
- 1x 3-State Output;
- 2x 3-State Output;
- Analog input;
- Analog output;

Also, Pull-Up/Pull-Down resistors are configurable. To configure the pin component, open its parameters to set a desired mode and pull-up/pull-down resistor. I/O pin components have **input (IN)**, **output (OUT)** and **output enable (OE)** pins. These pins are one-way directed, so you need to configure the pin component and connect the proper pin. OUT pin is an output signal from the pin component. It corresponds to the signal from the input buffer. IN pin is an input to the pin component. It accepts a signal from internal components. Output Enable (OE) signal defines the Push-pull buffer state. Low OE signal switches buffer to Hi-Z state. High OE signal enables Push-pull buffer regardless of selected component operating mode. It could be used for applications where bidirectional pins are needed.

3.6 Navigation

To navigate through project workspace use the **View** menu or toolbar. Use **Zoom In** , **Zoom Out** buttons or slider to zoom workspace. If you want to see all project components click on **Fit work area** or **Zoom 1:1** . To navigate through work area you can use **Pan mode** . Pan mode also activates by using middle mouse button.

To enable block's hint, press **Show item hints** button. A hint box pops up next to the item when the mouse moves over the block.

3.7 Keyboard commands

To navigate through GreenPAK Designer use specific keyboard commands or shortcuts. List of commands specified in the table:

Table 3-1. Keyboard commands

Keyboard command	***Action***
Block moving on the scene	
Alt+Arrow Keys	Moves selected block on 1 pixel
Ctrl+Arrow Keys	Moves selected block on 10 pixels
Connecting/Erasing wires	
Hold *Shift*	Forces Set wire while using Erase Wire
Hold *Alt*	Forces Erase wire while using Set Wire
Hold *Ctrl+mouse cursor*	Adds multiple wires from the same source
Hold *Ctrl+Shift+mouse cursor*	Forces add of multiple wires from the same source while using Erase Wire
Hold *Ctrl+Alt+mouse cursor*	Forces remove of all wires from the network while using Set Wire
Standard hotkeys	
Ctrl+Z	Undo
Ctrl+Y	Redo
Ctrl+N	New project
Ctrl+O	Open project
Ctrl+S	Save project
Ctrl+P	Print Editor

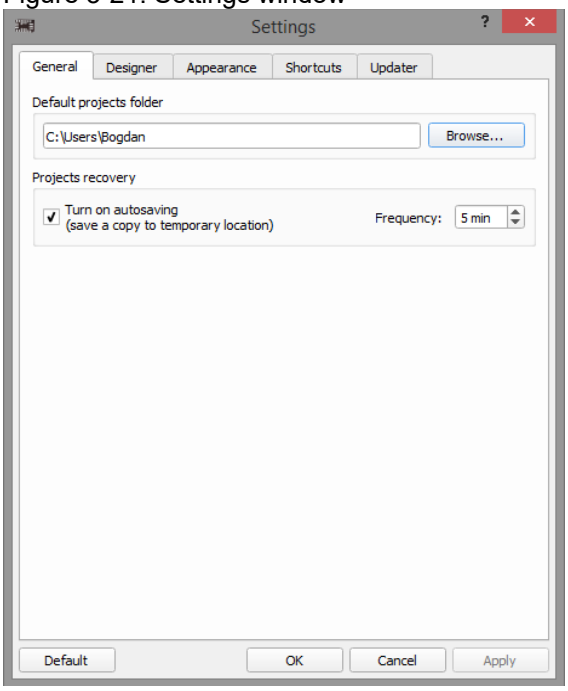
Ctrl+Q	Exit program
Ctrl+L	Rotate component Left
Ctrl+R	Rotate component Right
Ctrl+H	Flip component Horizontal
Ctrl+V	Flip component Vertical
Ctrl+W	Set Wire
Ctrl+E	Erase Wire
Ctrl+F	Filter on Components List
H	Hide component
+	Zoom in
-	Zoom out
F1	Help
F2	NVM Viewer
F3	Properties of component
F4	Components List
F5	Rules Checker
F9	Debug
F11	Fullscreen Mode
Debug hotkeys	
Shift+E	Emulation
Shift+I	Info
Shift+L	Log
Shift+N	NVM Data
Shift+P	Program
Shift+R	Read
Shift+S	Save Settings
Shift+T	Test Mode

All other Designer main window actions can be configured by entering specific key sequence in Settings window on Shortcuts tab. For ASM Editor user should enter key sequence in format: Ctrl+key.

3.8 GreenPAK Designer Settings

GreenPAK Designer settings configure all basic options of program in several tabs (Figure 3-21). To open settings select Options-> Settings in main menu.

Figure 3-21. Settings window

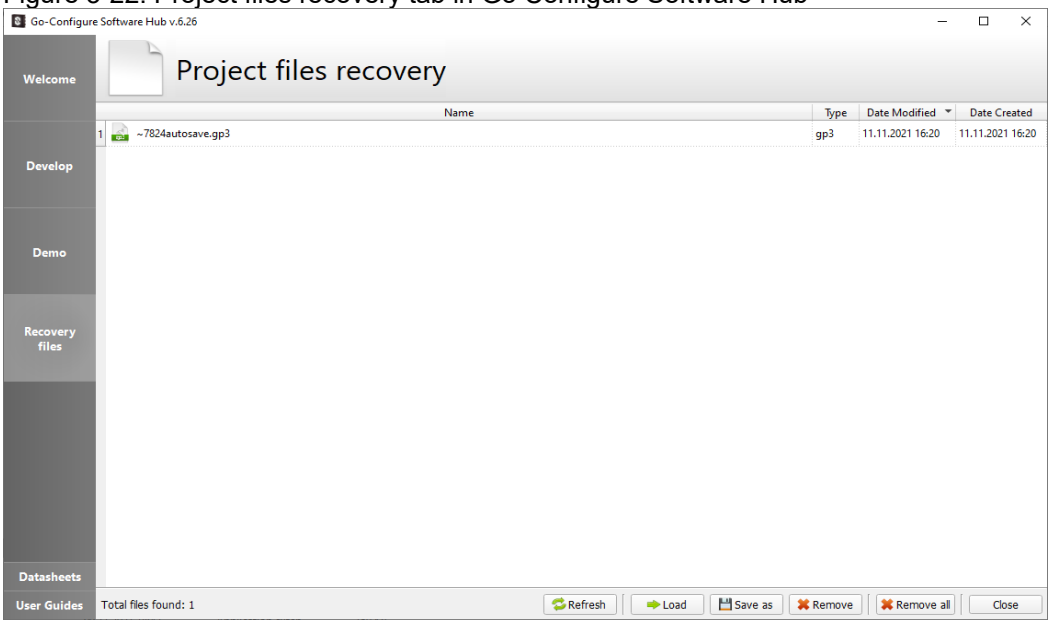


GreenPAK Designer settings window contains of tabs:

General:

- Default projects folder – defines path to users project files;
- Projects recovery – activates autosave function, which allows to reduce the risk or impact of data loss in case of a crash or freeze. Autosave function in predetermined time intervals will save your files and after a critical problem will offer to restore these files in Go Configure Software Hub on Projects files recovery tab (Figure 3-22);

Figure 3-22. Project files recovery tab in Go Configure Software Hub



Designer:

- Pin hints – shows pin hints while block is selected or properties panel of component is visible:

- Look-Up Table (LUT) – allow usage of regular shape by default. For example, regular shape of NXOR:

Appearance:

- Window appearance – saves positions of toolbars/dock widgets and window geometry of GP Designer work area;
- High DPI displays – enables GP Designer scalling on high DPI displays;

Shortcuts:

- On Shortcuts tab all GP Designer actions can be configured by entering specific key sequence. For ASM Editor user should enter key sequence in format: Ctrl+key;

Updater:

- Scheduler – determines check for updates time: after Designer starts or Once per 1-7 days;
- Path – defines server for update and destination to download updates;
- Proxy – allow user to configure proxy for updates;
- Check configuration button – checks connection to server.

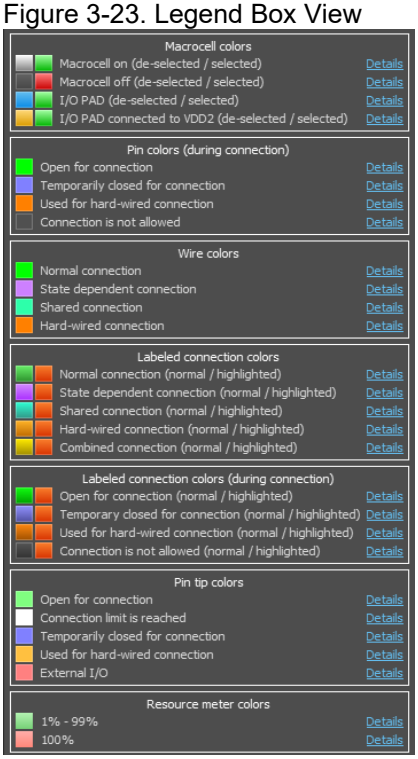
Default button:

- Resets settings to default parameters by categories or all at once.

3.9 Legend Box

Legend box shows the color scheme of GreenPAK designer.
The user can open this window by clicking 'Legend box' button in 'Help' menu.

Figure 3-23. Legend Box View

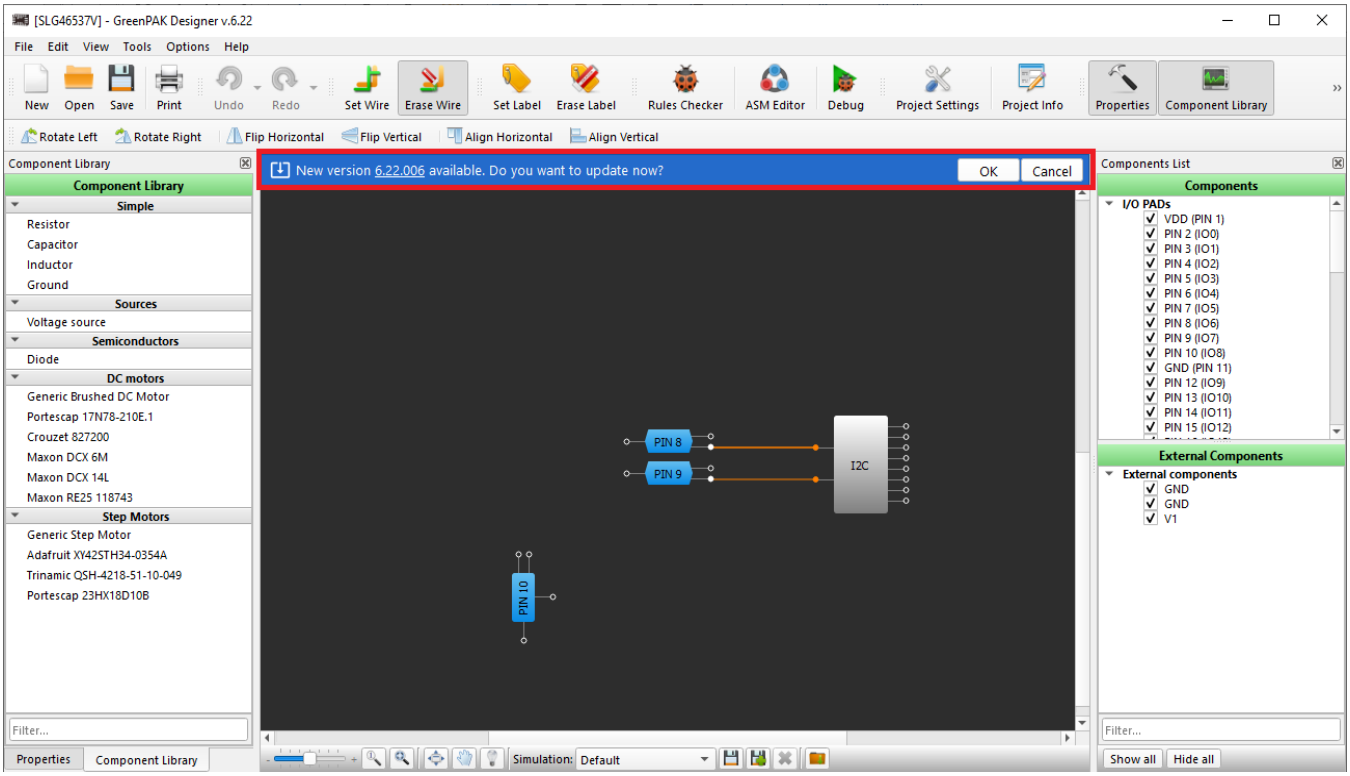


3.10. Updating GreenPAK Designer

There are two ways of updating the GreenPAK Designer:

- When updates are available – this information will be displayed. The user will get a chance to either download a new version using the “OK”, or the “Cancel” to delay the update until the next program start. After the download is finished, Designer automatically installs the software.
  IMPORTANT: After finishing installation, restart the GreenPAK Designer software.

Figure 3-24. Updating GreenPAK Designer



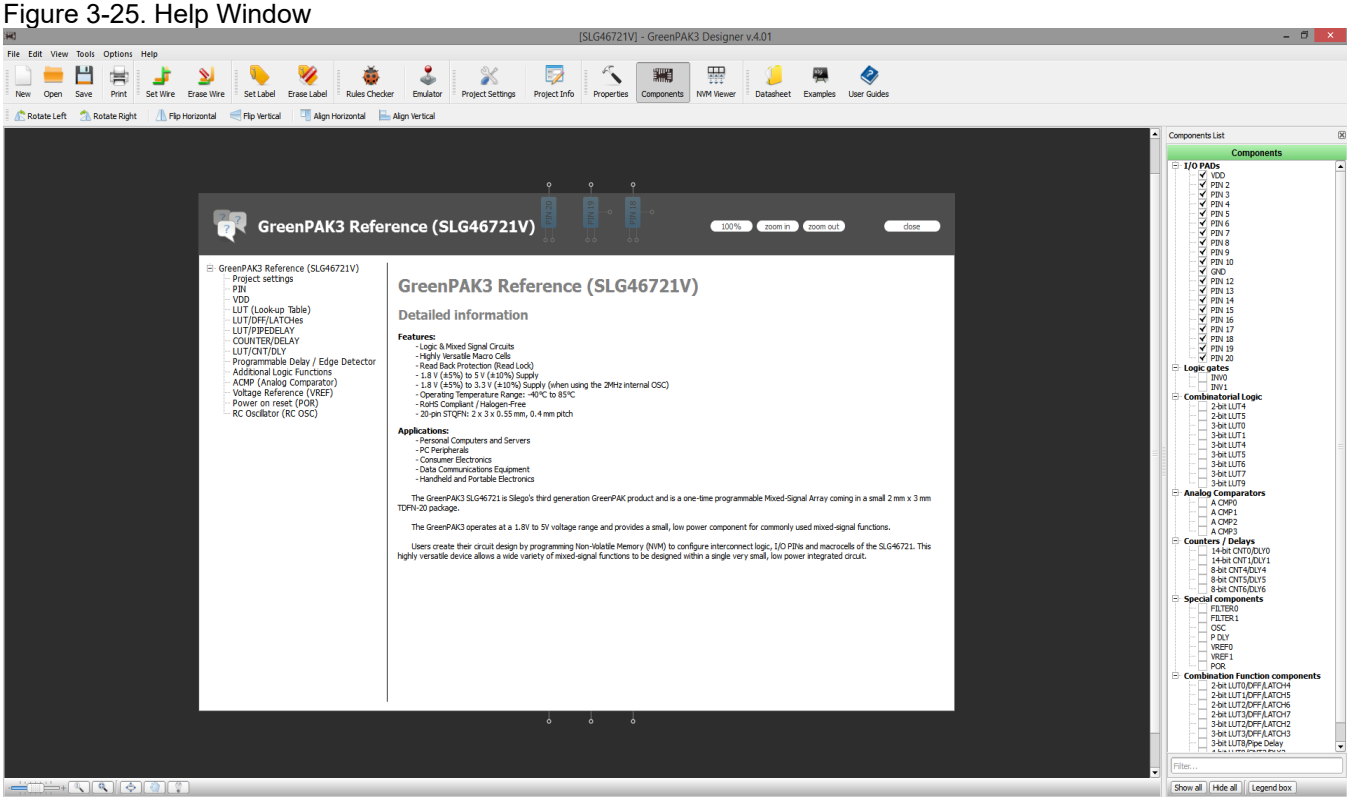
- You can also find the latest GPD version at Software page of Dialog web site. For the best user experience, keep your GreenPAK Designer up to date. Feel free to email suggested updates to the developer to improve this program (Please refer to **“About GreenPAK ”** section of **Help** menu).

Configure Updater options on Updater tab in Designer Settings window (see Section 3.8 GreenPAK Designer Settings)

3.11 Help Window

To view information about a specific block, select the block and click 'Help' from the Help menu or press the 'F1' button. A window will list the information about each block ('short info'). Press the 'detailed info' button for more detailed information (Figure 3-25). If you don't select any block, you will be shown the information about all the blocks. The 'Help' button on the property panel of each block provides the same information about the current block.

Figure 3-25. Help Window



3.12 Demo board and Demo mode

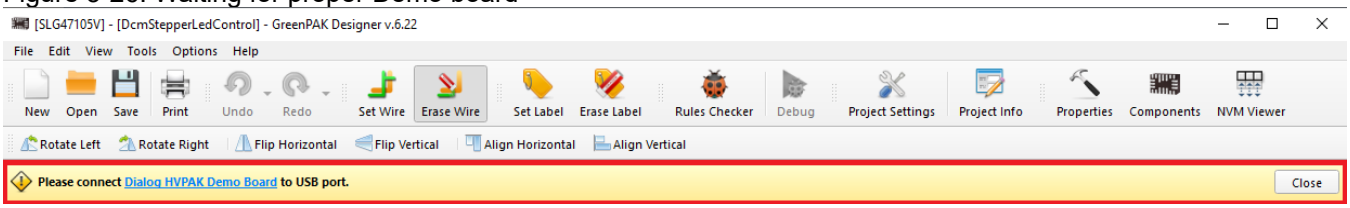
Demo board

Demo board is a special hardware with a mission to demonstrate some specific application of GreenPAK chip. They have GreenPAK chip soldered on the board, already programmed with some specific project. They also support I2C transferring that allows Designer to communicate with GreenPAK chip and change it NVM.
The exception is I2C Bridge which doesn't have specific project and allows any project to be loaded in.

Demo board connection

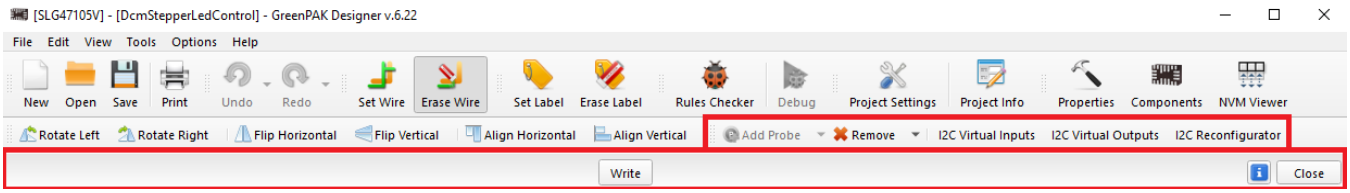
After opening of Demo project, Designer waits for connecting of proper device:

Figure 3-26. Waiting for proper Demo board



Demo board detection

Figure 3-27. Demo board detected



After Proper Demo board detected (Figure 3-27), simple I2C tool activated:

- Write - sends current project's NVM to device;
- I2C Virtual Inputs, I2C Virtual Outputs (with Add/Remove Probe tool) - opens I2C tools (see Section 6.4 Control Panel);
- I2C Reconfigurator – starts shapshots reconfigurator tool (see Section 6.4 Control Panel);
- Close – exits demo mode;
- Info button – shows all system and hardware information;

Designer's Demo Mode

User can start designer in Demo mode, which means it will show specific project for selected demo and provide tool to operate with demo board. Demo mode applies some restrictions on Designer functions as well as adds of a new features. In general case it opens specific demo project, shows specific tool control panel and limits operations with project file.

Exit demo mode

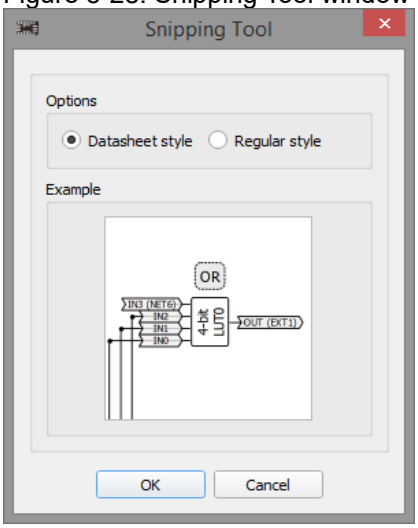
User can exit demo mode. This will decline all restrictions applied by demo mode, but keep current project open.

3.13 Snipping Tool

Snipping Tool is screenshot tool for software tools workarea. It allows scene selection, copying or saving as a file.

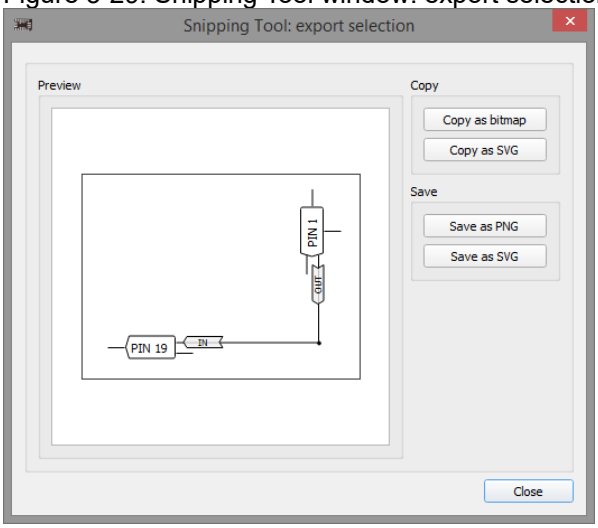
Click Tools → Snipping Tool, select style of screenshot area (Figure 3-28)

Figure 3-28. Snipping Tool window with style selection



Select area and copy to clipboard or save image in Bitmap/PNG/SVG format (Figure 3-29)

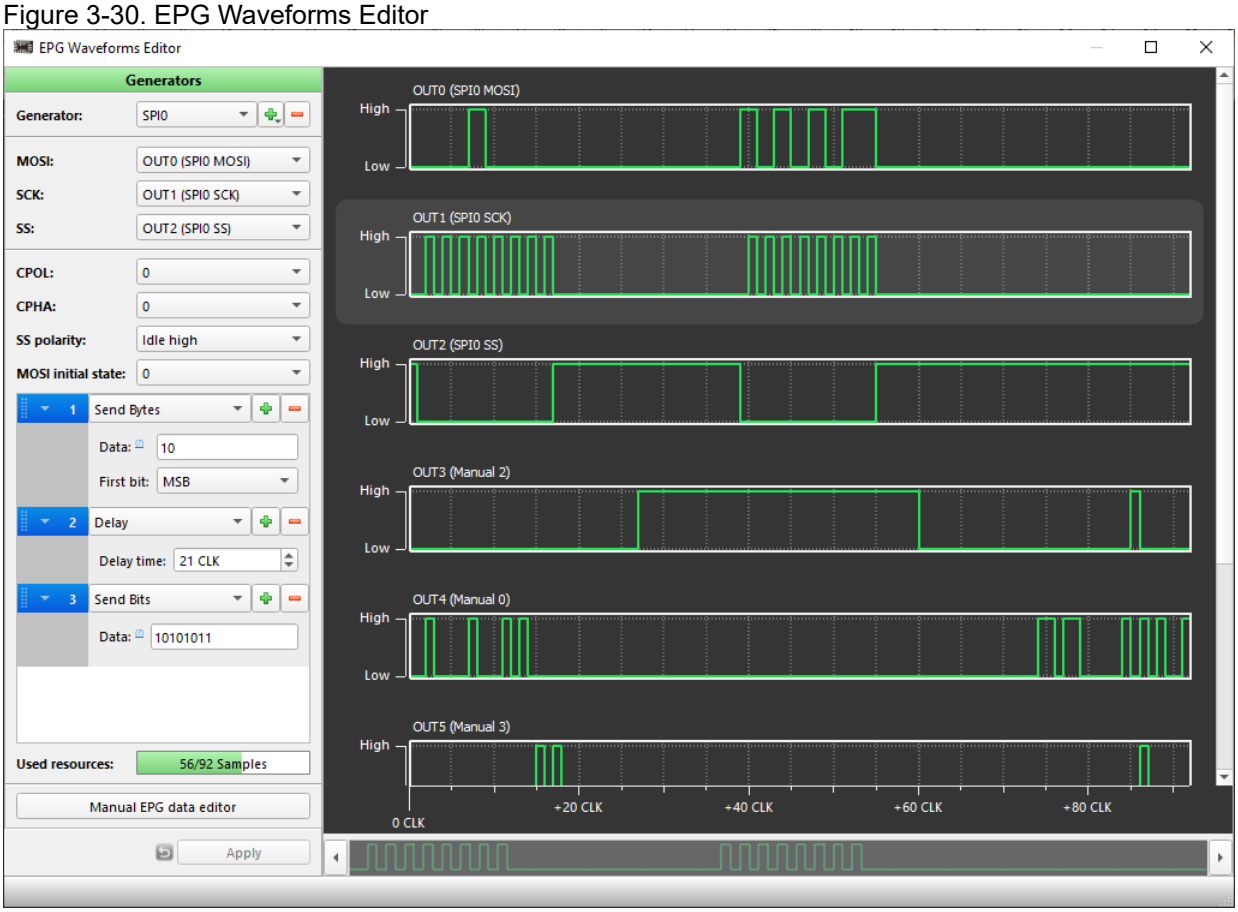
Figure 3-29. Snipping Tool window: export selection



3.14 EPG Waveforms Editor

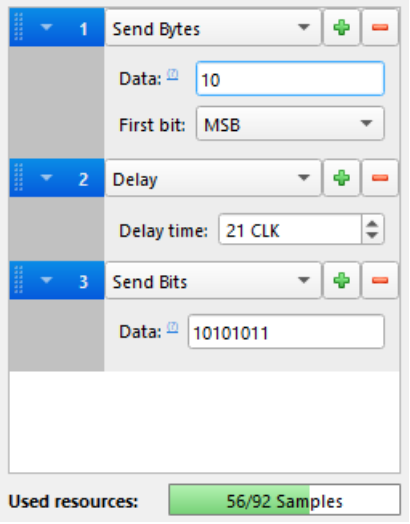
The Extended Pattern Generator (EPG) range is 92 bytes and at every rising edge of CLK input a byte from NVM appears at the output. EPG shares its outputs with I2C Virtual Inputs. Configure Extended Pattern Generator in EPG Waveforms Editor(Figure 3-30) using SPI, I2C, PWM or Manual type of generator.

Figure 3-30. EPG Waveforms Editor



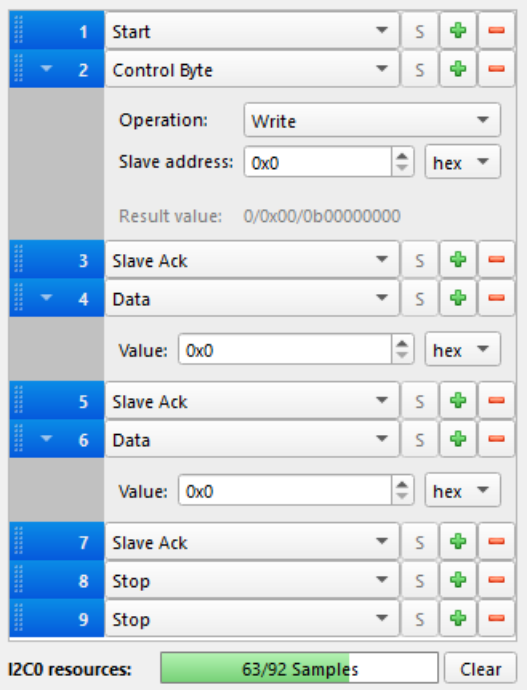
SPI generator has output pins, commands editor and samples resources meter (Figure 3-31).

Figure 3-31. SPI generator commands editor and resources meter



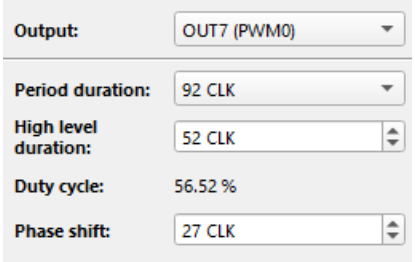
I2C generator has SDA and SCL output pins, commands editor and samples resources meter (Figure 3-32). In editor user selects commands from list of Basic (start, stop, etc.) and GreenPAK access (Read/Write). “S” button splits composite GP access commands in sequence of Basic commands.

Figure 3-32. I2C generator commands editor and resources meter



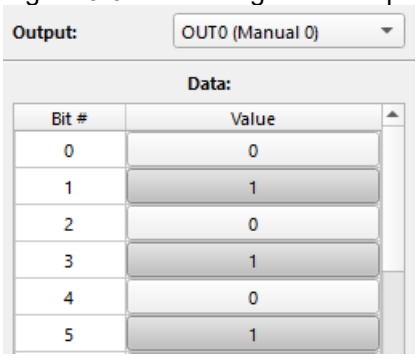
PWM generator configures output pin, period duration, high level duration and phase shift (Figure 3-33).

Figure 3-33. PWM generator options



Manual generator configures output pin and bits values (Figure 3-34).

Figure 3-34. Manual generator options



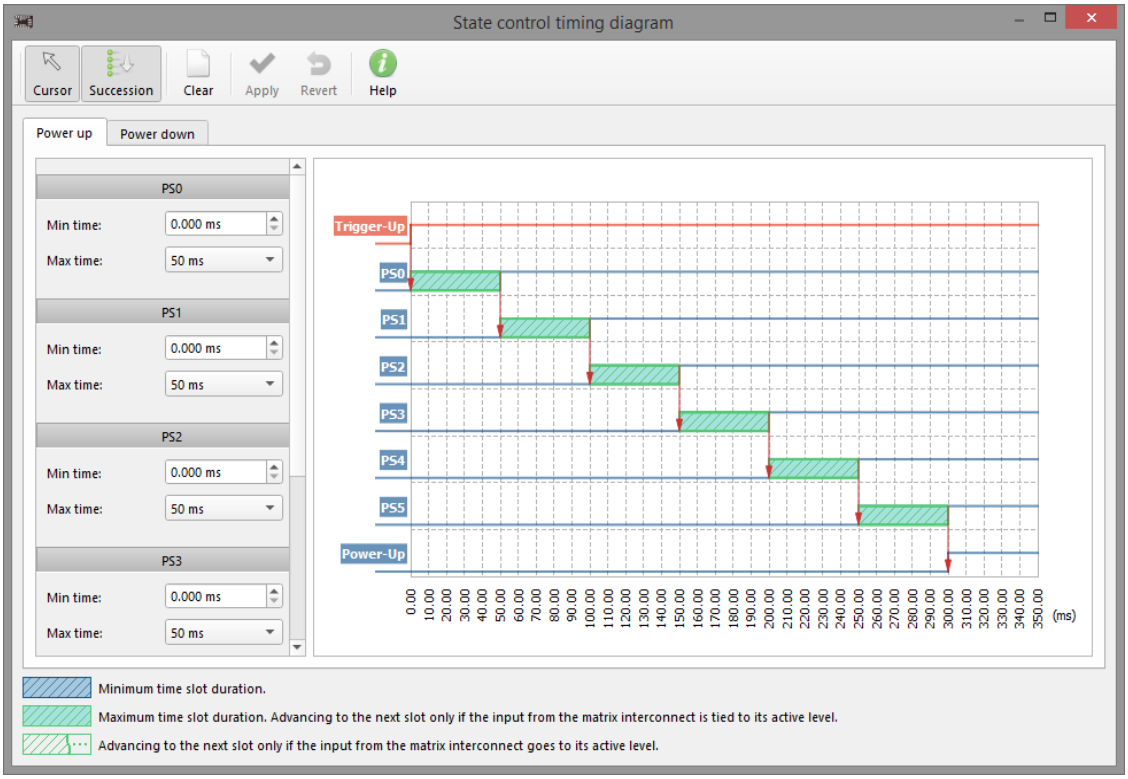
3.15 Timing diagram

The supply controller provides a flexible power sequencer that controls the power-up and power-down timings for the six resource enable outputs feeding the matrix interconnect. The timing sequence is divided into six slots or discrete periods of time between events. There are two dedicated configurable sequences (up and down). Initiation of a sequence is performed with the trigger-up and trigger-down control signals from the matrix interconnect.

The Power sequencer supports, One-Time Programmable (OTP) configurable, minimum and maximum slot duration limits for each slot in each of the power sequences.

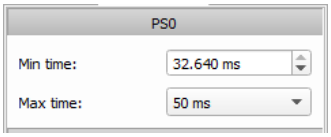
Timing diagram state control allows to configure the Power sequencer minimum and maximum slot durations provided for each slot and for both the up and down sequences (Figure 3-35).

Figure 3-35. Timing diagram window



The time duration in a given slot is limited to the minimum duration regardless of the state of the power-up and power-down signals from the matrix interconnect. The time duration in a given slot is limited to the maximum slot duration, when maximum duration control is configured as enabled in OTP.
Slot duration minimum timers support a range of 0 ms to 32.64 ms with a resolution of 128 µs.
Slot duration maximum timers support options of (0, 10, 30, or 50) ms (Figure 3-36).

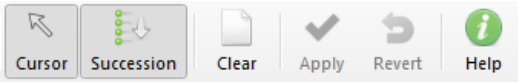
Figure 3-36. State minimum and maximum time



Timing diagram options (Figure 3-37):

- Cursor – shows time label when cursor is placed over the timing diagram;
- Succession – shows Power sequencer state transitions on timing diagram;
- Clear – clears all PS time settings to default;
- Apply – saves all min and max time parameters of PS, changed by user;
- Revert – discards all min and max time parameters of PS, changed by user;
- Help – shows help.

Figure 3-37. Timing diagram options



4. Print Function

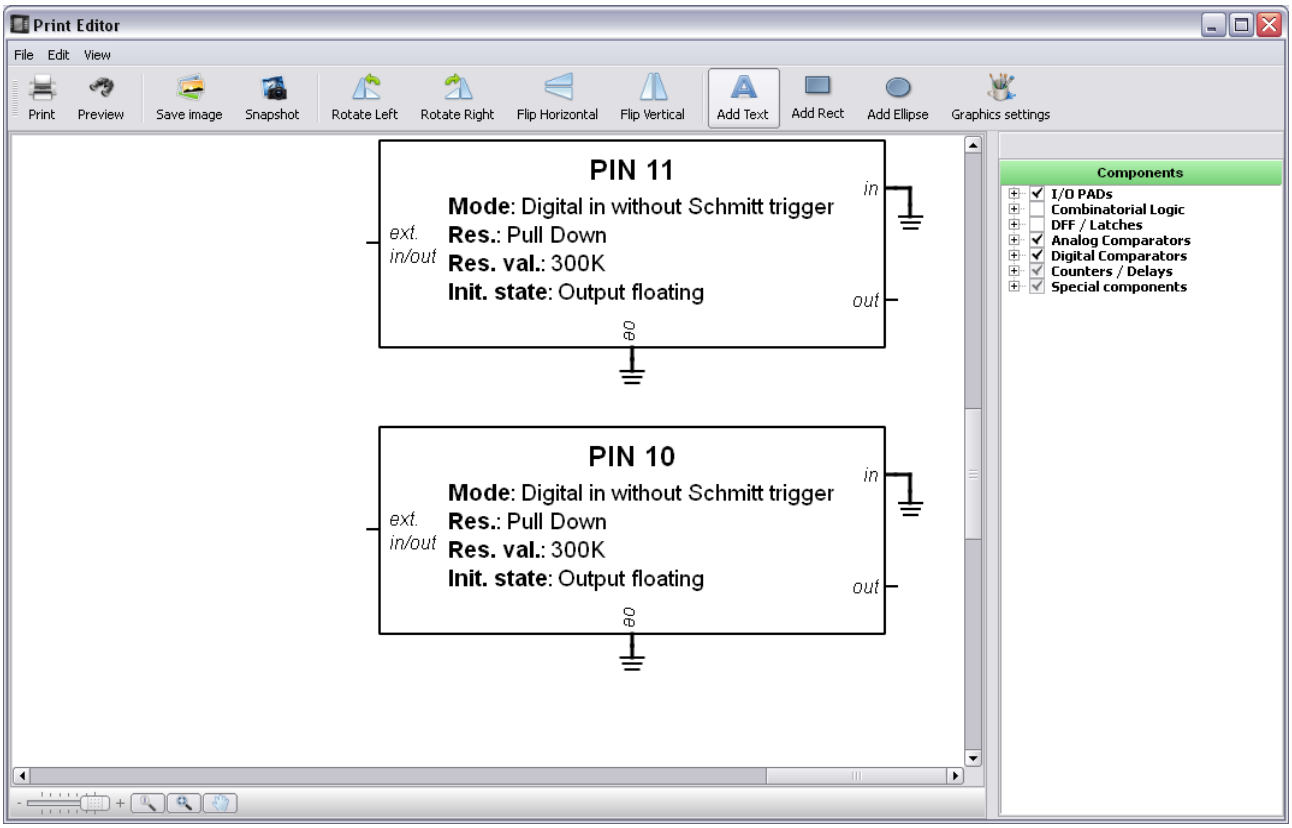
4.1. Print Editor

Print Editor feature consists of two main parts:

- Editable working area, where the user can customize positions, view of components, and lines connecting them.
- Preview window where the user can set up the print preferences.

Editable working area shows all components which where used in the design.

Figure 4-1. Print Editor



The Main Actions:
- User can hide or display any component using the Components list on the right.
- Each component in the work area is selectable and movable.
- Any component can be rotated or flipped.

Note: print editor settings will be saved before print editor's window is closed. It allows the user to repair previous state during next opening.

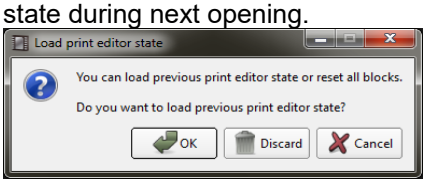


Figure 4-2. Preview Window

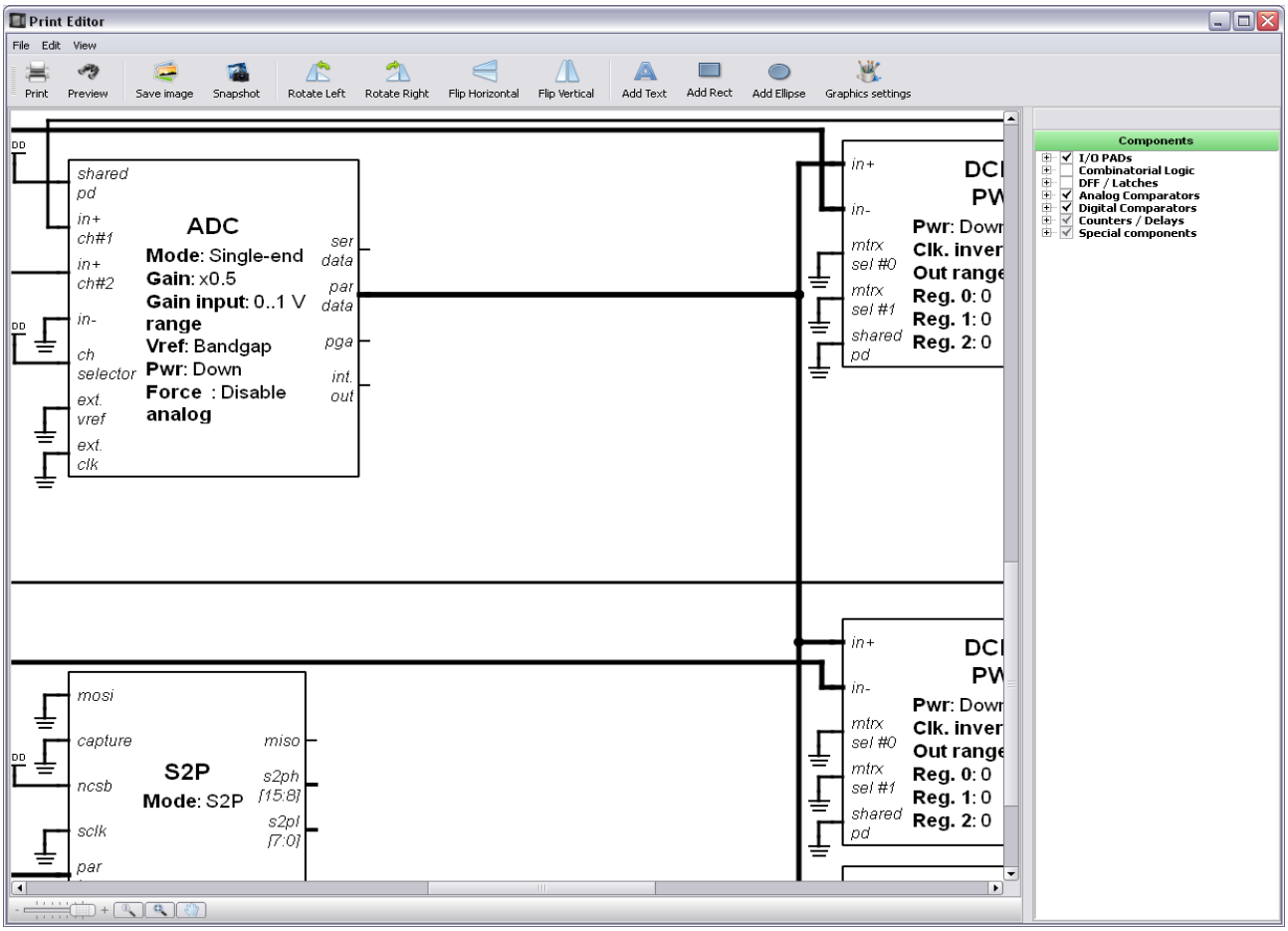
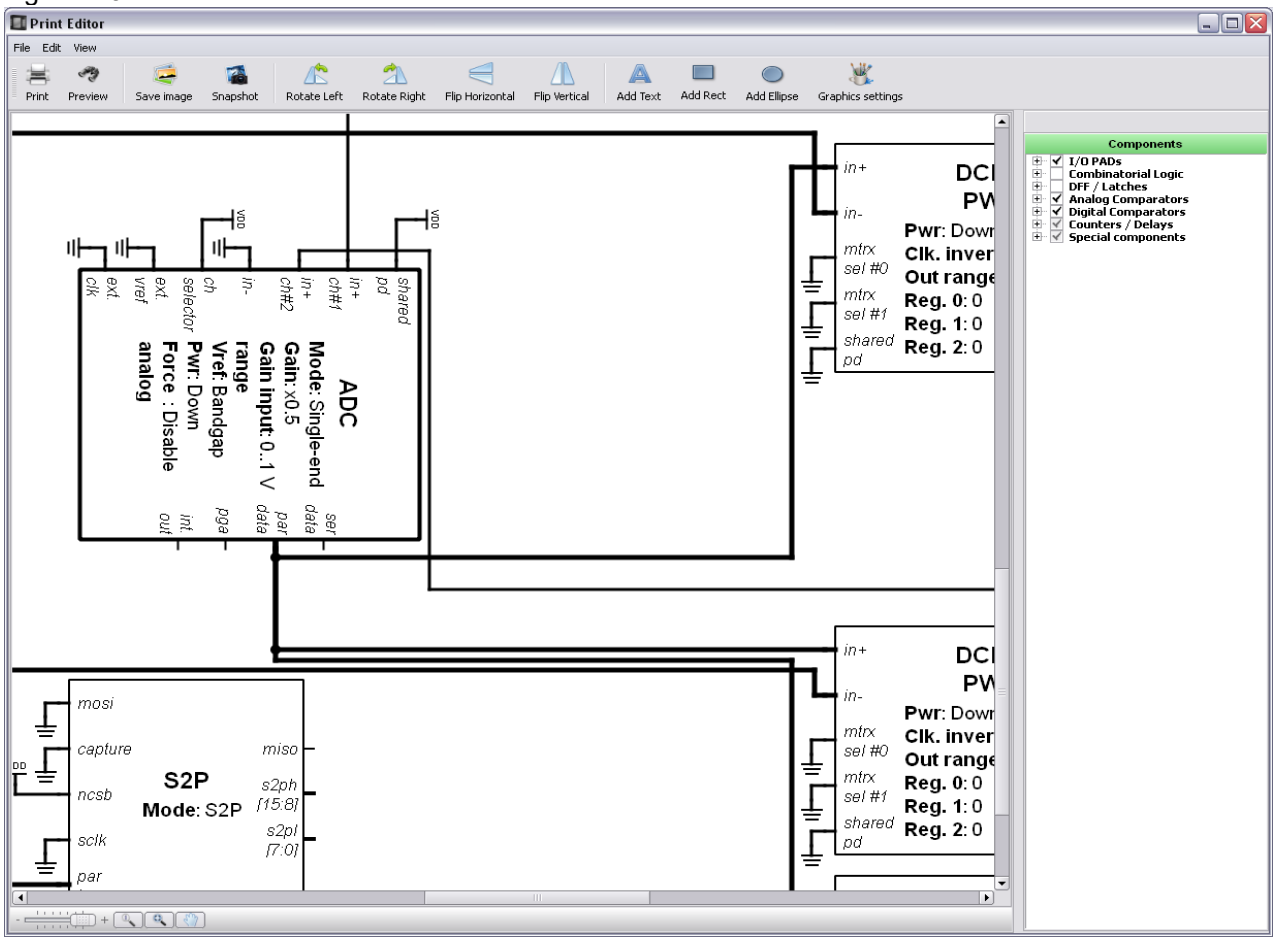
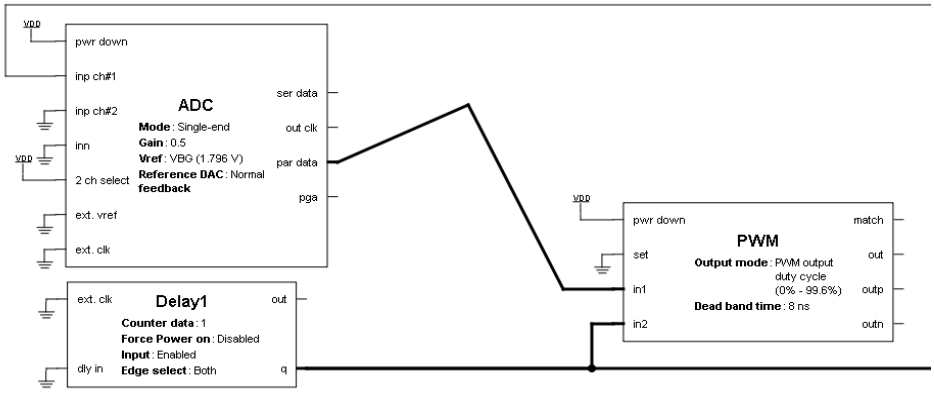


Figure 4-3. Preview Window



User can move lines and points to correct odd angled appearance.

Figure 4-4. Work sheet.



Working area

Working area can be zoomed in or zoomed out.
User can add a text label to the schematics using the text tool.
Figure 4-5. Text Label

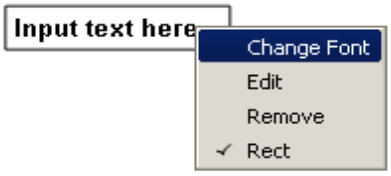
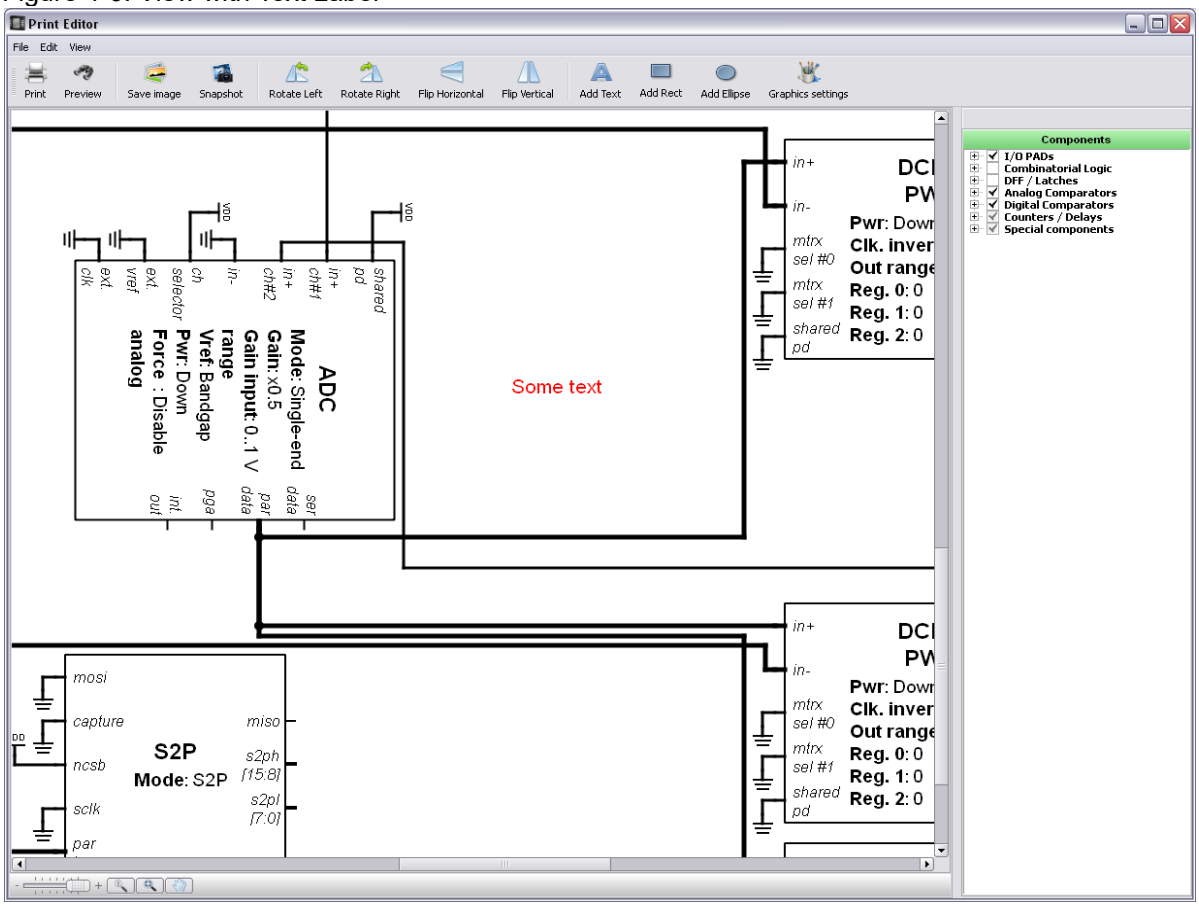
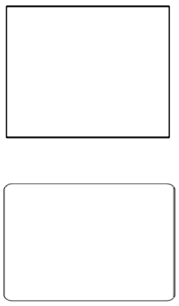


Figure 4-6. View with Text Label



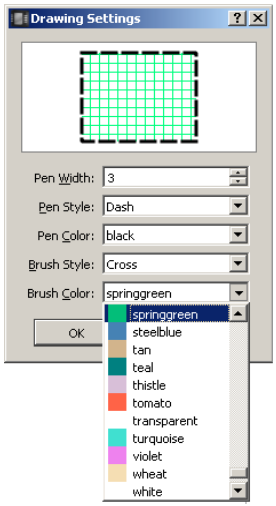
The user can add to the working area the custom figures including rectangle, rounded rectangle, ellipse etc.

Figure 4-7. Custom Figure



The user can also customize the main paint parameters.

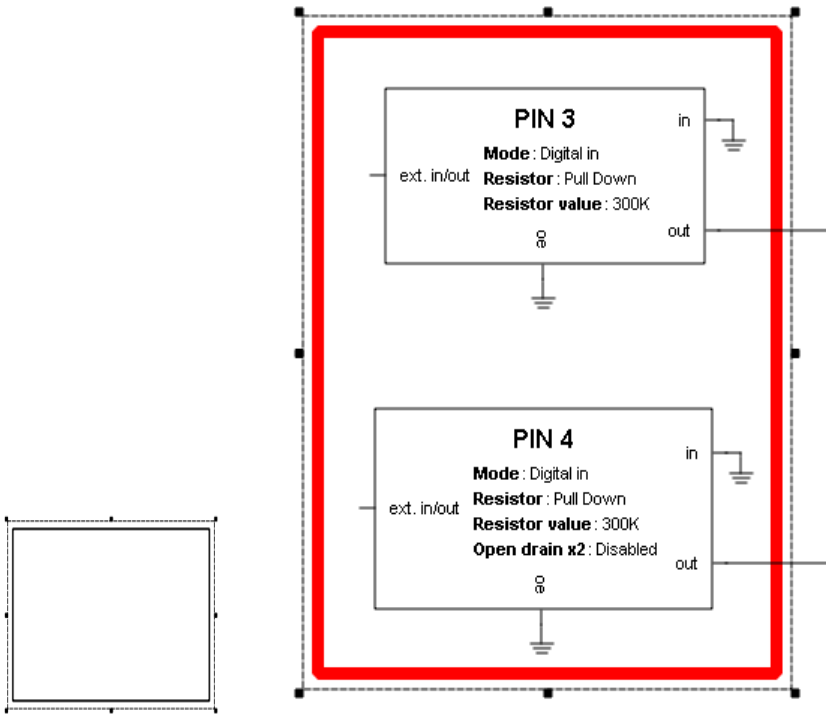
Figure 4-8. Paint Parameters



You can see a small preview window which includes a painted rectangle with user-parameters.

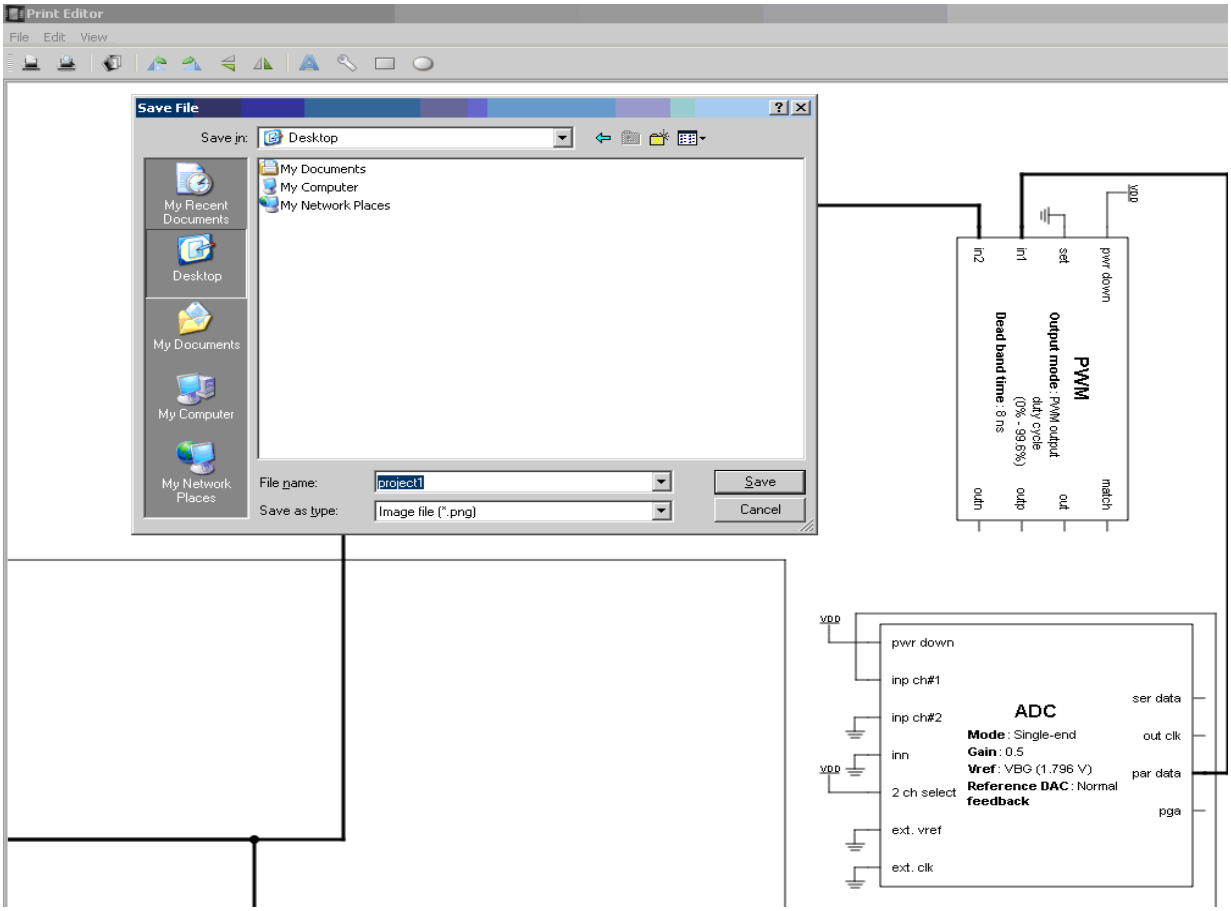
When the user adds a figure to the working area, one can customize the figure size by dragging black points on the corners and sides. The user can view it only by moving the mouse pointer up to the figure.

Figure 4-9. Work Area



The user can save a composed diagram into a graphics file or directly send it to the printer.

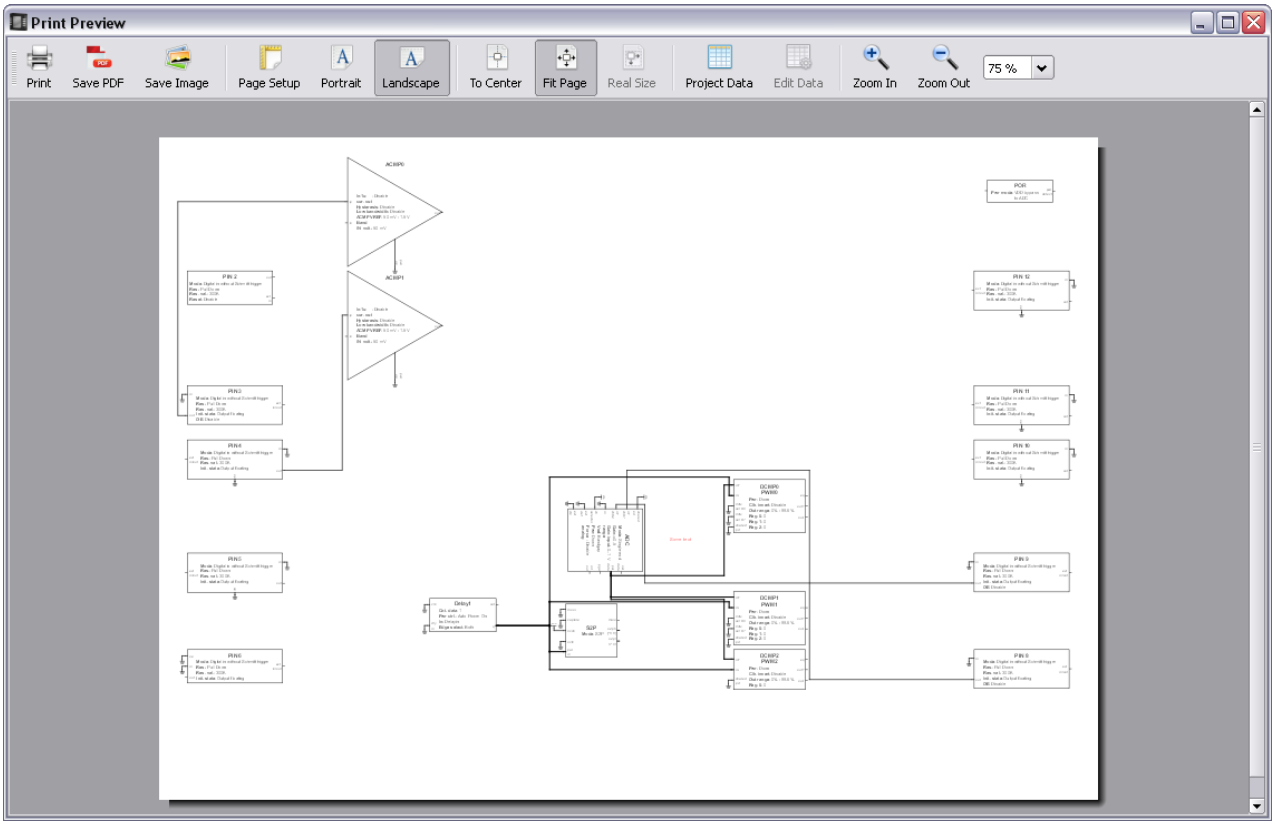
Figure 4-10. Save diagram



4.2. Print Editor Preview Window

Preview window shows the composed and ready-to-print diagram. In this window, the user cannot change the position of the components or the other elements in the diagram. The user can only choose the advanced settings for printing or saving to the file.

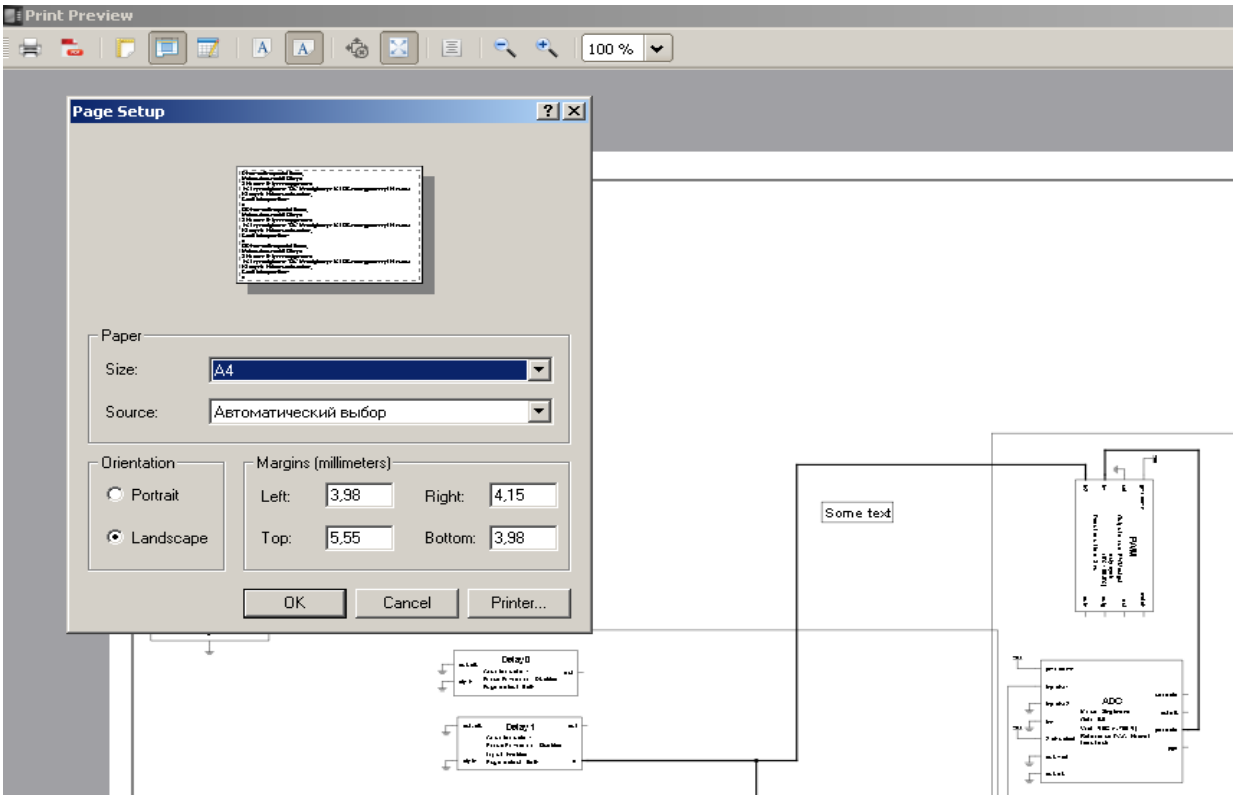
Figure 4-11. Preview Window



Main Actions
The user can:

- Choose orientation of the diagram on a paper (landscape or portrait)
- Fit the diagram to a page or keep the real size
- Fit to center
- Zoom in or zoom out
- Choose the size or type of paper
- Save the finished diagram into a PDF/Image file or print it out

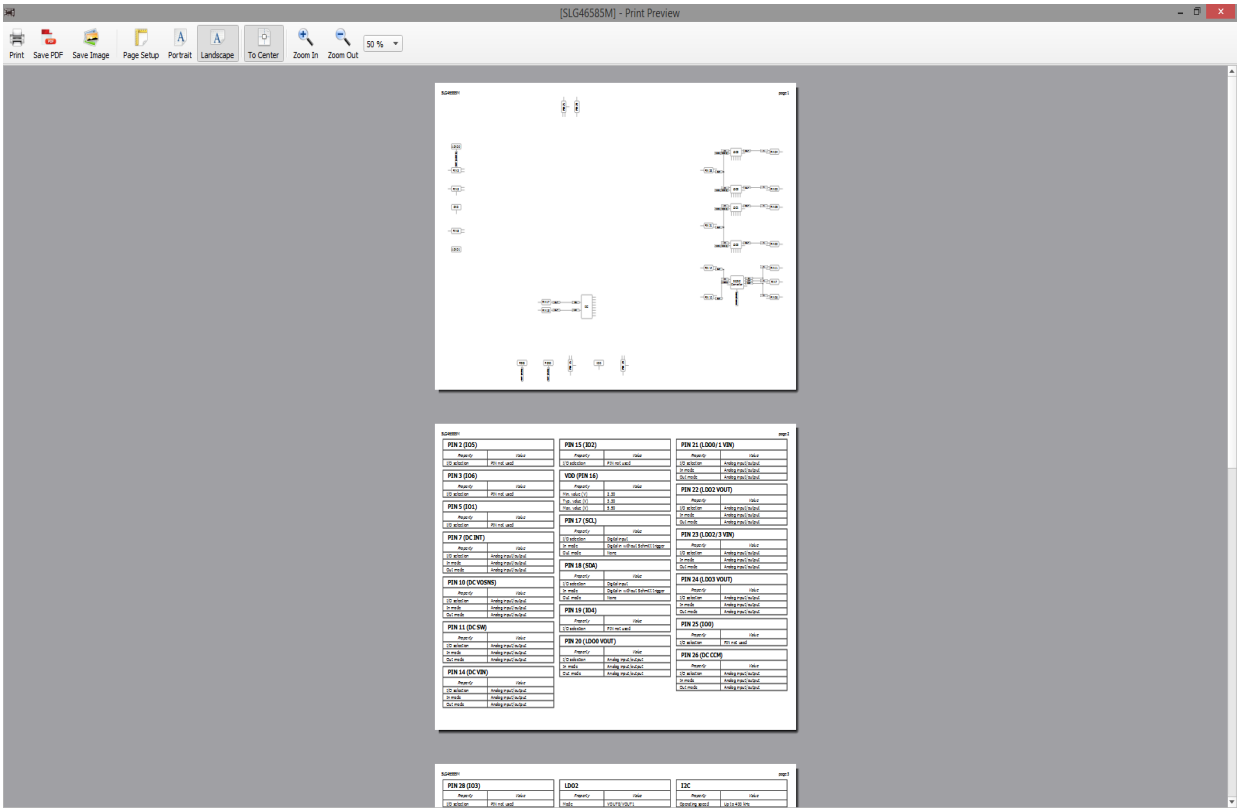
Figure 4-12. Page Setup



4.3. Print

Print window shows the composed and ready-to-print diagram with block properties and its values. In this window, the user cannot change the position of the components or the other elements in the diagram. The user can only choose the advanced settings for printing or saving to the file.

Figure 4-13. Print window with block properties and values



Print options:

- Choose orientation of the diagram on a paper (landscape or portrait)
- Fit diagram to center
- Zoom in or zoom out
- Choose the size or type of paper
- Save the finished diagram into a PDF/Image file
- Print diagram and block properties

5. Rules Checker

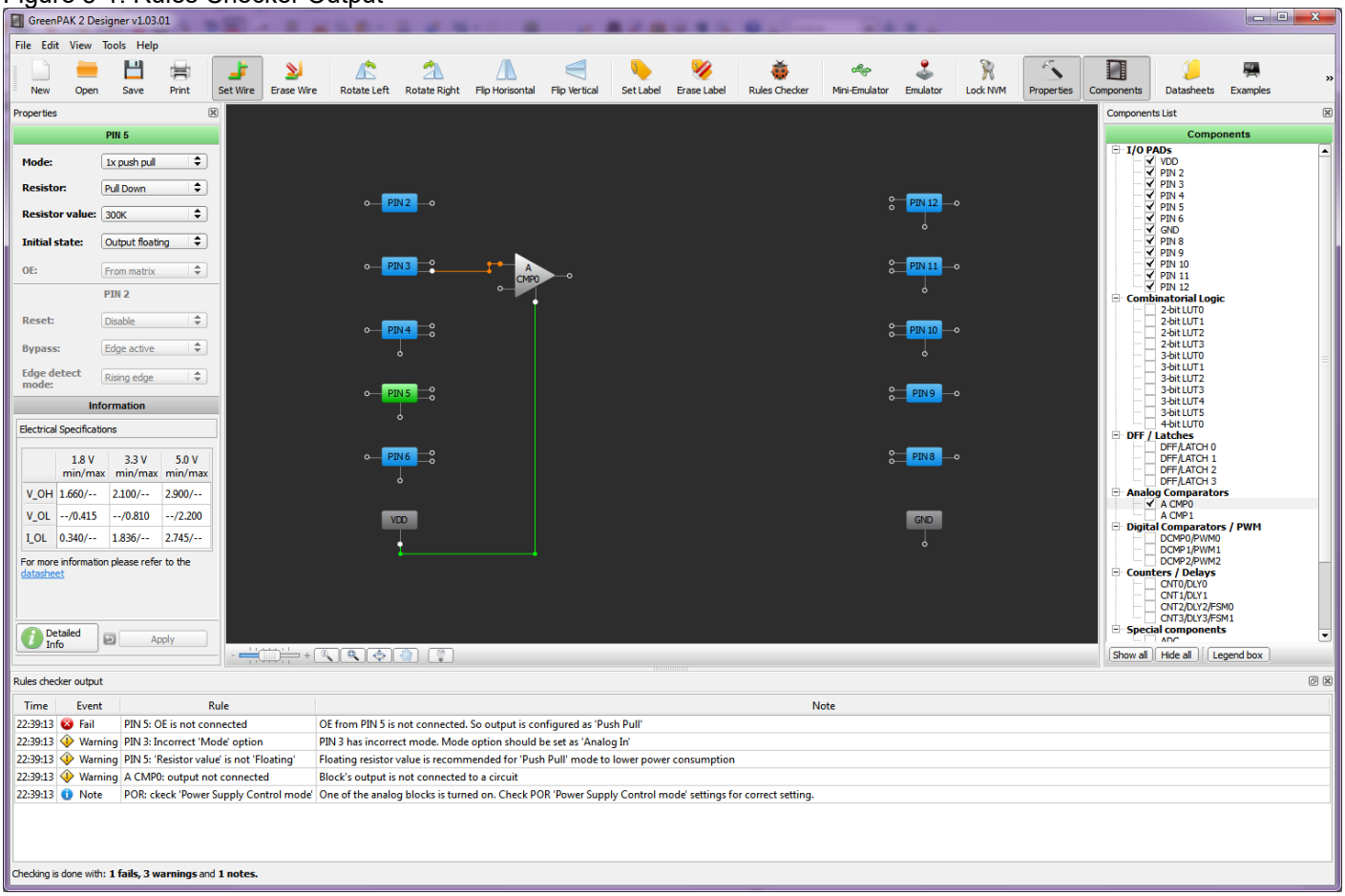
This tool allows checking current project errors, for example, incorrect block connections or settings. Rules Checker has three types of messages:

Fail - this message is generated when there is a significant error in design that will not work under any conditions.

Warning - this message is generated when one or more blocks may contain incorrect connections or settings in the design. This does not mean that there is an error. It only notifies the user to check the connections or settings of the blocks.

Note - this message is generated to remind the user to check for correct settings.

Figure 5-1. Rules Checker Output

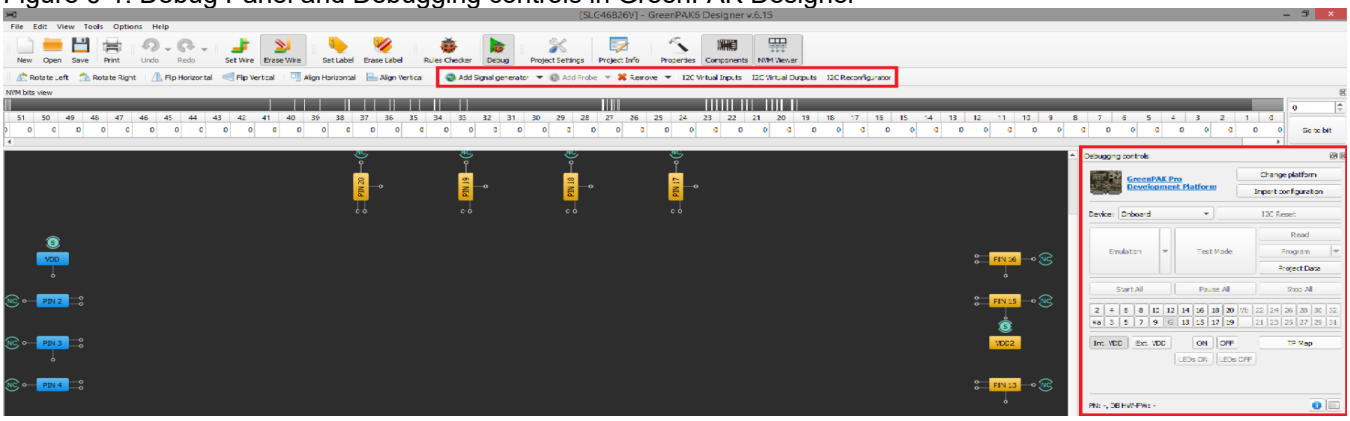


In order to check the design, click the Rules Checker button on the tool bar in Tools menu.
Rules Checker Window can be called by clicking Rules checker output in View menu.
Rules checker output consists of three parts:

1. Event – shows message type (Fail, Warning, Note).
2. Rule – information about the message.
3. Note – recommendations on how to correct the error or error explanation.

6. GreenPAK Debug Tool

Figure 6-1. Debug Panel and Debugging controls in GreenPAK Designer



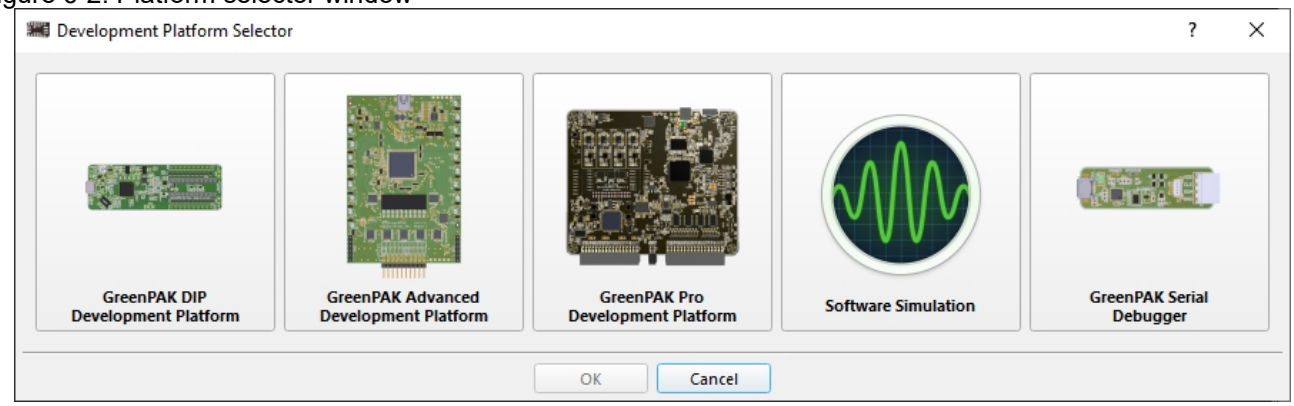
Debug

Debug button starts Debug tool in GreenPAK Designer (Figure 6-1). The Debug tool enables electronic circuit emulation and chip programming, which uses specific hardware platform to replicate the behavior of chip components included in GreenPAK. Before starting the emulation process, add test points controls to configure the emulation process.

Type of hardware platform

After start of Debugging tools select type of hardware platform with supported features (Figure 6-2):

Figure 6-2. Platform selector window



Platfrorm Configuration Guide

Recommended Platform Configuration Guide contains information about suitable sockets, adapters and boards for the specific chip. The user can pop up the guide by clicking on platform's name into Debugging controls panel.

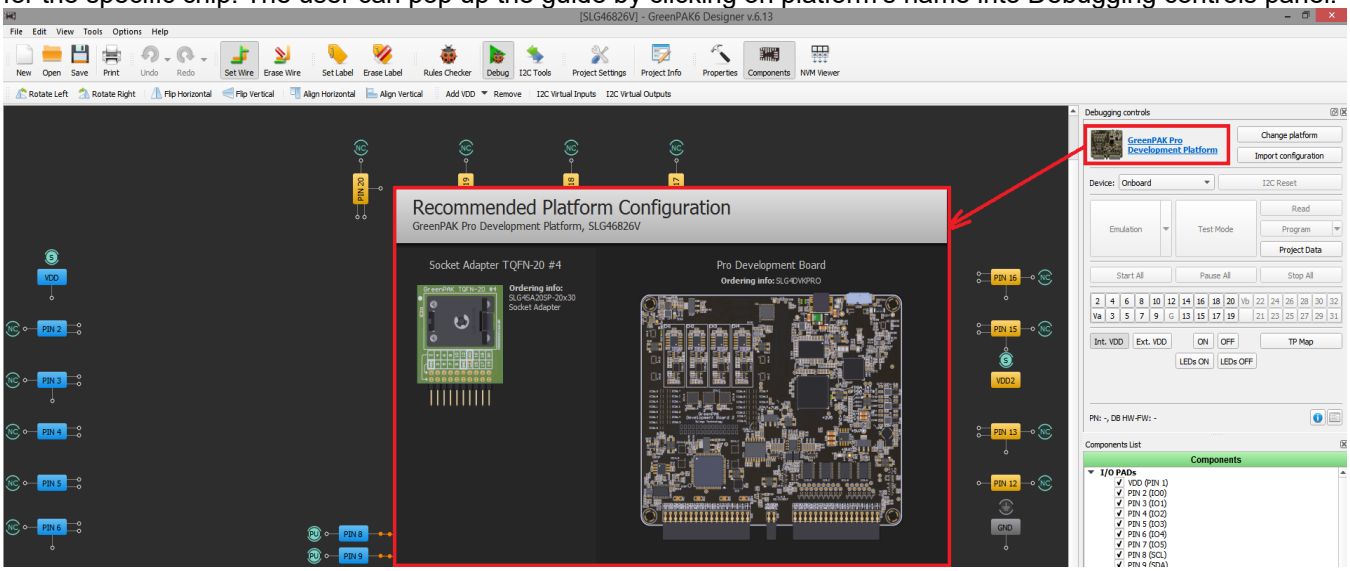


Figure 6-2. Recommended Platform Configuration Guide

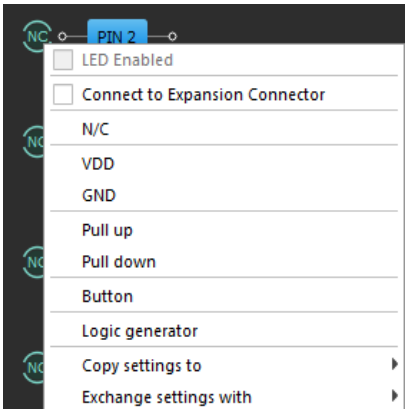
Add test points controls

Debugging tool controls are used to configure input signals on external inputs of chip (proper test points on development platform).

To manage chip input signals:

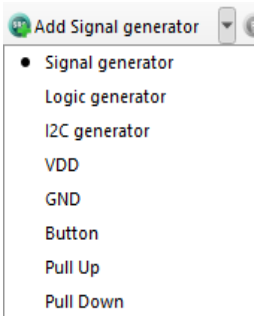
- Use the context menu on input pin with right mouse button click on NC (not connected) symbol (Figure 6-3).

Figure 6-3. Add control context menu



- Add control to TP with Add button on Debug panel (Figure 6-4). List of available controls on TP: Signal generator/Logic generator/VDD/GND/Button/Pull up/Pull down.

Figure 6-4. Add button on Debug panel



Remove test points controls

Remove button removes test points controls by mouse click on it. TP become NC. Also user can set TP control as N/C from context menu by clicking with right mouse button.

6.1. Types of Areas

Fixed Inputs (Figure 6-5 – 6-9).

Figure 6-5. N/C (not connected)

Figure 6-6. Set to VDD

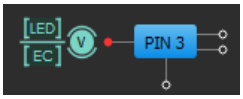


Figure 6-7. Set to GND

Figure 6-8. Pull Up

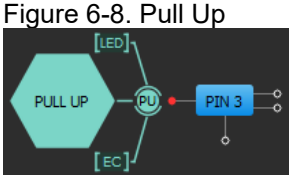
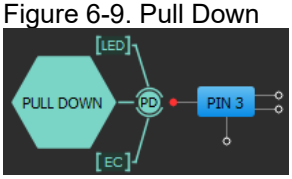


Figure 6-9. Pull Down

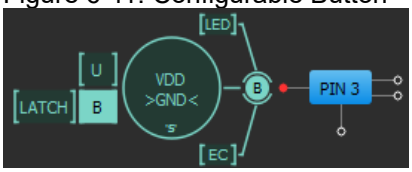


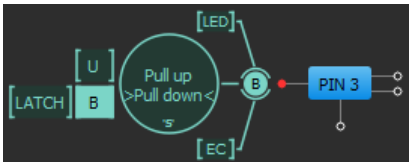
LED's (Figure 6-10)

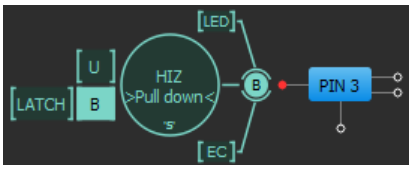
Figure 6-10. Buffered LED, Buffered LED+Pull Up, Buffered LED+Pull Down

Configurable Input (Figure 6-11)

Figure 6-11. Configurable Button

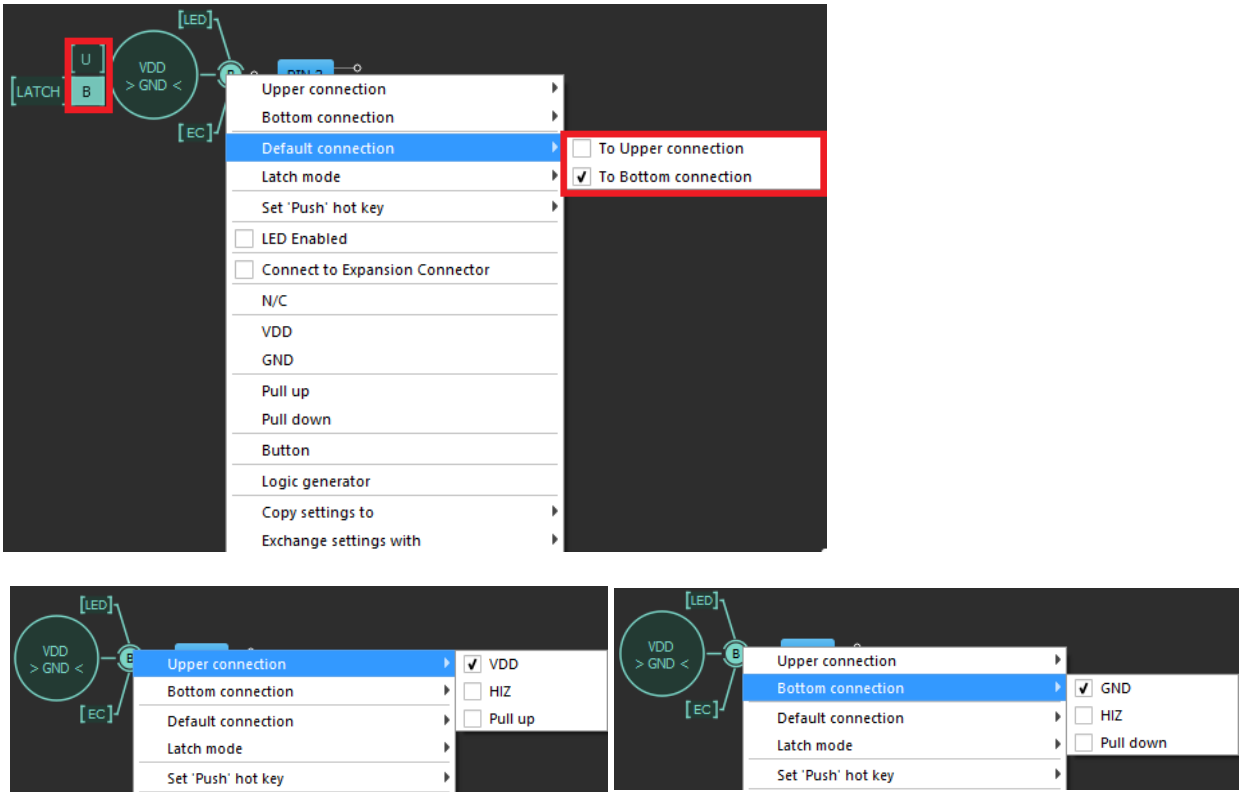






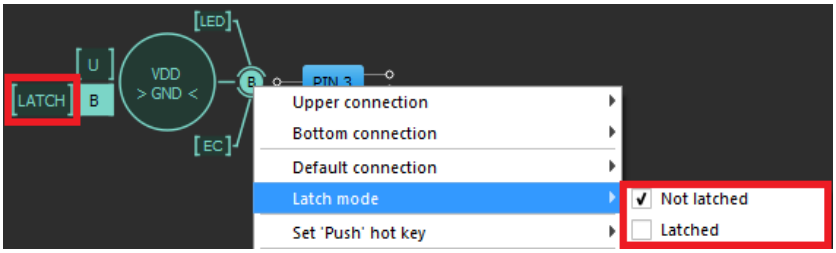
The default connection can be set to either Upper connection or Bottom connection. Click your mouse over the key U or B to change the value.
The user can configure each connection to VDD/GND, High-Z or Pull Up/Down.

Figure 6-12. Default Key Connection



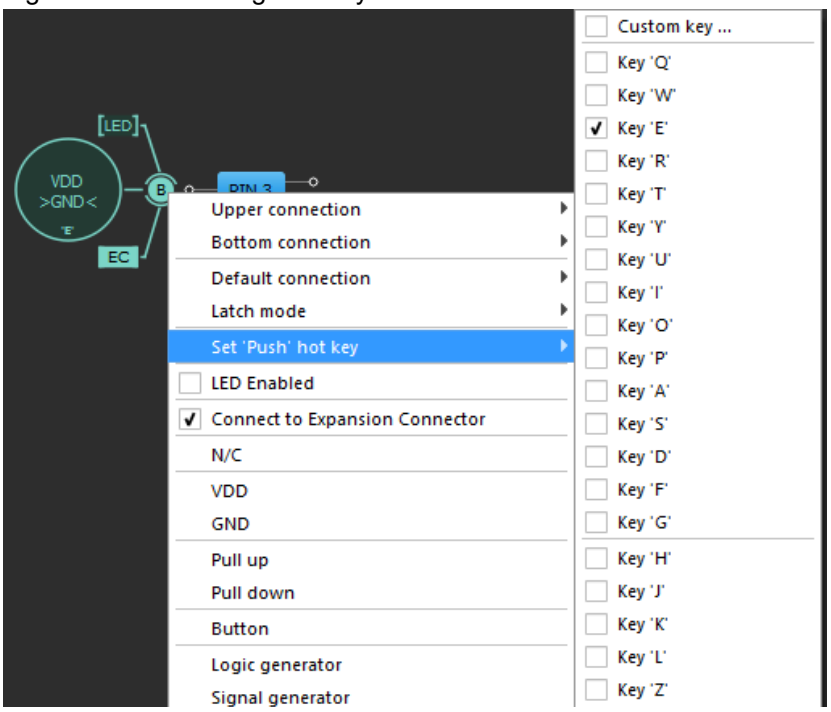
The switch has 2 modes: Latched, Unlatched, which can be configured from the context menu or click your mouse over the key LATCH to change the value.

Figure 6-13. Key Mode



User can assign Hot Key for 'Push' action. The assigned key will simulate mouse click over the key:

Figure 6-14. Choosing Hot Key

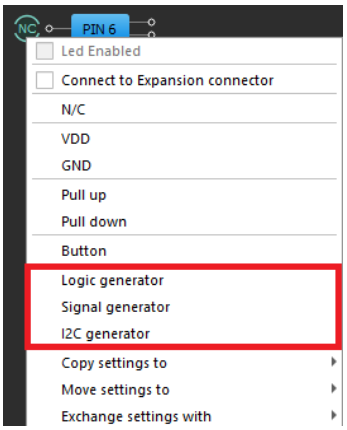


User can assign the same hot key to other Switches which allows changing the key values of all the Switches with the same hot key at once.

6.2. Generators

To defined TP can be connected 3 types of generators: Logic generator, Signal generator or I2C generator. User can add generator with Add button to highlighted green pins or use context menu of the TP.

Figure 6-15. Choosing Generators

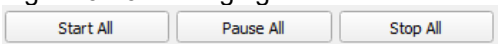


Each generator has its own settings. For the settings window to appear press the Edit button or double click on S or L or SDA/SCL symbol at TP.
On the left, you can see the options table divided into 2 groups:

1. General – applied to all types of generators
2. Special for each generator

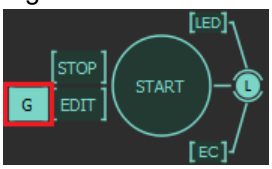
Start all generators with buttons at Debugging controls panel (Figure 6-16)

Figure 6-16. Managing Buttons



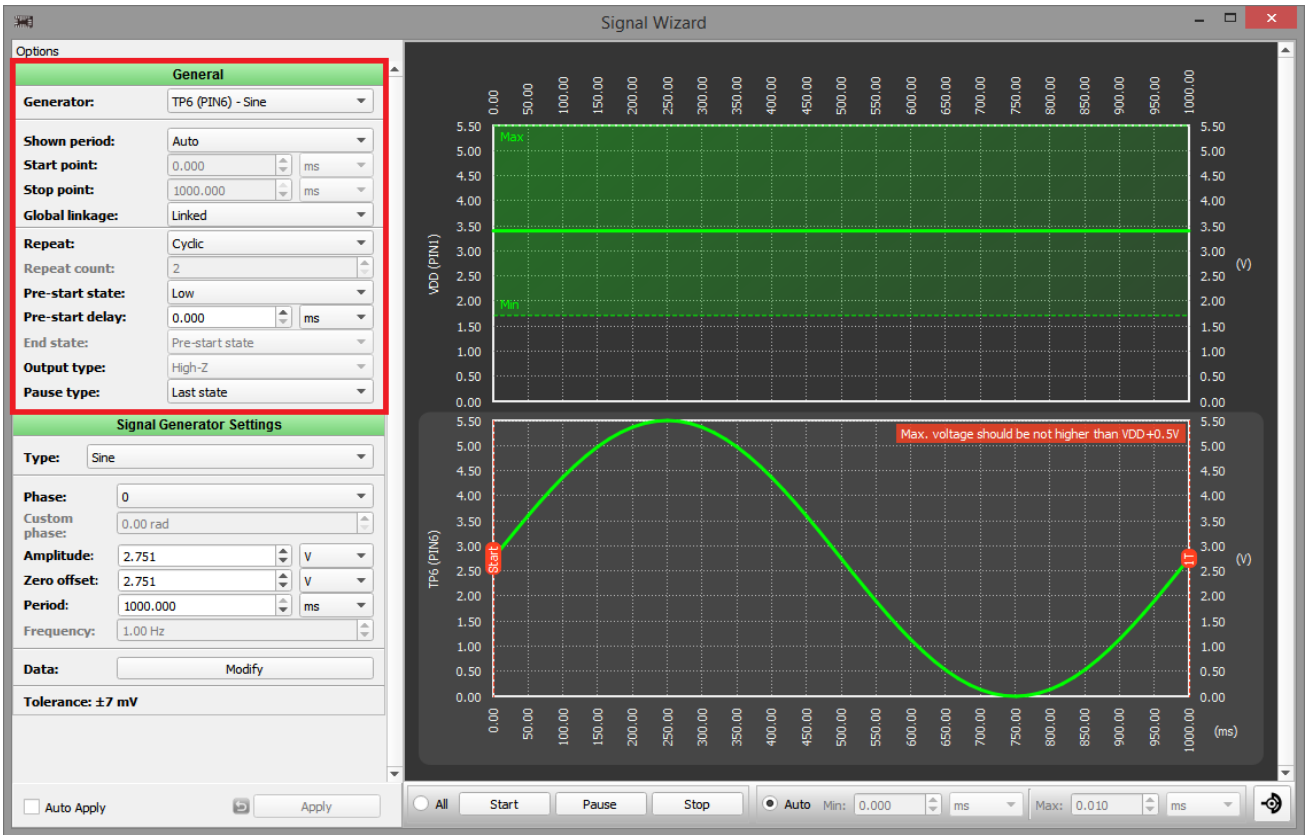
Note: these buttons can be controlled only by generators with an installed Global Linkage flag.

Figure 6-17. Global Linkage



6.2.1. General Options in a Signal Wizard Mode

Figure 6-18. General Option



Generator:		- generator selector
Shown period:	Auto/Custom/1T/2T/3T/4T	- set the period of a current generator to be displayed
Global linkage	Linked/Unlinked	-if generator is linked, it will be controlled by buttons “Start”, “Stop” and “Pause” on the Debugging controls
Repeat	One shot/Cyclic/Custom	- repeat option
Prestart state	Low/Start point(V0)/High-Z	-state before start
Prestart delay		-delay before start
End state	Keep last state/Prestart state	-pin state after generation
Output type	High-Z/Strong Drive/Open Drain, Drives High/ Open Drain, Drives Low/Resistive Pull Up/ Resistive Pull Down/ Resistive Pull Up/Down	- type of output
Pause type	Last state/Low/High/High-Z	-state when it is paused

6.2.2. Period Modes

AUTO Mode

All generators with 'AUTO' option have one scale; this scale = MAX period of all generators with 'AUTO' option.

Figure 6-19. One Scale for All Generators

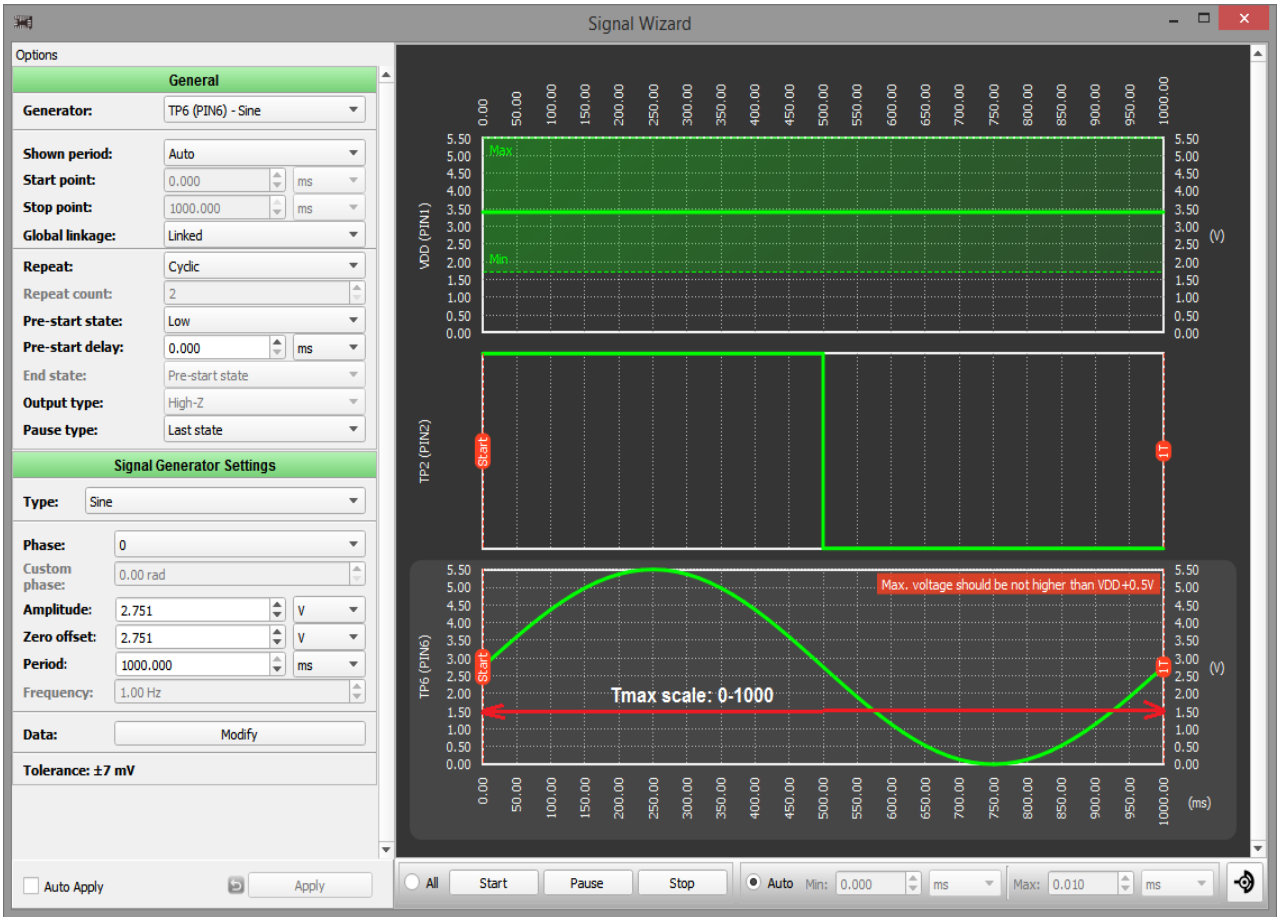
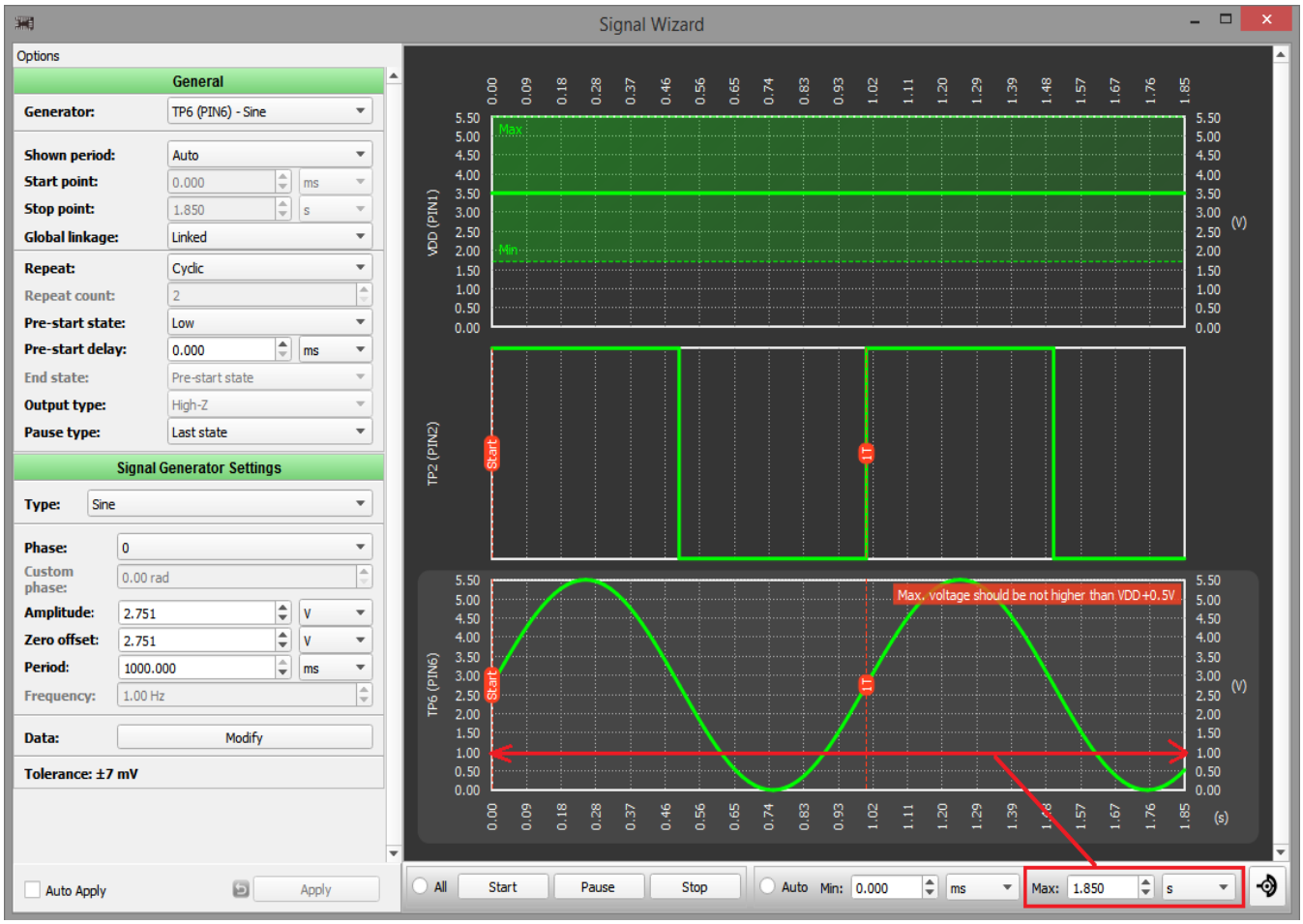


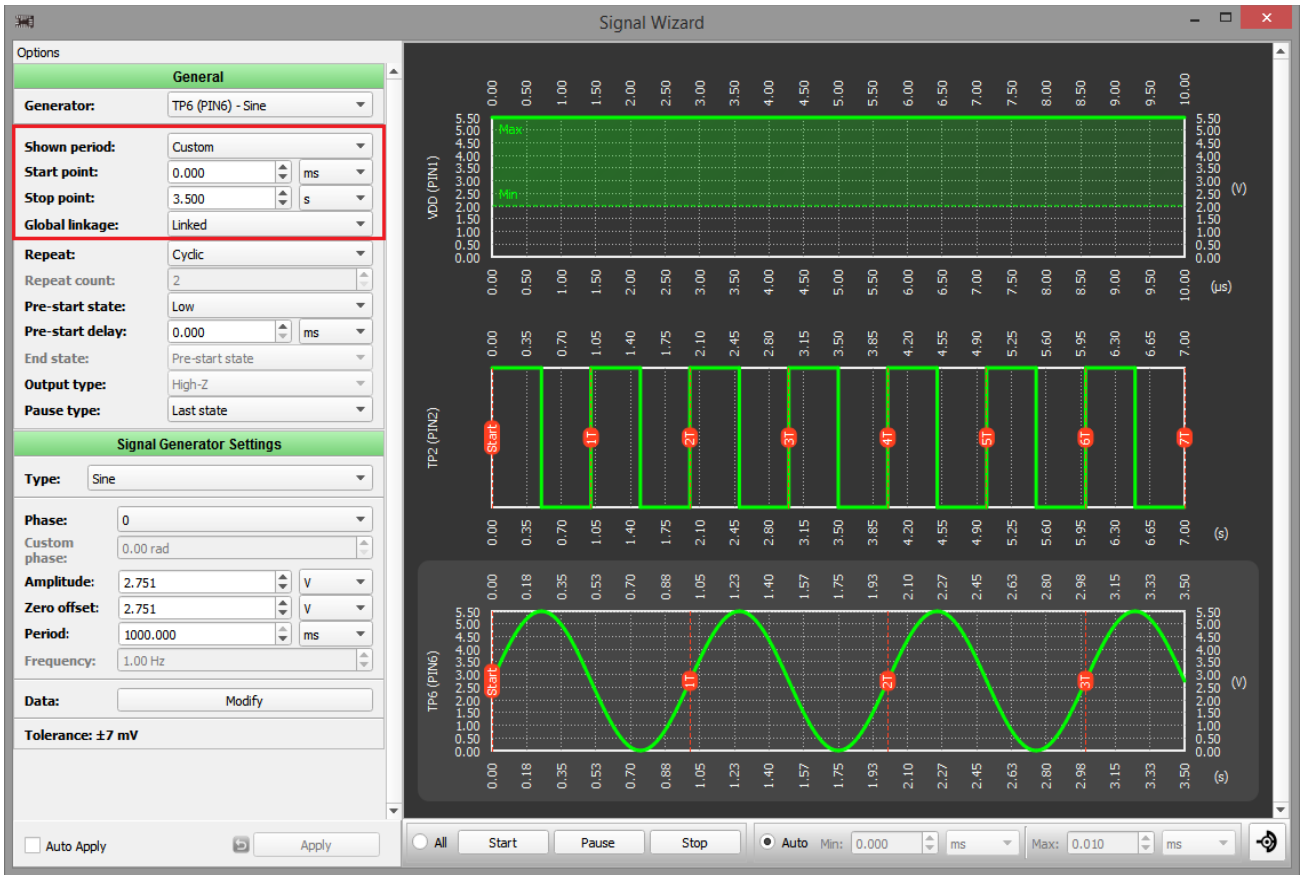
Figure 6-20. User Can Change the Scale Manually



button turns on/off the mouse coordinates in the timing diagrams.

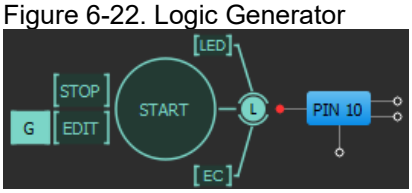
CUSTOM Mode

Figure 6-21. Choosing Period Options



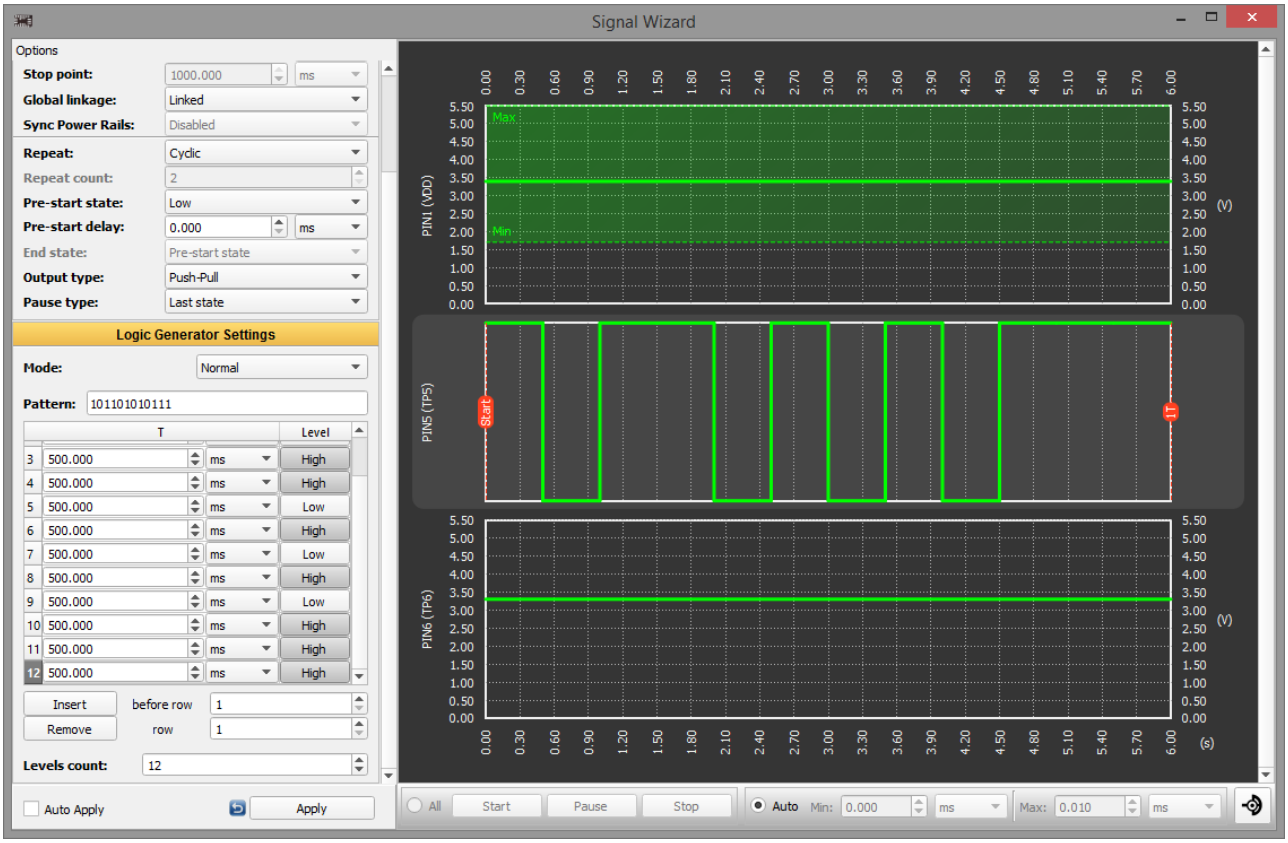
6.2.3. Logic Generator

Figure 6-22. Logic Generator



Logic generator is used for generating the logic pulses.

Figure 6-23. 'Edit' button allows configuring the signal.

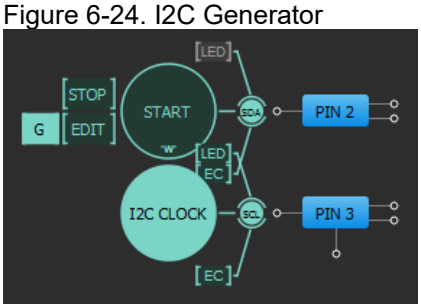


Configuration options:

Mode:	Normal/Invert	- signal mode
Pattern:	0 – low/ 1- high level	- pattern of pulse levels
Repeat:	One shot/Cyclic/Custom	- repeat option
T/Level:		- sets duration of level
Insert:		- insert pulse before selected position;
Remove:		- remove pulse from the selected position;
Levels count:		- pulse count

6.2.4. I2C Generator

Figure 6-24. I2C Generator



I2C generator allows a user to make an I2C signals based on logic generators. There are two logic generators combined together as SDA and SCL lines. User can combine predefined I2C primitives to generate the needed waveform in an appropriate way and choose SCL frequency.

SCL

Generates SCL signal applicable for I2C. SCL signal is the special kind of logic generator applicable only for board configuration. SCL is only 'read-only'. The SCL clock configured by choosing predefined frequency. The set of those frequencies depends on the development platform.

SDA

Generates configured SDA signal applicable for I2C. SDA signal is the special case of logic generator applicable for sending data via I2C. In Signal Wizard special editor shows sequence of commands. User can do some actions in command editor:

- change SCL pin for selected SDA;
- change Speed of I2C clock;
- control resource meter;
- change Slave Address for all commands in list;
- clear list to basic commands: Start and Stop;
- add or remove commands by using +/- buttons;
- change command parameters;
- split Composite commands in sequence of Basic commands by using button “S”;

Figure 6-25. I2C Generator commands editor

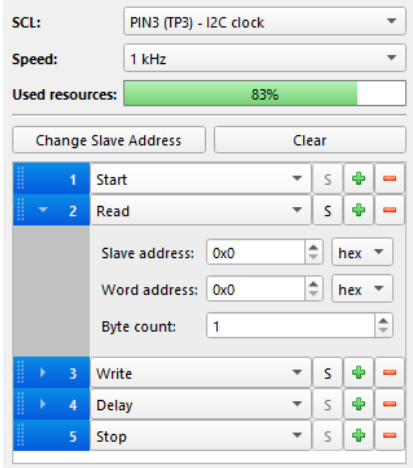
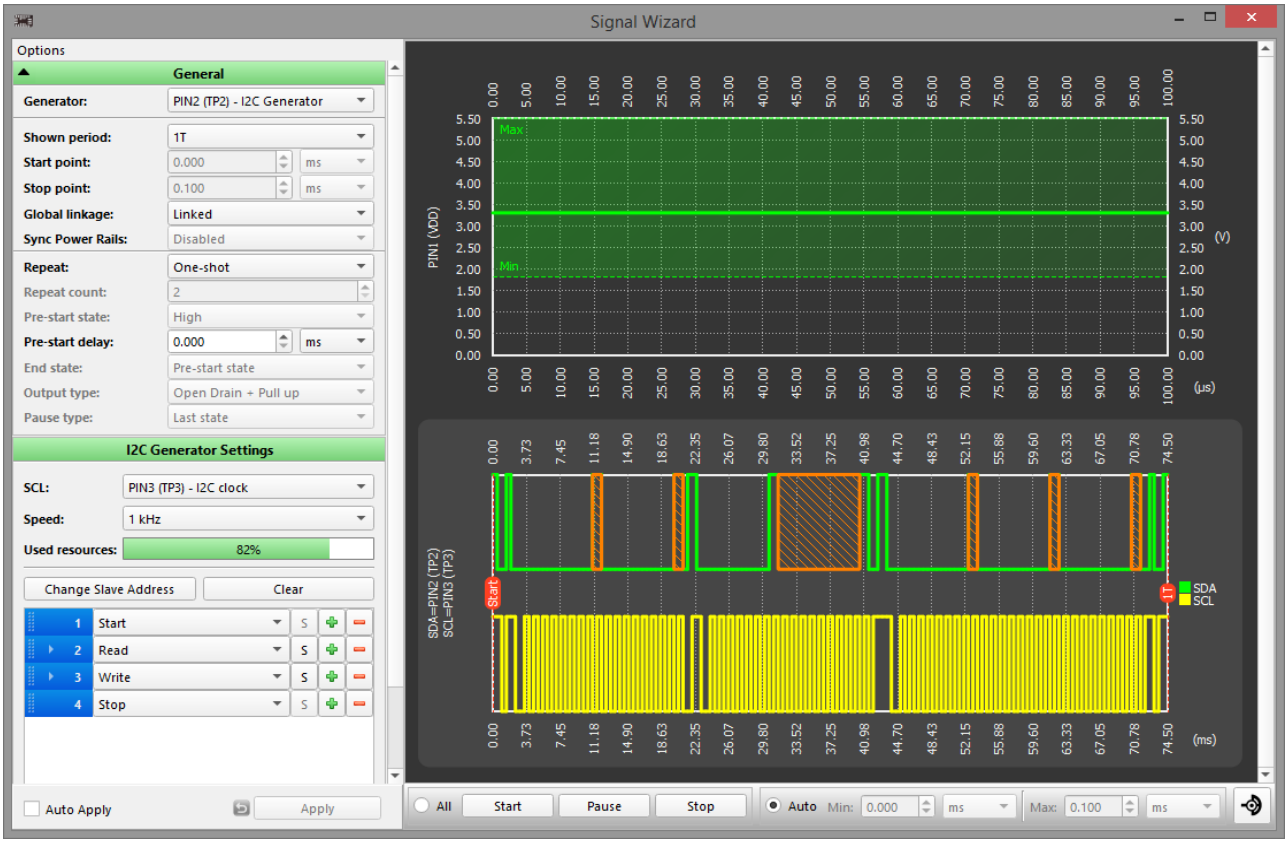


Figure 6-26. 'Edit' button or double click on SDA/SCL allows configuring the signal.

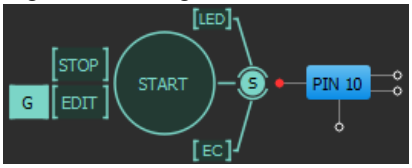


Configuration options:

SCL:	Any available pin	- selects pin to be the I2C clock channel with defined frequency
Speed:	1-1000kHz	- speed of I2C clock
Used resources:	%	- shows used resources after adding commands to list
Change Slave Address	hex	- changes slave address in all commands
Clear		- clears all commands from the list to Start and Stop
Commands		- list of I2C generator commands

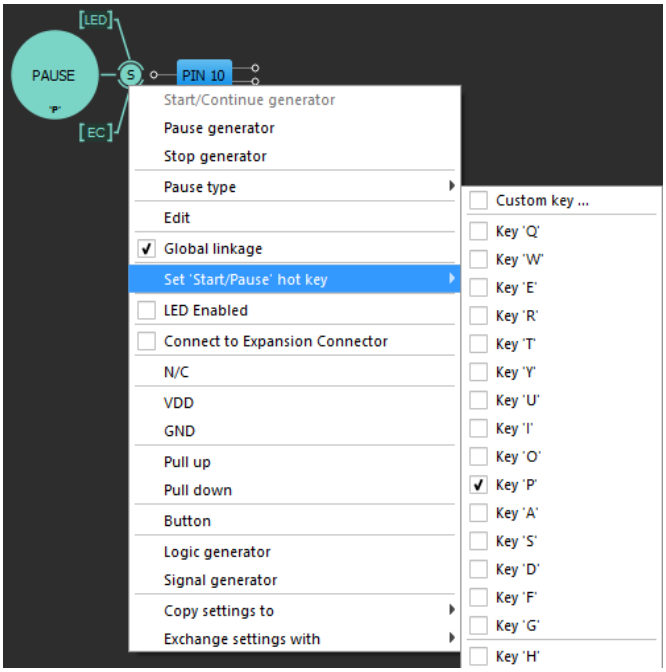
6.2.5. Signal (Analog) Generator

Figure 6-27. Signal Generator



Signal generator is used to generate analog signals: Constant Voltage level, Sine, Trapeze(Trapezoid), Logic pattern and User-defined.
Logic and signal generators can be started/paused/stopped using orange buttons or through the context menu. The user can also assign the hot keys for start/pause.
More than one generator can use the same hot button to start/pause at once. This is how to start more than one generator at the same time.

Figure 6-28. Sets Start/Pause Hot Key



Signal Generator Settings

Type: Const voltage level/Trapeze/Logic pattern/Sine/Custom - type of waveform;

Constant value:

U: - voltage level;

Figure 6-29. Constant Value

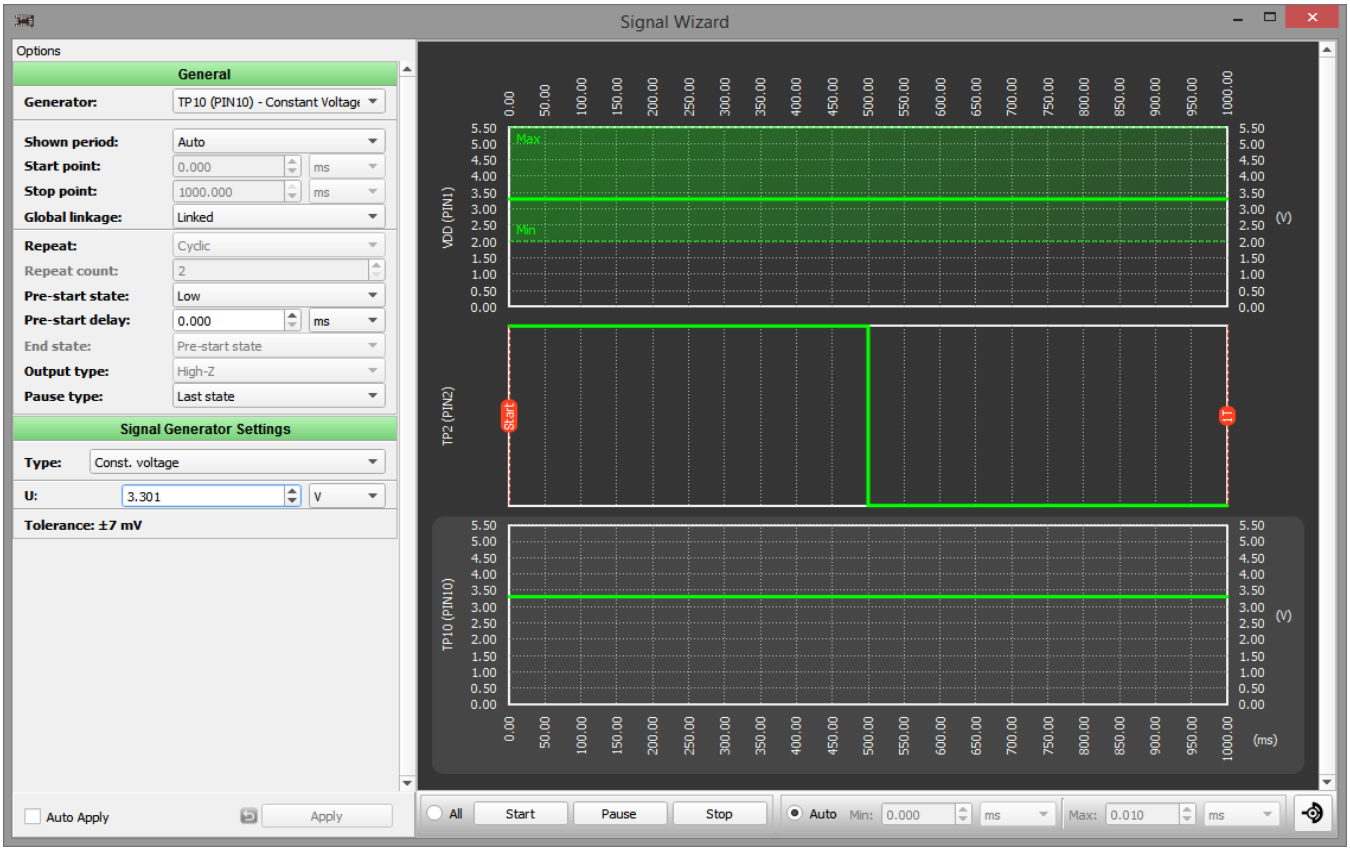
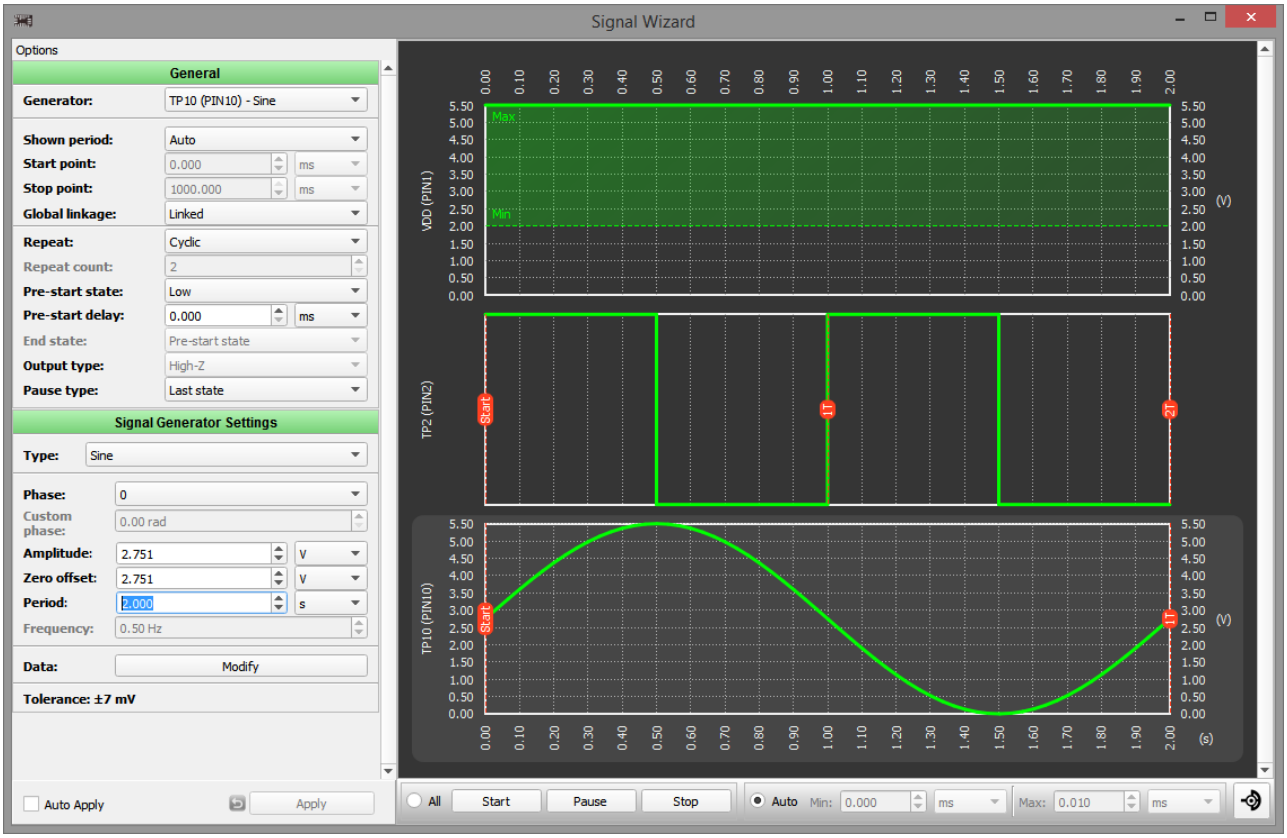


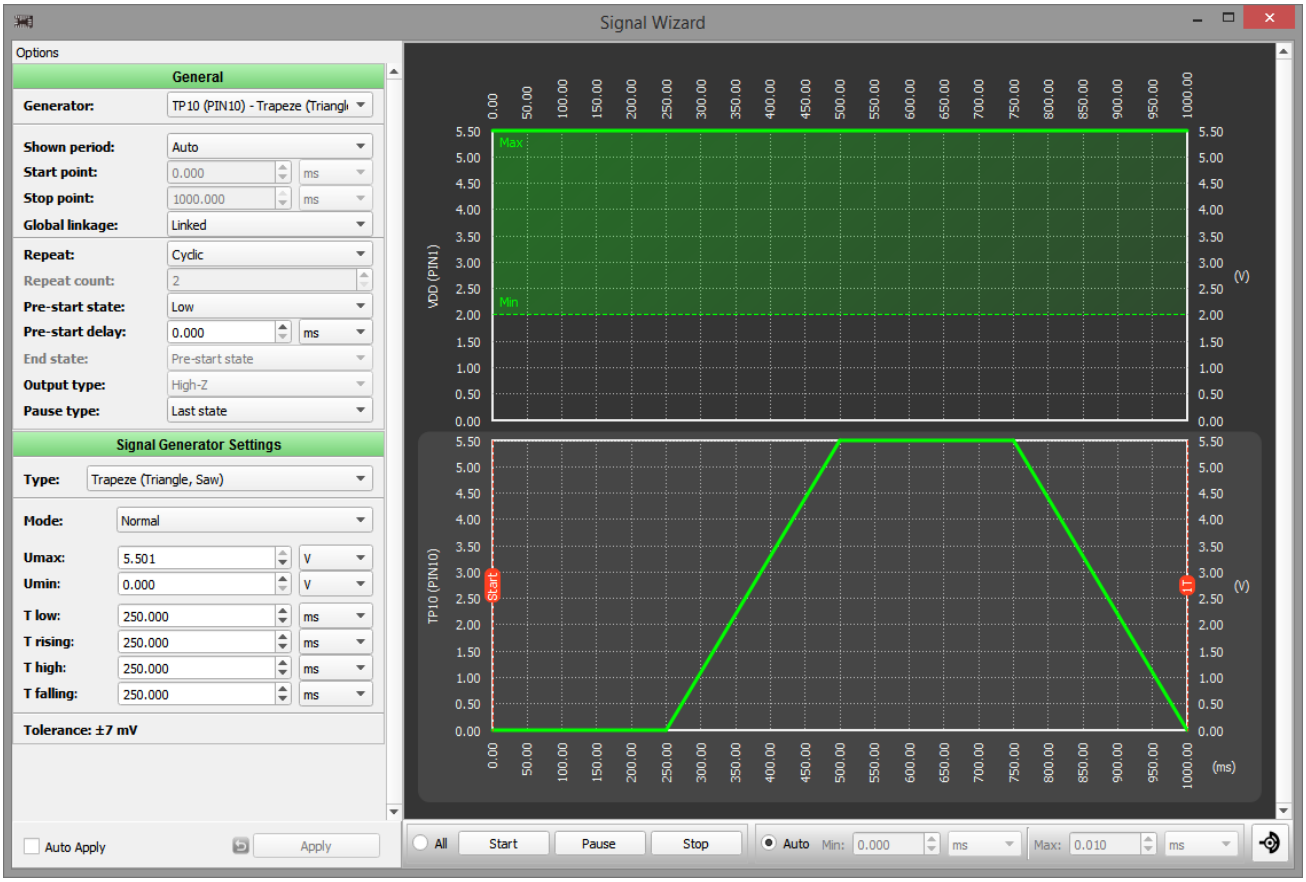
Figure 6-30. Sine



Sine settings:

Repeat:	One shot/Cyclic/Custom	- repeat option
Phase:	Custom/0/Pi:2/Pi/3Pi:2	- φ0
Custom phase:		- show phase in a radian
Amplitude:		- amplitude
Zero offset:		- zero offset
Period:		- period
Frequency:		- shows frequency
Data:		- change signal using Custom Signal Wizard

Figure 6-31. Trapezoid (Triangle, Sawtooth)



Trapeze Settings:

Mode:	Normal/Invert	-signal mode
Umax/Umin		-max/min voltage level
T1, T2, T3, T4		-duration of trapezoid

If T_3 = 1 signal is a triangle.
If T_3 = 1, T_2 = 2 or T_4 = 2 signal is a saw.

Figure 6-32. Duration of Trapezoid

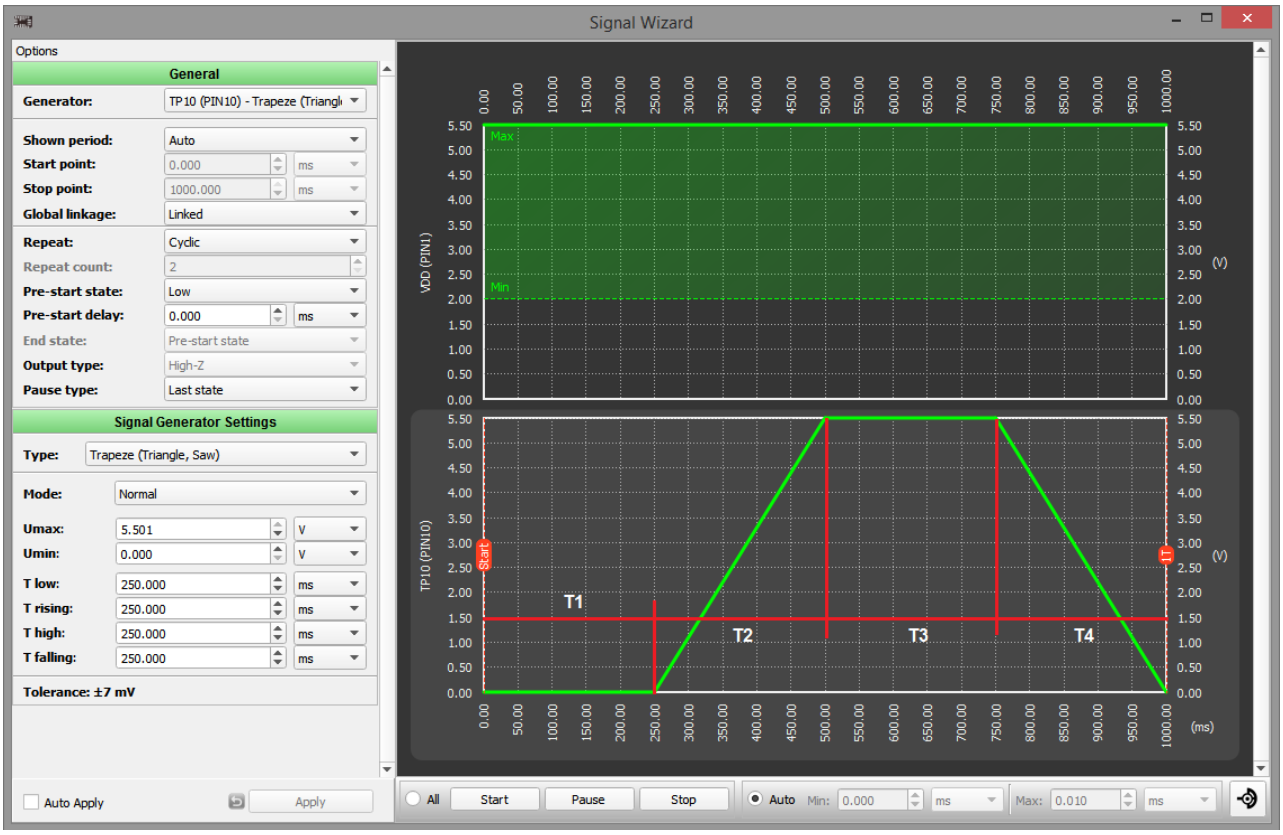
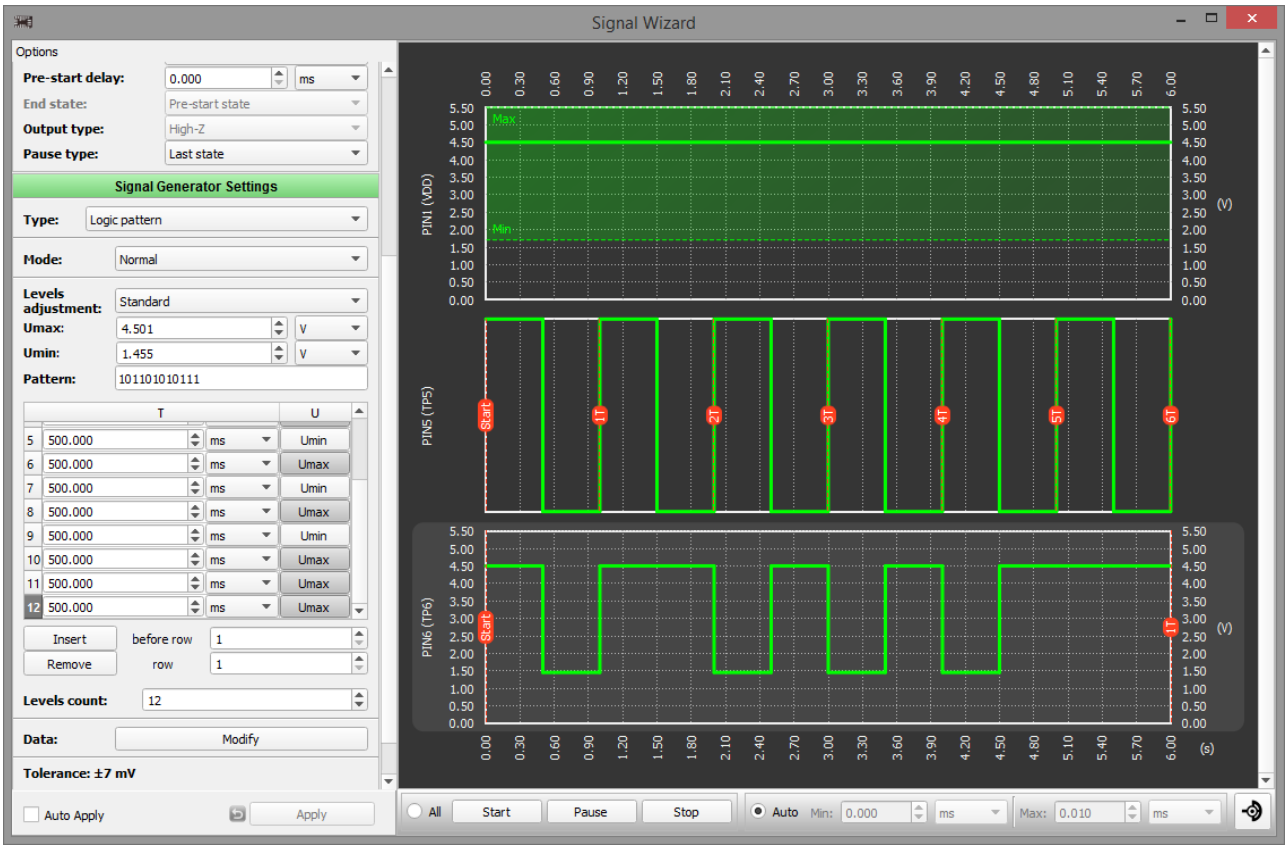


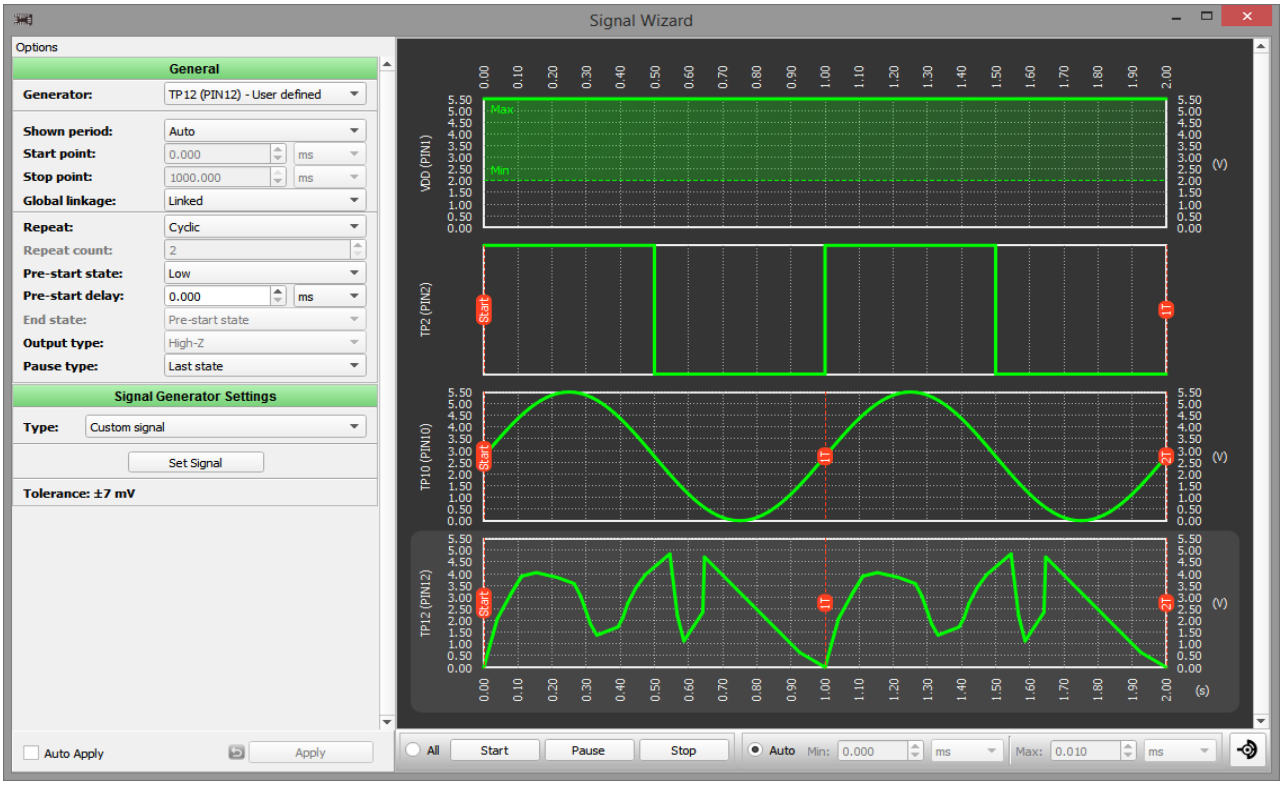
Figure 6-33. Logic pattern



Configuration options:

Mode:	Normal/Invert	- signal mode
Levels adjustment:	Standard/Custom	- sets standard Umax/min for all levels or U for each level
Umax/min:		- Umax/min for all levels
Pattern:	0 – low/ 1- high level	- pattern of pulse levels
Repeat:	One shot/Cyclic/Custom	- repeat option
T/Level:		- sets duration of level
Insert:		- insert pulse before selected position;
Remove:		- remove pulse from the selected position;
Count:		- pulse count

Figure 6-34. Custom Signal (Arbitrary waveform)



Custom Signal Settings:

Repeat:	One shot/Cyclic/Custom	- repeat option;
Data:	Set Signal	- change signal using Custom Signal Wizard

6.2.6. Custom Signal Wizard

Figure 6-35. Drawing Signal (Arbitrary waveform)

Toolbar:

Clear	- clear data
Add Point/Add Peak/Continuous Ramp/Remove Point	- draw mode
Data panel	- turn on/off the data table
Import points	- copy points from another application
Lock X/Y	- lock point position for X or Y axes
Close	- close window with current signal

Figure 6-36. Peak

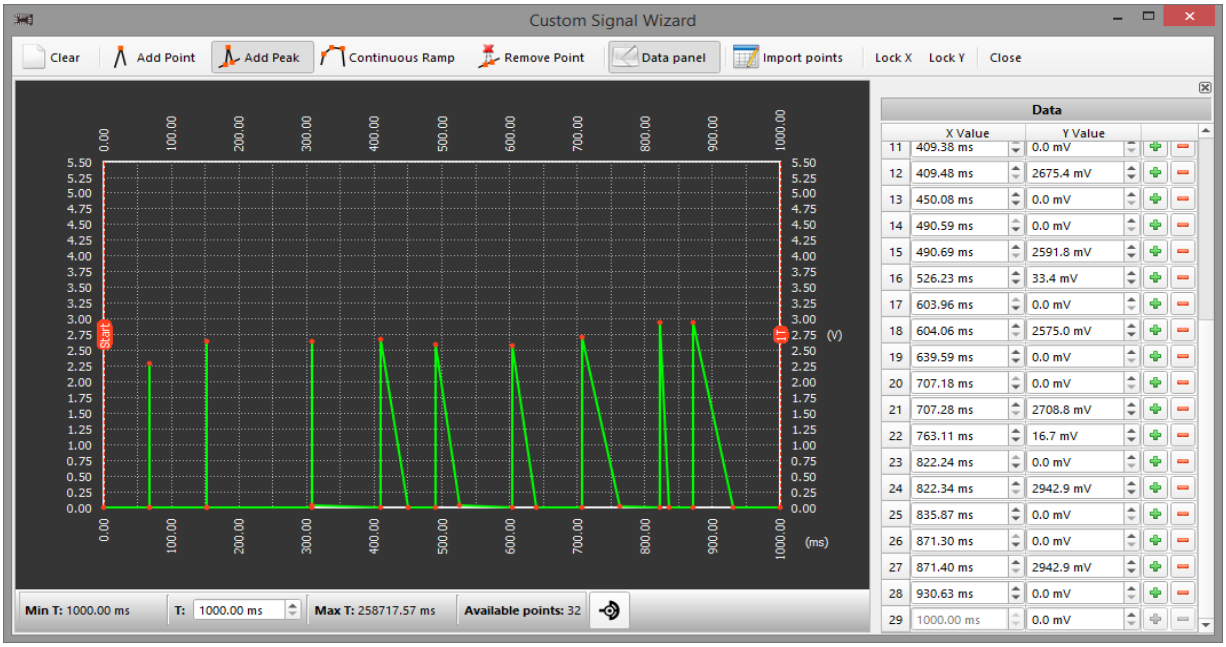
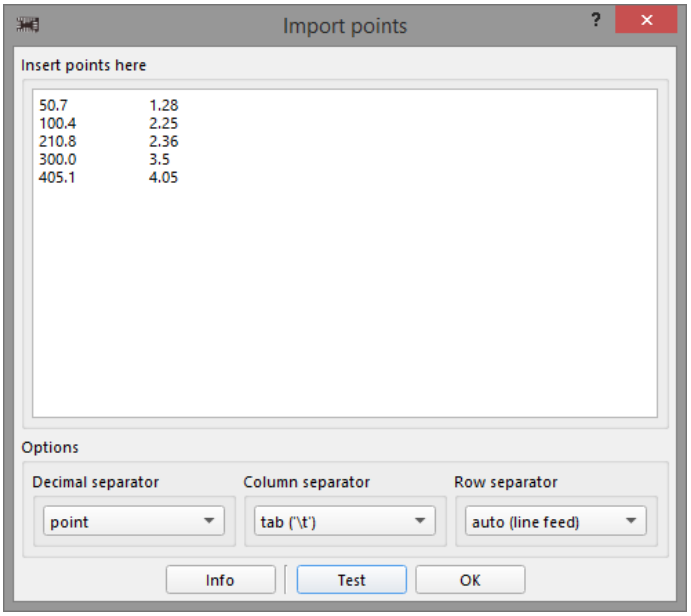


Figure 6-37. Continuous Ramp

Remove Point: Removes selected point. Double-clicking on the point will also remove it.
Data panel: Turns on/off the data table. User can remove/change values for selected point or add new point between two existing points.

Figure 6-38. Import Points



User can insert points from another application and set separators options:
Decimal separator: point/comma;
Column separator: auto/tab(“\t”)/other;
Row separator: auto(line feed)/ tab(“\t”)/other;

Figure 6-39. Data Panel

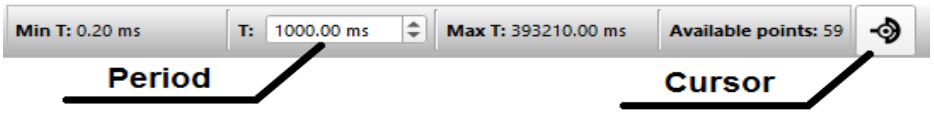
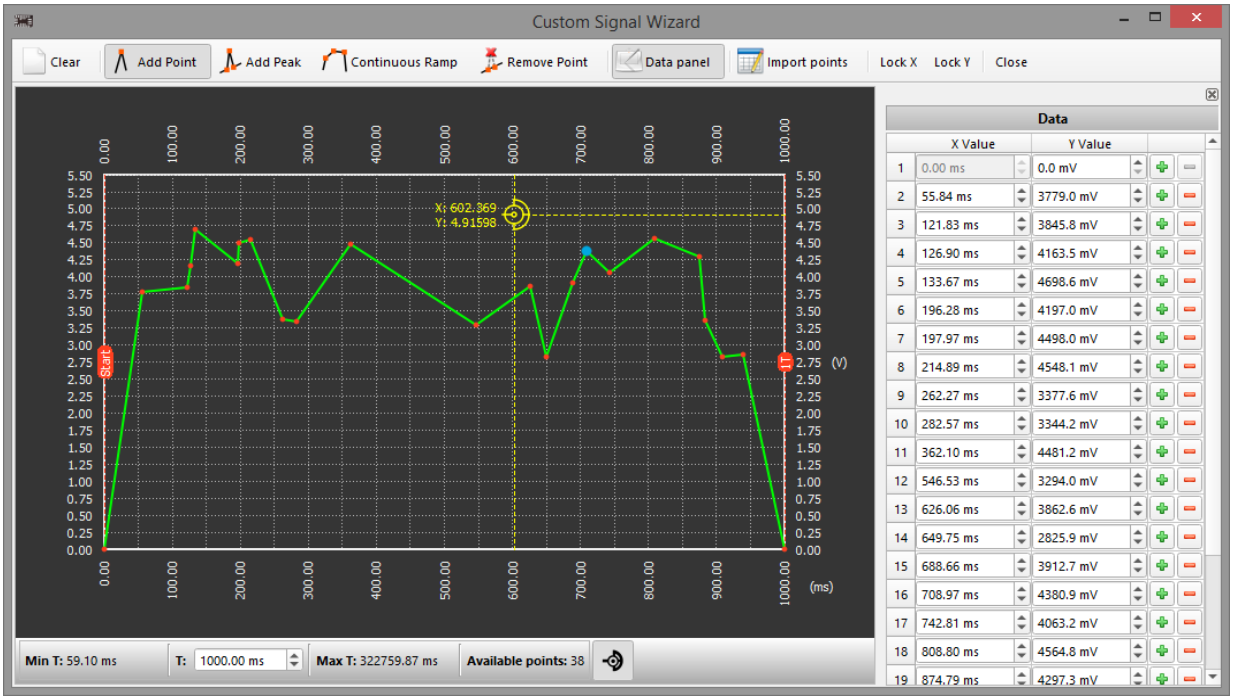
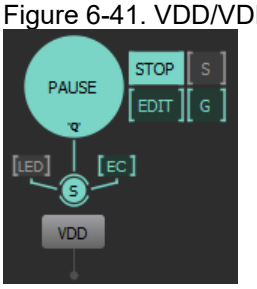


Figure 6-40. Cursor



6.2.7. VDD/VDD2 Power Signal Generator

Figure 6-41. VDD/VDD2 Power Signal Generator

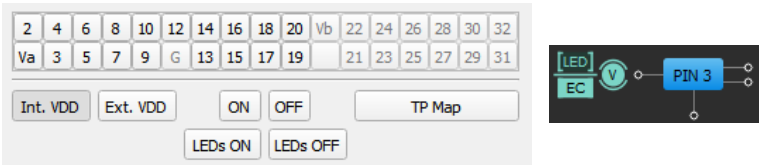


Simple signal generator for VDD/VDD2 with its own options.
The additional power supply (VDD2) on some chip revisions provides the ability to interface two independent voltage domains within the same design. Users can configure pins, dedicated to each power supply, as inputs, outputs, or both (controlled dynamically by internal logic) to both VDD and VDD2 voltage domains. Using the available macro-cells designers can implement mixed-signal functions bridging both domains or simply pass through level-translation in both HIGH to LOW and LOW to HIGH directions.
Sync Power Rails [S] mode means that VDD and VDD2 will share the same power settings. VDD2 will have the same power options as VDD. Sync power options is available only for VDD/VDD2 power generators.

6.3. Expansion Connector

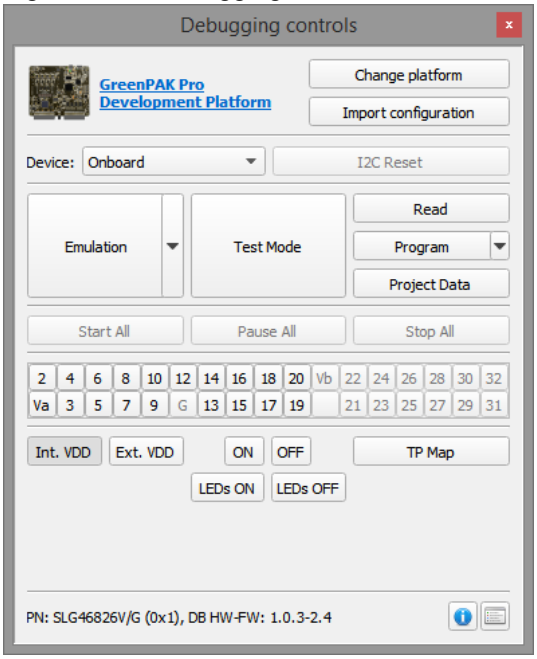
User can connect/disconnect I/O pads of GreenPAK with the expansion connector on the board.

Figure 6-42. Expansion Connector



6.4. Control panel

Figure 6-43. Debugging controls



Change platform:

Select type of hardware platform with supported features

Import configuration:

Allows user import configuration of test points from another platforms.

Device:

Allows user work with external chip on specified device address

I2C Reset:

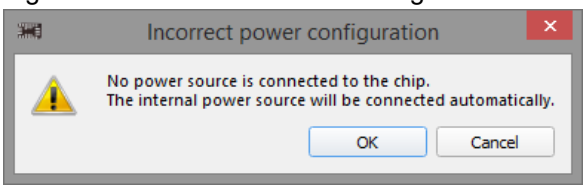
If I2C serial communication is established with the device, it is possible to reset the device to initial power up conditions, including configuration of all macrocells, and all connections provided by the Connection Matrix. This is implemented by setting register I2C reset bit to “1”, which causes the device to re-enable the Power-On Reset (POR) sequence, including the reload of all register data from NVM.

Emulation:

- Emulation - the current project will be loaded to the chip (but not programmed), and will be ready for test on the hardware board.
- Emulation (sync) – in addition to Emulation, each change that is made in the project, will be immediately loaded to the chip.

In the case when Int. VDD and VDD key on the Expansion connector are turned off, a warning message will pop up (Figure 6-44).

Figure 6-44. Incorrect Power Configuration



Test mode:

Test mode is used for connecting or disconnecting the chip's I/O pads to TP controls, configured by user. Also, a user can check the programmed chip using the test mode without emulation. In order to do this: turn on the test mode and internal VDD button. The test mode can work without power on the chip. User will control the power manually.

Read:

Read chip using hardware board.

Program:

Program chip with the current project. For some chip models user can configure programming process by clicking Programming options at Program button. Choose programming options:

- Program NVM - programs chip NVM;
- Program EEPROM - programs chip EEPROM;

Project Data:

The table of NVM and EEPROM(optionally for some chip) bits.

Pattern ID – gives an ID (1-255) to the project. The ID will be put in the chip after programming, and also will be read back while in "chip reading" operation.

Lock status – blocks NVM reading/writing. A programmed project becomes unavailable for chip reading/writing.

Use current project's sequence for Programming and Emulation process – user can choose to use current project's sequence for programing and emulation process.

Use this sequence for Programming and Emulation process – user can choose current sequence for programming and emulation process.

Reload from current project – user can load bit sequence from current project.

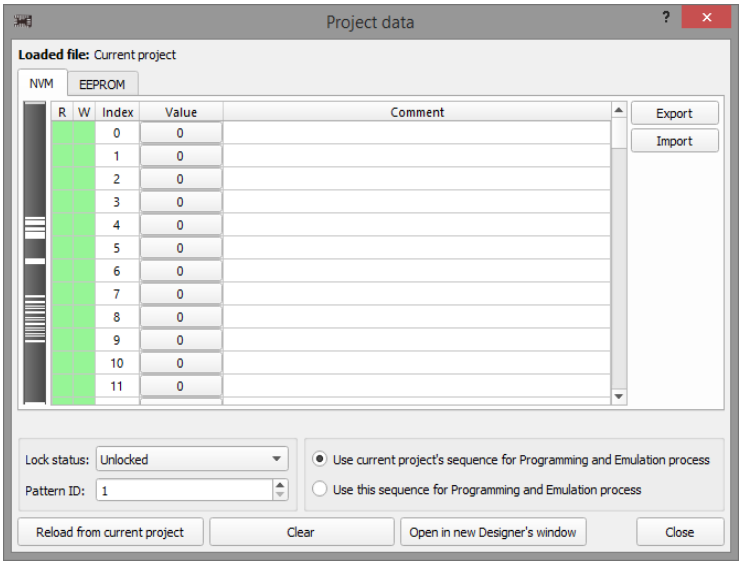
Clear – sets all bits to false.

Export – save data to text file.

Import – load data from text file.

Open in new application instance – open current bit sequence in new window.

Figure 6-45. Project Data



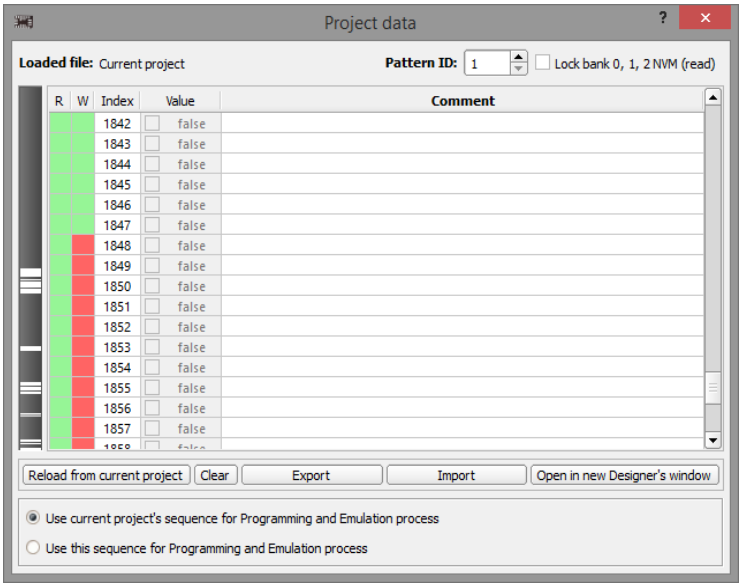
In GreenPAK Designer for chips with I2C Serial Communication new columns describes register flags in I2C tools and NVM Data window:
- I2C Legend:

- Green – Supported;
- Red - Not supported;
- Yellow - Partly supported (operation is supported for some part of data block);

- I2C Operations:

- R - I2C Read;
- W - I2C Write;

Figure 6-46. I2C operations in NVM Data window



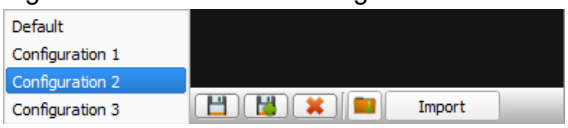
Chip Details:
Show details about chip and board

Log:
Show log.

Test points configuration:

- Save current configuration of a test points to the project file
- Delete selected configuration
- Import new configurations from project files or platforms

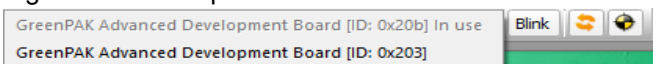
Figure 6-47. Test Points Configuration



Support for multiple devices:

This feature allows user to connect few supported hardware boards and select specific one for performing the operations.

Figure 6-48. Multiple hardware devices



Blink:

Selected board blinks to notify its selection

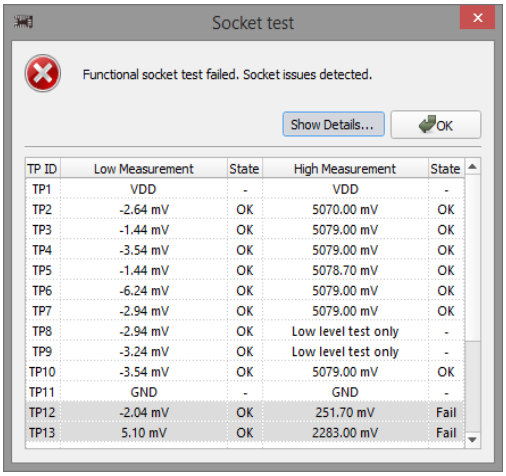
Refresh button:

Refresh button updates chip information in bottom right corner of the Debug tool.

Socket Test:

This feature allows testing the socket connectors to ensure that it works properly and doesn't have influence on emulation or reading/programming processes.

Figure 6-49. Socket Test Results

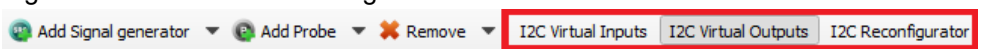


TP ID	Low Measurement	State	High Measurement	State
TP1	VDD	-	VDD	-
TP2	-2.64 mV	OK	5070.00 mV	OK
TP3	-1.44 mV	OK	5079.00 mV	OK
TP4	-3.54 mV	OK	5079.00 mV	OK
TP5	-1.44 mV	OK	5078.70 mV	OK
TP6	-6.24 mV	OK	5079.00 mV	OK
TP7	-2.94 mV	OK	5079.00 mV	OK
TP8	-2.94 mV	OK	Low level test only	-
TP9	-3.24 mV	OK	Low level test only	-
TP10	-3.54 mV	OK	5079.00 mV	OK
TP11	GND	-	GND	-
TP12	-2.04 mV	OK	251.70 mV	Fail
TP13	5.10 mV	OK	2283.00 mV	Fail

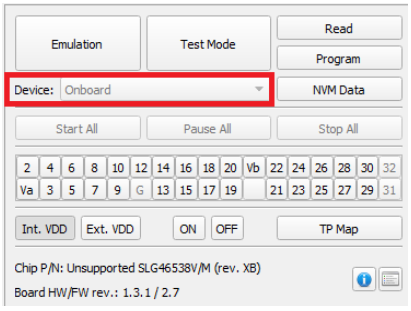
I2C Tools

Start Debug to start work with I2C tools:

Figure 6-50. I2C Tools in Debug tool



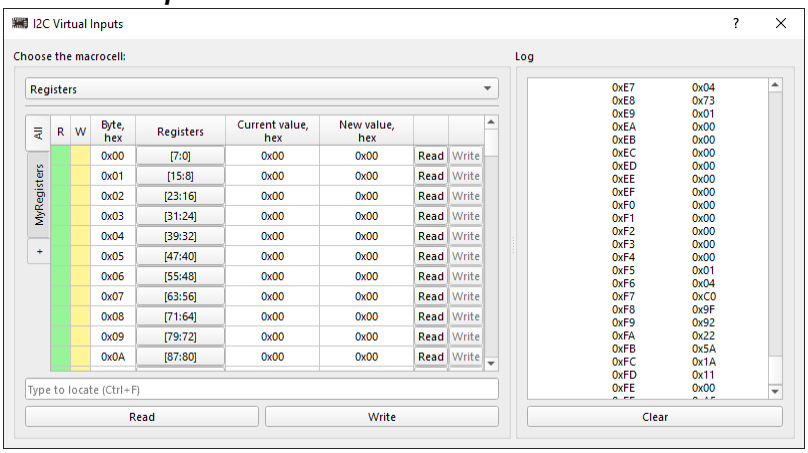
The I2C Serial Communication Block (I2C) in the chip allows an I2C bus Master to read and write information via a serial channel directly to the RAM registers. This allows the remote re-configuration of macrocells. The user has the flexibility to read and write registers not associated with NVM memory. Those registers are the signal outputs of macrocells in the device, giving the I2C bus Master the capability to remotely read the current value of any macrocell. Up to 16 (127 for specific revisions) GreenPAK I2C slave devices can share the same serial bus. You can choose any external device address and work with external chip using Device selector in Debug tool: Onboard or any of 16(127 for specific revisions) GreenPAK I2C slave devices:



The I2C cell also has dedicated registers (I2C Virtual Output bits) as well as dedicated arrays in memory (RAM Array Table) only accessible through I2C. The I2C Virtual Outputs are also signals that enter the matrix. To start I2C Tools Emulation or Test Mode is required.

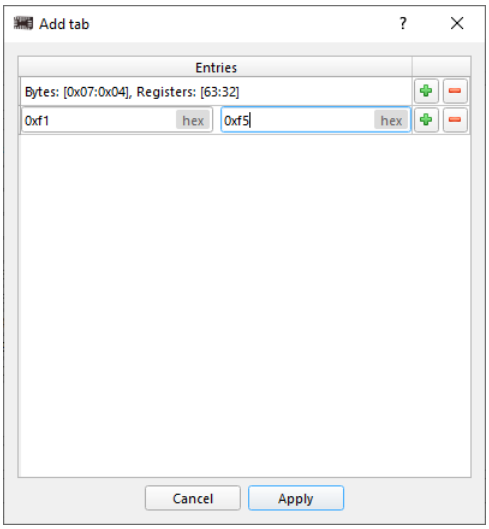
I2C Tools has 3 separate windows:

I2C Virtual Inputs:

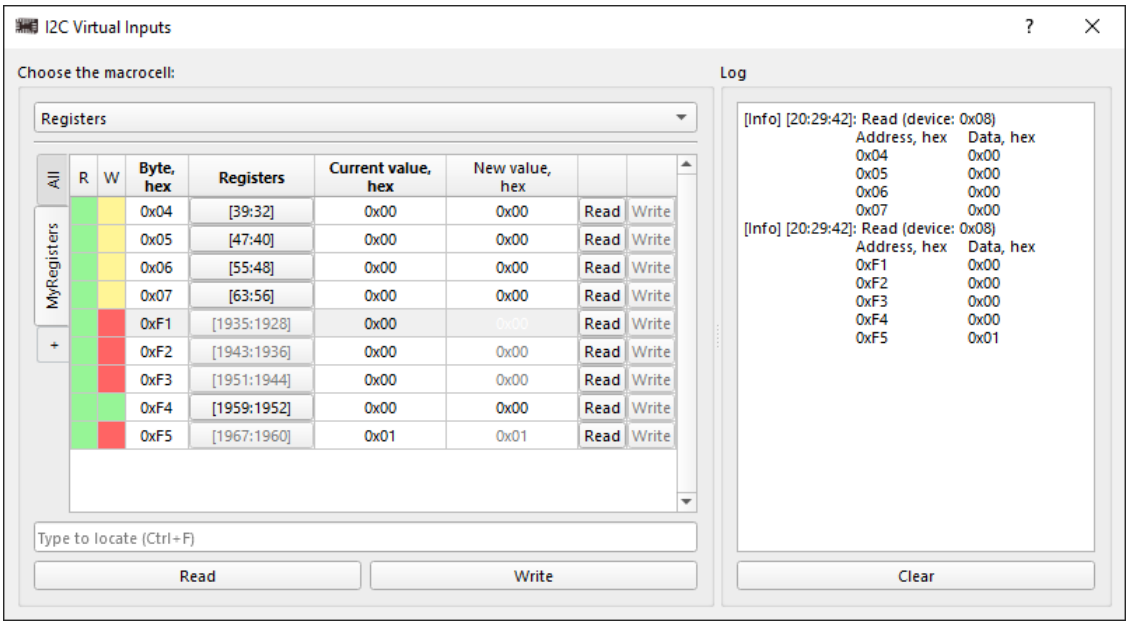


- Counters/Delays – the counter/delay data in the device can be read and write via I2C;
- I2C Virtual Inputs – the I2C Virtual Inputs OUT0-OUT7 value in the device can be read and write via I2C;
- Registers – the current value of all device registers can be read and write via I2C;
- Log – shows log of read/write operations.

Registers page allows to make tabs with a specific registers range. Simply click “+” button and add entries with registers ranges:



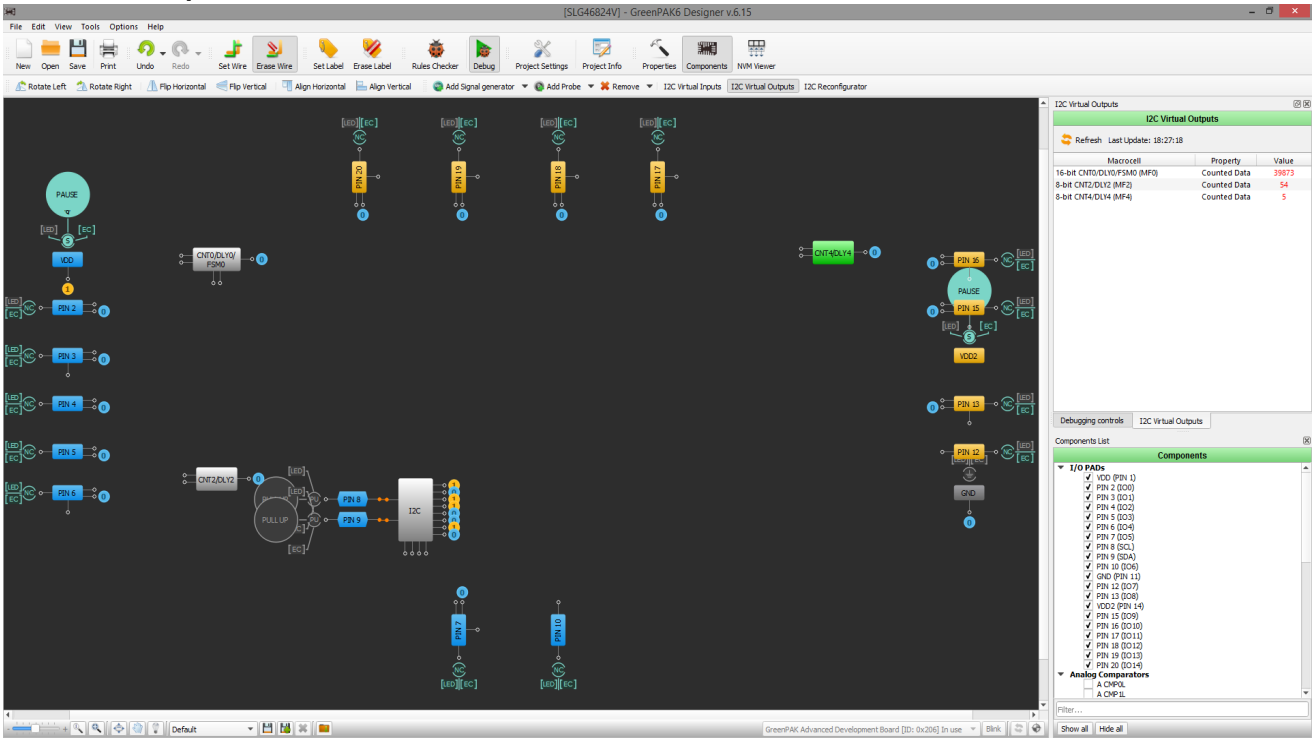
Read or Write buttons work with selected registers ranges. Any customized tab has context menu with operations Edit, Rename or Delete.



Registers page contains filter with 2 types of search:

- searching for bytes;
- searching for registers;

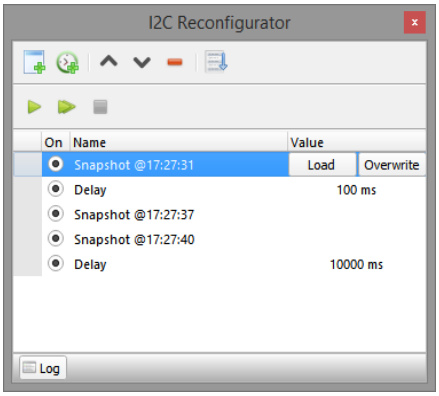
I2C Virtual Outputs:



- Probes (Matrix inputs) – the current level on blocks output pins in the device can be read via I2C. User can add probes to any pin(or to all visible pins) or remove probe(or all probes);
- Counted Data – the current count value in some counters (see Datasheet for selected chip revision) in the device can be read via I2C.

I2C Reconfigurator

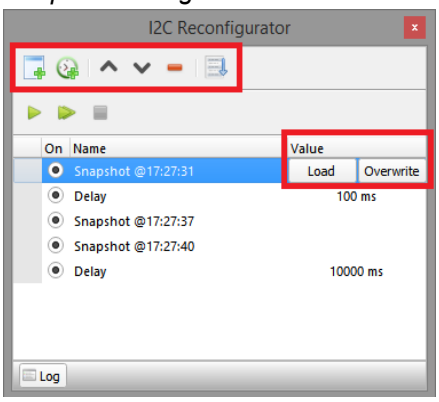
Figure 6-51. I2C Reconfigurator



I2C Reconfigurator allows user to change data dynamically on chip by sending NVM snapshots into chip. Snapshot of the NVM sequence saves configuration of macrocells and connections between macrocells. The user

can configure list of snapshots and send into the chip in two ways: send one by one or send all. Also delays between snapshots can be added.

Snapshot configuration:

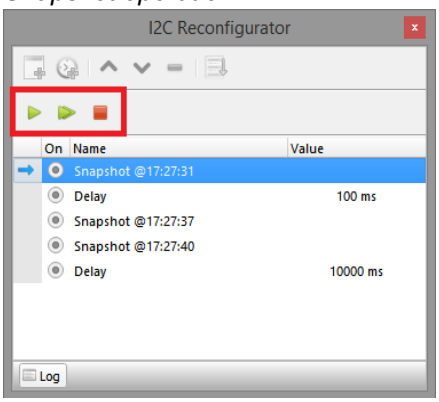


- Snapshot – button adds snapshot of workarea to the list;
- Delay – button adds delay between snapshots to the list;
- Up – moves selected list item up one level;
- Down – moves selected list item down one level;
- Remove – removes selected list item;
- Reconfiguration scenario – shows main snapshot and it changes.

Also snapshot has value:

- Load – loads snapshot data to the project;
- Overwrite – overwrites snapshot with project data;

Snapshot operation:



- Send one – send one snapshot to the chip;
- Send all – send all snapshots from the list to the chip;
- Stop – stops sending snapshots.

External chip

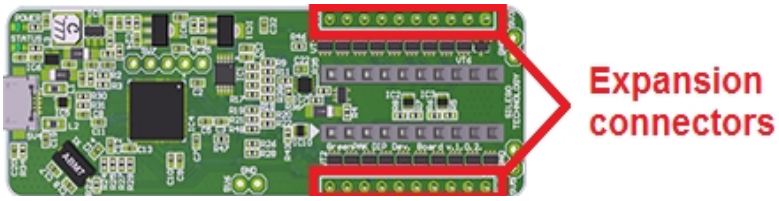
DIP/Advanced/Pro platforms

To start work with external chip for DIP, Advanced and PRO platforms:

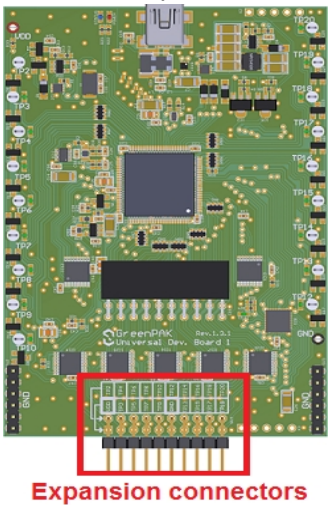
1. Connect socket with External chip to Development Board via pins:

- Corresponding I2C pins (SCL, SDA);
- VDD pin;
- GND pin;

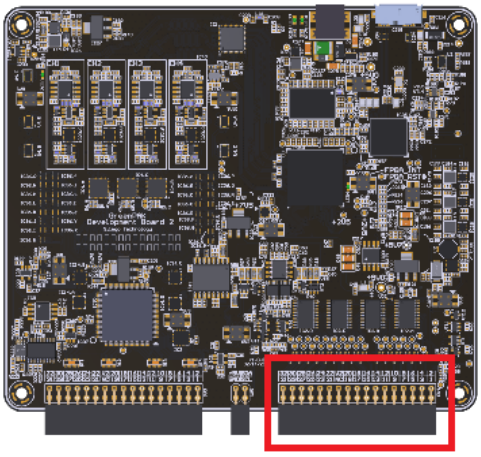
For DIP/Advanced Development board: connect socket with External chip to Expansion connectors pins:
DIP Expansion connectors:



Advanced Expansion connectors:

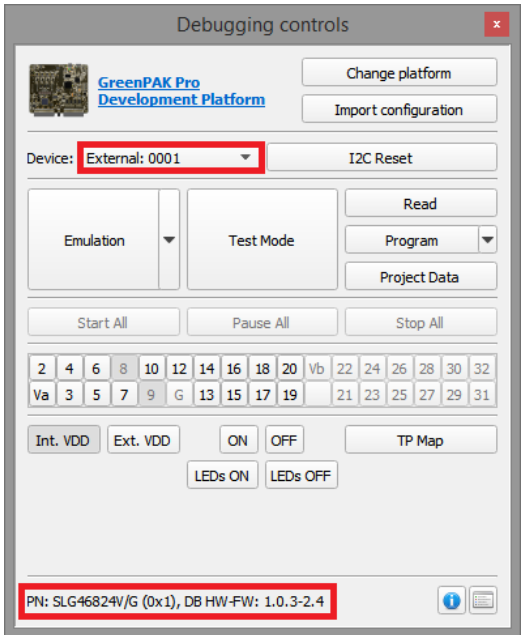


For PRO Development board: connect socket with External chip to external port on GreenPAK PRO Development Board (right connector):



2. Disconnect Onboard chip to proceed with the External chip (For DIP/Advanced/Pro platfroms)
3. Start GreenPAK Designer and select revision of connected External chip
4. Start Debug tool and select Development platform
5. Select Device address as 0001b (used by default for empty chip) or corresponding address programmed to the chip.

Attention: if external socket properly connected to development board and proper Device address selected - chip detects after clicking Update chip info button or automatically after starting any chip operation:



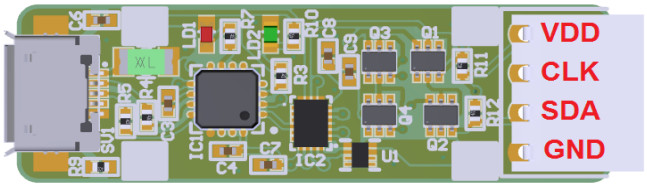
6. Now user can use all Debugging chip controls: Emulation/Test mode/Read/Program for external chip

Attention: Ext. VDD (Va) expansion should not be enabled if exist voltage on external VDD port. It automatically disconnects while starting Emulation/Test mode operations for chip with existing external voltage with warning message.

Serial Debugger

To start work with external chip on Serial Debugger platform:

1. Connect chip with wires to Serial Debugger pins (VDD, CLK, SDA, GND):

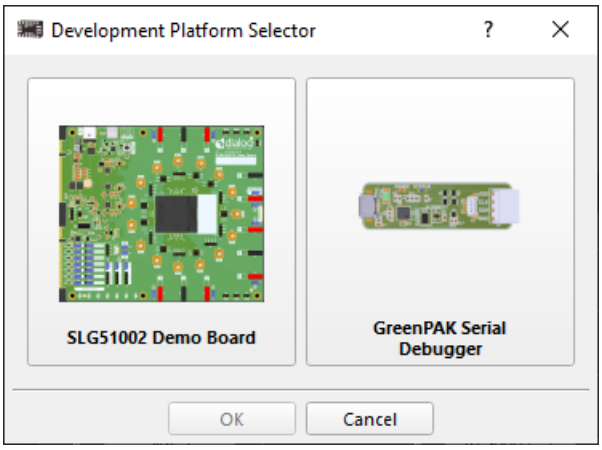


2. Start GreenPAK Designer and select revision of connected External chip
3. Start Debug tool and select Serial Debugger platform
4. Select Device address as 0001b (used by default for empty chip) or corresponding address programmed to the chip or select
5. Specify VDD configuration: Internal (chip will be powered from Serial Debugger board) or External (chip should be powered from external power source)
6. Now user can use all Debugging chip controls: Emulation/Test mode/Read/Program on Serial Debugger platform

6.5. PowerPAK Debug Tool

Type of hardware platform
After start of Debugging tools select type of hardware platform with supported features (Figure 6-1):

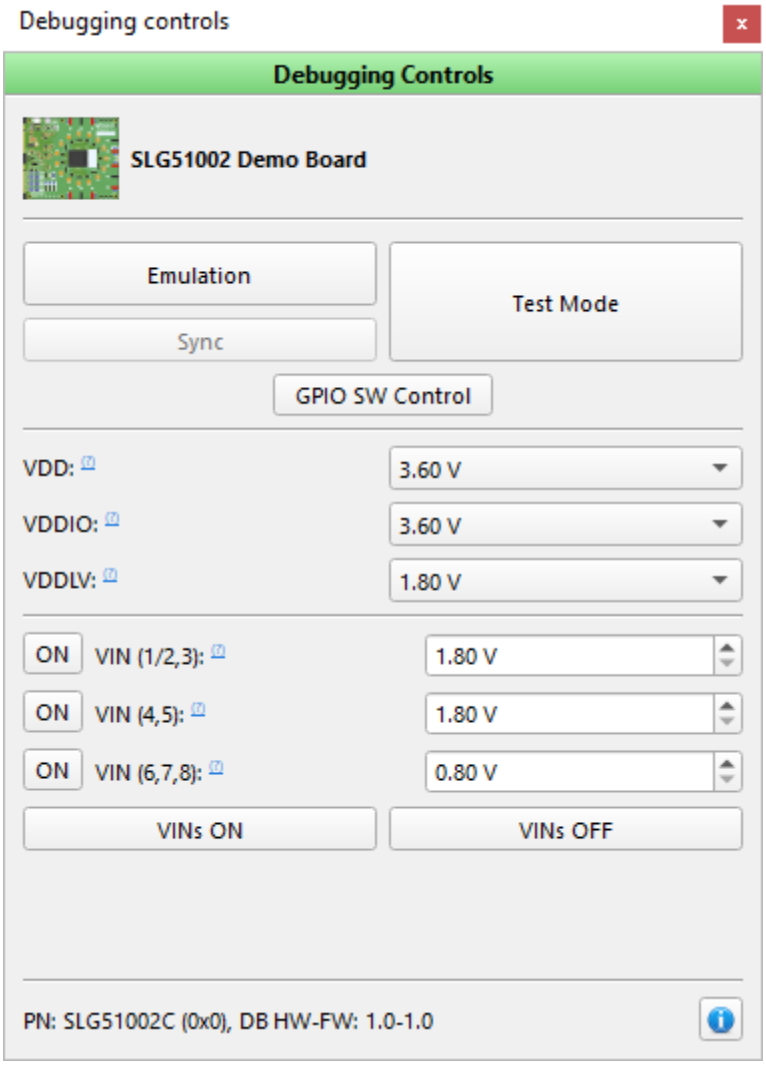
Figure 6-1. Platform selector window



Demo board
Demo board is a special hardware with a mission to demonstrate some specific application of SLG51002C chip. It has SLG51002C chip socket on the board. It also support I2C transferring that allows PowerPAK Designer to communicate with SLG51002C chip and temporarily change its NVM in Emulation. Emulation starts communication with SLG51002C chip from software. User can load any project data to chip by clicking Sync button and test configuration on hardware Demo board.

Demo board connection
After starting of Debug Tool, the PowerPAK Designer waits for connection of Demo board. All Debugging controls become available after connection of Demo board.

Figure 6-2. Demo board detected



After proper Demo board detected (Figure 6-2), simple Debug tool activated:

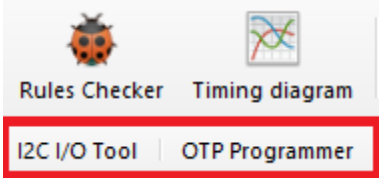
- Emulation – starts communication with SLG51002C chip;
- Sync – sends current project's NVM to device;
- Test mode – starts mode to work with or test project, programmed on chip;
- GPIO SW Control – adds buttons for software control of GPIO1-GPIO4, GPIO5(SDA), GPI6(SCL) and CS I/O pads (Figure 6-3). **Attention:** Please plug in all 'SWCTRL' jumpers to be able to control GPIO from the software.

Figure 6-3. Configurable Button

The default connection can be set to Upper or Bottom connection. Click your mouse over the key U or B to change the value. The user can configure each connection to High-Z, VDD or GND. The button has 2 modes: Latched or Not latched, which can be configured by clicking over the key LATCH.

- VDD, VDDIO, VDDLV – power supply voltages and voltage level of high state on I2C and CS pins;
- VIN – power supply voltage on relevant VINs divided in groups;
- Info button – shows all system and hardware information;

Figure 6-4. Debugging Tool toolbar

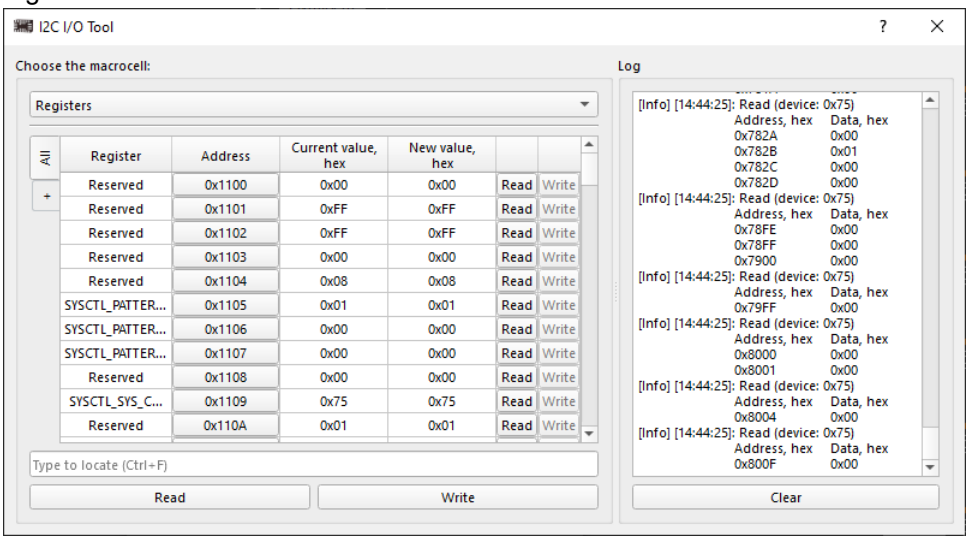


Debugging Tool toolbar contains (Figure 6-4):

- I2C I/O Tool;
- OTP Programmer;

I2C I/O Tool:

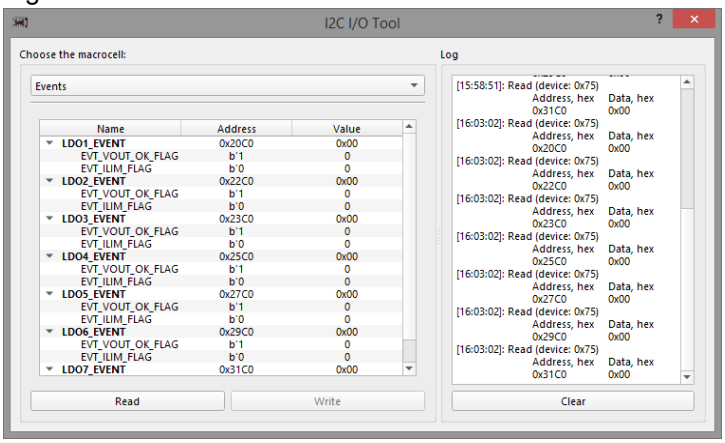
Figure 6-5. I2C I/O Tool window



Choose the macrocell:

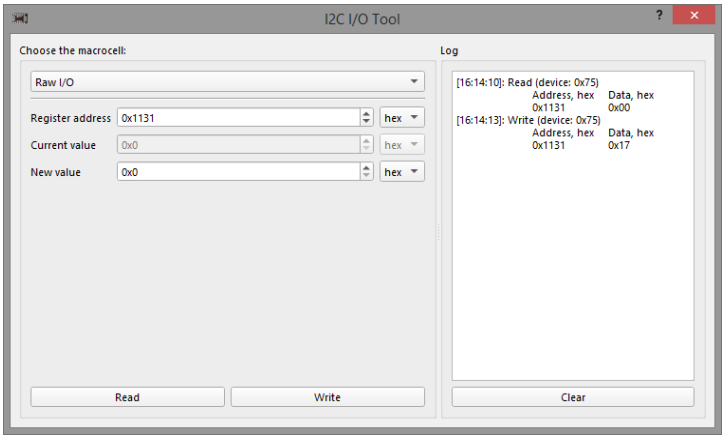
- Registers – the current value of all chip registers can be read and write via I2C. Click on any Register byte Address to change it value;
- Counters\Delays – shows current values for Counter/Delay blocks;
- Events – shows the list of LDO events: name, address and value (Figure 6-6);

Figure 6-6. I2C I/O Tool window with events



- Raw I/O – reads and writes any data from/to registers via I2C (Figure 6-7);

Figure 6-7. I2C I/O Tool window with Raw I/O



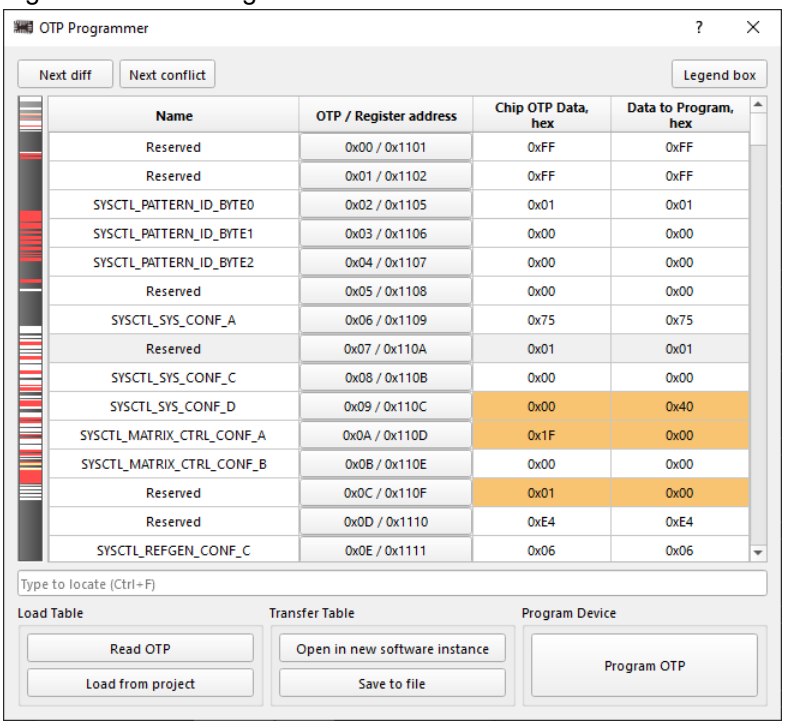
- Log – shows log of read/write operations;
- Clear – clears all Log information;

OTP Programmer:

OTP Programmer reads OTP memory from chip and programs chip with the current project data. The tool shows diffs and conflicts between chip and project data.

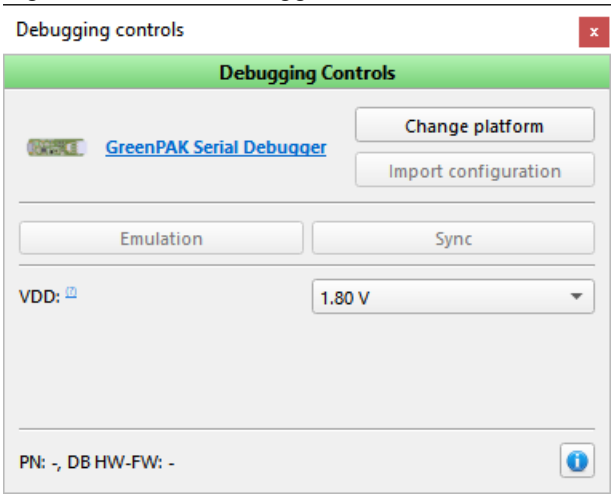
Attention: Users have the opportunity to reprogram bits changed from 0 to 1.

Figure 6-8. OTP Programmer



Serial Debugger

Figure 6-9. Serial Debugger tools



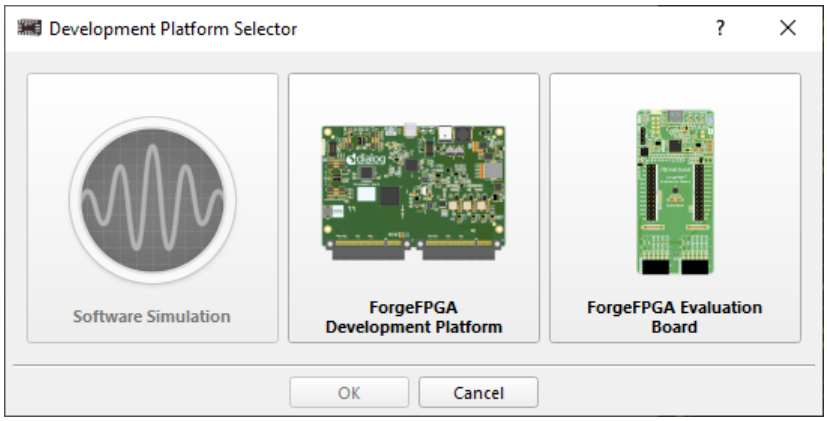
After Serial Debugger board detected, simple Debug tool activated (Figure 6-9):

- Emulation – starts Emulation: I2C transfer to communicate with SLG51002C chip;
- Sync – sends current project's NVM to device;
- VDD – selects operational Vdd for Emulation;
- Import configuration – allows user import configuration of test points from another platforms.
- Info button – shows all system and hardware information;
- I2C I/O Tool – appears on Debugging tool toolbar (Figure 6-4).

6.6. ForgeFPGA Workshop Debug Tool

Type of hardware platform
After the start of Debugging tools select the type of hardware platform with supported features (Figure 6-1):

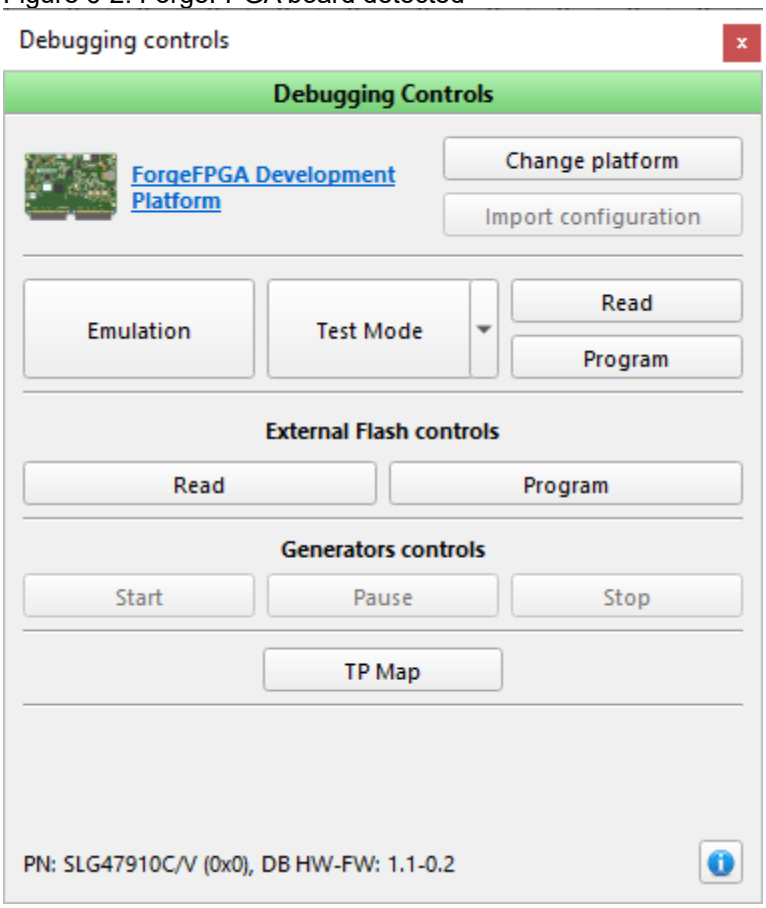
Figure 6-1. Platform selector window



ForgeFPGA Development Platform
A Development board is special hardware with a mission to demonstrate some specific application of the ForgeFPGA chip. It has a ForgeFPGA chip socket on the board. Debug tools start communication with the ForgeFPGA chip from software. Users can load any project data to the chip by clicking the Emulation button or work with programmed on a chip configuration using Test mode.

ForgeFPGA board connection
After starting of Debug Tool, the ForgeFPGA Workshop waits for the connection of the ForgeFPGA board. All Debugging controls become available after the connection of the ForgeFPGA board.

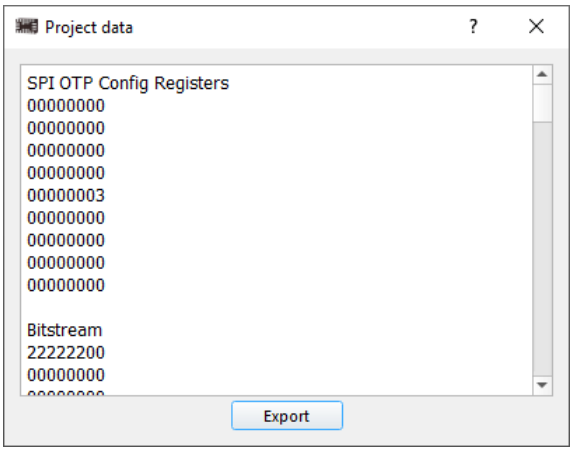
Figure 6-2. ForgeFPGA board detected



After the proper ForgeFPGA board is detected (Figure 6-2), Debug tool is activated:

- Emulation – starts communication with a chip, sends current project data to chip;
- Test mode – turns on the power supply on the chip, uses chip project, programmed on the chip;
- Test mode (*) – turns on the power supply on the chip, uses project, programmed on flash memory;
- Read – reads chip project, programmed on chip (Figure 6-3)

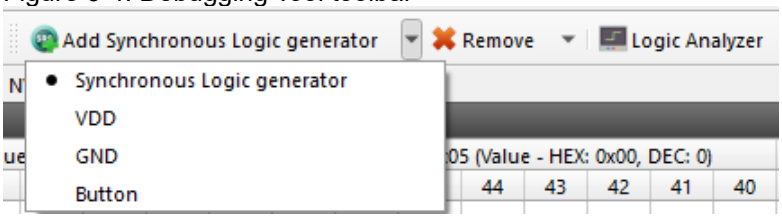
Figure 6-3. Project data window



- Program – programs chip with the current project data;
- External Flash Controls:
  - Read – reads chip project, programmed on flash memory(Figure 6-3);
  - Program – programs flash memory with the current project data;
- Generator Controls (Start/Pause/Stop) - these buttons manage all Synchronous Logic generators;
- TP Map – test point map, shows hardware board pads names;
- Info button – shows all system and hardware information;

Debugging Tool

Figure 6-4. Debugging Tool toolbar



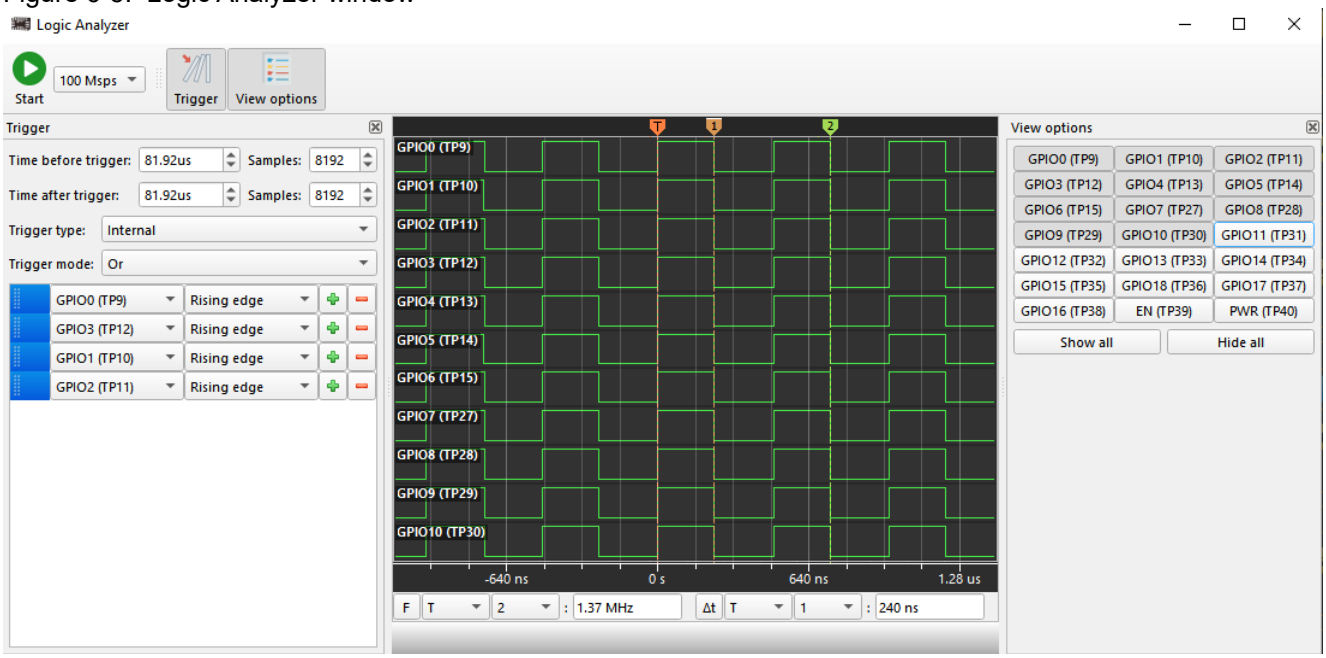
Debugging Tool toolbar contains (Figure 6-4):

- Add Sources to test points (Figure 6-4). List of available controls on TP: Synchronous Logic generators/VDD/GND/Button;
- Remove/Remove all Sources button removes test points controls by mouse click on it. Test point become NC;
- Logic Analyzer;

Logic Analyzer

This section describes the Logic Analyzer module and its features. After Emulation or Test Mode is started, the Start button in Logic Analyzer becomes active.

Figure 6-5. Logic Analyzer window



Logic Analyzer:

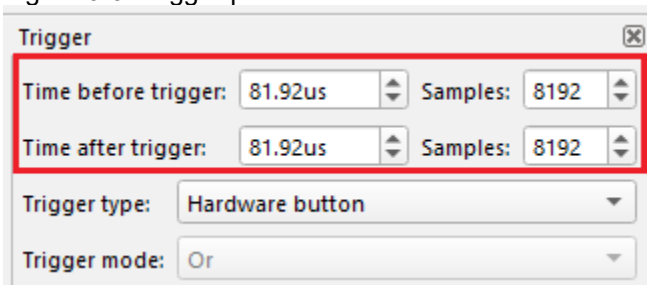
- Start – starts the data reading process.

Attention: Start Emulation or Test mode to activate the Start button in Logic Analyzer.

- Samples per second – data reading frequency;
- Trigger – opens Trigger configurations;
- View options – opens GPIO view options tab;
- Plot Widget.

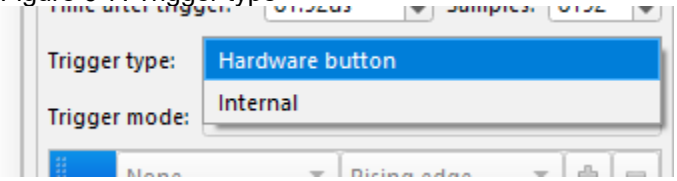
Trigger

Figure 6-6. Trigger parameters



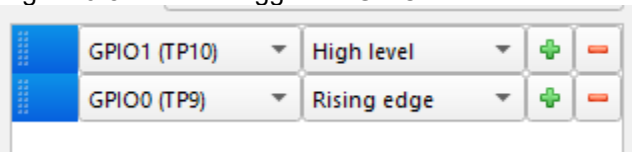
Set time parameters to choose a trigger time or a specific sample (Figure 6-6). By default, the trigger is activated when a physical button on the board is pressed. The trigger type can be changed in the drop-down list (Figure 6-7).

Figure 6-7. Trigger type



For the internal trigger, the channel and type (rising, falling, both edges, high, low) should be specified. Click the "plus" button to add a new item or the "minus" button to remove one. For the internal trigger, set the triggers to any number of test points and specify the trigger mode for them: Or, And.

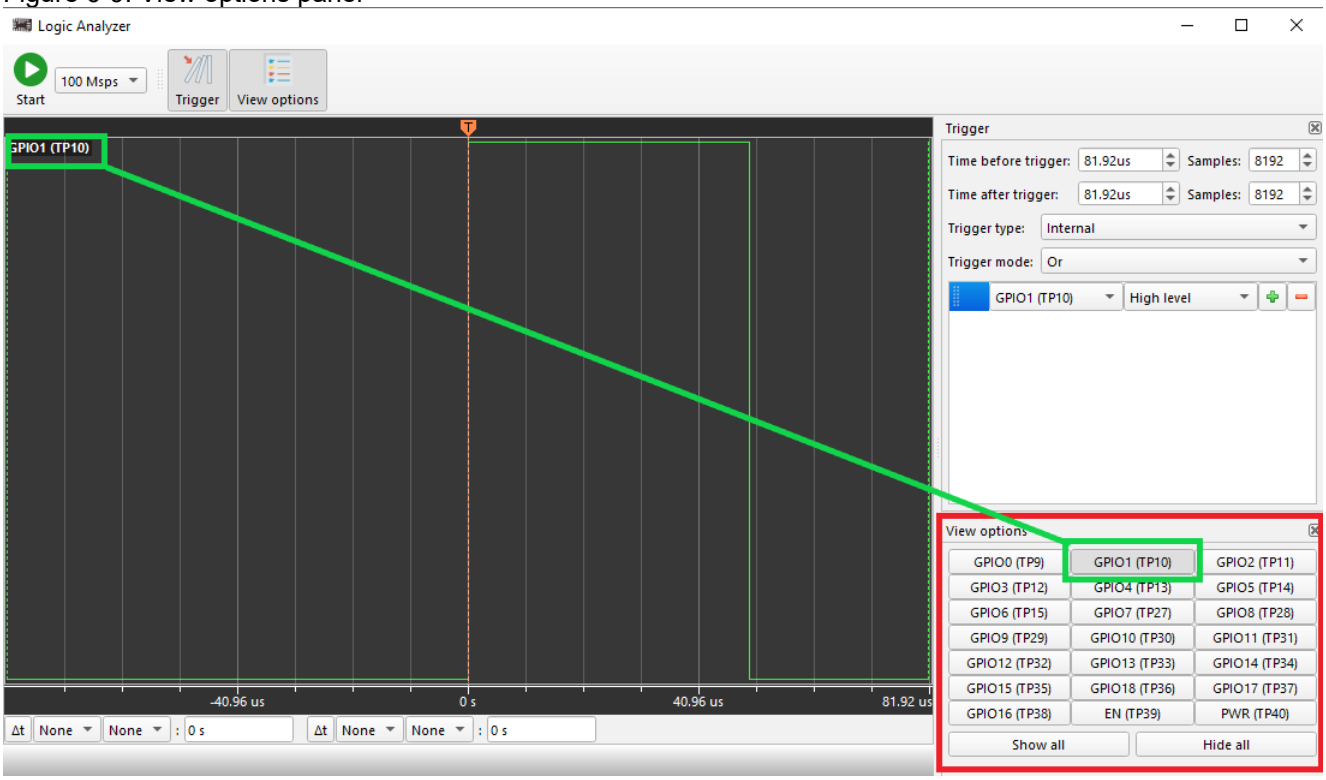
Figure 6-8. Internal trigger for GPIO



View options

In the "View options" panel, you can show or hide the graphs (Figure 6-9).

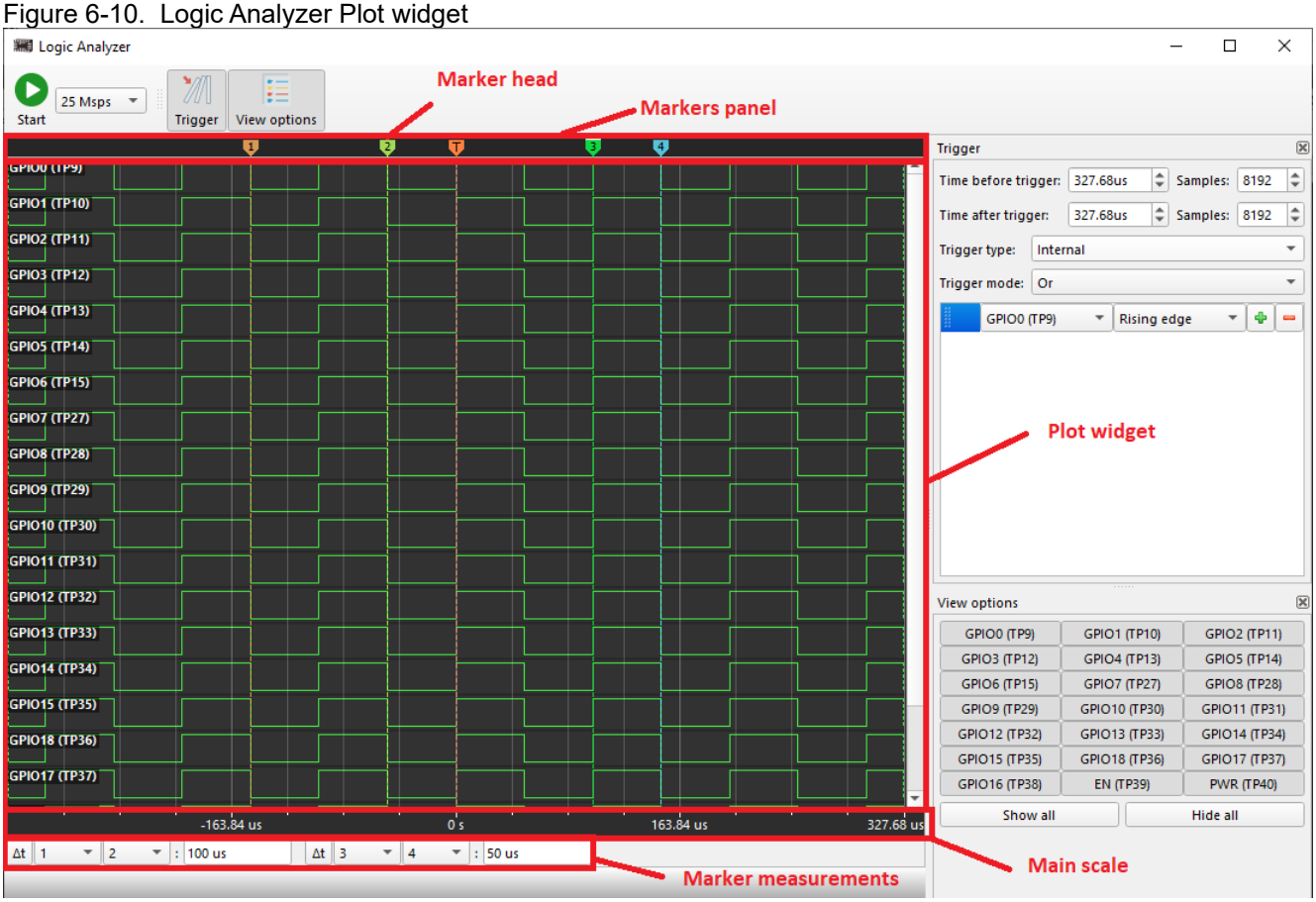
Figure 6-9. View options panel



Plot widget

The Plot widget displays the waveforms in the Logic Analyzer window(Figure 6-10).

Figure 6-10. Logic Analyzer Plot widget

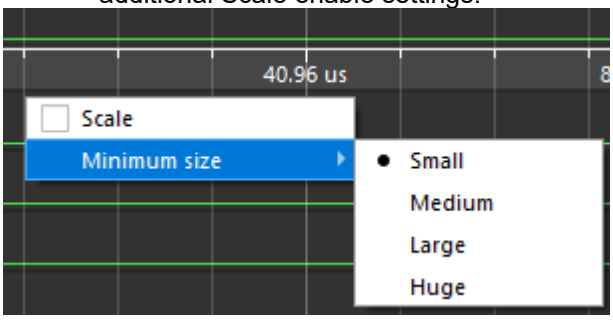


Markers:

- Add new markers by Ctrl + Left mouse click on the markers panel.
- Remove marker by Ctrl + Right mouse click on the marker head.
- Move marker by Left mouse click + drag.
- Select marker by Left mouse click on marker head. The selected marker has a white line on top.
- Move the selected marker to the closest visible left/right front by Ctrl + Left/Right.
- Move the selected marker left/right by one sample with Ctrl + Shift + Left/Right.

Plots and grid:

- The Plot has a context menu (Right mouse click) with minimum size (small, medium, large, huge) and additional Scale enable settings.



- Move left/right by Left mouse click + drag mouse left/right.
- Scroll up/down by mouse wheel Up/Down.
- Zoom in/out by Ctrl + mouse wheel Up/Down.
- Quick zoom in/out by Middle mouse click + drag Up/Down.

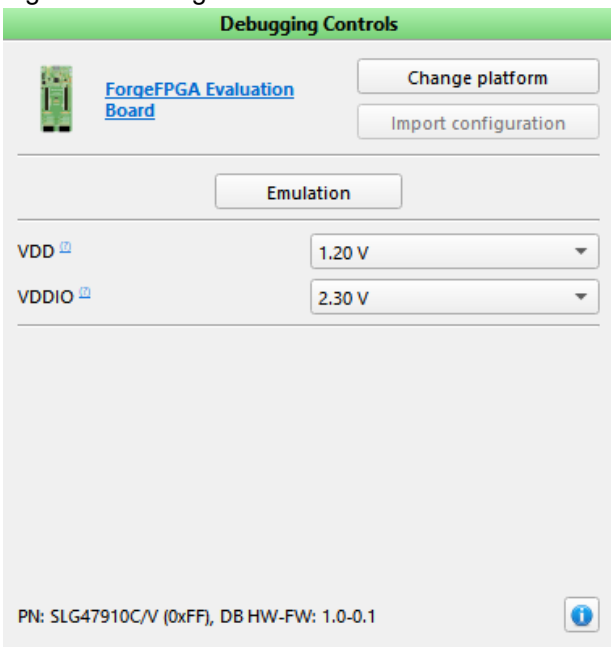
Markers measurements:

- Two modes: period and frequency. Press the button Δt/F to switch between them.
- Select two markers using combo boxes and calculate the time or frequency (1 / time in Hz) between them.

- In frequency mode value rounds to three digits after the comma.

ForgeFPGA Evaluation Board

Figure 6-11. ForgeFPGA Evaluation Board tools



After ForgeFPGA Evaluation Board detected, simple Debug tool was activated (Figure 6-11):

- Emulation – emulates chip with user project configuration data;
- VDD, VDDIO – selects operational VDD for Emulation for selected pins;
- Change platform – select type of hardware platform with supported features;
- Import configuration – allows user to import the configuration of test points from other platforms.
- Info button – shows all system and hardware information.

7. Asynchronous State Machine (ASM)

7.1 ASM block

The ASM block is an 8-state asynchronous state machine. There are 24 state transition inputs, one nRESET input, and 8 output lines. The ASM block is defined using state transitions and state outputs.

Figure 7-1. ASM block.

Transitions

The state machine is constructed by defining the one-way connections between two states gated by a transition control input. The state machine will transition to another state any time these two conditions are met: the current state has a transition connection to another state, and that transition enable signal is HIGH. The transition enable signals are controlled by the 24 inputs, and there are 3 transition enable signals per state.

Outputs

The ASM block has 8 outputs to the connection matrix. You can define the output of the 8 outputs based on the current state. This is done in the properties tab by double clicking the desired output for a given state and output line to invert its output.

Editing the ASM block

The ASM block can be defined through either the properties panel on the left, or the ASM editor. The ASM editor is a graphical interface for defining the ASM and editing labels. Double-click on any ASM component to open ASM Editor.

In the properties tab, you can select a state, edit its name, and configure all of its transition enable switches. Also, you can define the ASM Outputs, edit the label names for each output, and set the Initial/reset state. In the context menu of ASM block you can split ASM block to separate states blocks and to separate transitions (figure 7-2).

Figure 7-2. Split ASM block to States/Transitions.

Initial/reset state
The ASM block will power up to the state defined as the Initial/reset state. It is by default State 0, but can be changed to any state. It also defines which state the ASM will be reset to. When the logic on nRESET is LOW, the ASM will be kept at the Reset state until the logic at nRESET goes HIGH.

7.2. ASM Editor

ASM editor(figure 7-3) allows to configure the ASM block using state diagram and set the output configuration for ASM Output block ASM configuration should be applied using Apply button on the toolbar to be set to the NVM.

States
State colors:

- Green: regular state;
- Blue: selected state;
- Red: initial (reset) state;
- Gray: state connections limit reached;

Move
State can be moved by dragging it's central part using left mouse button. Dragging the border circle will have no effect.

Context menu

- Edit name: set state's name;
- Initial state: sets current state as initial;
- Hide: hides current state.

Transitions
Add/remove links

- To create new link turn on 'Set Link' mode and click on the border circle of source state, then click anywhere on state-destination or 'Esc' to cancel;
- To erase link - turn on 'Erase Link' mode and click on the link which should be erased.

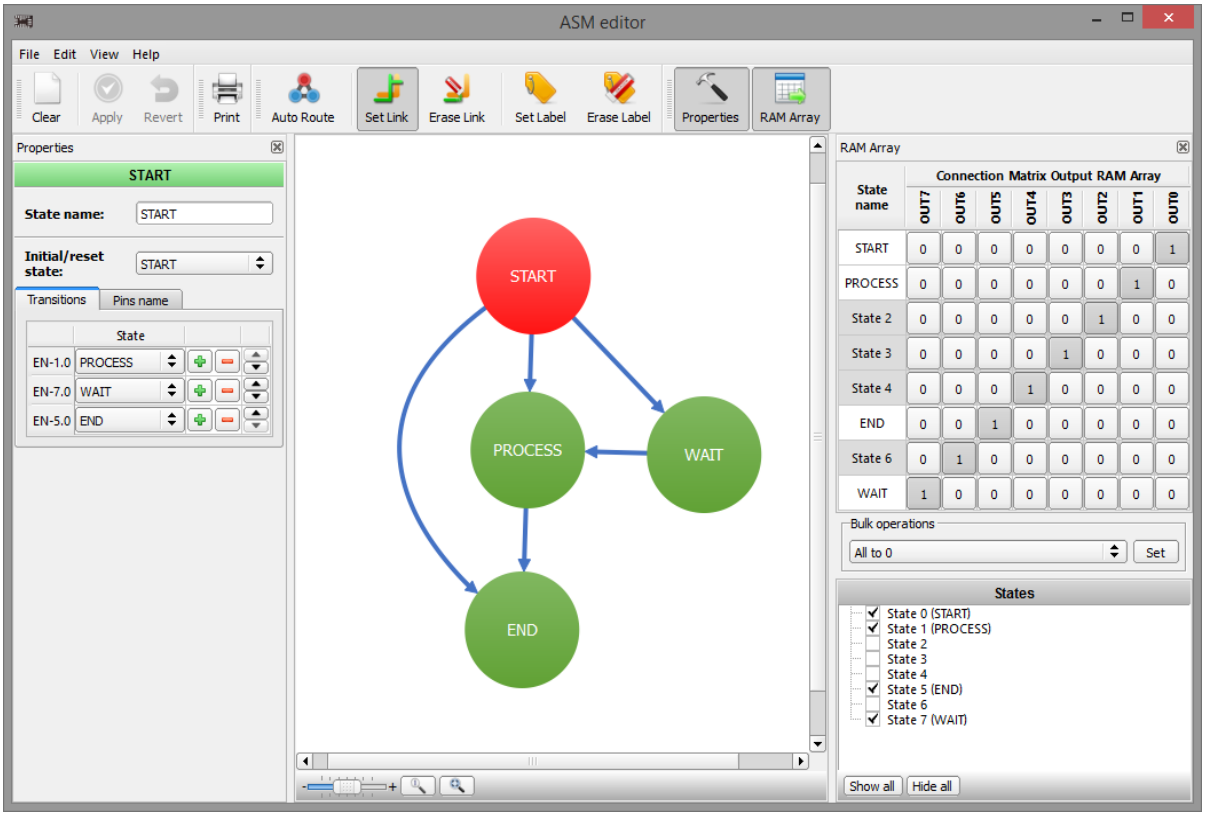
Edit shape
Link's shape sets automatically after states change their position. Modifying of the existing shape is available in two ways:

- Simple edit: one-point editing by dragging the link;
- Flexible edit: two-points editing by entering in 'Edit path' mode using link's context menu;

Context menu

- Set Label: sets the text label onto link;
- Erase Label: removes the text label;
- Edit path: flexible two-point editing of link's shape.

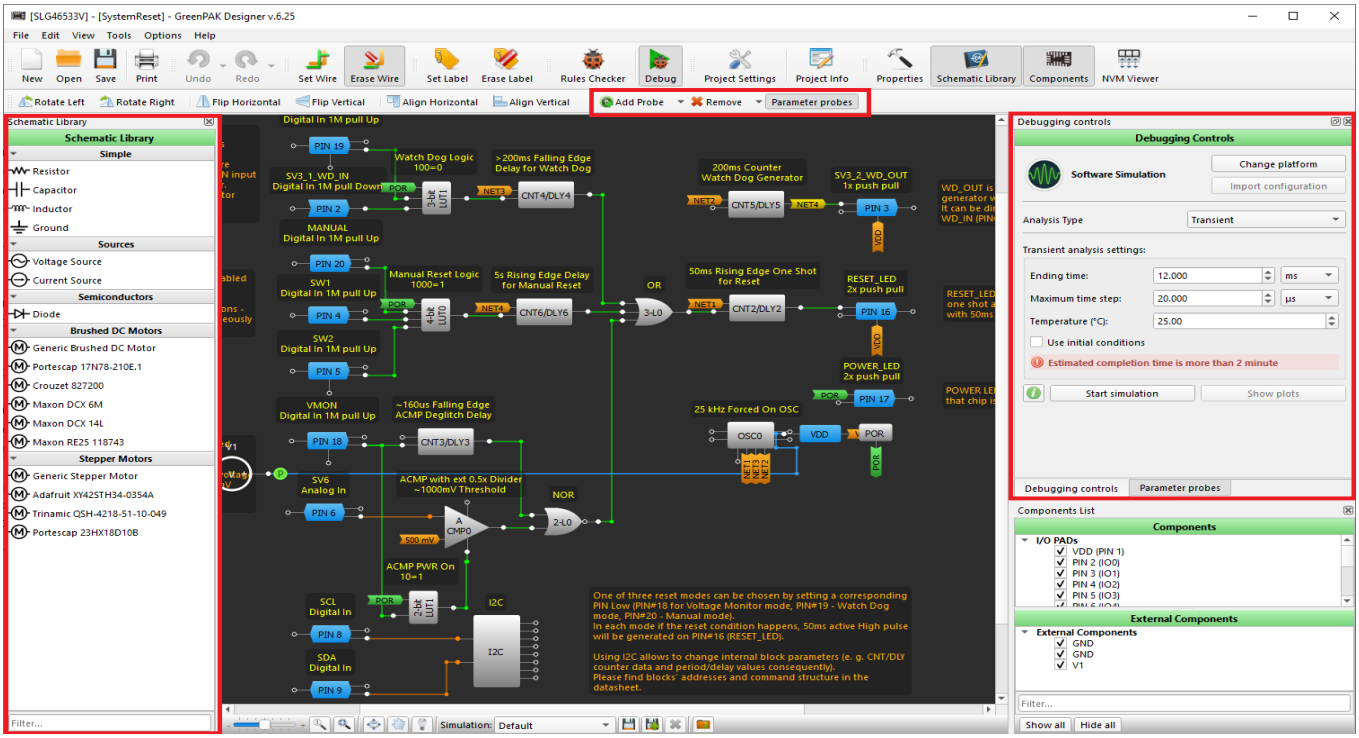
Figure 7-1. ASM Editor.



8. GreenPAK Simulation

Simulation Tools in GreenPAK Designer:

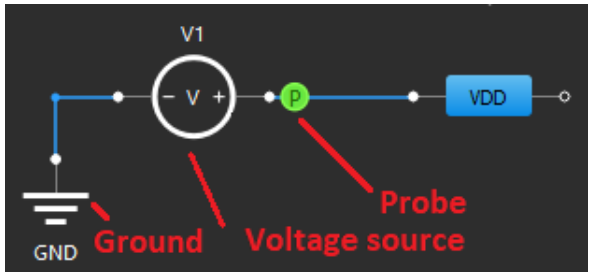
Figure 8-1. Software Simulation Tools.



Software Simulation

The Software Simulation mode enables electronic circuit simulation, which uses mathematical models to replicate the behavior of chip components included in GreenPAK and some external components. To start Software Simulation click on Debug button and select Software Simulation in Development Platform Selector. Before starting the analysis, use Schematic Library and Add Probes to configure the simulation environment parameters as shown on Figure 8-2.

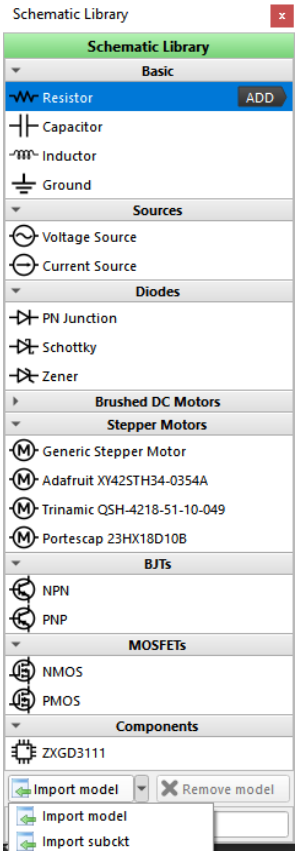
Figure 8-2. Basic Simulation Tools



Add Components from Schematic Library

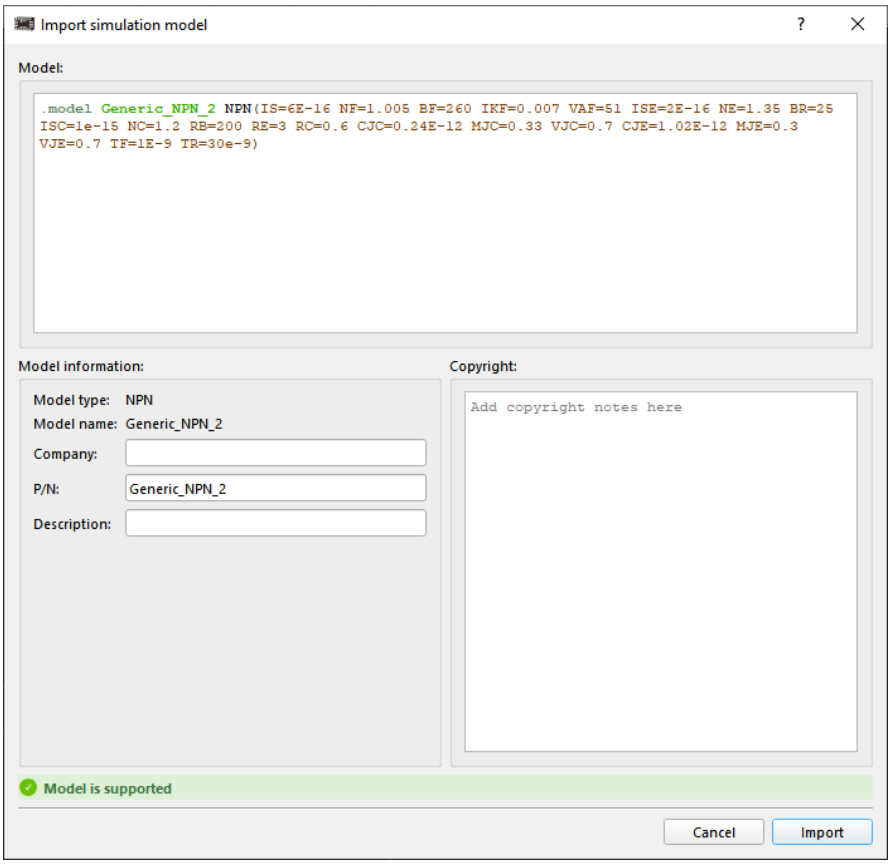
Click on component to add it to external I/O pin on chip (Figure 8-3). Voltage source(s) and GND added to corresponded pins in chip by default. Components cannot be added to any internal macrocell.

Figure 8-3. Schematic Library



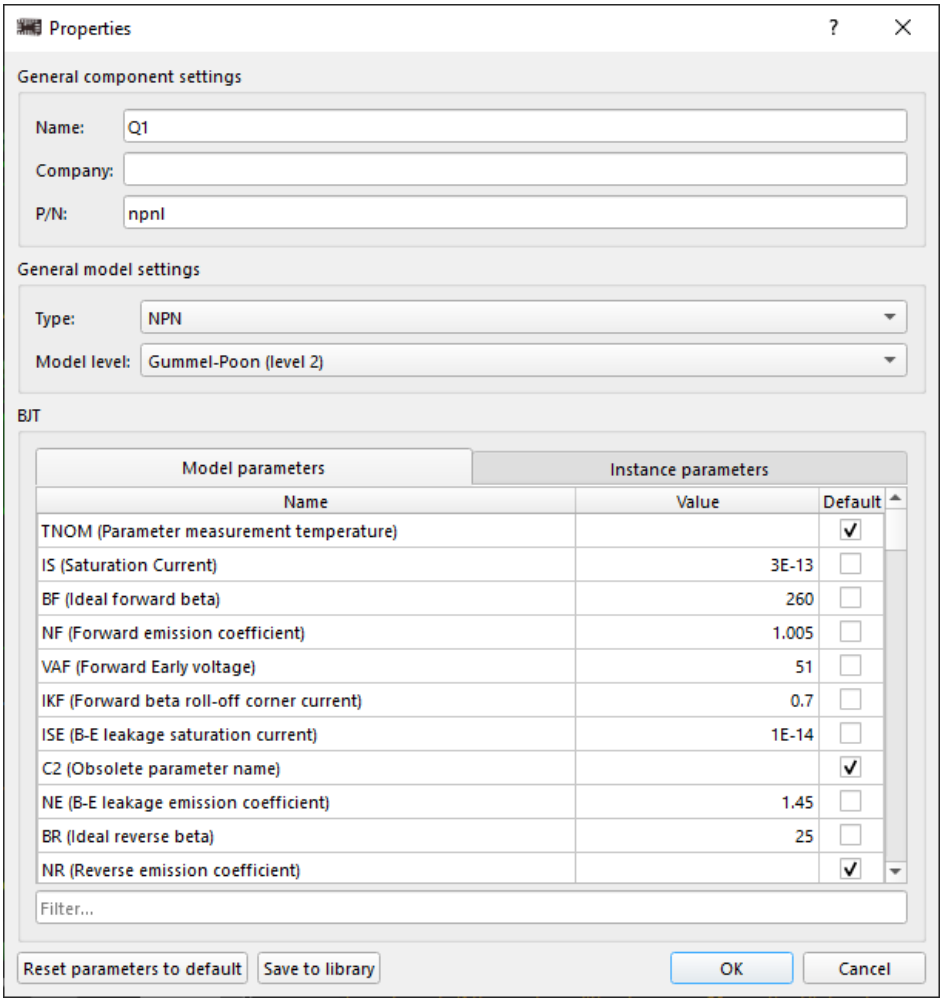
“Import model/subckt” button enables import of 3rd party models and subcircuits to GreenPAK project. Import model/subckt opens the import dialog where the user can insert the SPICE simulation model, fill in additional info about it, and import it to the library (Figure 8-4). The dialog validates a model and ensures the user imports exactly one .model or .subckt that uses valid SPICE syntax. Model validator accepts comments and multiline models. Ngspice model or subcircuit syntax is highlighted in editor window. Imported models are shared among all instances of the GPD. When the user imports the model to GPD it automatically becomes available in all started GPD applications. The user can remove the imported model or subcircuit from the library using the "Remove model" button.

Figure 8-4. Import Simulation Model Window



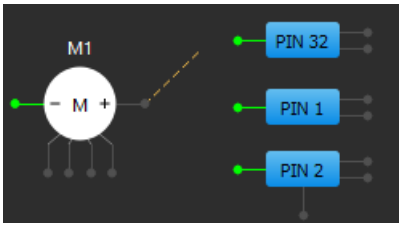
All imported models have dialog for model parameters editing (Figure 8-5).

Figure 8-5. Model Properties Window



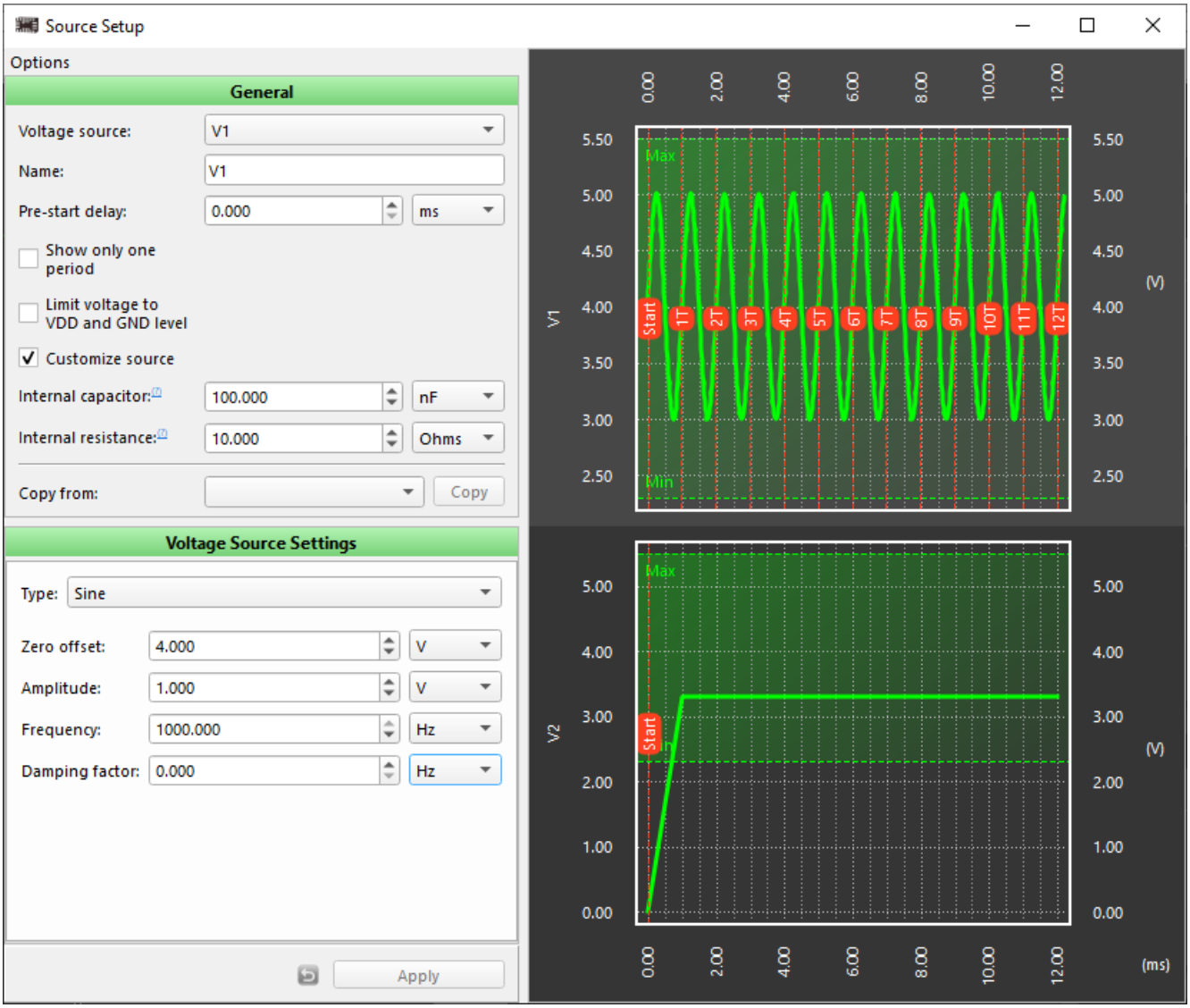
Component connects to external I/O pins with Set Wire tool. All available connections are highlighted with green color as on Figure 8-6.

Figure 8-6. Add wire to External Component

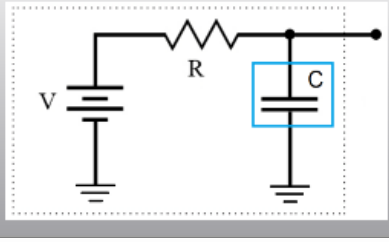
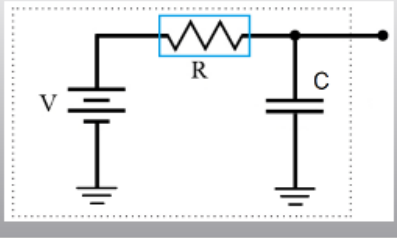


All components can be configured from Properties panel after select or by double click on it as on Figure 8-7.

Figure 8-7. Voltage Source Setup Window



General

Voltage source	Source selector
Name	Custom source name
Pre-start delay	Delay before start
Show only one period	Shows one period of voltage source
Customize source:	
Internal capacitor	This parameter defines value of the capacitance between voltage source terminal and ground. It can be used to model VDD bypass capacitor or IC pin parasitic capacitance. 
Internal resistance	This parameter defines the output resistance of the generator. Non-ideal (real) voltage source is modeled with two components: ideal voltage source and resistance connected in series. 
Copy from	Copy parameters from any voltage source

Voltage source settings

Type	DC, Trapeze, Sine, Exponential, Custom signal, Logic pattern, Clock generator	- type of waveform

Please see for the reference the voltage source settings in Emulation tool (Chapter 8, 10)

Add Probe Button

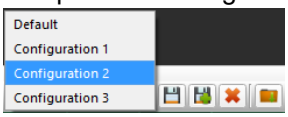
Adds probes to block outputs which shows simulation results graphs over time. Probes are used to capture the output signal from specific node. Probes can be added to the output from any pins, as well as the outputs for any internal resource. User can add probes to all visible pins from context menu.

Remove Button

Removes probe by mouse click on it or all probes from context menu.

Simulation pins configuration:

- Save current configuration of pins to the project file
- Delete selected configuration
- Import new configurations from project files of the same chip revision



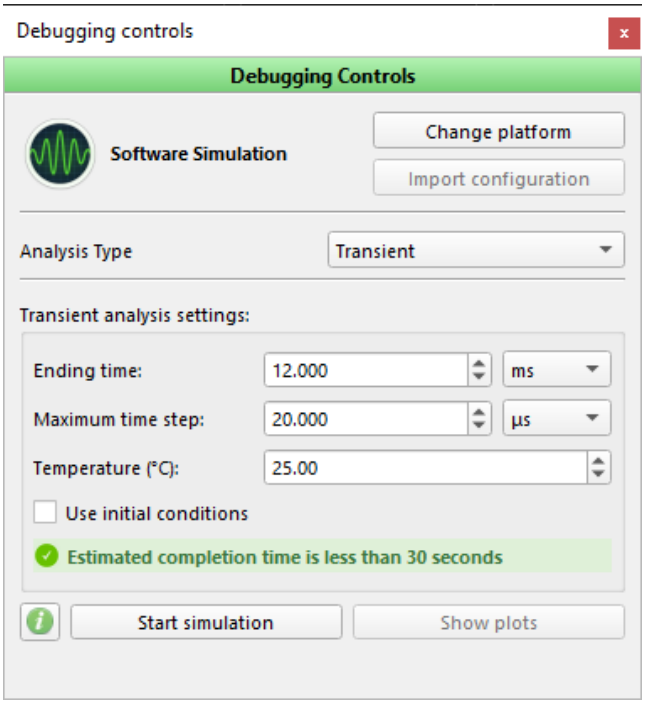
Simulation Debugging controls

Select Analysis Type before starting Simulation. There are 2 types of Software Simulation: Transient and Parametric DC.

Transient Analysis

Transient Analysis settings in Debugging controls window define period of time for simulation, maximum time step, source voltage and temperature (Figure 8-8). Note: If there is a change in the state of any probes, there will be a step generated at that time, the “Maximum time step” variable will only insure that if no probe data changes for this period, another step will be generated.

Figure 8-8. Transient Analysis Window



Transient analysis parameters:

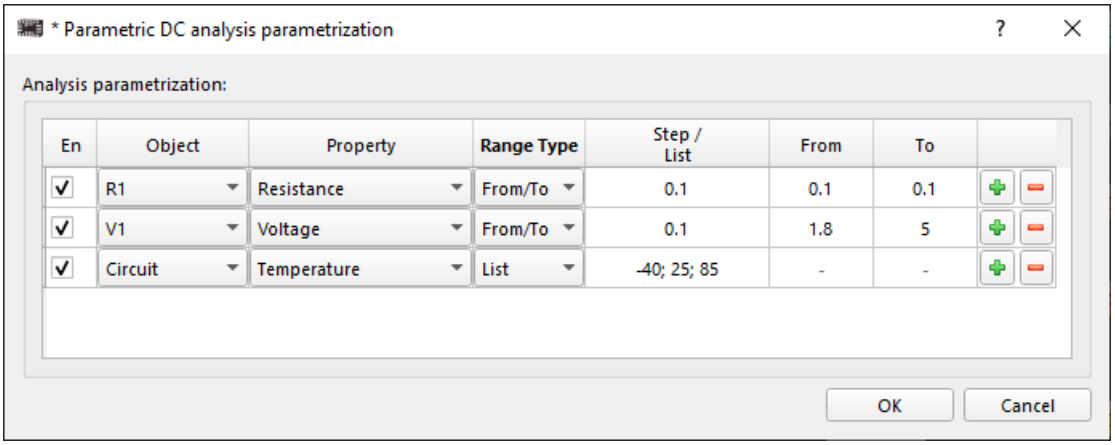
Parameter	Max time value	Min time value
Ending time	1000s	0.0001ms
Maximum time step	20s	0.0001us
VDD	6V	0.001mV
Temperature (°C)	-40	+85

Parametric DC Analysis

Customize Schematic Library objects properties by using Parametric DC analysis in Software Simulation. Parameter sweep can be configured in the Parametric DC analysis parametrization dialog. Open parametrization settings button on the Debugging Controls. Analysis parametrization can vary active parameters for external components, circuit temperature, and some GreenPAK macrocell properties (e.g. Resistance (initial data) for Digital Rheostats)(Figure 8-9). Parametric DC data has 2 range type:

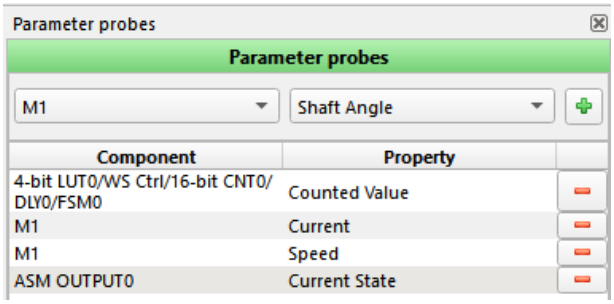
- From/To: Step, From, To parameters;
- List: list of values separated by semicolon;

Figure 8-9. Parametric DC Analysis Window



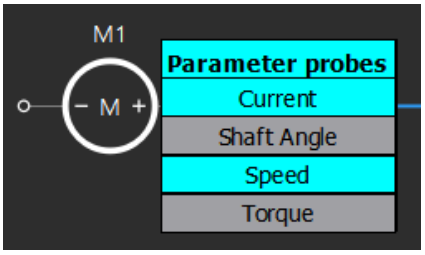
Parameter Probes

Figure 8-10. Parameter probes window



Parameter Probes gives ability to monitor the internal chip values in simulation, f. e. counted value in Counter/Delay blocks. Add parameter probe by selecting component and property (Figure 8-10) and start simulation.

Figure 8-11. Parametric Simulation Probes sticker



The sticker for parameter probes provides additional flexibility for adding/removing parameter probes for components. It represents the graphical interactive item object for the component that can react on mouse hover and allows automatically adding or removing parametric probes for the current component when clicking on it.

Estimated completion time

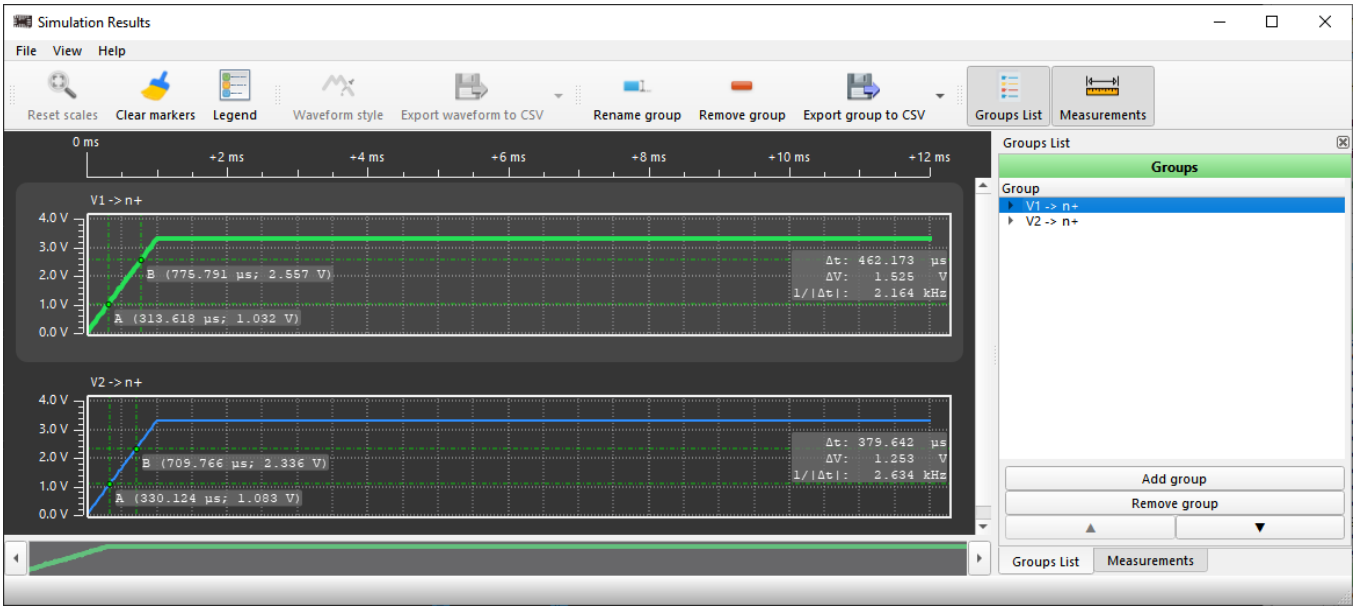
Software estimates the run time in three broad categories: green, yellow and red, based on sample points. As a general rule, green runs will complete quickly (less than 10 secs), yellow runs within one min, and red takes longer. Run time also varies based on the speed of your computer so the estimated times vary widely. Longer run times will also require greater resources on your computer, including CPU and memory resources.

Note that it is possible to define simulation runs that will exceed the available resources on your computer, which can potentially make your computer unstable. This may be true any time the estimated run time is categorized as “red”.

To start simulation, set Analysis Type, required parameters and click Start simulation. After simulation performed, user will see Simulation results (Figure 8-12).

Simulation Results Window

Figure 8-12. Simulation Results Window



The simulation result window has section:

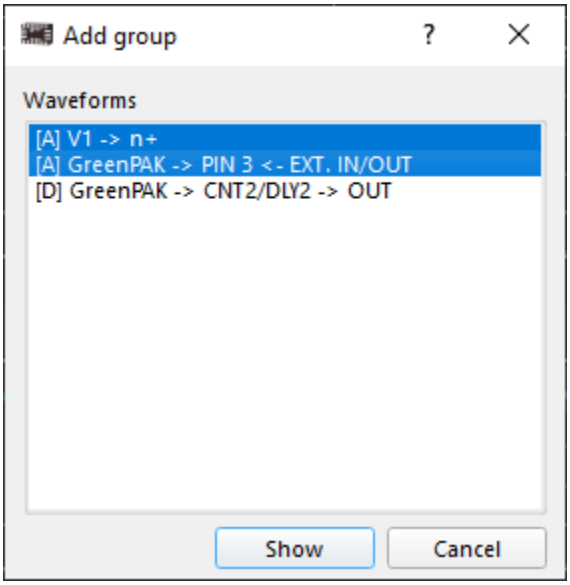
1. **Plot Widget**: displays the waveforms;
2. **Groups List**: contains a **list of all captured signals** (probes) **and source generators** in groups;

   Add group: contains a list of **signals that currently are displayed** on Plot widget.

   a. User adds the signal from Groups section to the custom group to display it.

   b. It calls ‘Groups’, because each waveform after adding to the Group section becomes a group of 1 waveform. User can select multiple waveforms at the same time and add them as a new group of analog[A], digital[D] or parameter probes[B] waveforms (Figure 8-13).
3. **Measurements:** contains a list of measurements for added plot markers;
4. **Toolbars:** contains operations with plot area, waveforms style, groups management and export waveforms/group to CSV, VCD, PNG.

Figure 8-13. Add group menu

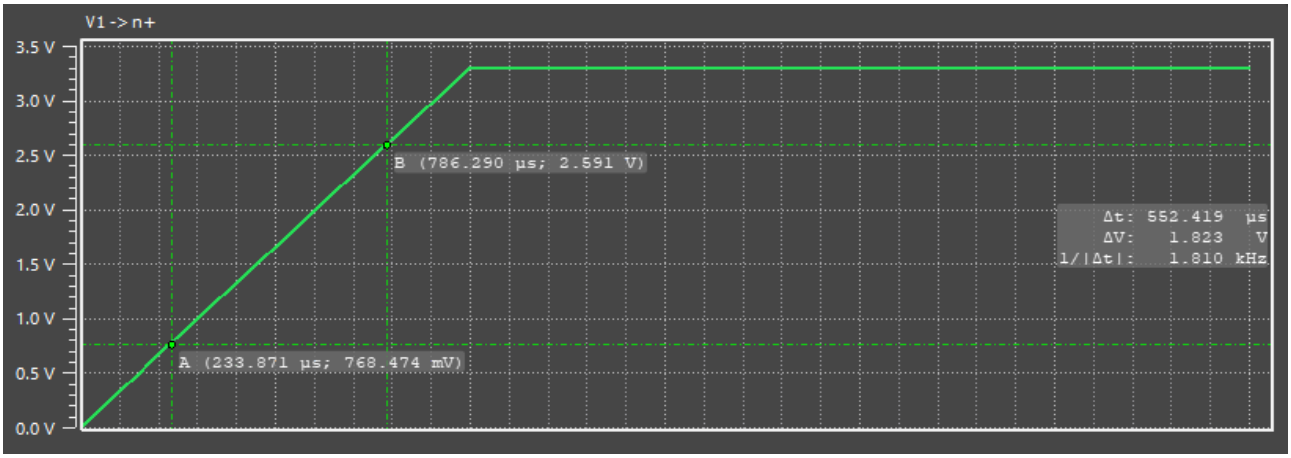


In the Plot Widget, the user has following controls:

Action	Hotkey
Pan mode	Press middle mouse button
Marker A	Ctrl + LMB
Marker B	Ctrl + RMB
Clear all markers	Esc or context menu → Clear markers
Zoom in/out X	Ctrl + mouse wheel
Zoom in/out Y	Shift + mouse wheel
Reset scales	Context menu → Reset scales

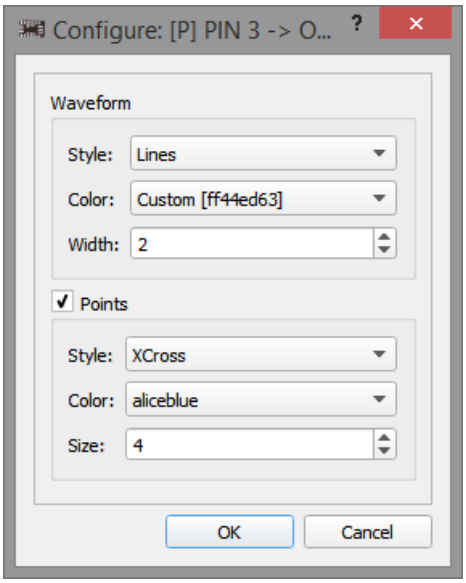
Markers and dt and dV parameters on plot widget showed in Figure 8-14 below:

Figure 8-14. Markers on Plot Widget



In Waveforms window user can configure waveforms scale and waveform base points parameters (style, color, width/size) by clicking on Waveform style button:

Figure 8-15. Waveform Plot Configuration Window



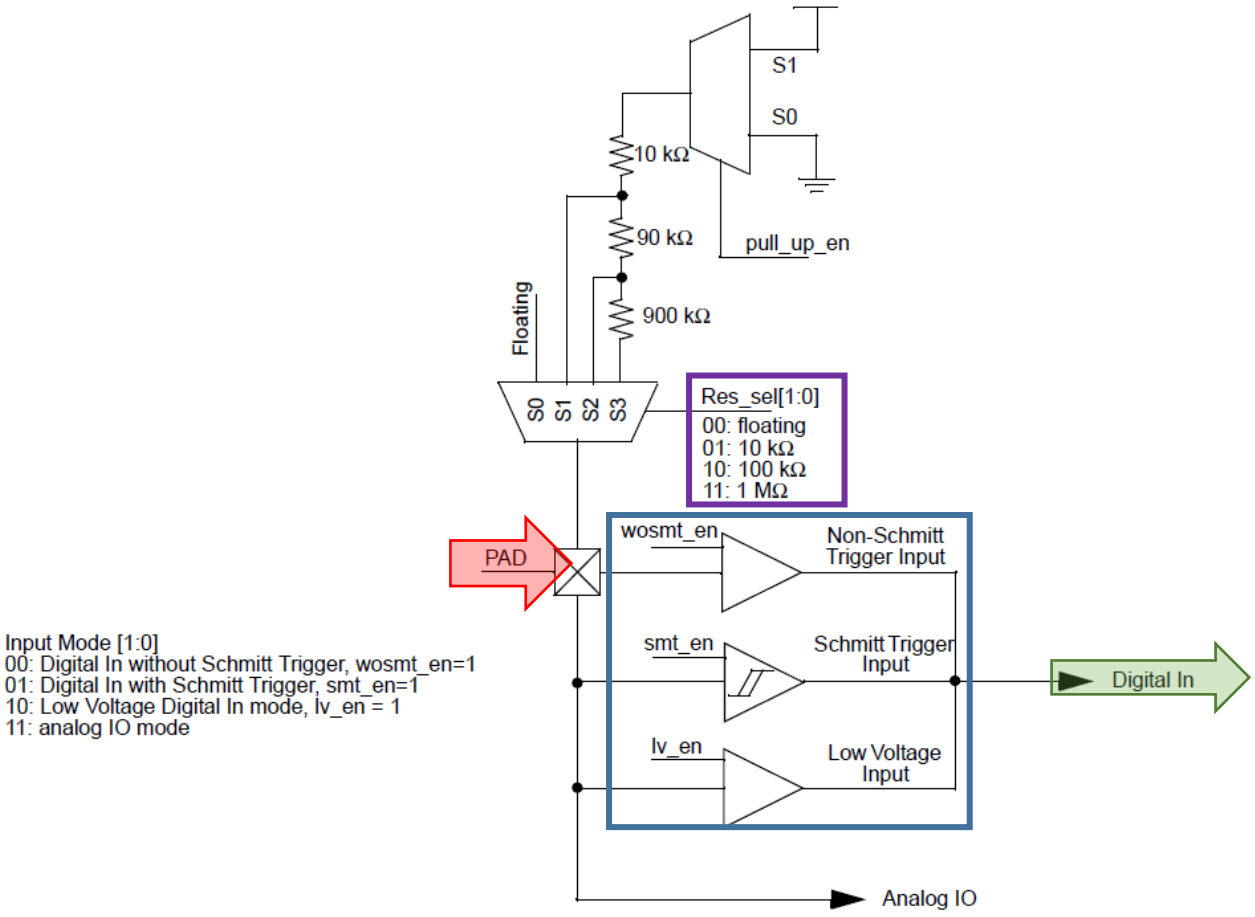
In Groups window user has the option to:

- Add group from analog[A] or digital[D] waveforms;
- Remove waveforms from plot widget;
- Organize order of waveforms;

9. Designing Overview

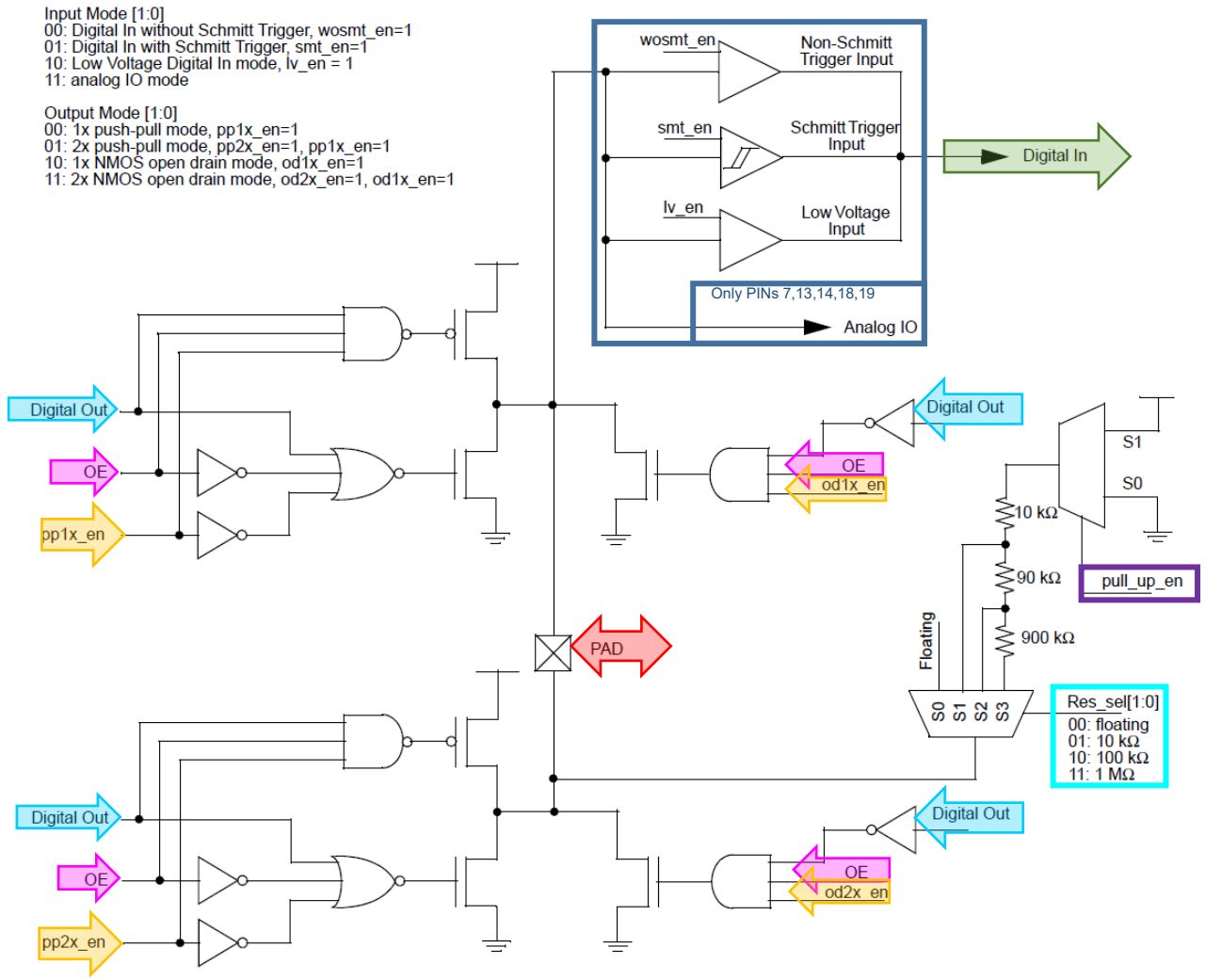
9.1. SLG4672x Properties Interpretation

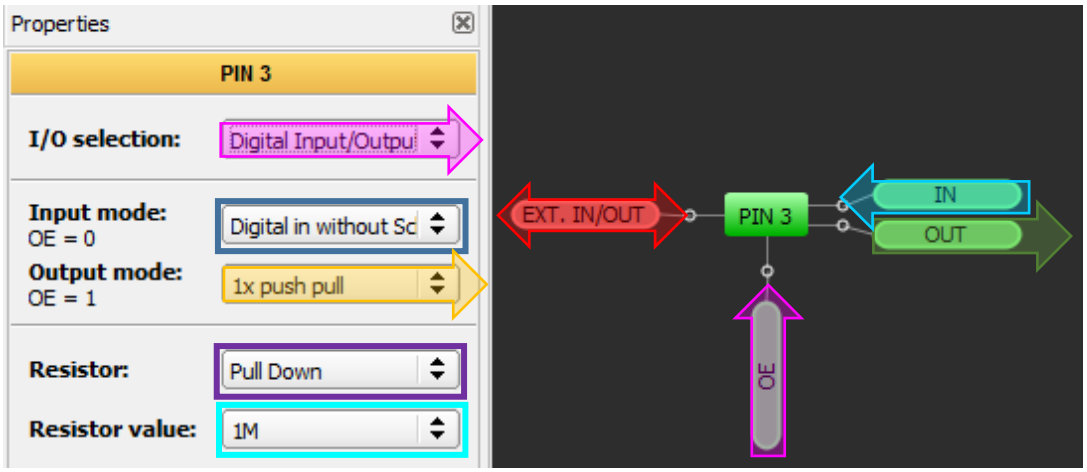
Pin2



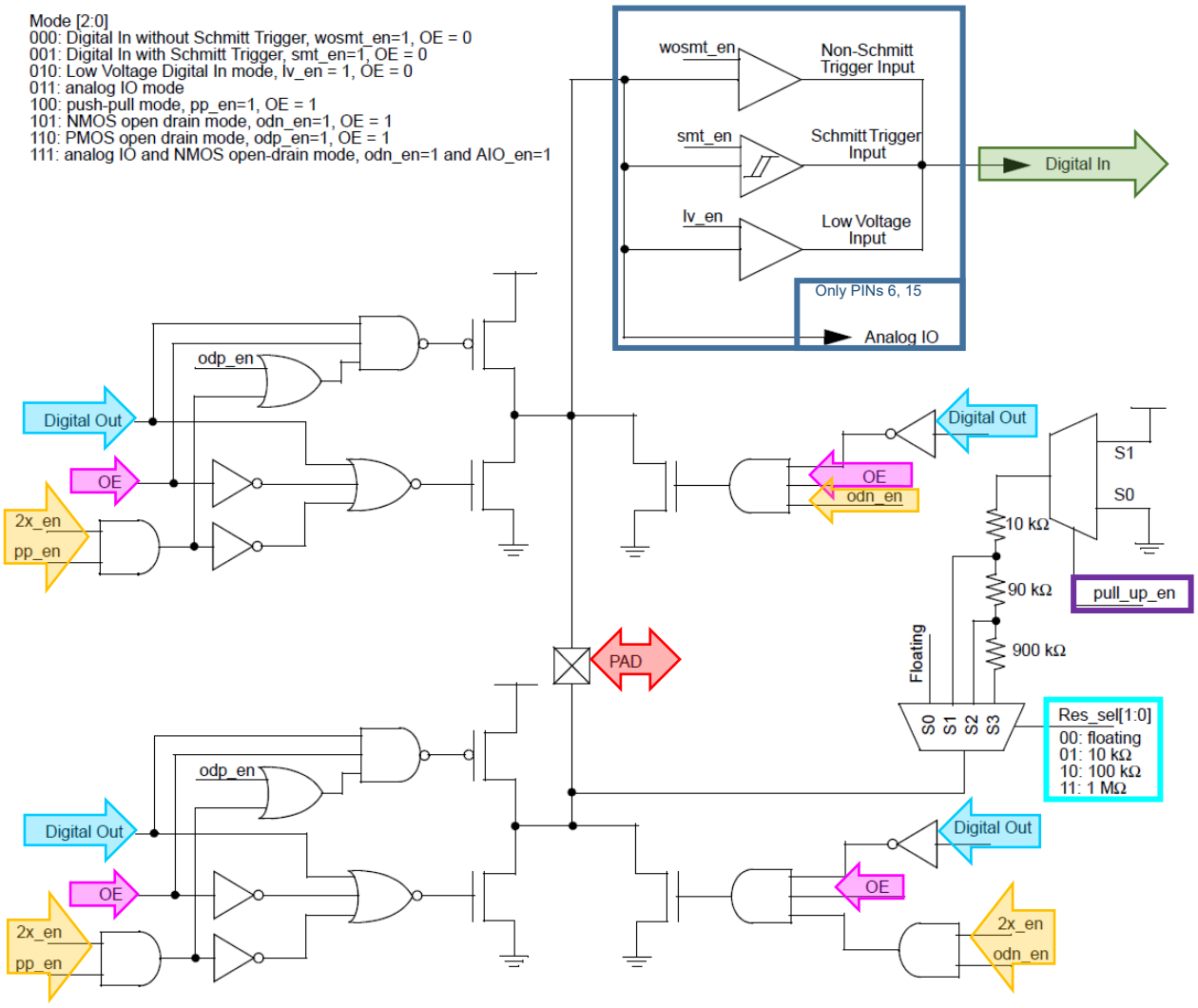

Properties
PIN 2
I/O selection:
Digital Input
Input mode:
OE = 0
Digital in without Sc
Output mode:
OE = 1
None
Resistor:
Pull Down
Resistor value:
1M
EXT. IN
PIN 2
OUT

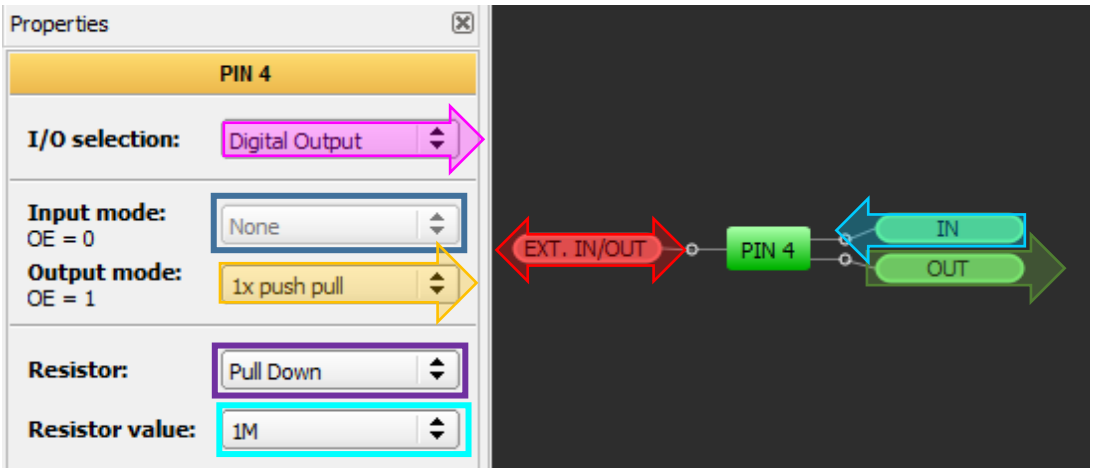
Matrix OE IO Structure (for Pins 3, 5, 7, 9, 13, 14, 16, 18, 19)



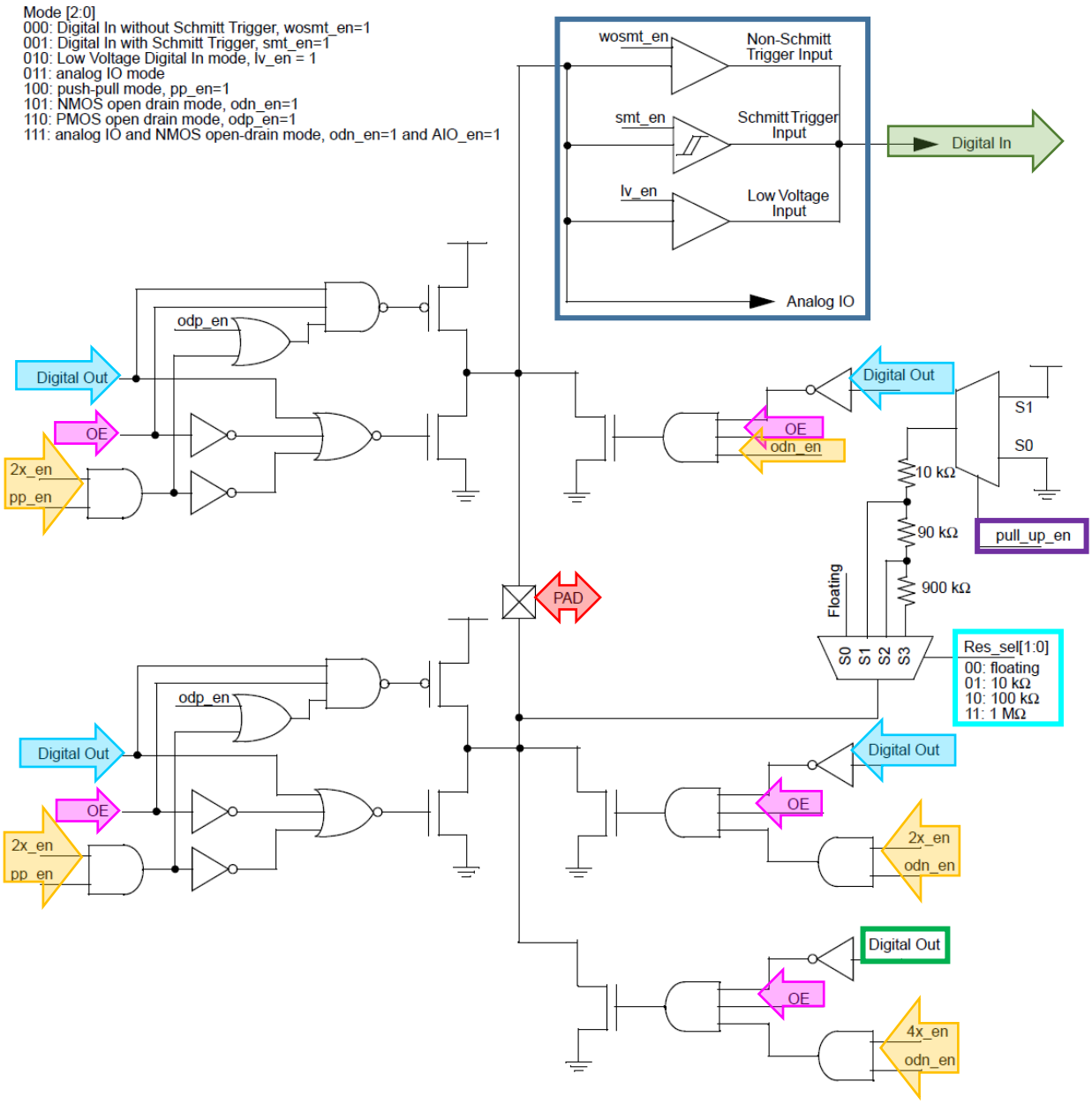

Properties
PIN 3
I/O selection:
Digital Input/Outpu
Input mode:
OE = 0
Digital in without Sc
Output mode:
OE = 1
1x push pull
Resistor:
Pull Down
Resistor value:
1M
EXT. IN/OUT
PIN 3
IN
OUT
OE

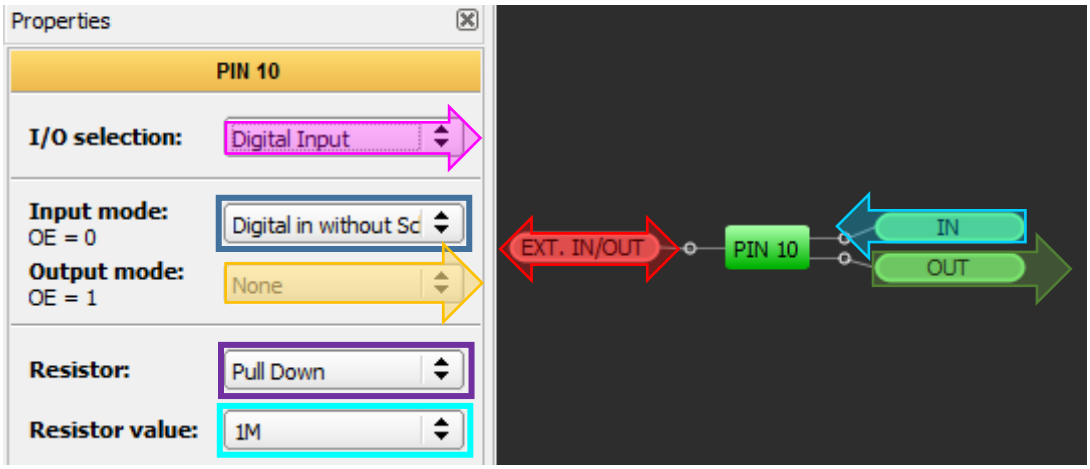
Register OE IO Structure (for Pins 4, 6, 8, 15, 17, 20)



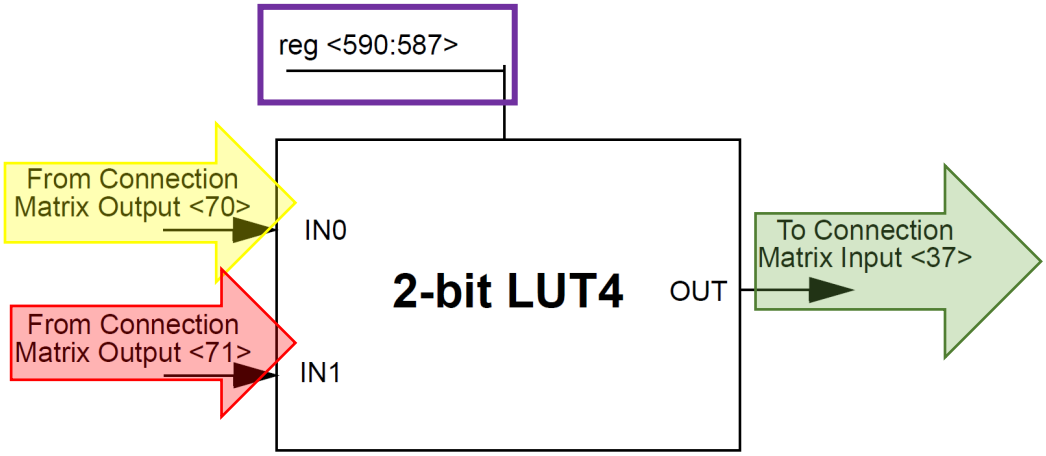

Properties
PIN 4
I/O selection:
Digital Output
Input mode:
OE = 0
None
Output mode:
OE = 1
1x push pull
Resistor:
Pull Down
Resistor value:
1M
EXT. IN/OUT
PIN 4
IN
OUT

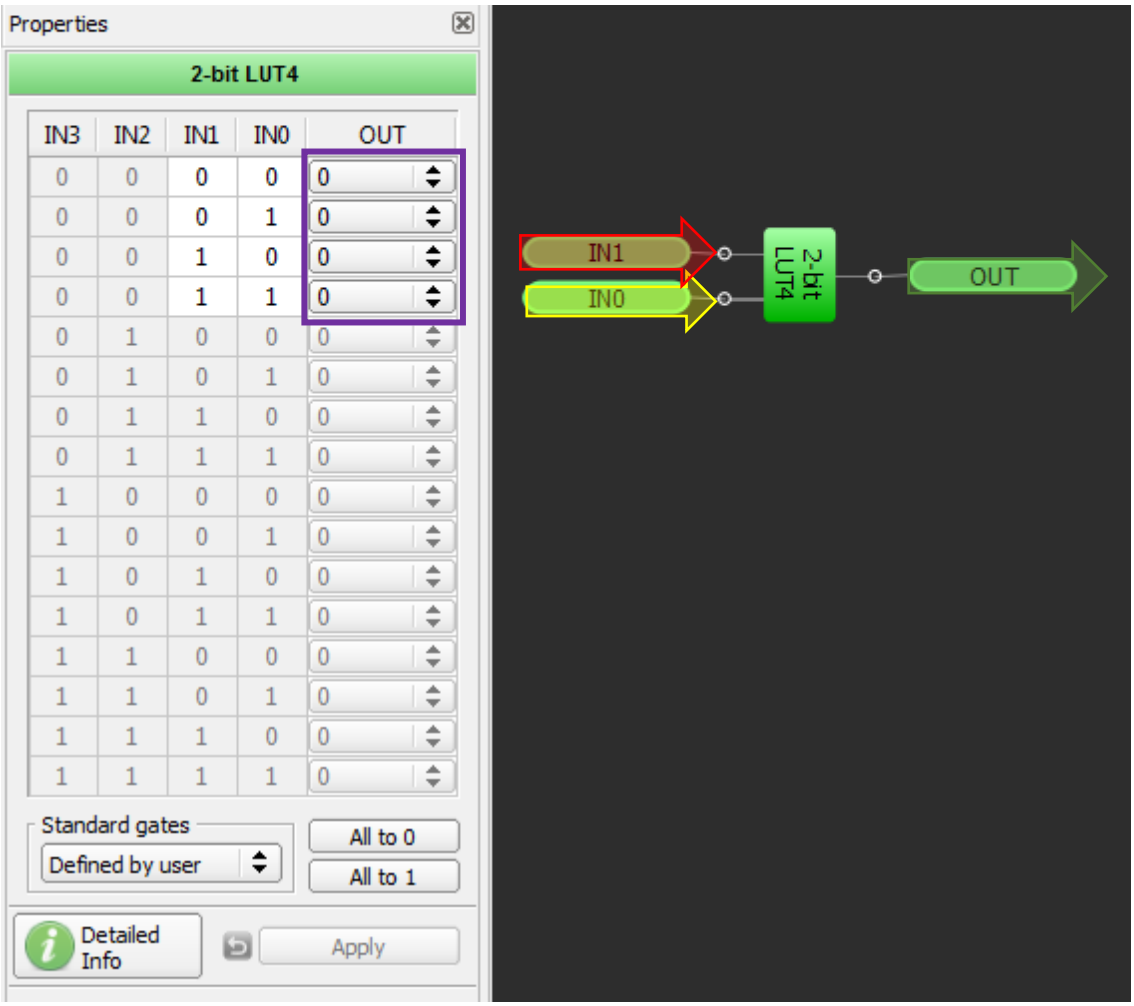
Register OE IO Structure with Super Driver (for Pins 10, 12)



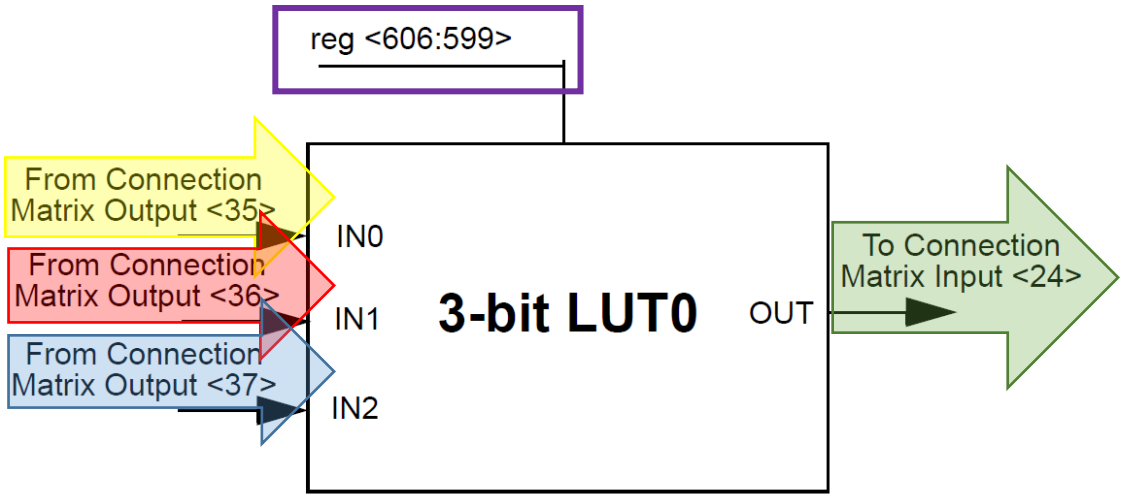

Properties
PIN 10
I/O selection:
Digital Input
Input mode:
OE = 0
Digital in without Sc
Output mode:
OE = 1
None
Resistor:
Pull Down
Resistor value:
1M
EXT. IN/OUT
PIN 10
IN
OUT

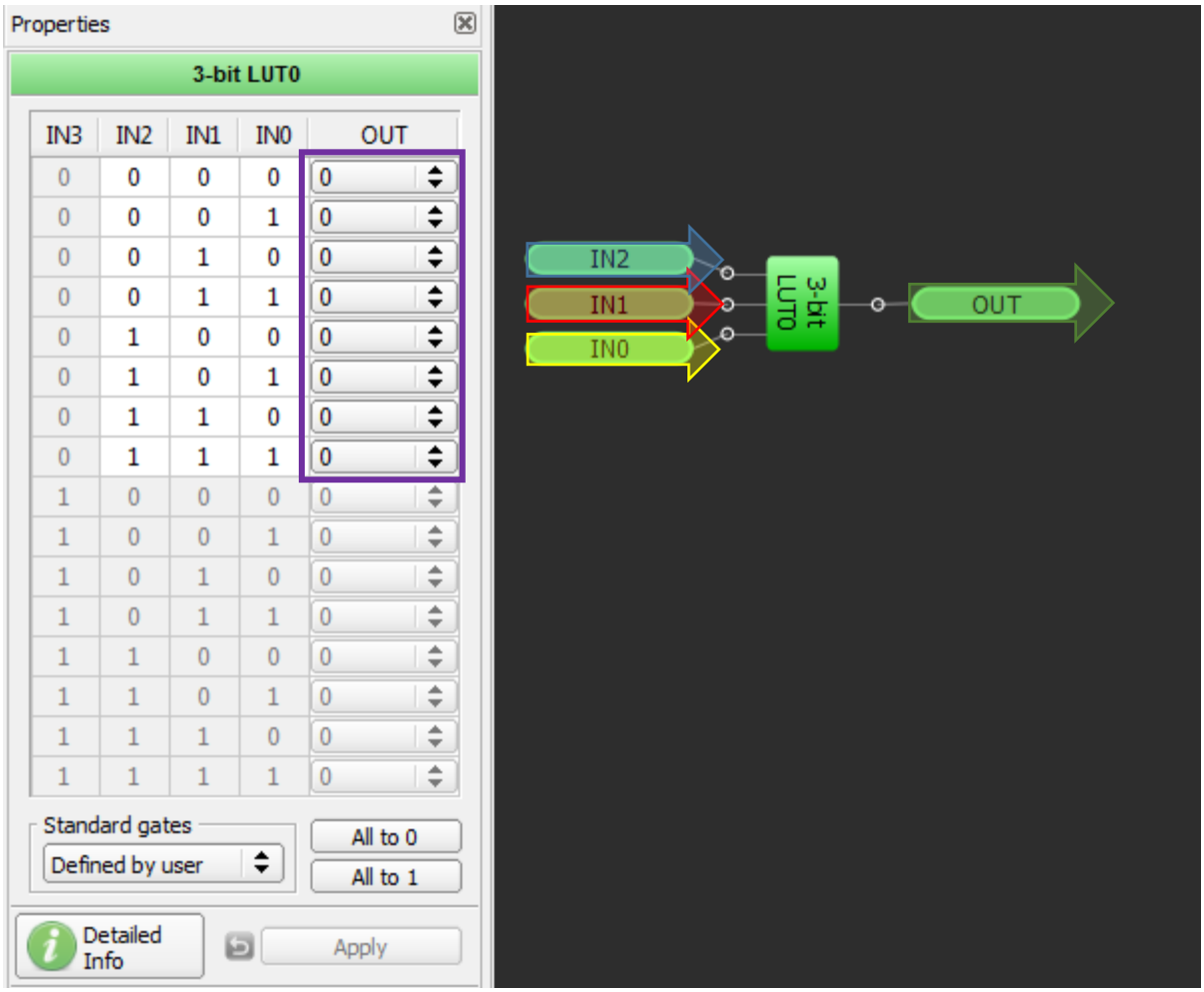
2-Bit LUT


reg <590:587>
From Connection Matrix Output <70>
From Connection Matrix Output <71>
IN0
IN1
2-bit LUT4
OUT
To Connection Matrix Input <37>

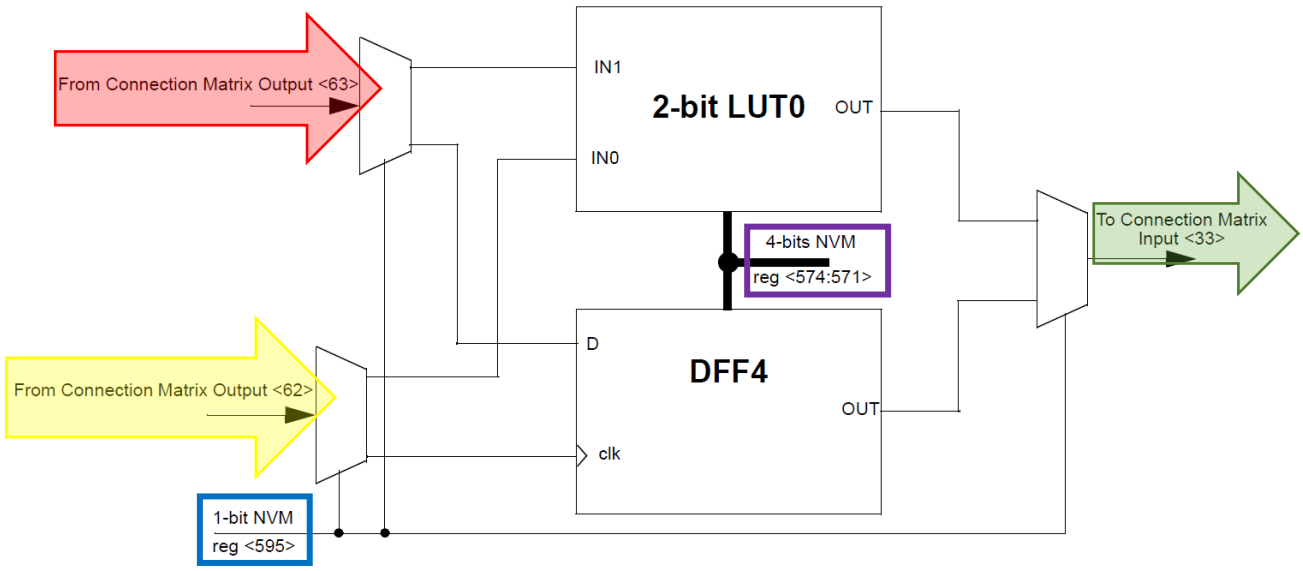

Properties
2-bit LUT4
IN3 IN2 IN1 IN0 OUT
Standard gates
Defined by user
All to 0
All to 1
Detailed Info
Apply
IN1
IN0
2-bit LUT4
OUT

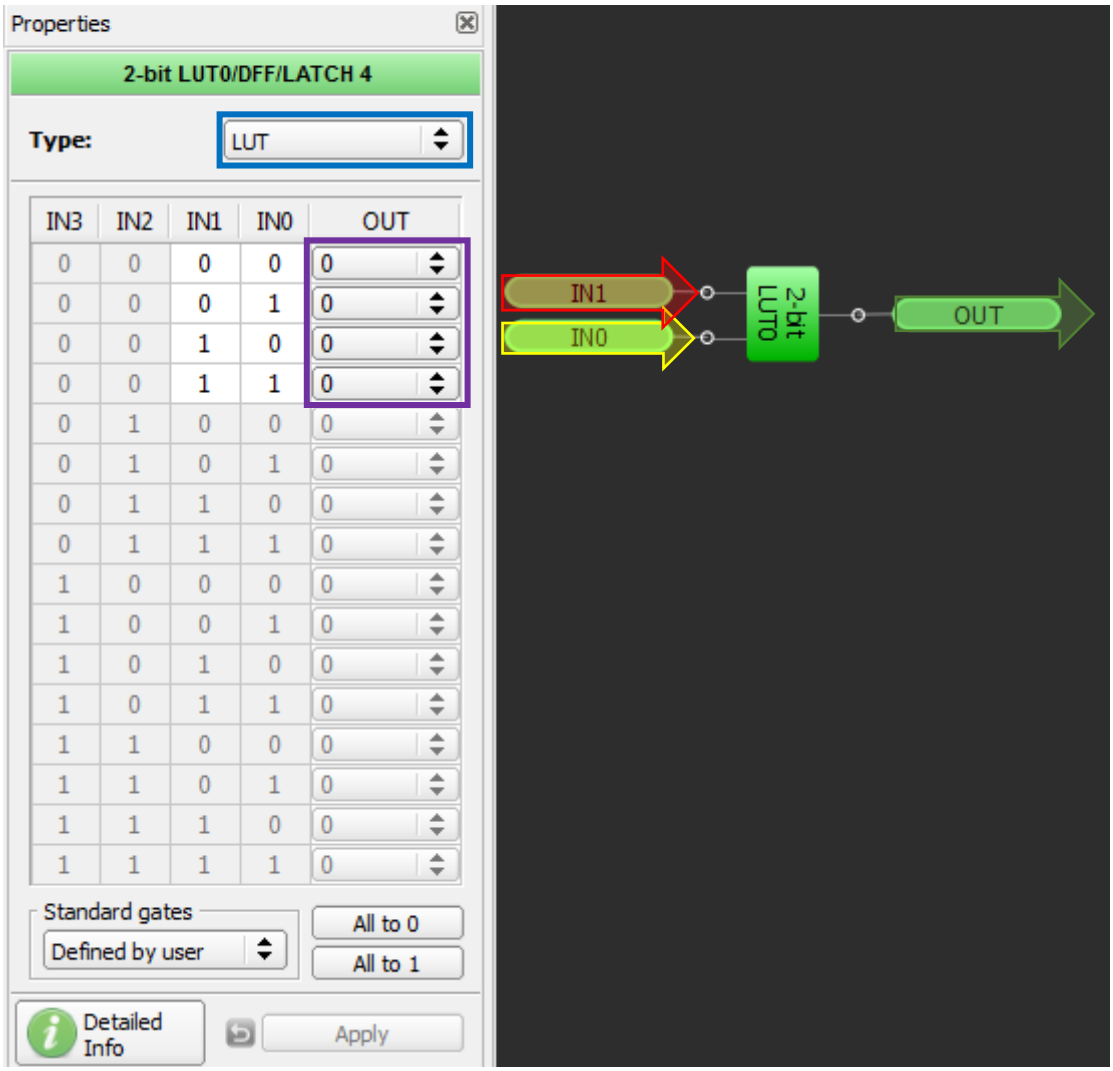
3-Bit LUT

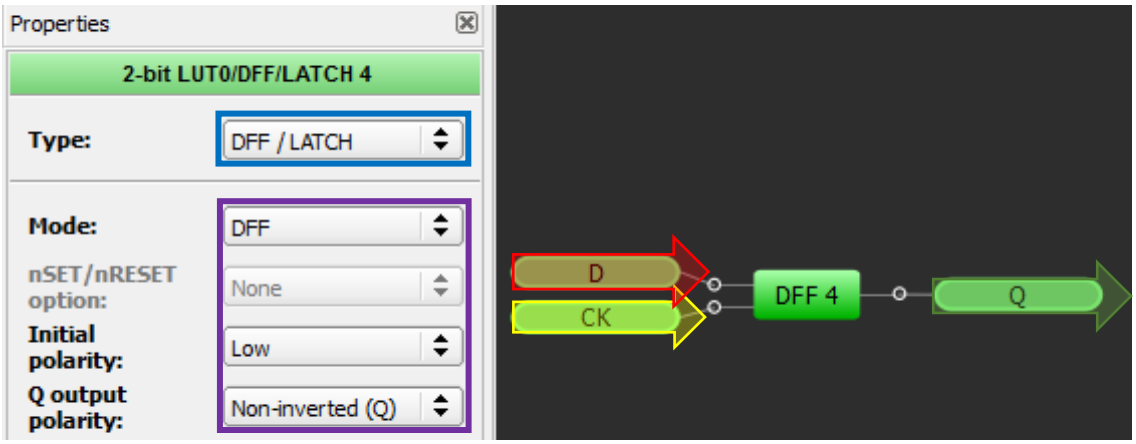

reg <606:599>
From Connection Matrix Output <35>
From Connection Matrix Output <36>
From Connection Matrix Output <37>
IN0
IN1
IN2
3-bit LUT0
OUT
To Connection Matrix Input <24>


Properties
3-bit LUT0
IN3 IN2 IN1 IN0 OUT
0 0 0 0 0
0 0 0 1 0
0 0 1 0 0
0 0 1 1 0
0 1 0 0 0
0 1 0 1 0
0 1 1 0 0
0 1 1 1 0
1 0 0 0 0
1 0 0 1 0
1 0 1 0 0
1 0 1 1 0
1 1 0 0 0
1 1 0 1 0
1 1 1 0 0
1 1 1 1 0
Standard gates
Defined by user
All to 0
All to 1
Detailed Info
Apply
IN2
IN1
IN0
3-bit LUT0
OUT

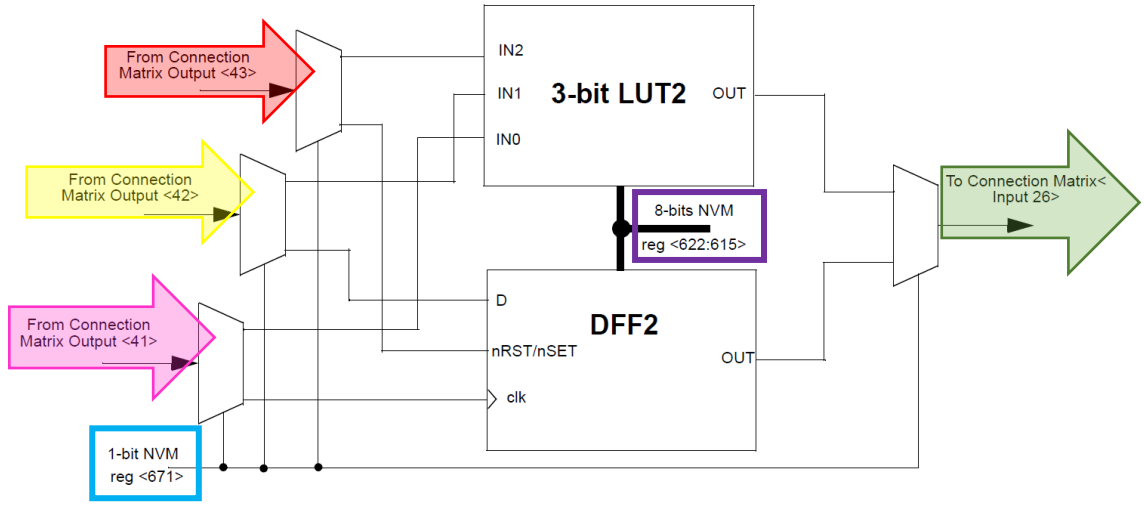
2-Bit LUT or D Flip Flop Macrocells

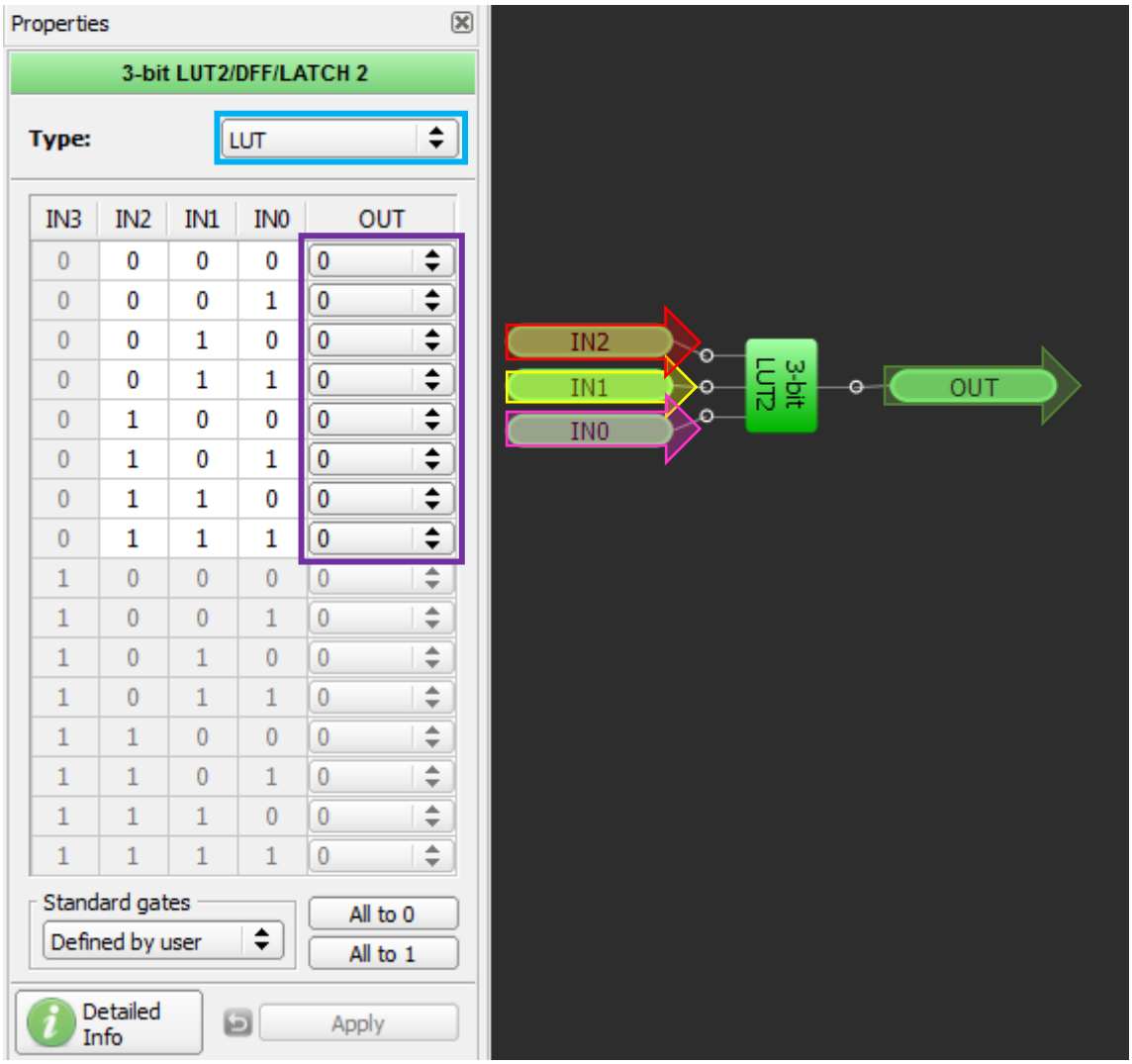


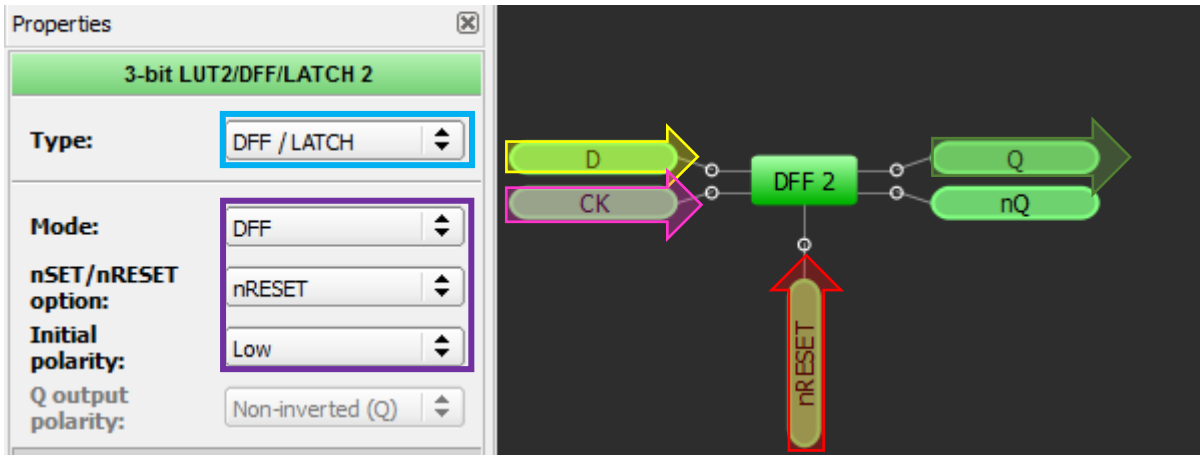

Properties
2-bit LUT0/DFF/LATCH 4
Type:
LUT
IN3 IN2 IN1 IN0 OUT
Standard gates
Defined by user
All to 0
All to 1
Detailed Info
Apply
IN1
IN0
2-bit LUT0
OUT


Properties
2-bit LUT0/DFF/LATCH 4
Type:
DFF / LATCH
Mode:
DFF
nSET/nRESET option:
None
Initial polarity:
Low
Q output polarity:
Non-inverted (Q)
D
CK
DFF 4
Q

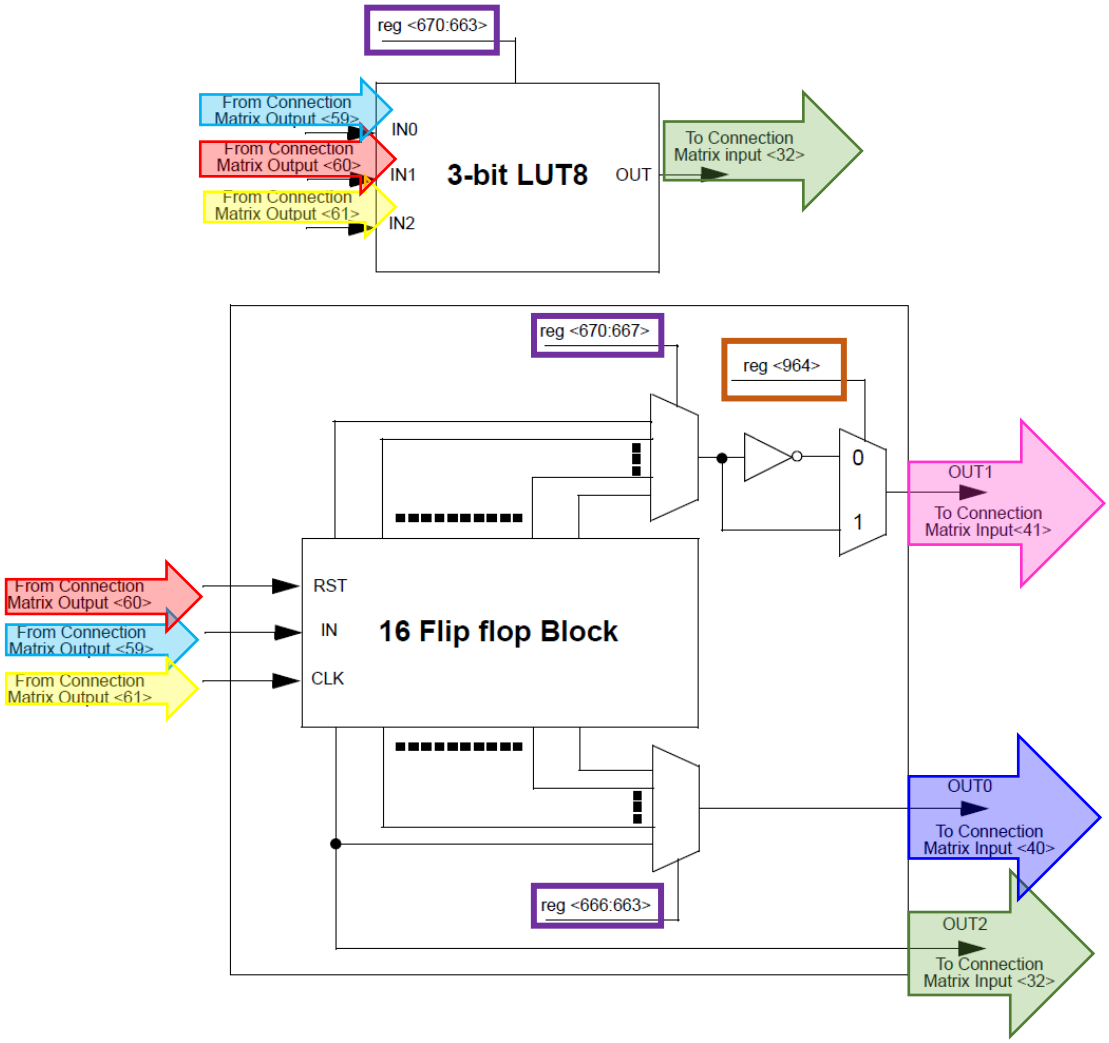
3-Bit LUT or D Flip Flop with Set/Reset Macrocells

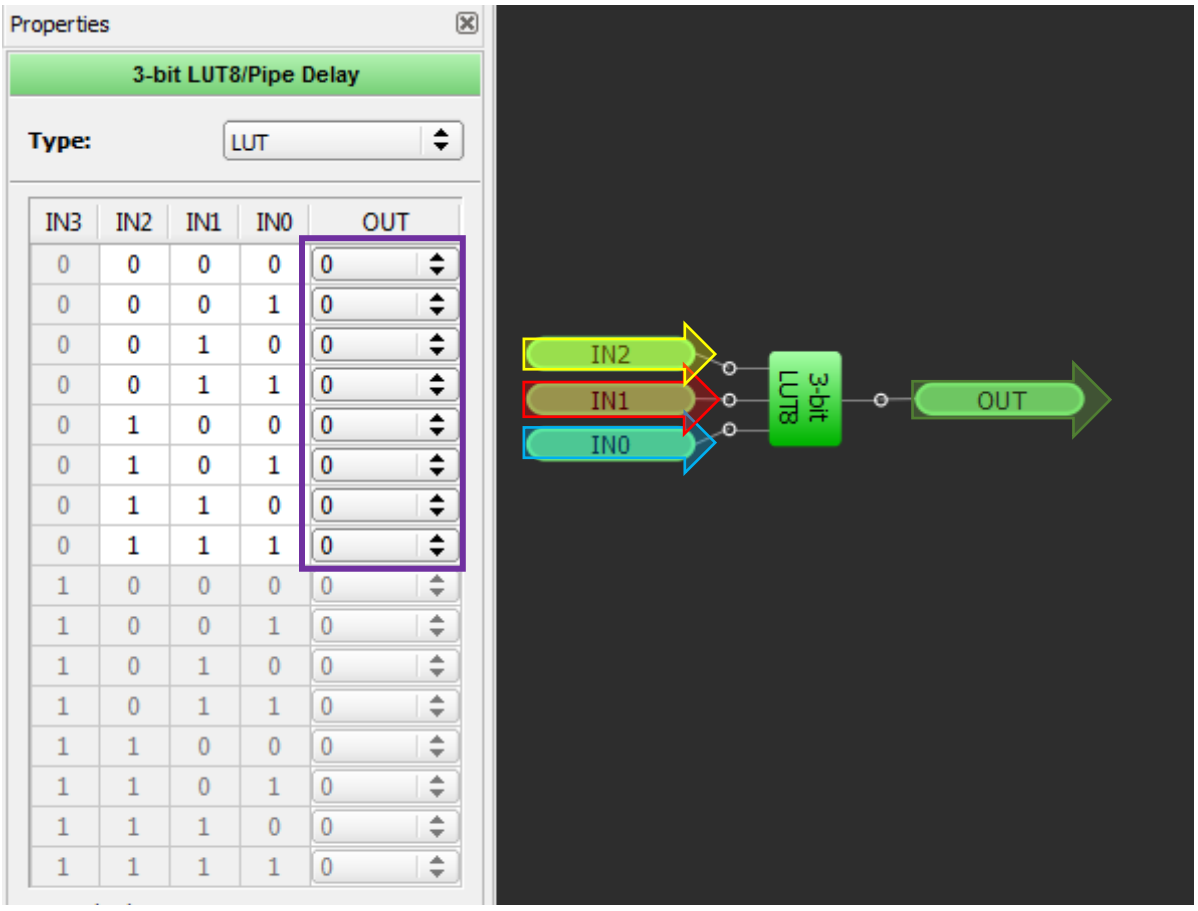


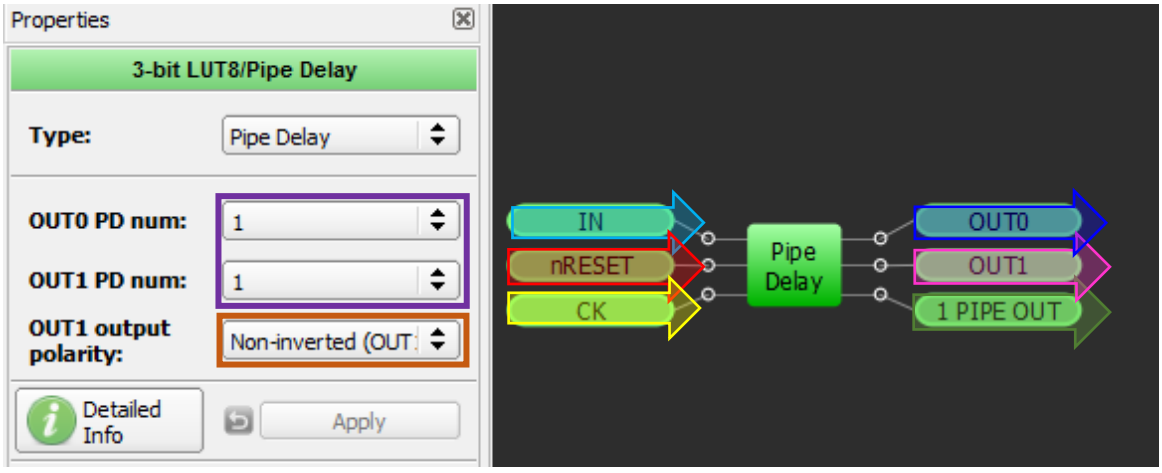

Properties
3-bit LUT2/DFF/LATCH 2
Type:
LUT
IN3 IN2 IN1 IN0 OUT
Standard gates
Defined by user
All to 0
All to 1
Detailed Info
Apply
IN2
IN1
IN0
3-bit LUT2
OUT


Properties
3-bit LUT2/DFF/LATCH 2
Type:
DFF / LATCH
Mode:
DFF
nSET/nRESET option:
nRESET
Initial polarity:
Low
Q output polarity:
Non-inverted (Q)
D
CK
DFF 2
Q
nQ
nRESET

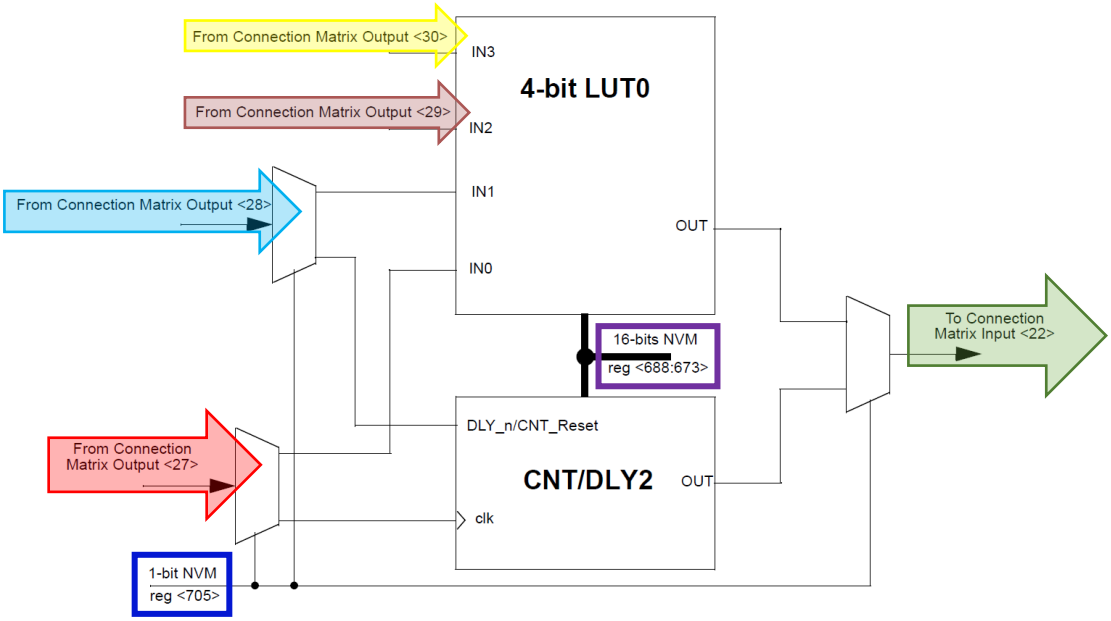
3-Bit LUT or Pipe Delay Macrocell

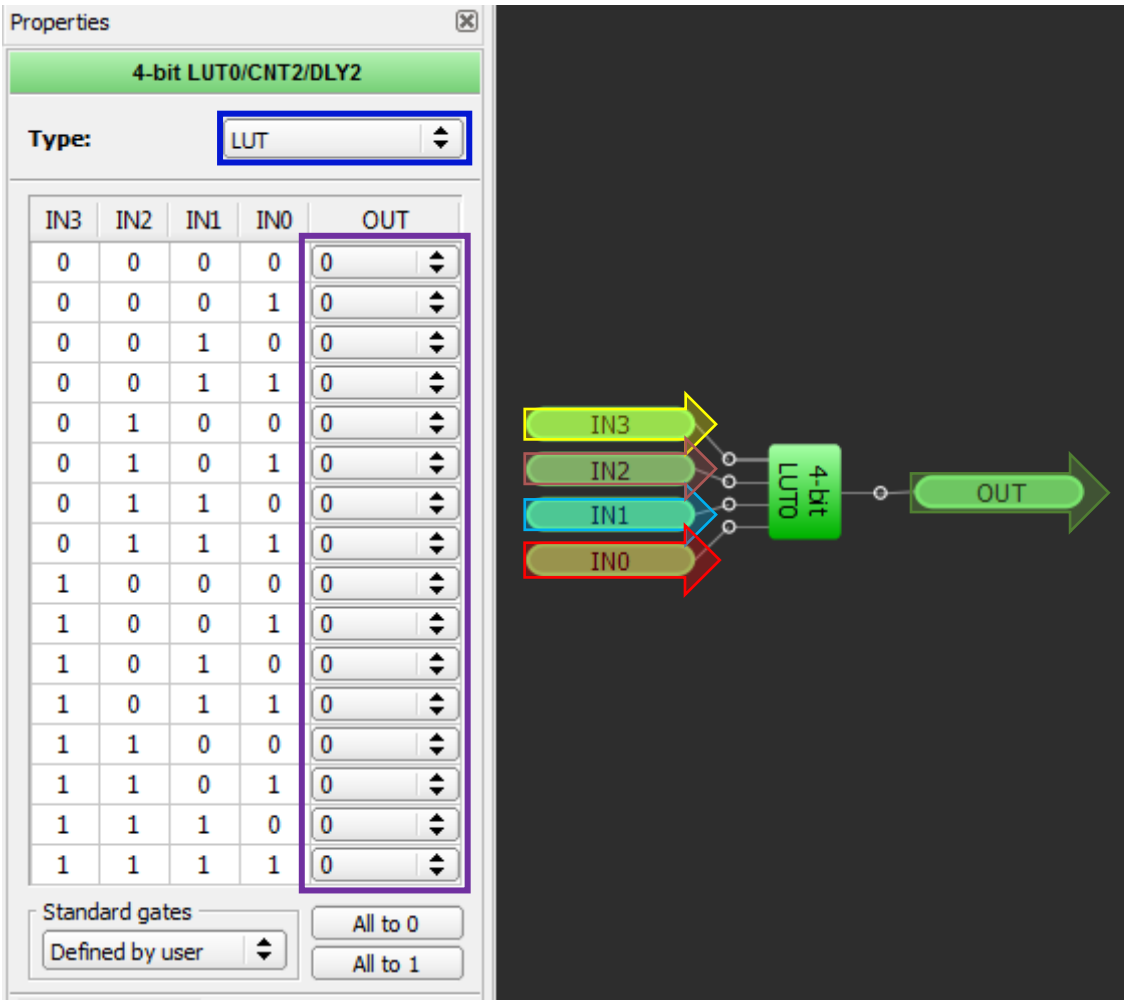


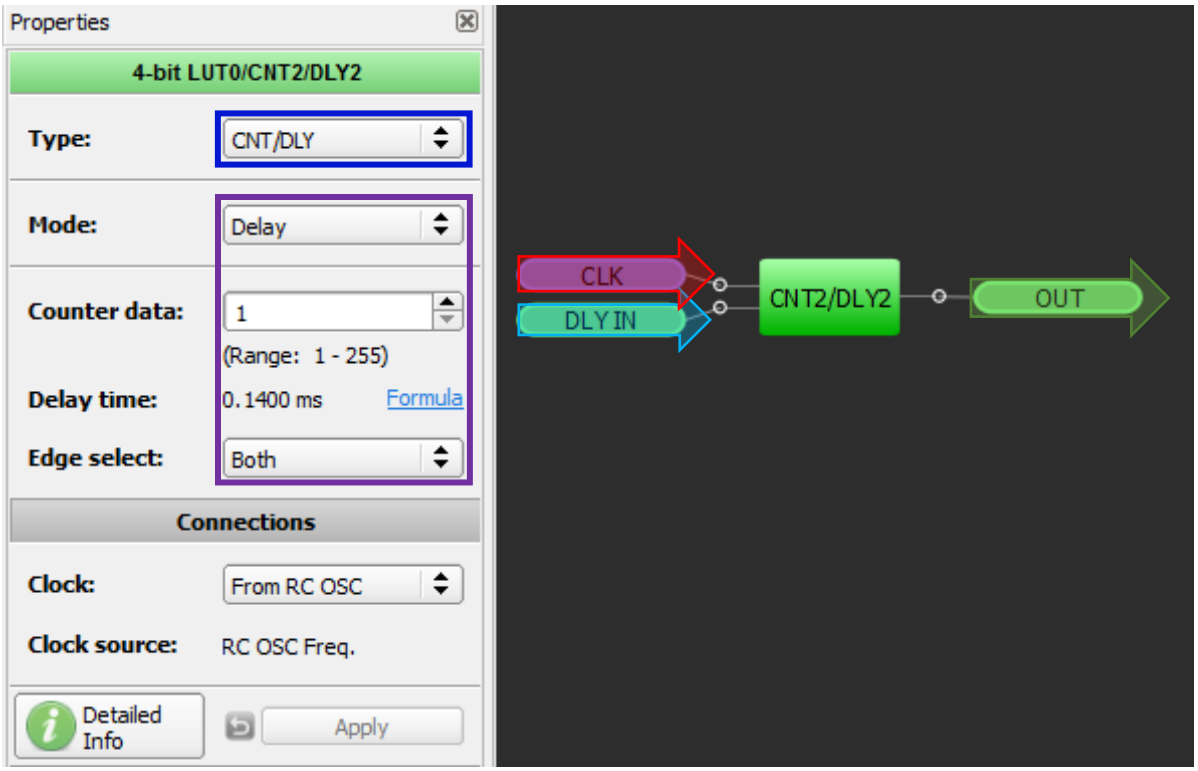

Properties
3-bit LUT8/Pipe Delay
Type:
LUT
IN3 IN2 IN1 IN0 OUT
IN2
IN1
IN0
3-bit LUT8
OUT


Properties
3-bit LUT8/Pipe Delay
Type:
Pipe Delay
OUT0 PD num:
1
OUT1 PD num:
1
OUT1 output polarity:
Non-inverted (OUT
Detailed Info
Apply
IN
nRESET
CK
Pipe Delay
OUT0
OUT1
1 PIPE OUT

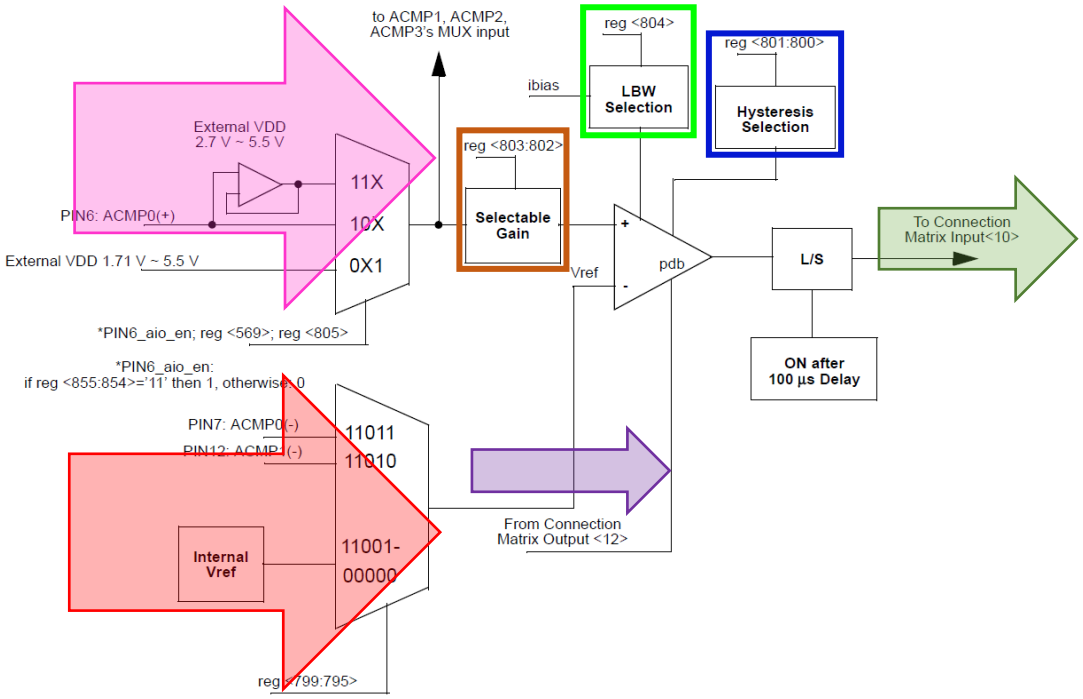
4-Bit LUT or 8- Bit Counter / Delay Macrocells

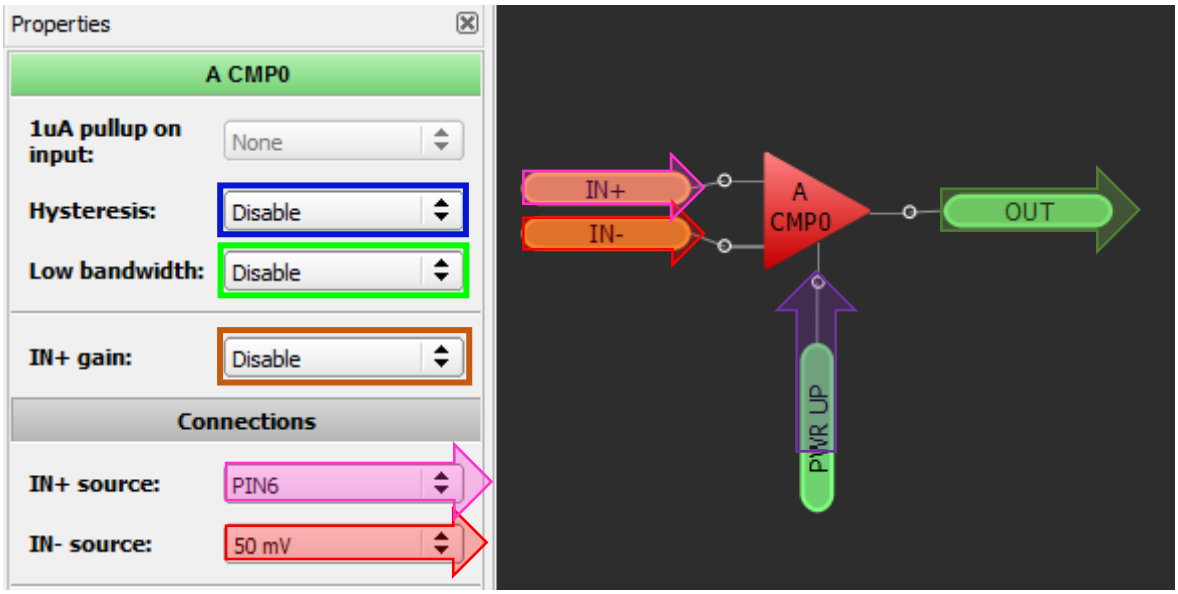



Properties
4-bit LUT0/CNT2/DLY2
Type:
LUT
IN3 IN2 IN1 IN0 OUT
Standard gates
Defined by user
All to 0
All to 1
IN3
IN2
IN1
IN0
4-bit LUT0
OUT

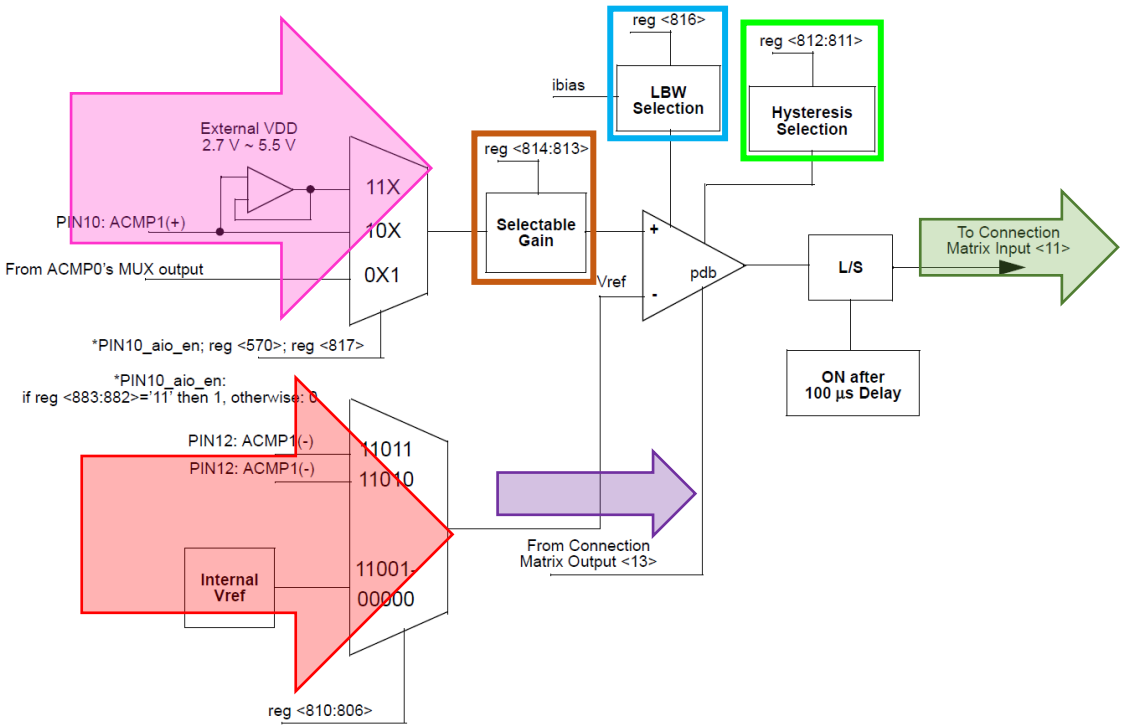

Properties
4-bit LUT0/CNT2/DLY2
Type:
CNT/DLY
Mode:
Delay
Counter data:
1
(Range: 1 - 255)
Delay time:
0.1400 ms
Formula
Edge select:
Both
Connections
Clock:
From RC OSC
Clock source:
RC OSC Freq.
Detailed Info
Apply
CLK
DLY IN
CNT2/DLY2
OUT

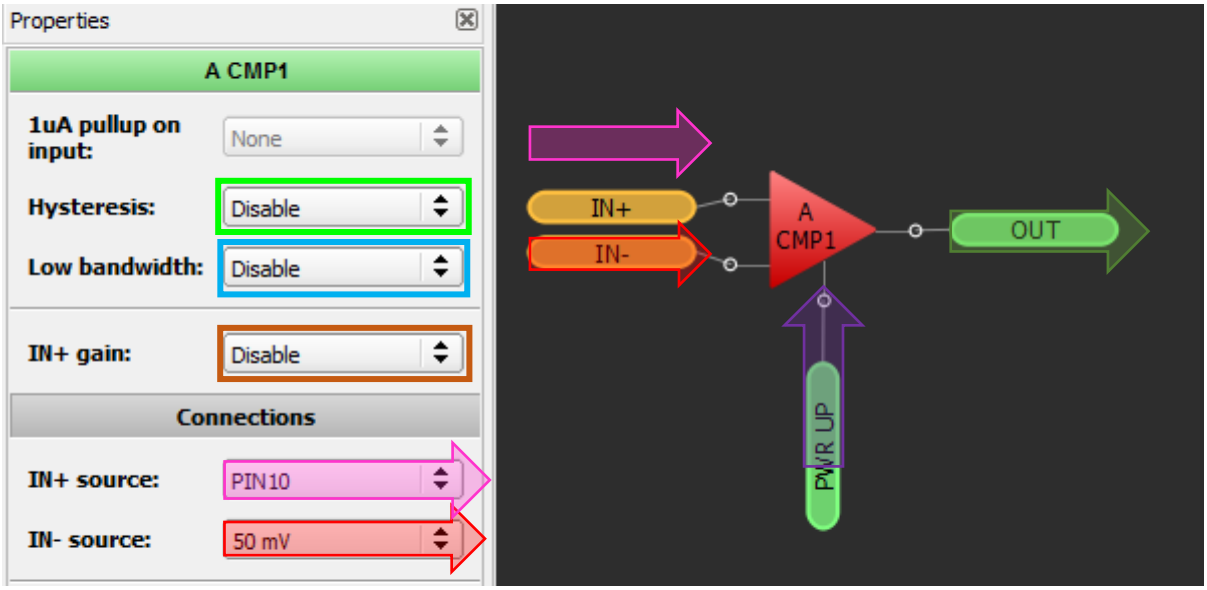
ACMP0 Block Diagram



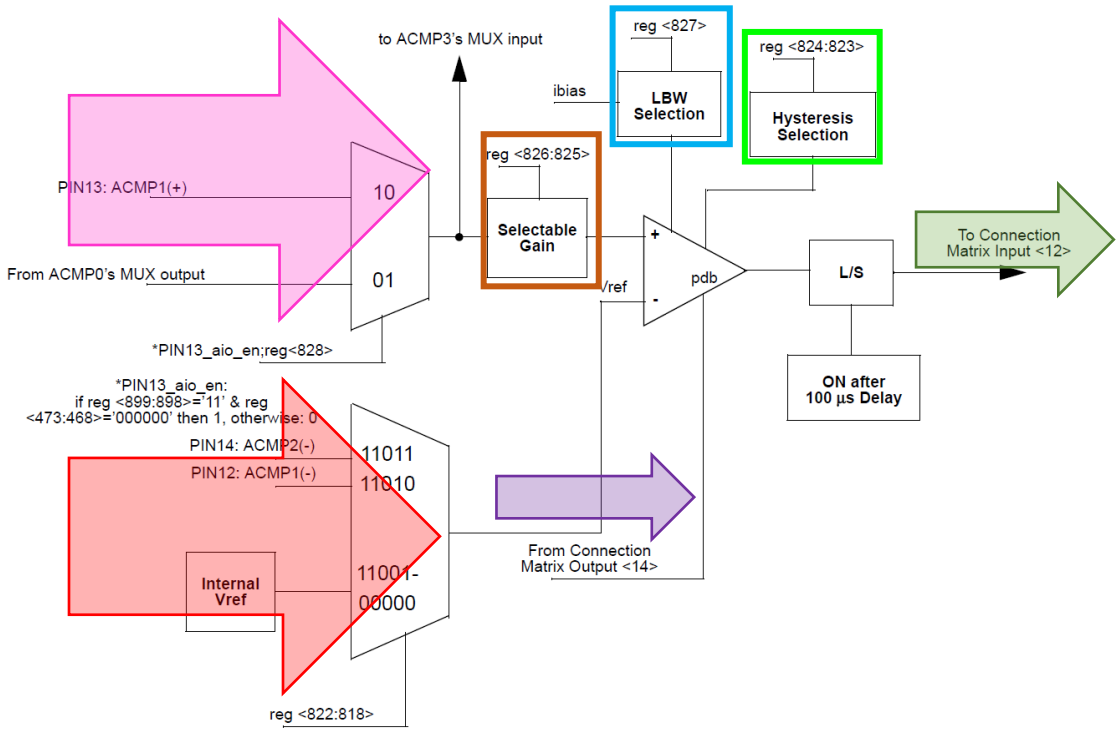


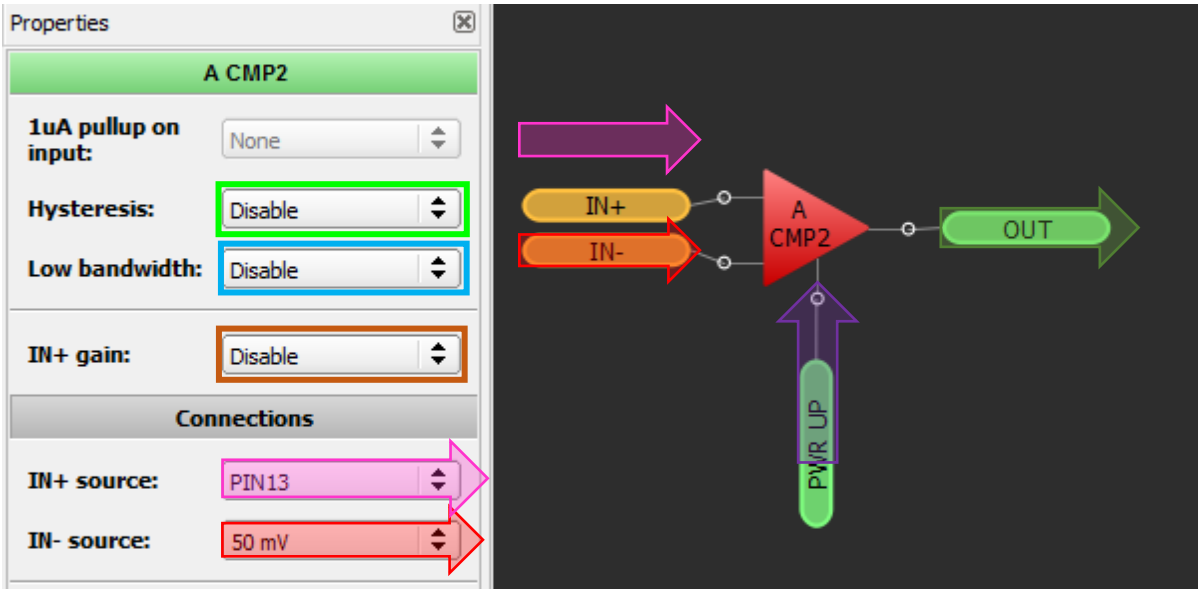
ACMP1 Block Diagram



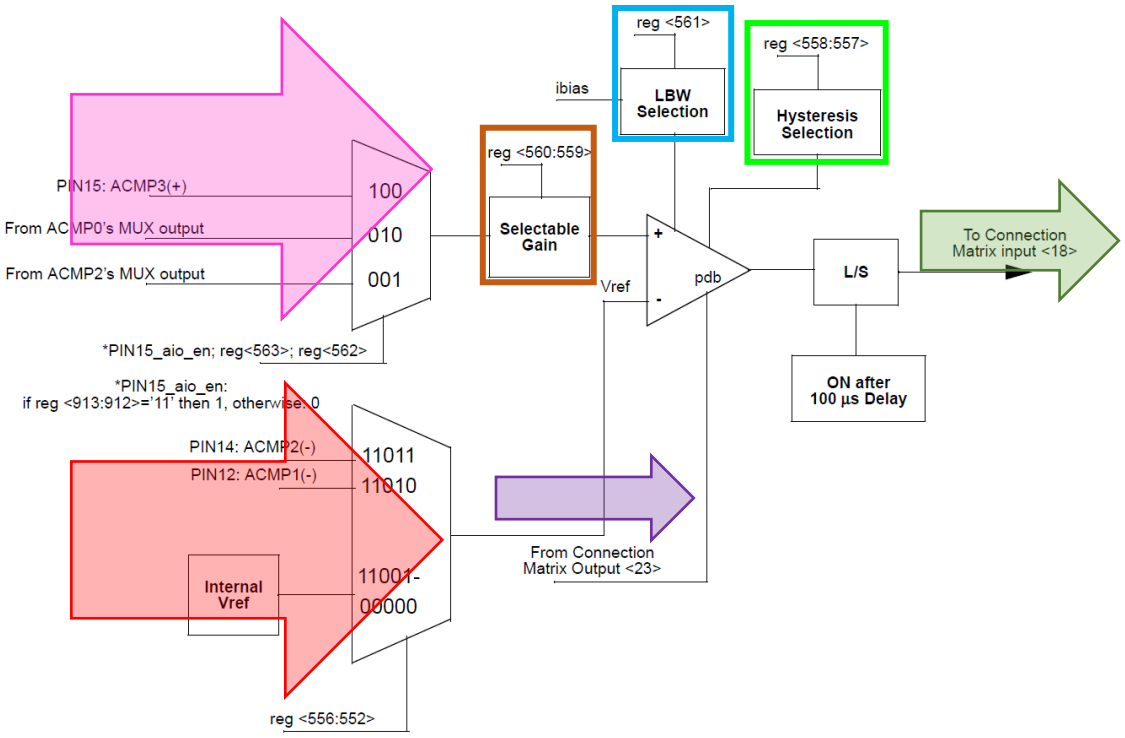


ACMP2 Block Diagram

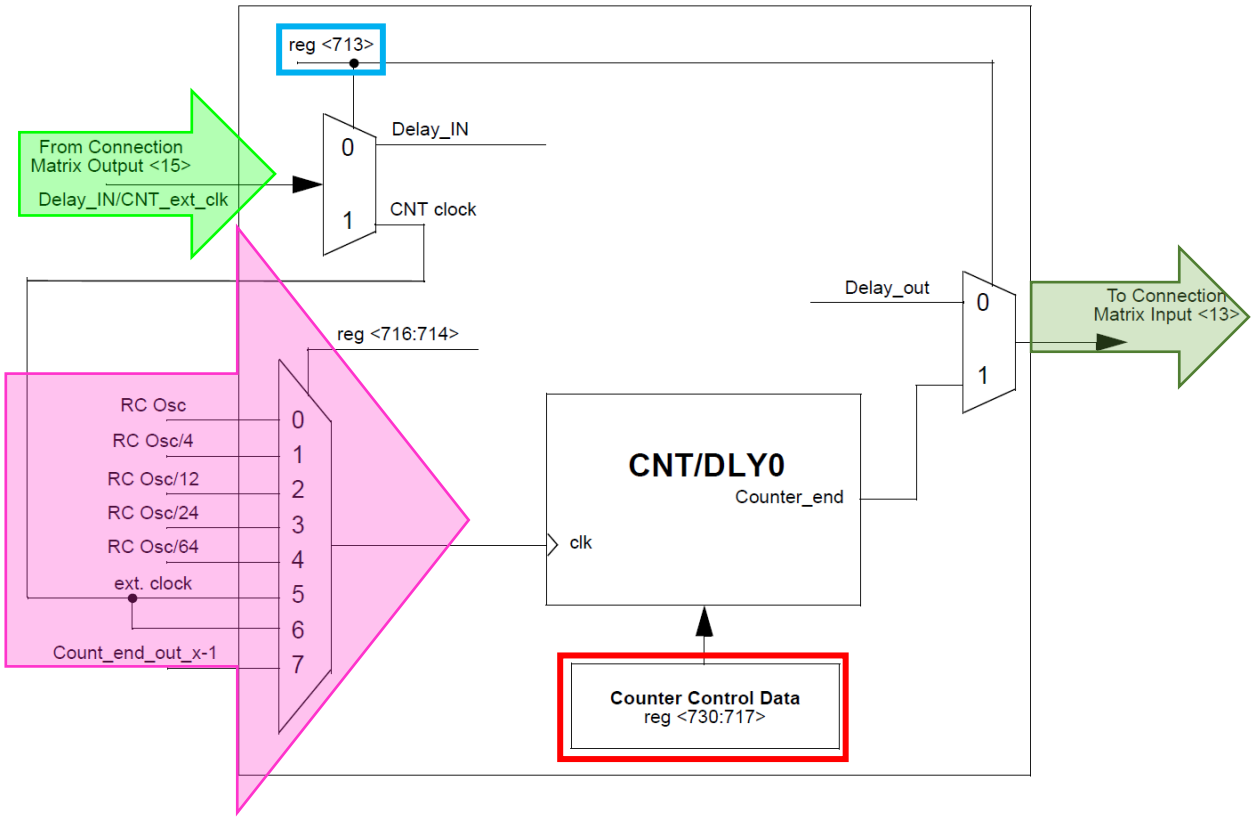


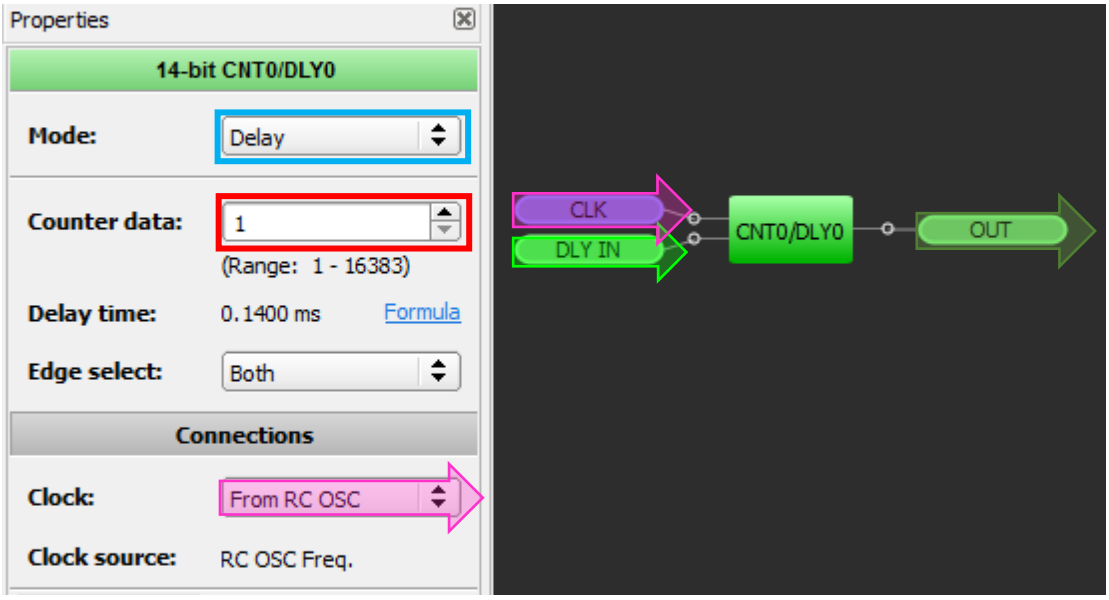


ACMP3 Block Diagram

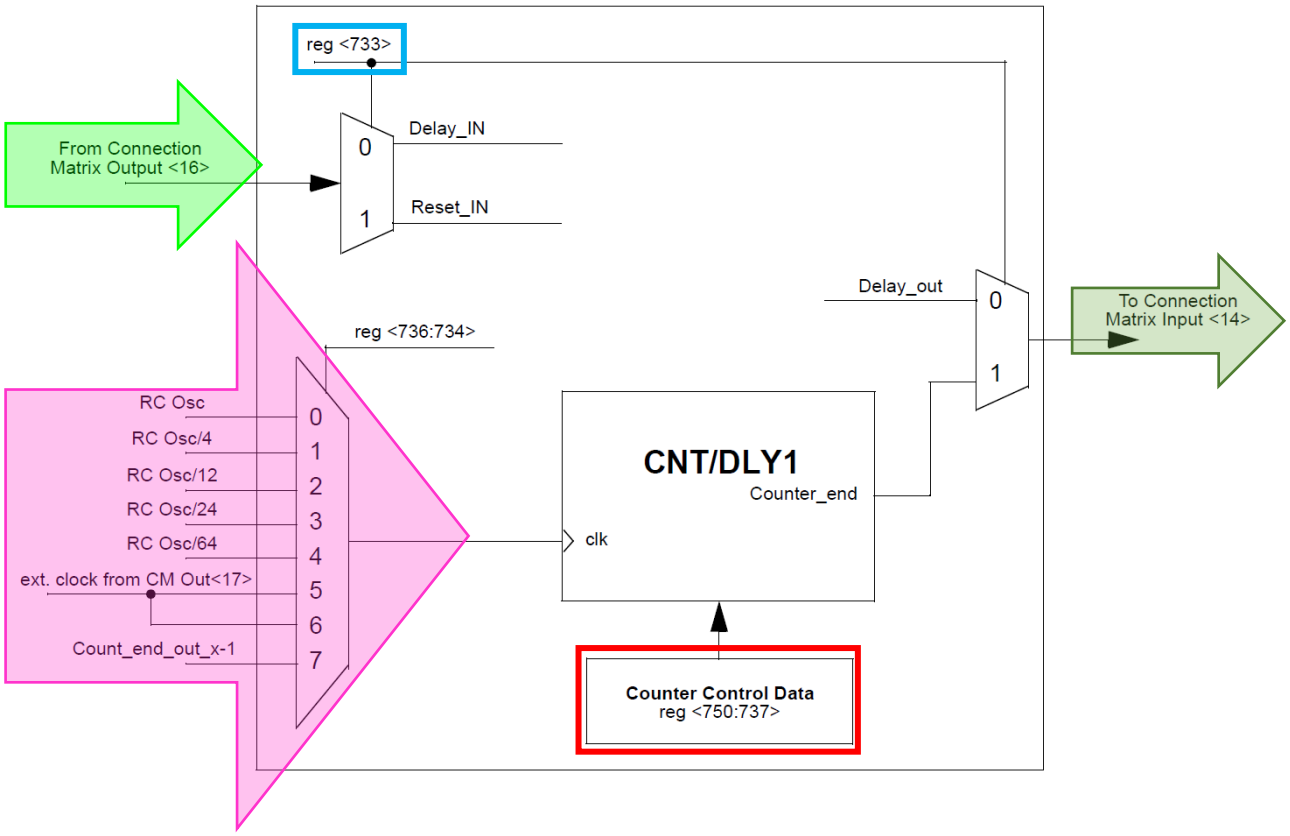


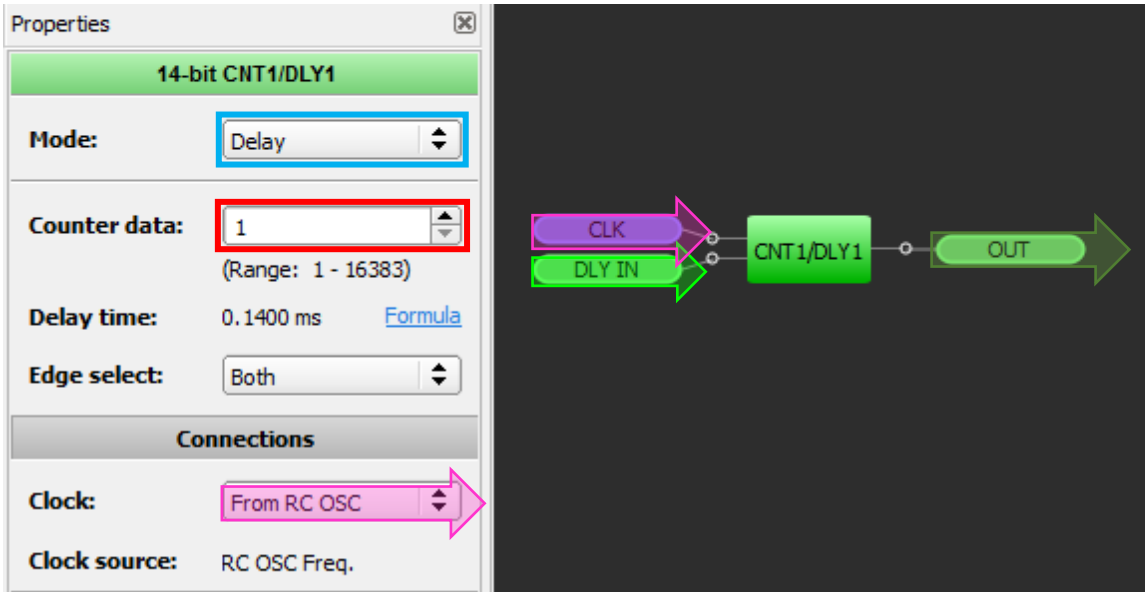
Counters/Delay Generators (CNT/DLY0, 4, 5, 6)



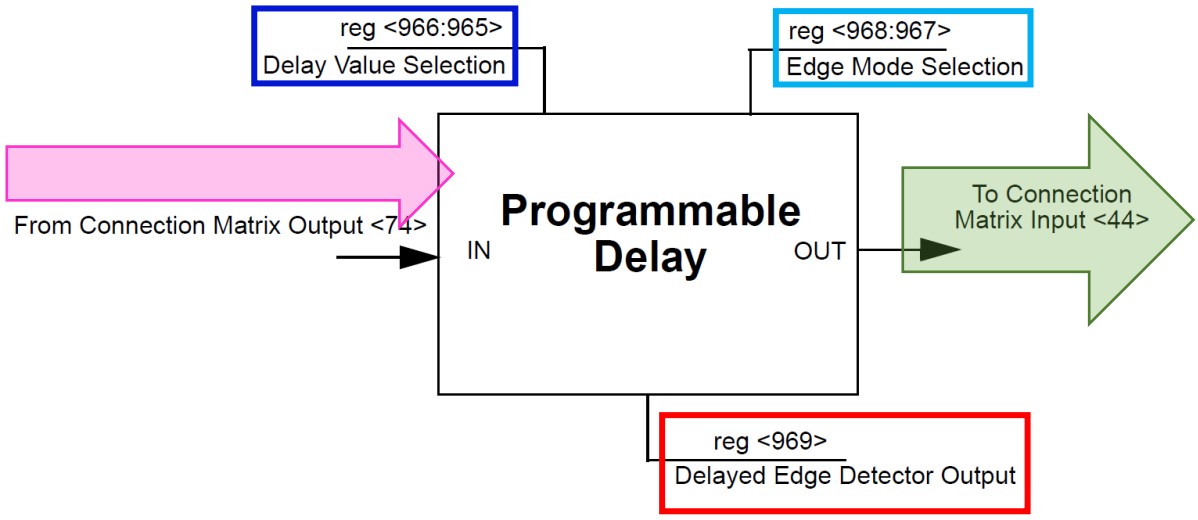


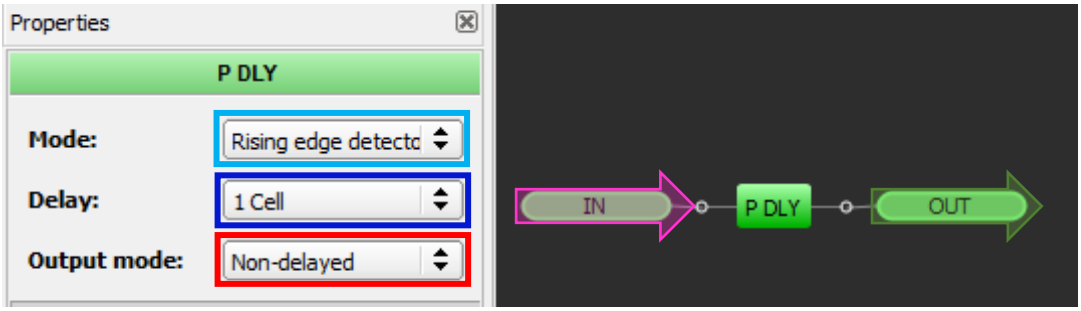
Counters/Delay Generators (CNT/DLY1)



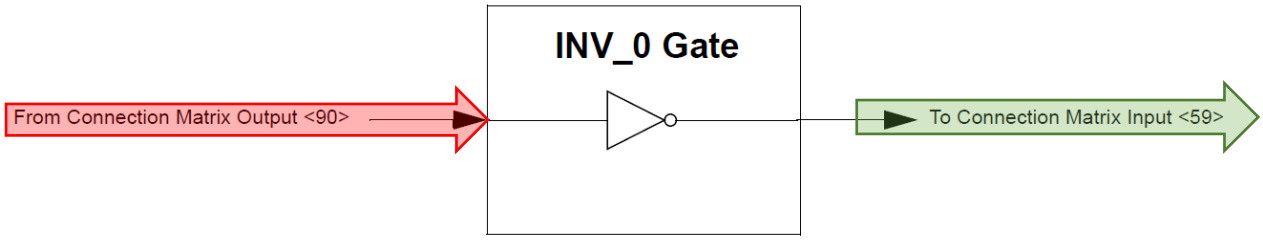


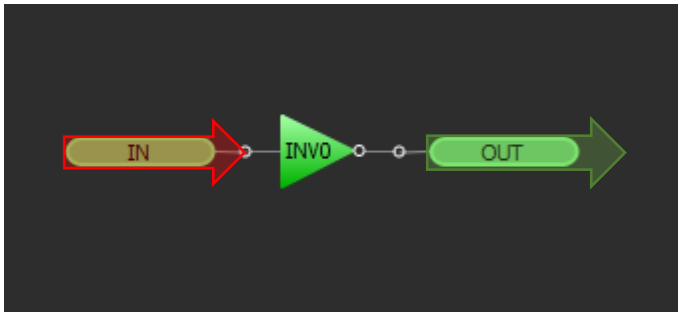
Programmable Delay / Edge Detector



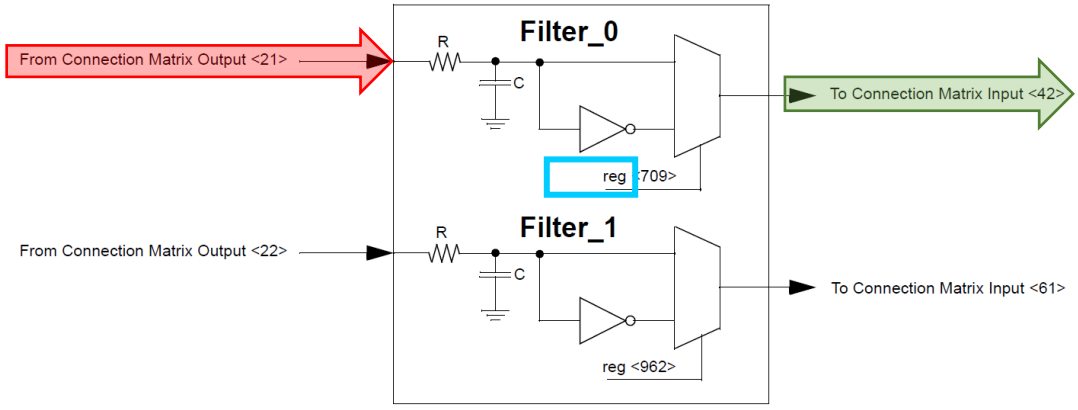


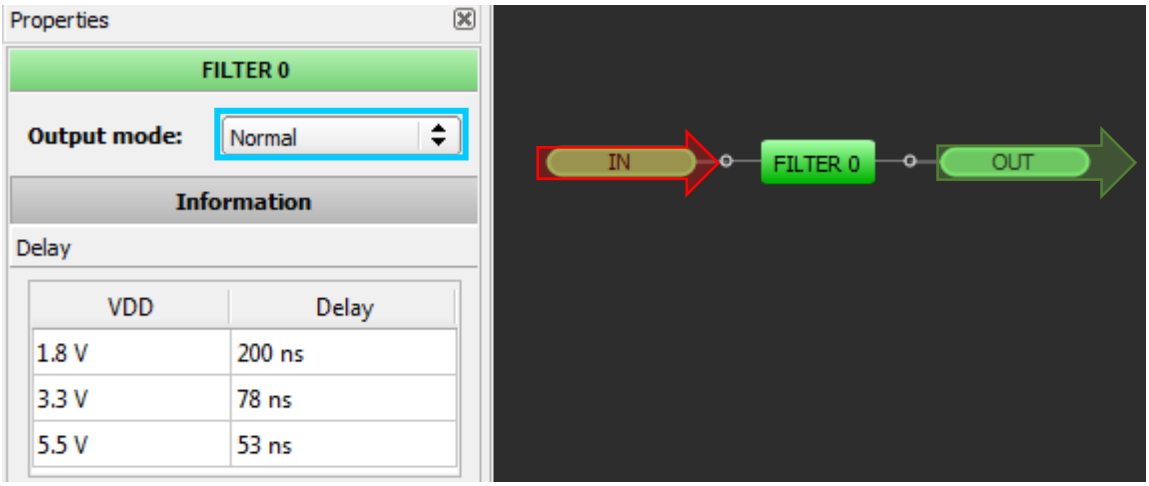
INV 0, INV 1 Gate



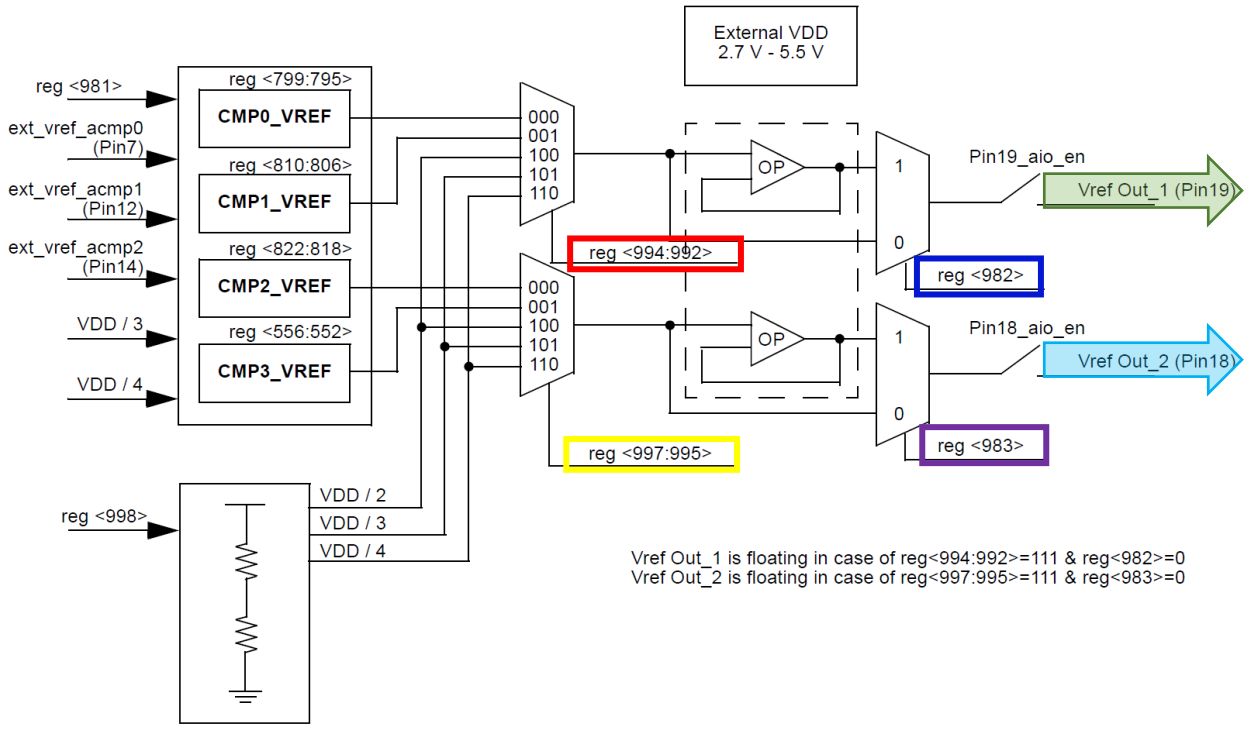


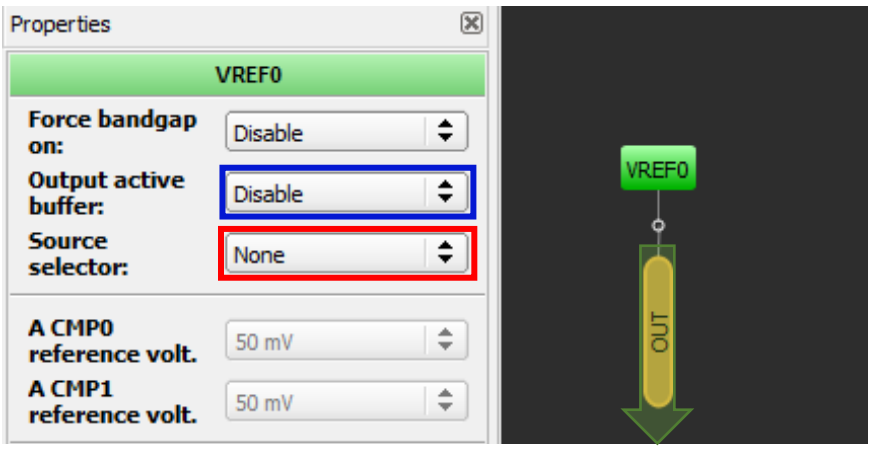
Deglitch Filter

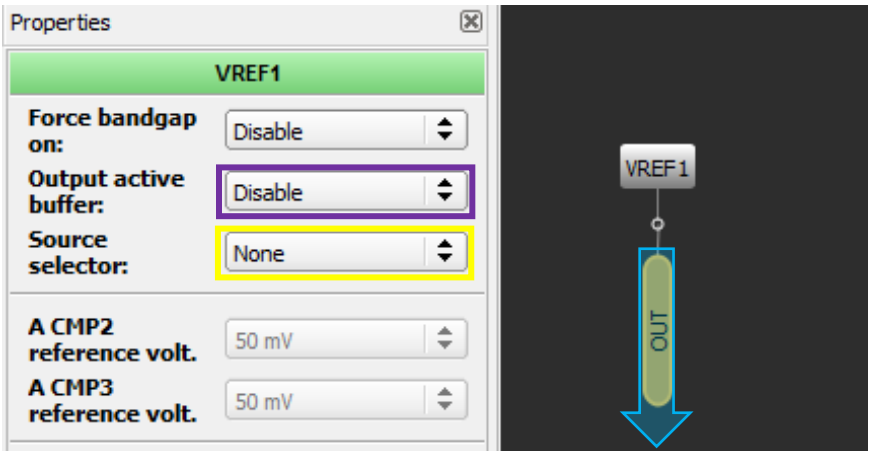




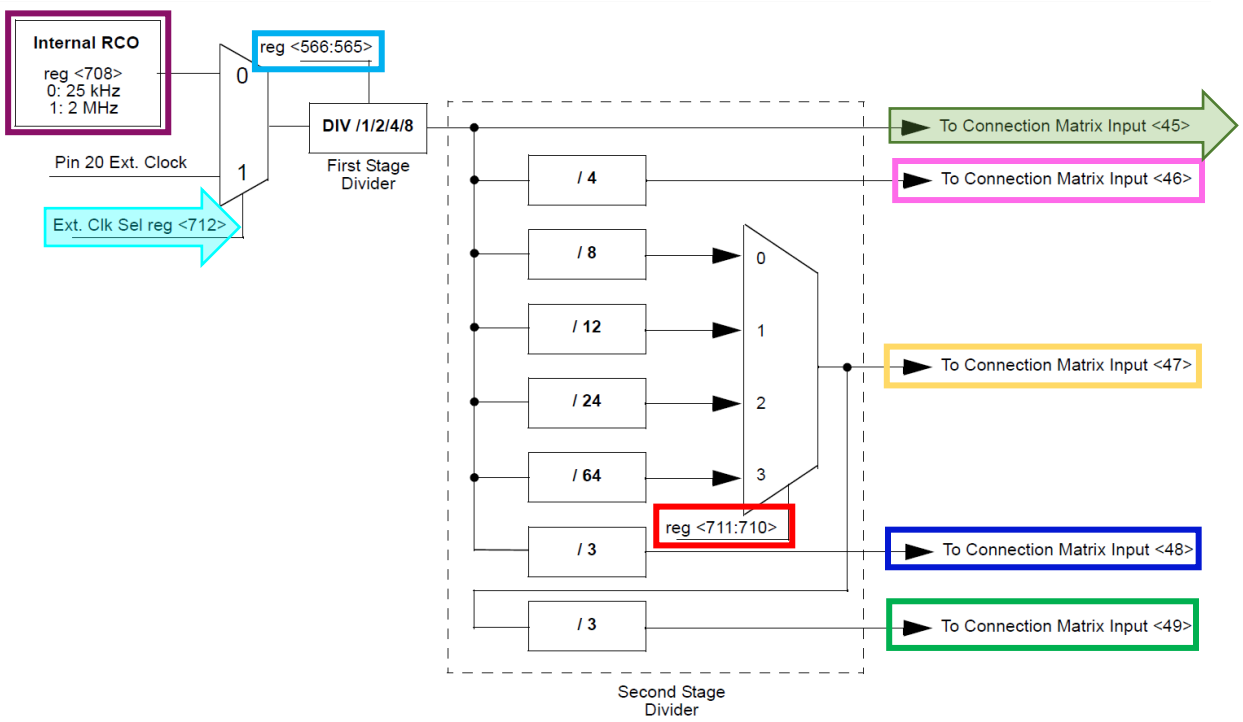
VREF Block Diagram

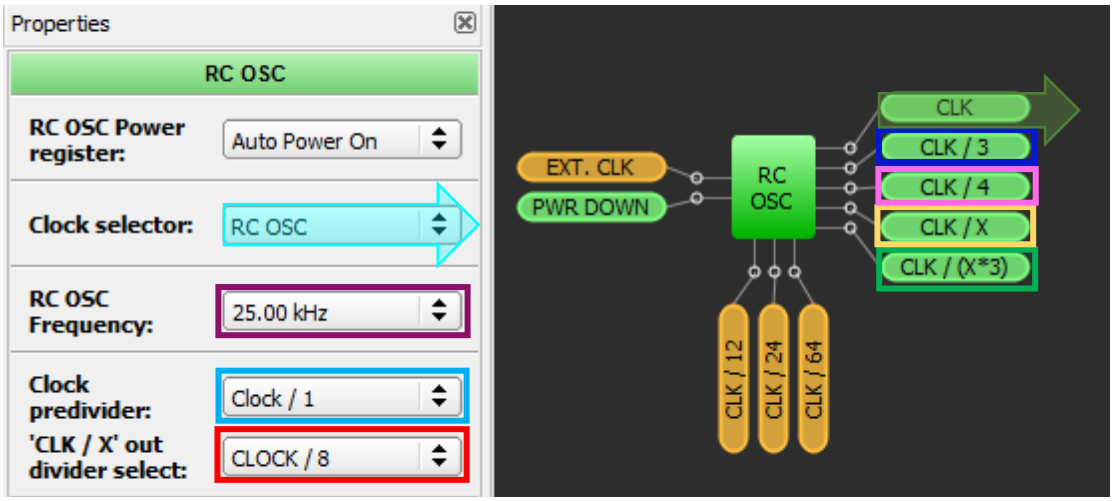





Properties
VREF1
Force bandgap on:
Disable
Output active buffer:
Disable
Source selector:
None
A CMP2 reference volt.
50 mV
A CMP3 reference volt.
50 mV
VREF1
OUT

RC OSC Block Diagram





A. Example projects.

In the GreenPAK Designer **Help** menu, you can find a link to the **Application Notes** web page for selected chip revision. There you can find fully configured examples which can help get your projects completed more quickly. Each example has documentation that contains diagrams and descriptions.

B. Major features history.

GPD 6.02:

- Demo page (Section 2);
- Demo mode (Section 3.12);

GPD 6.05:

- Simulation mode (Section 8);

GPD 6.09:

- Debug tool (Section 6);

GPD 6.17:

- Snipping tool (Section 3.13);

GPD 6.25:

- EPG Waveforms Editor (Section 3.14);

GPD 6.26:

- Timing diagram (Section 3.15);
- Debug Tool (Section 6);
- PowerPAK Debug Tool (Section 6.5);