ThinkRF WSA5000

Real Time Spectrum Analyzer
Version 0.7
User Guide

October 2014
Document no. 75-0024-141001

Important notice

The information in this guide is furnished for informational use only and is subject to change without notice. ThinkRF Corporation assumes no responsibility or liability for any errors or inaccuracies that may appear in this document.

Trademarks

ThinkRF Corp

390 March Road
Kanata, ON K2K 0G7
(613) 369-5104

Table of Contents

Preface

This section describes the audience for, the organization of, and conventions used in this document. It also identifies related documentation and explains how to access electronic documentation.

Audience

This document is written for technical people who have basic understanding, familiarity and experience with network test and measurement equipment.

Conventions

The following conventions and notations are used in this document:

Note: This symbol means **take note**. Notes contain helpful suggestions or references to additional information and material.

Caution: This symbol means **be careful**. In this situation, you might do something that could result in equipment damage or loss of data.

Warning: This symbol means **danger**. You are in a situation that could cause bodily injury. Before you work on any equipment, be aware of the hazards involved with electrical circuitry and be familiar with the standard practices for preventing accidents.

Obtaining the Latest Documentation and Software

Please visit the ThinkRF website at **http://www.thinkrf.com/documentation** to obtain the latest documentation. Latest software and firmware releases are also available on the ThinkRF website at **http://www.thinkrf.com/download**.

Document Feedback

Please send your comments about this document or any ThinkRF documentation to **support@thinkrf.com**.

We appreciate your feedback.

Obtaining Technical Assistance

For all customers who hold a valid end-user license, ThinkRF provides technical assistance 9 AM to 5 PM Eastern Time, Monday to Friday. Contact us at **support@thinkrf.com** or by calling **+1.613.369.5104**.

Before contacting support, please have the following information available:

- WSA5000 serial number which is located on the identification label on the WSA5000's underside.
- The firmware version running on the WSA5000.
- Versions of ThinkRF software you are using, including the API libraries to third-party applications.
- The operating system and version you are using.

Overview of the WSA5000 Real Time Spectrum Analyzer

The WSA5000 Real Time Spectrum Analyzer (RTSA) is a full-featured spectrum analysis application designed to work with the WSA5000 Wireless Signal Analyzer. The RTSA application provides standard spectrum analysis features including frequency controls, marker functions, and multi-trace capabilities.

The WSA5000 Wireless Signal Analyzer is a high-performance software-defined RF receiver, digitizer and analyzer. With patent-pending software-defined RF receiver technology, the WSA5000 provides industry leading combined sensitivity, tuning range, instantaneous bandwidth (IBW) and scan rate. It has a hybrid RF receiver front-end (RFE) combining super-heterodyne, direct-conversion and direct digitization technologies with both wide-band and narrow-band ADCs. Additionally, it provides hardware-based triggering and capture control, thereby enabling sophisticated sweeping and real-time capture.

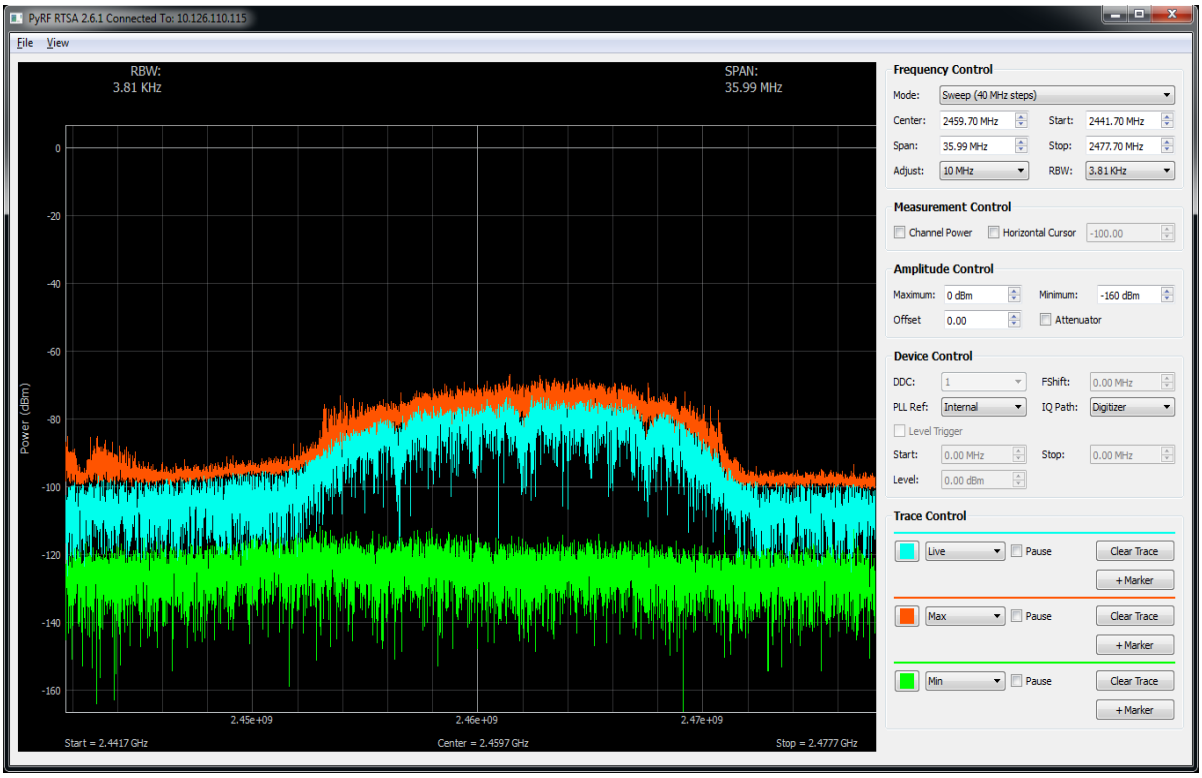
The WSA5000 RTSA can be used to measure signals such as Wi-Fi, Bluetooth, and LTE, and markers can be used to track specific frequencies. Sophisticated FPGA-based level triggers within the WSA5000 Wireless Signal Analyzer enables the capture of elusive, time-varying signals across an instantaneous bandwidth of up to 100 MHz. The record-and-playback feature allows you to capture signal data and store it, and replay the captured signal at a later time. In addition, you can export captured data to a CSV file for further processing.

The WSA5000 RTSA application includes an intuitive user interface that incorporates a full set of controls for fast and simple control of application features. Fluid mouse navigation over the plot area allows you to quickly view the entire spectrum and easily zoom into areas of interest.

The WSA500 RTSA application is built on the PyRF development framework, and is open-source under BSD licensing. PyRF handles the low-level details of real-time acquisition, signal processing and visualization, and provides feature-rich libraries, sample applications and source code, all specific to the requirements of signal analysis. For more information, visit the PyRF site at http://www.pyrf.org/.

WSA5000 RTSA User Interface

The WSA5000 RTSA user interface provides controls to manage the frequency and amplitude display, the RFE mode of the WSA5000 Wireless Signal Analyzer, trace capabilities, multiple plots such as spectrogram and raw IQ data, as well as measurement tools.



Note: This section provides an overview of the WSA5000 RTSA user interface. Please refer to the Configuring the WSA5000 RTSA Application section for details on how to use individual settings and the options for each control.

Frequency Control Area

The Frequency Control area contains parameters for defining the frequency bandwidth for signal monitoring.

Parameter	Function
Mode	Defines the Radio Front End (RFE) mode of the WSA5000 Wireless Signal Analyzer, and the instantaneous bandwidth. Available options are: • Sweep (40 MHz steps) • Sweep (100 MHz steps) • 100 MHz span • 40 MHz span • 10 MHz span • 0.1 MHz span (high dynamic range) • 0-50 MHz (no tuning) • 100 MHz IQ input (no tuning)
Center	Defines the center point of the frequency to be monitored, in MHz. Default value is 2450.00 MHz.
Span	Defines the frequency span to be monitored (sweep modes only). Changing this value changes the **Start** and **Stop** values Default value is 125.0 MHz.
Start	Defines the beginning of the frequency range to be monitored (sweep modes only). Changing this value changes the **Span** value.
Stop	Defines the end of the frequency range to be monitored (sweep modes only). Changing this value changes the **Span** value.
Adjust	Sets the interval of incrementation for the **Center**, **Span**, **Start** and **Stop** fields (for up/down arrows). Default value is 10 MHz.
RBW	Defines the Resolution Bandwidth (that is, determines the frequency between two adjacent bins in a given trace). Default value is 488.28 kHz.

Measurement Control Area

The Measurement Control area contains parameters for identifying measurements for a sub-section of the signal being monitored.

Parameter	Function
Channel Power	Enables or disables selection of a sub-band of the existing spectrum on the plot area, and measurement of the sub-band's channel power.
Horizontal Cursor	Enables or disables display of a horizontal marker on the plot area at a specific power level.

Amplitude Control Area

The Amplitude Control area contains parameters for defining the amplitude range for signal monitoring.

Parameter	Function
Reference Level	Defines the maximum power level for the monitored signal. Default value is 0 dBm.
Minimum Level	Defines the minimum power level for the monitored signal. Default value is -160 dBm.
Offset	Defines the offset power level applied to the signal. This offset is only applied in the software; the actual signal is not changed.
Attenuator	Enables or disables the RFE's 20 dB attenuation of the incoming signal.

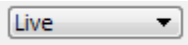
Device Control Area

The Device Control area contains parameters related to the WSA5000 Wireless Signal Analyzer operation.

Parameter	Function
DDC	Defines the decimation rate of the signal (that is, the decimation rate of raw samples applied in the FPGA). This parameter is disabled for Sweep SH and Sweep ZIF modes.
FShift	Defines the value used for frequency shifting. This parameter is disabled for Sweep SH and Sweep ZIF modes.
PLL Ref	Specifies the synchronization source for the PLL reference clock. Available options are: • Internal • External
IQ Path	Specifies the path for analog IQ signals. Available options are: • Digitizer (internal WSA5000 digitizer) • Connector (external digitizer through I/Q output connectors)
Level Trigger	Enables or disables triggering on signals of interest (only enabled for ZIF, SH and SHN modes).
Start	Defines the minimum frequency threshold for triggering signal capture.
Stop	Defines the maximum frequency threshold for triggering signal capture.
Level	Defines the power level for triggering signal capture.

Trace Control Area

The Trace Control area contains parameters for setting traces and adding markers on the signal display. You can add up to three traces to the signal display, and up to two markers.

Parameter	Function
	Defines the color displayed for the trace.
	Defines the display mode for the trace. Available options are: • Live • Max • Min • Average • Off
Pause	Pauses the trace display.
Clear Trace	Resets the trace display.
+ Marker	Adds a marker to the signal display.

Getting Started with the WSA5000 RTSA Software

Before installing the WSA5000 Real Time Spectrum Analyzer software, ensure that your PC meets the minimum system requirements:

- Operating System:
  - Windows XP
  - Windows 7 (32-bit or 64-bit)
  - Windows 8 (32-bit or 64-bit)
- RAM: 2 gigabytes (GB)
- Hard Disk: 1 gigabyte (GB) available

Note: You must have an Internet connection to download the WSA5000 RTSA software from the ThinkRF website.

Installing the WSA5000 RTSA Software

You must download the WSA5000 RTSA installer from the ThinkRF website to install the software on your computer.

1. Open a web browser, and go to http://thinkrf.com/download.
2. Click the **Download** button to download the zip file for the latest software and firmware release (for example, WSA5000 RTSA Release 140909).
3. When you have downloaded the zip file, extract the contents to a folder of your choice.
4. Double-click on the `pyrf_RTSA_Installer_x.y.z.exe` file to launch the Setup Wizard, and follow the prompts to select your installation options and install the software.
5. On the Welcome screen, click **Next**.
6. On the License Agreement screen, select “I accept the agreement” and click **Next**.
7. On the Select Destination Location screen, specify a path for software installation (or keep the default path) and click **Next**.
8. On the Select Component screen, select the software components you want to install and click **Next**.
9. On the Select Start Menu Folder screen, specify a path for the program shortcut in the Start Menu (or keep the default) and click **Next**.

10. On the Select Additional Tasks screen, specify whether you want the installer to create a desktop shortcut to the WSA 5000 RTSA application, and click **Next**.
11. On the Ready to Install screen, review your installation settings. Click **Install** to proceed with the installation, or **Back** to change your settings.
12. When the Setup Wizard has completed software installation, click **Finish** to exit the Wizard.

Connecting to a WSA5000 Device

You must connect to a WSA5000 Wireless Signal Analyzer to view data in the WSA5000 RTSA application. The application detects any WSA5000 devices that are on the same subnet of your network, but you must explicitly select one to complete the connection.

1. Launch the WSA5000 RTSA application from the Start menu or desktop icon (if installed).

   The application displays the “Open Device” window.

2. Select a device from the list of auto-discovered devices. You can click the **Refresh** button to obtain an updated list of devices.

   Alternatively, you can enter the IP address of the WSA5000 device manually in the “Manually Enter Device IP” field.

3. Click **Connect**.

   The WSA5000 RTSA application connects to the WSA5000 device and displays the user interface. By default, the WSA5000 acquires and measures signals automatically.

Configuring the WSA5000 RTSA Application

The WSA5000 Real Time Spectrum Analyzer application provides several options for controlling and displaying signal capture through the WSA5000 Wireless Signal Analyzer, including RFE mode, frequency and amplitude controls, trigger levels, and trace display.

Configuring Frequency Settings

You can set frequency for the signal to be monitored using the parameters in the Frequency Control area. You can enter values for these fields manually, or use the up/down arrows to increase or decrease the values. Alternatively, you can also click-and-drag the mouse over the plot display area to change the frequency to be monitored.

As you specify or change these values, the axes on the signal waveform display adjust accordingly. A vertical line on the plot display indicates the center point of the frequency band being monitored.

1. In the Frequency Control area, select the device RFE mode (with corresponding instantaneous bandwidth) from the drop-down list in the **Mode** field:

Device Mode	Description
Sweep (40 MHz steps)	40 MHz segments (instantaneous bandwidth), in sweep mode.
Sweep (100 MHz steps)	100 MHz segments (instantaneous bandwidth), in sweep mode.
100 MHz span	100 MHz span (instantaneous bandwidth) in fixed mode.
40 MHz span	40 MHz span (instantaneous bandwidth) in fixed mode.
10 MHz span	10 MHz span (instantaneous bandwidth) in fixed mode.
0.1 MHz span (high dynamic range)	100 kHz span (instantaneous bandwidth) in fixed mode; best for low-power signals.
0-50 MHz (no tuning)	0-50 MHz span; no tuning. Dedicated to RF IN connector (when device is used as a digitizer only).
100 MHz IQ input (no tuning)	100 MHz span; no tuning. IQ In mode (for external IQ input).

2. Set the central point of the frequency band you want to monitor by specifying a value in the **Center** field.
3. For Sweep modes only, specify the frequency sweep range. You can specify the range in two ways:

- specify start/stop frequency values in the **Start** and **Stop** fields (either manually or using the up/down arrows)
- specify a value in the **Span** field (either manually or using the up/down arrows) – this value is applied relative to the center frequency value

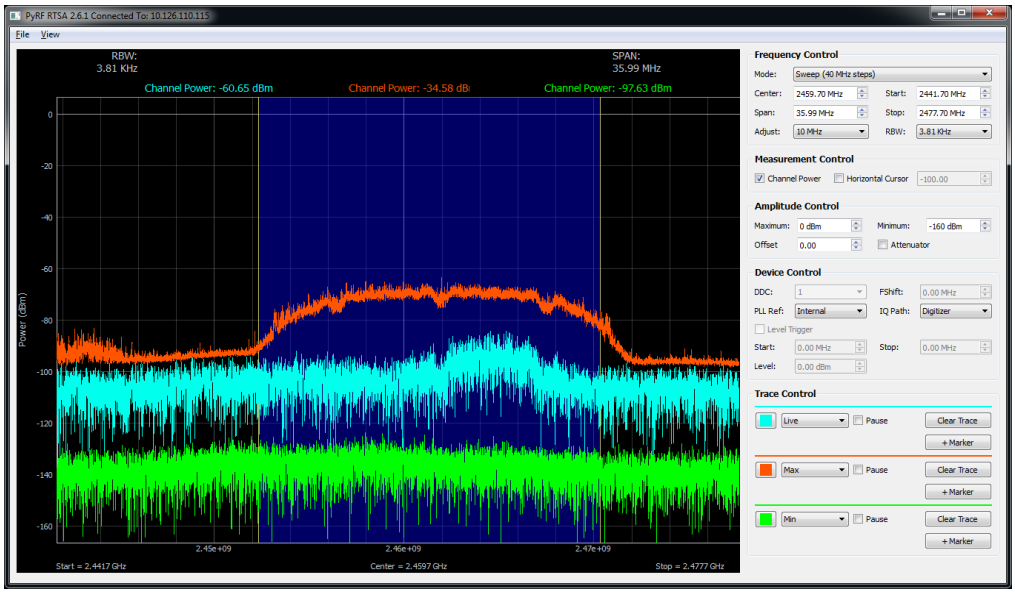
4. Optionally, specify the interval of frequency control field values by setting the **Adjust** field.
5. Set the Resolution Bandwidth for the signal by selecting a value from the drop-down list in the **RBW** field.

Configuring Channel Measurement Settings

You can define a section of the signal being monitored and view the channel power of that section, by enabling the **Channel Power** in the Measurement Control area.

1. In the Measurement Control area, select a sub-section of the signal by enabling the **Channel Power** field.

The plot display shows two vertical lines to define the frequency boundaries of the channel (default position is at the center of current spectrum), and the background color between the boundaries changes. The measured channel power is displayed above the signal plot.



2. Optionally, change the span of the channel by clicking and dragging the left or right boundary of the channel on the plot display.
3. Optionally, insert a horizontal line on the plot display to mark a specific power level by selecting the **Horizontal Cursor** box.

   You can change the position of the horizontal line by clicking and dragging the line on the plot display, or by changing the power level in the field beside by the **Horizontal Cursor** box.

Configuring Amplitude Settings

You can set the amplitude for the signal to be monitored using the parameters in the Amplitude Control area. You can enter values for these fields manually, or use the up/down arrows to increase or decrease the values.

1. In the Amplitude Control area, set the maximum power level for input signals by specifying a value in the **Reference Level** field.
2. Set the minimum power level for input signals by specifying a value in the **Minimum Level** field.
3. Optionally, configure a power offset for the signal display by specifying a value in the **Offset** field. Note that the offset is applied in software only and is reflected in the signal display (including any recorded signals for playback); however, the input signal itself is not affected.
4. Optionally, enable the device's 20 dB attenuation of the incoming signal by selecting the **Attenuator** box (enabled by default).

Configuring the Device Control Settings

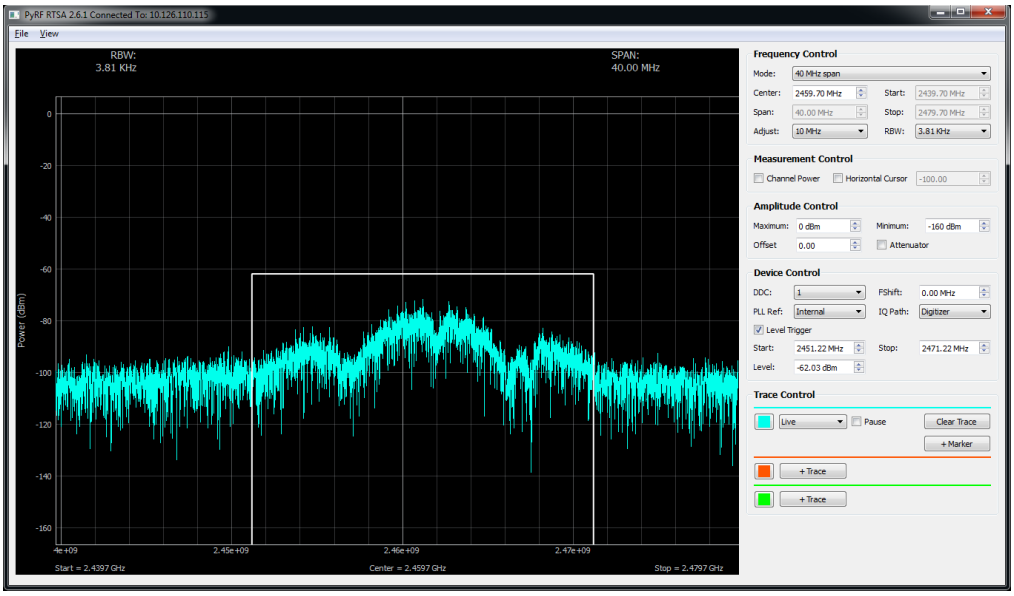
You can ensure signal information is captured and displayed in a manner appropriate to your application using the parameters in the Device Control area.

1. Set the decimation rate for the signal by selecting a value from the drop-down list in the **DDC** field. (Disabled for sweep modes.)
2. Set the value of the frequency shift in the **FShift** field. A negative value corresponds to a left shift. (Disabled for sweep modes.)
3. Specify the synchronization source for the PLL reference clock by selecting an option from the drop-down list in the **PLL Ref** field (either Internal or External).
4. Specify the IQ output path by selecting an option from the drop-down list in the **IQ Path** field (either Digitizer or Connector).

Note: If you select the “Connector” option for IQ Path, you are using the device in Down-Converter mode and bypassing all internal digitizing functions. The signal display area disappears from the interface when you select this option.

5. Enable signal capture triggering by selecting the **Level Trigger** box. (Enabled only for fixed modes of 100 MHz, 40 MHz or 10 MHz instantaneous bandwidth.)

   The plot display shows a rectangle representing default trigger parameters: minimum frequency (left boundary), maximum frequency (right boundary), and power level (top boundary).



6. If you have enabled signal capture triggering, set the trigger threshold values:
   - specify the minimum frequency in the **Start** field
   - specify the maximum frequency in the **Stop** field
   - specify the power level in the **Level** field

   You can also set the trigger parameters by clicking and dragging the left, right and top boundaries to change the minimum frequency, maximum frequency and power level respectively.

Configuring Traces for the Signal Display

You can configure up to three active traces in the signal display using the parameters in the Trace Control area. By default, the first trace is set to display all values captured for the signal. The following procedure applies to each trace.

1. In the Trace Control area, select the display mode for the trace from the drop-down list in the field beside the trace label:

Trace Mode	Description
Live	The trace plots all values captured for the signal.
Max	The trace plots only the maximum values captured for the signal.
Min	The trace plots only the minimum values captured for the signal.
Average	The trace plots the average values captured for the signal over the number of samples specified. When you select this mode, a new field appears beside the mode field where you must specify the number of samples to be used for averaging.
Off	Trace is removed from the display.

2. Optionally, change the color of the trace by clicking on the color icon beside the display mode button, and select a color from the palette.
3. Select the **Pause** box to freeze the trace display.
4. Click the **Clear Trace** button to reset the trace display.
5. Repeat steps 1 - 3 for additional traces.

Adding Markers to the Signal Display

You can add up to two markers to the signal display. The markers can be associated with any trace on the display.

By default, the marker is displayed at the center frequency, and moves on the waveform as it measures power levels. The frequency and power values at the marker point are displayed at the top of the signal display window.

You can change the marker position by clicking and dragging the marker to the left or to the right.

1. Select a trace in the Trace Control area and click the **+Marker** button.
2. Click the **Peak** button to position the marker at the highest peak signal.
3. Click the **Peak Left** button to position the marker at the highest peak signal to the left of the current marker position.
4. Click the **Peak Right** button to position the marker at the highest peak signal to the right of the current marker position.
5. Click the **Center** button to change the center frequency to match the current marker position.
6. Repeat steps 1 - 5 to add a second marker to the signal display.
7. To remove a marker, click on the **–** button in the M1 or M2 area.

Displaying I/Q Plots

You can view an I/Q plot display of captured signal information (this view is particularly useful when using the device in IQIN mode). The I/Q plots appear below the main signal display in the interface, and include two plots:

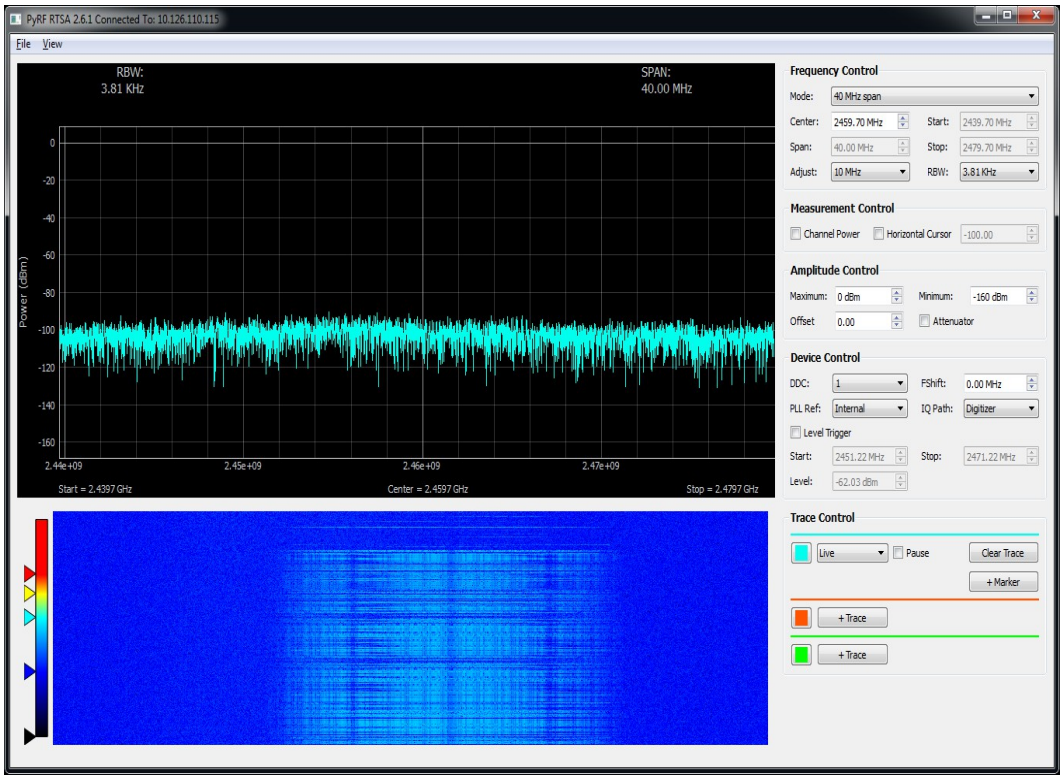
- Constellation plot (left side) shows the in-phase (I data) vs quadrature (Q data) plot
- Time domain plot (right side): shows a trace for the In-phase (I data in green) and a trace for the quadrature (Q data in red, if available in the mode)

1. Open the **View** menu, and select **I/Q Plots** from the drop-down menu.

   The I/Q plots appear below the main signal waveform display.
2. To hide the I/Q plots, open the **View** menu and deselect the **I/Q Plots** option.

Displaying the Spectrogram

You can view a spectrogram display of captured signal information. The spectrogram displays spectrum changes over time in a three-dimensional view. Time is displayed on the horizontal axis, and frequency on the vertical axis. The amplitude of the frequency components is expressed by color.



1. Open the **View** menu, and select **Spectrogram** from the drop-down menu.

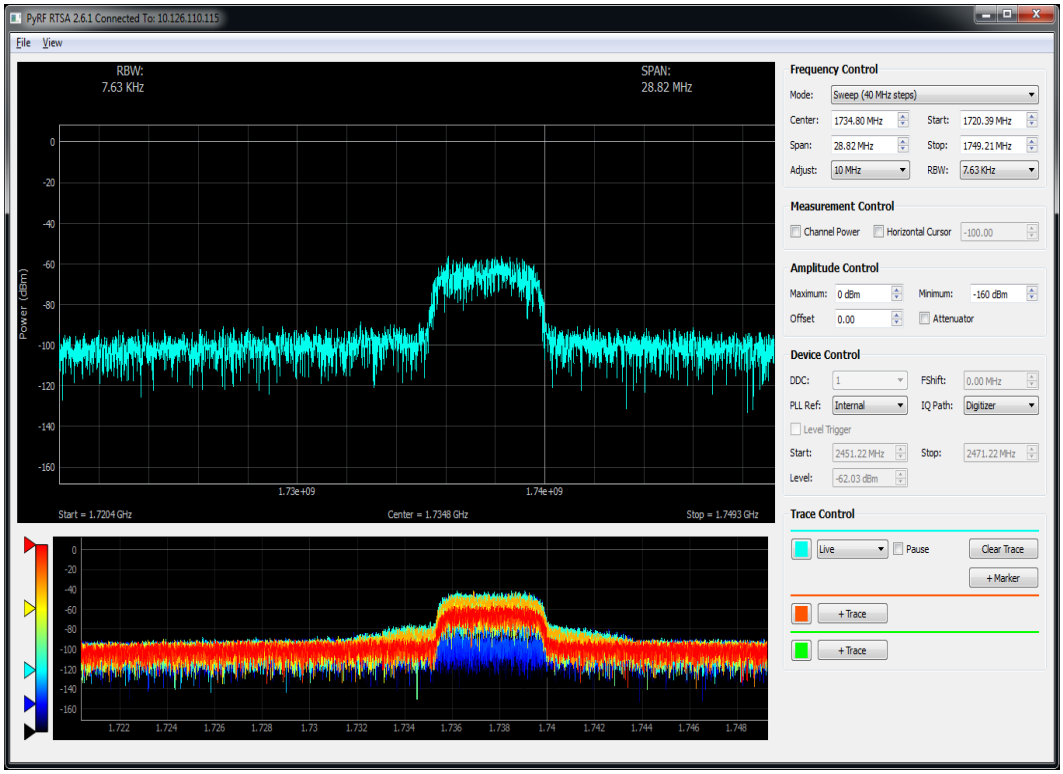
   The spectrogram plot appears below the main signal waveform display.

2. Optionally, you can enhance the display of the signal(s) of interest by sliding the triangles on the color palette (to the left of the display) to adjust the threshold at which the signal changes color. This allows the user to specify colors for each noise floor level.

3. To hide the Spectrogram plot, open the **View** menu and deselect the **Spectrogram Plot** option.

Displaying the Persistence Plot

You can view a persistence plot display of captured signal information. The persistence plot is a view of how long a signal remains present; as it grows weaker, it fades in color. Amplitude is displayed on the horizontal axis, and frequency on the vertical axis.



1. Open the **View** menu, and select **Persistence Plot** from the drop-down menu.

   The persistence plot appears below the main signal waveform display.

2. Optionally, you can enhance the display of the signal(s) of interest by sliding the triangles on the color palette (to the left of the display) to adjust the threshold at which the signal changes color. This allows the user to choose the colors that the persistence plot uses to represent the shades of persistence of the spectrum.

3. To hide the Persistence plot, open the **View** menu and deselect the **Persistence Plot** option.

Capturing Data from the WSA5000 RTSA Application

In addition to viewing signal data in real time on the WSA5000 Real Time Spectrum Analyzer application, you can also capture and store signal information for playback, or write the data to a CSV file for further processing by another application.

Caution: The WSA5000 RTSA application does not impose a limit on the file size for signal recording or writing to a CSV file. If recording is not explicitly stopped, the application continues to write data to the file until the local hard drive is full.

Recording a Signal for Playback

The WSA5000 RTSA application allows you to record a signal and save the file for future playback. All recorded signals are stored on the PC that is hosting the WSA5000 RTSA application.

1. To begin recording the signal data, open the **File** menu and select the **Start Recording** option.

   The RTSA application begins recording the signal data displayed in the interface.

2. To stop recording the signal data, open the **File** menu and select the **Stop Recording** option.

   The WSA5000 RTSA application stops recording the signal data displayed in the interface, and stores the recorded file (with .vrt extension) in the `pyrf/RTSA/Scripts/dist` directory.

Playing Back a Recorded Signal

The WSA5000 RTSA application allows you to play back a previously recorded signal. The .vrt file must be located on the PC that is hosting the RTSA application.

1. Open the **File** menu and select the **Playback Recording** option.

   The application opens a Explorer window, defaulting to the location of the `pyrf/RTSA/Scripts/dist` directory (Note you can move to another folder destination using the explorer window).

2. Select the .vrt file you want to play, and click **Open**.

   The signal display area plays back the recorded signal continuously, in loopback mode.

Note: You can add traces and markers to the signal playback display. Frequency and device controls are disabled.

3. To stop playback of the recorded file, you must do one of the following:
    - open a different file for playback (repeat step 1)
    - reconnect to the device (open the **File** menu and select the **Open Devic**e option)
    - exit the WSA5000 RTSA application

Exporting Signal Data to a CSV File

The WSA5000 RTSA application allows you to capture and export signal data to a CSV file for further processing by another application. All recorded data is stored on the PC that is hosting the WSA5000 RTSA application.

The CSV file contains the following information about the captured signal:

Parameter	Description
`mode`	The RFE mode of the device during signal capture. Possible values are: • ZIF (100 MHz instantaneous bandwidth) • SH (40 MHz instantaneous bandwidth) • SHN (10 MHz instantaneous bandwidth • HDR (100 kHz instantaneous bandwidth) • DD (0-50 MHz instantaneous bandwidth; no tuning) • IQIN (100 MHz instantaneous bandwidth; no tuning)
`fstart`	The start of the frequency range (Hz).
`fstop`	The end of the frequency range (Hz).
`size`	Number of data points (power values) captured for the signal in the CSV file.
`timestamp`	ISO time stamp representing time at which the data was displayed in the RTSA
<data>	A series of integers representing the power value of the captured signal for each sample. The number of data samples corresponds to the value specified in the `size` parameter.

1. To begin capture and export of the signal data, open the **File** menu and select the **Start Exporting CSV** option.

   The RTSA application opens an Explorer dialog box to specify the name and location of the CSV file.
2. Specify a name and storage location for the file and click **Save**. By default, the system stores the file (`csv-0000.csv`) in the the `pyrf/RTSA/Scripts/dist` directory. The system begins writing the signal data to the CSV file at the specified location.
3. To stop writing signal data to the CSV file, open the **File** menu and select the **Stop Exporting CSV** option.

   The WSA5000 RTSA application stops writing the signal data to the CSV file.